Semarak International Journal of Mechanical Precision Engineering

Journal homepage:
https://semarakilmu.my/index.php/sijmpe/index
ISSN: 3083-9564

A Proposed Procedure of a Numerical Simulation for the Fluid Flow through a Floating Valve based on Autodesk Computational Fluid Dynamics (CFD)

Nguyen Huu Tho[1,*]

[1] Faculty of Engineering and Technology, Nguyen Tat Thanh University, Ho Chi Minh City 70000, Vietnam

ARTICLE INFO	ABSTRACT

Article history:
Received 4 August 2025
Received in revised form 4 September 2025
Accepted 5 September 2025
Available online 11 September 2025

Keywords:
Floating valve; Solid modeling; numerical simulation; CAD/CAE

A floating valve is a valve that is capable of opening and closing automatically based on the level of water in the tank. This automatic opening and closing is driven by Archimedes' principle and does not require electric control signals. Numerous products within the domestic market have garnered significant consumer attention, yet they continue to exhibit several drawbacks, including the practical malfunctions of the floating valve. Accordingly, the present study employs a numerical simulation to analyze the flow of a fluid through a float valve. To this end, a CAD geometry was modeled in Autodesk Inventor for the floating valve configuration. Subsequent to this, the meshes and results in this paper were obtained through a numerical simulation of engineering fluid dynamics with Autodesk CFD. The numerical simulation results confirmed that the designed valve exhibited a strong correlation with the results obtained using Autodesk CFD. Furthermore, the results are integrated with a discussion, and the main findings are highlighted in the conclusion.

1. Introduction

Simulation-based experiments are an indispensable component of understanding the properties of fluids in the teaching and learning process [1], especially in several subjects such as applied fluid mechanics, modeling and simulation in engineering, mechanical design, etc. Simulation operations of fluid dynamics, such as Solidworks Flow Simulation and Autodesk CFD (computational fluid dynamics), provide a valuable practical application of theoretical concepts. These operations effectively support the improvement of learners' capacity, thereby enhancing their understanding of pressure drops, energy losses, pressures, and fluid flow velocities [2-4]. Kaiphanliam, Nazempour [5] examined the impact of low-cost desktop learning modules in a fluid mechanics class of 27 students. The integration of software into an engineering course offers two distinct benefits to learners. Firstly, the learning environment fosters a comprehensive comprehension of the course material. Secondly, it provides students with practical experience utilizing cutting-edge tools employed in industry. As presented by Untener, Mott [6], the integration of PIPI-FLO software into the mechanical fluid course

* *Corresponding author.*
E-mail address: nhtho@ntt.edu.vn

has been instrumental in preparing students for industrial skills. In the domain of mechanical engineering, the experimental science education method has been effectively implemented, contributing significantly to the advancement of knowledge [7]. Nguyen, Shirk [8] designed and constructed two pieces of laboratory equipment intended for use in fluid mechanics classes at the university. These devices were developed to provide opportunities for students to apply the concepts and principles learned in the classroom, thereby enhancing their understanding and experience. Hadžiahmetović, Blažević [9] conducted a numerical analysis of fluid flow through a valve using Ansys Fluent, comparing hexahedral and tetrahedral meshes with results from Solid Works Flow Simulation. The study revealed that both mesh types yielded analogous outcomes, with minor difference in maximum velocity indicative of laminar flow. The hexahedral mesh exhibited a marginally superior alignment with prior outcomes. While the selection of the appropriate mesh is paramount, both options yielded analogous pressure drop and flow characteristics at varying fixed flow rates. In the contemporary era, CFD-based simulations have witnessed a marked increase in their utilization for the design and optimization of valves. This increased use has enabled a more detailed examination of flow dynamics and facilitated the enhancement of performance. Their research has demonstrated the efficacy of CFD in predicting flow behavior, analyzing various valve types, and enhancing efficiency and safety [10]. Drzymalla, Lay [11] employed CFD simulations to optimize an aerosol chamber for the calibration of low-cost PM sensors by conducting a comprehensive analysis of airflow and particle movement. The optimal positioning of the outlet, which was found to be centrally placed and equipped with a straightener, was determined through numerical analysis. The recommendation of calibrating the zone at a height of 400 millimeters above ground was made, as this location was identified as the point where flow is known to be at its slowest and most symmetrical. Diring, Fromme [12] compared COMSOL Multiphysics (using finite element method [FEM]) and STAR-CCM+ (using finite volume method [FVM]) with experimental data for turbulent flow in a Venturi tube, applying k-ε and k-ω turbulence models. The two software tools provided simulations that exhibited a strong correlation with the measured results. The k-ω model yielded results that were more closely aligned with experimental data over a broader range of differential pressures compared to the k-ε model [13]. Kisiel and Szpica [14] employed the Finite Volume Method (FVM) and SolidWorks Flow Simulation to assess the static characteristics of a prototype differential valve. The study focused on how mesh size, valve opening, and nozzle diameter affect mass flow rate and pressure distribution. Li, Deng [15] employed CFD and orthogonal methods to optimize a pilot-operated control valve, thereby enhancing stability and preventing flow blockage. Their numerical simulations indicated that the pilot valve core's lifting height primarily affects the pilot regulating valve core force.

A significant number of studies have employed CFD to simulate the flow within and beyond the pipe. A subset of studies concentrates on the simulation of fluid flow within the pipe through a designated valve. This study employs numerical simulation to examine the flow through a floating valve.

2. Methodology

This section demonstrates the procedure for analyzing flow dynamics through the floating valve using CFD simulations. The text depicts the modeling approach, the boundary conditions employed in the Autodesk CFD simulation software, and the intricacies of the valve's design. A substantial body of research has been dedicated to the development and application of CFD methodologies. Numerous CFD codes have been developed, and a wide range of free and commercial software tools have been employed in the research and design of mechanical products. The CFD method is a group of numerical methods that are used to analyze, calculate, and predict parameters (e.g., temperature, velocity, pressure) of fluid flow. The computational fluid dynamics (CFD) method is predicated on the solution of the Navier-Stokes equations, which are the equations of conservation of mass, momentum, and energy. CFD techniques have found widespread application in a variety of scientific and technological domains, including: The field of aerodynamics plays a crucial role in the manufacturing of aircraft, while hydrodynamics is a pivotal aspect of valve design in the automotive industry.

Autodesk CFD software is developed by Autodesk. This software application is designed to facilitate the simulation of heat and fluid dynamics on computers. This software plays a pivotal role in the field of fluid-related industries, aiding designers in comprehending fluid processes during the product research and development stage. This tool assists engineers in making optimal design decisions prior to manufacturing products. It facilitates the assessment of energy efficiency and the identification of any potential risks in products before testing and manufacturing. This instrument is a valuable tool for designers in the process of creating new and superior features for products. Presently, Autodesk CFD has had the first versions since 2012 and has been updated annually until 2026. It boasts the following advantages: The interface is characterized by its friendliness and convenience for users in the process of entering input parameters. It facilitates the exchange of data with other graphics software or performance simulation software. The results about the flow through the valve are relatively complete and intuitive.

Autodesk CFD's primary fluid flow solver is based on the FVM, which is the industry standard for CFD because it is highly effective in preserving physical properties such as mass and momentum, which are critical for fluid flow. The FVM is a numerical technique employed to address the problem of an internal or external fluid flow. In this approach, the computational domain is subdivided into a finite number of non-overlapping control volumes (i.e., numerical mesh), with each control volume typically characterized by the physical property value at its center node. Two primary strategies are employed for the generation of a numerical mesh. The initial, more prevalent approach consists of the creation of the mesh domain, followed by the subsequent placement of calculation points at the centers of the control volumes. The alternative strategy involves the initial establishment of computational points and the subsequent construction of control volumes, with the objective of ensuring that the surfaces of these volumes lie at equidistant intervals between adjacent calculation points. The numerical algorithm is comprised of several key stages. The integration of fundamental fluid flow equations over the entire set of control volumes is imperative. The discretization of these equations through integral approximation is essential for transforming them into a system of algebraic equations suitable for nonlinear fluid flow modeling. The solution of the resulting algebraic system using iterative methods is crucial for the analysis. A series of numerical simulations were executed by employing Autodesk CFD software, a sophisticated tool designed for the modeling of fluid flow and heat transfer in complex geometries.

2.1 CAD (computer aided design) based solid modeling approach

The fluidic analysis of a floating valve is a multifaceted process, as illustrated in Figure 1. Initially, the valve's geometry is digitally modeled, with non-essential components being simplified to facilitate subsequent computations. Upon completion, the CAD model is exported and imported into the CFD software, where units are set and solids are grouped as components if necessary. The components are classified as either solid or fluid. Subsequently, the geometry undergoes meshing, which comprises the division of the geometry into elements with nodes for calculations. In addition, both solid and fluid regions are defined, including inlet, outlet, or symmetry surfaces. Structured meshes, such as tetrahedral and hexahedral grids, are advantageous in terms of accuracy and efficiency. The quality of a mesh is evaluated based on its orthogonality, aspect ratio, and skewness.

Following the meshing process, the model is parameterized in the Autodesk CFD. This encompasses the evaluation of geometry, the mesh, the selection of pressure and velocity at the inlet and outlet of the floating valve. Furthermore, the validity of the steady and transient simulations of laminar and turbulent flow was confirmed. Consequently, the identification of gravity's axis and suitable flow models is imperative. The selection of a viscous model is contingent upon the classification of the flow type. At this time, material properties for both solids and fluids have been established, thus enabling the initiation of flow simulation.

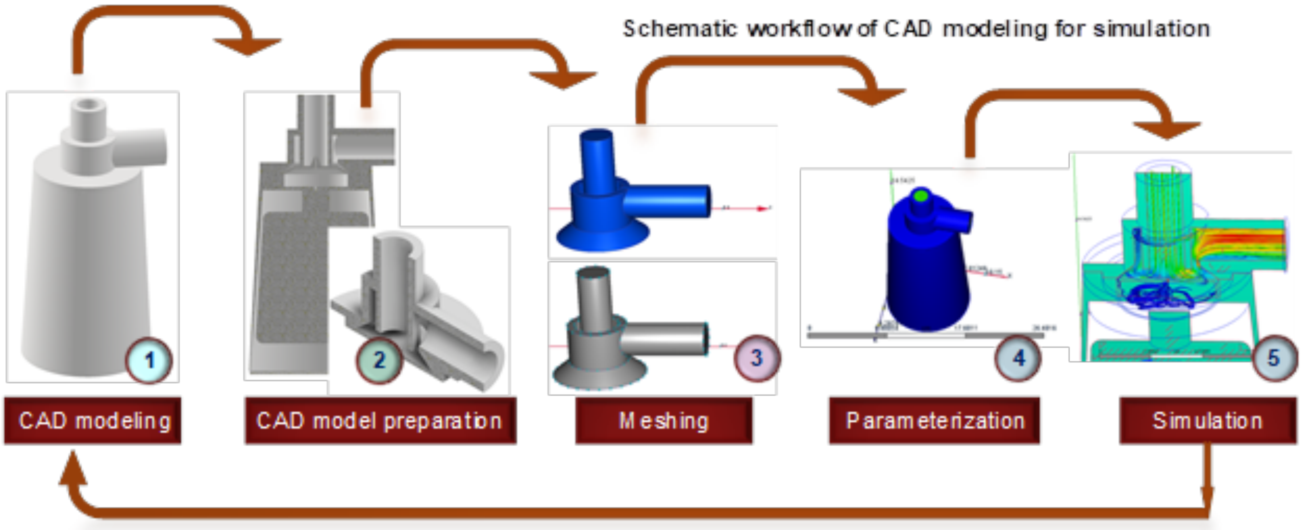


Fig. 1. General procedure of an internal flow simulation using CFD

2.2 General procedure of an internal flow simulation using Autodesk CFD

Step 1: CAD Model Preparation

CFD analysis focuses on the flow of fluid through a given space, neglecting the solid valve components. The creation of a three-dimensional model is necessary to depict the internal region of the valve. The most common method involves sealing the inlet and outlet apertures of the solid valve model and subsequently employing a Boolean operation (e.g., "subtract") to extract the internal volume. The resulting fluid volume is required to be a single, continuous, and fully enclosed solid body, with no gaps or overlapping surfaces. The removal of non-essential features is imperative; these features, while not directly affecting the flow, contribute to mesh complexity through elements such as external threads, small fillets, or part numbers. The configuration of the valve's moving components (e.g., the gate, ball, or plug) delineates the flow path. It is imperative to model the geometry with the valve set to a specific opening percentage for each simulation that is to be executed.

Step 2: Setup in Autodesk CFD

Subsequent to the completion of the CAD model, it is necessary to initiate its operation within the Autodesk CFD. The materials have been assigned accordingly. The user is then prompted to select the fluid volume that was previously created. The materials database should be consulted to determine the appropriate fluid (e.g., water, air, or oil). Subsequently, the boundary conditions will be delineated. This step instructs the software on the manner in which the fluid enters and exits the domain. The application of BCs to the surfaces of the model (i.e., the capped ends) is a critical step in the process.

The inlet face must be selected prior to proceeding. The following are some common BCs:
It is essential to specify a gauge pressure value in order to accurately measure and analyze pressure-related phenomena. This is advantageous when the upstream pressure driving the flow is known.

The volumetric flow rate is defined as the measurement of the rate at which a fluid of constant density and temperature changes in volume. It is imperative to specify the measurement of interest, such as Liters per Minute (LPM) or Gallons per Minute (GPM). The software will calculate the resulting pressure drop.

Velocity: It is essential to specify the velocity of the incoming fluid.

The following is an enumeration of the outlets: The outlet face must be selected. Typically, a pressure of 0 (gauge pressure) is applied, thereby simulating the fluid exiting to the atmosphere.

The following section will address the topic of walls. Conventionally, all other surfaces are regarded as walls with a "no-slip" condition, signifying that the fluid velocity at the wall surface is zero. In most cases, this adjustment is not necessary.

Step 3: Meshing

The mesh discretizes the large fluid volume into a multitude of small elements, where the governing equations (Navier-Stokes) are solved.

Automatic Sizing: Autodesk CFD is equipped with a robust automatic mesher. To initiate the process, select the "Auto Size" option, which will enable the software to determine an appropriate initial mesh size.

Mesh Refinement: To ensure the precision of the measurements, it is necessary to employ a finer mesh in regions characterized by intricating flow patterns or substantial high-velocity gradients. This is of particular importance in the vicinity of the valve seat, the narrowest passage (vena contracta), and any sharp turns. It is possible to select specific surfaces or regions and apply local mesh refinement to create smaller elements there without making the entire model's mesh excessively large.

Step 4: Solving

This section is dedicated to the computational aspect of the process, wherein the solver undertakes iterative calculations to determine the fluid's behavior.

Set Physics: It is essential to ensure that the analysis is set to "Flow." In instances where turbulence is anticipated (as is nearly invariably the case in a valve), it is imperative to ensure the activation of a turbulence model (e.g., k-epsilon or SST). Autodesk CFD generally addresses this automatically in most cases.

Set Iterations: The solver executes a predetermined number of "iterations" to achieve a stable solution. This number can be manually set (e.g., 200-500) or the software can be allowed to run until convergence is automatically detected. Convergence is characterized by a significant reduction in the variation between iterations of the solution.

Solve: To proceed, select the "Solve" button. The simulation can be executed on a local machine or on a cloud server, the latter of which is typically faster and frees up the computer's resources. The solver will generate a convergence plot, which illustrates the stabilization of key variables over time.

Step 5: Post-Processing & Results Analysis

Upon completion of the simulation, the results can be visualized and quantified to ascertain the performance of the valve.

It is essential to employ visualization tools to conceptualize the dynamic processes occurring within the valve.

Planes: The creation of 2D cut-planes enables the visualization of contour plots of pressure, velocity, and other variables. This approach facilitates the identification of regions where high-velocity zones and low-pressure areas exist, thereby allowing for the prediction of potential cavitation events.

Vectors: The display of velocity vectors is essential for the clear visualization of the direction and magnitude of the flow.
Traces (Streamlines): The release of virtual particles from the inlet facilitates the observation of their trajectory through the valve. This method is particularly effective in identifying turbulence and recirculation zones, also known as eddies.
Quantifying Performance: The primary objective is to extract key performance metrics.
The primary result of this phenomenon is referred to as "pressure drop." The calculation of the pressure gradient is achieved by determining the mean pressure on the inlet surface and then subtracting the mean pressure on the outlet surface.
Flow Coefficient: This is the universal standard for valve capacity. The relationship between the pressure drop and the flow rate is elucidated. The calculation of this index can be performed using the following formula, derived from the results of the simulation:

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

where,
Q is the volumetric flow rate (in US GPM).
ΔP is the pressure drop across the valve (in psi).
SG is the specific gravity of the fluid (1.0 for water).
We can now determine the hydraulic resistance ξ of the valve to be [16].

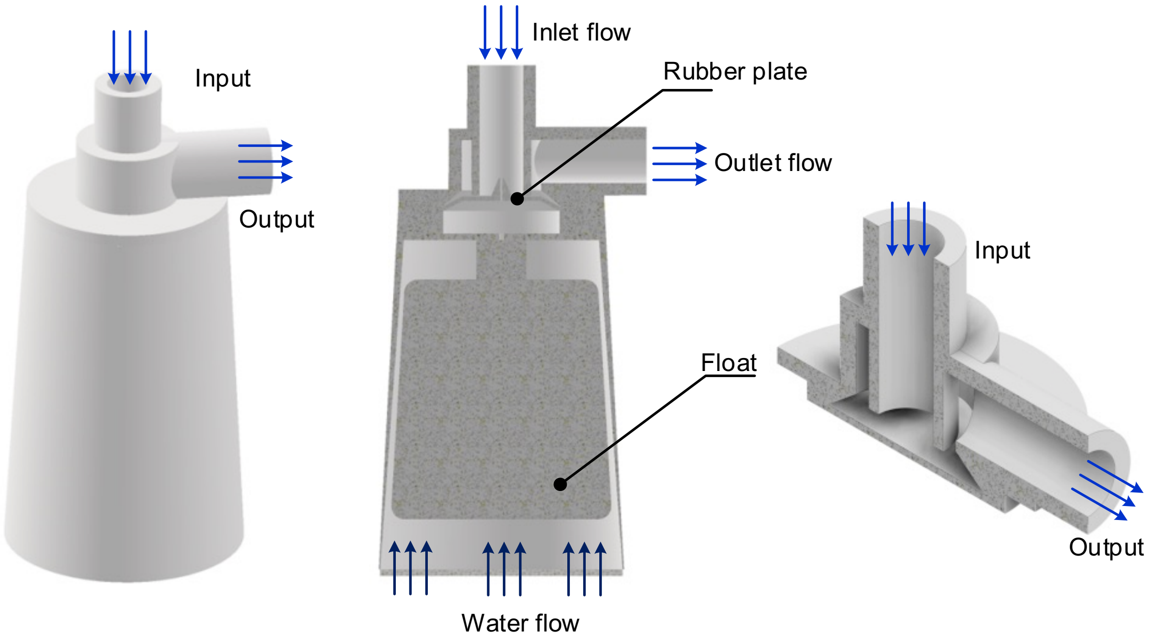
$$\xi = \frac{2\Delta P}{\rho V^2}$$

where, ρ is the density of the fluid, ΔP is the difference in total pressue over the valve and V is the average velocity in the pipe.

2.3 CAD model and numerical simulation of floating valve

The model's design is executed through the utilization of 3D CAD Autodesk Inventor Professional 2026, as depicted in Figure 2, adhering to the specified shape and dimensions. The device under consideration consists of a primary cylindrical/conical body with a vertical input port at the superior extremity and a lateral output port, as illustrated in the three-dimensional CAD model (Figure 2a). The primary component, which is of considerable size, should be placed within the primary chamber. The object's design is intended to facilitate buoyancy, thereby enabling vertical movement in response to changes in water level. The internal cross-section (Figure 2b) elucidates the core mechanism that enables its automatic function. The rubber plate (seal) is affixed to the upper portion of the float, where it fulfills the function of a seal. Its primary function is to exert pressure on the inlet pipe's opening, thereby impeding the flow of liquid or gas. Inlet and outlet ports serve as the entry and exit points for the fluid, respectively. The valve's operation is entirely mechanical and relies on the fluid level within its chamber/in the tank.

As illustrated in Figure 2b, the occurrence of a valve closing due to a high fluid level is indicated by the water level rising from the bottom and exerting an upward force on the float due to the principle of buoyancy. This upward movement exerts pressure on the rubber plate, firmly pressing it against the inlet opening. This effectively sealing of the inlet results in the cessation of inlet flow. The valve opens when the water level in the chamber/tank drops, resulting in a low fluid level. The float is no longer supported and moves downward due to the force of gravity. This process disengages the rubber seal from the inlet, thereby opening the valve and enabling fluid to flow from the input to the output. As illustrated in Figure 2c, the flow path is depicted when the valve is in the open state, indicating a downward position of the float. The fluid enters through the top Input, flows down past the unseated rubber seal, and then turns 90° to exit through the horizontal Output. In summary, this is an automatic control valve utilized for the maintenance of a predetermined fluid level within a tank or reservoir. The device opens to allow refilling when the level is low and closes to prevent overflow once the desired level is reached. The objective of this study is to determine the flow properties of the floating valve. This will be accomplished by testing the pressure applied to the pipe to establish a limit. Subsequently, a computational fluid dynamics simulation is performed to ascertain the water flow through the floating valve. This simulation is conducted using Autodesk CFD [11].



a. 3D CAD model   b. Principal model of flow   c. Flow model in pipeline of a floating valve

Fig. 2. CAD model of floating valve

K-ε turbulent flow model: In this model, the assumption is made that the water flowing through the valve is isothermal and turbulent, and the fluid is incompressible. The continuous equation, the Navier-Stokes equation for the turbulent model, simulates a single-phase fluid flow with the k-ε model used. The time-independent transfer equation for kinetic energy is calculated for unit mass, and the time-independent transfer equation for energy dissipation rate ε is set according to Diring, Fromme [12]. The coefficients of the k-ε perplexing model are predefined by default values [12, 13]:

$$C_{\varepsilon 1} = 1,44; \quad C_{\varepsilon 2} = 1,92; \quad C_{\mu} = 0,09; \quad \sigma_k = 1; \quad \sigma_{\varepsilon} = 1,3$$

The following steps must be taken in order to simulate CFD for the internal flow through the floating valve: The initial step in the procedure includes the establishment and subsequent adjustment of the boundary condition with the pressure at the two inlets, a process that is facilitated by the pressure gauge. The density of the fluid is measured at 1000 kg/m^3, and its dynamic viscosity is μ = 1.855×10^{-5} Pa·s.

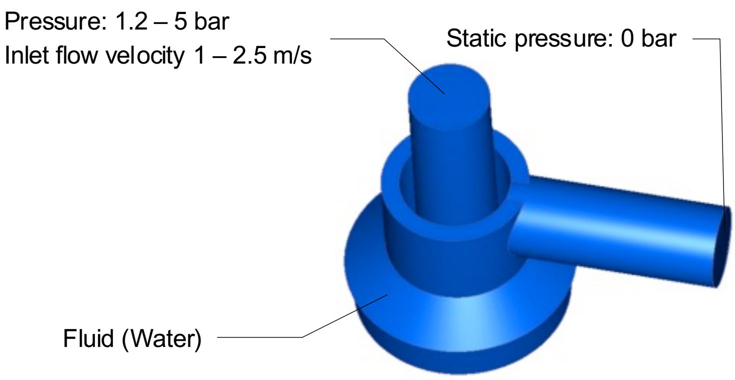


Fig. 3. Volume of fluid flow

The second step in the process entails the configuration of the computational domains for the CFD model of the floating valve. The computing space is defined as the three-dimensional volume encompassing the CAD model for the floating valve, as illustrated in Figures 3 and 4.

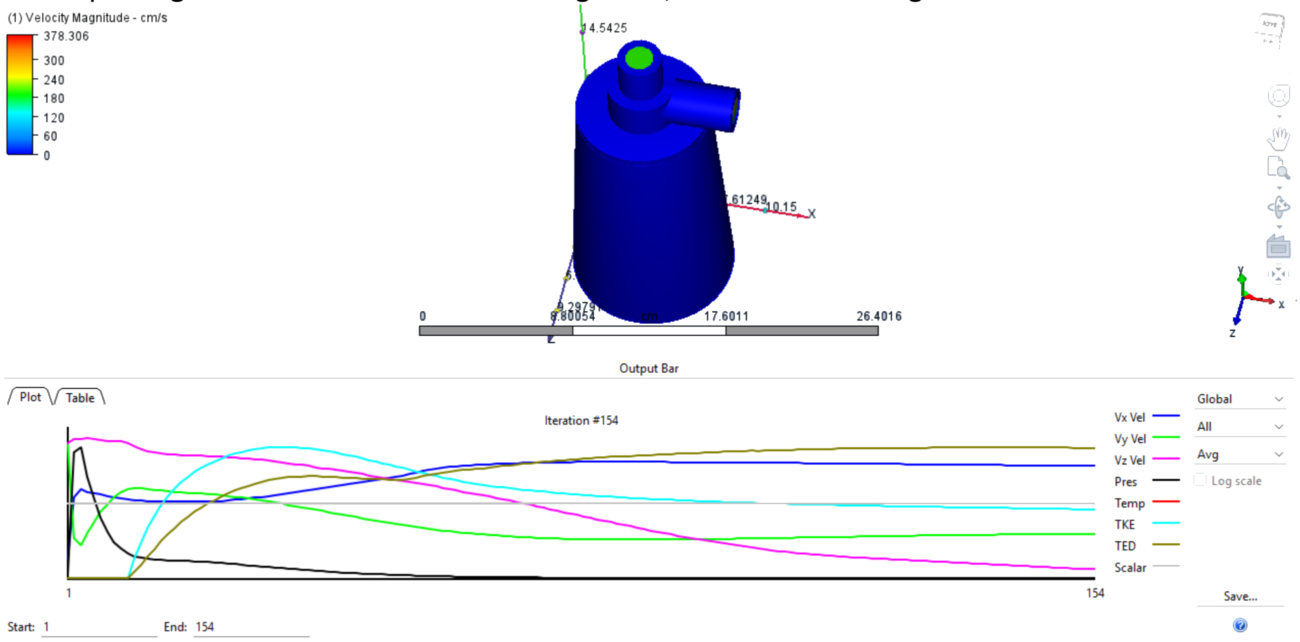


Fig. 4. Computation domain for CAD model of valve

The third step in the process is to establish calculation goals. The mean value of static pressure (Pa) and the mean value of velocity (m/s) at the inlet surface are demonstrated in Figures 3 and 4, respectively. Simultaneously, several observation objectives were established, including the velocity at the outlet and the pressure drop of the valve.

The fourth step 4 in process is meshing generation. In the CFD method, the study area is segmented into elements, with the corners of these elements designated as nodes. The nodes and elements constitute a mesh. The mesh solution selection is automatic, designated as Autosize. The independence of the mesh from the simulation results is ensured through the setting, Enable Adaptation. The meshing model is configured with automatic default modes and adapted in Autodesk CFD Simulation to expeditiously calculate results with acceptable accuracy. The meshing model contains a total of 59,749 nodes, including 12,536 fluid nodes, 47,213 solid nodes, and 250,274 total elements. Of these, 36,455 are fluid elements and 213,819 are solid elements. The model also includes one inlet and two outlets, in addition to zero unknowns. The simulations were executed on a computer with the following configuration: The Dell workstation is equipped with a 3.00-GHz central processing unit (CPU), the Windows 11 64-bit operating system, 4 GB of video memory, and 32 GB RAM. The meshing results are illustrated in Figure 5.

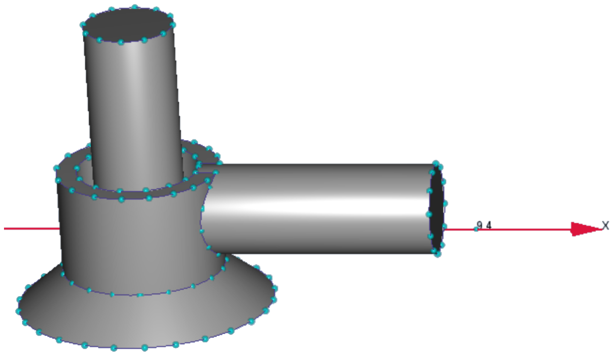


Fig. 5. Meshing model of internal flow through the floating valve

3. Results and Discussion

In this paper, the problem of numerical simulation of water liquid flow through a valve has been calculated and analyzed. Water flow through the valve was measured at inlet pressures ranging from 1.5 to 5 bar. The density of water is 1000 kg/m3, with the corresponding dynamic viscosity of 1.855×10−5 Pa·s. According to the established technical standards, the average velocity of water flow through the valve is measured at 2 meters per second. The input data for the numerical simulation are set by default according to Autodesk CFD 2026 with an educational license.
According to the continuity equation, the liquid flow through the valve is expressed as follows.

$$Q = Av = \frac{\pi d^2}{4} v = \frac{3.14(0.0166\,\mathrm{m})^2}{4} \cdot 2\mathrm{m/s} = 0.000433\mathrm{m}^3/\mathrm{s}$$

Reynolds of fluid flow through the floating valve:

$$\mathrm{Re} = \frac{V \cdot d \cdot \rho}{\mu} = \frac{2(\mathrm{m/s}) \times 0.0166\ \mathrm{m}\ \times 1000(\mathrm{kg/m}^3)}{1.855 \times 10^{-5}\ \mathrm{Pa} \cdot \mathrm{s}} = 17.90 \times 10^5$$

It has been determined through rigorous analysis and evaluation that, this study showed the established values of the Reynolds number and the velocity of the flow, the flow through the valve is turbulent. This determination is made on the basis that the Reynolds number, Re, is greater than 4000. The flow analysis through the valve is represented as an open valve structure with tetrahedral mesh mode. The meshing model contains a total of 59,749 nodes, including 12,536 fluid nodes, 47,213 solid nodes, and 250,274 total elements. Of these, 36,455 are fluid elements and 213,819 are solid elements.

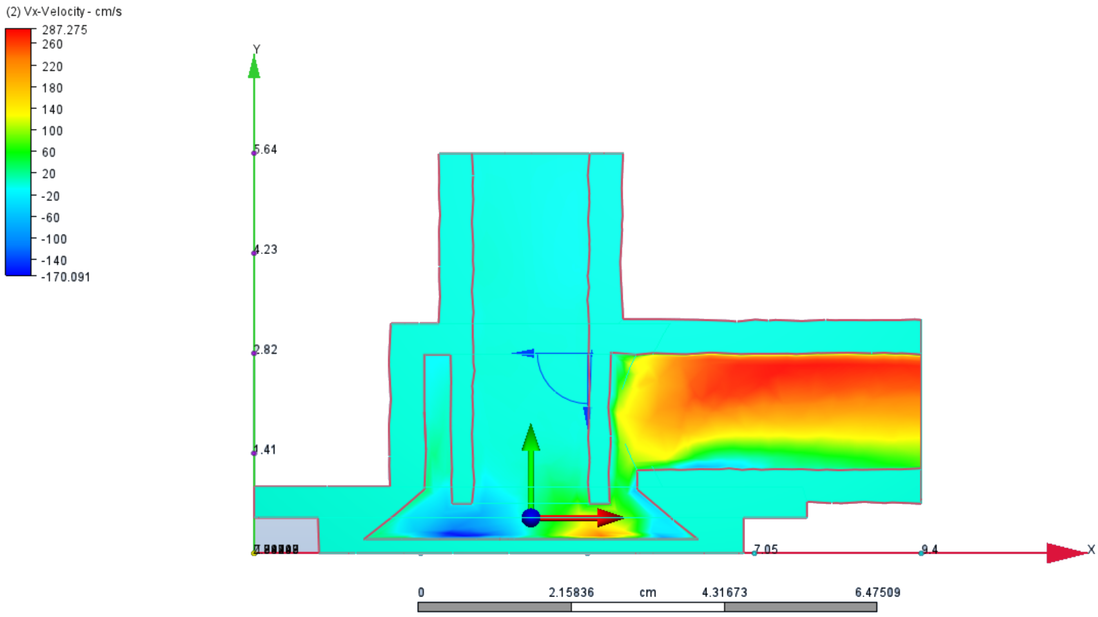


Fig. 6. The velocity distribution of fluid flow in the x direction

As illustrated in Figure 6, the simulation results of the flow velocity through the valve are presented. Given the negligible velocity in the x direction at the inlet pipe, the static pressure at the valve outlet is 0. Figure 5 demonstrates that following its passage through the valve core, the flow velocity exhibits an increase due to the emergence of turbulent flow. Consequently, the velocity can reach 2.87 m/s.

As illustrated in Figure 7, the simulation results of the flow velocity through the valve are presented. It has been established that, given a y-direction velocity of 2 m/s at the inlet pipe, the static pressure at the valve outlet is 0. Figure 6 demonstrates that, subsequent to passing through the valve core, the flow velocity tends to increase at the position where the area changes suddenly (the position in red as in Figure 6). At this point, the velocity can reach a maximum of 2.5 m/s in the y-direction. As illustrated in Figure 8, the trajectory of fluid flow through the float valve is delineated.

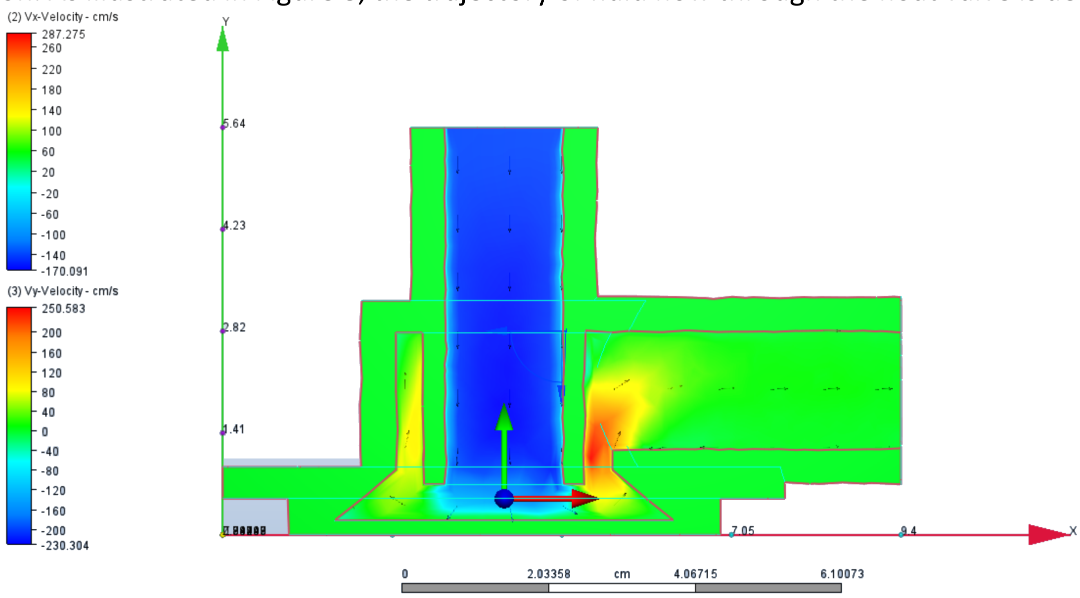


Fig. 7. The velocity distribution of fluid flow in the y direction

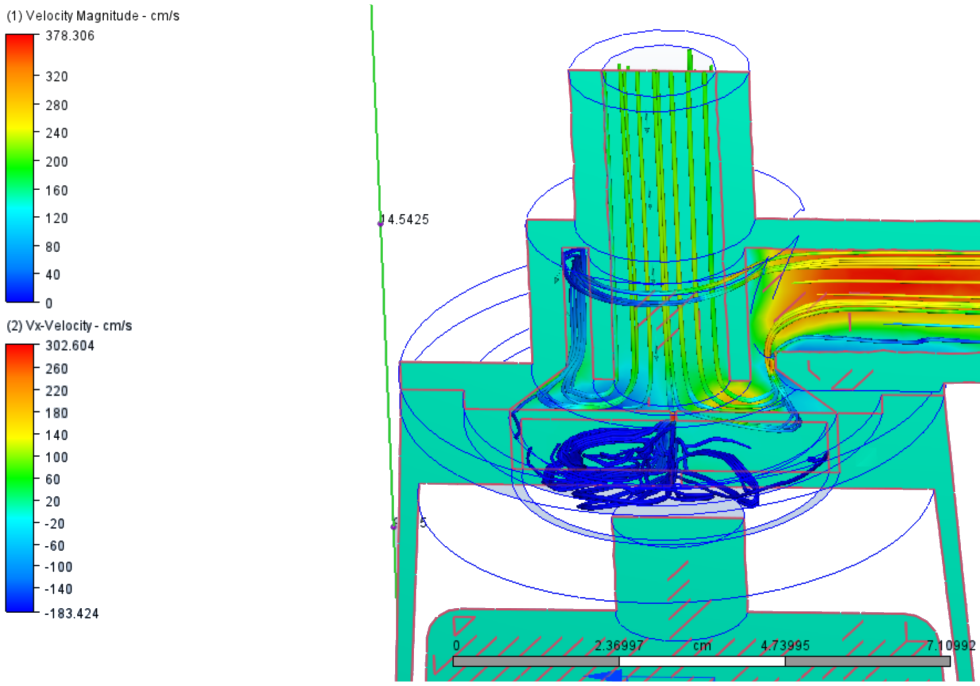


Fig. 8. The trace of fluid flow through the valve

$$\xi = \frac{2\Delta P}{\rho V^2} = \frac{2\times(5.02768-5.0)\times10^5 \text{ Pa}}{1000 \text{ (kg/m}^3)\times(2 \text{ m/s})^2} = 1.34$$

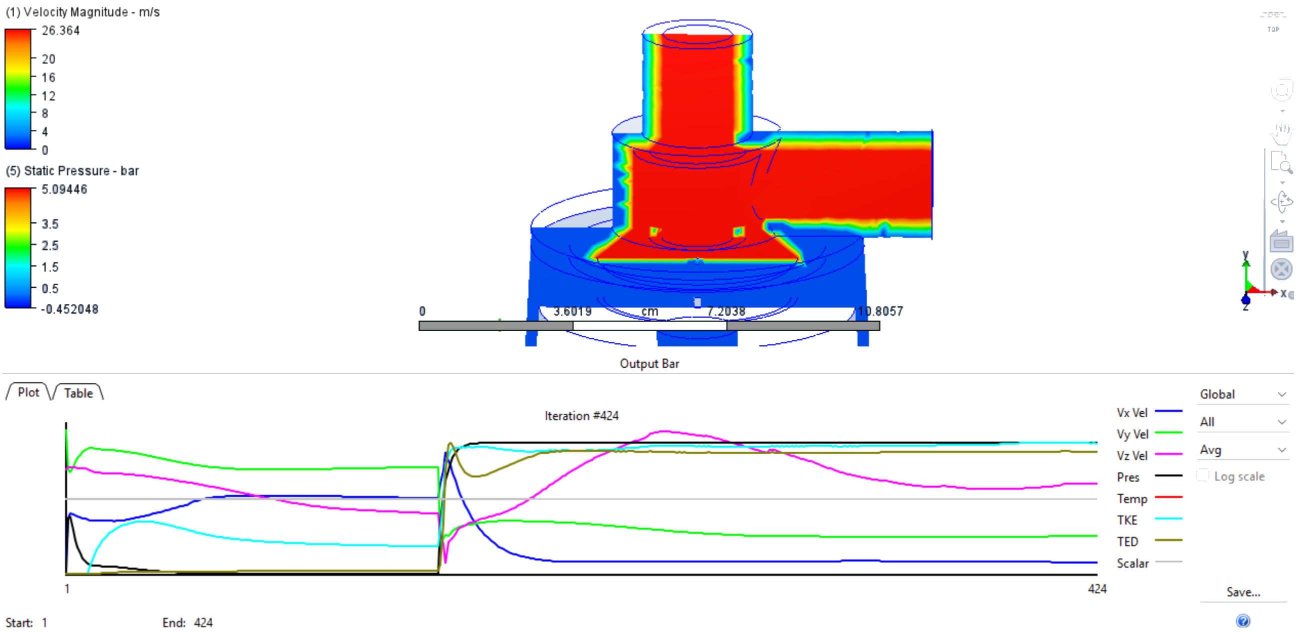


Fig. 9. The velocity distribution of fluid flow in the x direction with inlet parameters v = 2 m/s, p = 5 bar

Increasing the pressure at the inlet pipe to 5 bar results in the initial velocity remaining constant at 2 m/s, while the pressure at the valve outlet pipe is 0. Figure 9 presents the simulation outcomes of the flow through the converging valve following 424 calculation loops. The maximum velocity of the flow through the valve at the outlet in the x direction is 2.25787 m/s, and the static pressure in the valve outlet pipe also increases to nearly 5 bar.

It is now possible to ascertain the hydraulic resistance parameter, designated as ξ, of the valve. This resistance is determined to be equal to [16].

4. Conclusions

In the development trend of valve structures, the CFD method has emerged as the most prevalent approach in recent times. Autodesk CFD software is one of the CFD software options available, and it possesses numerous advantages and conveniences for users. The interface is characterized by its user-friendliness, facilitating seamless interaction with graphics software and other performance simulation software. It enables the acquisition of comprehensive and intuitive results concerning the flow through the valve. In order to ensure the broad applicability of the simulation results obtained from Autodesk CFD software, it is essential to evaluate their reliability through the utilization of analytical methods. The present study employed a numerical simulation of CFD to analyze the fluid flow through a floating valve. The simulation determined the velocity and pressure distribution of flow in the valve's internal structure. Furthermore, the hydraulic resistance of the floating valve was determined to validate the proposed design. These findings substantiated the efficacy of Autodesk CFD (employing the RNG k-ε turbulence model) in generating reliable results when investigating and simulating the flow through the valve. The software's versatility extends to its applicability in both research and development of mechanical engineering design. Moreover, this study also demonstrated the effectiveness of applying simulation tools in teaching modeling and simulation in engineering for students majoring in mechanical engineering technology.

Acknowledgement
This research was not funded by any grant.

References

[1] Andrew Garrard, Krys Bangert, and Stephen Beck. "Large-scale, multidisciplinary laboratory teaching of fluid mechanics." *Fluids* 5, no. 4 (2020): 206.

[2] Manuel Rodríguez-Martín, Pablo Rodríguez-Gonzálvez, Alberto Sánchez Patrocinio, and Javier Ramón Sánchez Martín. *Short simulation activity to improve the competences in the Fluid-mechanical Engineering classroom using Solidworks® Flow Simulation*. in *Proceedings of the Seventh International Conference on Technological Ecosystems for Enhancing Multiculturality*. 2019.

[3] DM Fraser, R Pillay, L Tjatindi, and JM Case. "Enhancing the learning of fluid mechanics using computer simulations." *Journal of Engineering Education* 96, no. 4 (2007): 381-388.

[4] Ronald R Gutierrez, Frank Escusa, Joseph A Lyon, Alejandra J Magana, Jose H Cabrera, Richard Pehovaz, Oscar Link, German Rivillas-Ospina, Guillermo J Acuña, and Julio M Kuroiwa. "Combining hands-on and virtual experiments for enhancing fluid mechanics teaching: A design-based research study." *Computer Applications in Engineering Education* 30, no. 6 (2022): 1701-1724.

[5] KITANA M Kaiphanliam, Arshan Nazempour, Paul B Golter, BERNARD J Van Wie, and Olusola O Adesope. "Efficiently assessing hands-on learning in fluid mechanics at varied Bloom's taxonomy levels." *International Journal of Engineering Education* 37, no. 3 (2021): 624-639.

[6] Joseph A Untener, Robert L Mott, and Buck Jones. *Preparing students for industry by integrating commercial software into coursework*. in *2015 ASEE Annual Conference & Exposition*. 2015.

[7] Daniel GF Huilier. "Forty years' experience in teaching fluid mechanics at Strasbourg University." *Fluids* 4, no. 4 (2019): 199.

[8] Thuc Nguyen, Ted Shirk, and Kyle Rufer. "Triple Falling-Ball Viscometer Small-Scale Pipeline System." ENT 497/498 Senior Design (Final Report), Department of Engineering Technology, College of Liberal Arts and Applied Science, Miami University, 2021.

[9] Halima Hadžiahmetović, Rejhana Blažević, and Sanda Midžić Kurtagić. "NUMERICAL SIMULATION OF FLUID FLOW THROUGH THE VALVE." *Annals of DAAAM & Proceedings* 34 (2023):

[10] Nguyen Huu Tho, Phan Hoang Phung, Huynh Van Nam, and Nguyen Vu Anh Duy. "Geometry Improvement and Flow Simulation in the water level control valve based on the CAD/CAE and DOE integrated system." *Jurnal Polimesin* 19, no. 2 (2021): 98-115.

[11] Jan Drzymalla, Yannic Lay, Marc Sauermann, and Andreas Henne. "CFD Simulations Of An Aerosol Chamber For Calibration Of Low-Cost Particulate Matter Sensors." (2023):

[12] A Diring, L Fromme, M Petry, and E Weizel. *Comparison Between COMSOL Multiphysics® and STAR-CCM+® Simulation Results and Experimentally Determined Measured Data for a Venturi Tube*. in *Proceedings of the Excerpt from the Proceedings of the 2017 COMSOL Conference in Rotterdam*. 2017.

[13] David C Wilcox, *Turbulence modeling for CFD*. Vol. 2. 1998: DCW industries La Canada, CA.

[14] Marcin KisielDariusz Szpica. "Determination of static flow characteristics of a prototypical differential valve using computational fluid dynamics." *acta mechanica et automatica* 18, no. 4 (2024):

[15] Shuxun Li, Guolong Deng, Yinggang Hu, Mengyao Yu, and Tingqian Ma. "Optimization of structural parameters of pilot-operated control valve based on CFD and orthogonal method." *Results in Engineering* 21 (2024): 101914.

[16] John E Matsson, *An Introduction to SOLIDWORKS Flow Simulation 2025*. 2025: SDC publications.