Environment. Technology. Resources. Rezekne, Latvia
Proceedings of the 16th International Scientific and Practical Conference. 2025. Volume I, 583-587

Numerical Simulation of Fluid Flow Through a Non-Return valve: an Integrated Approach for teaching in Fluid Mechanics and Mechatronics

Cvetelina Velkova
Department of Mechatronmics.
Naval Academy N.Vaptsarov, Varna, Bulgaria
Varna, Bulgaria
cvvelkova@nvna.eu

***Abstract*—Non-return valve is a critical component in fluid mechanics and mechatronic systems, ensuring unidirectional flow while preventing backflow, which is essential for system efficiency and safety. These valves are widely used in hydraulic, pneumatic, and process control systems, playing a vital role in preventing pressure surges, cavitation, and mechanical failure. Understanding the fluid dynamics behavior inside a non-return valve is essential for optimizing its design and performance. However, due to the complex flow interactions within the valve, analytical solutions are often insufficient, necessitating numerical simulations for accurate analysis. This study presents a numerical simulation model of fluid flow through a non-return valve using Computational Fluid Dynamics (CFD) techniques. The primary objective is to provide an integrated learning approach that enhances students' and engineers' understanding of the interaction between fluid flow and valve dynamics. The research employs CFD simulations to visualize pressure distributions, velocity profiles, and turbulence effects under different flow conditions, offering insights into the valve's operational characteristics. The methodology includes Theoretical background on valve mechanics and fundamental fluid flow equations; Numerical modeling using ANSYS Fluent, where the valve geometry, boundary conditions, and turbulence models are defined; Simulation and analysis of pressure drop, velocity distribution, and flow separation within the valve. The results highlight how flow conditions influence valve efficiency, showing pressure losses, cavitation risks, and flow separation effects. The study demonstrates that numerical simulations provide a powerful tool for engineering education, allowing students to visualize complex flow behaviors and develop practical skills in CFD analysis. By integrating simulation-based learning into engineering curricula, this approach strengthens problem-solving abilities and prepares future engineers for real-world challenges in fluid mechanics and mechatronics**

Keywords— non-Return Valve, flow simulation, CAE, education, training simulation.

I. Introduction

Check valves, also known as non-return valves (NRV), are essential components in fluid mechanics and mechatronics, ensuring unidirectional flow while preventing backflow. These valves are widely used in pumps, compressors, hydraulic and pneumatic systems, where maintaining controlled flow is critical for system efficiency, safety, and reliability. The design and performance of check valves directly impact pressure losses, flow stability, and cavitation risks, making their study crucial for engineers working in fluid transport and control applications.

The ability to model and simulate fluid flow through check valves is essential for understanding and optimizing their performance. Due to the complex interactions between turbulent flows, pressure distributions, and vortex formations, analytical methods alone are insufficient for accurate analysis. Therefore, Computational Fluid Dynamics (CFD) has become an indispensable tool for predicting flow behaviour, pressure losses, and turbulence effects within check valves.

Integrating CFD-based simulations into fluid mechanics and mechatronics provides students with practical learning experience, bridging the gap between theoretical concepts and real-world applications. This study aims to develop an integrated learning approach that enhances students' and engineers' understanding of fluid

Online ISSN 2256-070X
https://doi.org/10.17770/etr2025vol1.8641

dynamics, numerical modelling, and control systems through numerical simulations of check valve flow.

A. Objectives of the Study

The primary objective of this research is to present a numerical simulation model for analysing fluid flow through a check valve using CFD techniques. This approach serves both as a research tool for valve optimization and an educational framework for teaching fluid mechanics and mechatronics. The key goals of this study include:

- Providing a detailed analysis of flow behaviour, pressure distribution, and turbulence effects inside a check valve.
- Demonstrating the use of CFD modelling techniques in ANSYS Fluent for valve flow simulation.
- Comparing numerical results with classical fluid mechanics principles to validate model accuracy.
- Enhancing engineering education by integrating simulation-based learning into fluid mechanics and mechatronics curricula.

By addressing these objectives, this study contributes to the optimization of check valve design, offering insights into flow regulation, energy efficiency, and system stability in fluid-based engineering applications.

A thorough review of fluid mechanics, numerical methods, and turbulence modelling is necessary to support the theoretical and computational framework of this research. The selected literature sources provide foundational knowledge on fluid flow, numerical modelling, and valve optimization:

Sources [1] and [2] provide essential background on fluid mechanics, conservation laws (mass, momentum, energy), and governing equations (Navier-Stokes equations). These principles are fundamental for understanding flow behaviour in check valves, particularly in relation to pressure variations and flow separation.

Source [8] offer an in-depth discussion on boundary layer theory, crucial for analysing flow interactions near valve walls and transition from laminar to turbulent flow.

Sources [3] and [7] provide detailed discussions on the Finite Volume Method (FVM), which is widely used in CFD solvers such as ANSYS Fluent. These sources emphasize the importance of mesh refinement, discretization techniques, and numerical accuracy in fluid flow simulations.

Sources [6] is a foundational reference in numerical heat transfer and fluid flow, introducing techniques for solving Navier-Stokes equations and turbulence modelling, both of which are crucial for CFD-based valve simulations.

Source [5] introduces computational fluid dynamics (CFD), including practical aspects of grid generation, turbulence models, and solver settings, making it a valuable resource for numerically analysing check valve flows.

Source [10] provide a comprehensive analysis of the k-ε turbulence model, which is widely used for simulating turbulent flows in engineering applications. This model is particularly relevant for checking valve simulations, where vortex shedding, flow separation, and recirculation zones play a significant role in pressure losses and performance evaluation.

Source [9] discuss aerodynamic design principles, which are applicable in the optimization of fluid systems, including valve geometry modifications to minimize pressure losses and enhance efficiency.

Source [4] explores modern control engineering, which is essential for understanding how check valves interact with mechatronic systems in fluid control applications.

Source [11] provides a practical tutorial on check valve simulation, demonstrating real-world applications of CFD tools in engineering design and analysis.

B. Importance of an Integrated Learning Approach

This research emphasizes the educational benefits of integrating numerical simulations into fluid mechanics and mechatronics training:

- Theoretical Understanding: Students develop a strong foundation in fluid mechanics through conservation equations, boundary layer effects, and turbulence modelling.
- Practical Application: By using CFD software (ANSYS Fluent), students gain hands-on experience in numerical modelling, mesh generation, and post-processing of simulation results.
- Interdisciplinary Approach: The study connects fluid dynamics with control engineering, allowing students to understand the role of check valves in automated fluid systems.

This paper presents an integrated approach for teaching fluid flow through check valves, combining theoretical foundations, numerical simulations, and practical applications. By leveraging CFD techniques, students and engineers can gain valuable insights into valve performance, turbulence effects, and energy efficiency, reinforcing the role of numerical modelling in modern engineering education.

II. MATERIALS AND METHODS

A. Preparing the CAd model for simulation

An assembly including the valve's components is part of the original shape. The fluid volume must be created before we can start a flow simulation. We will use the CAD edit functionality to accomplish this. At this point, our three primary goals are: establish an Internal Flow Volume, Eliminate the valve's solid parts, Optimize the geometry of the flow zone for simulation.

First, it is established an internal flow volume.

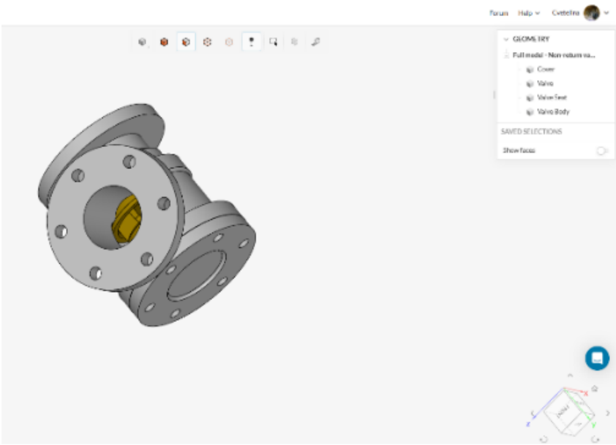

Fig. 1. Imported CAD model of non-return valve.

Fig. 1 shows the imported CAD model of non-return valve. The next step after importing the CAD model is performed elimination of the valve's solid parts. Next the geometry is optimized, and it is obtaining ned final CAD model that is used for fluid flow simulation.

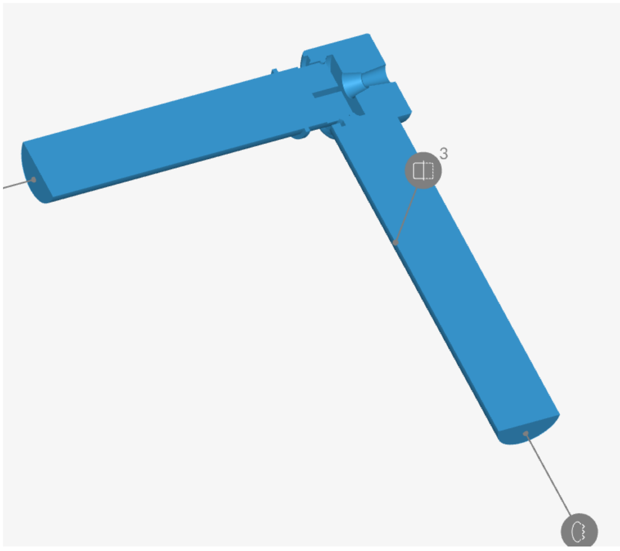

Fig. 2. Final perpared CAD model for fluid flow simulation.

B. Creating a simulation

The developed numerical model of non-return valve under this study is based on the relevant models of Sim Scale web, [11].

We determine the water flow through a valve in this scenario. We choose the incompressible analysis and build the simulation once the fluid speed is subsonic (Mach Number < 0.3).

C. Assigning the material and boundary conditions

Using [11], before assigning the boundary conditions, the material was assigned first. The assigned material for the fluid flow simulation is water with kinematic viscosity $\nu = 9.338e-7$ m^2/s, and density $\rho = 997.3$ kg/m^3.

Using [11], Fig. 3 shows an overview of boundary conditions.

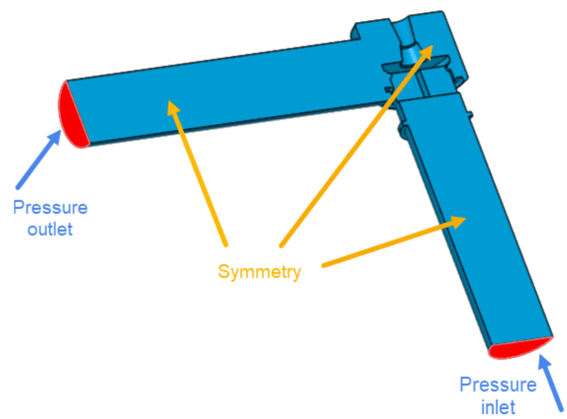

Fig. 3. Overview of the boundary conditions for the non-return valve, [11].

We assigned a pressure inlet condition with a total gauge pressure of 3e+5 Pa. Finally, we will set a pressure outlet condition to the outlet with a fixed value of 0 gauge pressure. This allows flow to easily depart by setting the outlet to atmospheric pressure. The geometry's symmetry condition can now be specified. All surfaces that are immediately on the plane of symmetry will now be given symmetry criteria. Create a new boundary condition of the Symmetry type and choose the three surfaces to do this.

D. Numerical and simulation control

Using the algorithm [11] under the Simulation control setting we change the End Time to 2500 seconds, and a total of 2500 iterations. Result controls can also be used to see how some things of interest behave when they converge. Let's set an Area average result control in this simulation to monitor the inflow values.

E. Mesh

According to [11] the Standard algorithm is recommended for creating the mesh, which is a solid choice in general because it is quite automated and produces decent results for most complex geometries, such as the non-return valve in our instance. Figure 4 depicts the geometry of a non-return valve with a produced mesh.

Frequently, just a few areas in the mesh exhibit significant variations in cell sizes. Cells rise sharply as global mesh refinements are increased.SimScale provides physics-based meshing as an alternative to the traditional Mesher. Using the specified border conditions, this method finds areas that need a finer resolution. One of the local refinement options, such local element size and region refinements, can also be used to accomplish this manually.

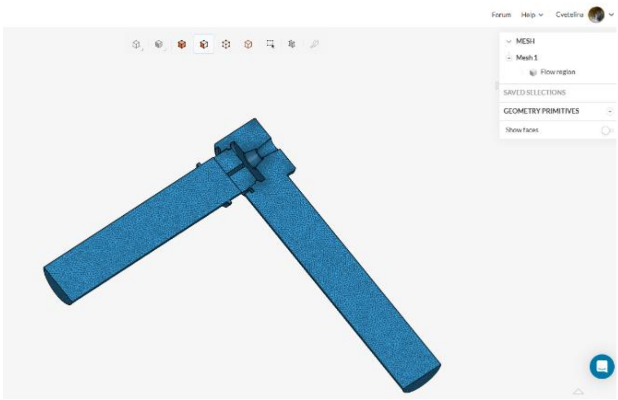

Fig. 4 View of mesh created around non-return valve.

III. Results and Discussion

Convergence is crucial for the fluid flow simulation. As the iterations go on, key parameters are expected to stop changing. At this point, the simulation is considered converged. For the flow simulation through non-return valve the key parameters for convergence that must be monitored are the pressure drops across the valve, and velocity vectors.

After your simulation run is finished, you can view the results using the online post-processor. It's a good idea to assess the simulation's convergence before using the post-processor.

Calculating the pressure drop, monitoring the velocity behavior throughout the system, and determining whether backflow occurs are all fascinating aspects of valve analysis. The outcome statistics calculator is used to calculate the pressure drop across the valve. Fig. 6 shows the pressure visualization through the non-return valve.

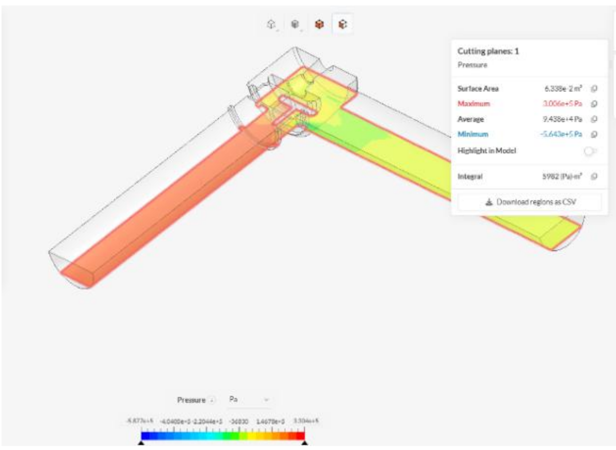

Fig. 5 The pressure visualization through the non-return valve.

From Fig. 5 we can see how the statics tool provides minimum, maxium, and average value over the selected faces from valve. You can now choose the model's inlet face and discover that the average pressure is 9.438e+4 Pa. The average pressure, which is the value imposed by our boundary condition, should be zero if you unselect the intake face and then select the outflow face. Consequently, 9.438e+4 Pa is the total pressure decrease over the valve. The pressure distribution across the valve provides information about the forces acting on the valve disc. The highest pressure is observed near the inlet, while a significant pressure drop is observed in the valve area. The valve disc experiences a force that balances the pressure differential and regulates the flow rate.

The average pressure at the inlet does not match the inlet pressure boundary condition we put in the simulation setup, as you may have noted, but it does at the outlet. You're probably thinking that something went wrong!

Actually, there isn't a mistake. We applied a total pressure condition as the boundary condition at the input, and we post-processed the static pressure as the output field. Because the dynamic pressure from the flow speed is added to the static pressure, the total pressure is higher.

Another significant occurrence to consider while assessing a valve is backflow. Fig. 6 shows the cutting plane with velocity vectors visualizing backflow.

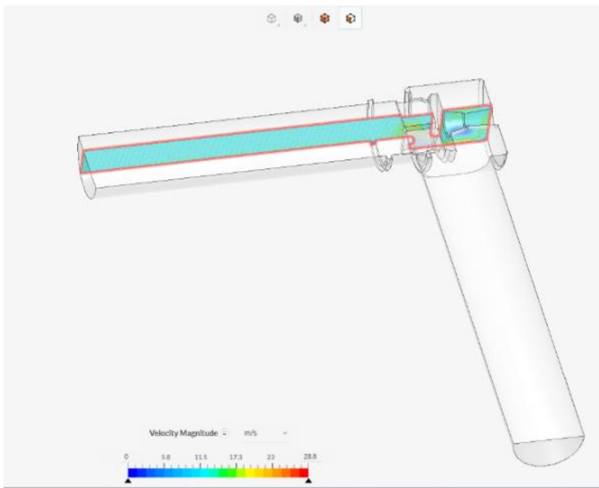

Fig. 6 Cutting plane with V vectors visualizing backflow.

The aforementioned Fig. 6 shows that there are swirls inside the valve cavity and backflow close to the valve's entrance. Because it obstructs flow patterns at crucial points and raises the pressure drop, this may affect the valve's function. The simulation shows that the check valve effectively prevents backflow when the outlet pressure exceeds that at the inlet. The valve disc closes rapidly in response to the pressure difference, preventing any backflow. This behavior highlights the reliability of check valves for maintaining unidirectional flow in fluid systems.

IV. Conclusion

This study demonstrates the effectiveness of numerical simulations in analyzing fluid flow through a non-return (check) valve, offering valuable insights into its performance, pressure distribution, and flow characteristics. The integration of Computational Fluid Dynamics (CFD) simulations in engineering education bridges the gap between theoretical understanding and practical applications, providing an interactive learning approach for students and engineers in fluid mechanics and mechatronics

A. Theoretical Contributions

1. Validation of Fluid Mechanics Principles:

- The numerical results confirm the fundamental principles of fluid dynamics, including mass conservation, momentum balance, and energy conservation.
- The Navier-Stokes equations were successfully applied to model flow through a check valve, validating classical fluid mechanics theories such as boundary layer interactions and turbulence formation.

2. Application of Numerical Methods:

- The study employs the Finite Volume Method (FVM) for numerical discretization, demonstrating its accuracy and efficiency in simulating internal fluid flows.
- The use of turbulence models (k-ε model) effectively captures vortex formations and flow separation, which are crucial for valve performance analysis.

3. Impact of Boundary Layer Effects:

- The simulation results illustrate how boundary layers develop along the valve walls, affecting pressure distribution and velocity profiles.
- The study aligns with Schlichting & Gersten (2016) [8], reinforcing the importance of boundary layer theory in valve design and optimization.

B. Practical Applications

1. Optimization of Valve Design:

- The analysis of pressure drops, and backflow prevention helps in improving valve efficiency and reliability.
- reducing energy losses and cavitation risks in fluid transport systems.

2. Industrial Engineering Applications:

- The findings are relevant to hydraulic, pneumatic, and process control industries, where check valves are critical for maintaining unidirectional flow.
- The study provides a CFD-based framework for engineers to evaluate and optimize valve designs, improving operational efficiency in industrial applications.

3. Engineering Education and Training:

- The study highlights the importance of simulation-based learning, enabling students to visualize complex flow behaviour that are difficult to observe in experimental settings.
- The integration of ANSYS Fluent simulations into academic curricula fosters problem-solving skills and hands-on experience, preparing students for real-world engineering challenges.
- The interdisciplinary approach linking fluid mechanics and mechatronics encourages a deeper understanding of fluid-structure interactions and control systems.

By utilizing numerical simulations, this research provides a comprehensive approach to studying fluid flow in check valves, demonstrating both academic relevance and industrial significance. The integration of CFD techniques into engineering education not only enhances theoretical comprehension but also prepares future engineers for advanced computational analysis in real-world applications.

Future research can expand on this work by investigating:

- Turbulent and unsteady flow conditions in check valve.
- Comparison with experimental data to further validate simulation accuracy.

The study concludes that CFD-based modeling is an indispensable tool for optimizing engineering systems, reinforcing its role in both educational and professional engineering disciplines.

REFERENCES

[1] F. M. White, Fluid Mechanics, New York, NY, USA: McGraw-Hill, 2011.

[2] B. R. Munson, T. H. Okiishi, and W. W. Huebsch, Fundamentals of Fluid Mechanics, Hoboken, NJ, USA: John Wiley & Sons, 2012.

[3] H. K. Versteeg and W. Malalasekera, An Introduction to Computational Fluid Dynamics: The Finite Volume Method, Harlow, UK: Pearson Education, 2007.

[4] K. Ogata, Modern Control Engineering, Upper Saddle River, NJ, USA: Prentice Hall, 2010.

[5] J. D. Anderson, Computational Fluid Dynamics: The Basics with Applications, New York, NY, USA: McGraw-Hill, 2012.

[6] S. V. Patankar, Numerical Heat Transfer and Fluid Flow, Washington, DC, USA: Hemisphere Publishing Corporation, 1980.

[7] J. H. Ferziger and M. Perić, Computational Methods for Fluid Dynamics, Berlin, Germany: Springer, 2002.

[8] H. Schlichting and K. Gersten, Boundary-Layer Theory, Berlin, Germany: Springer, 2016.

[9] A. M. Kuethe and C.-Y. Chow, Foundations of Aerodynamics: Bases of Aerodynamic Design, Hoboken, NJ, USA: John Wiley & Sons, 1997.

[10] B. Mohammadi and O. Pironneau, Analysis of the K-Epsilon Turbulence Model, Hoboken, NJ, USA: Wiley, 1994.

[11] SimScale, "Tutorial: Fluid Flow through a Non-Return Valve," SimScale Docs. [Online]. Available: https://www.simscale.com/docs/tutorials. [Accessed: Jan. 13, 2025].