EVALUATING INFLUENCE OF CFD MESH ON FLOW CHARACTERISTICS OF PNEUMATIC BRAKE VALVE DIFFERENTIAL SECTION

Marcin Kisiel, Dariusz Szpica
Bialystok University of Technology, Poland
mail_marcin.kisiel@sd.pb.edu.pl, d.szpica@pb.edu.pl

Abstract. The selection of the nozzle diameter of the differential section of the trailer air brake valve is crucial for two-range operation of the system adopted by the designers. Using CFD tools, it is possible to determine the flow characteristics of the components with reference to the mode of operation and the valve switching times from tracking to accelerating function. Taking into account the shape of the working chambers of the differential section of the conceptual brake valve and the small effective flow area resulting from the nozzle diameter, assessment of the influence of the mesh on the flow characteristics was carried out. The use of different meshes can be a cause in the discrepancies between the results obtained. The selection of a suitable mesh can reduce computation time and provide flow characteristics comparable to the results determined on a laboratory bench. For different nozzle diameters, both in SolidWorks Flow Simulation and Ansys Fluent, the influence of variable mesh parameters - element shape and number of nodes, on the flow characteristics is compared. The influence of the mesh was assessed on the orthogonal mesh in SolidWorks and on the different types of mesh available in Fluent. Evaluation of the quality of all finite element mesh variants considered was carried out using basic statistical indicators. The determined flow characteristics will allow evaluation of the influence of the nozzle diameter on MFR (mass flow rate) of the differential section, resulting in the selection of a nozzle that meets the design and functional objectives. The selection of an appropriate finite element mesh will enable further extension of the model to include the main and auxiliary feeders, taking into account the movement of the elements (dynamic mesh), resulting in a complete numerical model of the valve, ensuring a reduction in the response time of the system to the operator's excitation in relation to solutions available on the market.

Keywords: pneumatic brake valve, CFD mesh, SolidWorks Flow Simulation, Ansys Fluent.

Introduction

The computational mesh [1] in numerical fluid mechanics (CFD), its proper selection and optimization in terms of the calculation time [2] and hardware resources, are key aspects when preparing and analysing the computational domain. The correct determination of boundary conditions and parameters associated with the calculation flow, together with a properly selected computational mesh [3], are the basis for obtaining results comparable, or at least close, to those determined experimentally. The use of numerical fluid mechanics precludes the creation and production of physical prototypes prior to the introduction of an upgrade to an existing product or a completely new pneumatic or hydraulic component. An important feature of CFD calculations [4] is the possibility of local qualitative analysis, which cannot be obtained using empirical models such as the lumped method (LM) [5].

Most CFD software uses the Finite Volume Method (FVM) [6-7]. The computational mesh allows the volume of a gas or liquid in the domain under consideration to be divided into cells of user-defined shape and density. The subdivision of the computational domain into smaller cells makes it possible, on the basis of the conservation of the mass (1) and RANS (2) - Reynolds Average Navier-Stokes equations [8], to determine selected flow parameters, such as the values of pressure, density, velocity and temperature in each cell of the computational mesh, without having to search for a general solution.

$$\frac{\partial p}{\partial t} + \nabla\left(pv\right) = 0\,, \tag{1}$$

$$p\frac{\partial v}{\partial t} = -\nabla p + pg + \mu\nabla^2 v\,. \tag{2}$$

Available numerical environments have implemented automated meshing tools, but the results are not always satisfactory, even for simple geometries. Software developers are introducing further functionality to reduce the time needed to prepare a model for meshing, but subjecting different geometries to numerical analysis still requires individual user intervention. A key element of computational fluid dynamics is becoming both the choice of the finite element shape and the degree of mesh refinement, which can be variable for different flow domains, and it has been shown from literature analysis [9] that optimal mesh refinement with appropriate values of quality indices allows more accurate results to be obtained.

The quality of the computational mesh can be assessed by the values of quality parameters such as the aspect ratio, orthogonal quality and skewness. Among the mesh evaluation criteria, the minimum assumptions are that the value of orthogonal quality should be greater than 0.1 and skewness should not exceed 0.9, while quality indicators close to the best should be sought. It should be noted that the evaluation of the above-mentioned mesh quality indices can only be applied to one of the two CFD environments considered in the study – Ansys Fluent.

Materials and methods

The subject of the CFD study was the differential section [10-11] of the Visteon brake valve modified by the authors. The calculations were carried out using two CFD numerical environments (SolidWorks Flow Simulation 2022 and Ansys Fluent 16.0) to determine and compare MFR – Mass Flow Rate values under the same boundary conditions, which enabled the static characteristics of the differential section to be plotted. A PC equipped with an AMD Ryzen 7 5800X3D processor and 32GB of DDR4 RAM memory was used for the calculations.

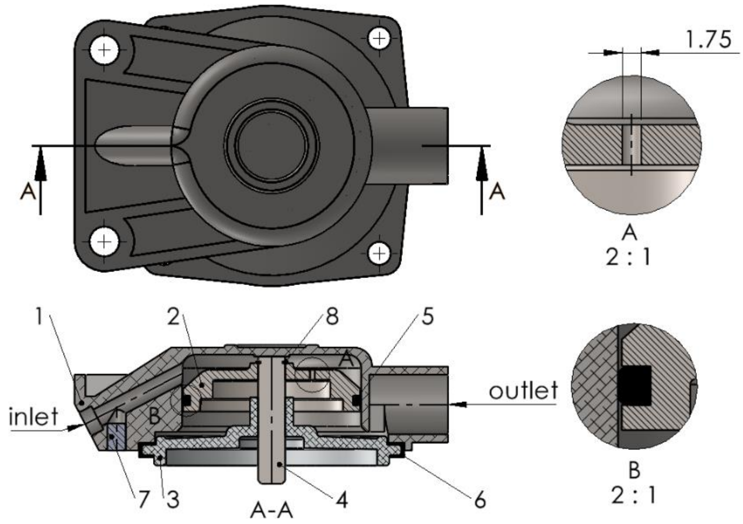

Fig. 1. **Differential section of pneumatic braking valve**: 1 – valve body; 2 – differential piston; 3 – lower piston; 4 – piston guide; 5,6 – piston seals; 7 – plug; 8 – circlip

The valve model was recreated and modified by introducing a differential section (Fig. 1) in SolidWorks CAD software, simplified and subjected to airflow simulation with assumed boundary conditions using the Flow Simulation add-on. In the second step, the model was exported to Ansys Fluent and simulated to determine MFR using the same boundary conditions but different mesh types. In order to be correctly implemented in both numeric environments, the model has been simplified:

- the nozzle has been replaced by a hole with a diameter from 0.75 mm to 5 mm in a differential piston 2 (Fig. 1);
- the o-rings on pistons 2,3 (Fig. 1) were replaced by square piston seals 5,6 (Fig. 1) with rounded edges in SolidWorks, and without roundings in Ansys Fluent CAD model due to mesh skewness higher than 0.97;
- the compressed air reservoir supply connection has been plugged using the plug 7 (Fig. 1);
- the circlip 8 (Fig.1) was replaced by a solid ring.

The research scenario in SolidWorks Flow Simulation [12] involved carrying out a static CFD simulation of the differential section for all available nozzle diameters. For this purpose, all the global mesh sizes offered by the software (1-7) were used with the simulation parameters shown in Table 1.

The results of the simulations carried out for the *k-ε* turbulence model, inlet pressure equals $p_{inlet} = 9 \times 10^5$ Pa and outlet pressure equals $p_{outlet} = 1 \times 10^5$ Pa were summarised in the static flow characteristics presented in Fig. 2.

Table 1

Parameters of internal flow of differential section in SolidWorks Flow Simulation

Analysis type	Internal
Fluid	Air
Flow type	Laminar and turbulent
Default wall thermal condition	Adiabatic wall
Pressure	p_0 = 101325 Pa
Temperature	T_0 = 293.2 K
Turbulence intensity	2%
Turbulence length k	0.0010745 m

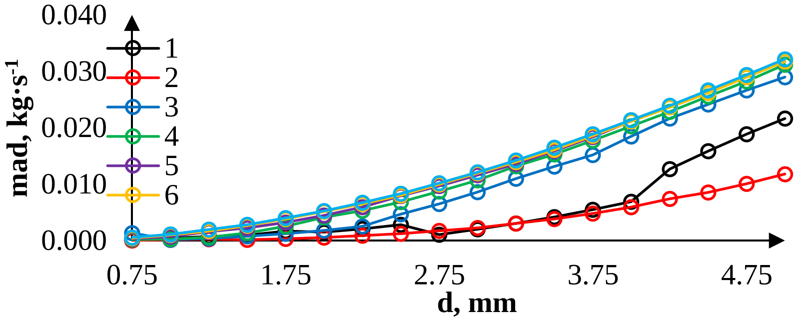

Fig. 2. **Flow characteristics of differential section using global meshes:** 1-7 mesh size

The next step involved the use of a global mesh of size 6 to optimise the computation time and a local mesh in the hole volume responsible for the differential action of the valve. For this purpose, built-in tools were used to locally compact the mesh in the hole volume, according to the parameters shown in Table 2.

Table 2

Parameters of local meshes using different combinations of refinement tools

Local mesh		**Equidistant refinement**		**Advanced refinement**		**Channels**	
Level of refining cells (1-9)	3	Number of shells	1	Small solid feature refinement level (1-9)	3	Number of cells across channel	10
Level of refining cells at fluid boundary (1-9)	3	Maximum equidistant level (1-9)	2	-	-	Maximum channel refinement level (1-9)	2
-	-	Offset distance	0.0002 5 m	-	-	-	-

Based on the obtained results, static flow characteristics were determined using a local mesh as shown in Fig. 3.

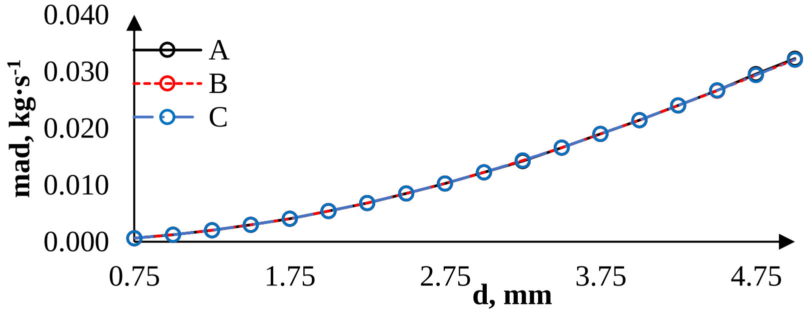

Fig. 3 **Flow characteristics of differential section using local meshes:** A – equidistant and advanced refinement; B – channels; C - refinement and channels

An example of the global mesh distribution in size 7 can be seen in Fig. 4a, while a view of the local mesh in the key area is shown in Fig. 4b. In both cases, this distribution was presented for a 1.75 mm diameter hole (nozzle).

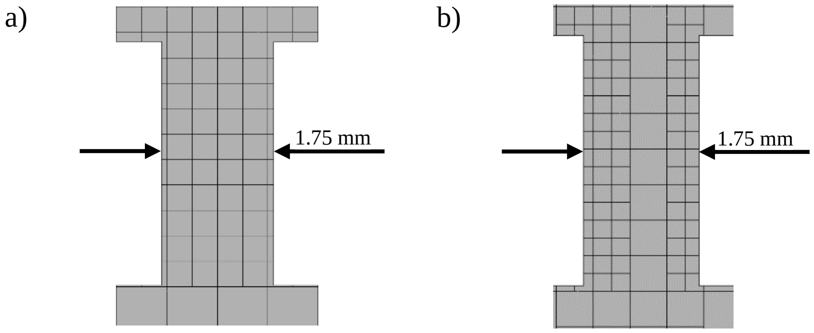

Fig. 4. **Distrubution of meshes (Ø1.75 mm):** a – global; b – local using refinement tools

The procedure path in Ansys Fluent required additional model preparation. Before starting the definition of the boundary conditions, the gas volume was extracted [13] and then the supply $p_{inlet} = 9\times10^5$ Pa and outlet $p_{outlet} = 1\times10^5$ Pa pressure boundary conditions were defined. The simulation was carried out for the five mesh variants shown in Table 3. The number of iterations in each case was limited to 200.

Table 3

Parameters of internal flow of differential section in Ansys Fluent

Name	I	II	III	IV	V
Turbulence model	*k-ε*				
Advanced size function	Proximity and curvature				
Relevance center	Medium				
Smoothing	Medium				
Min size	Default	Default	0.1 mm	0.1 mm	0.1 mm
Proximity min size	Default	Default	0.1 mm	0.1 mm	0.1 mm
Max face size	Default	Default	0.5 mm	0.5 mm	0.3 mm
Max size	Default	Default	0.5 mm	0.5 mm	0.3 mm
Growth rate	1.20				
Wall function	Standard Wall Function				
Method	Tetrahedrons	Automatic	Tetrahedrons	Automatic	Tetrahedrons
Polyhedra domains	No	No	No	Yes	Yes

Based on the results obtained, as in the previous cases, the static characteristics were determined and presented in Fig. 5.

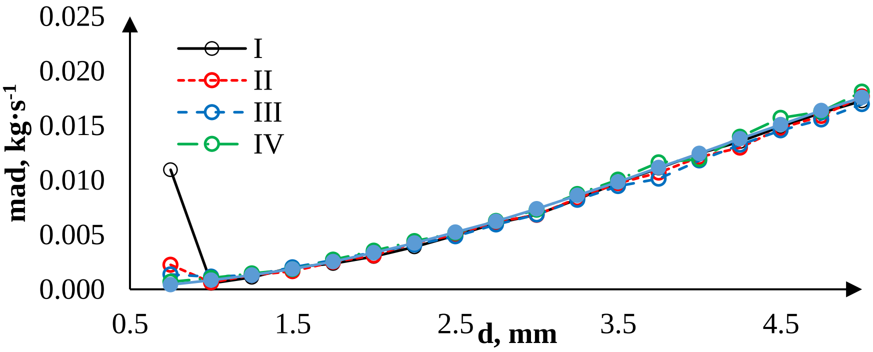

Fig. 5. **Flow characteristics of differential section using different meshes in Ansys Fluent:** I – tetrahedrons; II – automatic refinement; III – 0.1 mm to 0.5 mm edge length; IV – polyhedral mesh; V – 0.1 mm to 0.3 mm edge length and polyhedral domain

The model mesh distribution using a tetrahedral mesh for a 1.75 mm diameter hole is presented in Fig. 6a. The polyhedral mesh that was created after converting the computational domain into polyhedral is shown in Fig. 6b.

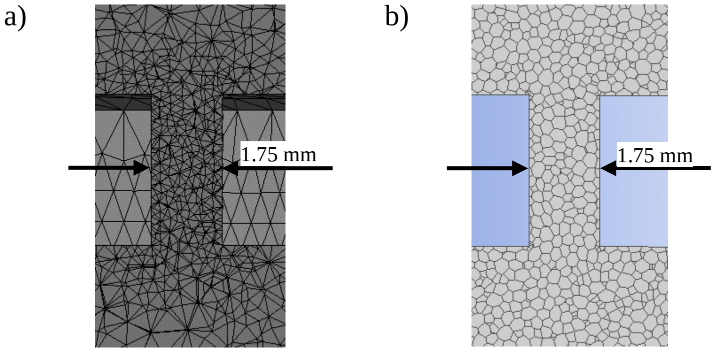

Fig. 6 **Distribution of meshes (Ø1.75 mm):** a – tetrahedrons; b - polyhedral

Results and discussion

The use of local meshes in SolidWorks Flow Simulation resulted in a six-fold reduction in the calculation time compared to the global mesh model in size 7. The results of the determined MFR value for the local grid were 0.22% lower (5.0 mm nozzle) than the value calculated using the global grid, but for the other nozzle diameters the MFR values from the local grid were higher from 0.15% to 12.40%, compared to the model with denser global grid.

The internal flow simulation in Ansys Fluent requires the generation of the fluid volume during the import of the model geometry. The high mesh skewness of the original model required a modification of the geometry in sensitive areas, which ensured that a maximum mesh skewness of about 87% was obtained for 1% of the model volume, while the average skewness varied around 40%. The value of the orthogonal quality ratio varied between 0.15 and 0.86, while the aspect ratio varied between 1.15% and 17.27%, depending on the grid settings adopted. The determined grid parameters are within the ranges specified by the manufacturer of Ansys Fluent software and confirmed in the literature.

The MFR values obtained using the polyhedral mesh in Ansys Fluent with node edges in the range 0.1 mm to 0.3 mm were lower, ranging from 32.51% to 45.23% compared to the results obtained with the compacted local mesh in SW Flow Simulation. The use of the polyhedral mesh in Ansys Fluent for the considered geometry resulted in an average fourfold reduction of the computation time.

Flow disturbances for the 0.75 mm bore valve variant resulted in significant discrepancies when determining MFR values using Ansys Fluent.

Considering the case of the Ansys Fluent mesh without the use of additional compaction tools with a node edge in the range 0.1 mm to 0.5 mm and the polyhedral mesh based on these settings, flow disturbance resulted in a decrease in MFR values of 9% and 50% for the hole diameters (0.75, 1.00 and 1.5 mm), while in the other cases the polyhedral mesh resulted in an increase in the determined MFR value of 1-15%.

The obtained flow characteristics in SW Flow Simulation were close to linear, whereas the characteristics determined from the results with Ansys Fluent are characterised by non-linearity due to the limited number of iterations - Ansys Fluent 200 iterations, Flow Simulation 300-650, depending on the case.

Conclusions

1. The results obtained showed a large variation depending on the software used. Mesh densification using the built-in SW Flow Simulation tools resulted in an increase in the determined MFR value in the range of 0.15% to 12.40%.
2. In the case of Ansys Fluent, mesh densification and the use of polyhedral domains resulted in an increase of the MFR in the range of 1% to 15%, while in some cases this led to an underestimation of the results after using this method.

3. A comparison of the results obtained from the densest meshes considered shows an underestimation of the MFR in Ansys Fluent of 32.51% to 45.23% compared to the values obtained in SW Flow Simulation.
4. The results provide a starting point for the laboratory bench experiments planned in the next steps to determine the influence of the nozzle diameter on the differential function of the valve. The changes in performance characteristics under real conditions determined in this way will make it possible to assess the validity of the use of the differential section in the acceleration action of the valve. The results of the experimental tests will make it possible to evaluate the discrepancy between the numerical and experimental values of MFR.

Acknowledgements

This research was financed through subsidy of the Ministry of Science and Higher Education of Poland for the discipline of mechanical engineering at the Faculty of Mechanical Engineering Bialystok University of Technology WZ/WM-IIM/5/2023.

Author contributions

Conceptualization, M.K and D.S.; methodology, M.K and D.S.; formal analysis, M.K and D.S.; writing – original draft preparation, M.K.; writing – review and editing, D.S.; funding acquisition, D.S. All authors have read and agreed to the published version of the manuscript.

References

[1] Pisarciuc C., Dan I., Cioară R., The Influence of Mesh Density on the Results Obtained by Finite Element Analysis of Complex Bodies. Materials, vol. 16, p. 2555, Sep. 2023, DOI: 10.3390/ma16072555.
[2] Jurkowski S. Janisz K. Analiza wpływu parametrów siatki obliczeniowej na wynik symulacji przepływomierza. (Analysis of the influence of computational mesh parameters on the flowmeter simulation result). AUTOBUSY. Eksploatacja i testy, vol. 12, 2019, DOI: 10.24136/atest.2019.238. (In Polish).
[3] Sencic T., Mrzljak V., Bukovac O., Batista J. Influence of Mesh and Combustion Parameters on a Spark Ignition Engine CFD Simulation. FME Transactions, vol. 51, Mar. 2023, pp. 374-385. DOI: 10.5937/fme2303374S.
[4] Filo G., Lisowski E., Rajda J. Design and Flow Analysis of an Adjustable Check Valve by Means of CFD Method. Energies (Basel), vol. 14, p. 2237, Jul. 2021, DOI: 10.3390/en14082237.
[5] Kaminski Z. A simplified lumped parameter model for pneumatic tubes. Math Comput Model Dyn Syst, vol. 23, no. 5, Sep. 2017, pp. 523-535. DOI: 10.1080/13873954.2017.1280512.
[6] Jing F., Han W., Kashiwabara T., Yan and W. On Finite Volume Methods for a Navier–Stokes Variational Inequality. J Sci Comput, vol. 98, Mar. 2024, DOI: 10.1007/s10915-023-02408-x.
[7] Schäfer M. Finite-Volume Methods for Incompressible Flows, 2022, pp. 247-282. DOI: 10.1007/978-3-030-76027-4_11.
[8] Basara B., Krajnović S., Pavlovic Z., Ringqvist P. Performance analysis of Partially-Averaged Navier-Stokes method for complex turbulent flows, Sep. 2011. DOI: 10.2514/6.2011-3106.
[9] Aqilah F., Islam M., Juretic F., Guerrero J., Wood D., Ani F. Study of Mesh Quality Improvement for CFD Analysis of an Airfoil. IIUM Engineering Journal, vol. 19, Mar. 2018, pp. 203-212. DOI: 10.31436/iiumej.v19i2.905.
[10] Kisiel M., Szpica D., Czaban J., Koten H. Pneumatic brake valves used in vehicle trailers – A review, Eng Fail Anal, vol. 158, p. 107942, Mar. 2024, DOI: 10.1016/j.engfailanal.2023.107942.
[11] Mystkowski A., "Zastosowanie zaworów różniczkujących w pneumatycznych układach napędowych,"(The use of differential valves in pneumatic drive systems). Pneumatyka, vol. 3, 2004, pp. 21-23. (In Polish).
[12] Salmat S., Sari D., Fernanda Y., Prasetya F. SolidWorks Flow Simulation: Selecting the optimal mesh for conducting CFD analysis on a centrifugal fan. Journal of Engineering Researcher and Lecturer, vol. 2, Mar. 2023, pp. 52-61. DOI: 10.58712/jerel.v2i3.104.
[13] Dongpo W., Hongxia Z. Research on Flow Characteristics of Throttle Valve Based on Fluent. J Phys Conf Ser, vol. 1676, p. 12021, Mar. 2020, DOI: 10.1088/1742-6596/1676/1/012021.