Journal of Thermal and Fluid Science
Volume 3, Issue 2, December 2022, pp 59 - 72
https://doi.org/10.26706/jtfs.3.2.20221205

Study for the Comparison of CFD Analysis of Flow through Valves

Prathamesh S. Indule

Department of Mechanical Engineering, JSPM Narhe Technical Campus, Narhe, Pune 2021- 22

Corresponding Author: prathameshindule2000@gmail.com

Abstract. Control valves are generally flow control equipment is utilized in a variety of sectors. The authors' modeling and simulation studies of various control valves are reviewed in this study. The findings of this research enable users to comprehend the flow pattern of a valve at various flow rates, as well as identify approaches for improving the valve's function. A gate valve was designed using Computational Fluid Dynamics (CFD) and CFD-powered software. The valve coefficient (Cv), often known as the flow coefficient, is a commonly used valve parameter. This paper presents the results of Flow Coefficient Analysis research.

Keywords. CFD, gate valve, Cage type Valve, valve coefficient.

Article – Peer Reviewed
Received: 9 Dec 2022
Accepted: 20 Dec 2022
Published: 31 Dec 2022

Cite this article: Prathamesh. S. Indule, "Study for the Comparison of CFD Analysis of Flow through Valves", *Journal of Thermal and Fluid Science*, RAME Publishers, vol. 3, issue 2, pp. 59-72, 2022. https://doi.org/10.26706/jtfs.3.2.20221205

1. Introduction

A gate valve, also known as a sluice valve, is a type of valve that opens by lifting a round or rectangular gate/wedge out of the way of the flowing fluid. Gate valves are commonly used when a straight-line flow of fluid with minimal restriction is desired because the sealing surfaces between the gate and seats are planar. Gate valves are typically used to allow or prevent liquid flow, but they should not be used to regulate flow unless specifically designed for that purpose. When used to regulate flow, the majority of the flow change occurs near shutoff at a relatively high fluid velocity, resulting in gate and seat wear and eventual leakage. Gate valves are widely used in industry because of their ability to cut through liquids.

When fully open, a typical gate valve has no obstruction in the flow path, resulting in very low friction loss. The majority of the flow change occurs at a relatively high fluid velocity near shutoff, causing gate and seat wear as well as eventual leakage. When used to regulate flow, the majority of the flow change occurs near shutoff at a relatively high fluid velocity, resulting in gate and seat wear and eventual leakage. A gate valve is a device that allows or prevents the flow of liquid. When fully open, a typical gate valve has no obstruction in the flow path, resulting in very low friction loss.

Because of their ability to cut through liquids, gate valves are widely used in the petroleum industry. When the gate valve is opened, the flow path is nonlinearly widened.

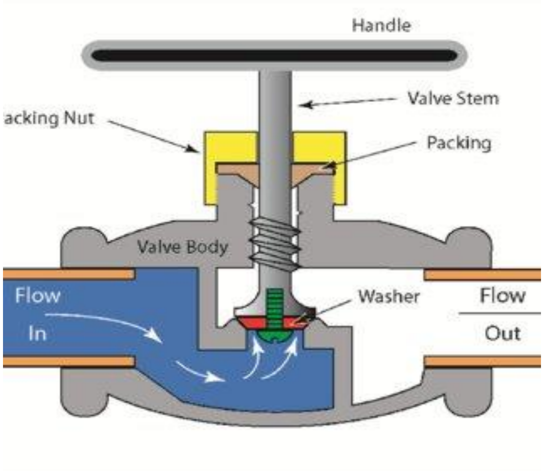


Figure 1. Gate Valve [4]

The actuator (e.g., hand wheel or motor) is connected to the gate valve via a threaded stem, allowing it to be opened and closed. The stem can be threaded in a variety of ways depending on which end is threaded. They are classified as having a rising stem or not. When you turn on the valve, rising stems are attached to the gate, which you can raise and lower at the same time, displaying a visual representation of the valve position. The actuator takes the form of a nut that is rotated around the threaded stem to move it. Non-rising stem valves are threaded into the gate and are attached to and rotate with the actuator. Because the gate's motion is buried inside the valve, they may have a pointer threaded onto the upper end of the stem to indicate valve position. Non-rising stems, such as underground, are used when vertical space is limited. The primary function of gate valves is isolation. In service, these valves are typically fully open or completely closed. When the valve is fully open, the fluid or gas flows in a straight line through the valve path with little resistance. Because precise control is impossible, flow regulation or throttling should not be used. Furthermore, partially opened valves with high flow velocity can cause disc and seating surface erosion. Vibration can also cause the partially opened valve disc to chatter. Specially designed low velocity throttling gate valves, such as guillotine gate valves for pulp stock, are an exception.

1.1. CFD (Computational Fluid Dynamics)

CFD stands for Computational Fluid Dynamics an effective and powerful tool to simulate fluid flow and heat transfer numerically. Researchers have created a variety of numerical approaches to use this resilient tool to simulate a large selection of complex flows and heat transfer problems over the years. There are two types of methods that can be used. major groups as “conventional methods” and “accelerated methods”

In the present contribution only, major methods are mentioned. Most CFD methods use either a Lagrangian or an Eulerian approach to The Navier-Stokes equations must be solved. Aside from that, some methods use Boltzmann equations rather than Navier-Stokes equations to solve problems.

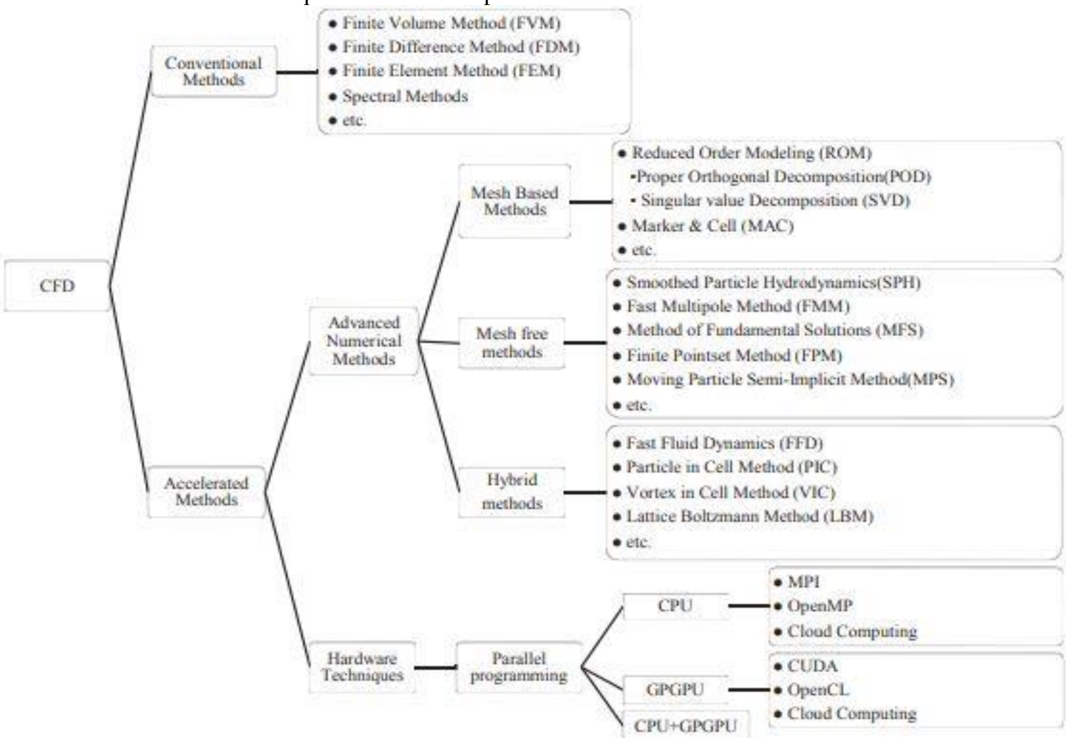


Figure 2. CFD Classification [4]

Traditional methods are the most frequently utilized, extremely accurate, and commonly employed in most commercial software products. Traditional methods, on the contrary, are exceedingly slow in terms of calculation time, making it nearly impossible to solve huge issues in a reasonable amount of time to use online. We conduct a literature review of many available and popular advanced approaches that can provide moderate acceleration over conventional methods in this study. After that, the Acceleration Methods are divided into two groups: Numerical Methods and Hardware Techniques Advanced Numerical Methods and Hardware Techniques In most cases, hardware acceleration techniques are employed in conjunction with both traditional and advanced numerical approaches. Mesh-based, mesh-free, and hybrid numerical methods are the three types of advanced numerical methods. Figure 1 shows the literature reviews on the popular advanced numerical methods utilized in this document. This short study will not be able to go into the technical intricacies of each approach or conduct a comprehensive analysis of all the material available for each method. In this study, we evaluate the existing approaches other than standard methods for achieving real-time CFD, describe their application, and list some basic and noteworthy literatures for multiphase flow, free surface flow, and other related topics. Flow and heat transfer.

1.2 Methodology

This section highlights the research methodology for present study to explain research objectives and suitable methodology to achieve those objectives. This project's goal is to research is to make analysis gate valve using ansys and cfd analysis of flow through valve. This case study is carried for a Flow control valve. The various stages planned for present research work are as follow.

1. Study of various flow control valve.
2. Collection of data for analysis of CFD.
3. Analytical Gate valve analysis for different openings.
4. Selection of Gate valve disc of different sizes.
5. Analytical analysis of CFD flow through control valve.

1.3 Problem Statement

Flow measurement through different gate valve may change due to their shape and opening and hence comparison of the work presented by different researches is important for their study. This study includes the comparative analysis of gate valve; cage-type valve using the approach Computational Fluid Dynamics is a branch of computer science that deals with the simulation of fluid (CFD) is a technique for simulating fluid dynamics.

1.4 Objectives of Project

1. To check fluid flow through valve.
2. To check velocity, pressure and mass flow rate through gate valve at different openings.
3. To determine the streamline of flow through different opening of gate valve and modified valve.

2. CFD Analysis for Flow through Gate valve

2.1. Analytical analysis

Numerical analysis is a branch of mathematics and computer science that develops analyses, and implements algorithms for solving problems with continuous variables numerically. Such issues emerge in the natural sciences, social sciences, engineering, medicine, and business, among other fields. Since the mid-20th century, the rising power and availability of digital computers has led to a rise in the usage of realistic mathematical models in science and engineering, necessitating increasingly sophisticated numerical analysis to solve these more detailed world models. Numerical analysis is a formal academic field that includes everything from mathematical studies to computer technology difficulties. Because of the increased availability of computers in the 1980s and 1990s, the new subject of scientific computing, or computational science, arose. Numerical analysis, symbolic mathematical computations, computer graphics, and other computer science fields are used to make it easier to set up, analyses, and interpret big mathematical models of the real world.

Velocity calculation,

$$\rho = 998.2\left(\frac{kg}{m^3}\right) \& \ \mu = 0.001003\left(kg \cdot m^{-1} \cdot s^{-1}\right).$$

Boundary condition

$$R_e = \frac{\rho . V . L}{\mu} => $$

Flow coefficient (Cv)

$$C_v = Q\sqrt{\frac{\mathrm{SG}}{\Delta P}}$$

As all the analysis is done by considering the fluid i.e. water so all the values are considered accordingly suitable for the same.

Table 3. Parametric Calculations [17]

Lift Disk (mm)	Mass Flow Rate (Kg/s)	Flow Coefficient (Cv)	Q1 = Rate of Flow (m^3/s)	Q = m/Sp. Gravity of the fluid (water)	ΔP = Pressure drop across the valve
10	0.1439	0.0038	0.0014129	1	0.0014
20	0.2257	0.0060	0.0002261	1	0.0014
30	0.3338	0.0081	0.0003344	1	0.0014
40	0.4195	0.0112	0.0004202	1	0.0014
50	0.5124	0.0133	0.0005133	1	0.0014
60	0.6893	0.0184	0.0006905	1	0.0014
70	0.7164	0.0189	0.0007177	1	0.0014

2.2. CFD Anaysis of Fluid (water) flow through Gate Valve

2.2.1 Velocity Contours on different openings

The velocity contour is the velocity field for that iteration in steady state. Data sampling can also be done on a steady case where we may see some slight changes in the flow (you'll have poor convergence) which can then produce an averaged velocity field.

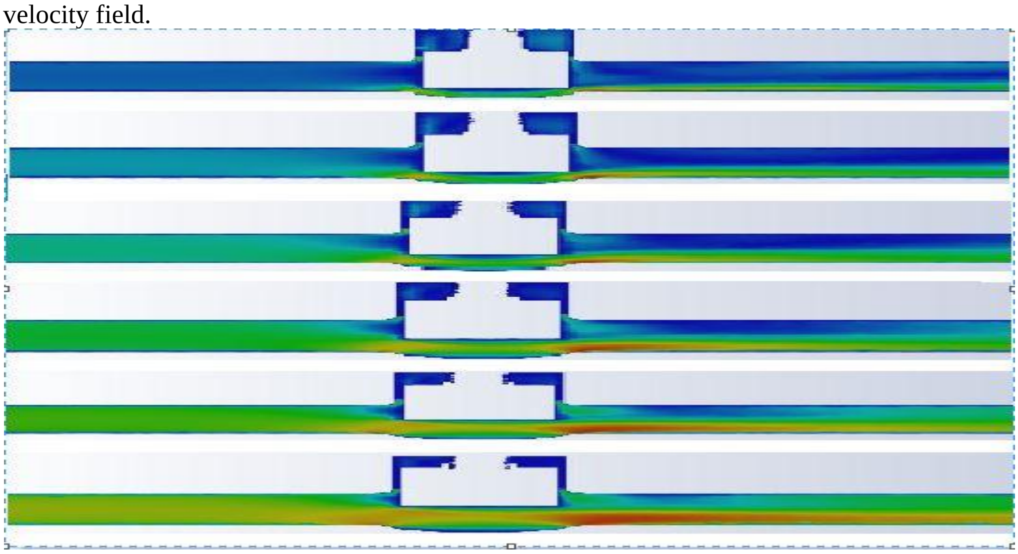


Figure 3. Velocity counters on different openings (10, 20, 30, 40, 50, 60, &70mm). [17]

2.2.2 Fluid flow velocity

A longitudinal view of the gate valve model at various degrees of opening based on the 150 streamlines. The first four models show models with an inlet velocity of 1.0 m·s–1 and the last four models with an inlet velocity of 1.5 m/s.

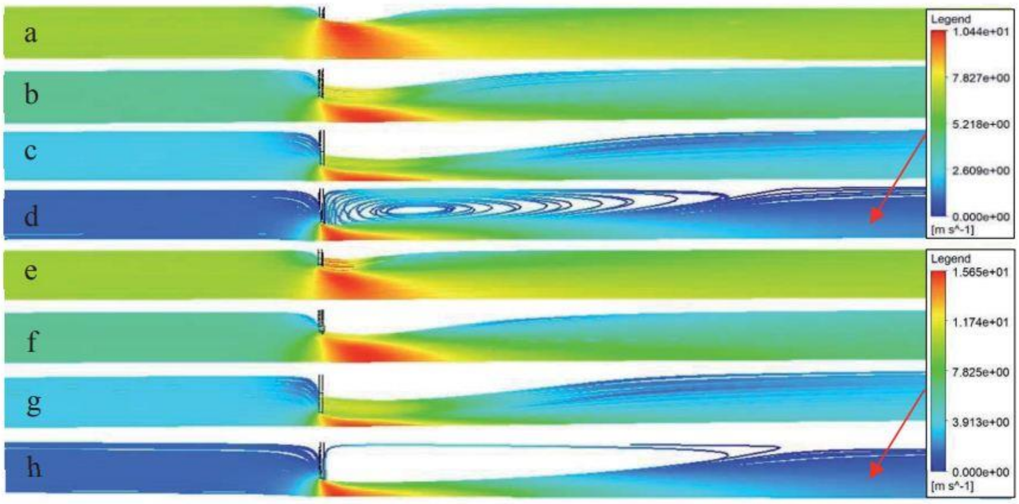


Figure 4. Fluid flow velocity [17]

Table. 4. For each of the gate valve models, the maximum, minimum, and average flow velocities [m•s–1] are shown. [17]

Percentage of gate valve closure [%]	v = 1.0 m·s–1			v = 1.5 m·s–1		
	max	min	Avg	max	min	Avg
20	1.564	0.007	1.035	2.337	0.021	1.553
40	2.533	0.003	1.195	3.797	0.004	1.795
60	4.415	0.004	1.594	6.633	0.003	2.390
80	10.585	0.002	2.780	15.884	0.004	4.220

2.2.3 Relative pressure

Table 5 shows the maximum, minimum, and average relative pressures [Pa] for all gate valve submodels based on the k- turbulent model. As the valve is closed, the maximum, lowest, and average relative pressure readings all rise exponentially. A valve that is 80 percent closed has a relative pressure that is 75 times greater than a valve that is 20 percent closed. Consider the maximum and lowest pressures when dimensioning the valve as a function of pressure, as these can cause the pipe to expand or twist, causing damage and breaking.

Table 5. View of maximum, minimum and average relative pressures [Pa] for each of the gate valve sub models. [17]

Percentage of gate valve closure [%]	v = 1.0 m·s–1			v = 1.5 m·s–1		
	Max	min	Avg	max	min	Avg
20	895	–1053	171	19.83	–2.501	355
40	1 989	–2689	407	4. 433	–6.057	886
60	7 223	–8209	1 897	16.195	–18.347	4228
80	56 948	–46156	12 890	127.831	–103.401	29080

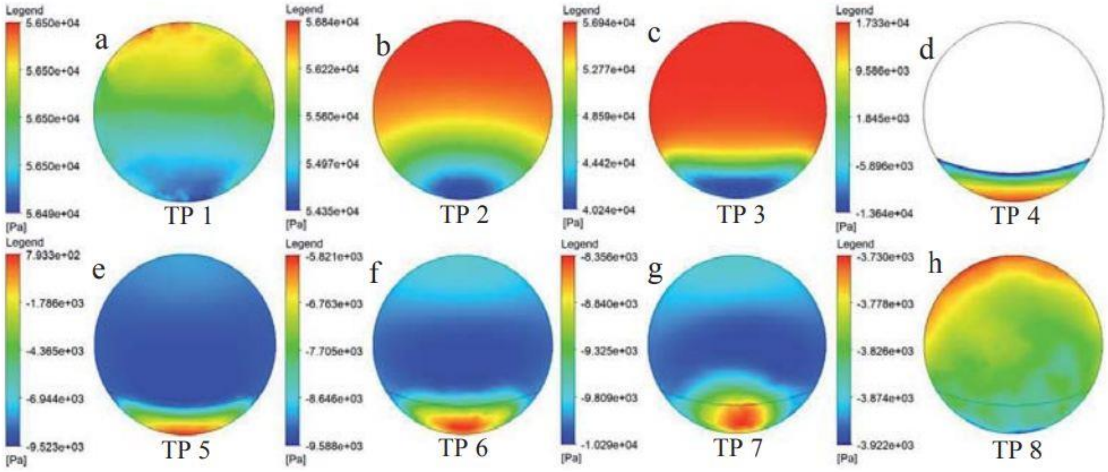


Figure 5. Relative pressure distribution on transverse profiles of gate valve sub models with 80% valve closure and 1.0 m•s–1 inflow velocity [17]

2.3 Results

As per the analysis the rate of mass flow plots and Residual plots are occurred.

2.3.1 Residual Plots

The residuals are shown on one axis, and the independent variable is shown on the other. A linear regression model is appropriate for the data if the dots in a residual plot are randomly distributed across the horizontal axis. Otherwise, a nonlinear model is preferable.

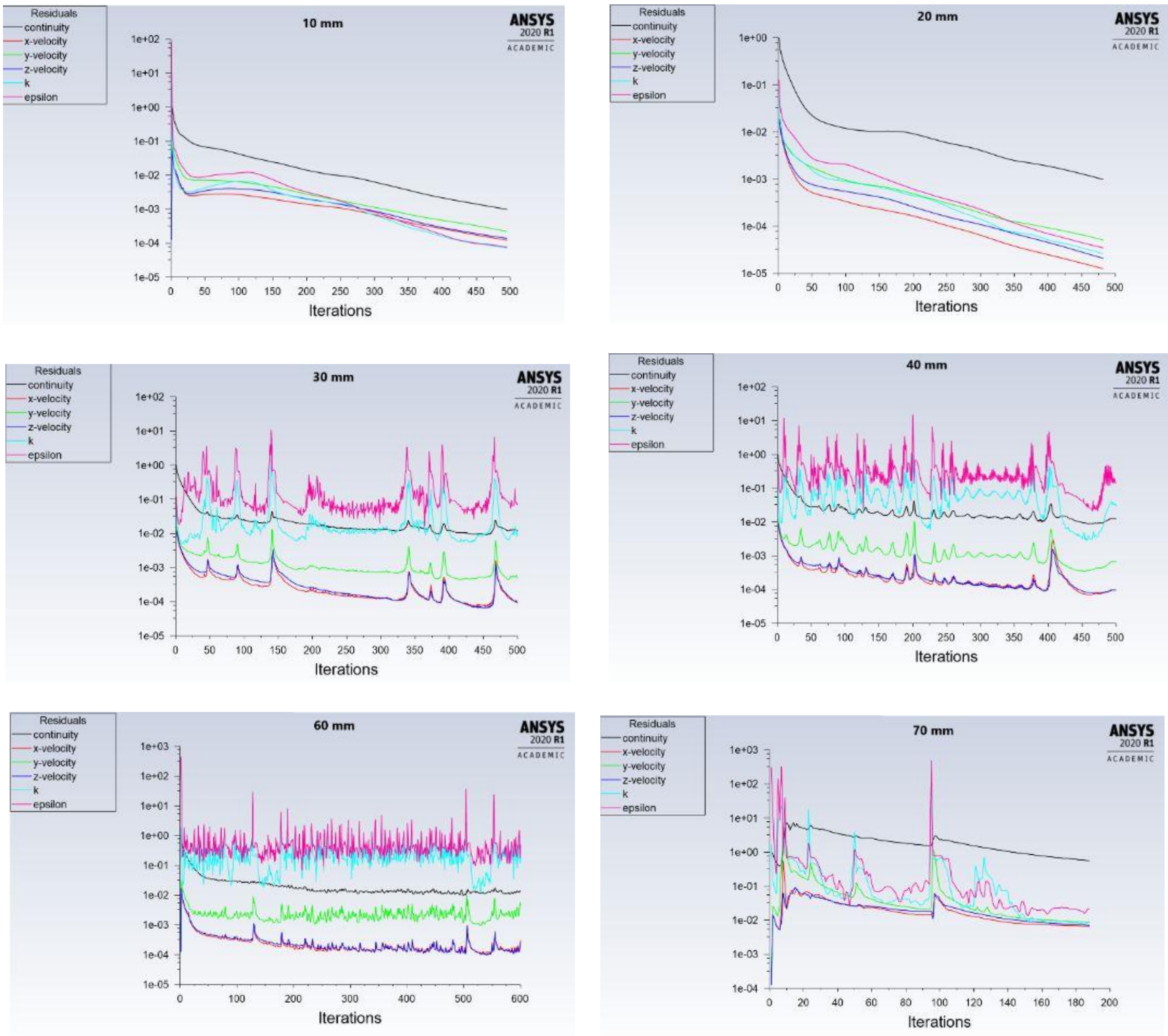
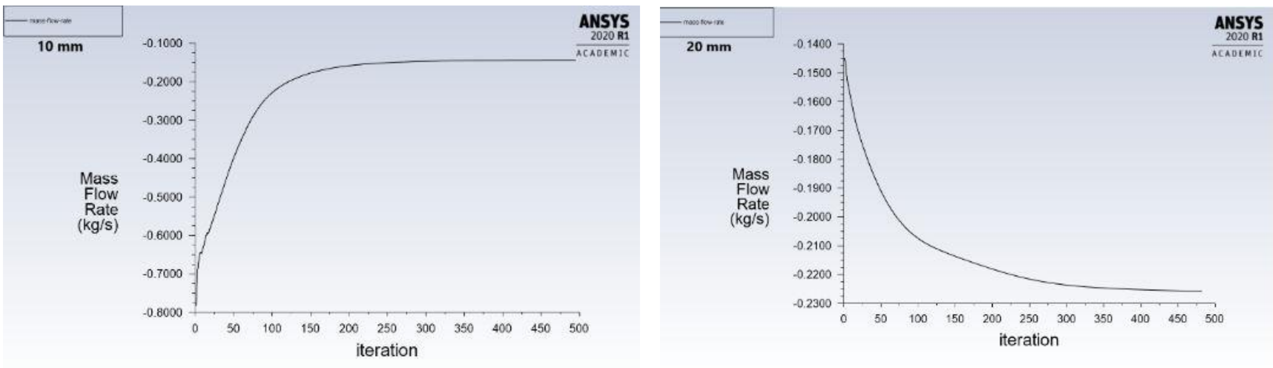


Figure 6. Residual plot for the percentage of gate valve closure from 10 mm to 70 mm

2.2.2 Mass Flow Rate Plots

The mass flow rate is the mass of a liquid substance passing per unit time. In other words, the mass flow rate is defined as the rate of movement of liquid pass through a unit area. The mass flow is directly proportional to the density, velocity, and cross-sectional area of the liquid.



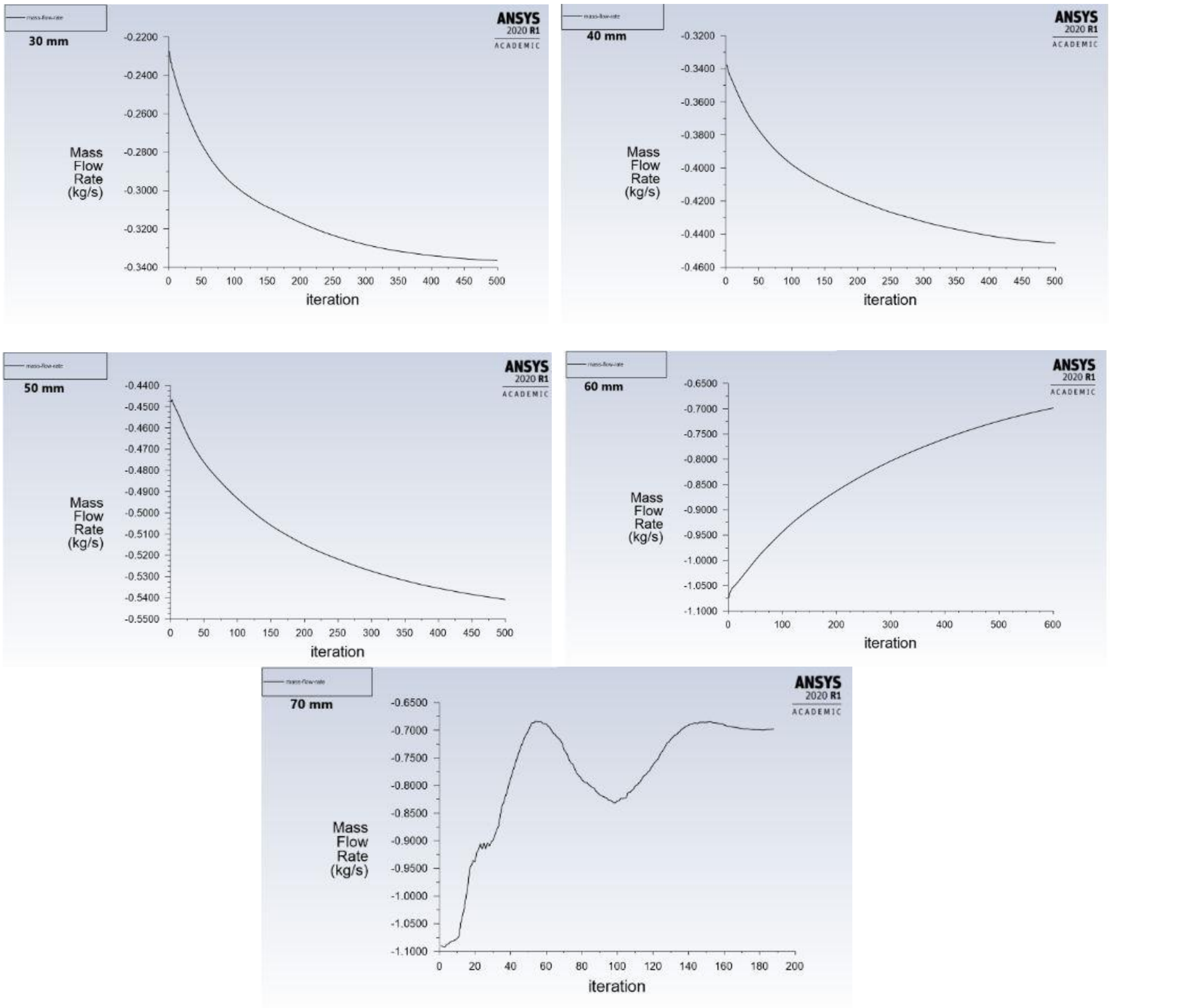


Figure 7. Residual plot for the percentage of gate valve closure from 10 mm to 70 mm

3. CFD Analysis of flow through Cage type Valve

3.1 Physical Model of Valve

Control valves are essential components in many process industries, such as nuclear and thermal power plants, to manage flow and pressure. Normally, the volume flow rate of control valves grows monotonously as the pressure drop between the valve inlet and the valve outlet increases, but when the pressure drop is substantial, the volume flow rate of control valves decreases. A choked flow emerges. Cavitation or flashing is the most common cause of blocked flow in incompressible working fluids, which can lead to Valve vibration, loudness, or even valve rupture are all possible outcomes.

The flow coefficient under various valve openings is shown by the flow characteristics curve. Is typically used as a crucial indicator to identify whether a control valve is suited for a certain system and has good performance. Many studies have been conducted on control valves' flow properties to date, with an emphasis on three aspects: the effects of control valve construction, the effects of manufacturing method, and methods to directly compute the flow characteristics. The computational fluid dynamics (CFD) method is the most used, and machine learning is also used to speed up calculation times.

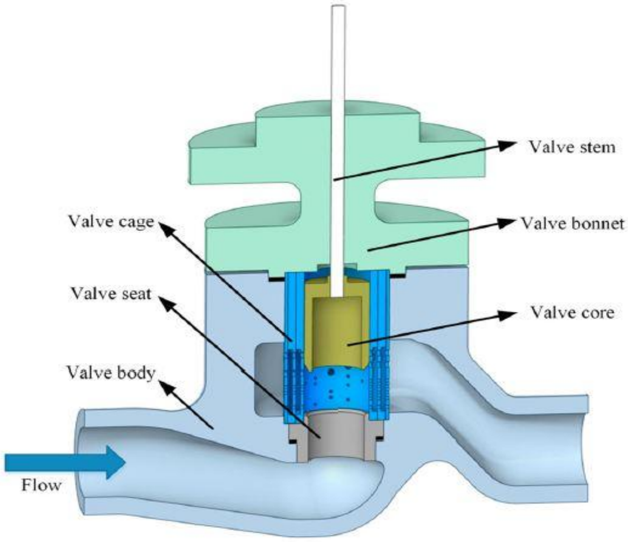


Figure 8. Model of Valve [8]

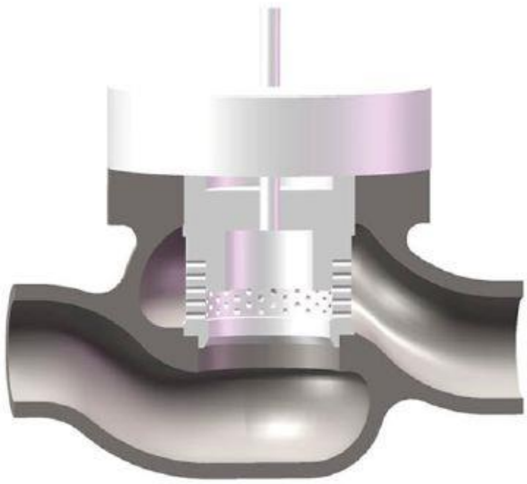


Figure 9. 3D model of valve [8]

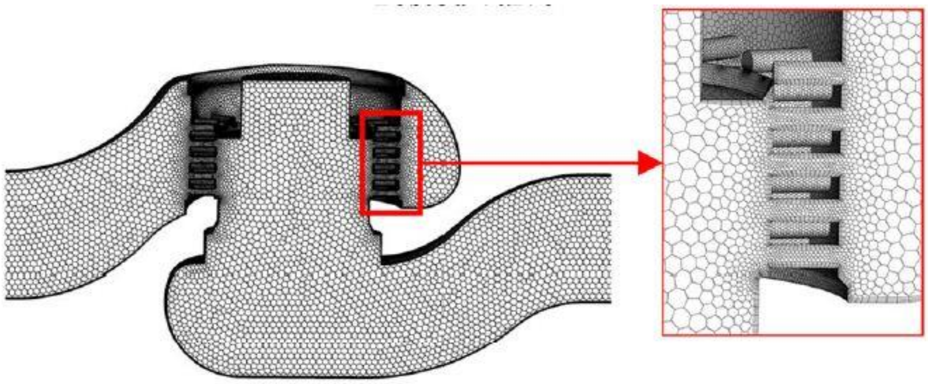


Figure 10. Mesh model of valve [8]

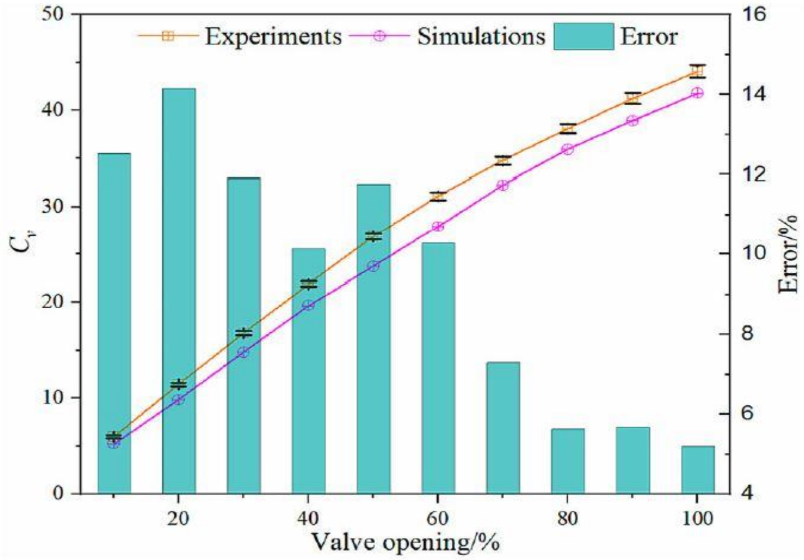


Figure 11. Valve opening % [8]

3.1.1 When the valve opening is 50% and the cage number is two, the pressure distributions are as follows.

Figure 12 shows when the valve is open 50% of the time and the cage number is two, pressure distributions are shown in Figure 12. There is a substantial pressure decrease after water runs n cage-type control valves with valve cage 1 through the first valve cage; however, the pressure variation is not visible. This is due to the second cage's low decompression ability, which is equivalent to a perforated plate that is fixed for cage cages 2 and 4. Figure 12 shows that the pressure difference between the inner and outer cages is greatest in controls with cage 2 and lowest in those with cage 1.

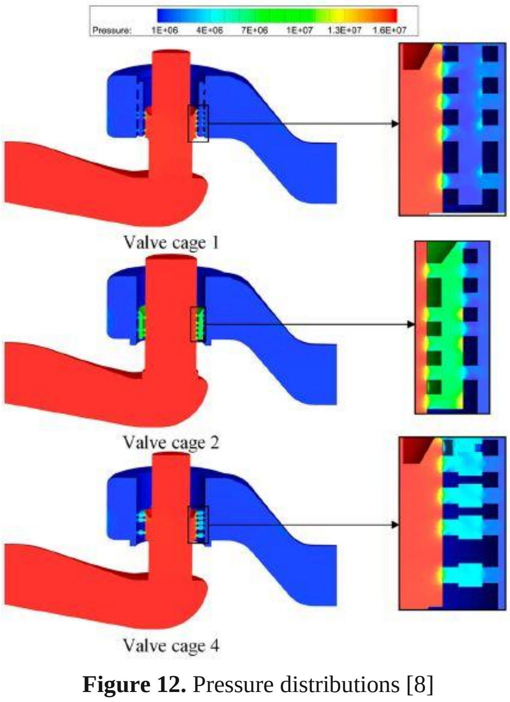


Figure 12. Pressure distributions [8]

3.1.2 When the valve is open 50% and the cage number is three, the pressure distributions are as follows.

Figure 13 shows the pressure distributions when the valve is 50% open and the cage number is three. A large pressure drop occurs before and after the first valve cage in cage-type control valves with valve cage 1, but the pressure fluctuation is minimal for the outer valve cage. Figure 13 also shows the decompression capability of cage- type control valves increase with groove height.

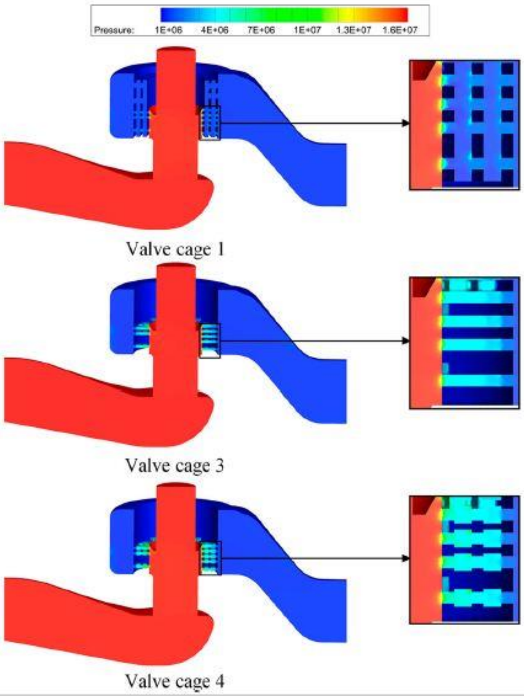


Figure 13. Pressure distributions when valve is open 50% [8]

3.1.3 Pressure along a specific streamline in different cage type control valves.

Cage-type control valves with valve cages 2 and 4 have two distinct decompression processes when the cage number is two, with pressure in valve cage 2 being higher. Cage number has little effect on pressure distribution, as does groove height. Despite pressure changes within the valve cages, pressure distributions are nearly equal before and after water flows into the valve cage.

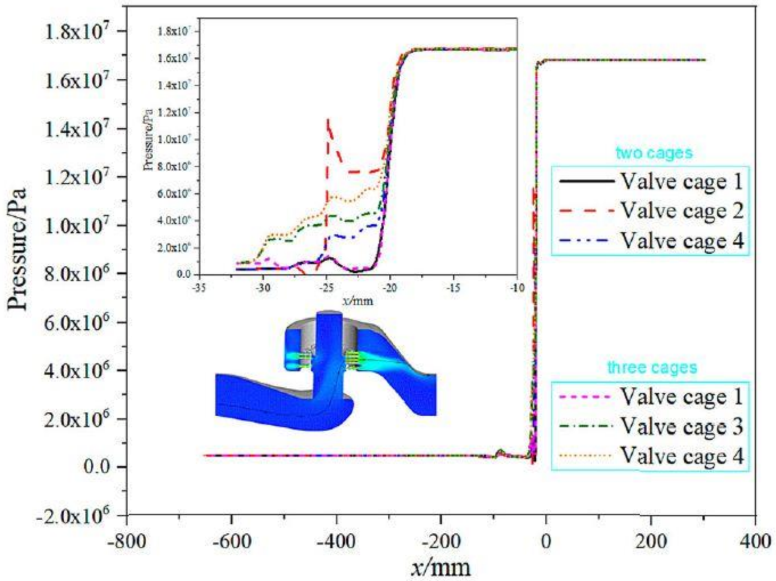


Figure 14. Pressure distributions [8]

3.2 When the cage number is two, velocity and streamlines in distinct cage-type control valves on the symmetry plane.

Figure 15 depicts the flow coefficients for various valve openings when the valve cage number is two. Figure 16 depicts the effects of the cage number on the flow coefficient in greater detail. Flow coefficient for cage-type control valves with valve cage 4 obviously increases as cage number decreases.

The flow coefficient of control valves is greatest with valve cage 1, and decreases as the cage number increases. For cage-type control valves with valve cages 3 and 4, the flow coefficient is medium and decreasing as the cage number or groove height increases. Figure 16 shows flow characteristics for various valve cages.

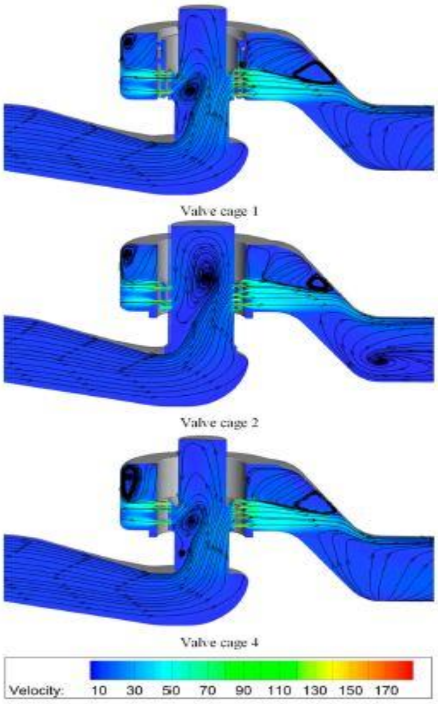


Figure 15. Pressure distribution in cage 1, 2, 4 [8]

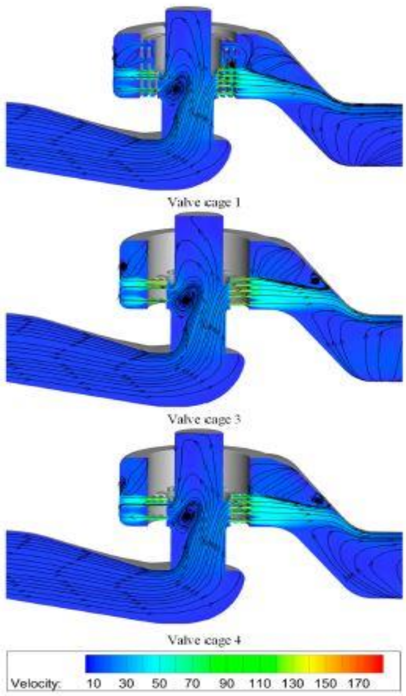


Figure 16. Pressure distribution in cage 1, 3, 4 [8]

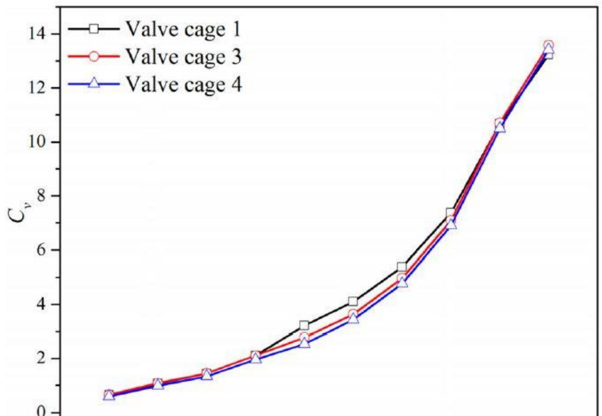


Figure 17. Valve Opening [8]

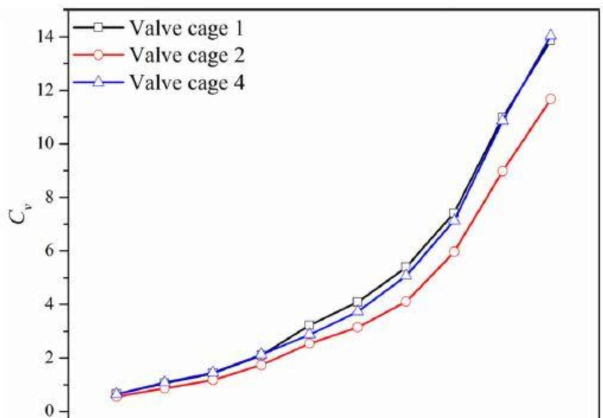


Figure 18. Valve Opening 2 [8]

4. Result and Discussion

Here all the calculations and graphical representation and various further results or outputs are mentioned above; so we differentiate both the valves as per the given mathematical numeric expression and calculations as well as it attributes respectively. So as per the given data mentioned above in the tables related (Pressure and Velocity) so we done comparison between the two valves regarding to the pressure and velocity at the different opening and closing parameters.

So, we analyzed in cage type valve which is not compatible for higher pressure drops and so it requires more space to get more pressure drop. Therefore, its compatible lower space and more pressure drop.

So, in gate type valve we analyzed that the decrease in pressure higher in comparison of cage type valve. And in cage type valve the velocity drop is higher in comparison of gate type valve.

So, we can compare in pressure the gate type valve is best in performance and while in velocity the cage type valve is best in performance as shown in graphical representation results.

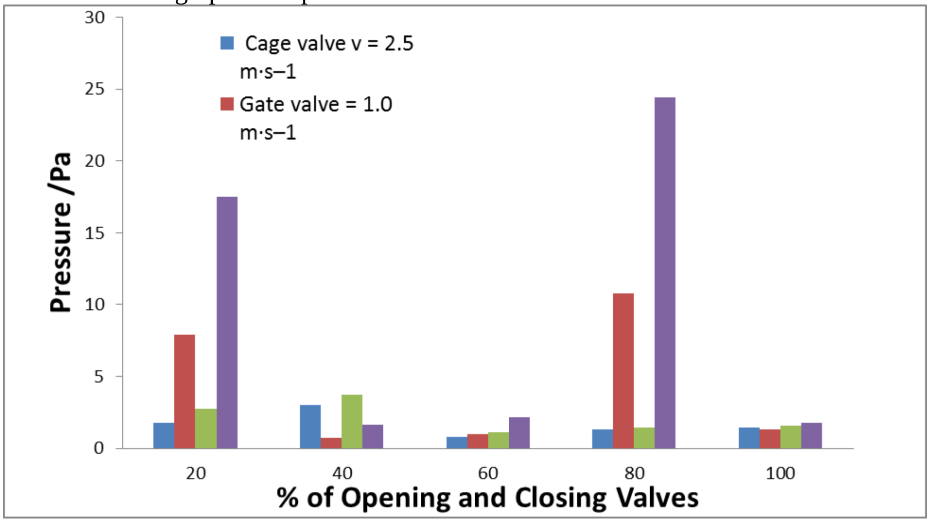


Figure 19. Graphical representation of Pressure vs. % of opening & closing valve.

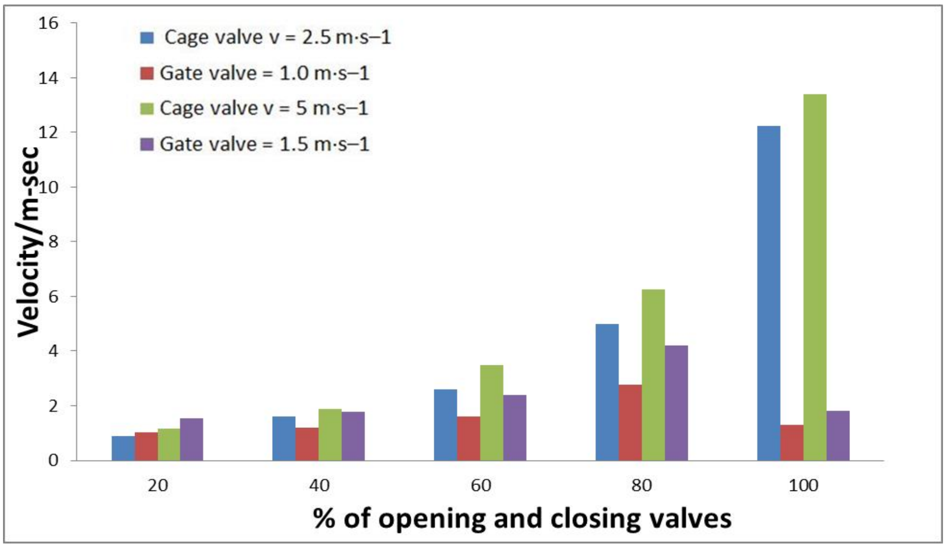


Figure 20. Graphical representation of Velocity vs. % of opening & closing valve

5. Conclusion

The commercial tools Ansys Fluent 19.1 and Ansys CFX 19.1 were used to do hydraulic study models of gate valves in this paper. The analyses were carried out with a gate valve opening angle of 4° and inflow gate valve types with velocities of 1.0 and 1.5 m•s–1. Following the hydrodynamic research, it was discovered that all models, especially at smaller opening degrees, Vortices in the area behind the gate valve are shown. The vortex's appearance and travel down the pipe are readily visible on the pipe system's core longitudinal profiles. The greatest velocity that occurs for a gate valve with a 40 percent closing degree and an intake flow velocity of 1.0 m•s–1 is 2.53 m•s–1, but it is 3.80 m•s–1 for the same model with an inlet flow velocity of 1.5 m•s–1. The investigation reveals that models with a lower valve opening degree have the highest velocities, pressures, and other physical variables. In the studied models, the physical quantities' maximum values occur primarily in or behind the valve. This research demonstrates that hydrodynamic analysis may be applied to a variety of valve geometries. Correct numerical modeling using CFD technology allows the generated results to be used to improve the valve's design and operation characteristics. This paper introduces cage-type control valves with improved valve cages, which have the advantages of no dead zones in the regulation process and are better suited to higher pressure drop settings. In terms of flow and cavitation, different cage-type control valves with no dead zones in the regulation process are compared. The cage with a full groove in both the circumferential and axial directions between two neighboring cages, the cage with a full groove but movable innermost cage, and two other types of upgraded cages are also studied. The flow coefficient tests are run first to check that the numerical methods employed are correct, and then the flow and cavitation distributions are calculated using the verified numerical methods.

References

[1] Asim, T., Mishra, R., Oliveira, A., &Charlton, M. (2019). The flow capacity of a control valve trim is affected by the geometrical aspects of flow routes. The Journal of Petroleum Science and Engineering is a peer-reviewed journal that publishes research on 172, 124–138. https://doi.org/10.1016/j.petrol.2018.09.050
[2] Chern, M. J., Hsu, P. H., Cheng, Y. J., Tseng, P. Y., & Hu, C. M. (2013). The occurrence of cavitation in a globe valve was studied numerically. The Journal of Energy Engineering is a publication dedicated to the study of energy, 139(1), 25–34. https://doi.org/10.1061/(ASCE)EY.1943-7897.0000084
[3] Chern, M. J., Wang, C. H., Lu, G. T., Tseng, P. Y., Cheng, Y. J., Lin, C. A.,&Hu, C.M. (2015). In a globe valve, the cages are designed. Part C: Journal of Mechanical Engineering Science, Proceedings of the Institution of Mechanical Engineers, 229(3), 476–484. https://doi.org/10.1177/0954406214535387
[4] Gao, H., Lin, W., & Tsukiji, T. (2006). Cavitation around the orifice of hydraulic valves is being investigated. Part G: Journal of Aerospace Engineering, Proceedings of the Institution of Mechanical Engineers, 220(4), 253–265. https://doi.org/10.1243/09544
[5] Han, M., Liu, Y., Wu, D., Zhao, X., & Tan, H. (2017). The properties of flow force in a cavitation state inside a water hydraulic poppet were studied numerically. Valves. International Journal of Heat and Mass Transfer, 111, 1–16. https://doi.org/10.1016/j.ijheatmasstransfer.2017. 03.100
[6] Jin, Z. J., Gao, Z. X., Qian, J. Y., Wu, Z., & Sunden, B. (2018). Inside globe valves, a parametric investigation of hydrodynamic cavitation. ASME Journal of Fluids Engineering, 140(3), 031208.
https://doi.org/10.1115/1.4038090
[7] Jin, Z. J., Qiu, C., Jiang, C. H., Wu, J. Y., & Qian, J. Y. (2020). Cavitation flow through a sleeve regulating valve is affected by valve core shapes. Journal of Zhejiang University-SCIENCE A, 21(1), 1–14. https://doi.org/10.1631/jzus.A1900528
[8] Li, J. Y., Gao, Z. X., Wu, H., & Jin, Z. J. (2020a). Numerical investigation of cavitation suppression techniques in globe valves. Applied Sciences, 10(16), 5541(1–13). https://doi.org/10.3390/app10165541.
[9] Li, S., Li, C., Li, Z., Xu, X., Ye, C., & Zhang, W. (2020b). Dynamic flow balance valve design optimization and experimental performance testing. Eng. Appl. Comput. Fluid Mech, 14(1), 700–712. https://doi.org/10.1080/19942060.2020.1756914
[10] Lin, Z., Ma, C., Xu, H., Li, X., Cui, B., & Zhu, Z. (2017).
[11] Numerical and experimental studies of sleeve regulating valve hydrodynamic properties. Instrumentation and Flow Measurement, 53, 279–285. https://doi.org/10.1016/j.flow measinst.2016.12.001
[12] Lisowski, E., & Filo, G. (2017). The use of a CFD approach was used to analyses a proportional control valve flow coe [15] coefficient. Flow Measurement and Instrumentation, Flow Measurement and Instrumentation, Flow Measurement and Instrumentation53, 269–278. https://doi.org/10.1016/j.flowmeasinst.2016. 12.009
[13] Liu, X., Wu, Z., Li, B., Zhao, J., He, J., Li, W., Zhang, C., & Xie, F. (2020). Inlet pressure has an effect on the cavitation characteristics of a regulating valve. Fluid Mech. Eng. Appl. Compute. Fluid Mech., 14(1), 299–310. https://doi.org/10.1080/19942060.2020. 1711811
[14] Mosavi, A., Shamshirband, S., Salwana, E., Chau, K. W., & Tah, J. H. (2019). A unique hybrid model of computational fluid dynamics and machine learning was used to predict the performance of a multi-input bubble column reactor. Fluid Mech, 13(1), 482–492. https://doi.org/10.1080/19942060. 2019.1613448
[15] Qian, J. Y., Gao, Z. X., Hou, C. W., & Jin, Z. J. (2019Mechanical heart valves and control valves: A detailed overview of cavitation in valves. Bio-Design and Production, 2(2), 119–136. https://doi.org/10.1007/s42242-019-000 40-z
[16] Qian, J. Y.,Gao, Z. X., Li,W.Q., & Jin, Z. J. (2020aSuppression of the bileaflet Cavitation Mechanical heart valves Cardiovasc. Eng. Tech, 11(6), 1–12. https://doi.org/10.1007/s13239-020- 00484-w
[17] Qian, J. Y., Hou, C.W., Mu, J., Gao, Z. X., & Jin, Z. J. (2020b). Flux through control valves in nuclear power plants: Valve core shapes analysis. Nuclear Engineering and Technology, 52(10), 2173–2182. https://doi.org/10.1016/j.net.2020. 03.008
[18] Qian, J. Y., Wu, J. Y., Gao, Z. X., & Jin, Z. J. (2020c). Effects of a throttling window on feed-water valve flow rates. ISA Transactions, 104, 393–405. https://doi.org/10.1016/ j.isatra.2020.05.017
[19] Qian, J. Y.,Wu, J. Y., Gao, Z. X., & Jin, Z. J. (2020d). Analysis of a Pilot-control globe valve's pilot pipe and damping orifice arrangements. ASME J. Fluids Eng, 142(10), 101210(1–12). https://doi.org/10.1115/1.4047533
[20] Qiu, C., Jiang, C. H., Zhang, H., Wu, J. Y., & Jin, Z. J. (2019). Analysis of a sleeve regulating valve's pressure drop and cavitation. Processes, 7(11), 829(1–16). https://doi.org/10.3390/ pr7110829.

[21] Ramezanizadeh, M., Alhuyi Nazari, M., Ahmadi, M. H., & Chau, K. W. (2019). A Nano fluidic thermo syphon heat exchanger was studied experimentally and numerically. Eng. Appl. Comput. Fluid Mech, 13(1), 40–47. https://doi.org/10.1080/ 19942060.2018.1518272
[22] Saha, B. K., Peng, J., & Li, S. (2020). Cavitation phenomena inside the Pilot stage of the deflector Jet servo-valve have been studied numerically and experimentally. IEEE Access, 8, 64238–64249. https://doi.org/10.1109/ACCESS.2020.298 4481
[23] Singh, D., Charlton, M., Asim, T., Mishra, R., Townsend, A., & Blunt, L. (2020). The effects of additive manufacturing on the global and local performance characteristics of a complicated multi-stage control valve were quantified. Trim. Journal of Petroleum Science and Engineering, 190, 107053(1–13). https://doi.org/10.1016/j.petrol.2020.107053.
[24] Sun, X., Kim, H. S., Yang, S. D., Kim, C. K., & Yoon, J. Y. (2017). The influence of surface roughness on the flow coefficient of an eccentric butterfly valve was investigated numerically. Journal