ISSN 1453 – 7303 **"HIDRAULICA" (No. 3/2015)**
Magazine *of Hydraulics, Pneumatics, Tribology, Ecology, Sensorics, Mechatronics*

Flow through Command Hydraulic Resistance

Dipl. Eng. **Ioana SFARLEA[1],** PhD. Eng. **Lucian MARCU[1],** PhD. Eng. **Daniel BANYAI[1]**

[1] Technical University of Cluj-Napoca, sfarlea.ioana@termo.utcluj.ro

Abstract: *The growing usage of hydraulic servo technique involves optimizing the elementary components of these types of devices. The paper presents the results of a study on hydraulic resistance, achieved by mathematical modeling and numerical simulation, CFD. We have studied three types of bushings and 3 types of piston forming different geometric resistant fed in turn with three different pressure values. There are shown the formation and dissipation of fluid jets that appear in the flow through the variable resistance.*

***Keywords**: CFD, hydraulic resistance, servo-hydraulics*

1. Introduction

Precision hydraulic actuators have the beginnings in 1955, when J.F. Blackburn publish at Massachusetts Institute of Technology the "Fluid Power Control". The importance of this research lies in the fact that the work founded the servo-hydraulics, marking the moment when the conventional hydraulic actuators were developed for servo drives (servo-hydraulic) and then for proportional hydraulics. The difference between proportional hydraulic equipment or servo-hydraulic and classic hydraulics is in the first two cases, by opening a variable hydraulic resistance the flow can be controlled continuously, eventually, the velocity of the fluid in a hydraulic system. This cannot be achieved in conventional hydraulics. The difference between proportional and servo-hydraulic equipment is represented by technological achievements and the coverage of the hydraulic resistance (Fig.1).

As shown in Fig. 1, for zero coverage, the width of the sleeve, t, and the width of the spool, a, are equal, t>a of the negative cover and t <a for positive coverage. The appearance of the coverage is reflected in the variation of the flow rate, according to the opening resistance. Thus, for negative coverage, the middle position is characterized by a flow, Q_0.Positive coverage is characterized by a movement, y_0 for which there is no flow. It is noted that the only situation in which for a reference displacement, y occurs flow and movement of the spool without parasite flow Q_0 if coverage is "0". The usage of zero coverage resistances was limited at first by the technological possibilities for achieving them. This is why the proportional hydraulics technique was used. The years 1970-1990 were characterized by proportional hydraulics research. The technological advances of the last decades have led to the use of systems "servo" research in medical and energy production directions [1],[4],[5].

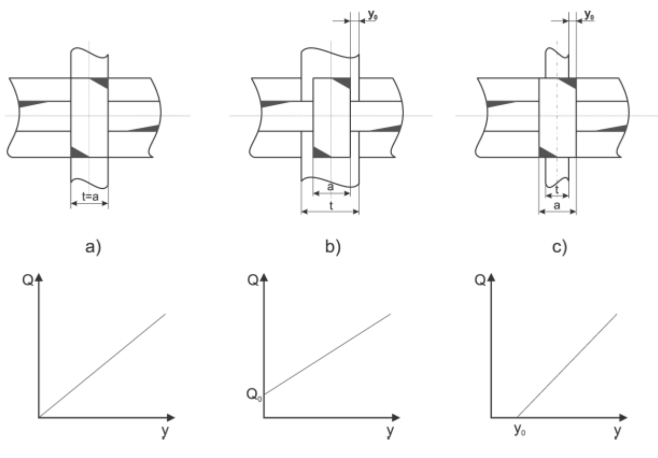


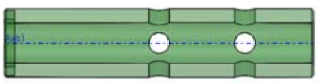
Fig. 1. Flow variation depending on spool displacement a) zero coverage (servo-hydraulics); b) negative coverage (conventional hydraulics); c) positive coverage (proportional hydraulics)

2. Objectives

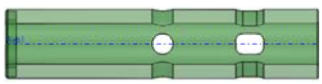
At the base of the manufacture of hydraulic energy control equipments, also called hydraulic control valves are the resistances. Given this and the fact that there are targeted superior static and dynamic performances, there is made an in-depth study and research of all aspects influencing parameters for fluid flow through the hydraulic resistance. The need to know these issues is given primarily by the automation of the industrial process, where hydraulic systems are in big number of applications. Also the predictability of the static and dynamic behaviour of hydraulic power control devices require knowledge of these issues.
This paper proposes to study the static and dynamic behaviour of a hydraulic resistance with spool and sleeve with holes, based on numerical simulations. The numerical simulations conducted will help to understand, visualize and analyze the phenomena occurring, during the fluid flow through hydraulic resistance formed. Thus, they are traced and can be interpreted the fields and gradients of pressure and velocity. It will be analyzed the formation of fluid flow and of vortices, where the situation will be such.
The program involves geometric modeling numerical simulation of hydraulic resistance sleeve with holes and spool design using SolidWorks software and then was used the module Fluent, ANSYS.
The study objective is to optimize the sleeve's holes geometry, so for small openings command to have turbulent regime, the hydrodynamic forces as smaller as possible, and, where possible, to avoid the appearance of cavitation. Numerical analysis of CFD (Computational Fluid Dynamics) helps us in this regard by viewing flow and phenomena that occur during it without actually manufacture the resistance.

3. Mathematical modeling

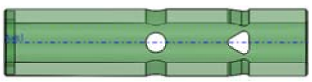
To study the flow through hydraulic resistance spool and sleeve with holes were designed several CAD models. A body has been modeled, three types of sleeve holes (circular, rectangular and triangular), and three patterns of piston (control edge bevel at 30°, 45° or 60°), there were made various combinations of these, to study the flow through them. The inlet pressure was considered 20 bar, 30 bar, 35 bar and 40 bar, the control openings of 0.1 mm, 0.2 mm, 0.3 mm, 0.4 mm and 0.5 mm. Was chose, small openings control, because in this area the effects of hydrodynamic force disrupts the dynamic behaviour but also because of the potential of cavitation due to pressure drop.
The simulations that will be presented below are made for the following combinations: circular holes and piston 30º, 45º or 60º,rectangular holes and piston 30º, 45º or 60º and triangular holes and piston 30º, 45º, 60º respectively at all openings and input pressures considered above.
The geometric models were designed using SolidWorks software and the simulations being carried with Fluent, ANSYS. The steps that were followed are: Defining the computational domain (representing the physical space), Defining the hydraulic oil used, Meshing the fluid domain. Figures 2 and 3 show the models of sleeve or piston used.



a) cylindrical holes



b) rectangular openings



c) triangular openings

Fig. 2. Sleeves used

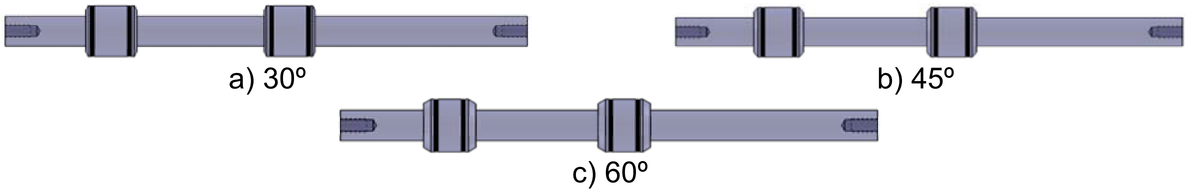


Fig. 3. Pistons

From the imposed boundary conditions point of view, which have been implemented only by the values of the input or the output of the fluid. In this case, the input pressures were 20e5 Pa, Pa 30e5, 35e5 Pa, Pa 40e5 and 101325 Pa output pressure (atmospheric pressure); was considered a hydraulic fluid density ρ_{oil} = 876 kg/m^3 and kinematic viscosity v = 45cSt.
Turbulent flow is characterized by the variations in the speed fields, which mix the transported sizes, such as momentum and energy. The turbulence models help to simulate these fluctuations. The turbulence model chose for this study was, k-ε RNG (Renormalization Group). This model of turbulence arising from model k-ε, bringing the latter improved by including a term in addition to the equation ε, which gives a significant increase in precision and by implementing the theory of renormalization, which lets us to get satisfactory results if there are small Reynolds numbers [2].
In addition to the turbulence equations proposed, for the accurate modeling of flow the mathematical model contains also the continuity of mass equations, the Navier-Stokes equations (conservation of momentum) and the equation of speed in any point of the fluid.

$$\frac{\partial x}{\partial t}+\frac{\partial y}{\partial t}+\frac{\partial z}{\partial t}=0$$

$$\begin{cases} \dfrac{\partial v_x}{\partial t}+v_x\dfrac{\partial v_x}{\partial x}v_y\dfrac{\partial v_x}{\partial y}+v_z\dfrac{\partial v_x}{\partial z}=f_x-\dfrac{1}{\rho}\dfrac{\partial p}{\partial x} \\ \dfrac{\partial v_y}{\partial t}+v_x\dfrac{\partial v_y}{\partial x}+v_y\dfrac{\partial v_y}{\partial y}+v_z\dfrac{\partial v_y}{\partial z}=f_y-\dfrac{1}{\rho}\dfrac{\partial p}{\partial y} \\ \dfrac{\partial v_z}{\partial t}+v_x\dfrac{\partial v_z}{\partial x}+v_y\dfrac{\partial v_z}{\partial y}+v_z\dfrac{\partial v_z}{\partial z}=f_z-\dfrac{1}{\rho}\dfrac{\partial p}{\partial z} \end{cases} \qquad (1)$$

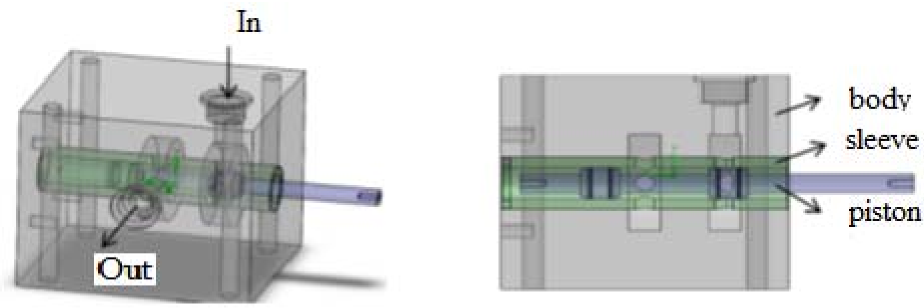
$$v=v_0 * tgh\left(\frac{t}{T}\right)$$

$$v=v_0\left(\frac{r}{R}\right)^{\frac{1}{7}}$$

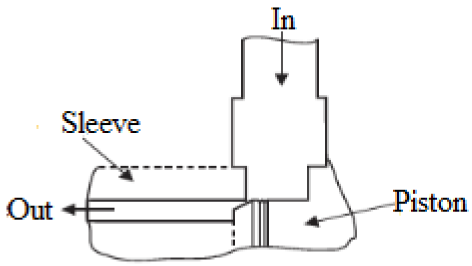
where: v_0 the velocity of fluid at input;
$v_{x,y,z}$ – the velocity about Ox, Oy and Oz.

4. Numerical analysis

Regarding the flow regime, investigations start from the need of a turbulent regime, a flow coefficient (α) constant with a value as close to 1, it includes the value all hydraulic resistance. It is known that the proportionality between flow and the opening control is ensured only if both the pressure difference and flow coefficient are constant [3].
Flow coefficient, α, is constant only in turbulent flow regime. It will not depend on Re, but will be heavily influenced by the pressure difference. It is necessary to avoid laminar regime, as it depends on the temperature and thus the viscosity of the oil [3].
In the following part will be presented numerical simulation of the studied models. The fields are shown in a longitudinal section through the center of interest. Figure 4 is intended to highlight the fields studied.



a) b)

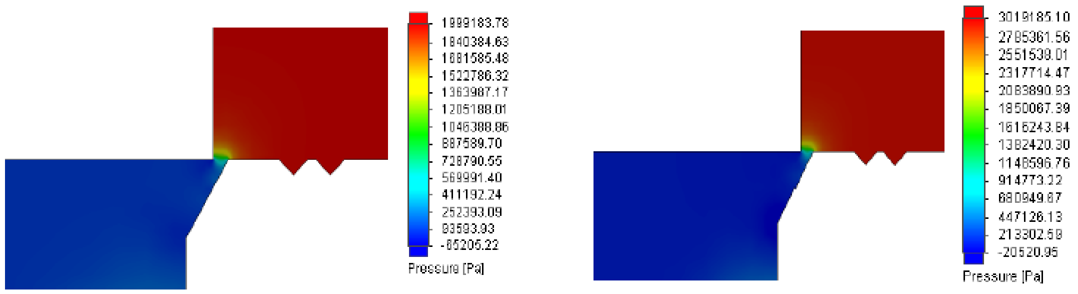


c)

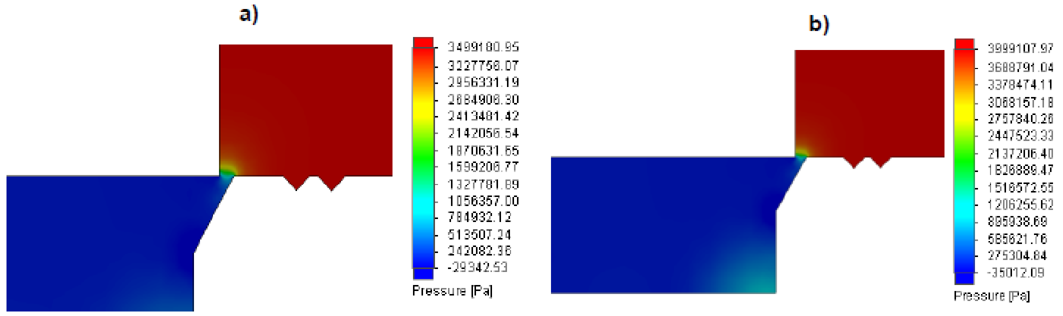
a) Overview b) Longitudinal section through the middle of the aperture c) Section for displaying the results

Fig. 4. Hydraulic resistance assembly

Fig. 5 shows the distribution of the pressure inside the hydraulic resistance that is formed in the whole body, sleeve with rectangular holes and piston with the control edge chamfered at 30°. Representations are available for opening 0.2 mm at various inlet pressures.



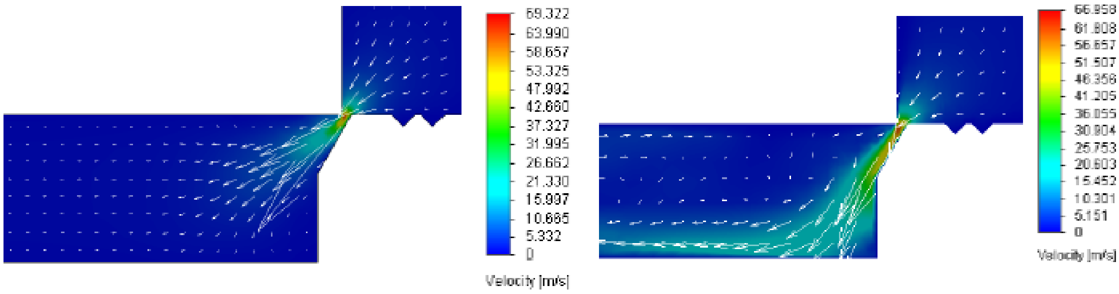
a) b)



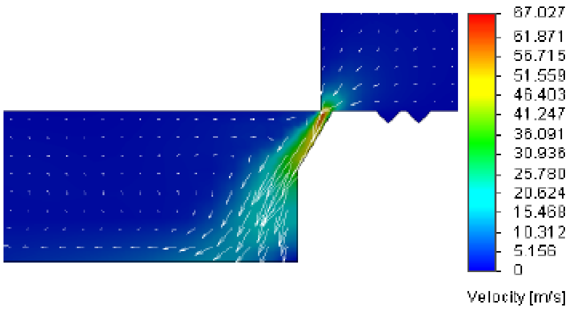
c) d)

Fig. 5. Pressure field (rectangular holes, and the active edge chamfered at 30 °. Inlet pressure: a) 20 bar; b) 30 bar; c) 35 bar; d) 40 bar.

In Fig. 6 is shown the speed distribution of the assembling with three sleeves having holes with a control edge chamfered at 30°. Opening command is 0.2 mm and the inlet pressure is 20 bar. The highest values were recorded for the combination of the sleeve with cylindrical apertures, about 69 m/s (Fig. 5a), then assembling the sleeve with triangular openings, about 67 m/s (Fig. 5c), and the low speed values occurred at the rectangular holes, 66 m/s (Fig. 5b). It is noted that the values are close but very different the form of the jet that cross the considered resistance. As if a) the jet is dispersed almost immediately after passing the resistance, in the other two situations the jet is "sticking" to the walls of the spool, much more when the sleeve have rectangular holes. Also in the latter two cases, recirculation zone is observed, which leads to slower the oil flow.



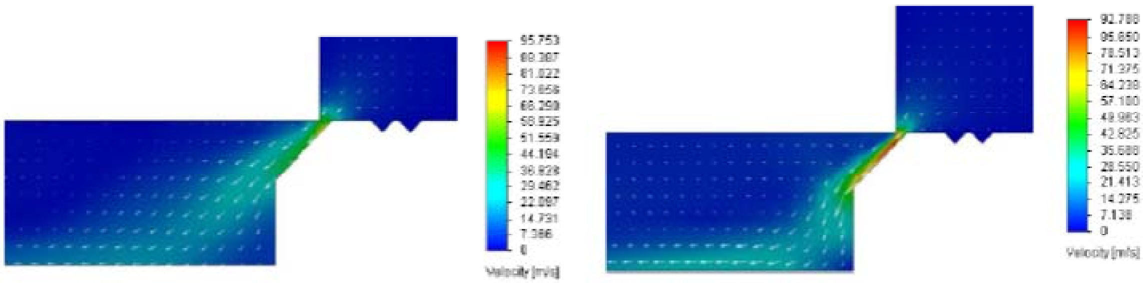
a) Circular holes b) Rectangular holes



c) Triangular holes

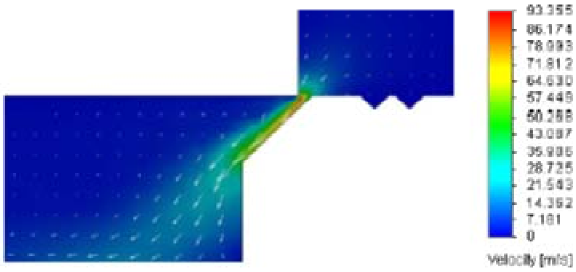
Fig. 6. Velocity distribution for 30^{o} chamfer

The velocity field is compared if inlet pressure is 40 bar and active edge sloping at 45°; the conclusions drawn from the analysis of all results obtained by combining together the 3 pistons 3 bushings, with a pressure of 20, 30, 25, 40 bar, for a small opening of 0.2 mm are presented in the following.



a) Circular holes b) Rectangular holes

Fig. 7. Velocity distribution for 45^{o} chamfer



c) Triangular holes

Fig. 7. Velocity distribution for 45^o chamfer (continued)

5. Conclusions

As seen in the pressure field in each case there is a sudden pressure drop in the minimum section, reaching negative values. This is an alarm in the occurrence of cavitation phenomenon.
As the value of the inlet pressure increases, it is evident that the velocity of the oil will increase the value. Regarding the speed values recorded, the results provided by simulations is not much different between them, regardless of the geometry of the hole.
According to the orientation of the velocity vector and the jet shape, it can be seen that in the case of circular apertures the fluid jet is dispersed almost uniformly on the surface of the sleeve and spool, when leaving the beveled edge. In the other two cases, the fluid stream tends to be closer to the edge of the control spool; in the assembly of sleeve with rectangular apertures, the fluid jet is observed as it is joined to both the sleeve and the plunger. This aspect allows the formation of vortices in the upper part of the enclosure.
In the case of the sleeves with circular and triangular holes, the fluid jet is sticking by the control edge of the hydraulic resistance. When the jet reaches the stem of the spool, it begins to fall apart. For the case of the sleeve with rectangular apertures, the jet of fluid is attached to both: the control edge and its rod. It also appears, a fluid recirculation due to the meeting with the second control section of the spool, eddies areas where fluid decelerates.
For chamfered edge at a greater angle, the oil speed increases through resistance which leads to a decrease in static pressure in the minimum section pass.

References

[1] C. Dumitrescu, C. Cristescu, I. Nita, G. Matache, I. Ilie, "Considerations regarding the use of hydraulic and pneumatic trackers for photovoltaic panels to convert solar energy directly into electric energy", Proceedings of International Multidisciplinary Scientific Geo-Conference SGEM-2013, 2013, Albena Co., Bulgaria, Vol.: "Energy and clean technologies", pp. 77 – 84, ISBN 978-619-7105-03-2, ISSN 1314-2704;
[2] C.H. Shin, "A numerical study on the characteristics of transient flow in a pressure regulator resulting from closure of the pressure control valve", Journal of Mechanical Science and Technology, 02/2013; 27(2). DOI: 10.1007/s12206-012-1257-y;
[3] D. Opruta, L. Vaida, H. Hedesiu, "The influence of the geometric configuration of the hydraulic command resistances upon the cavitational phenomena", Proceedings of First National Conference on Recent Advances in Mechanical Engineering, September 17-20, 2001, University of Patras, Greece, Volume: 1;
[4] G. Matache, P. Drumea, M. Comes, I. Ilie, "Echipament proportional de reglare a presiunii", Hidraulica Magazine no. 1 /2005 – ISSN 1453-7303, pp.76-79;
[5] L. Vaida, L. Nascutiu, D. Potolea, C. Vaida, D. Opruta, "Techniques for the reduction of noise and vibrations for axial piston pumps", Conference: Experimental Fluid Mechanics 2006, EMT06, Liberec.