한국동력기계공학회지 제19권 제2호 pp. 22-26 2015년 4월 ISSN 1226-7813(Print) ISSN 2384-1354(Online)
Journal of the Korean Society for Power System Engineering http://dx.doi.org/10.9726/kspse.2015.19.2.022
Vol. 19, No. 2, pp. 22-26, April 2015

Study of the Fluid Flow on Proportional Valve in Spool Displacement using CFD

Kui-Ming Li*, Yoon-Hwan Choi**, Ill-Yeong Lee***, Yeon-Won Lee*†

(Received 05 November 2014, Revision received 04 April 2015, Accepted 06 April 2015)

Abstract: The main objective of this work is to estimate the fluid flow of a proportional valve. The study is based on the classical compressible flow theory and the computations with the help of CFD based commercial software - ANSYS CFX. The fluid flow with the movement of spool along the sleeve is simulated. To change the spool moving from 0.4mm to 2.0mm, the moving mesh method with different condition of orifice is considered here. The results show that it is the highest at the 80 % (1.6mm) opening and at the 20 % (0.4mm) opening, is the lowest.

Key Words : CFD, Servo Valve, Spool Displacement, Sleeve, Flow rate, Pressure Drop

1. Introduction

A hydraulic servo-valve plays an important role in hydraulic control system. Its characteristics significantly influence the performance of the whole system. Servo-valves are used to control the position, velocity, or force of an actuator. Electro-hydraulic servo-drives are widely used in industrial applications like machine tools, testing equipment and autonomous manufacturing systems. It is a device that takes an electrical current and turns it into hydraulic flow which can then create linear, rotational, uni-directional or reciprocating mechanical motion[1]. Many articles regarding various aspects of servo-valve have appeared alongside their development[2,3].

Recently, the improvement of computer performances and advances in CFD enables further investigations of the fluid dynamic phenomena into the valve[4-6]. Deep insights of the flow behavior inside hydraulic devices such as turbulence, cavitation, particle paths, and velocity or pressure distributions have become more and more relevant in the attempts to improve their hydraulic characteristics[7]. Together with this, the development of the computational calculations using numerical methods has made its way to solving complex flows.

In this study, the main objective is to calculate the fluid flow in a servo valve. The study is based on classical incompressible flow theory using the numerical program - ANSYS CFX. The movement of spool along the sleeve is simulated. Here we focus on two main factors, namely flow rate and flow force. Also, the relationship between pressure drop and mass flow rate is investigated. To realize

*† Yeon Won Lee, Department of Mechanical Design Engineering, Center for Marine-Integrated Biomedical Technology, Pukyong National University.
E-mail: ywlee@pknu.ac.kr, Tel:051-629-6162
*Kui-Ming Li, Energy Marine Research Division, Korea Marine Equipment Research Institute
**Yoon-Hwan Choi, Department of Mechanical Design Engineering, Pukyong National University.
***Ill-Yeong Lee, Department of Mechanical Design Engineering, Pukyong National University.

the spool movement, the moving mesh method was used, where the spool movement was from 0.4mm mm to 2.0mm.

2. Analysis Model

A servo-valve consists of a polarized electrical torque motor and two stages of hydraulic power amplification. The motor armature extends into the air gaps of the magnetic flux circuit and is supported in this position by a flexure tube member. The flexure tube acts as a seal between the electromagnetic and hydraulic sections of the valve. The two motor coils surround the armature, one on each side of the flexure tube. The flapper of the first stage hydraulic amplifier is rigidly attached to the midpoint of the armature. The flapper extends through the flexure tube and passes between two nozzles, creating two variable orifices between the nozzle tips and the flapper. The pressure controlled by the flapper and nozzle variable orifice is fed to the end areas of the second stage spool. The second stage is a conventional 4-way spool design in which output flow from the valve, at a fixed valve pressure drop, is proportional to spool displacement from the null position. A cantilever feedback spring is fixed to the flapper and engages a slot at the centre of the spool. Displacement of the spool deflects the feedback spring which creates a force on the armature/flapper assembly. Input signal induces a magnetic charge in the armature and causes a deflection of the armature and flapper. This assembly pivots about the flexure tube and increases the size of one nozzle orifice and decreases the size of the other. This action creates a differential pressure from one end of the spool to the other and results in spool displacement. The spool displacement transmits a force in the feedback wire which opposes the original input signal torque. Spool movement continues until the feedback wire force equals the input signal force[8].

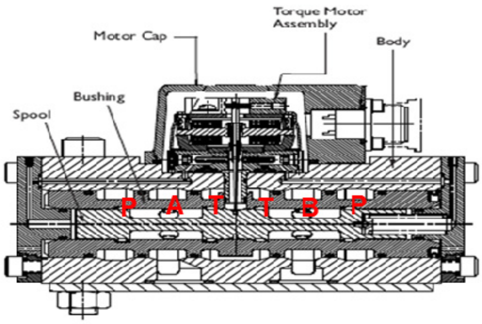


Fig. 1 Electrohydraulic Servo valve Cut-Away [8]

Fig. 1 shows the modelled fluid domain of a servo-valve. In order to visualize the working process of valve, we have modelled the connection between port A and port B. If the spool move to right side, the working fluid enter from port P pass through orifice, and then out from port A, after working, come back to the port B, then the fluid pass from orifice out from port T. On the contrary, if the spool moves to the left side, the working fluid enter from port P, pass from B, then arrived port A to T port. Since the movement of the spool on both sides are the same, we consider only one sided movement, in this case, the right side and ignore the movement on the left side.

3. Numerical Analysis

The compressibility of fluid is considered here with change of pressure by the following equation,

$$\rho = \frac{\rho_0 (V_{l0} + V_{g0})}{V_{l0} \exp(-\frac{P - P_0}{K_l}) + V_{g0} \left(\frac{P_0}{P} \right)^{1/n}} \tag{1}$$

Where ρ_0 = $855kg/m^3$, V_{g0} = 0.1% is the gas volume percentage inside the working fluid, V_{l0} = 99.9% is the water volume percentage. $P_0 = 10^5 Pa$ the atmospheric pressure, P is fluid pressure. $K_l = 1.6\times10^4 Pa$, and $n = 1.4$ is Poly Tropic Index.

Fig. 2 indicates the model of the calculation domain. It is assumed that the inlet condition is 70bar with different conditions as shown in Table 1. Outlet condition is 17.5bar. In this study, we use the steady state simulation. The shear–stress transport (SST) model is used for the turbulence modeling. The SST model unifies the advantages of the most widely employed two-equation (k-ω and k-ε) models and is the most reliable model for fluids with flow separation. And the wall boundary conditions are set to no-slip wall.

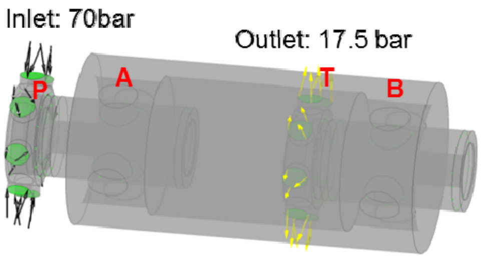


Fig. 2 Servo valve Boundary condition

Table 1 Flow conditions

Inlet	70bar	70bar	70bar	70bar	70bar
Outlet	17.5bar	17.5bar	17.5bar	17.5bar	17.5bar
Opening	0.4mm	0.8mm	1.2mm	1.6mm	2.0mm
Value	(20%)	(40%)	(60%)	(80%)	(100%)

4. Result and discussion

Fig. 3 illustrates the streamline for the case of orifice at 20% (0.4mm) opening. It can be seen that the high pressure fluid flow enter from port of inlet P, and through from A port to port B smoothly, then flows out from port of outlet T in low speed.

Fig. 4 explains the absolute pressure comparison at different orifice opening condition. It can be seen that pressure increases with the increased of the opening of orifice, especially near the left orifice. On the contrary, pressure decrease with increase of the opening of orifice in right.

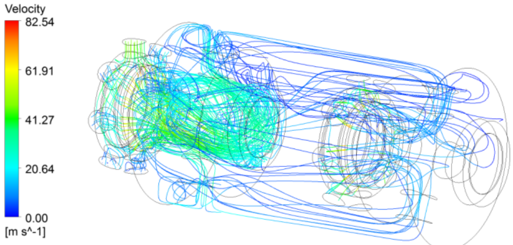
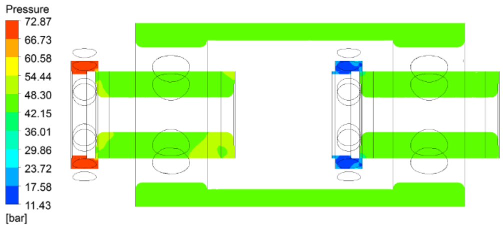
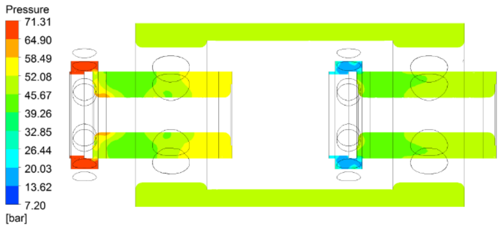


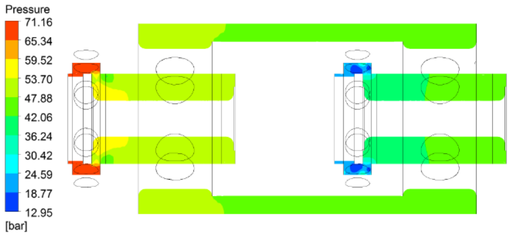
Fig. 3 Streamline in orifice 20% (0.4mm) opening along x axis



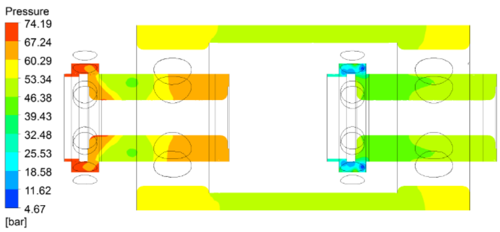
20% Opening



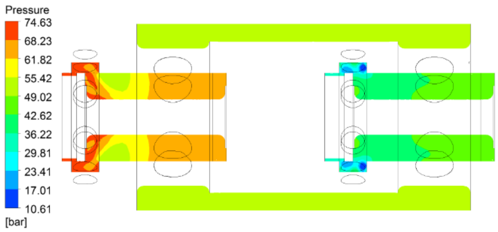
40% Opening



60% Opening



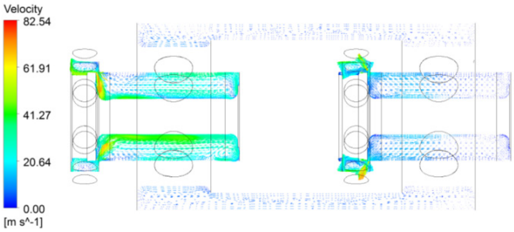
80% Opening



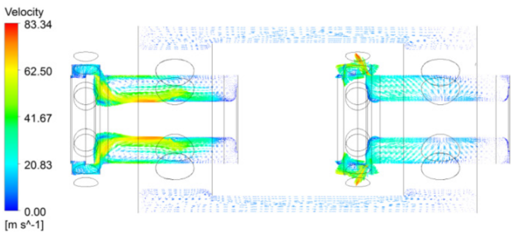
100% Opening

Fig. 4 Pressure on middle section in different orifice opening along x axis

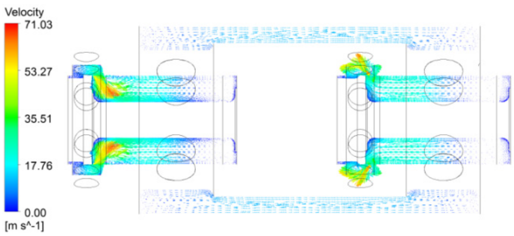
Fig. 5 describes the velocity vector at different orifice openings. It can be seen that in the case of 20% opening, the velocity near left orifice is much higher than the others, and incident angle of velocity with vertical plane is very small when compared with the other cases. With the increase of orifice opening, incident angle becomes larger. The area of high velocity is far away from orifice.



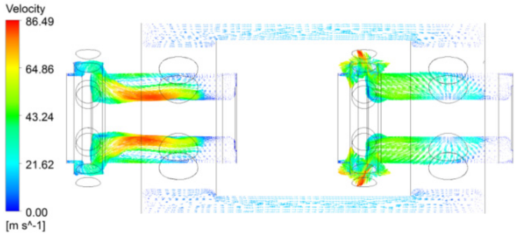
20% Opening



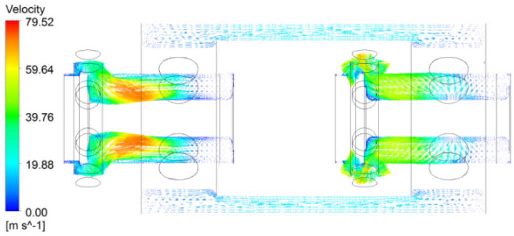
40% Opening



60% Opening



80% Opening



100% Opening

Fig. 5 Velocity on middle section in different orifice opening along x axis

The force in spool along the x direction has been studied. Fig. 6 shows the force along x direction. The total force in both spool is the highest in 80% (1.6mm) opening, where the value is approximately -100N, and the total force in both spool is lowest in 20% (0.4mm) opening, where the value is -40N.

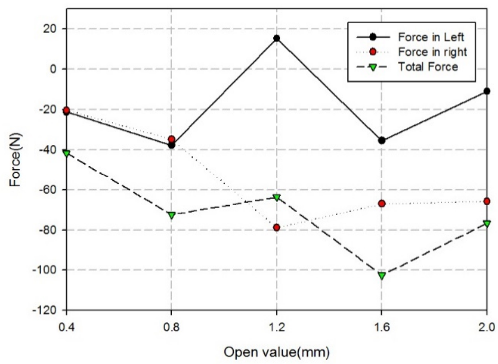


Fig. 6 The total force in different orifice opening along x axis

5. Conclusions

The fluid motion within the servo-valve domain of high pressure by CFD has been studied in the present paper. The velocity vector, pressure, and force of different conditions of orifice opening, such as 20%, 40%, 60%, 80%, 100% are investigated separately and finally compare the results among them. And the force in 80% (1.6mm) opening is the highest. It is approximately 100N, and the force is the lowest in 20% (0.4mm) opening, value being 40N.

References

1. R. Poley, 2005, “DSP Control of Electro-Hydraulic Servo Actuators”, Application Report.
2. R. Amirante, D. Vescovo and A. Lippolis, 2006, “Flow forces analysis of an open centre hydraulic directional control valve sliding spool”, Energy Convers Manage, Vol. 47, No. 1, pp. 114-31.
3. R. Amirante, D. Vescovo and A. Lippolis, 2006, “Evaluation of the flow forces on an open centre directional control valve by means of a computational fluid dynamics analysis”, Energy Convers Manage, Vol. 47, pp. 1748-1760.
4. T. J. Chung, 2002, “Computational Fluid Dynamics”, Cambridge University Press, Cambridge.
5. R. Amirante, PG. Moscatelli and LA. Cartalano, 2007, “Evaluation of the flow forces on a direct (single stage) proportional valve by means of a computational fluid dynamics analysis”, Energy Convers Manage, Vol. 48, pp. 942-53.
6. J. C. Renn and T. C. Kao, 2003, “Application of CFD to design a power-saving hydraulic directional two-land-four-way-valve”, In: Proceedings of the 1st international conference on computational methods in fluid power technology, pp. 26-28.
7. H. K. Versteeg and W. Malalasekera, 1995, “An Introduction to Computational Fluid Dynamics. The Finite Volume Method”, Longman, England.
8. 72 Series installation and Operation Instruction, Moog-Inc, www.moog.com.