

JANUSZ RAJDA*, EDWARD LISOWSKI**

FLOW FORCES ACTING ON THE SPOOL OF DIRECTIONAL CONTROL VALVE

ODDZIAŁYWANIE PRZEPLYWAJĄCEJ STRUGI NA SUWAK ROZDZIELACZA HYDRAULICZNEGO

Abstract

This paper presents a method with the use of CFD analysis for calculating the forces acting on the spool during the flow of working liquid stream through the electro-hydraulic pilot operated directional control valve. The liquid stream flowing through the directional valve causes hydrodynamic response according to the principle of conservation of momentum. Liquid pressure acting on the spool surface affects the forces balance of valve spool as well. An object of the research is the pilot operated directional control valve WEH22 type with flow rate up to 450 dm³/min. For the analysis ANSYS/Fluent software was used.

Keywords: CFD modelling, directional control valve, Fluent/Ansys

Streszczenie

W artykule przedstawiono metodę obliczania sił powstających na skutek oddziaływania strumienia przepływającej cieczy na suwak rozdzielacza z wykorzystaniem analizy CFD. Obiektem badań jest rozdzielacz hydrauliczny sterowany pośrednio elektrohydraulicznie. Struga przepływająca przez rozdzielacz hydrauliczny zgodnie z zasadą zachowania pędu wywołuje reakcję hydrodynamiczną. Również ciśnienie cieczy, działając na powierzchnię suwaka wpływa na bilans sił działających na suwak rozdzielacza. W artykule przedstawiono analizę sił związanych z przepływem dla rozdzielacza hydraulicznego typu WEH22 dla przepływu w zakresie do 450 dm³/min. Do analizy zastosowano program ANSYS/FLUENT.

Słowa kluczowe: modelowanie CFD, rozdzielacz hydrauliczny, Fluent/Ansys

* MSc. Janusz Rajda, Ponar Wadowice.

** Prof. DSc. PhD. Edward Lisowski, Institute of Applied Informatics, Cracow University of Technology.

1. Introduction

Development of hydraulic machine drive is connected with the development of its individual elements. The main aim of development of hydraulic components is to increase their parameters, including the range of flow and working pressure [1, 4]. Increasing the flow and pressure result in an increase of forces acting on the valve spool. It may happen that at high flow rate, the increase of flow forces affects adversely the total balance of forces acting on the spool and thus interferes with the correct operation of the valve [4, 5]. This paper presents a method for calculating the forces connected with the flow of working liquid with the use of ANSYS/Fluent software.

2. Object of the research

An object of the research is a popular type of hydraulic directional control valve WEH22 produced by several manufacturers [7]. It is designated for hydraulic systems in which pressure does not exceed 35 MPa, and flow rate is up to 450 dm³/min. Design of the directional control valve WEH22 type is shown in figure 1. The valve consists of the main body (1) with the valve spool (2) and the pilot valve (4). The pilot valve is controlled by the solenoids (5) with manual override buttons (6). When one of electromagnets is energized the pilot valve causes the spool (2) in the main body to move to the extreme position and it allows the flow from port P to A and from B to T (or P to B and A to T). The paper presents the analysis of flow from P to A (P-A) and B to T ports (B-T) as representing the basic flow path for this directional valve.

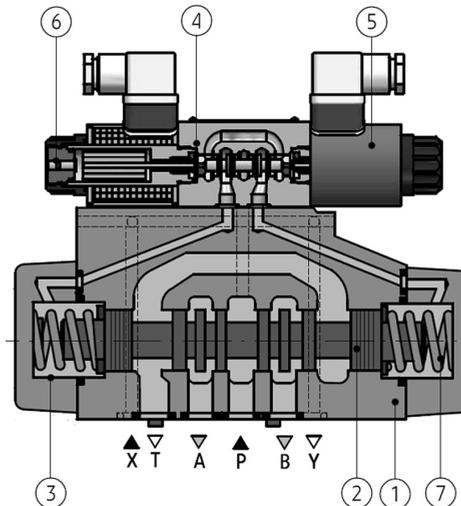


Fig. 1. Design of directional control valve WEH22J type
Rys. 1. Budowa rozdzielacza hydraulicznego typu WEH22J

3. Discrete model of flow paths

3D models of the body and spool created in Solid Edge for technical documentation are used to built stream geometry. For this purpose, 3D models of the body and spool are loaded to the CREO software. The surfaces are copied from these 3D models and irrelevant components of non-geometric nature of the channel surface are remove. Then solids are created using the CREO Solidify command. For these components the logical operations are performed which give as a result geometric model of the streams. FEM mesh and flow models are made using ANSYS/Fluent software. The individual flow path have a different geometric structure and different flow paths within the valve body. Figure 2 shows the discrete model of P-A flow path and in figure 3 model of B-T path. These models are made using the advanced features of the grid compaction. In the areas of complex geometry the grid is properly compacted. The model for P-A path includes 358 800 cells and 121 081 nodes and for B-T path 1 917 405 cells and 414 778 nodes.

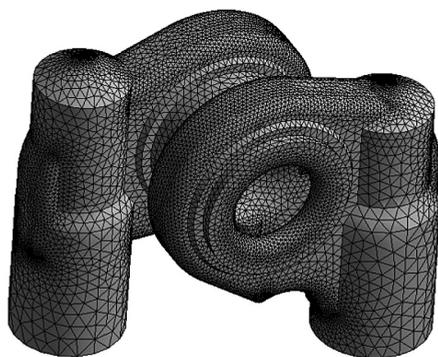


Fig. 2. Mesh of P-A flow path

Rys. 2. Siatka MES drogi przepływowej P-A

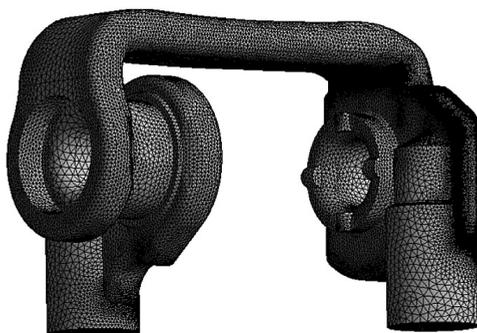


Fig. 3. Mesh of B-T flow path

Rys. 3. Siatka MES drogi przepływowej B-T

4. CFD model

In order to perform the analysis flow kind must be determined. Due to the fact that at the directional control valve there are no conditions for the formation of laminar flow, turbulent flow pattern was assumed to simulate flow. The ANSYS/FLUENT [6] program allows to choose from a variety of models available including: k -epsilon, K-Omega and Reynolds. In the case of flow through the directional control valve k -epsilon model works well enough due to the fact that in the valve there are no conditions for the formation of laminar flow [2, 3]. Thus, for the simulation study turbulence model k -epsilon was chosen. The kinetic energy of the turbulence and dissipation factor are computed from the following equations [1]:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left(\alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right) + G_k + G_b - \rho \varepsilon - Y_M \quad (2)$$

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

In these equations G_k represents the increase in the kinetic energy of turbulence caused by gradient of average velocities. G_b is the energy generated by the phenomenon of buoyancy. Y_m is the energy associated with the compressibility of liquids, $C_{1\varepsilon}$, $C_{2\varepsilon}$ and $C_{3\varepsilon}$ are constants of the model, s_k , s_ε are Prandtl's numbers, respectively.

Turbulent viscosity, m_t , is calculated as follows:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (4)$$

where model constants [1]: $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$, $C_m = 0.09$, $s_k = 1.0$, $s_\varepsilon = 1.3$.

To carry out the calculation in ANSYS Fluent program after generating grid, boundary conditions in *Setup* module of the program must be defined. In the option *Boundary Conditions* as inlet velocity magnitude normal to boundary is assumed. Value Velocity magnitude is introduced as input parameter to *Workbench*. As outlet condition outlet pressure is taken. Value k and $epsilon$ are introduced as intensity and length scale. Using model *Results* of ANSYS Fluent program, with the command *Calculators* and *Expressions* to *Workbench* there are introduced as parameters F_x , F_y and F_z forces acting on the valve spool. In the module *Expressions* the following variables specified by *Calculators* are defined:

- force_x()@spool – force in X axial direction (axis along port),
- force_y()@spool – force in Y axial direction (axis along the spool),
- force_z()@spool – force in Z axial direction (axis normal to port),
- ave(Pressure)@a_inlet – average pressure at the inlet to the flow path.

5. Results of calculations

The P-A flow path

The results of the CFD analysis provide information about the values of hydraulic forces, as well as the distribution of flow velocity and pressure in the analysed flow paths. Figure 4

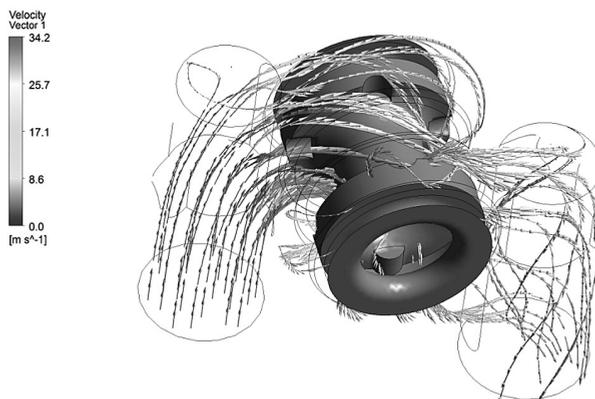


Fig. 4. Velocity vectors on the stream lines for P-A flow path

Rys. 4. Wektory prędkości na liniach prądu dla drogi przepływowej P-A

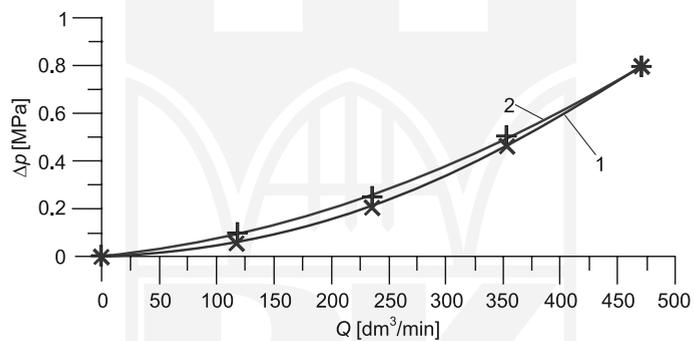


Fig. 5. Pressure loss for flow path P-A

Rys. 5. Straty ciśnienia dla drogi przepływowej P-A

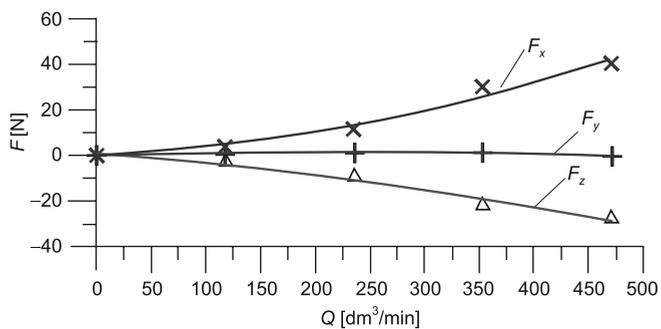


Fig. 6. F_x, F_y, F_z forces acting on the spool for P-A flow path

Rys. 6. Siły F_x, F_y, F_z działające na suwak dla przepływowej P-A

shows distribution of the velocity on streamline acting on the walls for $Q = 450$ [dm³/min]. for P-A flow path and the spool surface which the stream flows around. Figure 5 – curve 1 – shows the pressure drop during the flow while Figure 6 shows the relation of forces F_x , F_y , F_z acting on the spool. Figure 5 – curve 2 – shows the pressure losses took from the manufacturer's catalog [7]. As can be seen from this figure, the catalogue data (received from test bench) are consistent with the ones obtained from the CFD analysis. The following force values are estimated for $Q = 450$ dm³/min.: $F_x = 40$ [N], $F_y = -0.8$ [N], $F_z = -28$ [N]. Thus, F_y axial force obtains the lowest value, and F_z axial force obtains the highest value. This result informs us that the influence of flow is very small on balance force acting on the spool.

B-T flow path

Similarly, the analysis for B-T flow path is conducted. For this flow path Figure 7 shows velocity vectors on the stream lines and an outline of the spool surface flown around by the stream of liquid.

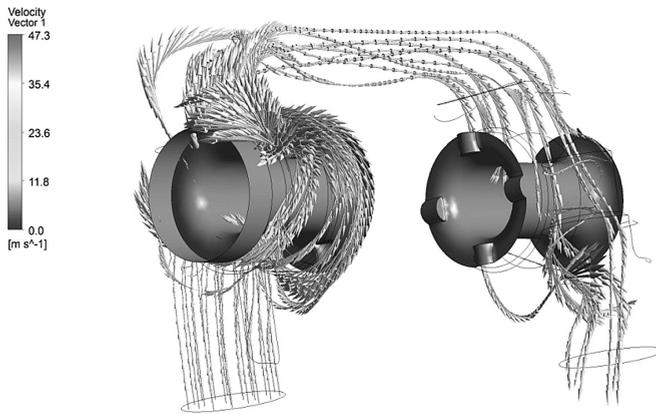


Fig. 7. Velocity vectors on the stream lines for B-T flow path

Rys. 7. Wektory prędkości na liniach prądu dla drogi przepływowej B-T

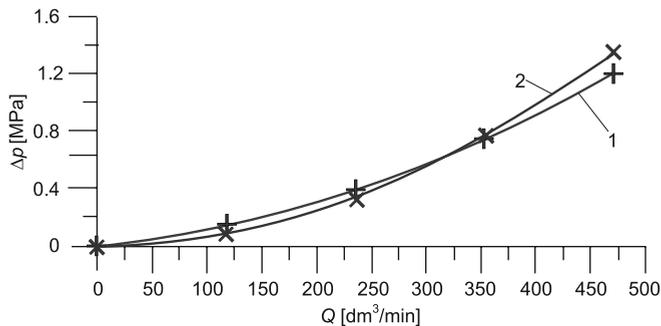


Fig. 8. Pressure losses for B-T flow path

Rys. 8. Straty ciśnienia dla drogi przepływowej B-T

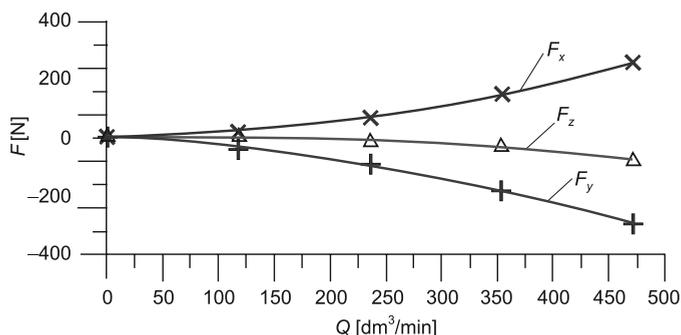


Fig. 9. F_x , F_y , F_z forces acting on the spool for B-T flow path
Rys. 9. Siły F_x , F_y , F_z działające na suwak dla przepływowej B-T

Figure 8 shows the comparison of pressure losses during the flow through the path obtained from CFD calculations and catalogue data. Figure 9 shows the change of F_x , F_y , F_z forces acting on the spool. This flow path is more complicated. Stream of liquid on its way flows the spool around twice and the velocity vectors change its direction several times. The forces which arise at the flow through this path are several times larger than those of P-A path. The following force values are estimated for $Q = 450$ [dm³/min]: $F_x = 252$ [N], $F_y = -99$ [N], $F_z = -83$ [N]. Thus, F_y axial force obtains the highest value, and F_z axial force obtains the lowest value. This result informs us that the influence of flow on balance force acting on spool is very high.

6. Conclusions

The paper considers the issue of determination of the forces acting on the valve spool of directional control valve derived from the liquid stream. The analysis concerns the hydraulic directional control valve of flow range up to 450 [dm³/min]. The article shows that the forces derived from the stream interaction can obtain high values interfering the valve operation. By shaping the geometry of the flow paths, the negative effects can be minimized. With several flow paths in the body, this task may be difficult. Using CFD method research can be conducted to seek compromise solutions.

References

- [1] Launder B.E., Spalding D.B., *The Numerical Computation of Turbulent Flows*. *Computer Methods in Applied Mechanics and Engineering*, no. 3, 1974, 269-289.
- [2] Lisowski E., Domagala M., *Determination of static characteristic of direct acting relief valve*, Proceedings of the 1st international conference on computational methods in fluid power technology, Melbourne 2003, 211-220.
- [3] Lisowski E., Rajda J., *CFD analysis of pressure loss during flow by hydraulic directional control valve constructed from logic valves*, *Energy Conversion and Management* 65, 2013, 285-291.

- [4] Renn J-C., Kao T-C., *Application of CFD to design a power-saving hydraulic directional two-land-four-way-valve*, Proceedings of the 1st international conference on computational methods in fluid power technology, Melbourne 2003, 26-28.
- [5] Del Vescovo G., Lippolis A., *Three-dimensional analysis of flow forces on directional control valves*, International Journal of Fluid Power, 4(2), 2003.
- [6] *ANSYS/Fluent Inc. 13.0 users guide*; 2011.
- [7] Catalogue *Directional spool valve type WEH22 electrically operated*, Ponar Wadowice, WK 491 800 01.2013.

