Numerical investigation of cavitation flow inside spool valve with large pressure drop

Jian Deng, Dingyi Pan, Fangfang Xie and Xueming Shao
State Key Laboratory of Fluid Power Transmission and Control, Department of Engineering Mechanics, Zhejiang University, Hangzhou 310027, China

Abstract. Spool valves play an important role in fluid power system. Cavitation phenomena happen frequently inside the spool valves, which cause structure damages, noise and lower down hydrodynamic performance. A numerical tools incorporating the cavitation model, are developed to predict the flow structure and cavitation pattern in the spool valve. Two major flow states in the spool valve chamber, i.e. flow-in and flow-out, are studies. The pressure distributions along the spool wall are first investigated, and the results agree well with the experimental data. For the flow-in cases, the local pressure at the throttling area drops much deeper than the pressure in flow-out cases. Meanwhile, the bubbles are more stable in flow-in cases than those in flow-out cases, which are ruptured and shed into the downstream.

1. Introduction
Spool valves are widely used in fluid power system. However, due to the complicated internal structure and high speed flow stream, cavitation phenomena are found in these valves [1]. It is normally harmful that cavitation may cause structure damages, produce noise [2], and lower down system hydrodynamic performance. Experimental and numerical endeavors has been given to predict the cavitation inception and effects of cavitation, and correlate the hydraulic parameters, valve structure and cavitation phenomena [3, 4].

In current study, concerning typical throttling valves with pairs of U-shaped grooves, a fully numerical simulation tool has been developed, and it has been applied to predict the viscous heating phenomenon and spool deformation [5, 6]. By incorporating the cavitation model, we are aiming to investigate the correlation between cavitation phenomena, e.g. cavitation inception, evolution of cavitation, and relevant hydraulic parameters, e.g. pressure drop, local structure, valve opening, etc. Two major classes of cases are investigated, flow-in cases and flow-out cases, corresponding to different local structures. Comparison with the experiment are given, which convinced the reliability of the numerical tool. In the following, the physical model and numerical method are introduced, and then, numerical results and discussion are presented.

2. Physical model and numerical method
In current study, each spool consists two pairs of symmetric U-shaped grooves on both sides of the neck. The corresponding schematic slices of the valve section of two typical cases are shown in Fig. 1. For the flow-in cases, the left pair of grooves and the sleeve forms the throttling area, and for flow-out cases, the right pair of grooves take places. The geometry of the groove

1 To whom any corresponding should be addressed. Email: dpan@zju.edu.cn
Figure 1. Schematics of the valve structure and working condition. Cut-view for flow-in (a) and flow-out (b) cases. Locally enlarged view for flow-in (c) and flow-out (d) cases.

is described by the following parameters, i.e. groove length \((L)\), end radius \((r)\) and depth \((h)\). These parameters are fixed as \(L = 5\, \text{mm}\), \(r = 1\, \text{mm}\) and \(h = 1.5\, \text{mm}\). The leakage between the spool and the sleeve is neglected. It is worthy to mention that there are two possible narrowest areas in the chamber, i.e. \(A_1\) and \(A_2\). With the variation of the opening, \(A_2\) increases monotonously while \(A_1\) remains constant.

The hydraulic oil in the system, is assumed to be incompressible, viscous and isothermal. The Navier-Stokes equations are solved numerically with commercial software FLUENT. The finite volume method (FVM) and the SIMPLEC algorithm are adopted \cite{7}. The structured meshes are generated, yielding precise numerical accuracy. There are 290000 – 346000 cells and the throttling area is refined along the short edge of the groove. The Renormalization-group \(k - \epsilon\) model is employed for turbulence modelling, which has a good adaptability for flow separation. The fluid is assumed as a mixture of liquid and vapor. The transport equation of vapor mass fraction is

\[
\frac{\partial}{\partial t} (\rho f) + \nabla \cdot (\rho u f) = \nabla \cdot (\mu \nabla f) + R_e - R_c, \tag{1}
\]

where \(u\) is the velocity, \(\mu\) the effective exchange coefficient, \(f\) the vapor mass fraction, and \(R_e\) and \(R_c\) the vapor generation and condensation rate, derived from the Rayleigh-Plesset equation, taking account of the effects of turbulence-induced fluctuation and non-condensable gas \cite{8}

\[
R_e = C_e \frac{\sqrt{k}}{\sigma} \rho_l \rho_v \left[ \frac{2}{3} \frac{P_v - P}{\rho_l} \right]^{\frac{3}{2}} (1 - f) \quad (P < P_v); \quad R_c = C_c \frac{\sqrt{k}}{\sigma} \rho_l \rho_v \left[ \frac{2}{3} \frac{P - P_v}{\rho_l} \right]^{\frac{3}{2}} f \quad (P > P_v).
\]

Here, \(\sigma\) represents the surface tension, \(P_v\) is the saturated vapor pressure, and the values of the empirical constants of \(C_e\) and \(C_c\) are 0.02 and 0.01, respectively. To validate our numerical method, a comparison with the experiment result \cite{4} is given in Fig. 2. The bubble interfaces are demonstrated as \(\rho = 700\, \text{kg/m}^3\), and it is shown that the bubble sizes at different outlet pressures are quite close to the experiment results.

3. Results and discussions

The fluid properties are: liquid density \(\rho_l = 889\, \text{kg/m}^3\) and kinetic viscosity \(\nu_l = 4 \times 10^{-5}\, \text{m}^2/\text{s}\), vapor density \(\rho_v = 10.95\, \text{kg/m}^3\) and kinetic viscosity \(\nu_v = 7 \times 10^{-6}\, \text{m}^2/\text{s}\), and saturated vapor pressure \(P_v\) = 50kPa. The inlet pressure is set to \(P_1 = 3\, \text{MPa}\), and the outlet pressure \((P_2)\) varies, leading to different pressure drops. The turbulence intensity at inlet is set to 5\%, and the hydraulic diameter is set to 0.02m. Standard wall function is applied, corresponding to a fully developed turbulent flow in a circular tube. For each major class, i.e. the flow-in or flow-out cases, results of two typical pressure drops are compared.
Figure 2. Comparison of the bubbles patterns of the experimental [4] and numerical results at different outlet pressures.

Figure 3. Pressure distribution along the spool direction ($L_x$) of flow-in cases with two typical outlet pressure constants, $P_2 = 0.98$MPa and $P_2 = 0.1$MPa (a), and the cavitation bubbles pattern for low outlet pressure case ($P_2 = 0.1$MPa) with different openings (b) and (c).

3.1. Flow-in cases

Fig. 3 (a) shows the local pressure along spool wall in throttling area. $x$ represents opening as shown in Fig. 1. The pressure drops quickly at $L_x = 0$, where the cross section area of the flow chamber turns to be minimal ($A_1$ in left sub-figure of Fig. 1). After then, as the cross section area increases, the pressure recovers quickly to the background pressure (outlet pressure) with fluctuation. Relevant experimental data are also given as solid symbols in Fig. 3 (a), and the results agree well with the experiment.

The patterns of the cavitation bubbles at $P_2 = 0.1$MPa are given in Fig. 3 (b) and (c), corresponding to different openings. It is shown that cavitation bubbles develop at the minimal pressure point, and are elongated at downstream. The larger opening the longer bubble.

3.2. Flow-out cases

The pressure distribution of flow-out cases are given in Fig. 4. For all cases, the pressure first keeps constant close to the inlet pressure, and then, decreases at $L_x = 4$, where the cross section area becomes minimal. Compared to flow-in cases, the pressure only decreases to the back pressure, without a recovery process from a low pressure to the back pressure. The decrease slopes are much smoother the those of the flow-in cases. It is believed that, the local structures around the throttling area of the major classes are quite different with each others, and hence, resulting in different flow structures with different pressure distributions.

According to the pressure results, we conjecture that it is difficult to induce the cavitation inception in flow-out cases, and the numerical results confirm this. It is difficult to capture the cavitation bubble structures if the interface is set to $\rho = 700$kg/m$^3$, indicating that there is only
Figure 4. Pressure distribution along the spool direction ($L_x$) of flow-out cases with two typical outlet pressure constants, (a) $P_2 = 0.98$MPa and (b) $P_2 = 0.1$MPa.

Figure 5. Cavitation bubbles pattern for low outlet pressure ($P_2 = 0.1$MPa) of flow-out cases with different openings.

little amount of liquid has phase changed. Instead, the interface with $\rho = 200$kg/m$^3$ are given in Fig. 5. Snapshots with three different openings are presented. The larger opening the bigger bubble. In the downstream the bubbles are first narrowed and then expanded, and are ruptured at the narrow neck and then shed into the downstream. We does not capture this phenomenon in flow-in cases, in which the cavitation bubbles are more stable than those in flow-out cases.

4. Conclusions
Two major flow states in the spool valve are studies numerically. The cavitation model is applied to predict the evolution of cavitation bubbles. The pressure distributions along the spool wall are investigated, and the results agree well with the experiment data. For the flow-in cases, the local pressure at the throttling area drops much deeper than the pressure in flow-out cases, inducing the cavitation inception more easily. The bubbles are more stable in flow-in cases than those in flow-out cases, which are ruptured and shed into downstream. Correlations between local flow structures and cavitation phenomena are required deep investigation in the future.

Acknowledgments
This work is supported by the National Natural Science Foundation of China (Nos: 11272283, 11402230), Zhejiang Provincial Natural Science Foundation of China (No: LY12A02006).

References
[1] McCloy D and Beck A 1967 Proc. Inst. Mech. Eng. 182 163–174
[2] Martin C S, Medlarz H, Wiggert D C and Brennen C 1981 J. Fluids Eng. 103 564–576
[3] Oshima S, Leino T, Linjama M, Koskinen K T and Vilenius M J 2001 Int. J. Fluid Power 2 5–13
[4] Fu X, Du X, Zou J, Ji H, Ryu S and Ochiai M 2007 Int. J. Fluid Power 8 29–37
[5] Deng J, Shao X M and Fu X 2009 Proc. Inst. Mech. Eng., C: J. Mech. Eng. Sci. 223 2571–2581
[6] Deng J, Shao X M, Fu X and Zheng Y 2009 Energy Conversion and Management 50 947 – 954
[7] Patankar S V 1980 Numerical heat transfer and fluid flow (Taylor and Francis)
[8]Singhal A K, Athavale M M, Li H Y and Jiang Y 2002 J. Fluids Eng. 124 617–624