DOI: 10.2507/34th.daaam.proceedings.xxx

NUMERICAL SIMULATION OF FLUID FLOW THROUGH THE VALVE

Halima Hadžiahmetović, Rejhana Blažević & Sanda Midžić Kurtagić

This Publication has to be referred as: Hadžiahmetović, H[alima]; Blažević, R[ejhana] & Midžić Kurtagić, S[anda] (2023). Numerical simulation of fluid flow through the valve, Proceedings of the 34th DAAAM International Symposium, pp.xxxx-xxxx, B. Katalinic (Ed.), Published by DAAAM International, ISBN 978-3-902734-xx-x, ISSN 1726-9679, Vienna, Austria
DOI: 10.2507/34th.daaam.proceedings.xxx

Abstract

The main goal of this paper is to present an analysis of the complex problem of fluid flow through the valve with a seat type A 10 FS1.0364 of a variable cross-section and the obtained results. For this purpose, a geometry was created in Ansys Fluent for an "open" valve configuration with different types of meshes, tetrahedral and hexahedral. The paper also presents a comparison of different configurations and meshes, and a comparison of the results obtained in this paper using numerical simulations and ANSYS Fluent, with the previously published results of the group of authors obtained from Solid Works Flow Simulation. Numerical simulations carried out in this paper using Fluent showed a good agreement of the results with previously obtained results of the group of authors in Solid Works Flow Simulation. Also, an interpretation of the results with a discussion is given and the main findings are highlighted in the conclusion.

Keywords: valve; numerical mesh; volume flow; velocity magnitude; numerical simulation;

1. Introduction

In modern hydraulic and pneumatic components for direct and control, the valves are defined as devices whose main task is to direct and control hydraulic energy in the process of transfer from the source (hydraulic pump) to the hydraulic motor. At variable flow, there is an additional mechanism that adjusts the size of the valve opening. An essential element of the hydraulic system is the working fluid. The main function of fluid in the hydraulic system is the transfer of energy. Fluid currents transfer energy from the source (hydraulic pump) to the consumers (motors). The working fluid lubricates the moving parts in the hydraulic system so they do not have special equipment for lubrication. In addition, the working fluid also serves as a means of preserving the hydraulic system, but also washes outflow channels in the hydraulic system. Different fluids can be used as working fluids in the hydraulic system. They were selected so that their physical and chemical properties comply with the conditions of exploitation [1]. Today, as a working fluid in hydraulic systems, 90% of mineral oil used is pure either with water or a mixture of synthetic fluids. Therefore, the common name for the working fluid of hydraulic systems is hydraulic oil.

Limited capabilities of analytical methods, in practice very often lead to the use of experimental solutions to fluid flow problems, i.e. an approach to measuring velocity, pressure, etc. real progress, or in their model, which uses so-called similarity theory. But in some cases, it is very difficult to conduct an experiment and sometimes even impossible. Metering equipment can disrupt the flow, or it is such a fluid flow that it is impossible to set the measuring sensors. Progress in technology and increased competition require optimized design and high technology such as a mechanism which will predict the fluid flow. Experimental design and its realization require considerable financial support and time to get results. An alternative experiment, or at least a complementary method, is the advent of fast, inexpensive digital computers, which also leads to a significant (although most of the key ideas were founded more than a century) application of numerical solutions of partial differential equations. When the possibility of using computers in fluid flow analysis was recognized, interest in numerical methods increased significantly. Solving the equations of fluid mechanics using computer programs has become an important field of research known as Computational Fluid Dynamics (CFD) analysis [2], [3]. Numerical simulations are used in many industrial areas, such as power plants, automotive, chemical engineering, aviation, petroleum refining, etc [4], [5], [6]. Therefore, many studies have used numerical simulations and various computer programs to solve the problem of fluid flow through a valve.

Vedova et al. [5] conducted research on the 3D model of the full geometry of the valve (OpenFOAM) that has been developed to predict the distribution of pressures (hence forces) inside it to make an optimization process possible. Simic and Herakovic [7] investigated the reduction of the flow forces in a small hydraulic seat valve in detail by using a commercial simulation tool, Ansys CFX. Banaś et. al. [8] compared the results of numerical calculations obtained by using ANSYS® R17.2 with experimental studies for a prototype of a hydraulic throttle valve made from transparent plastic. Szpica [9] presented the numerical research methodology to designate the throttle valve flow characteristics using SolidWorks Flow Simulation as the alternative to long and expensive experimental research. Polášek et. al. [10] investigated fluid flow through the throttle valve for the laminar model, RANS models, and LES model using numerical simulation in ANSYS Fluent. Hodzic et al. [11] investigated the possibility of applying numerical simulations to solve the problem of calculation and determination of the fluid flow through the hydraulic valve. Results of numerical simulation by using Solid Works Flow Simulation of hydraulic oil flow through a valve with seat surface type A 10 FS1.0364 were taken from paper [11], and the own results obtained in this paper are compared with them.

So, this paper aims to present the results obtained by numerical simulations in Fluent and compare them with the previously published results by Hodzic et al. obtained in Solid Works Flow Simulation [11] for the same valve type with the same input data for the analysis, and to provide interpretation of the results with discussion.

2. Methodology

The finite volume method (FVM) is a numerical method used to solve the problem of mineral oil flow. According to this method, the calculation domain is divided into a finite number of control volumes that do not overlap (numerical mesh), each of which is represented by the value of the physical size of the node that is within the control volume (usually in the middle).

There are two different approaches to generating numerical mesh. The first approach generates the appropriate mesh domain, and then the nodes (calculation points) are placed in the center of the control volume. This approach is used more often. The second approach is based on the creation of computational points and a mesh of control volumes created so that the surface of the control volume lies in the middle between two calculation points [12].

The numerical algorithm consists of the following stages:

- proper integration of the main equations of fluid flow overall (finite) control volumes domain solutions,
- discretization, which involves solving some integral in discrete form (by use of integral approximation), which converts the integral equation into a system of algebraic equations for the nonlinear modelling of fluid flow,
- solving algebraic equations using final iterative methods.

Numerical simulation was carried using the Ansys Fluent CFD software. Fluent is a state-of-the-art computer program for modelling fluid flow and heat transfer in complex geometries. Fluent provides complete mesh flexibility, including the ability to solve flow problems using unstructured meshes that can be generated in complex geometries with relative ease. Supported mesh types include 2D triangular/quadrilateral, 3D tetrahedral/hexahedral/pyramid/wedge/polyhedral, and mixed (hybrid) meshes. Fluent also allows the use of refined or coarse grids based on the flow solution [12].

3. Results and discussion

In the paper, the problem of the flow of mineral oil through the valve was analysed, therefore, certain assumptions were given that were used in solving the problem. The mineral oil HL-ISO VG 32 is released into the atmosphere at pressure p = 1 bar. Density and viscosity coefficient were assumed as $\rho = 887$ kg / m3, $\mu = 0{,}057655$ Pa·s, respectively. Volume flow has fixed values of 5, 9, 13, 16 and 20 l/min. These are necessary input data for numerical simulations in Fluent.

The first step is to calculate velocity for volume flow values of 5, 9, 13, 16 and 20 (l/min). The following expression can be used for calculation:

$$\dot{V} = u \cdot A \quad \Rightarrow \quad u = \frac{\dot{V}}{A} \tag{1}$$

where u is velocity in m/s and A is area m^2.

The second step is to calculate the Reynolds number (Re) for volume flow values 5, 9, 13, 16 and 20 l/min with known velocities. The equation for calculating Reynolds number is:

$$\mathrm{Re} = \frac{\rho \cdot u \cdot d}{\mu} \tag{2}$$

where ρ is density in kg/m^3, d is diameter with value d= 0,007 m and μ is dynamic viscosity in kg/m·s.
Table 1 shows velocities values and Reynolds numbers for different values of volume flow.

Volume flow [l/min]	Velocity [m/s]	Re-number	Flow
5	2,19	235,84	laminar
9	3,96	426,461	laminar
13	5,716	615,56	laminar
16	7,037	757,830	laminar
20	8,797	947,36	laminar

Table 1. Velocities and Reynolds numbers for different values of volume flow

Based on the obtained values of velocities and Reynolds numbers, it can be concluded that laminar flow occurs. Laminar flow occurs when the calculated Reynolds number is less than 2300.
The analysis of fluid flow through the valve was performed for the "open" valve configuration and for two different meshes, tetrahedral and hexahedral. Figure 1 and Figure 2 show the geometry and tetrahedral mesh "open" configuration.

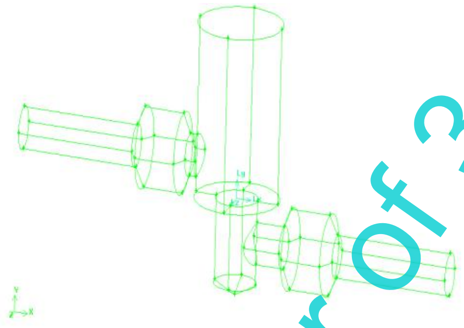

Fig. 1. Geometry

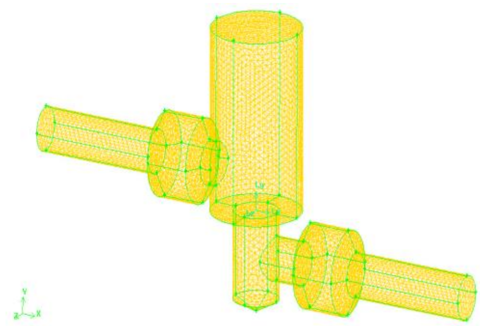

Fig. 2. Tetrahedral mesh "open" configuration

Figure 3 shows the quality of the mesh with a total number of elements of 91.000 tetrahedral cells. Figure 4 shows the position of the worst element and a quality value of 0,8037.

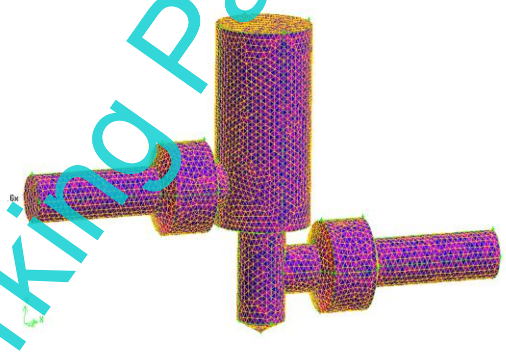

Fig. 3. Quality of tetrahedral mesh

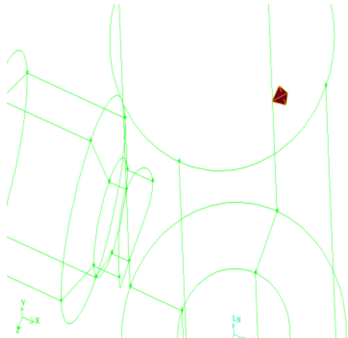

Fig. 4. Position the worst element in geometry

Figure 5 and Figure 6 show the geometry and hexahedral mesh "open" configuration. Geometry with hexahedral mesh is possible and complex.

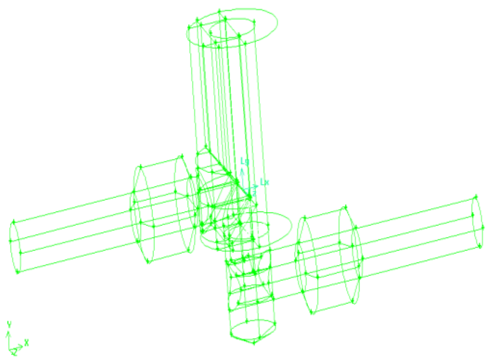

Fig. 5. Complex geometry

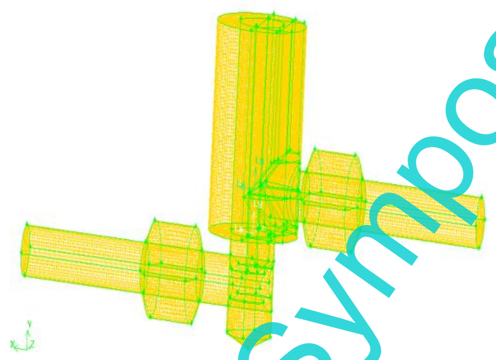

Fig. 6. Hexahedral mesh "open" configuration

Figure 7 shows the quality of the mesh with a total number of elements of 151.894 hexahedral cells. Figure 8 shows the position of the worst element and the quality value of 0,7858.

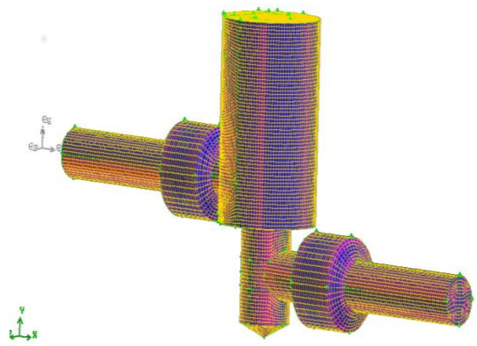

Fig. 7. Quality of hexahedral mesh

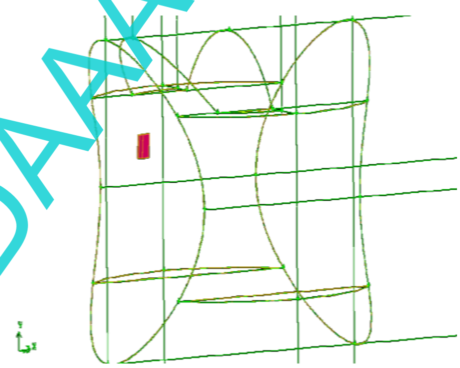

Fig. 8. Position the worst element in geometry

Figure 9 shows the contours of the velocity magnitude results in tetrahedral mesh. The minimum velocity value is 0 m/s, while the maximum is 3,58 m/s. From Figure 10, it can be seen the contours of velocity magnitude results in hexahedral mesh. The minimum value of velocity magnitude is 0 m/s, while the maximum is 3,97 m/s.

When comparing velocities magnitude, in both cases, the minimum velocity is 0 m/s, and the difference between the maximum velocities is very small, velocities are almost the same (3,58 m/s for tetrahedral mesh and 3,97 m/s for hexahedral).

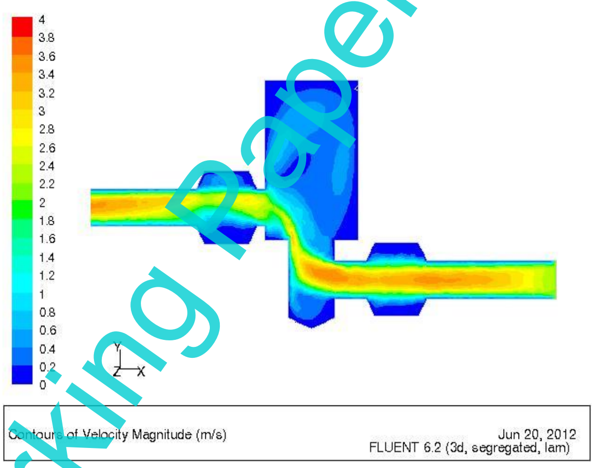

Fig. 9. Contours of velocity magnitude in tetrahedral mesh

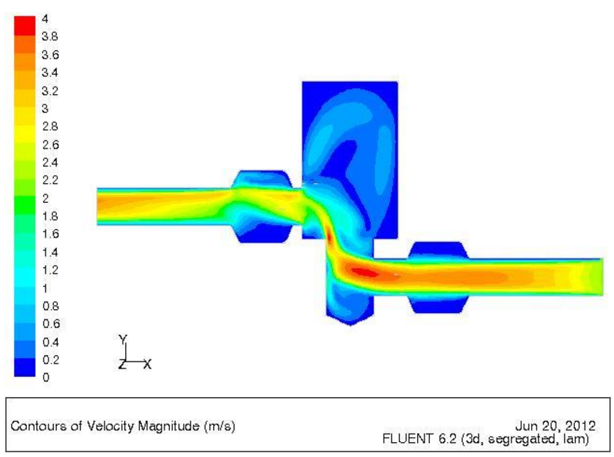

Fig. 10. Contours of velocity magnitude in hexahedral mesh

Table 2 shows the results of the mass flow rate for the inlet and outlet, the area-weighted average for static pressure in Pascal, and the area-weighted average for velocity magnitude in m/s, for two different meshes, and as it can be seen the difference between them is not significant.

		Tetrahedral (worst geometry)	Hexahedral (complex geometry)
Mass flow rate (kg/s)	Outlet	-0,07352	- 0,07450
	Inlet	0,07353	0,07450
	Net mass flow	1,9803643e-05	-6,7055225e-08
Area-weighted average static pressure (Pa)	Outlet	0	0
	Inlet	16529	15791
	Cross-section	8892	8584
Area-weighted average velocity magnitude (m/s)	Outlet	1,84	2,11
	Inlet	2,19	2,19
	Cross-section	1,34	1,36
Velocity magnitude (m/s)	Min.	0	0
	Max.	3,58	3,97
Static pressure (Pa)	Min.	0	0
	Max.	17099	16611

Table 2. Calculation results for two different meshes and volume flow 5 l/min

Figure 11 shows the diagram of pressure drop and volume flow dependencies obtained by numerical simulation in Fluent for hexahedral and tetrahedral mesh and the results of numerical simulation Solid Works Flow Simulation [11]. The green line shows the results of the hexahedral mesh, while the red line shows the results of the tetrahedral mesh. When comparing these lines, the difference is not significant. Also, the results obtained in Fluent for both meshes have a good match with the existing results from Solid Works Flow Simulation. However, the results obtained in Fluent with the hexahedral mesh have a better match with the results from Solid Works Flow Simulation.

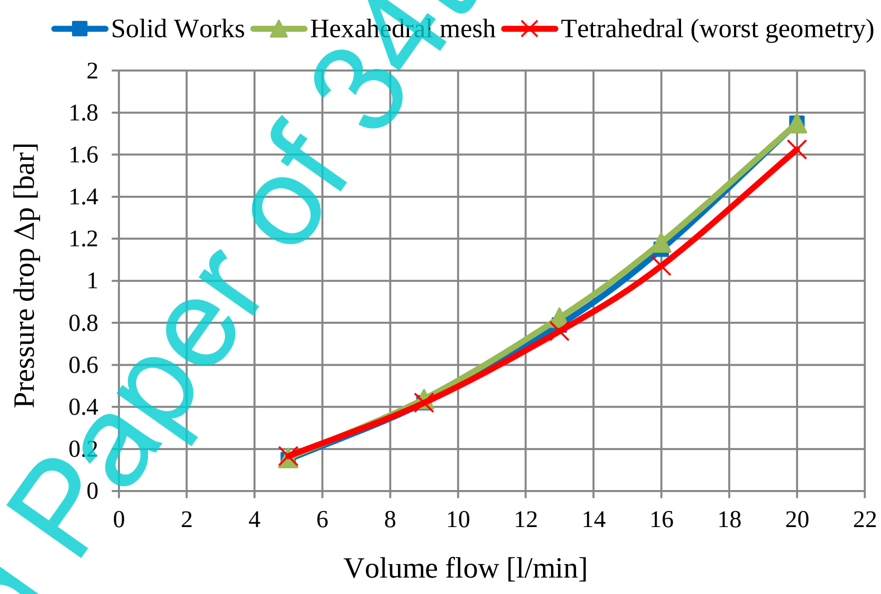

Fig. 11. Diagram of pressure drop and volume flow dependencies, comparison numerical simulation results from Fluent with existing results from Solid Works

4. Conclusion and further work

This paper provides the numerical calculation results of pressure drop obtained in Fluent for two different meshes, hexahedral and tetrahedral, for fixed volume flow values 5, 9, 13, 16 and 20 l/min through the valve. The main characteristics of the results of numerical simulations through the valve can be highlighted as:

- By comparing the results obtained for two different meshes, less deviations between them can be noticed. Minimum velocity values for both meshes are the same (0 m/s), while difference between the maximum velocity values is very small, about 9,8% (for tetrahedral mesh maximum velocity is 3,58 m/s and for hexahedral 3,97 m/s).
- When comparing results for configuration "open" valve with hexahedral mesh and tetrahedral mesh, the results of the mass flow rate for inlet and outlet, the area-weighted average for static pressure, and the area-weighted average for velocity magnitude, it can be observed that the difference between them is not significant.
- The results obtained in Fluent for both meshes have a good match with the existing results from Solid Works Flow Simulation, but the results obtained with the hexahedral mesh have a better match.

In some of the following research, as further work, other models will be developed and experimental studies will be included.

5. References

[1] Hadziahmetovic, H; Dzaferovic, E. & Cohodar, M. (2010). Description Of Technological Process Of Hydraulic Transport From Thermal Power Plant, Annals of DAAAM & Proceedings, 20-23rd October 2010, Zadar, ISSN 1726-9679, Katalinic, B. (Ed.), pp 685-686, DAAAM International Vienna Audience, Zadar, Croatia.

[2] Ferziger, J.H. & Peric, M. (1999). Computational Methods for Fluid Dynamic, Third Edition, Springer-Verlag Berlin Heidelberg, ISBN 3-540-42074-6, Germany

[3] Krieger, M.; Mehrle, A. & Thumfart, M. (2012). Practical Training in Computational Fluid Dynamics, Institute of Fluid Mechanics and Heat Transfer, Johannes Kepler University Linz, Linz, Austria

[4] Gevorkov, L.; Smidl, V. & Sirovy, M. (2018). Stepper Motor Based Model of Electric Drive for Throttle Valve, Proceedings of the 29th DAAAM International Symposium, 24-27th October 2018, Zadar, ISBN 978-3-902734-20-4, ISSN 1726-9679, Katalinic, B. (Ed.), pp.1102-1107, Published by DAAAM International, Vienna, Austria, DOI: 10.2507/29th.daaam.proceedings.157

[5] Vedova, D.; Maggiore, M.D.L. & Riva, P.G. (2017). A new CFD-Simulink based systems engineering approach applied to the modelling of a hydraulic safety relief. International Journal Mechanics 11, pp. 43–50

[6] Hadziahmetovic, H.; Hodzic, N.; Kahrimanovic, D. & Dzaferovic, E. (2014). Computational fluid dynamics (CFD) based erosion prediction model in elbows. Tehnički Vjesnik – *Technical Gazette*, 21 (2), 275–282.

[7] Simic, M. & Herakovic, N. (2015). Reduction of the flow forces in a small hydraulic seat valve as alternative approach to improve the valve characteristics. Energy Conversion Management, 89, 708–718, https://doi.org/10.1016/j.enconman.2014.10.037.

[8] Banaś, M.; Antoniak, P.; Marciniak, L. & Stryczek, J. (2018*). Visualization of flow phenomena in hydraulic throttle valves of plastics*, MATEC Web Conferences, 211, 19001, DOI: https://doi.org/10.1051/matecconf/201821119001.

[9] Szpica D. (2015). Simplified numerical simulation as the base for throttle flow characteristics designation, Mechanika, 21(2), pp. 129–133.

[10] Polášek, T.; Ledvoň, M.; Hružík, L. & Bureček. A. (2023). Numerical simulation and experimental verification of flow through throttle valve. AIP Conference Proceedings, 2672 (1): 020017, https://doi.org/10.1063/5.0119995

[11] Hodžić, N.; Spahić, D. & Skopljaković, M. (2010). Numerical simulation of hydraulic oil flow through a valve with seat area type A 10 FS1.0364, 14th International Research / Expert Conference "Trends in the Development of Machinery and Associated Technology" TMT 2010, Mediterranean Cruise, 11-18 September 2010., ISSN 1840-4944, pp. 533 - 536, University of Zenica, Faculty of Mechanical Engineering in Zenica, Bosnia and Herzegovina.

[12] Fluent Inc. FLUENT 6.3 User's Guide. Fluent Inc., 2006.