

# Modeling flow ratios of the slide gate valve in hydraulic and pneumatic piping systems

Lubomíra KMEŤOVÁ<sup>1</sup>\* and Marek MORAVIČ<sup>2</sup>

## Authors' affiliations and addresses:

<sup>1</sup> Faculty of Mechanical Engineering, Technical University of Košice, Department of Energy Engineering, Vysokoškolská 4, 042 00 Košice, Slovak Republic  
e-mail: lubomira.kmetova@tuke.sk

<sup>2</sup> Faculty of Mining, Ecology, Process Control and Geotechnology, Technical University of Košice, Park Komenského 14, 042 00 Košice, Slovak Republic  
e-mail: marek.moravic@tuke.sk

## \*Correspondence:

Lubomíra Kmeťová, Faculty of Mechanical Engineering, Technical University of Košice, Department of Energy Engineering, Vysokoškolská 4, 042 00 Košice, Slovak Republic  
tel.: +421 55 602 4361  
e-mail: lubomira.kmetova@tuke.sk

## Funding information:

This article was supported by the projects VEGA 1/0728/24 and APVV-23-0342.

## Acknowledgement:

This article was funded by the Slovak Research and Development Agency under contract No. APVV-23-0342 "New structural and material elements for sustainable transport of bulk materials" and VEGA 1/0728/24 "Research on the possibilities of implementing sensory technologies and visualization methods for the needs of digital transformation of pipeline conveyor systems".

## How to cite this article:

Kmeťová, L. and Moravič, M. (2025). Modeling flow ratios of the slide gate valve in hydraulic and pneumatic piping systems. *Acta Montanistica Slovaca*, Volume 30 (4), 864-874

## DOI:

<https://doi.org/10.46544/AMS.v30i4.03>

## Abstract

This study investigates the influence of the slide gate valve's closure degree on hydraulic performance parameters, specifically the loss coefficient and pressure losses during water flow. The research combines analytical calculations with numerical simulations using Computational Fluid Dynamics (CFD) in ANSYS CFX to evaluate flow behaviour at three valve opening positions: 25 %, 50 %, and 75 %. The selected valve model, IKO-Plus – DN50 – PN-R14, was analysed under varying inlet velocities ( $0.5 \text{ m}\cdot\text{s}^{-1}$ ,  $1.0 \text{ m}\cdot\text{s}^{-1}$ , and  $2.0 \text{ m}\cdot\text{s}^{-1}$ ) to assess its behaviour in a hydraulic pipeline system. The results demonstrate that increasing the closure degree leads to a significant rise in pressure losses and loss coefficients, primarily due to flow acceleration, vortex formation, and turbulence in the narrowed sections of the valve. These findings highlight the critical impact of valve geometry and operating conditions on system efficiency and component wear. CFD simulations provided detailed visualizations of velocity streamlines and pressure distributions, revealing the formation of low-pressure zones and intense vortex structures downstream of the valve. The study confirms a high level of agreement between analytical and numerical approaches, validating the accuracy of the simulation model. The insights gained from this research can be applied to optimize gate valve design, improve material selection, and enhance the reliability of hydraulic and pneumatic systems. The methodology is also adaptable for analysing other types of valves and fittings exposed to similar flow conditions.

## Keywords

Slide valve, gate valve, fluid flow, water.



© 2025 by the authors. Submitted for possible open access publication under the terms and conditions of the Creative Commons Attribution (CC BY) license (<http://creativecommons.org/licenses/by/4.0/>).

## Introduction

In modern hydraulic and pneumatic pipeline systems, slide gate valves serve an important regulatory and safety function. These components allow the control of fluid and gas flow through pipelines while ensuring a secure system closure in the event of shutdown or failure. Particularly in high-pressure systems with large flow rates, the proper function of a slide valve depends not only on its mechanical durability but also on understanding the complex flow phenomena occurring within it (Žic et al., 2020).

The study by Žic et al. (2020) demonstrated that even with partial valve opening, intense flow zones with increased velocity, pressure gradients, and turbulent vortices emerge, which can lead to dynamic stress on the slide valve, erosion, and a reduction in its regulatory properties. The flow is nonlinear, locally accelerated, and accompanied by irregular pressure and turbulence distribution.

Beyond liquid environments, slide valves are often exposed to more complex gas-solid two-phase flows, such as in pneumatic transport systems. As shown in the work of Ma et al. (2020), the presence of solid particles in the gas significantly alters the flow: particles cause turbulent disturbances, wall impacts, friction, and erosion, leading to reduced system lifespan and reliability. An important phenomenon is also the formation of vortices and secondary flows due to geometric irregularities, such as the groove at the lower part of the slide valve, which is often the site of the most intense wear.

The groove, commonly located at the lower part of the slide valve, creates a low-pressure and reduced velocity zone within the flow. These areas not only induce vortex formation and particle deposition but also represent a site of increased solid particle impact against the wall. The result is localized wear, material loss, and decreased sealing capability. Additionally, prolonged exposure leads to the accumulation of triangular-shaped particles, which can secondarily affect flow dynamics (Ma et al., 2020).

Recent studies have further emphasized the importance of understanding valve-induced flow dynamics. Javanshir et al. (2015) developed a nonlinear self-excited vibration model for sliding gate dams, showing that fluid-induced vibrations are governed by reduced velocity and damping parameters, and that traditional linear models fail to capture the complex oscillatory behavior of gate structures. Their work highlights the need for dynamic modeling that accounts for non-harmonic and multi-frequency responses, especially under turbulent flow conditions.

A team of authors (Stosiak et al., 2023) focused on identifying mechanical vibrations occurring in the operating environment of hydraulic systems used in marine engineering, specifically the impact of periodic vibrations on changes in the amplitude-frequency spectrum. To achieve this, they employed a modified mathematical model, which was then compared with the experimental results. Their result was a proposed method for limiting the transmission of vibrations to the directional valve body.

Hydrodynamic load characteristics and structural response under flow-induced vibrations are mainly influenced by inertia, damping, elastic and geometric properties, as well as hydraulic parameters (Mehrzhad et al., 2015).

In the study by Mehrzhad et al. (2015), an analytical and numerical method was developed to simulate the dynamic and vibrational behavior of sliding gate valves in response to flow excitation. The simulation results show that, under optimal design requirements for the valves, the vibration amplitude can be reduced on average by up to 30 %.

Gonzalez-Trejo et al. (2024) conducted an experimental analysis of sliding-gate valves in submerged entry nozzles (SENs) used in steel casting, revealing that valve obstruction levels significantly affect jet turbulence and flow distribution. Their findings are particularly relevant for industrial applications where flow uniformity impacts product quality and system stability.

Additionally, research (Javanshir et al., 2015) explored fluid-elastic instabilities in gate valves, confirming that valve geometry and flow conditions can trigger complex vibrational modes. These insights are crucial for predicting wear and ensuring long-term reliability in high-pressure systems.

Therefore, studying flow conditions in slide valves is essential not only from the perspective of hydraulic efficiency but also in terms of operational safety. Computational Fluid Dynamics (CFD) provides tools for accurate numerical simulations of these complex phenomena (Sozinando et al., 2023).

Methods based on turbulence models (such as the Reynolds Stress Model – RSM) and particle tracking (Lagrangian approach) allow for the analysis of pressure distribution, velocity, particle trajectories, and wear prediction (Ma et al., 2020).

The aim of this article is to numerically analyse the flow conditions in a selected type of gate valve exposed to the flow of clean water in a hydraulic system. The study focuses on identifying critical areas, quantifying the influence of geometry on the distribution of physical parameters, and assessing the potential wear of the valve's inner walls. The findings will serve as a basis for optimizing the gate valve design and selecting materials depending on the type of flowing medium.

## Materials and methods

### The gate valve description

Gate valves, together with other shut-off and control valves, represent an important element in piping systems used to transport various types of fluids. Gate valve is also a term used in different contexts and meanings, depending on the field it is in. In construction, technology, and engineering, it can refer to various components and devices.

They can be used (Company Atria Europe, 2025):

1. Ventilation and air conditioning: In these industries, a gate valve is often used to refer to an adjustable flap device or closure that can be used to adjust the flow of air or other fluids in a pipe.
2. Plumbing systems: Water and sewage installations may include a gate valve as a valve used to regulate the flow of water or other fluids.
3. Fluid and gas control: In industry, a gate valve is used to regulate or shut off the flow of various fluids and gases in pipes or tubes.
4. Laboratory applications: In laboratory environments, a gate valve can be part of experimental equipment that is used to regulate the flow of fluids or gases.
5. Solids Control: Some valves may be designed to control the flow of solids, such as powders or particles.
6. Automotive: In cars, gate valves may be found in systems that provide flow for liquids or gases, such as cooling systems or fuel systems.

The gate valve consists of several basic parts (Fig. 1) with the following functions of the individual parts (Little P.Eng., 2025):

- a) The valve body houses the internal valve parts and provides the passage for fluid flow.
- b) The bonnet or cover is fastened to the valve body to complete the pressure-retaining shell. In case of gate, globe, stop check, and diaphragm valves, it contains an opening for the valve stem to pass through.
- c) The valve stem imparts the required motion to the disc for opening or closing the valve. It is connected to the valve handwheel, actuator, or the lever at one end and the valve disc on the other.
- d) The gate or disc is the part which allows, throttles, or stops flow, depending on its position.
- e) Stem sealing performs one or both of the following two functions, depending on the application:
  - Prevent leakage of flow medium to the environment.
  - Prevent outside air from entering the valve in vacuum applications.

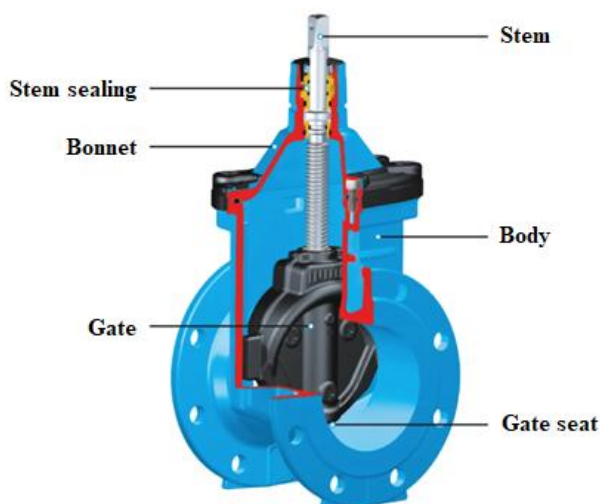


Fig. 1. The major parts of gate valve (IPL group, 2025).

For the analysis of flow conditions, determination of the loss factor, and evaluation of construction parameters such as force and moment, the IKO-Plus – DN50 – PN-R14 gate valve from the Czech company VAG s.r.o. was selected (Fig. 2). It is a metal-sealing gate valve with a non-rising stem, designed for use in hot water, steam, or non-aggressive media distribution systems. This model is designed to provide reliable shut-off and long service life in industrial applications (VAG s.r.o., 2025).

Main features of the selected gate valve (VAG s.r.o., 2025):

- Metal-sealing gate valve according to EN 1171.
- Nominal diameter (DN): 50 mm.
- Nominal pressure (PN): R14.
- Cast iron construction with stainless steel seat rings for universal use.
- Flanged ends according to EN 1092-2.
- O-ring rubber (fluor elastomer) sealing, asbestos-free, maintenance-free, resistant to high temperatures and aggressive media.
- Operated by a handwheel.
- Stem thread inside the valve chamber, stem forged from a single piece with cold-rolled thread.
- Maintenance-free stem packing.
- Pressed seats of the body and wedge.

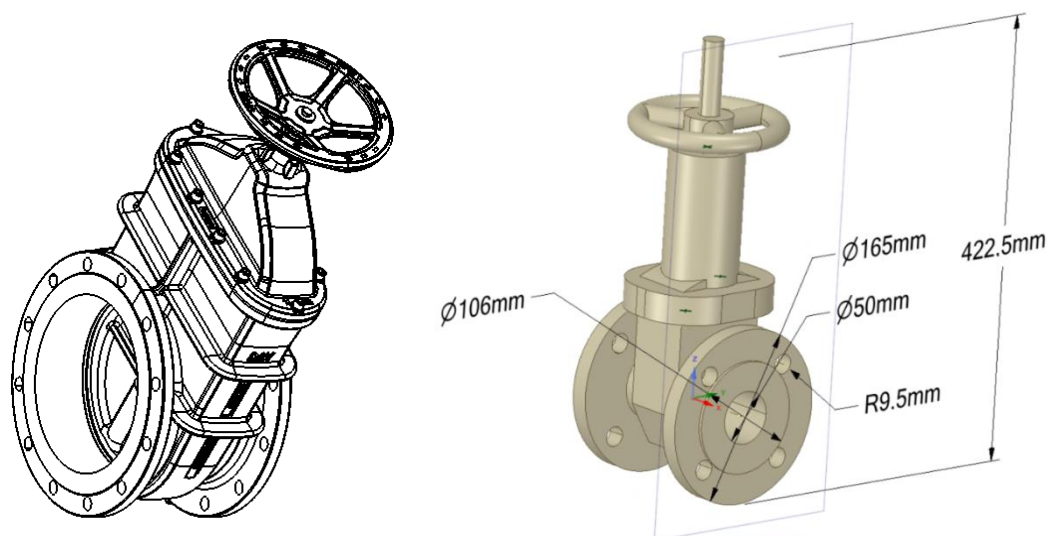


Fig. 2. The slide valve IKO-Plus – DN50 – PN-R14. On the right, a drawing of the slide valve in ANSYS-SpaceClaim (VAG s.r.o., 2025).

### Loss and design factor in a gate valve

The loss factor is a quantity that characterizes pressure losses in a flowing medium due to resistance in the piping system or fittings. Pressure losses in hydraulic systems are caused by several factors, including (VAG s.r.o., 2025 and Tech-science, 2020):

- Friction losses – occur due to the viscosity of the fluid and the roughness of the pipe walls.
- Local losses – caused by changes in the geometry of the piping, such as bends, valves, contractions, or expansions of the cross-section.
- Losses due to turbulence – arise from sudden changes in flow direction or obstacles within the piping.
- Losses due to cross-section changes – sudden narrowing or widening of the pipe can lead to additional pressure losses.

Each of these factors is important in calculating the total hydraulic resistance and energy losses in the system.

When fluid flows through the gate valve, the flow cross-section narrows, causing an acceleration of the stream, followed by its expansion beyond the valve. At this point, the flow may detach from the pipe walls (Fig. 3), leading to vortex formation and an increase in pressure losses (Čarnogurská et al., 2021).

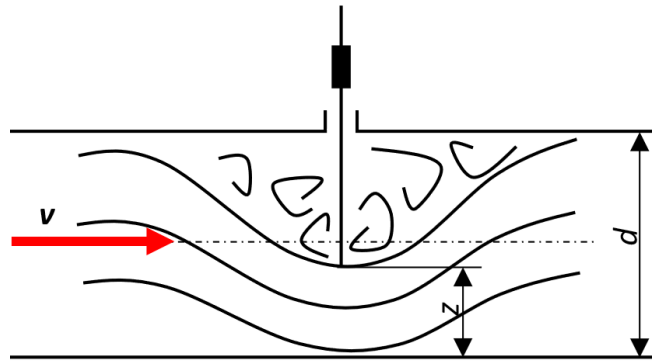


Fig. 3. Fluid flow through the gate valve (Čarnogurská et al., 2021).

The rate of losses is determined using the loss coefficient  $\zeta$  (Fig. 4), which depends on (Young, 2025):

- The relative lift  $z/d$ , i.e., the ratio of the lift height to the pipe diameter.
- The Reynolds number  $Re$ , meaning that the higher the  $Re$ , the smaller the effect of friction may be.

Experimental measurements show that at a small gate valve lift, significant losses occur due to the narrow flow channel and vortex formation. At a greater lift, the flow stabilizes, and the losses decrease.

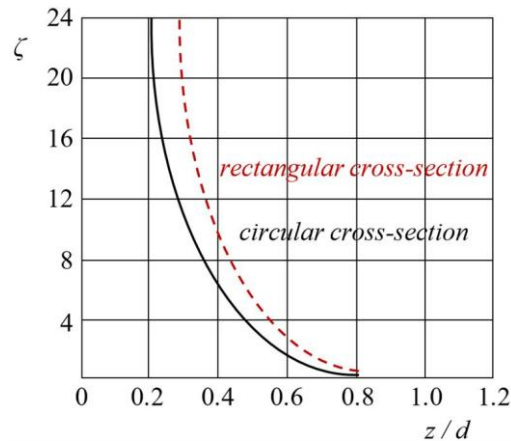


Fig. 4. Loss coefficient in the gate valve (Čarnogurská et al., 2021).

The loss coefficient for the gate valve varies depending on its lift and the specific geometry of the flow profile. It is an important parameter in assessing pressure losses occurring during fluid flow through the valve, and its analysis allows for a better understanding of the hydraulic behaviour of this component within the pipeline system.

## Calculations and results

### Solution of Flow Conditions in the Gate Valve in ANSYS (CFX)

As previously mentioned, the IKO-Plus – DN50 – PN-R14 gate valve was selected for the analysis of flow conditions, determination of the loss factor, and evaluation of construction parameters. The geometry of the computational model was rendered for three different wedge positions within the valve, corresponding to an opening of the flow cross-section at 75 %, 50 %, and 25 %. A simplified diagram of the wedge position within the valve when opened in the pipeline is shown in Fig. 5. The diagram illustrates the actual displacement values of the valve wedge at various opening positions along the axis of the pipeline.

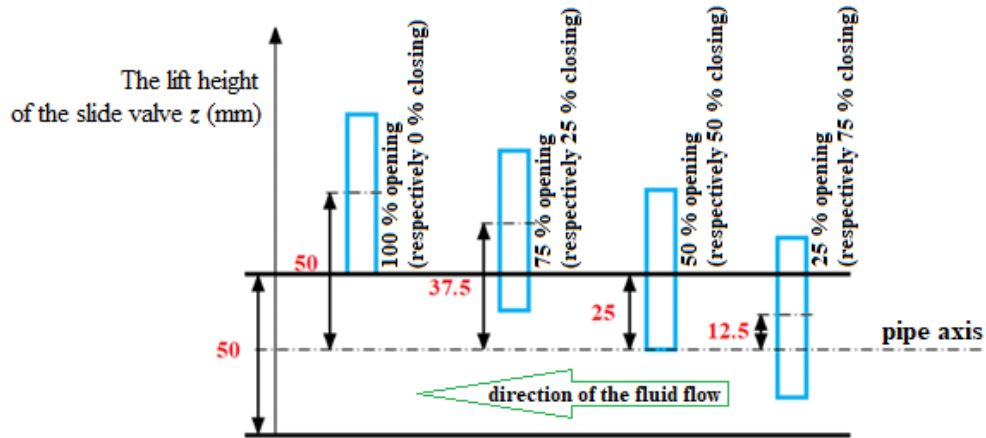


Fig. 5. Representation of the gate valve wedge in the analyzed opening positions (Kušnir, 2025).

Some dimensions in the rendered geometry were simplified compared to real values for numerical analysis. In model creation, only the internal geometry of the pipeline and gate valve was depicted, representing the space through which water flows. This approach allows for a more precise analysis of flow conditions and pressure losses while eliminating unnecessary structural details that would increase computational complexity without affecting simulation results.

The geometry was constructed as a "fluid" region – essentially a hollow space through which the flowing medium, water, passes, with the wedge area at a given opening deducted from it. The start and end of the selected pipeline section, where pressure is measured before and after the valve installation, define the geometric boundary conditions: inlet and outlet.

The same process was applied to all wedge opening positions.

3D Computational Grid Settings:

- The grid was refined in areas critical for calculation accuracy.
- A higher number of cells were added in boundary layers around all walls.

The simulation, performed using ANSYS CFX, provides a detailed view of pressure conditions within the gate valve at different closure levels (25 %, 50 %, 75 %) and varying inlet velocity values ( $0.5 \text{ m}\cdot\text{s}^{-1}$ ,  $1.0 \text{ m}\cdot\text{s}^{-1}$ , and  $2.0 \text{ m}\cdot\text{s}^{-1}$ ) for water as the flowing medium.

For clarity, Fig. 6 shows the dependence of the pressure change on the water flow velocity in the pipe through the gate valve at 75 % closure and a velocity of  $2.0 \text{ m}\cdot\text{s}^{-1}$ .

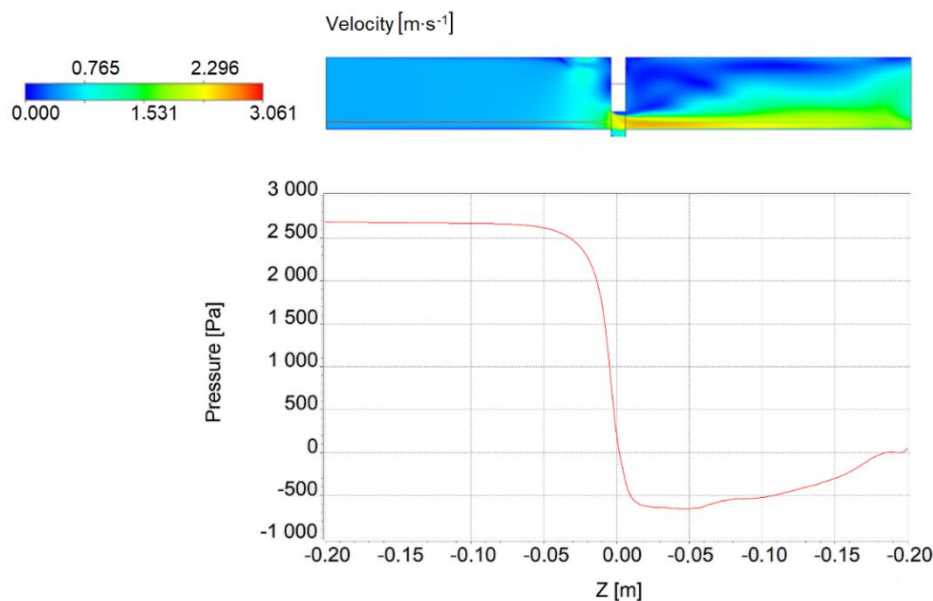


Fig. 6. Plotting the dependence of pressure change on water flow velocity in the pipeline through the gate valve at 75 % closure and a velocity of  $2 \text{ m}\cdot\text{s}^{-1}$ .

Based on the simulation, it can be concluded that with increasing closure degree and higher flow velocity, there are more significant pressure losses, a pressure rise in front of the gate valve (0.2 m), and the formation of low-pressure regions with stronger vortex structures behind the wedge (0.2 m).

At lower degrees of closure, the flow transitions more smoothly, with only mild vortex structures forming around the gate valve. As the closure increases, the flow accelerates in the narrowed regions, leading to stronger vortices and higher energy losses.

The streamline distribution in the pipe profile also shows (Fig. 7) that at lower velocities, the flow remains stable, whereas at higher velocities and greater closure, the flow becomes significantly disrupted, resulting in increased pressure losses. These changes reflect the influence of the gate valve geometry on hydrodynamic parameters.

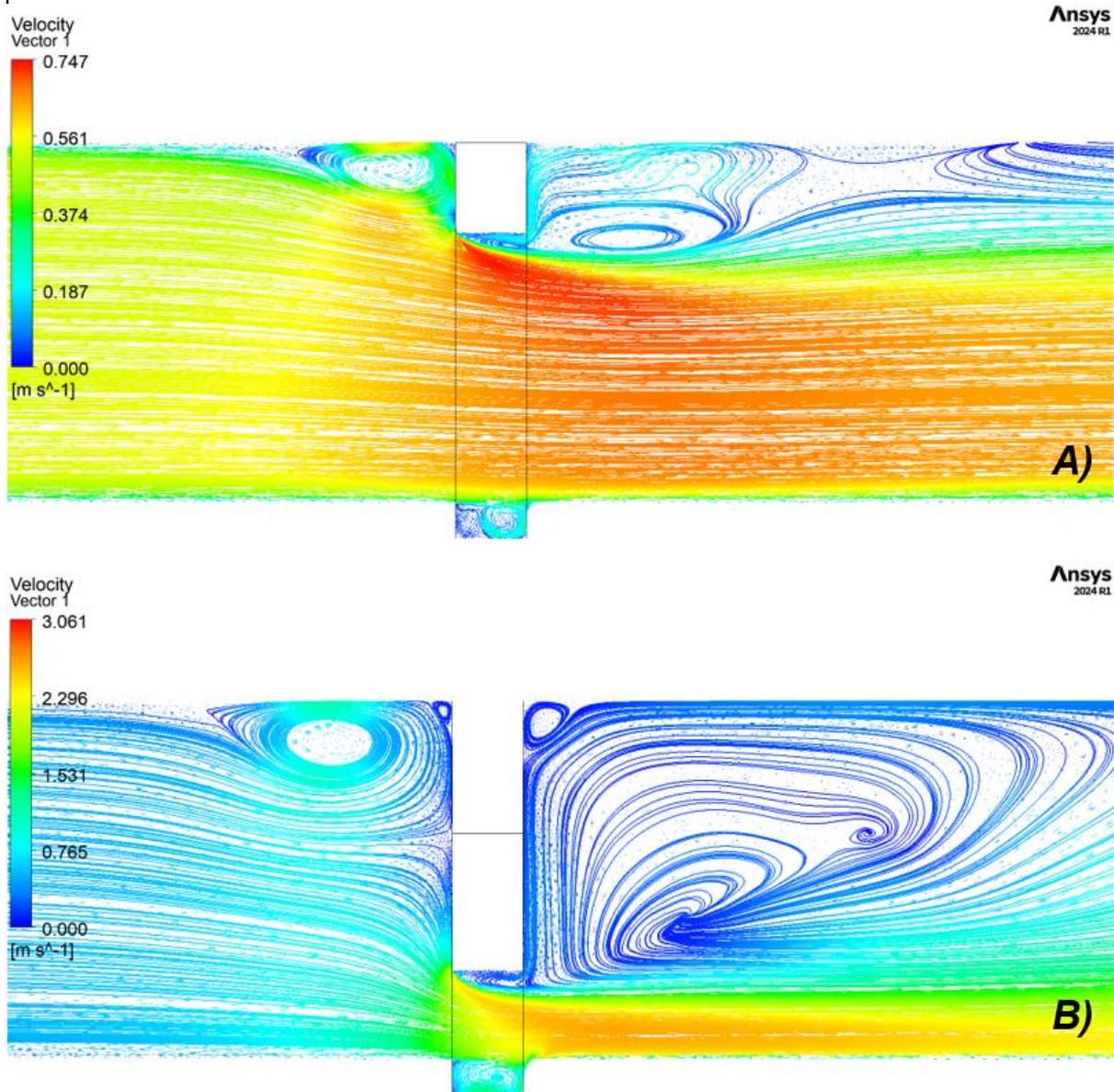


Fig. 7. Visualization of velocity streamlines during water flow at a velocity of  $0.5 \text{ m}\cdot\text{s}^{-1}$  in the pipeline with the flow cross-section closed by the gate valve to 25 % (A) and 75 % (B).

For comparison, velocity streamlines during water flow in the pipeline with the flow cross-section closed by the gate valve to 75 % are shown in the figure below (Fig. 8):

- A. Flow at a velocity of  $0.5 \text{ m}\cdot\text{s}^{-1}$ .
- B. Flow at a velocity of  $2.0 \text{ m}\cdot\text{s}^{-1}$ .

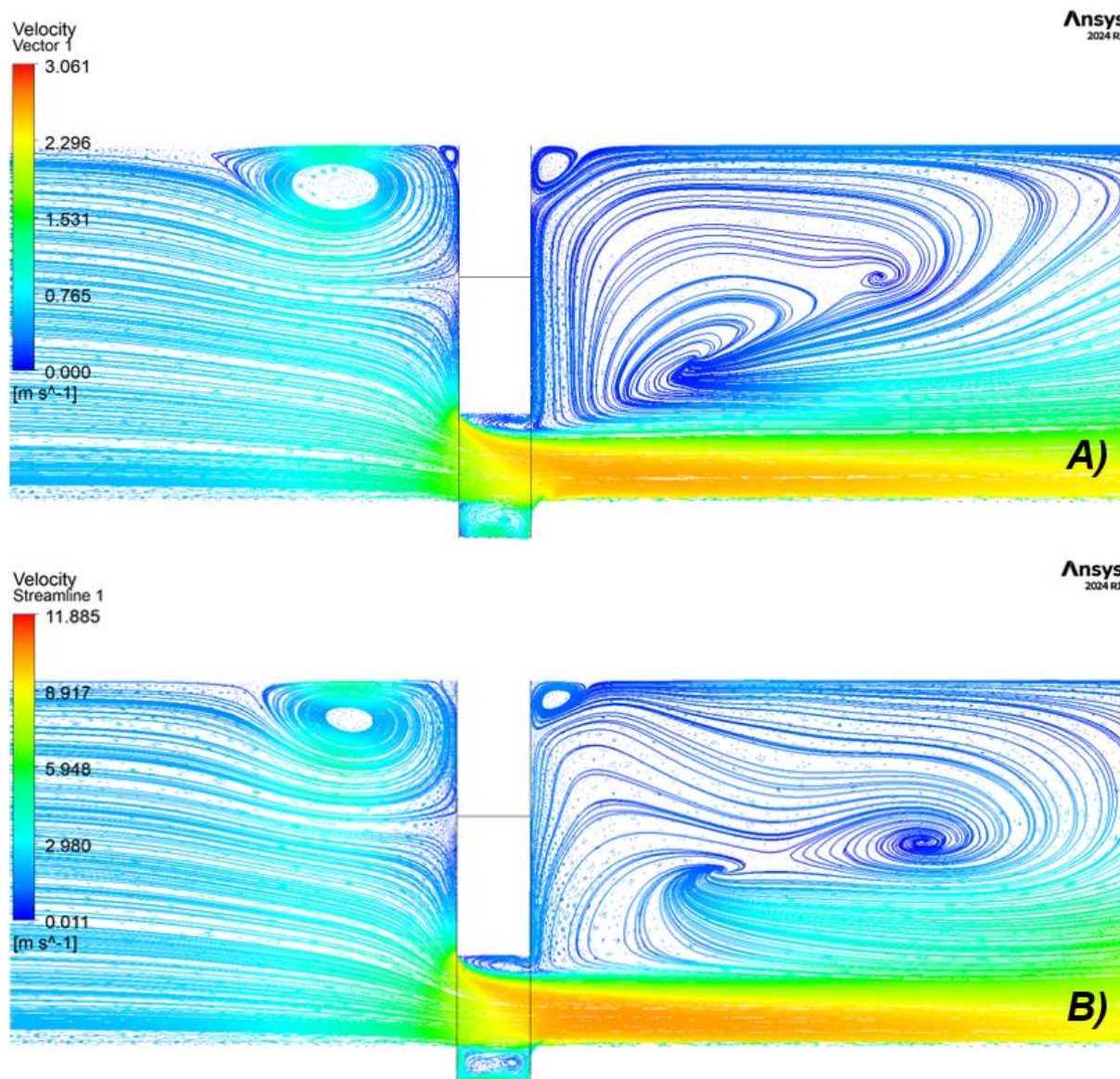


Fig. 8. Visualization of velocity streamlines during water flow in the pipeline with the flow cross-section closed by the gate valve to 75 % at a velocity of  $0.5 \text{ m}\cdot\text{s}^{-1}$  (A) and at a velocity of  $2.0 \text{ m}\cdot\text{s}^{-1}$  (B).

The flow velocities changed significantly due to this degree of flow cross-section narrowing. From an initial velocity of  $0.5 \text{ m}\cdot\text{s}^{-1}$ , the velocity increased to approximately  $3.0 \text{ m}\cdot\text{s}^{-1}$ . In the case of an initial velocity of  $2.0 \text{ m}\cdot\text{s}^{-1}$ , the velocity increased to around  $12 \text{ m}\cdot\text{s}^{-1}$  when the flow cross-section was reduced by 75 %, representing a sixfold increase in both cases.

At 75 % closure, we observe significant changes in the flow characteristics, including intense turbulence, substantial acceleration of the flow in the narrowed section, and a subsequent velocity drop downstream of the gate valve due to the pipe widening (Fig. 9).

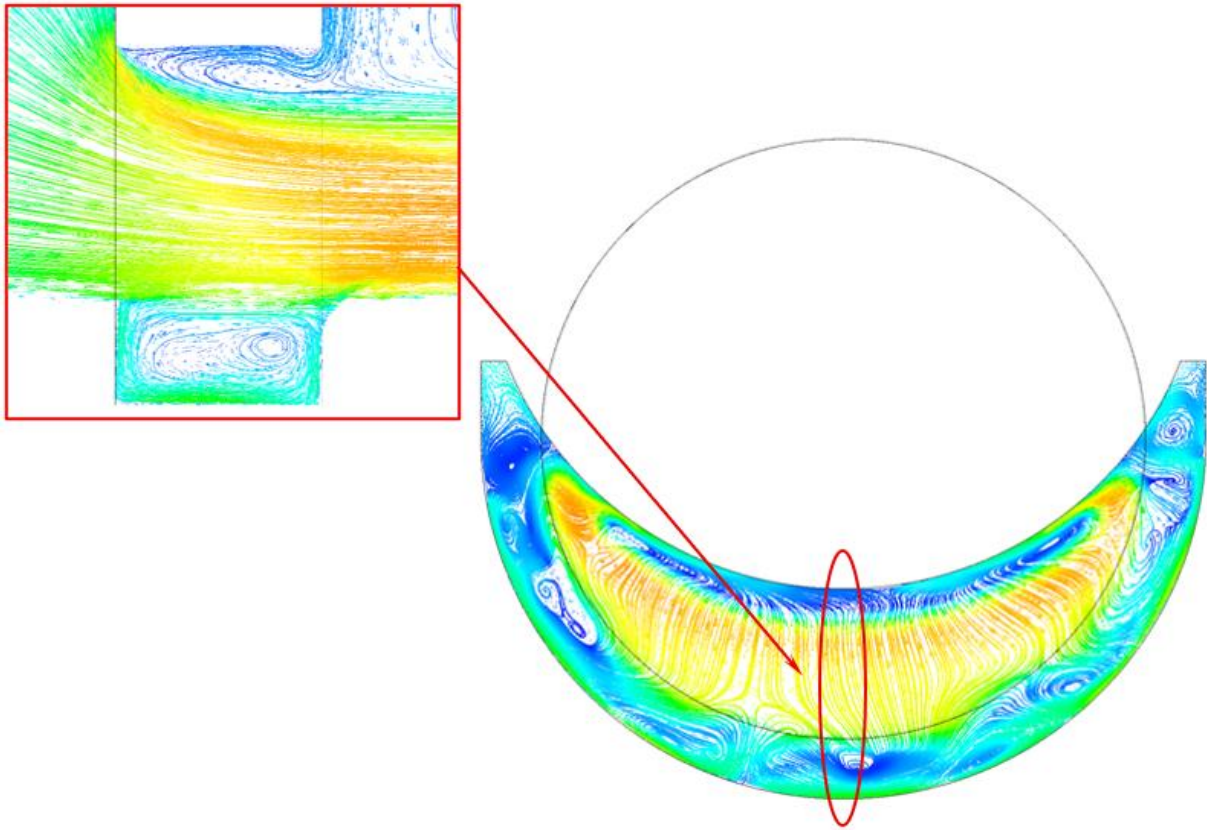


Fig. 9. Detailed view of velocity streamlines during water flow in the pipeline with the flow cross-section closed by the gate valve to 75 % at a velocity of  $2.0 \text{ m}\cdot\text{s}^{-1}$ .

The analytical procedure for determining the structural parameter  $\pi$  and the loss factor  $\zeta$  is demonstrated for the 75 % closure position of the wedge. The flowing medium is water, with a velocity of  $v = 2.0 \text{ m}\cdot\text{s}^{-1}$ . The pipeline diameter is  $D = 0.05 \text{ m}$ . The pressure drop  $\Delta p$  was determined as the difference between the inlet pressure of the valve and the pressure behind it, obtained from numerical analysis. The solution was conducted using the relationships outlined below in MS Excel. The same approach used for this state was applied in the calculations for other opening positions and all selected flow velocities.

Basic flow parameters (Kušnír, 2025):

1. Volumetric flow rate at the given flow velocity:

$$Q_V = v \cdot S = 2 \cdot \frac{\pi \cdot D^2}{4} = 0.00393 \text{ m}\cdot\text{s}^{-1}. \quad (1)$$

2. Mass flow rate at the given flow velocity:

$$Q_m = \rho \cdot v \cdot S = 995.87 \cdot 2 \cdot \frac{\pi \cdot D^2}{4} = 3.911 \text{ kg}\cdot\text{s}^{-1}. \quad (2)$$

3. Reynolds number:

$$Re = \frac{v \cdot D}{\nu} = \frac{2 \cdot 0.05}{8.01 \cdot 10^{-7}} = 124844. \text{ (It is turbulent flow)} \quad (3)$$

4. Pressure drop:

$$\Delta p = p_{inlet} - p_{outlet} = 43468 - 0 = 43468 \text{ Pa}. \quad (4)$$

5. The loss factor  $\zeta$  is expressed by the relation:

$$\Delta p = \zeta \cdot \frac{v^2}{2} \cdot \rho . \quad (5)$$

From where:

$$\zeta = 2 \cdot \frac{\Delta p}{\rho \cdot v^2} = 2 \cdot \frac{43468}{995.87 \cdot 2^2} = 21.824 . \quad (6)$$

The results of the numerical simulation of water flow through the gate valve at different closure levels are summarized in Tab. 1, Tab. 2, and Tab. 3. Three levels of flow cross-section closure were considered (25 %, 50 %, and 75 %) along with three different flow velocities (0.5 m·s<sup>-1</sup>, 1.0 m·s<sup>-1</sup>, and 2.0 m·s<sup>-1</sup>). The main flow parameters were evaluated.

Tab. 1. Evaluation of solution results for 25 % closure.

Physical parameters	$v$	$Q_V$	$Q_m$	$Re$	$\Delta p$	$\zeta$
Units	[m·s <sup>-1</sup> ]	[m <sup>3</sup> ·s <sup>-1</sup> ]	[kg·m <sup>-3</sup> ]	[1]	[Pa]	[1]
Numerical values	0.5	0.00098	0.978	31211	73.0	0.586
	1	0.00196	1.955	62422	268.0	0.538
	2	0.00393	3.911	124844	1002.0	0.503

Tab. 2. Evaluation of solution results for 50 % closure.

Physical parameters	$v$	$Q_V$	$Q_m$	$Re$	$\Delta p$	$\zeta$
Units	[m·s <sup>-1</sup> ]	[m <sup>3</sup> ·s <sup>-1</sup> ]	[kg·m <sup>-3</sup> ]	[1]	[Pa]	[1]
Numerical values	0.5	0.00098	0.978	31211	442.5	3.555
	1	0.00196	1.955	62422	1778.0	3.571
	2	0.00393	3.911	124844	7141.0	3.585

Tab. 3. Evaluation of solution results for 75 % closure.

Physical parameters	$v$	$Q_V$	$Q_m$	$Re$	$\Delta p$	$\zeta$
Units	[m·s <sup>-1</sup> ]	[m <sup>3</sup> ·s <sup>-1</sup> ]	[kg·m <sup>-3</sup> ]	[1]	[Pa]	[1]
Numerical values	0.5	0.00098	0.978	31211	2684.0	21.561
	1	0.00196	1.955	62422	9919.5	19.921
	2	0.00393	3.911	124844	43468.0	21.824

From the results, it can be observed that as velocity increases, the expected rise in pressure loss  $\Delta p$  occurs, with the highest values reached at 75 % closure. At a velocity of  $v = 2.0$  m·s<sup>-1</sup>,  $\Delta p$  reaches 43468.0 Pa, representing a significant load on the gate valve.

The loss coefficient  $\zeta$ , representing the measure of hydraulic resistance caused by the valve, increases with higher closure levels. The highest value was recorded at  $v = 2.0$  m·s<sup>-1</sup> and 75 % closure, where  $\zeta = 21.824$ .

### Discussion and conclusions

The results of numerical simulation and analytical calculations confirmed that the geometry of the slide valve and its degree of closure significantly affect pressure losses and the loss coefficient. As the degree of

closure increases, the intensity of turbulent flow and vortex formation rises, resulting in more pronounced low-pressure zones. This leads to an increase in hydraulic losses and potential structural wear. The highest values of the loss coefficient were recorded at 75 % closure and a flow velocity of  $2.0 \text{ m}\cdot\text{s}^{-1}$ . CFD analysis allowed precise localization of areas with the greatest dynamic stress, which are critical for the slide valve's service life. A high level of agreement between the numerical and analytical approaches was also confirmed, verifying the accuracy of the model used. The acquired insights can be applied to optimize the valve geometry, select appropriate materials, and improve pipeline system design in terms of efficiency and reliability. The proposed methodology is also applicable to other types of fittings exposed to similar flow conditions.

## References

- Čarnogurská, M., Příhoda, M., Brestovič, T. (2021). *Aplikovaná mechanika tekutín*. University textbook. Tipopress – Tlačiareň, s. r. o., Košice
- Company Atria Europe. (2025). Glossary of company portfolio terms: Gate valve. Available at: <https://www.atria.sk/slovník-pojmov/supatko/> (in Slovak)
- Gonzalez-Trejo, J., Miranda-Tello, R., Gabbasov, R., Real-Ramirez, C. A., & Cervantes-de-la-Torre, F. (2024). Experimental Analysis of the Influence of the Sliding-Gate Valve on Submerged Entry Nozzle Outlet Jets. *Fluids*, 9(1), 30. <https://doi.org/10.3390/fluids9010030>
- IPL group. (2025). E1+ VALVE – DN 50-300, PN10/PN16. Available at: <https://www.iplgroup.com/hawle-e1plus-valve.php>
- Javanshir, I., Javanshir, N., Barmaki, R., Mahmoodi, M. (2015). Modeling of the fluid-induced vibrations in sliding gate dams. *Journal of Vibroengineering*. Vol. 17, No. 1, pp. 478–486
- Kušnír, S. (2025). Prúdové pomery vo vybranom posúvači a ich vplyv na stratový faktor. Diploma thesis. Technical University of Košice, 79 s. (in Slovak)
- Little P.Eng. (2025). For Engineering Consultancy Services. Major valve parts. Available at: <https://www.littlepeng.com/single-post/2017/06/29/major-valve-parts>
- Ma, G., Lin, Z., Zhu, Z. (2020). Investigation of transient gas-solid flow characteristics and particle erosion in a square gate valve. *Engineering Failure Analysis*. Volume 118, 104827, ISSN 1350-6307. <https://doi.org/10.1016/j.engfailanal.2020.104827>.
- Mehrzad, S., Javanshir, I., Ranji, A.R. and Taheri, S.H. (2015). Modeling of fluid-induced vibrations and identification of hydrodynamic forces on flow control valves. *J. Cent. South Univ.* 22, 2596–2603. <https://doi.org/10.1007/s11771-015-2789-y>
- Sozinando, D.F., Tchomeni, B.X., Alugongo, A.A. (2023). Modal Analysis and Flow through Sectionalized Pipeline Gate Valve Using FEA and CFD Technique. *Journal of Engineering*. Volume 2023, Issue 1, 2215509. <https://doi.org/10.1155/2023/2215509>
- Stosiak, M., Karpenko, M., Deptuła, A., Urbanowicz, K., Skačkauskas, P., Deptuła, A. M., Danilevičius, A., Šukevičius, Š., & Łapka, M. (2023). Research of Vibration Effects on a Hydraulic Valve in the Pressure Pulsation Spectrum Analysis. *Journal of Marine Science and Engineering*. 11(2), 301. <https://doi.org/10.3390/jmse11020301>
- Tech-science. (2020). Pressure loss in pipe systems (Darcy friction factor). Available at: <https://www.tech-science.com/mechanics/gases-and-liquids/pressure-loss-in-pipe-systems/>
- VAG s.r.o. (2025). IKO-Plus gate valve. Available at: <https://www.vag-group.com/cz-cs/vyrobky/detail-vyrobku/ikoplus-kovotesnici-soupatko>
- Young, Ch. (2025). EngineerExcel. Loss Coefficients: A Practical Guide for Engineers. Available at: <https://engineerexcel.com/loss-coefficient/>
- Žic, E., Banko, P., Lešnik, L. (2020). Hydraulic analysis of gate valve using computational fluid dynamics (CFD). *Scientific Review Engineering and Environmental Sciences*. Vol. 29 (3), 275–288. <https://doi.org/10.22630/PNIKS.2020.29.3.23>