TECHNICAL TRANSACTIONS | CZASOPISMO TECHNICZNE
MECHANICS | MECHANIKA
2-M/2015

EDWARD LISOWSKI*, JANUSZ RAJDA**

CFD ANALYSIS OF FLOW FORCES ACTING ON THE SPOOL OF DIRECTIONAL CONTROL VALVE TYPE WE10J

ANALIZA CFD SIŁ HYDRAULICZNYCH DZIAŁAJĄCYCH NA SUWAK ROZDZIELACZA TYPU WE10J

A b s t r a c t

The paper presents a CFD analysis of flow forces acting on a spool of directional control valve type WE10J. This is a four-port, three-position, solenoid directly operated valve with a nominal diameter of 10 mm and working pressure up to 35 MPa. The aim of the study was to investigate whether hydraulic forces acting on the spool during the flow of working fluid through the valve could be reduced by changing the geometry of that spool. ANSYS/Fluent software was used for the CFD analysis.

Keywords: *CFD analysis, hydraulic directional control valve, flow forces, ANSYS/Fluent*

S t r e s z c z e n i e

W artykule przedstawiono analizę CFD sił hydraulicznych działających na suwak rozdzielacza hydraulicznego typu WE10J. Jest to rozdzielacz czterodrogowy, trzypołożeniowy, sterowany bezpośrednio elektromagnetycznie o nominalnej średnicy przelotu 10 mm i ciśnieniu roboczym do 35 MPa. Celem analizy było zbadanie, czy poprzez zmiany geometrii suwaka możliwe jest zmniejszenie wartości sił hydraulicznych działających na ten suwak przy przepływie cieczy roboczej przez rozdzielacz. Do analizy CFD zastosowano program ANSYS/Fluent.

Słowa kluczowe: *analiza CFD, rozdzielacz hydrauliczny, siły hydrauliczne, ANSYS/Fluent*

* Prof. Ph.D. D.Sc. Eng. Edward Lisowski, Institute of Applied Informatices, Faculty of Mechanical Engineering, Cracow University of Technology.
** Ph.D. Janusz Rajda, Ponar Wadowice S.A.

1. Introduction

Directional control valves direct solenoid operated are used in the drive and control of various machines and equipment. They change the direction movement of the working element. Their relatively simple design and low power consumption by the solenoid makes them readily used, especially in hydraulic systems with not a very high fluid flow rate. For large values of the flow rate, pilot operated directional control valves are used where a solenoid directly operated valve is a pilot. The design of pilot operated directional control valves is more complicated, as they have a greater weight and larger dimensions. Hence, the producers of directional control valves try to extend the scope of the flow rate for directional control valves directly operated. This is not a simple task, because on the one hand, it requires optimization of the flow channels to reduce the flow resistance, while on the other hand, we must seek to limit the flow forces associated with the flow. Their growth significantly affects the balance of forces acting on the spool, causing finally inability to control the valve. The valve in question WE10J was initially produced for a flow range of 80 dm^3/min., and subsequently that range was increased by 50% [5] without changing the external dimensions. Now, the aim is to double the stated initial scope, and even more. In this paper a CFD analysis using ANSYS/Fluent will be presented, which focuses on the study of the impact of the spool geometry for the hydrodynamic reaction.

2. The object of the study

Directional control valves WE10 are produced with various configurations of flow paths. This paper refers to one type, marked WE10J [5], which is often used to control the direction of movement of the actuator or hydraulic motor. This valve is built of a body 1 made of casting, spool 2, centering springs 4, and two solenoids 3. The standard design of the valve spool is shown in Table 1, marked as *A*1. For the study, variations of design solutions have been selected to investigate which one would have the best performance.
They are symbolized by *A*1 to *A*6.

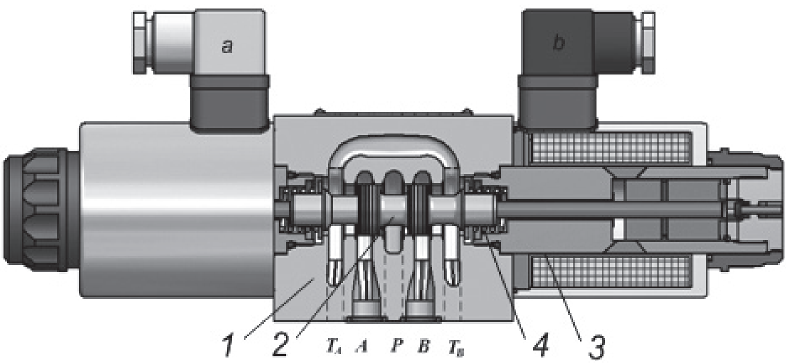
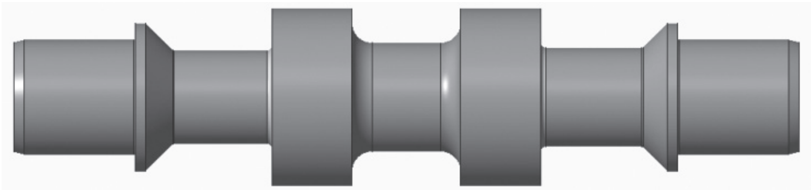
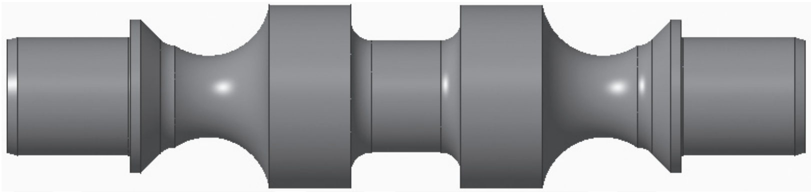
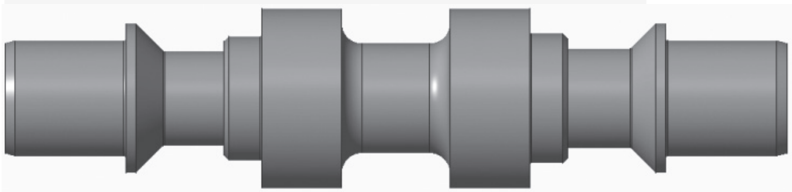
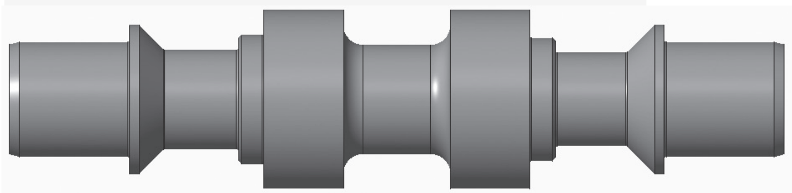
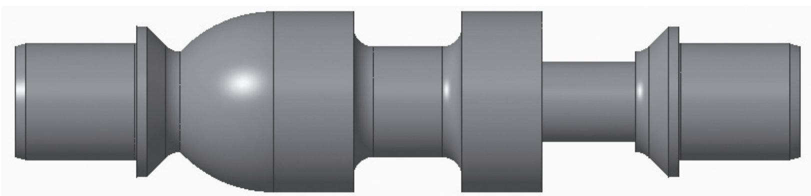


Fig. 1. chematic overview of directional control valve WE10J

Table 1

Various spool designs selected for the study

A1	
A2	
A3	
A4	
A5	
A6	

3. Discrete model of flow paths

For the purposes of the CFD analysis, it was necessary to build a geometric model of the fluid filling communication channels of the valve. That was done in the Creo Parametric software using Boolean operations on properly prepared components representing respective volumes of the valve [4]. Geometric models of the flow paths resulting from logic operations are shown in Figure 2. Due to the fact that paths *PA* and *BT* are identical with paths *PB* and *AT*, path *PB* and *AT* were selected to study. Because the individual paths are disjoint geometrically, their analysis may be carried out individually for each flow path. In the paper the analysis process will be shown on the example of *AT* flow path. FEM mesh and flow models were performed using ANSYS/Fluent software. Fig. 3 shows a discrete model of the

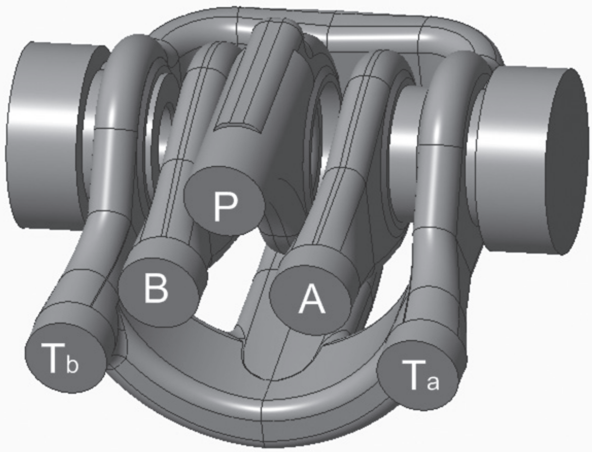


Fig. 2. Fluid geometrical model

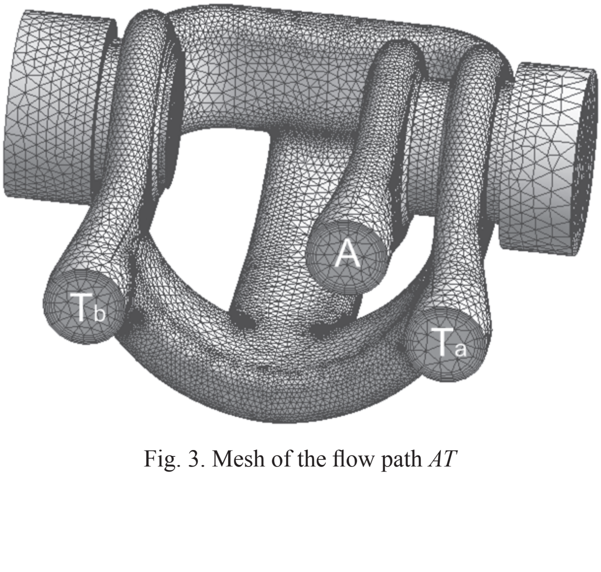


Fig. 3. Mesh of the flow path *AT*

flow path *AT*. This model was made by means of the advanced features of compaction of the net. Five boundary layers were used for the channel surfaces, and in areas with complex geometry the grid was properly compacted. The path model *AT* includes 2 601 524 cells and 802 625 nodes.

4. CFD model

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

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left(\alpha_k \mu_{eff}\frac{\partial k}{\partial x_j}\right)+G_k+G_b-\rho\varepsilon-Y_M \tag{1}$$

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

In these equations *Gk* represents the increase in the kinetic energy of turbulence caused by a 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 respectively Pandtl's numbers.

Turbulent viscosity, μ_t, is calculated as follows:

$$\mu_t=\rho C_\mu \frac{k^2}{\varepsilon} \tag{3}$$

Model constants $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$, $C_m = 0.09$, $s_k = 1.0$, $s_\varepsilon = 1.3$.

To carry out the calculation in the ANSYS/Fluent program after generating the grid, boundary conditions in the Setup module of the program must be defined. In the option *Boundary Conditions* as inlet velocity magnitude is assumed to be normal to boundary. *Value Velocity* magnitude is introduced as an input parameter to Workbench. As an outlet condition the outlet pressure is taken. Value *k* and ε are introduced as intensity and length scale. Using model Results of the 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.

5. Results of calculations

The CFD analysis was performed for the spool in a position allowing the flow from port *P* to *B* and from *A* to *T*. The results of the analysis showed that the greatest force values associated with the fluid flow were on the way from port *A* to *Tb*. Therefore, in the paper test the results for this flow path are shown. As a result of the conducted study, distributions of the fluid flow velocity and pressure on the walls were determined. Fig. 4 shows the velocity distribution along the stream lines, and Fig. 5 in the longitudinal section of the flow path. Calculations were made for the following boundary conditions:

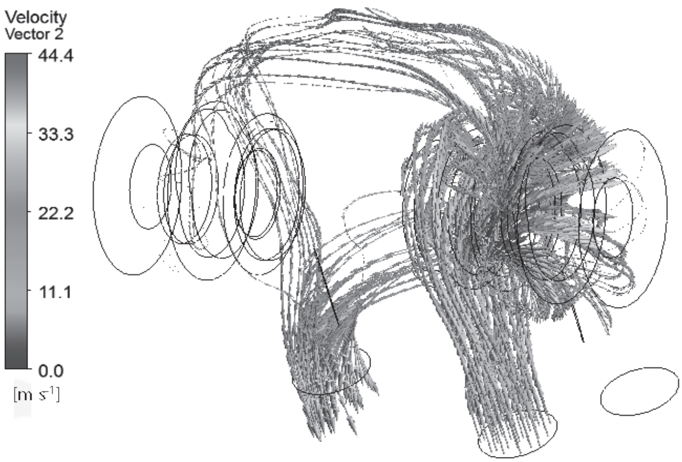


Fig. 4. Fluid velocity vectors along the stream lines for flow path *ATa*

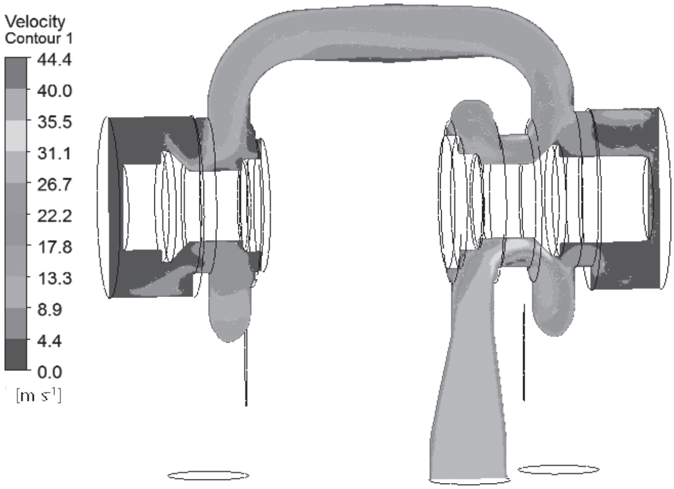


Fig. 5. Fluid velocity vectors in the longitudinal section for flow path *ATa*

– the average velocity of the fluid in the plane of port *A*: $v = 28$ m/s, which corresponds to the flow rate $Q = 160$ dm^3/min,
– the outlet pressure in the plane of port *Ta*: $p = 0.1$ MPa.

As it is clear from the results, the maximum value of the local velocity does not exceed 1.6 average value at the inlet to port *A*.

Fig. 6 shows the distribution of pressure on the walls of the spool. During the flow from port *A* to *Tb* the fluid flows in both ends of the spool. As it is evident from the pressure distribution, higher pressure at the spool is from channel *A*, what results in the formation of the resultant axial force directed opposite to the direction of the solenoid force. Table 1 shows the pressure loss at fluid flow from port *A* to *Tb* and the value of hydraulic forces obtained for flow rate $Q = 160$ dm^3/min. *F1* stands for the value of force acting on the surface of the spool from port *A*, while *F2* for the value of hydraulic force acting from port *Tb*. *F* is the value of the resultant of the two forces acting on the valve spool.

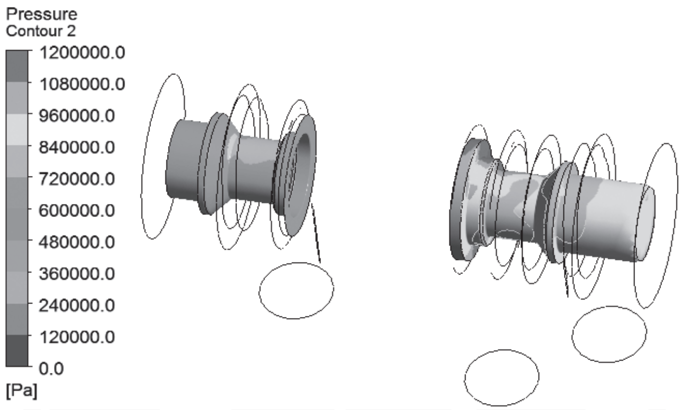


Fig. 6. Pressure distribution on the spool surface

Table 2

Pressure losses and forces associated with the flow from port a to *Tb*

Variant	Δ*p* [MPa]	*F*1 [N]	*F*2 [N]	*F* [N]
A1	0.92	–224.94	165.84	–59.10
A2	1.10	–229.10	167.68	–61.43
A3	2.26	–224.96	164.48	–60.48
A4	0.95	–212.22	145.72	–66.50
A5	0.90	–213.05	149.46	–63.59
A6	0.91	–220.12	149.53	–70.58

As it can be seen from the data presented in Table 2, the resulting hydraulic forces *F*1 and *F*2 associated with the fluid flow have opposite directions, what makes their operation to a large extent mutually compensating. There remains, however, some value of unbalance. Table 1 shows various designs of spools but they do not fully solve the problem. The lowest value of the resultant force was obtained for variant A1, while the lowest flow rate loss for variant A5.

6. Conclusions

The paper has presented a CFD analysis of the four port directional control valve WE10J of nominal size 10, intended for the use at flow up to 120 dm^3/min. The study was conducted for a wider range of up to 160 dm^3/min. in order to estimate the permissible scope of its work. The CFD analysis has shown a possibility of a much greater range of the flow rate. That is, however, connected with the formation of the fluid forces which act on the valve spool in the direction opposite to the solenoid force. The formation of fluid forces is inevitable for a wide range of flow rate. On the basis of the flow forces defined by the CFD analysis, a solenoid and return spring of respectively sufficient strength can be chosen. Furthermore, the range of parameters for proper performance of the valve can be determined.

References

[1] Launder B.E., Spalding D.B., *The Numerical Computation of Turbulent Flows*. *Computer Methods in Applied Mechanics and Engineering*, 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] Lisowski E., Czyżycki W., 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*, Volume 70, June 2013, 220-229.
[5] *Directional spool valve type WE10 electrically operated*, Catalogue, Ponar Wadowice, WK 499 780, 2013.