Journal of Applied Sciences

ISSN 1812-5654

Journal of Applied Sciences 13 (10): 1677-1683, 2013
ISSN 1812-5654 / DOI: 10.3923/jas.2013.1677.1683

Relationship Between the Forces on the Spool and Anti-cavitation Performance and Throttling Effect after the Notch of the Valve

Zhu Yu and Gao Xiao-rui
Marine Engineering Institute, Jimei University, Xiamen, People Republic of China

Abstract: In order to reduce the flow force and radial jammed force on the spool and raise the anti-cavitation performance of the spool valve, CFD simulations were adopted to analyze the influences for the forces (include the flow force and radial jammed force) on the spool and the lowest pressure value around the notch by the factors such as the ratio of the areas (the ratio of the flow area through the notch and the flow area through the annular space after the notch), the axial length of the annular space after the notch and the arrangement of the inlet and outlet passages of the spool valve. The result indicates that: along with the increase of the ratio of the areas, the radial jammed forces on the spool increase in one direction and the flow force on the spool decrease in the notch closing direction first and then increase in the notch opening direction; the spool valve has the strongest anti-cavitation performance when the ratio of the areas is 0.5; when the length of the annular space after the notch increases, the variation law of the forces on the spool is the same with the increase of the ratio of the areas but there is the optimum length with which the valve has the best anti-cavitation performance; when the inlet and outlet passages are in opposite direction, the radial jammed force on the spool reduce a lot. This research provides a theoretic base for the designers of the spool valve on board the ship.

Key words: Hydraulic spool valve, flow force, radial jammed force, anti-cavitation performance

INTRODUCTION

Hydraulic spool valve is one of the most common used valves in marine hydraulic system. The flow area is adjusted by sliding the spool in the valve body with the consequent control on the flow rate and flow direction through the valve. Normally the spool is kept in neutral position by the spring forces and is pushed forward and afterward by the magnetic forces. The magnitude of the magnetic force depends on the resistance force of the spool movement. In the view of the energy saving this force should be small as possible as it can. The analyses for the resistance forces become very important. According to the book (Li, 2006), the forces on the spool when it moves include flow force, spring force, friction force and radial jammed force; the flow force is related to the spool structure and the notch geometry; The radial jammed force is related to the notch arrangement, machining precision, the arrangement of the flow passage and so on; the viscosity friction force is related to the fluid viscosity and also the machining precision of the spool. The researches on the flow force and cavitation have been accumulated more at home and abroad. In the papers (Ji *et al*., 2003; Lei and Quan, 2006; Wang *et al*., 2000; Amirante *et al*., 2006a, b; Fang *et al*., 2010; Wang *et al*., 2010), the flow field inside the spool valve has been analyzed and the change laws of the flow force has been gained. In the paper (Zhao *et al*., 2008), the effect of the spool movement and flow rate on the flow force and the radial jammed force has been analyzed. In the paper (ZHU, 2012), the effect of notch type on the internal flow field has been studied.

About 6-12% air can be dissolved in mineral oil at ambient temperature and atmospheric pressure. When the pressure is lower than the air apart pressure the dissolved air will escape and the air bubbles form. In addition, when the pressure continues to decline to the saturated vapor pressure the oil itself will vaporize and a lot of steam bubbles form. Because the saturated vapor pressure is much lower than the air apart pressure, the fluid pressure normally should not be less than the air apart pressure to avoid air bubbles. CFD simulation and transparent model experiment are adopted to analyze the anti-cavitation performance for the hydraulic elements by which the pressure distribution and the lowest pressure point are obtained. In paper (Ji *et al*., 2002; Du *et al*., 2007), the effect studies of the internal structure of the flow passage on cavitation generation are carried out; in study (Weber *et al*., 2001; Timo *et al*., 2003), the lowest pressure point in the simulation flow field is verified by the model

Corresponding Author: ZHU Yu, Marine Engineering Institute, Jimei University, Xiamen, People Republic of China

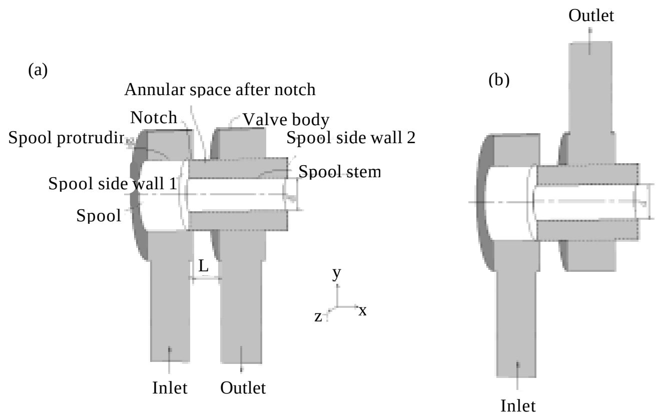

Fig. 1(a-b): 3D model of the valve, (a) Same side of Inlet and Outlet passage and (b) Opposite of Inlet and Outlet passage

experiment; In study (Stares and Roth, 2001; Liao *et al.*, 2003), second or multi stage throttling are put forward in order to improve the anti-cavitation performance of the valve.

So far, researches on flow force and anti-cavitation performance mainly focus on the effects of notch geometry, opening degree and flow passage structure. In the paper (Wang *et al.*, 2000), the effects on the flow force have been analyzed for the dimension of the same side flow passages and the position relative to the slots on the valve body. This study mainly studies the effects on spool force and anti-cavitation performance of the annular space after the notch, the inlet and outlet passage arrangement. It provides theoretic foundation for the valve's design.

GEOMETRIC MODEL FOR THE SPOOL VALVE

First, a real spool valve has been selected and simplified to a theoretic spool valve; Second, the model has been further simplified to single throttling valve chamber base on the flow similarity inside the chamber. The Fig. 1 shows the 3D model of the valve in Pro/E software; L is the length of the annular space after the notch; d is the diameter of the spool stem.

The throttling action of the annual space after the notch is relative to the notch; the different diameters of the spool stem are adopted in this paper; so the flow area (A) of the annual space after the notch is different; if A_v is taken to be the notch flow area, A_v/A is taken as a factor which will affect the flow force and radial jam force and anti-cavitation performance of the valve; the other two factors taken into account in this paper are the length of the annual space and the inlet and outlet arrangements.

MATHEMATIC MODEL CHOSEN

Because the internal flow of the spool valve includes strongly jet-flow, standard k-εand RNG k-ε models are often used to solve this kind problem according to (Ji *et al.*, 2002; Du *et al.*, 2007); but the latter is better when there are strongly bending of the flow lines; so RNG k-ε turbulent model is selected in this paper without considerations of the heat conduction and the compressible of the fluid. General transport equations for RNG k-ε model are as follows:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left(\alpha_k \mu_{eff}\frac{\partial k}{\partial x_j}\right)+G_k+G_b+\rho\varepsilon-Y_M+S_k$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}\left(\alpha_k \mu_{eff}\frac{\partial\varepsilon}{\partial x_j}\right)+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}-R_\varepsilon+S_\varepsilon$$

Here: ρ is the density of the fluid; k is turbulent kinetic energy; ε is turbulent kinetic energy dissipation rate; μ is the velocity; The quantities $\alpha_k\alpha_\varepsilon$ are the inverse effective Prandtl numbers for k and ε, respectively; G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients; G_b is the generation of turbulence kinetic energy due to buoyancy; Y_m represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate; S_k, S_ε are user-defined source terms. $C_{1\varepsilon}$, $C_{2\varepsilon}$, $C_{3\varepsilon}$ are model constants.

NUMERICAL SIMULATION AND RESULTS

Gridding: Before carrying out the CFD analysis, Gambit is used to generate quality computational grid, the area

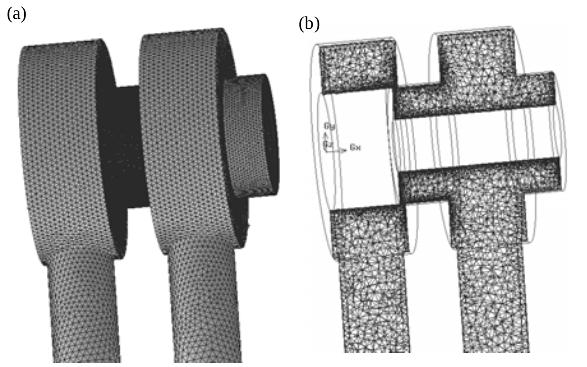

Fig. 2(a-b): Computational grid of the valve, (a) Computational grid of the whole valve and (b) Meshing on the symmetry plane

near the notch has been refined and self-adapted. In order to short the computing time, a half model can be used with a symmetry plane shown on the Fig. 2.

Computational and boundary conditions: Assuming that the fluid is Newtonian liquid, incompressible and without temperature change; the walls the fluid contacts are static; on inlet surface, the pump flow rate value has been enforced; on the outlet surface, the pressure value has been imposed. The fluid density and the kinematic viscosity have been set to 860 kg m^{-3} and to 29 mm^2 sec^{-1} respectively.

Result analysis: (1) Effects of the area ratio A_v/A: Fig. 3 shows the contours of pressure and velocity vectors with different A_v/A.

From the above Fig. 3, the followings can be obtained:

- The pressure distribution near the notch has changed obviously as the value of A_v/A increases; the position of the lowest pressure point transfers from the spool side wall 1 to the valve body and the value of it changes; the following Fig. 4 shows the relationship between the lowest pressure values and the ratio of A_v/A; further refined calculations had been carried out, we found that the lowest pressure reaches the highest value when the area ratio is equals to 0.5, which means the anti-cavitation performance is best at this area ratio
- With the increase of the A_v/A, the pressure drop of the valve increases; so is the throttling loss; the notch throttling transfers to the combined throttling of the notch and the annual space after the notch; according to the flow path inside the valve, the notch and the annual space after the notch can be regarded as two throttling elements in series; the notch can be treated as thin-wall orifice and the annual space can be treated as elongated hole; the pressure drop through the valve is taken

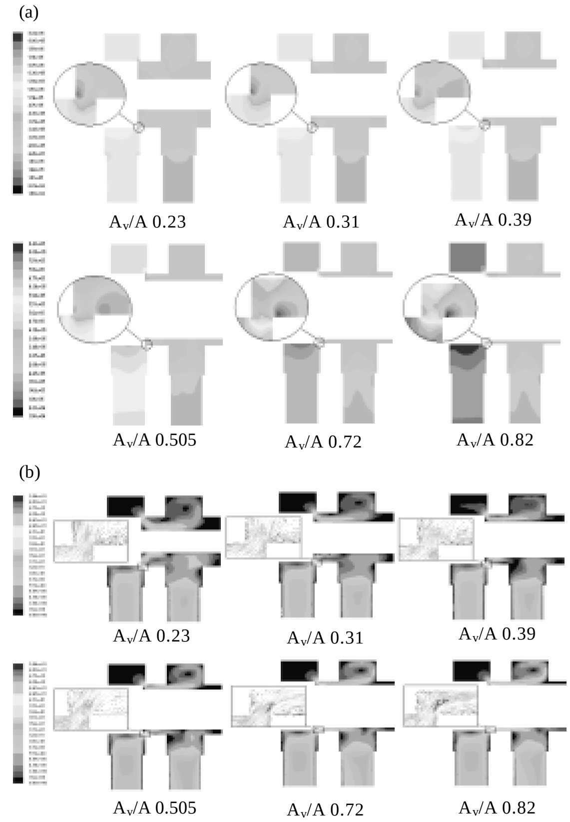

Fig. 3(a-b): Pressure contours and velocity contours, (a) Pressure contours with local enlargement and (b)Velocity contours with local enlargement of velocity vector

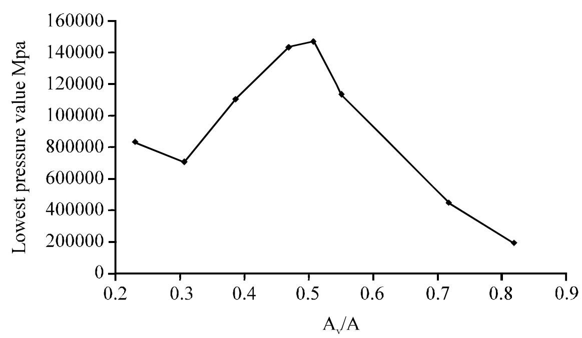

Fig. 4: Relationship between the lowest pressures and area ratio

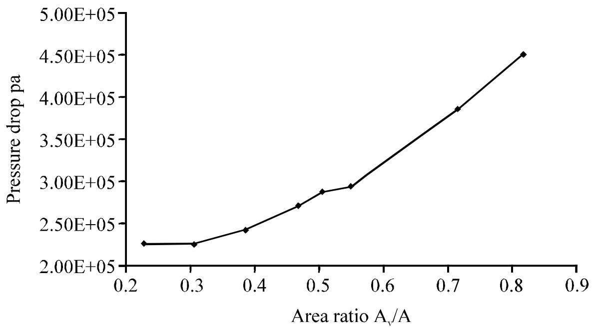

Fig. 5: Pressure drop of the valve

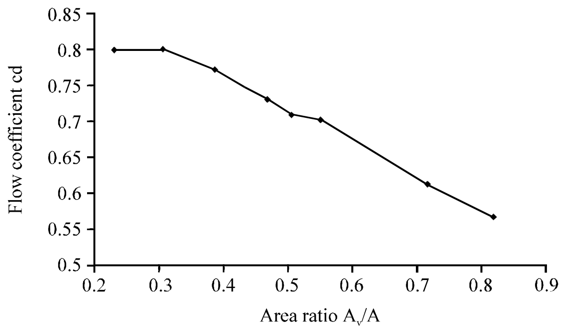

Fig. 6: Flow coefficient of the valve

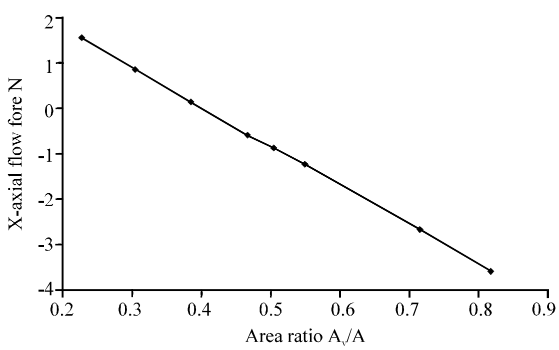

Fig. 7: X-axial flow force varying with the area ratio

as the pressure difference between the inlet and outlet. The following Fig. 5-6 show the changes of the pressure drop and the flow coefficient of the valve; when the area ratio is less than 0.31 there are no changes because the throttling effect of the annual space can be negligible

- X-axial flow force on the spool can be obtained by integral of the pressure on the side wall; the magnitude and orientation of the flow force change with the area ratio A_v/A shown in Fig. 7; the positive value represents the flow force tends to close the notch and the negative value represents the flow force tends to open the notch; the flow force and the area ratio is linear relation

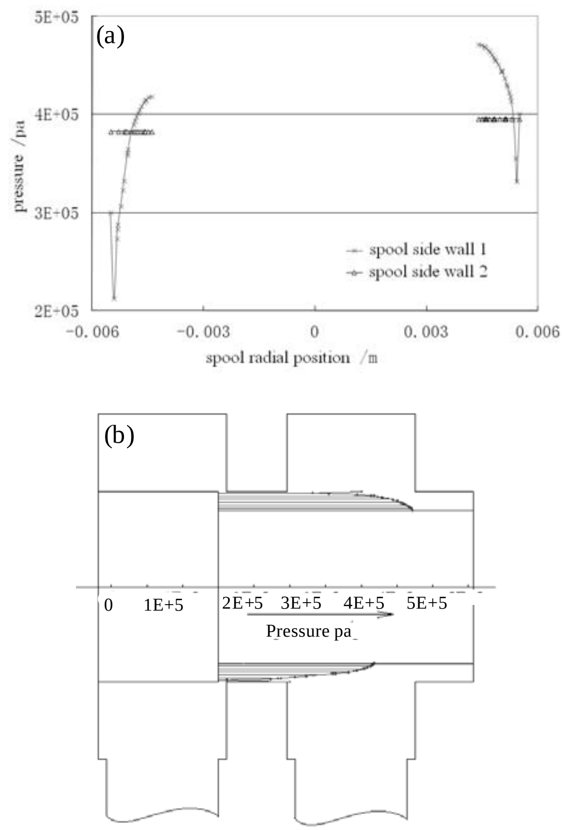

Fig. 8(a-b): Pressure distribution on the side walls of the spool, (a) Pressure distribution on side wall 1 and 2, (b) Pressure distribution on side wall 1

According to the momentum equation of steady incompressible flow: the force action on the fluid of the control volume is equal to the net outflow momentum of the control volume per unit time; this force and the flow force on the spool are equal and opposite. Along with the area ratio increase, the annual space throttling effect increases; the X-axial component of the outflow momentum of the control volume (surrounded by the dashed line in Fig. 1) increases gradually. When the area ratio increases to about 0.4, the X-axial component of the net outflow moment is close to zero and so is the spool flow force; when the area ratio increases continuously, the X-axial component of the net outflow moment increases in opposite direction, so the spool flow force becomes negative.

By drawing the pressure distribution on the spool side walls (Fig. 8), we found that the point of application

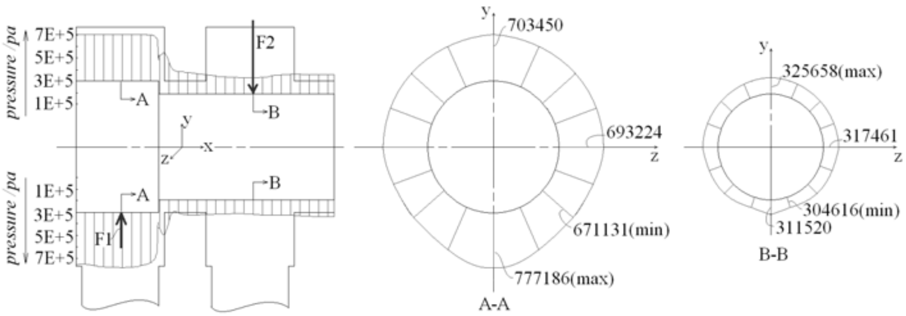

Fig. 9: X-axial and radial pressure distributions on the spool

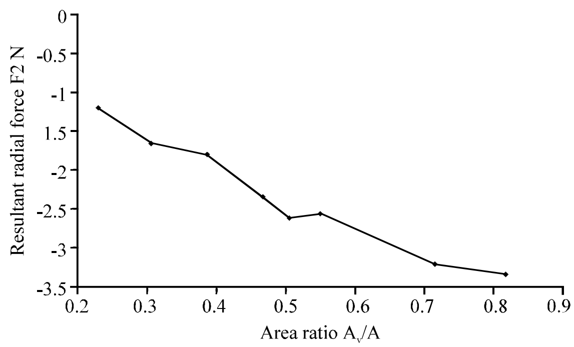

Fig. 10: Relationship between F2 and the area ratio

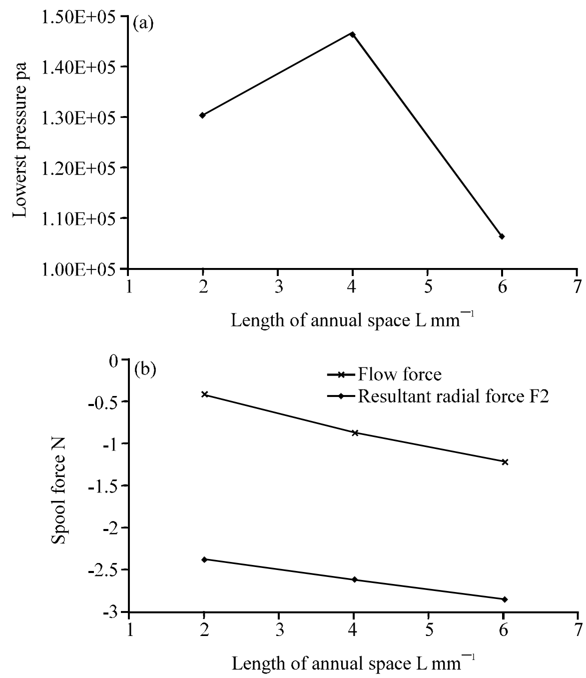

Fig. 11(a-b): Lowest pressure value and forces on the spool relation to the length, (a) Lowest pressure value and (b) Forces on the spool

of the flow force is not the center of the spool; so a rotating moment generates for the spool which will cause the radial jam of the spool; the moment tries to rotate the spool clockwise with small area ratio and counterclockwise with large area ratio:

- Because the flow field layout inside the valve is asymmetric in radial, the axial and radial pressure on the spool surface distribute asymmetrically; for the details please see Fig. 9

The pressure near the notch steeply decreases; the pressure distributions on the spool protruding and stem are symmetric with respect to the Y-axial but asymmetric with respect to the Z-axial. Resultant radial forces F1 and F2(also called radial jam forces) can be obtained by integral of the pressures on the spool surface which are 0.42N and -2.6N separately; positive value means in Y-direction; negative value means in opposite. The forces F1 and F2 cause the axis of the spool rotate clockwise, which leads to radial jam of the spool; the magnitudes of F1 and F2 vary with the area ratio. The following Fig. 10 shows the relationship between F2 and the area ratio.

Effects of the length (L) of the annual space: Three lengths were selected for the simulation; the lowest pressure value near the notch, the spool flow force and radial jam force F2 are shown in the Fig. 11.

Along with the increase of the length L, the values of the flow force and the radial force F2 increase; the value of the lowest pressure with 4 mm length of the annual space is the highest which means the anti-cavitation performance of the valve with this length is the best; The flow coefficient of the valve decreases with the length L increase, which is fully consistent with the theoretical analysis.

Effects of the layout of the inlet and outlet passages: So far, the analyses are mainly based on the same side inlet and outlet passages; however, in practice, the inlet and outlet passages are arranged on opposite sides; two different layouts for the inlet and outlet passage, i.e. same side and opposite sides, are analyzed in this paper. We

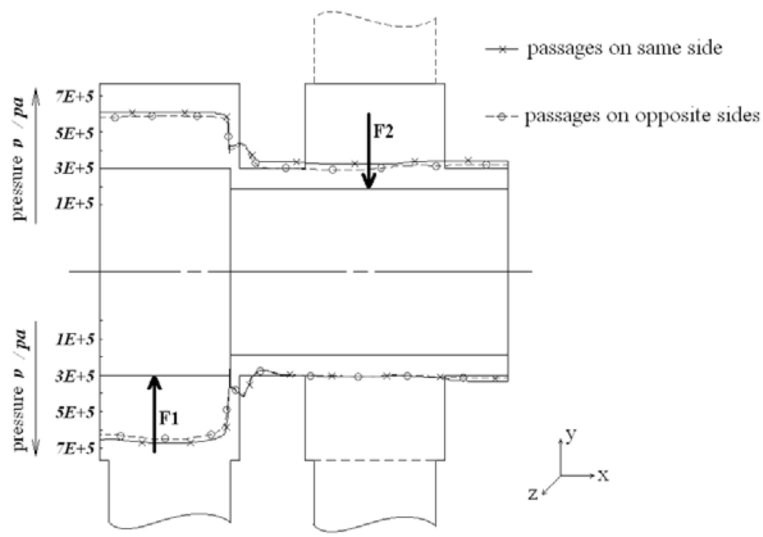

Fig. 12: Comparison of the axial pressure distributions

found that: the layout of the inlet and outlet passage affects the spool flow force and the lowest pressure value near the notch little and affects radial jam forces a lot. The radial jam forces F1 and F2 decrease from 0.42N and -2.6N to 0.24N and -0.19N when same side layout of the inlet and outlet passage changes to opposite layout. The following Fig. 12 shows the comparison of the axial pressure distributions.

CONCLUSION

By means of CFD simulation for the flow field inside the spool valve, detailed effects analyses have been carried out on flow force and anti-cavitation performance of the valve. The followings are concluded:

- The position of the lowest pressure point transfers from the spool side wall 1 to the valve body as the value of A_v/A increases; the lowest pressure reaches the highest value when the area ratio is equals to 0.5, which means the anti-cavitation performance is best at this area ratio
- The spool flow force varies with the area ratio; When $A_v/A<0.4$, the flow force tends to close the notch; when $A_v/A>0.4$, the flow force changes direction and tends to open the notch; the flow force and the area ratio is linear relation; Because the flow field layout inside the valve is asymmetric, the point of application of the flow force is not the center of the spool, the resultant moment on the spool generates which cause radial jam of the spool; the moment will changes direction as the area ratio increase
- The radial jam forces F1 and F2 form a clockwise rotation moment which intensifies the radial jam of the spool
- The effects of the length(L) of the annual space after the notch are similar to the area ratio; there is a length(L) with which the anti-cavitation performance of the valve is the best
- The layout of the inlet and outlet passages affects the spool flow force and the lowest pressure value near the notch little; the radial jam forces F1and F2 decrease a lot with opposite inlet and outlet passages

REFERENCES

Amirante, R., G. Del Vescovo and A. Lippolis, 2006a. Evaluation of the flow forces on an open centre directional control valve by means of a computational fluid dynamic analysis. Energy Conv. Manage., 47: 1748-1760.

Amirante, R., G. Del Vescovo and A. Lippolis, 2006b. Flow forces analysis of an open center hydraulic directional control valve sliding spool. Energy Conv. Manage., 47: 114-131.

Du, X., J. Zou, X. Fu, H. Ji and H.Y. Yang, 2007. Effect of throttling grooves structure on cavitation noise. J. Zhejiang Univ., 41: 456-460.

Fang, W.M., L.L. Cheng, X. Fu, H. Zhou and H.Y. Yang, 2010. Investigation on steady-state flow force of spool valve with U-grooves. J. Zhejiang Univ., 44: 574-580.

Ji, H., X. Fu and H.Y. Yang, 2002. Effects of internal structure on cavitating noise of hydraulic relief valve. Chinese J. Mech. Eng., 38: 19-22.

Ji, H., X. Fu and H.Y. Yang, 2003. Study on steady flow force. Chinese J. Mech. Eng., 39: 13-17.

Lei, H. and L. Quan, 2006. Three-dimensional numerical simulation and visual analysis of the F low field in a hydraulic poppet valve. Mech. Sci. Technol., 25: 426-429.

Li, Z., 2006. Hydraulic Element and System. 2nd Edn., China Machine Press, China, Pages: 159.

Liao, Y., Y. Liu, Y. Huang and Z. Li, 2003. Experimental research on anti-cavitation characteristics of two-step throttle in water hydraulic system. Fluidmachinery, 31: 1-3.

Stares, J.A. and K.W. Roth, 2001. Avoid control valve application problems with physics-based models. Hydrocarbon Proc.

Timo, L., T.K. Kari and V. Matti, 2003. CFD-modelling of a water hydraulic poppet valve-comparison of different modeling parameters. Proceedings of 8th Scandinavian International Conference on fluid Power, May 7-9, 2003, Tampere, Finland, pp: 277-286.

Wang, A.L., X.F. Wu and C.L. Zhou, 2010. Multidisciplinary Optimization of a hydraulic slide valve based on Cfd. J. Shanghai Jiaotong Univ., 44: 1767-1772.

Wang, L., M. Zhang and Z. Fang, 2000. Effects on the flow force of flow passage inside the valve. Mach. Tool Hydraulics, 6: 27-28.

Weber, S.T., D.N. Johnston and K.A. Edge, 2001. Flow visualization of a load control valve. Proceedings of the 7th Scandinavian International Conference on Fluid Power, May 30-June 1, 2001, Lynkoping, Sweden, pp: 87-102.

Zhao, L., Q. Chen and L. Quan, 2008. Visual ization analysis of the flow field in a moving spool valve. Trans. Chinese Soc. Agric. Mach., 39: 142-145.

Zhu, Y., 2012. Numerical simulation of flow field and dynamic characteristics of pilot operated directional valve. J. Harbin Institute Technol., 44: 133-139.