Application of CFD Numerical Simulations and Shape Optimization to Modify the Flow Characteristics of Throttle Valves

Alicja Dykas, Urszula Warzyńska*

Wrocław University of Science and Technology, Faculty of Mechanical Engineering, 5 Łukasiewicza St., Wrocław, 50-371, Poland

Received: April 17, 2025. Revised: May 31, 2025. Accepted: June 23, 2025.

Abstract

The aim of the study was to perform a numerical analysis using the CFD method of oil flow through a hydraulic valve gap and to perform an optimisation of the gap shape with a view to linearising the valve characteristics. As part of the work, a flow analysis of the valve was carried out using numerical simulations. This made it possible to develop the characteristics of the studied valve. The optimisation process started with a shape sensitivity analysis to determine the effect of geometry on key flow parameters such as pressure drop. One of the resulting solutions selected on the basis of its functionality and technological manufacturing possibility was further analysed. The flow characteristics determined for the optimised design were compared with those of the original valve using statistical tools. It was shown that optimised geometry achieved a more linear characteristic, which will enable more precise throttle control using this valve.

Keywords: shape optimisation; computational fluid dynamics; pressure valve; pressure drop; flow characteristics; finite volume method; hydraulic systems.

1. Introduction

The development of modern hydraulic power systems is associated with increasing demands for control precision, energy efficiency and compactness of design solutions. A key role in achieving these goals is played both by new valve concepts and the development of numerical methods – primarily computational fluid mechanics (CFD), which supports the design and optimization of hydraulic components. In the technical literature, several major areas of research can be distinguished that are part of these issues.

Innovative approaches to valve design are an important direction in the development of power hydraulics. Ongoing research focuses, inter alia, on control valves [1] for which a method of active control of differential pressure in control valves has been proposed, which allows for a significant improvement in their accuracy and dynamic performance, or on proportional valves [2] where a flow coefficient analysis has been carried out using the CFD method, identifying the effect of internal geometry on flow characteristics. Both approaches show that properly selected design and numerical support can significantly improve the functionality of hydraulic valves.

Numerical simulations have also been successfully used in the analysis of flows in hydraulic pumps, motors and actuators, for example for the aerospace industry [3], where an electro-hydraulic pump was studied, focusing on the analysis of dynamic flow properties. Studies using simulation are also applicable to pumps in the automotive industry [4], where very high speeds are involved. Also, simulations using deforming mesh and mesh replacement [5] show possibilities to accurately represent geometry and boundary conditions for realistic flow analyses. Another example is the possibility of studying fluid-structure interactions, which increase the accuracy of the representation of real operating conditions [6].

* Corresponding author. Email address: urszula.warzynska@pwr.edu.pl

This paper should be cited as: A. Dykas, U. Warzyńska. (2025) Application of CFD numerical simulations and shape optimization to modify the flow characteristics of throttle valves. *Energy Engineering and Control Systems*, Vol. 11, No. 1, pp. 33 – 43. https://doi.org/10.23939/jeecs2025.01.033

2. Analysis of recent research and publications

An important area of research is the application of shape and topology optimization methods in the design of hydraulic components. Studies of the effect of the diameter of the inlet nozzle on the efficiency of fluid exchange in a hydraulic cylinder [7] show that optimizing the shape of the inlet elements significantly improves the efficiency of the actuator. Also exemplified is a comprehensive approach to the design of gerotor and orbital hydraulic machines [8], which uses advanced reverse engineering, 3D modelling and flow analysis to improve performance. This approach is in line with current trends of combining simulation techniques with geometric optimization, which is the foundation of modern design.

Shape optimisation became one of the most popular directions in components design thanks to many modern calculation tools many of which contain optimisation modules already built in. One of the examples that used CFD method to optimise shape is spool valve grove optimisation [9] using method of inner surface response checking. Shape optimisation of the valves may be used in many fields and may have many different objective functions, such as improving the electromagnetic force in electromagnetic valves [10] or temperature distribution efficiency [11] that can be used in heat control valves. The next advantage of shape optimisation is a fact that multiple objectives can be set for one optimisation, so that a compromise between several desirable properties is reached. Examples can be multi-objective optimization to improve structural safety and sealing performance of butterfly valves [12] or optimization of channel shape with Tesla valve [13] taking into account both temperature uniformity and pressure drop.

3. Formulation of the goal of the paper

On the background of the above studies, the present work focuses on using CFD methods and shape optimization to modify the flow characteristics of a throttling valve. The goal is to improve its efficiency and match its precise operating conditions by analysing the velocity distribution, pressure losses and the effect of geometry on hydraulic parameters.

4. Theoretical analysis

The velocity of the receiver in a hydrostatic drive system is dependent on the rate of fluid supplied to it. The use of throttling valves in the hydrostatic drive system in a suitable arrangement allows us to control the speed of the receiver when using a fixed displacement pump (throttle control or throttle regulation). The efficiency of systems with such a solution is low, because of power loss caused by intentionally draining part of the fluid flow back to the tank. The second solution for controlling the speed of the receiver is the use of a variable displacement pump (volumetric control or regulation). However, variable displacement pumps are often much more expensive than fixed-displacement. Throttle control is more often selected in low-power systems and in cases of long downtimes of the throttle-controlled receiver or in the case of a short period of time in the operating cycle that requires choking the flow.

To describe the behaviour of a fluid and its parameters within a throttle valve, two extreme cases of throttling gaps are considered: a sharp-edged orifice and a capillary. In the first case, the flow is turbulent, whereas in the second, it is laminar. Laminar flow through a circular cross-section, such as a capillary, can be described using the Hagen-Poiseuille equation, which is expressed as follows [14]:

$$Q = k_1 f_d \Delta p, \tag{1}$$

where k_1 is proportionality coefficient depending on the viscosity of the liquid; f_d is flow area of the capillary (m^2); Δp is pressure difference between the inlet and outlet of the capillary (Pa).

We can describe the coefficient k_1 as [14]:

$$k_1 = \frac{d^2}{32\mu l}, \tag{2}$$

where d is diameter of the capillary (m); μ is dynamic viscosity of the liquid (Pa·s); l is length of the capillary (m).

Turbulent flow, which occurs when the fluid flows through an orifice, can be described for this case by the equation [14]:

$$Q = k f_d \sqrt{\Delta p}. \tag{3}$$

In this case, the k-factor can be described by the equation [9]:

$$k = \sqrt{\frac{2}{\zeta \varrho}}, \tag{4}$$

where ζ is flow resistance coefficient for the local resistances, which depends, among other things, on the geometry of the orifice; ϱ is the density of the fluid (kg/m^3).

Based on equations (1) and (3), the general equation for flow through any type of throttling gap may be determined. This equation is as follows [14]:

$$Q = k_1 f_d \Delta p^n. \quad (5)$$

In this equation, the power exponent n will take the value $n = 1$ for a capillary, while $n = 1/2$ for a sharp-edged orifice. Creating from these equations the flow characteristics for both these types of flow resistances as a function of pressure difference Δp, it can be concluded that the greater variability will be characterized by the flow through the capillary. The slots of actual throttling valves, which are most often the geometric connection of an orifice and a capillary, will be characterized by flows whose characteristics will lie between these two extremes, their power exponents n will be in the range of $½ > n > 1$.

Obtaining the desired flow characteristic of a throttle valve depends mainly on the shape of the flow gap, which can be improved by shape optimization methods. Modern optimization methods, based on algorithms and numerical techniques, make it possible to identify the optimal solution more efficiently without having to analyse all possible alternatives. Optimization is a separate theory that lies within the field of applied mathematics. It aims to adequately formulate the object of optimization with a set of solution searches and to define the objective function. Once this problem has been formulated, the next step is to find the optimal solution that satisfies the accepted criterion. The following steps of optimization can be presented as [15]:

- Development of a mathematical model of the problem under analysis.
- Determination of the objective function.
- Searching for the optimal solution using the selected optimization method.

There are some basic concepts associated with optimization, which are discussed below.

- The set of admissible solutions $X_d \subset X$ is defined as the set of points $x \in X$ that are taken into account in the optimization process. It is defined by specifying conditions that must be satisfied by a vector x in order for it to belong to the set X_d. When no constraints are specified and $X_d = X = R_n$, where R_n is an n-dimensional space of real vectors, optimization without constraints occurs. Otherwise, there is optimization with constraints.
- The objective function is as follows:

$$f : X \to R$$

For each solution $x \in X$ there is assigned some numerical value $f(x) \in R$ from the set of admissible solutions. This gives the possibility to compare different solutions, since this assigned value expresses the quality of the solution against the specified optimization criterion. Depending on the problem, one seeks maximization or minimization. In minimization problems, the objective function is often referred to as the cost function. The goal of optimization in such cases is to identify, among the possible options defined by the given constraints, the solution that minimizes this cost, which may be, for example, the pressure output.

Any optimization problem requires finding a solution x that minimizes (or maximizes) the value of the objective function while satisfying certain constraints that define the set of feasible solutions. For the vast majority of practical applications of optimization methods, it is unlikely to find an exact minimum (or maximum) solution. This is due to the limited accuracy of numerical methods. Therefore, a solution x close to assumed is sufficient, which is determined by satisfying the stop conditions. Stop conditions are most often convergence tests, when the point obtained in successive iterations differs little from the previous one, it can be assumed that subsequent iterations will not lead to a much closer solution [15].

There are many optimization methods and criteria for their division. However, the remainder of this paper focuses only on the methods used by the optimization module of Ansys Fluent software in conjunction with Adjoint Solver. These are gradient methods. They use not only knowledge of the current values of the objective function, but also its gradient and associated values. Therefore, the objective function in the case of gradient methods must be defined and differentiable throughout space. When searching for the minimum solution, gradient methods analyse- -the upward trend of the objective function, since the gradient $\nabla f(x)$ indicates the direction of the greatest increase in the objective function, while the vector- $\nabla f(x)$ indicates the direction of the greatest decrease.

In general, the optimization problem can involve both minimization and maximization of observable shapes, as well as constraints localized in space. This requires an approach that can provide a deformation field that is well-behaved and consistent with production requirements, while allowing locally sharper deformations when required to satisfy the imposed constraints. For Adjoint solver, Fluent provides three different morphing techniques: polynomial-based, using direct interpolation and using a radial basis function. These methods are based on different design spaces and therefore produce different morphing results. Key information about them is presented below [16]:

- The polynomial method uses polynomial functions to model changes in the geometry in the design space. In this technique, shape modifications are defined using low-order polynomials, which preserves the smoothness and continuity of geometric surfaces. The advantage of this method is its simplicity and low computational requirements, making it suitable for relatively simple geometries or preliminary stages of optimization. However, a limitation is the difficulty in accurately representing complex shape changes, which may require the use of higher-order polynomials, increasing computational complexity.
- The direct interpolation method involves directly displacing selected points on a geometric surface based on specified displacement values. In this technique, displacement values at intermediate points are calculated by linear or higher order interpolation. This method is intuitive and allows for precise control of changes in selected areas of the geometry, making it useful for local optimizations. However, the lack of global control over shape smoothness can lead to discontinuities, especially with large changes in geometry.
- The radial basis function (RBF) method uses mathematical functions that are defined with respect to the distance from a central point (known as a node). These functions are used to smoothly transform the entire geometric mesh based on the displacement values at selected nodes. RBF is very flexible and does a good job of mapping complex shape changes while keeping the geometry smooth and continuous. This makes the method particularly effective for complex geometries and global optimizations. The disadvantage of this technique is its higher computational cost compared to other methods, especially for large meshes.

5. Results of the study

5.1. Original valve flow characteristics

The CFD flow simulations were performed in Ansys Fluent software with the use of finite volume method. The original geometry of the valve was prepared based on technical documentation [17] using SpaceClaim software. Based on 2D technical drawings, the internal 3D domain of the fluid filling the interior of the valve structure was modelled (Fig.1 and Fig.2). The discrete model was composed of tetrahedral elements on surface mesh and poly-hexcore elements in the interior domain with the maximum size of 2 mm, and mesh refinement rate of 1.2 in small gaps (Fig.3).

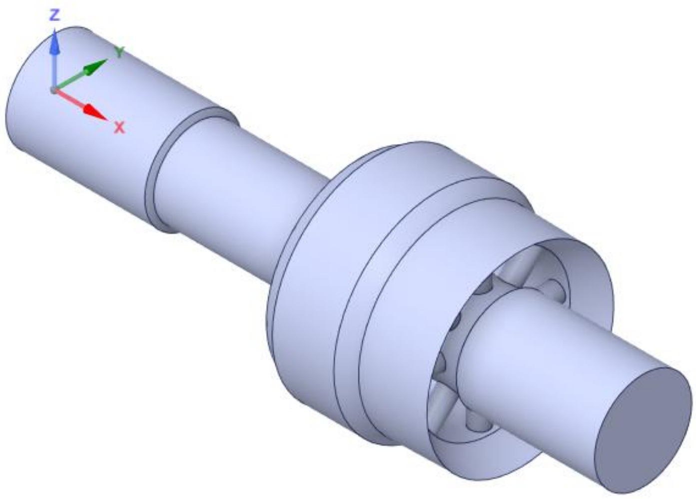


Fig.1. Geometrical model of the original valve fluid domain geometry.

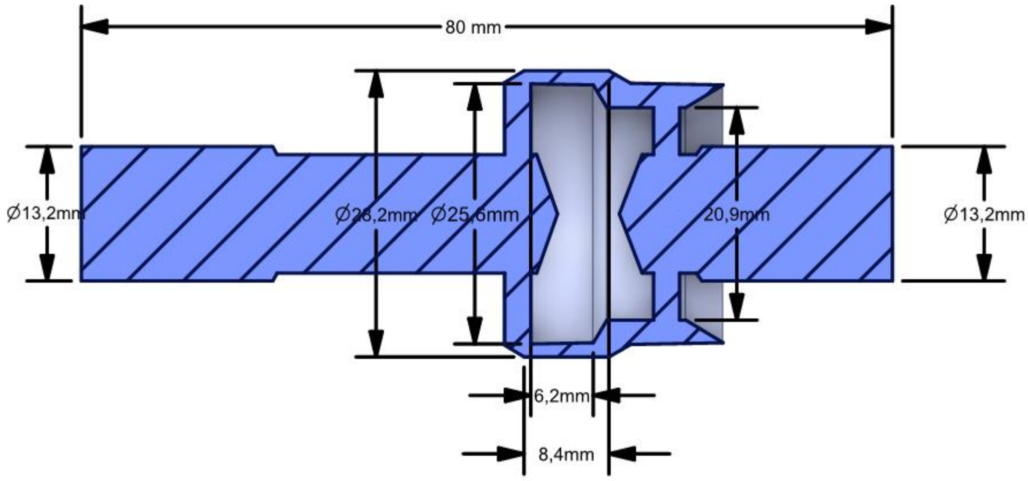


Fig.2. Basic dimensions of the original valve fluid domain geometry.

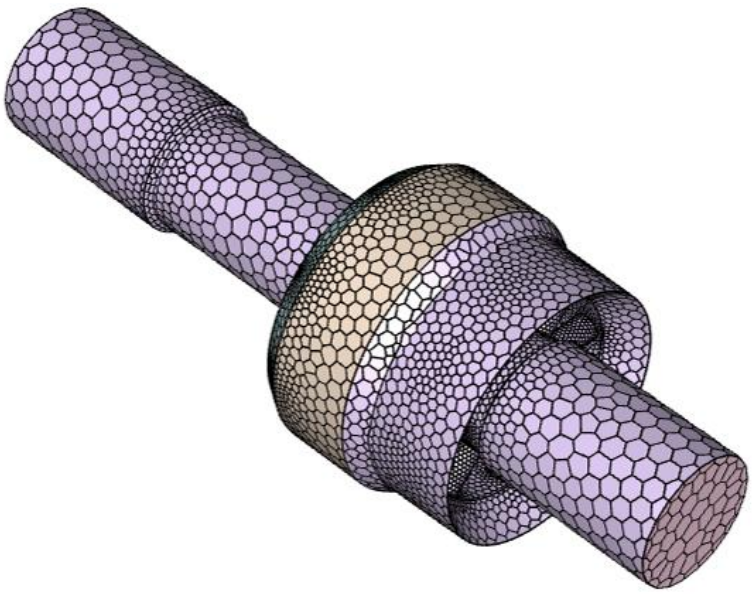


Fig.3. Discrete model of the original valve fluid domain geometry.

In the flow simulations, hydraulic oil Hydrol L-HL 46 was used with constant parameters of density equal to 864.7 kg/m^3 and viscosity 0.0393 Pa·s [18]. The inlet boundary condition was set to velocity varying with the analysed case. Pressure and flow velocity distributions were analysed for five flow rates, ranging from 10 to 50 l/min, with a step of 10 l/min, which translated into inlet flow velocities ranging from 1.2 to 6.1 m/s. The outlet was set to a pressure outlet with free outflow. The standard k-ε turbulence model was chosen because of reported reliable results in engineering applications [19]. The following k-ε model parameters values were used. For Cmu, which is the k-ε turbulence model constant associated with the modeling of turbulent viscosity the value Cμ=0.09 is a standard value calibrated experimentally and theoretically for most turbulent flows. For C1-Epsilon which is the calibration constant in the transport equation for ϵ, the turbulence dispersion rate the standard value is C_{ϵ}^{1}= 1.44 and it results from calibration for fully turbulent flows. C2-Epsilon is the damping constant in the transport equation for ϵ. It controls the decay of turbulence where the production rate of ϵ is low, such as near the laminar-turbulent flow boundary. The standard value is C_{ϵ}^{2} = 1.92. The first step of the study was to perform steady-state flow simulations in the original geometry of the valve. The results of pressure and velocity fields in the example of the highest flow rate of 50 l/min are shown in Fig.4 and Fig.5. The simulation results for all tested cases (with different flow rates) allowed us to develop the flow characteristics $\Delta p = f(Q)$ for the original geometry of the valve. Convergence was reached after about 70 iterations. During this time, the values of the finite residuals declined. The residuals for velocities in the x, y, and z directions and the continuity equation decreased to values on the order of 10^{-4}. In addition to the residuals, the stability and consistency of physical parameters such as wall forces, pressure losses and flow development were monitored. Finite residuals of 10^{-4} are acceptable for most engineering applications, suggesting that the solution is accurate, and further iterations would not result in significant changes in the results. The reliability of these results is also affected by the quality of the grid, the choice of the turbulence model and the use of second-order schemes, which makes it possible to accurately represent the distribution of flow parameters in sensitive zones.

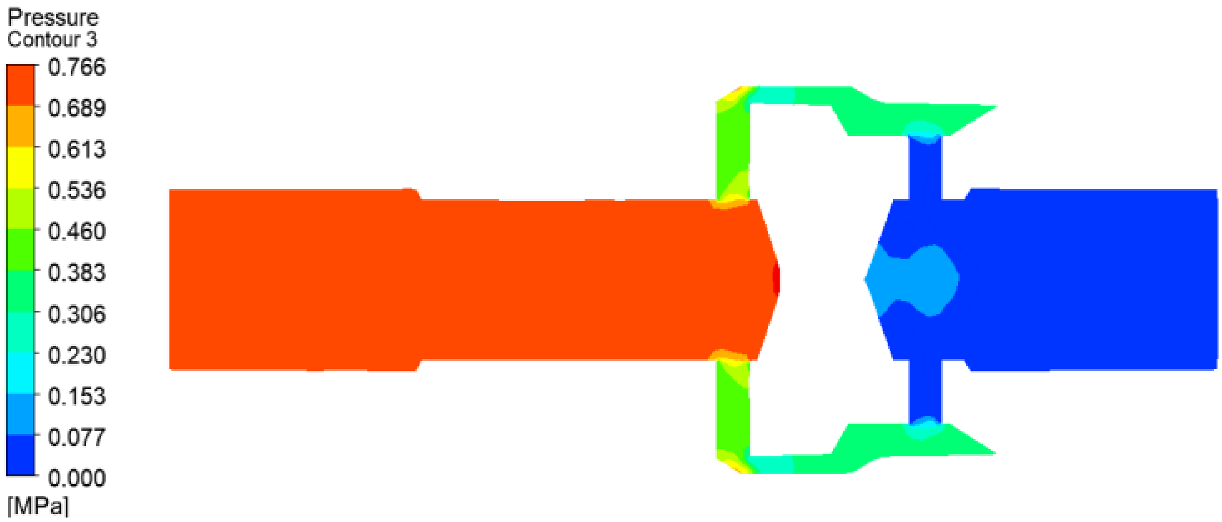


Fig.4. Contour pressure diagram for a flow rate of 50 l/min.

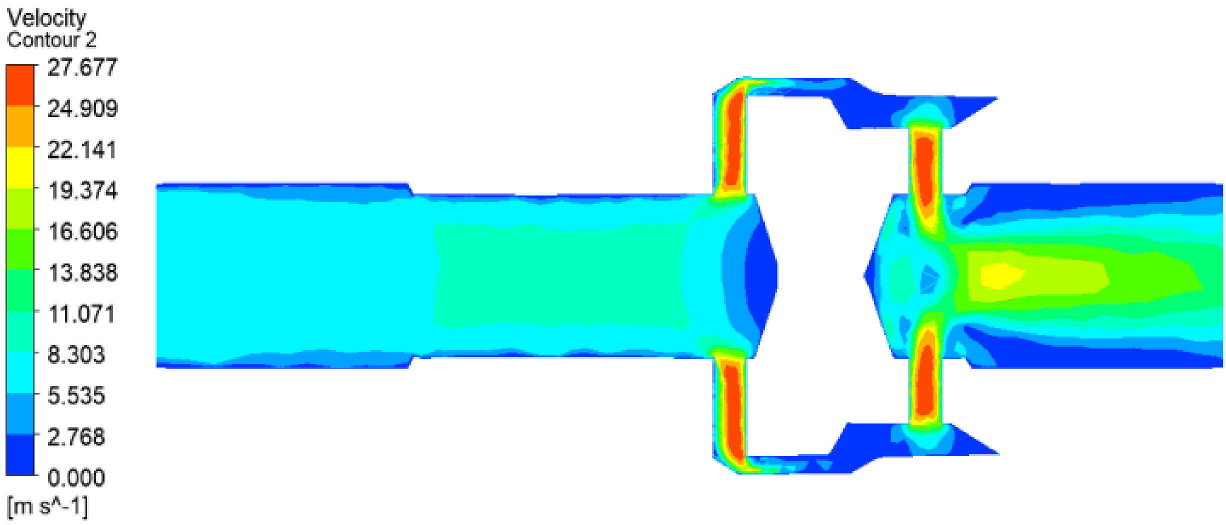


Fig.5. Contour flow velocity diagram for a flow rate of 50 l/min.

Fig.6, in addition to the characteristic itself, shows the value of the R^2 coefficient. This is the coefficient of determination used to evaluate the fit of the linear regression model to the data. The value of the coefficient R^2= 0.9718 indicates high agreement of the studied characteristics with the characteristics determined by linear regression. However, to determine the reliability of this coefficient, it is worth supplementing it with the value of the root mean square error RMSE, which gives information about the difference between actual and predicted values. For this case, the calculated value was RMSE = 0.04234. The RMSE value representing less than 10% of the spread of characteristic values can be considered low. With this information, the reliability of the coefficient of determination R^2 has been confirmed, so further analysis and comparison of subsequent characteristics can be carried out on its basis.

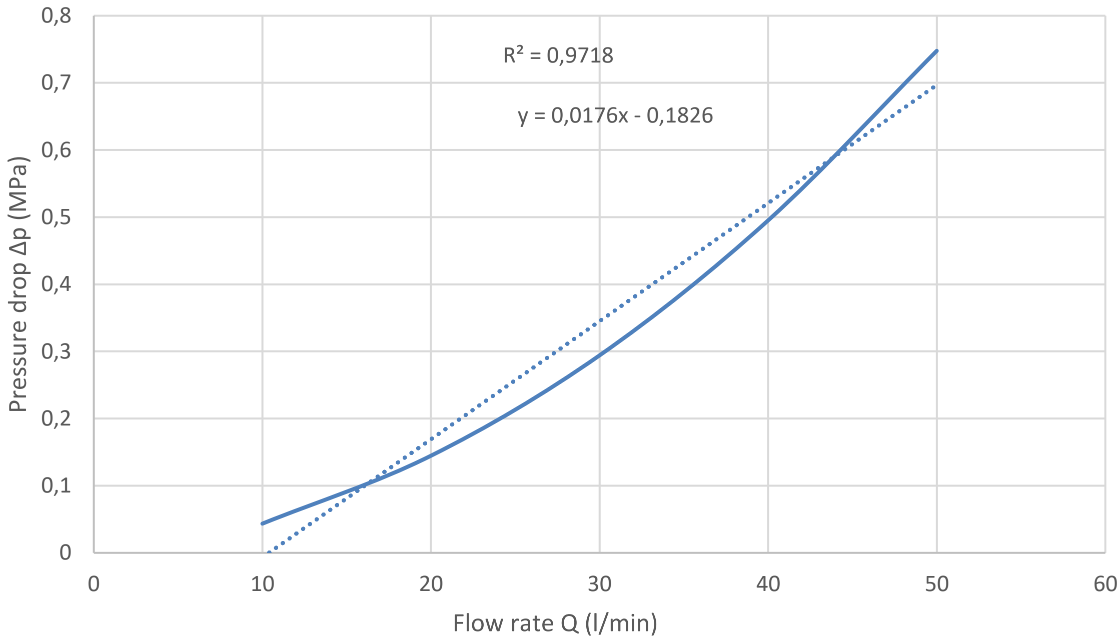


Fig.6. Flow characteristics of the original geometry of the valve.

5.2. Valve shape modification

Modification started on calculating the sensitivities of the observable variable to individual flow parameters. These are used as input data for the optimization module in the further step. Pressure drop was chosen as an observable.

Fluent's shape optimization module – Design Tool – based on the coupled sensitivities determined by Adjoint Solver performs shape optimization of the user-selected part of the geometry to achieve the optimization goal. A change in pressure drop is chosen as the optimization goal. At this stage, the desired value of this drop will be determined. However, the first step is to determine the method of morphing, that is, changing the mesh.

Part of geometry that was chosen to be modified is inner surface of the throttling gap. Defined value of change in pressure drop was set as -6% and -8% for each morphing method. Achieved geometries are compared in Table 1 along with analysis of functionality of changes and technological feasibility. The technological feasibility of the valve modification was determined by analysing its geometry, the materials used for such components and the available processing methods, taking into account the applicable tolerances and parameters of the manufacturing processes.

Table 1. Comparison of modification methods and resulting geometries.

Morphing method	Percentage of change in pressure drop	Functionality	Technological feasibility	Selection for further analysis
Polynomials	-6%	No information- invisible change	Mesh displacement values too small, below possible manufacturing tolerances	Rejected
	-8%	No information- invisible change	Mesh displacement values too small, below possible manufacturing tolerances	Rejected
Radial basis function	-6%	Operation for one valve setting only	The need for complex operations	Rejected
	-8%	Operation for one valve setting only	The need for complex operations	Rejected
Direct interpolation	-6%	Change in the whole circuit, influence in the whole range of the valve setting	Mesh displacement values close to manufacturing tolerances	Rejected
	-8%	Change in the whole circuit, influence in the whole range of the valve setting	Mesh displacement values within the manufacturing tolerances of CNC lathes	**Chosen**

As a result of modification by Direct interpolation by the value of the change in pressure drop of -8%, a gap surface geometry technologically feasible in a simple operation of turning the rounded surface of the valve inner sleeve was obtained. Displacements on the order of tenths to hundredths of a millimetre are also within the manufacturing tolerances for CNC lathes [20]. The functionality of this solution also extends to the remaining valve settings. For these reasons, the geometry was further analysed. The geometry before optimisation is shown in Fig.7. And the optimised geometry selected for further analysis is shown in Fig.8.

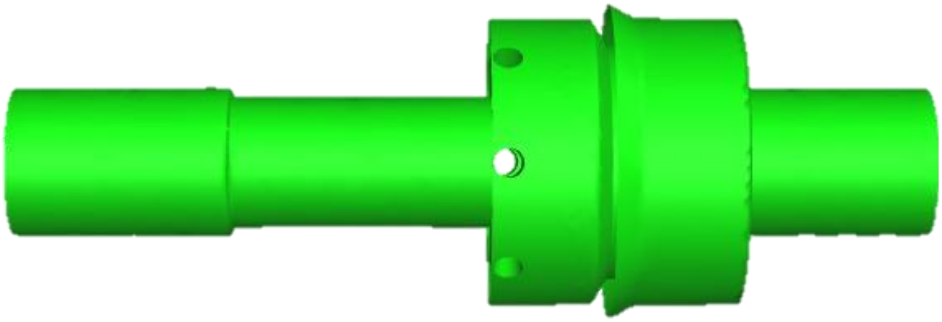


Fig.7. Geometry of the original valve (inner surface shown).

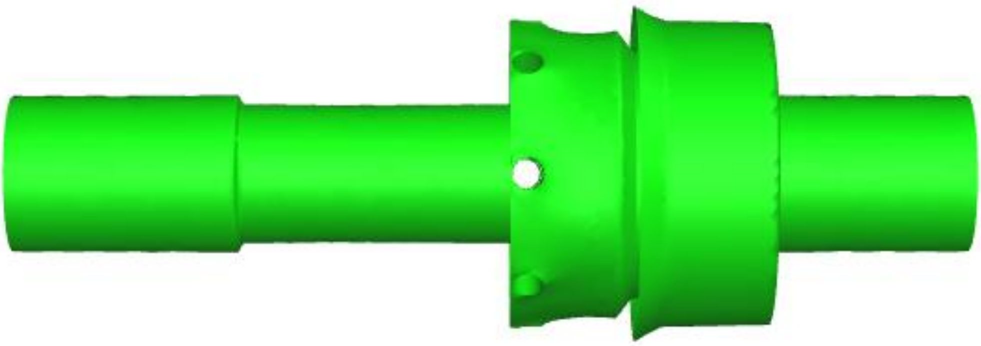


Fig.8. Geometry obtained in the modification (inner surface shown).

For the optimised geometry, flow calculations were performed again to determine its flow characteristics. Boundary conditions were the same as for the original geometry to achieve meaningful comparison. Results in the form of contour plots of pressure and velocity fields are presented in Table 2. The flow characteristics of the optimised geometry are shown in Fig.8.

Table 2. Pressure and velocity charts for optimised valve geometry.

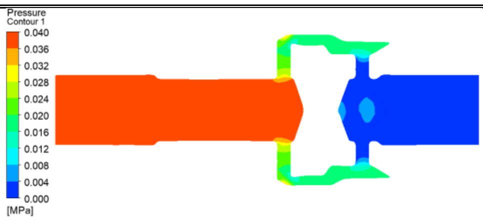
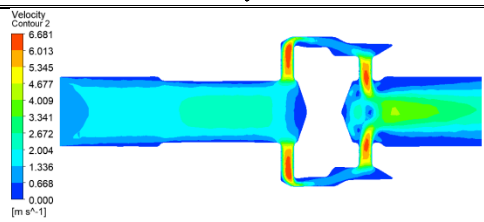
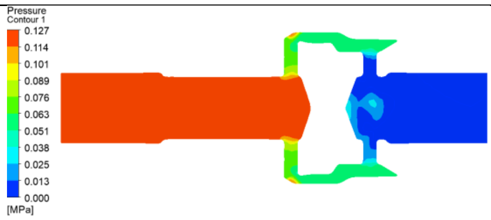
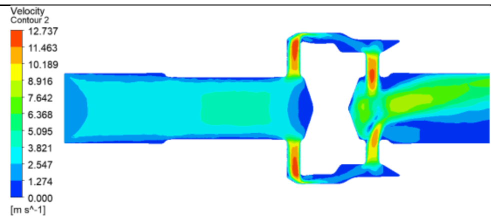
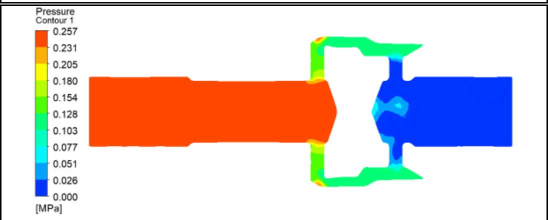
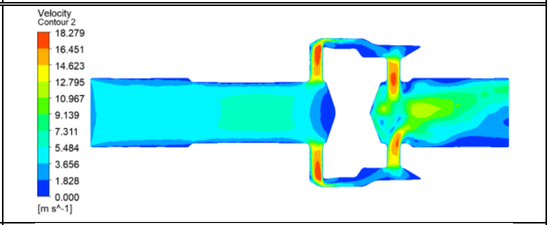
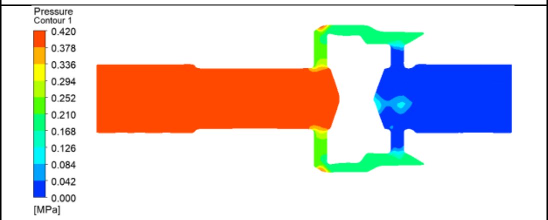
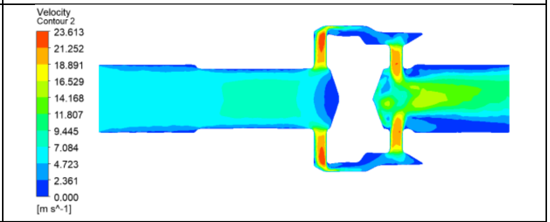
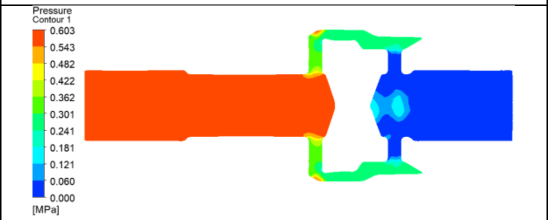
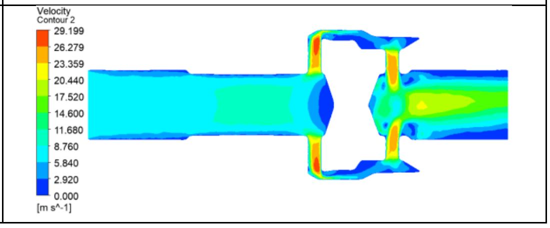
Flow rate	Pressure charts	Velocity charts
$10 \frac{l}{min}$		
$20 \frac{l}{min}$		

Table 3 (continued)

Flow rate	Pressure charts	Velocity charts
$30\ \frac{l}{min}$		
$40\ \frac{l}{min}$		
$50\ \frac{l}{min}$		

In order to compare the obtained characteristics (Fig.9), the coefficient of determination R^2 was determined. For the optimised case, it was equal to 0.9825 with RMSE = 0.026059, while for the original geometry $R^2 = 0.9718$ with RMSE = 0.04234.

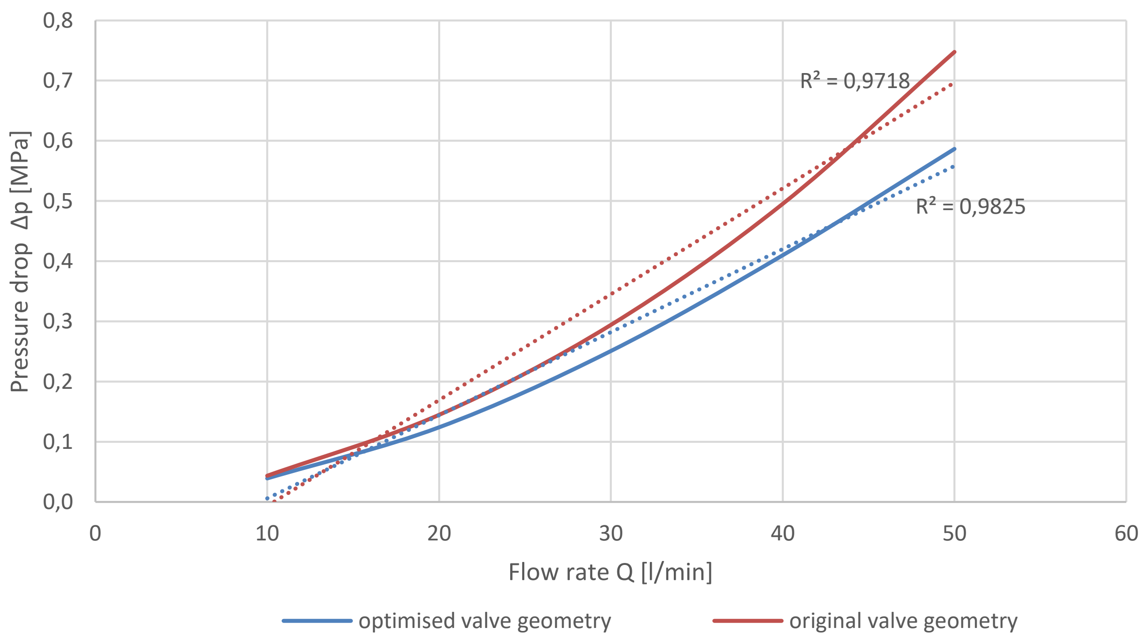


Fig.9. Flow characteristics of the optimised and original geometry of the valve.

6. Conclusion

The study demonstrates that optimizing the shape of the throttling gap in a throttle valve significantly improves the linearity of its flow characteristics. By employing computational fluid dynamics simulations and optimization algorithms, the geometrical modifications that reduce flow nonlinearity and enhance control precision were identified.

The valve with the original geometry has an R^2 of 0.9718, indicating that its characteristics are well matched to linear characteristics, but not perfectly. There is some nonlinearity in the performance of the valve in this configuration.

The valve with the modification of the internal gap area has a higher R^2 value, equal to 0.9825. This indicates that the change in the internal gap geometry improves the fit of the valve's characteristics to the linear characteristic, which may mean a more stable and predictable performance.

Compared to using complex mechanical compensation mechanisms, shape optimization offers a simpler and more economical solution, reducing manufacturing and maintenance costs. Further research could explore adaptive shape control and real-time optimization techniques to enhance valve efficiency under varying operating conditions.

References

[1] Wang, B., Zhao, X., Quan, L., Li, Y., Hao, Y., & Ge, L. (2023). *A method for improving flow control valve performance based on active differential pressure regulation*. *Measurement, 219*, 113271. https://doi.org/10.1016/j.measurement.2023.113271

[2] Lisowski, E., & Filo, G. (2017). *Analysis of a proportional control valve flow coefficient with the usage of a CFD method*. *Flow Measurement and Instrumentation, 53*(Part B), 269–278. https://doi.org/10.1016/j.flowmeasinst.2016.12.008

[3] Zhu, D., Fu, Y., Han, X., & Li, Z. (2020). Design and experimental verification on characteristics of electro-hydraulic pump. *Mechanical Systems and Signal Processing, 144*, 106771. https://doi.org/10.1016/j.ymssp.2020.106771

[4] Milani, M., Montorsi, L., & Paltrinieri, F. (2024). Experimental investigation of the suction capabilities of an innovative high speed external gear pump for electro-hydraulic actuated automotive transmissions. *International Journal of Fluid Power, 25*(2), 243–272. https://doi.org/10.13052/ijfp1439-9776.2527

[5] Castilla, R., Gamez-Montero, P. J., Ertürk, N., Vernet, A., Coussirat, M., & Codina, E. (2010). Numerical simulation of turbulent flow in the suction chamber of a gear pump using deforming mesh and mesh replacement. *International Journal of Mechanical Sciences, 52*(10), 1334–1342. https://doi.org/10.1016/j.ijmecsci.2010.06.001

[6] Pellegri, M., Manne, V. H. B., & Vacca, A. (2020). *A simulation model of Gerotor pumps considering fluid–structure interaction effects: Formulation and validation*. Mechanical Systems and Signal Processing, 140, 106720. https://doi.org/10.1016/j.ymssp.2020.106720

[7] Siwulski, T., & Warzyńska, U. (2021). Numerical investigation of the influence of the inlet nozzle diameter on the degree of fluid exchange process in a hydraulic cylinder. *Engineering Applications of Computational Fluid Mechanics, 15*(1), 1243–1258. https://doi.org/10.1080/19942060.2021.1958379

[8] Stryczek, J., & Stryczek, P. (2021). Synthetic approach to the design, manufacturing and examination of gerotor and orbital hydraulic machines. *Energies, 14*(3), 624. https://doi.org/10.3390/en14030624

[9] Li, R., Wang, Z., Xu, J., Yuan, W., Wang, D., Ji, H., & Chen, S. (2024). *Design and optimization of hydraulic slide valve spool structure based on steady state flow force*. Flow Measurement and Instrumentation, 96, 102568. https://doi.org/10.1016/j.flowmeasinst.2024.102568

[10] Zhang, C., Zhao, Y., Jiang, C., Guo, J., & Li, W. (2024). *Structure optimization of electromagnetic valve to improve electromagnetic force*. Journal of Magnetism and Magnetic Materials. https://doi.org/10.1016/j.jmmm.2024.171600

[11] Moayedi, H., Chen, Y.-C., Liu, C.-Y., & Weng, C.-I. (2024). *Geometry optimization of a vortex tube for use as a throttling device in natural gas liquefaction process*. Cryogenics. https://doi.org/10.1016/j.cryogenics.2024.103366

[12] Meng, H., Zuo, S., Ren, W., & Li, Z. (2024). *Multi-objective optimization design of triple-eccentric butterfly valve considering structural safety and sealing performance*. Engineering Failure Analysis. https://doi.org/10.1016/j.engfailanal.2024.107280

[13] Xie, B., Guo, S., Zhang, Q., Zhang, X., & Chen, H. (2025). *Multi-objective optimization of Tesla valve channel battery cold plate*. Results in Engineering, 100052. https://doi.org/10.1016/j.rineng.2025.1006826

[14] Stryczek, S. (2017). *Hydrostatic drive. Vol. 2: Systems* (2nd ed.). Polish Scientific Publishers PWN (in Polish)

[15] Danielewska-Tułecka, A., Oprocha, P., & Kusiak, J. (2009). *Optimization*. Warsaw: Wydawnictwo Naukowe PWN. (in Polish)

[16] ANSYS Inc. (2017). *ANSYS Fluent User's Guide* (Version 18.2). ANSYS Inc.

[17] https://www.ponar-wadowice.pl/!uploads/attachments_prod/mg_wk496290_pl_11.2020.pdf

[18] Orlen Oil, „HYDROL L-HL", Wersja 1 / 2023.07.17

[19] Blazek, J. (2005). *Computational fluid dynamics: Principles and applications* (2nd ed.). Elsevier

[20] Zalewski A., Grzesik W., Deja M., et al., *CNC Machine Tools: Fundamentals of Operation and Programming*, WNT, Warsaw 2024. (in Polish)

Застосування чисельного CFD моделювання та оптимізації форми для модифікації витратних характеристик дросельних клапанів

Аліція Дикас, Урсула Важинська

Вроцлавський університет науки і техніки, факультет машинобудування,
вул. Лукасевича, 5, Вроцлав, 50-371, Польща

Анотація

Метою дослідження було виконати числовий аналіз потоку оливи через прохідний отвір гідравлічного клапана за допомогою методу CFD та виконати оптимізацію профілю клапана з метою лінеаризації його характеристик. В рамках роботи було проведено аналіз витратних характеристик клапана за допомогою числового моделювання. Це дозволило розробити характеристики досліджуваного клапана. Процес оптимізації розпочався з аналізу чутливості форми, щоб визначити вплив геометрії на ключові параметри потоку, такі як перепад тиску. Одне з отриманих рішень, вибране на основі його функціональності та технологічної можливості виробництва, було додатково проаналізовано. Характеристики потоку, визначені для оптимізованої конструкції, були порівняні з характеристиками вихідного клапана за допомогою статистичних методів. Було показано, що оптимізована геометрія досягла більш лінійної характеристики, що дозволить забезпечити точніше керування процесом дроселювання за допомогою цього клапана.

Ключові слова: оптимізація форми; обчислювальна гідродинаміка; напірний клапан; перепад тиску; характеристики потоку; метод скінченних об'ємів; гідравлічні системи.