Generation Mechanism of Flow Force Acting on Spool Valve

Fumio SHIMIZU¹, Kazuhiro TANAKA¹

¹ Kyushu Institute of Technology (Kawazu 680-4 Iizuka Fukuoka, Japan) shimizu@phys.kyutech.ac.jp

Received January 20, 2022 / Accepted August 15, 2022

In hydraulic systems, spool valves play an important role. When the spool moves to the left or right, the flow path in the control valve changes, and the flow rate of hydraulic oil is controlled by the valve opening. Since high-pressure fluid flows into the valve, it is necessary to predict the behavior of the spool with high accuracy. One of the problems that spool valves have is the hydraulic vibration caused by the high-pressure fluid. There is a need for a control valve that can suppress the hydraulic vibration even if it is used stand-alone. The purpose of this study is to visualize the flow field in the spool valve using three-dimensional computational analysis, and to clarify the generation mechanism of the flow force acting on the spool. The flow force acting on the spool was caused by the pressure difference between the high-pressure region generated by the impact of the jet and the low-pressure region through which the high-speed jet passed. Fluctuation of the jet flow occurred when the valve opening became narrow. The unsteady fluctuation of the jet flow had a significant effect on the unsteady variation of the flow forces.

Keywords: Spool valve, Flow force, Generation mechanism, Validation, Jet flow

1. Introduction

Hydraulic circuits are widely used in many large machines, such as transportation and construction machinery. The elements of a hydraulic circuit can be classified into five categories: tanks, pumps, piping, hydraulic control valves, and actuators. The most important of these elements is the hydraulic control valve, which is responsible for controlling the hydraulic circuit system. One of the typical hydraulic control valves is a spool valve, in which a small spool is installed inside the valve. When the spool moves to the left or right, the flow path inside the valve changes, and the flow rate of hydraulic oil is controlled by the valve opening. A small spool, about one centimeter in diameter, controls the operation of a hydraulic circuit with high internal pressure of several MPa. Therefore, the behavior of the spool needs to be predicted with high accuracy. One of the problems with the use of spool valves is the hydraulic vibration caused by the high-pressure fluid. A combination of multiple valves is commonly used to dampen hydraulic vibration, but this method has the disadvantage of increasing the size of the hydraulic system. Recently, there is a need to reduce the overall size of the hydraulic machine, and a stand-alone control valve that can suppress hydraulic vibration is required. Therefore, it is necessary to understand the generation mechanism of the flow force acting on the spool, which is the cause of hydraulic vibration, and to study the suppression method of the flow force.

Many researchers have reported the flow fields and flow forces in a spool valve using three-dimensional computational fluid dynamics (CFD). Kondo et al. investigated flow pattern and flow forces for a spool valve with basic geometry. Lisowski et al. compared with the numerical and experimental results for a 3D spool valve with near real geometry, Niko et al. and Amiratnte et al. also reported 3D spool valves with different shapes. Chen et al. reported CFD analysis of a spool valve with a more complex geometry. Most of these studies have focused on the axial flow force acting steady state on the spool, and the flow force in perpendicular to the axial direction has not been well studied. Few reports have focused on unsteady flow forces acting on the spool. No analysis has been found that investigates the effect of the narrow clearance between the spool and the sleeve.

The purpose of the present study is to visualize the flow field inside a spool valve using CFD simulation and to clarify the generation mechanism of flow forces acting on the spool in two directions. The unsteady fluctuation inside the valve is investigated, and a numerical analysis method is developed to account for the narrow clearance between the spool and the sleeve. The final goal of the present research is to identify the causes of unsteady flow forces and to propose methods to suppress the hydraulic vibration.

2. Nomenclature

| Symbol | Description |
|--------|-------------|
| A      | opening area = πDS₁ |
| D      | diameter of the thick shaft |
| p      | pressure |
| Q      | volume flow rate |
| Re     | Reynolds number = Q/νπD |
| S₁     | valve opening on the inlet side |
| S₂     | valve opening on the outlet side |
| t      | time |
| u      | mean velocity at the throat = Q/A |
| U      | velocity vectors |
| x      | displacement |
| X,Y,Z  | Cartesian coordinate |
| μ      | viscosity |
| ν      | kinematic viscosity |
| ρ      | density |

3. Computational Methods

In the present study, numerical analysis was conducted using ANSYS CFX, that was thermal and fluid analysis software. CFX is a commercial software that is widely used around the world. The finite volume method is used as a discretization method, which provides excellent robustness and fluid conservations in numerical calculations. The governing equations are shown below.
\[
\n\nabla \cdot \mathbf{U} = 0 \quad (1)
\]

\[
\frac{\partial \mathbf{U}}{\partial t} + \nabla \cdot (\mathbf{U} \times \mathbf{U}) = -\frac{1}{\rho} \nabla p + \mu \nabla^2 \mathbf{U} \quad (2)
\]

Eq. (1) is the continuity equation for the conservation of mass, and Eq. (2) is the Navier-Stokes equations for the conservation of momentum.

4. Research Target and Computational Conditions

In the present study, a two-port spool valve with a basic structure was chosen as the research target. The schematic figure of the spool valve is shown in Fig. 1. Fig. 1 (a) shows the structure and dimensions of the control valve. For the spool valve, one port was connected to the inlet side and another one to the outlet side. The cross-section of both ports was square configuration, with a side length of ten millimeters on the inlet and eight millimeters on the outlet. There was a spool with large and small diameter shafts, the diameter size was 16 and 8 millimeters, respectively. The flow rate through the inside of the valve was controlled by the movement of the spool. Expressing the valve opening on the inlet side as “S1” and the valve opening on the outlet side as “S2”, the following relationship was obtained,

\[
S_1 + S_2 = 10 \text{ mm.} \quad (3)
\]

There is an inlet throat (S1 ≤ 5.0 mm) that narrowed the flow path on the inlet side, and an outlet throat (S2 ≤ 5.0 mm) that narrowed the flow path on the outlet side. In the present paper, numerical analysis was conducted focusing on the inlet throat (S1).

Fig. 1 (b) shows the computational domain from bird’s-eye view. The computational domain was created by enclosing area of the dashed lines in Fig. 1 (a). The diameter of the thick shaft of the spool was D = 16 mm, and this length was used as the reference length. The lengths of the inlet and outlet ports were set to 3D for both. In numerical calculations, the spool was treated as having a fixed position. Five types of computational grids with different valve openings S1 were created. The eccentricity and tilt of the spool were not considered. Each computational grid had a total number of about 2 million grid points.

Table 1 summarizes the computational conditions used in the numerical calculations. To explore the possibility of the flow fluctuates with time, an unsteady analysis was performed. The time increments were set so that the CFL condition would be about 10. Oil was used as the working fluid, and the density and kinematic viscosity of the oil were given. The volume flow rate on the inlet was 20 L/min. The Reynolds number using the inlet throat and mean velocity at the throat was defined as \(Re = \frac{Q}{\nu D}\). In the present study, the Reynolds number was 214. The Shear Stress Transport model was used for the turbulence model. Numerical calculations were carried out for five different channel geometries with different inlet throat S1.

The flow force acting on the spool can be decomposed into an axial force acting in the direction of the spool axis (Z-axis) and a lateral force acting in the radial (Y-axis) direction. Kondo et al. experimentally measured two types of flow forces acting on a spool\textsuperscript{4}. A three-dimensional numerical analysis was also performed using an original numerical code. In the present paper, we compared the experimental and numerical results of Kondo et al. with the computational results obtained by our calculations and confirmed the validity of the present computational method.
5. Computational Results and Discussions

5.1 Flow force acting on spool

Fig. 2 shows the results of the present calculations compared with the experimental and numerical results of Kondo et al. Fig. 2 (a) shows the results of the axial force acting on the spool for five kinds of valve opening $S_1$. Fig. 2 (b) shows the results of the lateral force. The plotted marks in the figures were all time-averaged values, and the error bars indicated the time variability range of the computational results. In Fig. 2 (a), the axial force was almost zero for $S_1 \geq 1.0$ mm. However, as the inlet throat became narrow, the axial force increased rapidly. The monotonic increase in axial force with decreasing inlet throat was quantitatively good agreement with all three results. The error bars were expanded when the throat was small, so it suggested that time variability occurred. For the lateral forces, present results were compared with the numerical results of Kondo et al.

In Fig. 2 (b), the value of lateral force was generally smaller than that of axial force. As the throat length decreases, the lateral force tended to increase and then decrease. When the throat length was narrow, unsteady fluctuations appeared in the present results. A comparison of the time-averaged values showed good agreement with the results of Kondo et al., the present numerical analysis is considered to validity.

5.2 Flow pattern inside valve

To investigate the generation mechanism of axial and lateral forces, the relationship between the flow pattern inside the valve and the pressure distribution on the spool surface was examined. Fig. 3 shows the instantaneous streamlines and the pressure distribution on the spool surface. The case of $S_1 = 0.2$ mm is shown. By observing the streamlines, the flow inside the valve can be explained as follows.

(A) The hydraulic fluid that passed through the inlet throat flowed along the end face of the large-diameter shaft of the spool.

(B) After that, the hydraulic fluid collided with the small-diameter shaft on the inlet side.

(C) The hydraulic fluid traveled downstream along the thin shaft.

(D) The hydraulic fluid collided with the end face of the thick shaft on the outlet side.

(E) The hydraulic fluid flowed out to the outlet port with forming some vortex.

Fig. 3 (b) shows an enlarged view near the inlet throat region. The jet flowed in vigorously. The outer circumference parts of the thick shaft were at the root of the jet, which were a low-pressure region due to the fast velocity of the jet. On the thin shaft, high-pressure region appeared because of the collision of the jet. The pressure difference between low- and high-pressure regions created the flow force.

5.3 Generation mechanism of axial force

To clarify the mechanism of axial force generation, the flow field...
inside the valve and the pressure distribution acting on the end faces of
the thick shaft were investigated. The results of \( S_1 = 0.5 \text{ mm} \) are shown
in Fig. 4. The velocity vectors in the YZ section and the pressure
distribution at the two end faces of the thick shaft are shown. As \( S_1 \)
changed, the direction and velocity of the jet from the inlet throat
changed and the pressure on the spool surface changed. However, the
basic structure of the flow field inside the valve and the pressure
distribution on the spool surface remained the same. The result of
\( S_1 = 0.5 \text{ mm} \) was easily visible the pressure difference on the left and
right surfaces. Therefore, this result is used for the explain. On the left
side surface, a low-pressure region was extended by the jet blow. In
particular, the pressure was extremely low at the outer circumference
part, which was just fast jet area. On the other hand, the right-side end
face of the thick shaft was a high-pressure region because the flow
along the thin shaft collided with the end face. It was found that the
axial force was generated by the pressure difference acting on the left
and right end faces.

5.4 Generation mechanism of lateral force

To clarify the mechanism of lateral force generation, the velocity
vectors inside the valve and the pressure distribution acting on the
spool surface were investigated. The results for \( S_1 = 0.2 \text{ mm} \) are
shown in Fig. 5. Fig. 5 (a) shows the velocity vectors in the XY cross
section at the inlet port side. Fig. 5 (b) shows the velocity vectors in

the YZ cross section at the center plane. Fig. 5 (c) shows the pressure
distribution on the spool surface from the views of top and bottom

---

**Fig. 4 (a) Velocity vectors plot**

**Fig. 4 (b) Pressure distribution on the end faces**

**Fig. 4 Velocity vectors plot and pressure distribution for \( S_1 = 0.5 \text{ mm} \)**

**Fig. 5 (a) Velocity vectors plot in XY cross section**

**Fig. 5 (b) Velocity vectors plot in YZ cross section**

**Fig. 5 Velocity vectors plot and pressure distribution for \( S_1 = 0.2 \text{ mm} \)**

**Fig. 5 (c) Pressure distribution on thick and thin shafts**
As can be seen in Fig. 5 (a), the downward flow from the inlet port flowed around the thick shaft and filled the valve chamber of the inlet side with fluid. In this case, high velocity region was formed near the inlet port. As the static pressure decreased due to the increase in dynamic pressure, a slightly lower pressure region was created at the side of the inlet port. This can be seen from the pressure distribution in the top view of Fig. 5 (c). On the other hand, the flow velocity was widely slow in the bottom region of the thick shaft. Therefore, the pressure distribution on the bottom surface of the thick shaft was uniformly high. Since the lateral force acting on the thick shaft was caused by the pressure difference between the upper and lower surfaces, we can see that it was acting in +Y direction.

Next, we focus on the flow field around the thin shaft as shown in Fig. 5 (b). The jet from the inlet throat impinged the thin shaft and fast flow was formed along the narrow spool axis. Part of the velocity vectors in the lower region flowed out toward the outlet port. On the top view of the thin shaft, the low-pressure region was spread out over a wide area, except for the area of the jet impacts. On the other hand, on the bottom view of the thin shaft, several high-pressure regions were scattered and spread out. Therefore, it can be seen that the lateral force on the thin shaft also acted in the +Y direction.

5.5 Small clearance between spool and sleeve

A numerical analysis was performed considering small clearance between the spool and the sleeve\(^{13}\). The clearance width was set to 19 micrometers and inlet throat was set to \(S_1 = 0.5\) mm. To confirm that the flow inside the clearance is properly captured, the pressure distributions inside the valve chamber at \(X = 0\) cross section and on the spool surface were plotted and shown in Fig. 6. Complex pressure changes were observed inside the valve chamber and on the thin shaft. On both sides of the thick shaft, gradual pressure change was observed. The pressure was gradually decreasing toward the clearance exit. On the left side of the spool, the pressure decreased linearly in the axial direction and no pressure change occurred in the lateral direction. Since the pressure inside the valve chamber was high and the clearance exit was set to atmospheric pressure, the pressure decrease toward the clearance outlet was correctly captured. The velocity vectors in the small clearance were also checked, and the flow toward the clearance exit was observed on both sides. We would like to consider the quantitative variation of the flow force acting on the spool with the small clearance in the future work.

5.6 Unsteadiness

The time variations of the flow field were investigated for \(S_1 = 0.2\) and \(S_1 = 0.5\) mm. The velocity distributions at three different times in the YZ section are shown in Fig. 7. The high-pressure area on the spool surface was indicated with solid triangle marks, and the low-pressure area was indicated with blank triangle marks. First, we focused on \(S_1 = 0.2\) mm. The hydraulic fluid that flowed in from the inlet throat became a jet flow and flowed along the end face of the thick shaft and the thin shaft body. The flow formed some vortex near the end face of the thick shaft on the outlet side. Although the jet shape at the upper side remained almost unchanged with time, the jet shape at the lower side changed its direction significantly with time. The lower jet collided with the thin shaft or not collided the shaft. Next, we focused on the pressure distribution acting on the spool. The end face of the thick shaft showed a low-pressure region on the inlet side (indicated by the blank marks), while the outlet side showed a high-pressure region. As a result, there was the pressure difference acting on the left and right surfaces, which acted on an axial force in the +Z (right) direction. The axial force did not change much over time. On the other hand, the pressure on the surface of the small-diameter shaft showed time variation depending on the presence or absence of the jet impact. When the vortex formed on the outlet side collided with the thin shaft, it was observed that a high-pressure region was also formed on the outlet side. The pressure distribution on the thin shaft surface changed significantly with time. Therefore, it was confirmed that unsteady changes in the lateral force occurred.

When \(S_1 = 0.5\) mm, the obliquely oriented jet impacted the thin shaft. It was also observed that the flow along the thin shaft changed the direction with time. However, the change was slight, indicating that the unsteady variation was reduced.

Since the spool position is fixed in the present study, the spool vibration cannot be reproduced. However, spool vibration may occur in actual spool valves due to the action of unsteady flow forces. To investigate the state of spool vibration using CFD, it is necessary to perform fluid/structure coupled analysis that takes spool movement into account.

6. Concluding Remarks

A two-ports spool valve was numerically investigated using three-dimensional computational fluid dynamics. Flow forces acting on the spool in two directions were investigated. As the results of investigating the magnitude of the flow forces against the change in inlet valve opening, the results of the present calculations were in good agreement with the results of the previous study. The present calculations were confirmed to be valid. It was found that the flow force acting on the spool was caused by the pressure difference between the high-pressure region generated by the impact of the jet and the low-pressure region through which the high-speed jet passed. Fluctuation of the jet flow occurred when the valve opening became narrow. The high-pressure region changed with time due to the time variation in the jet flow direction. It was found that the unsteady fluctuation of the jet flow had a significant effect on the unsteady variation of the flow forces. Numerical analysis was also performed considering the narrow clearance between the spool and the sleeve, and it was confirmed that reasonable results were obtained.
Acknowledgements

We would like to express our gratitude to Mr. Saiki Tanaka for his great cooperation in the present research.

References

1) Lee, S.Y. and Blackburn, J.F.: Steady-State Axial Forces on Control-Valve Pistons, Transactions of the American Society of Mechanical Engineers, Vol. 76, pp. 1005-1011 (1952)
2) Kumagaya, H. and Masaki, K.: Your First Hydraulic System (in Japanese), Gijutsu-Hyohon Co., Ltd., pp. 100-153 (2009)
3) Ichikawa, T. and Hibi, A.: Oil-Hydraulic Engineering (in Japanese), Asakura Publishing, pp. 88-118 (1979)
4) Kondoh, Y., et al.: Analysis of Flow Forces Acting on a Spool Valve (1st Report, Non-Uniformity of Flow Pattern and Momentum Flux in the Azimuthal Direction) (in Japanese), Transactions of the Japan Society of Mechanical Engineers, Series B, Vol. 65, No. 639, pp. 3577-3585 (1999)
5) Lisowski, E., Czyżycki, W. and Rajda, J.: Three Dimensional CFD Analysis and Experimental Test of Flow Force Acting on the Spool of Solenoid Operated Directional Control Valve, Energy Conversion and Management 70, pp. 220-229 (2013), http://dx.doi.org/10.1016/j.enconman.2013.02.016
6) Lisowski, E. and Rajda, J.: CFD Analysis of Flow Forces Acting on the Spool of Directional Control Valve Type WE10J, Engineering, Biology, pp. 133-140 (2015)
7) Herakovic, N., Duhovnik, J. and Simic, M.: CFD Simulation of Flow Force Reduction in Hydraulic Valves, Technical Gazette, Vol. 22, No. 2, pp. 453-463 (2015)
8) Amirante, R., Del Vescovo, G. and Lippolis, A.: Evaluation of the Flow Forces on an Open Centre Directional Control Valve by Means of a Computational Fluid Dynamic Analysis, Energy Conversion and Management 47, pp. 1748-1760 (2006)
9) Amirante, R., Moscatelli, P.G. and Catalano, L.A.: Evaluation of the Flow Forces on a Direct (Single Stage) Proportional Valve by Means of a Computational Fluid Dynamic Analysis, Energy Conversion and Management 48, pp. 942-953 (2007)
10) Amirante, R., Catalano, L.A. and Tamburrano, P.: The Importance of a Full
3D Fluid Dynamic Analysis to Evaluate the Flow Forces in a Hydraulic Directional Proportional Valve, International Journal for Computer-Aided Engineering and Software, Vol. 31, No. 5, pp. 898-922 (2014)

11) Chen, Q., Ji, H., Zhu, Y. and Yang, X.: Proposal for Optimization of Spool Valve Flow Force Based on the MATLAB-AMESim-FLUENT Joint Simulation Method, IEEE Access, Vol. 6, pp. 33148-33158 (2018)

12) ANSYS Inc.: ANSYS CFX-Solver Modeling Guide Release 13.0, pp. 13-15, pp. 149-152 (2010)

13) Shimizu, F., Ueki, M. and Tanaka, K.: Consideration of Axial Fluid Force Acting on Spool Valve with Small Clearance (in Japanese), Proceedings of 2016 JFPS Autumn Conference, pp. 35-37 (2016)