EPJ Web of Conferences **347**, 02001 (2026) https://doi.org/10.1051/epjconf/202634702001
SCMERD 2025

Experimental Validation of CFD–FEA Flow and Structural Performance Analysis in a Non-Return Multi-Door Reflux Valve

Xolani Prince Hadebe [1*], *Bernard Xavier Tchomeni* Kouejou[1], *Alfayo Anyika* Alugongo[1] and *Desejo Filipeson* Sozinando[1]

[1]Department of Industrial Engineering, Operation Management, and Mechanical Engineering, Vaal University of Technology, Vanderbijlpark, 1911, South Africa

Abstract. This study provides a validated numerical analysis of the hydraulic and structural performance of a non-return multi-door reflux valve, using computational fluid dynamics (CFD) and finite element analysis (FEA), and compares it with experimental tests. Under varying flow conditions, the numerical model predicts important performance parameters such as pressure drop, flow coefficient, velocity distribution, and stress concentration. The pressure difference, flow capacity, and response times of the valve were measured using a large-scale experimental setup; the results demonstrated a strong correlation with the simulation data. While the FEA method successfully captured the high-stress regions without exceeding the material yield limits, the CFD-predicted pressure drop values remained within ±5% of the experimental results. These results validate the accuracy and cost-effectiveness of numerical simulations as an alternative to extensive physical prototyping for valve design and performance evaluation. The study also highlights areas where future work, such as fluid-structure interaction modelling, is needed to more effectively capture transient dynamics.

1 Introduction

Non-return multi-door reflux valves are critical components in large-scale water supply systems, designed to prevent reverse flow and maintain unidirectional fluid movement. Their proper operation is vital for safeguarding infrastructure against contamination, mechanical damage, and operational inefficiencies. Such valves must exhibit low pressure loss, rapid closure, structural robustness, and minimal vibration during operation. These characteristics are essential in applications such as municipal water treatment plants, stormwater management facilities, and high-capacity industrial fluid control systems where backflow could lead to severe consequences [1-2].

To improve valve design and performance, engineers increasingly rely on computational tools such as Computational Fluid Dynamics (CFD) and Finite Element Analysis (FEA). CFD enables detailed investigation of flow characteristics, turbulence, and pressure distribution, while FEA provides insights into structural stresses, deformation, and vibration responses. Together, these methods offer a cost-effective and accurate means of predicting valve behaviour before manufacturing and testing [1]. Studies such as those by Dallstream et al. [3], who applied CFD to analyse swing check valves in nuclear applications, and research on wafer-type check valves validated through SOLIDWORKS Flow Simulation and Fluent (IJSER, 2015), have demonstrated the value of such computational techniques in valve performance evaluation. More recent investigations, such as the adjustable check valve analysis by Filo et al. [4] using ANSYS/Fluent, highlight how turbulence model selection can affect simulation accuracy. In contrast, a 2025 study of a three-way control valve in [5] illustrates CFD's effectiveness in predicting pressure losses and flow coefficients in complex valve geometries. CFD simulations were used in [6] to analyse the hydraulic performance of ball valves under cavitating flow conditions, highlighting the impact of cavitation on pressure distribution, flow characteristics, and valve efficiency. Despite these advances, relatively few studies have integrated CFD and FEA simultaneously and validated both against experimental results for large-scale multi-door re-flux valves. This is particularly true for valves of substantial diameter and pressure rating, such as the DN 1400 PN 17.5 model examined in this study. Manufactured and tested in South Africa, this valve type is specifically engineered for high-volume, high-pressure water conveyance. Its 1.4 m nominal bore and 17.5 bar pressure rating make it suitable for utility-scale waterworks, including major pumping stations, raw water transfer schemes, and large municipal distribution mains. In Southern Africa, where intermittent water supply, pressure fluctuations, and the risk of cross-contamination are prevalent, reliable reflux valves are indispensable for protecting potable water systems against back-siphonage and reverse flow. The present research builds upon a CFD-FEA study that modelled the hydraulic and structural performance of a DN 1400 PN 17.5 non-return multi-door reflux valve

* Corresponding author: xolanih@ithubavalves.co.za

and an experimental testing campaign on an identical valve under controlled hydraulic conditions. By comparing numerical predictions with empirical measurements, this work seeks to bridge the gap between simulation and reality. Similar to recent validation efforts, such as the flow-induced excitation analysis of a four-way reversing valve by Zhang et al. [7], this study not only confirms the predictive reliability of modern simulation methodologies but also identifies their limitations and opportunities for refinement. In doing so, it provides both a benchmark for future valve design optimisation and a practical contribution to improving large-scale water infrastructure performance in the Southern African context.

This work is structured into five sections. Section 2 briefly describes the experimental setup and the detailed operation of the valve in various scenarios. Section 3 presents the flow analysis for different parameter sets, followed by the valve performance results of the first and second test series, and concludes with a discussion of the experimental results. Finally, Section 4 presents concluding remarks and perspectives.

2 Methodology

2.1 Valve Configuration

The valve analysed and schematically presented in Figure 1(a) is a DN1400 PN17.5 multi-port reflux valve composed of several semi-circular flaps articulated inside a circular conduit. This retrofit multi-way check valve is designed to allow fluid flow in one direction while effectively blocking reverse flow. Its multi-hinged doors ensure one-way passage and prevent backflow. The commercial multiport reflux valve used in this study, shown in Figure 1(b), conforms to industry standards, with its material composition, dimensions, and operating mechanism being clearly specified.

The multi-door check valve represents an advanced engineering solution to address backflow challenges in pipelines. Its design minimises pressure losses, mitigates water hammer effects, and enhances overall system efficiency. However, the symmetrical geometry and dynamic door behaviour introduce specific challenges for both numerical modelling and experimental validation.

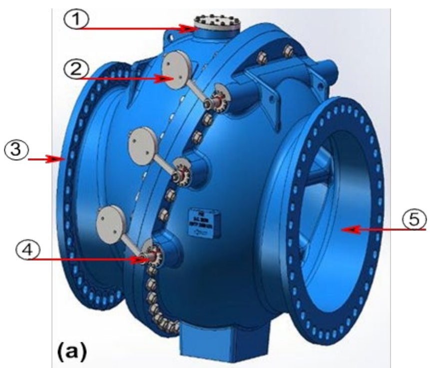


Fig. 1. Non-return multi-door reflux valve (a) schematic multi-door valve (1)-Inspection cover. (2)-Door's lever arm & counterweights, (3)-Outlet body, (4)- Door's main spindle (b) Physical valve.

2.2 Numerical Approach (CFD-FEA)

2.2.1 Geometry and Mesh

The valve geometry is carefully modelled, taking into account both its variable opening angles up to a maximum of 65.5 degrees (Appendix A.2), depending on the fluid flow rate and the 1400 mm diameter. Dynamic meshing is performed to achieve detailed resolution around valve elements, using smoothing, layering, and remeshing techniques to accommodate variations in valve position. Additionally, a user-defined function is applied to control specific simulation actions, thus improving the accuracy of fluid-structure interaction representation. These simulations provide valuable information, which is often expensive or difficult to obtain through simple physical experiments. A precise 3D CAD model was developed. Fluid volume meshing used tetrahedral elements (SolidWorks 2022 - 3×10^6 tetrahedral elements; $Y_+ \approx 30$), while the structural mesh was optimised for stress convergence (SolidWorks 2022).

2.2.2 CFD Parameters

Computational fluid dynamics (CFD) numerical simulation techniques allow the analysis of flow patterns, pressure distribution, turbulence intensity, backflow tendencies, and overall fluid efficiency under various operating conditions. The process began with the development of discrete fluid models based on the valve geometry, followed by the selection of turbulence models and the definition of boundary conditions (Figure 2). A 3D model of the multi-door check valve is used, incorporating key parameters and clearly defined boundary conditions. Software: SolidWorks 2022.

The CFD analysis used the achievable k-ε turbulence model with improved wall treatment, using inlet conditions based on varying flow rates to simulate realistic hydraulic loads. Boundary conditions included a no-slip wall, a

velocity inlet, and a pressure outlet. For the finite element analysis (FEA), SolidWorks 2022 was used with ductile iron as the main material and rubber gaskets modelled by hyperelastic elements. Constraints were applied to the hinge boundaries and flange attachment, and the applied loads were based on static pressure data derived from CFD.

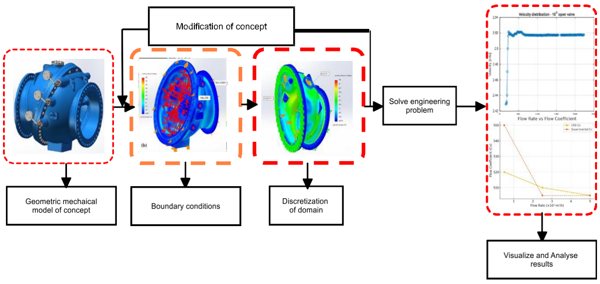


Fig 2: Flowchart of FEA simulation technique

2.3 Experimental Methodology

2.3.1 Test Bench

To assess the flow performance of the multi-way reflux valve, a comprehensive experimental setup was established in a controlled test bay, as depicted in Figure 3(b). The test rig featured a closed-loop piping system with the valve positioned at a critical location. Flow rates were precisely regulated using a motorised pump (Figure 3(a)), and pressure sensors were strategically placed to measure pressure differentials across the valve under various water pressures and response conditions. The study emphasised pressure testing and the evaluation of the effectiveness of reflux prevention (Figure 3(c)). Water served as the test fluid to simulate real-world scenarios relevant to both potable water supply and industrial applications. Key components included a motorised pump test fixture, a temporary sealing plate for valve isolation, a blanking flange functioning as a drain plug, and two certified glycerine pressure gauges rated up to 25,000 kPa. This configuration enabled precise control of flow conditions and facilitated detailed analysis of valve performance.

Simulations. The main results of the study included velocity profiles, pressure maps, flow coefficients, and pressure-drop curves, as well as the maximum stress and displacement across the valve door structures.

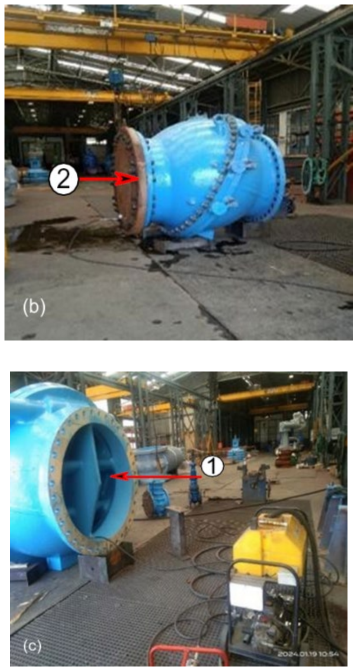


Fig. 3. Experimental setup: (a)-Motorised pump, (b)-2-Outlet valve side view, (c) 1-Inlet front view

2.3.2 Measurement Focus

To ensure the integrity and reliability of the high-pressure multi-door check valve, the detailed procedure should be followed. Initially, the ball isolation valve located on the inlet test plate is opened. The motorised pump is activated to fill the valve body with water, allowing trapped air to escape through the connection point on the upper test plate. Once the valve is filled, a pressure gauge is connected to the upper test plate, and the water pressure is gradually increased to 3,500 kPa, ensuring that both pressure gauges read the same pressure. The test pressure in the valve body and upstream of the doors is maintained for at least 15 minutes, during which time the system is inspected for leaks at flanges, including end covers, universal joints, and packing rings.

The test pressure in the valve body and upstream of the doors is maintained for at least 15 minutes, during which time the system is inspected for leaks at flanges, including end covers, universal joints, and packing rings. In the event of excessive leakage, the pressure is reduced by reopening the ball isolation valve (Fig. 4).

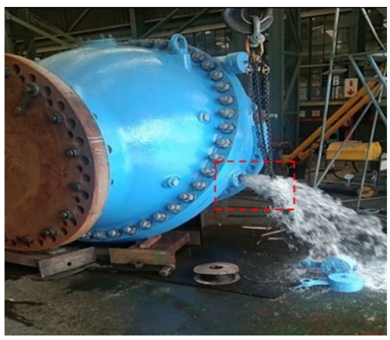


Fig. 4. Failure observed at high shaft bush leak.

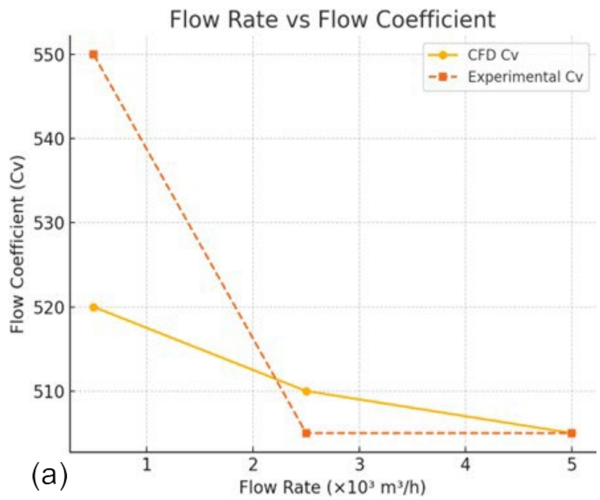
Test pressure is recorded from pressure gauges at the start and end of the test, and it is checked that no leaks are present at the valves or shut-off flanges. Finally, the valve body is carefully examined for any signs of bleeding or leakage, marking the identified leak points with a permanent white highlighter. Validation was performed by comparing these metrics with CFD and FEA results.

3 Pressure Drop (ΔP) and Flow Coefficient (Cv)

To evaluate the hydraulic performance of the non-return multi-door reflux valve, three key parameters were analysed: the flow coefficient (Cv), the pressure drop (ΔP), and the flow rate (Q). Figure 5 presents a comparison between CFD predictions and experimental measurements for Cv and ΔP across varying flow rates. Table 1 shows the simulation and experimental results due to tightness and low pressure loss.

Table 1. Pressure Drop (ΔP) and Flow Coefficient (Cv) relationship (Numerical vs Experimental)

Flow rate ($\times 10^3$ m³/h)	**CFD ΔP (kPa)**	**Exp. ΔP (kPa)**
0.5	2.1	2.3
2.5	8.4	8.7
5.0	14.2	14.9
%Error	**CFD. Cv**	**Exp. Cv**
–9 %	520	550
–3 %	510	505
–5 %	505	505



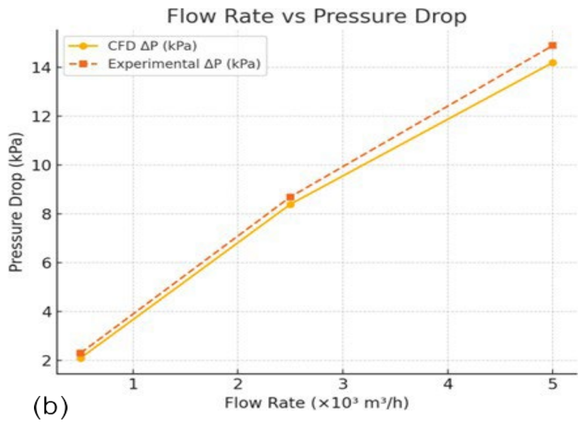


Fig. 5. Relationship between flow rate and (a) flow coefficient, and (b) pressure drop, based on numerical and experimental data

The plot in Fig. 5(a) shows the relationship between flow rate and flow coefficient. Both CFD and experimental data reveal a decreasing trend in Cv as flow rate increases, indicating reduced valve efficiency at higher flow rates. While CFD results predict a smoother and more gradual reduction, experimental values show a more marked drop, particularly between 1×10^3 m³/h and 2×10^3 m³/h. Despite this difference, both methods converge toward a similar Cv value at higher flow rates, confirming general consistency.

The graph in Fig. 5(b) illustrates the relationship between flow rate and pressure drop. As expected, ΔP increases almost linearly with flow rate in both CFD and experimental results, reflecting the quadratic dependence of pressure loss on velocity. The CFD predictions slightly underestimate ΔP compared to experimental values, but the overall trend and slope remain well aligned. This close agreement validates the ability of the CFD model to reproduce real valve performance, particularly for pressure drop behaviour, although minor deviations exist in the prediction of Cv at lower flow rates.

3.1 Flow Field Behaviour

3.1.1 Simulated velocity contours indicated

The CFD analysis displayed in Figure 6 provided a comprehensive view of fluid behaviour in the valve, including velocity profiles, pressure distributions, and shear stresses, facilitating fine-tuning of the valve design. Through extensive simulations and analyses, the research study provides valuable information on the flow performance of the multi-port check valve under pressure test conditions and the effectiveness of backflow prevention (Table A.1 of Appendix A.1). A total flow analysis is performed for a full door opening angle from 0° to 65.5°. Flow streamlines revealed that the valve maintained consistent and reliable performance over a range of flow rates, and no significant areas of flow separation or stagnation were observed, confirming its suitability for various operational scenarios and suggesting that the valve maintains a smooth and controlled flow. The arrows in Figure 6(a)-6(b) indicate the flow velocity streamlines of water inside the valve.

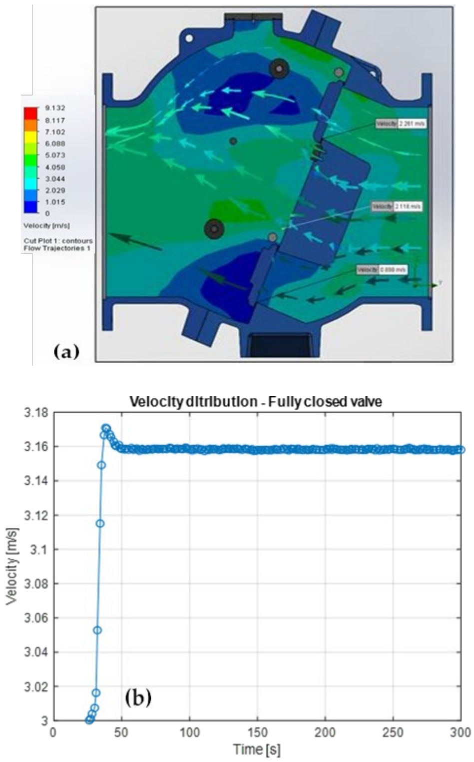
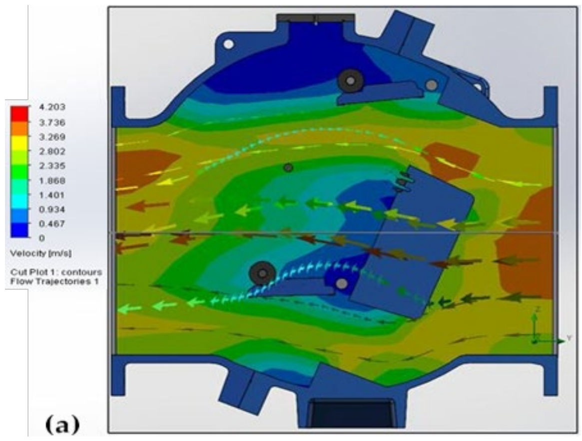


Fig. 6. Velocity distribution – Fully Closed

The velocity contour plot in Figures 6 and 7 reveals that the fluid flow is directed towards the valve doors, partially obstructed by them, causing the formation of a downstream vortex that increases the fluid velocity behind the doors. As the flow moves toward the seat-seal arrangement area downstream of the valve plugs, the velocity increases rapidly until it reaches a maximum downstream of the valve plugs. Beyond this point, the speed gradually increases until it reaches the outlet boundary condition downstream of the valves. The flow velocity decreases as the opening angle of the valve door increases.



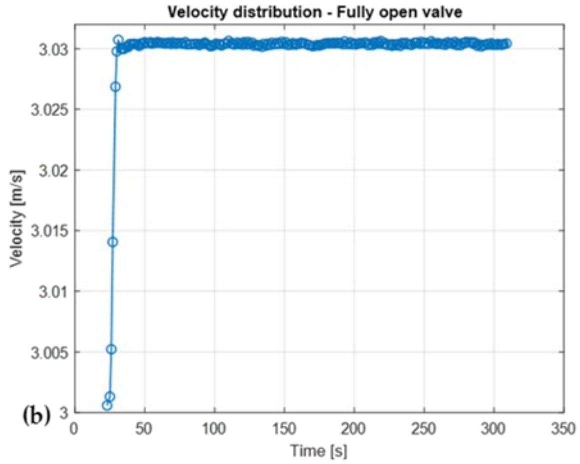


Fig. 7. Velocity distribution: (a) water flow, (b) velocity map of fully open valve

The velocity distribution results displayed in Figure 7(a), shown for both the water flow rate and the fully open valve, show that the valve maintains uniform axial flow at small opening angles, ensuring smooth passage of water through the duct. At intermediate openings, slight flow separation occurs near partially open doors, accompanied by localised vortices; however, the symmetric multi-door configuration effectively balances the flow, reducing large-scale disturbances. As a result, recirculation areas are minimised, highlighting the effectiveness of the design in promoting stable and streamlined flow patterns under different operating conditions (in Figure 7(b)). These observations correlate with transient flow noise and minor oscillations seen in the experimental setup. The velocity contours plotted in Figure 5 reveal that the fluid flow is directed towards the valve doors and partially obstructed by them, causing the formation of a downstream vortex that increases the velocity of the fluid behind the doors. As the flow moves toward the area of the seat–seal arrangement downstream of the valve doors, the velocity rapidly increases until it reaches a maximum downstream of the valve doors. Beyond this point, the speed gradually increases until reaching the outlet boundary condition downstream of the valve doors. The flow velocity decreases as the valve door opening angle increases.

3.1.2 Structural Integrity, simulated stress

The stress distribution on the valve body highlights areas of high stress located mainly near the hinges and sealing surfaces. The deformation behaviour under applied pressure loads is shown in Figures 8(c) and 8(d). For simulation purposes, the material properties shown in Table 3 are defined based on the actual material specifications used in the construction of the valve. Stress analysis identified maximum concentrations around the hinges and sealing surfaces, with maximum stresses of 206 MPa at the inlet body and 103 MPa at the outlet body (Table 2), which is well below the material yield of 310 MPa. Although some regions experienced measurable deformation, the peak displacements of 0.0178 mm at the inlet body and 0.0631 mm at the outlet body remain well within the allowable safety margin, indicating structural integrity under the conditions tested.

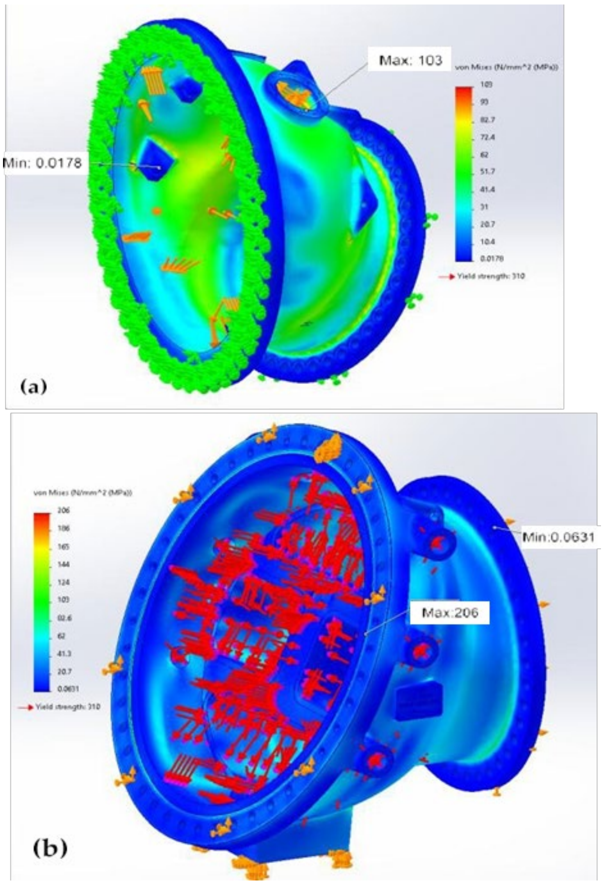


Fig. 8. Maximum hinge region stresses: (**a**) outlet stress distribution; (**b**) inlet stress distribution.

Table 2. Maximum hinge region stresses

Condition (Applied Pressure: 3500 kPa)	Max σ_Von Mises (MPa)	Margin to Yield (%)
Inlet Body	206	66
Outlet Body	103	62

Table 3. Material properties

Properties	Value and Unit
Material	Ductile Iron AS1831 Gr.450-10
Material damping ratio	206800
Poisson's ratio	0.28
Thermal Conductivity	75 W/(m.K)
Specific Heat	450 J/(kg.K)
Elastic Modulus	169 GPa
Shear modulus	77 GPa
Tensile strength	450 MPa
Yield strength	310 MPa
Thermal coefficient	1.1 e^{-5} /K
Mass density	**7100** Kgm^{-3}

3.1.3 Experimental observations

During the tests, the valve showed no visible deformation, and the structure maintained full dimensional stability under the applied loads. Repeated opening and closing cycles revealed no signs of hinge misalignment or seal fatigue, demonstrating long-term durability under operational conditions. These results confirm that the FEA correctly captured the worst-case loading scenarios, thus confirming the mechanical robustness of the valve when subjected to its design pressure. Based on these structural observations, the study also examined the transient behaviour of the valve to assess its real-time operating performance, including its dynamic response during opening and closing cycles.

3.2 Transient Valve Behaviour

The transient response of the valve was also evaluated experimentally, with the results summarised in Table 3. The measured opening time was approximately 0.45 ± 0.02 s, while the closing time was significantly faster, at 0.18 ±
0.01 s. The faster closing can be attributed to the predominance of gravitational forces acting on the doors, while the slower opening response is governed by inertial resistance and fluid acceleration effects. Additionally, door flapping was observed at specific flow thresholds, indicating localised unstable flow interactions that could influence long-term operational stability.

Table 4. Valve response times

Mode	Experimental Time (s)	CFD/FEA Estimate	Notes
Opening	0.45 ± 0.02	N/A	CFD steady-state only
Closing	0.18 ± 0.01	N/A	Gravity dominated

3.3 Numerical Limitation

It is important to note that the steady-state CFD models used in this study did not capture the dynamic aspects of transient valve behaviour, such as door flutter and time-dependent opening/closing responses. Although CFD and FEA analyses provided reliable predictions of steady-state flow and stress distribution (Table A.2 of Appendix A.1), they were unable to resolve the fluid-structure interactions inherent in transient operations. Future work should therefore incorporate fully coupled fluid-structure interaction (FSI) solvers to capture these dynamic effects more accurately and obtain more precise information on real-time valve performance.

3.4 Discussion

The validated results confirm that numerical models, particularly CFD for flow performance and FEA for structural integrity, can reliably predict the operational behaviour of a multi-door non-return reflux valve under steady-state conditions. The close agreement between numerical and experimental values, with pressure drop deviations of ± 5%, confirms the accuracy of the selected turbulence model and boundary conditions.

Moreover, stress distributions obtained via FEA aligned with observed structural resilience, highlighting the model's ability to capture high-stress regions such as hinge zones. Furthermore, the stress distributions obtained via FEA align with the observed structural resilience, highlighting the model's ability to capture high-stress regions such as hinge zones. These results highlight the value of simulation-assisted valve design to minimise prototyping costs while ensuring safety and performance.

Compared to published valve validation studies, the current approach achieves competitive accuracy and demonstrates scalability to larger valve diameters. Finally, the results confirm the importance of combining experimental validation with advanced simulation to strengthen confidence in design decisions of critical infrastructure systems.

4 Conclusions

The validated CFD–FEA framework developed in this study offers a powerful tool for improving future valve designs while significantly reducing the need for extensive field testing. By achieving a close match between simulated and experimental results within 3.2% for pressure drop and a 66% safety margin for stress predictions, the model enables engineers to evaluate multiple design iterations virtually before committing to physical fabrication. This capability allows for optimisation of geometry, materials, and flow paths early in the design process, reducing the reliance on costly and time-consuming prototype construction. Because static flow and mechanical durability can be predicted with high accuracy, most design refinements can be completed computationally, with physical tests reserved for final verification and regulatory approval. The approach also accelerates development timelines by replacing weeks of field trials with rapid, high-fidelity simulations capable of assessing performance under varied operating conditions. Furthermore, validated mechanical stress predictions allow manufacturers to forecast service life and maintenance needs more accurately without long-term endurance testing. The framework provides a strong foundation for integrating dynamic Fluid–Structure Interaction (FSI) modelling in future work, enabling the capture of transient phenomena such as pressure surges, water hammer effects, and door closure dynamics.

5 References

1. X.P. Hadebe, B.X.T. Kouejou, A.A. Alugongo, D.F. Sozinando, Finite element analysis and computational fluid dynamics for the flow control of a non-return multi-door reflux valve. Fluids **9**, 238 (2024). https://doi.org/10.3390/fluids9100238
2. X.P. Hadebe, B.X.T. Kouejou, A.A. Alugongo, D.F. Sozinando, Flow performance analysis of non-return multi-door reflux valve: experimental case study. Fluids **9**, 213 (2024).
https://doi.org/10.3390/fluids9090213
3. B.E. Dallstream, B.A. Fricke, B.R., Becker, Swing check valve design criteria and CFD validation. Proc. 14th Int. Conf. Nucl. Eng. (ICONE-14), Miami, USA (2006).
4. G. Filo, E. Lisowski, J. Rajda, Design and flow analysis of an adjustable check valve by means of CFD method. Energies **14**, 2237 (2021). https://doi.org/10.3390/en14082237.
5. A. Dhumal, N. Ambhore, G. Sanap, T. Patil, V. Sanap, A. Kadu, Investigating multi-directional flow dynamics in three-way ball valves: a CFD-based study. J. Eng. Appl. Sci. **72**, 61 (2025).
6. A.S. Tabrizi, M. Asadi, G. Xie, G. Lorenzini, C. Biserni, Computational fluid-dynamics-based analysis of a ball valve performance in the presence of cavitation. J. Eng. Thermophys. **23**, 27-38 (2014). https://doi.org/10.1134/S1810232814010044
7. K.-P. Zhang, D.-Z. Wu, Y.-D. Yuan, et al., Numerical simulation and experimental validation of the flow-induced excitation characteristics of a four-way reversing valve. Processes **11**, 2177 (2023). https://doi.org/10.3390/pr11072177.

Appendix

A.1 Discussion of the flow results of a multi-gate check valve

The results in Table A.1 present the parameters of different valve gate configurations, from a fully closed position at 0° to a fully open position at 65.5°.

Table A.1. Flow Characteristics and CFD Results

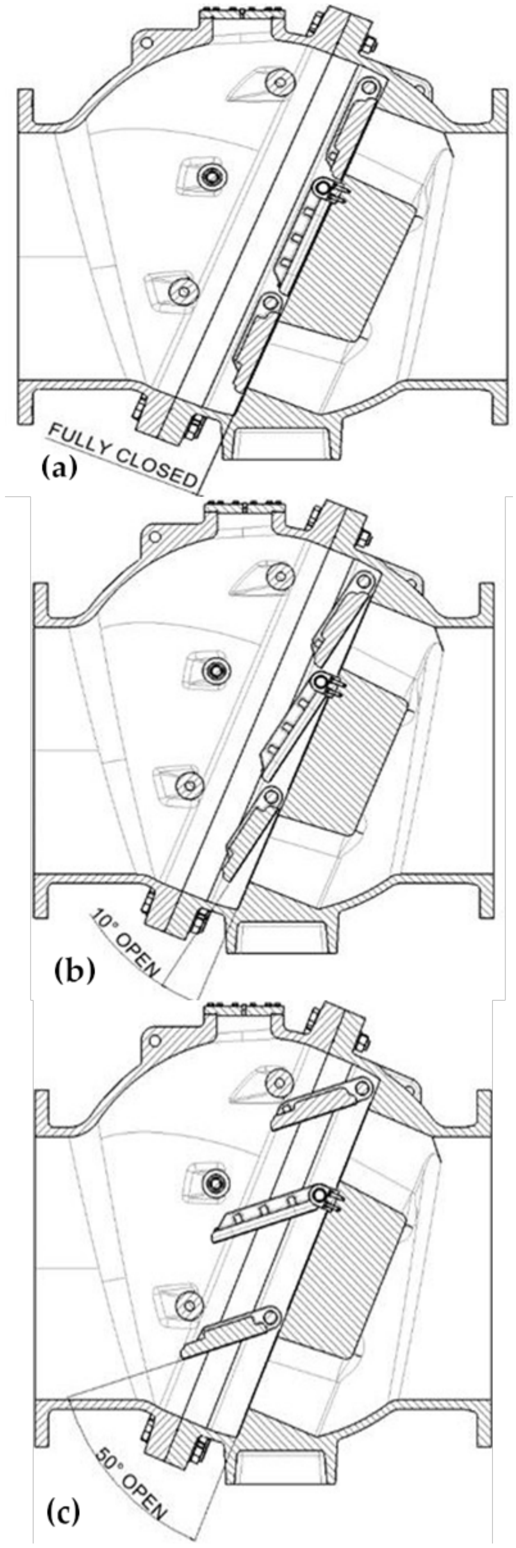
Parameter	Fully Closed(0°)	Fully Open (65.5)
Area, A(m^2)	1.54	1.54
Flow rate, Q(m^3/s)	4.62	4.62
Velocity at Inlet (m/s)	3	3
Velocity at Outlet (m/s)	2.5	3.03
Pressure at Inlet (kPa)	1750	1750
Pressure at Outlet (kPa)	1790	1754
Pressure Drop (kPa)	40	6
Mass Flow Rate (kg/s)	4610	4610
Flow Coefficient (Cv)	0.016	0.066
Head Coefficient (Kv)	0.019	0.076
Head Loss (m)	0.034	0.017
Reynolds Number (Re)	4.3 e^6	4.3 e^6
Laminar Flow, Le (m)	3.5 e^5	3.5 e^5
Turbulent Flow, Le (m)	78	62
Turbulent Energy (m^2 s^2)	0.494	0.032
Turbulence Length (m)	0.045	0.039
Turbulent Time (s)	0.399	0.627
Cavitation Index (σ)	20.75	338.51
Water Leak Rate (L/min)	0	0
Response Time (s)	5	20

Table A.2. CFD Meshing Specifications

Parameter	Details
SolidWorks Version	SolidWorks 2022
Solver	SolidWorks Flow Simulation (CFD)
Mesh Type	Computational Mesh (CFD)
Global Mesh Size	10 mm
Mesh Refinement Levels	3 Levels (Automatic)
Boundary Layer Mesh	5 layers, Growth Rate = 1.2
Element Type	Tetrahedral
Minimum Element Size	1 mm
Maximum Element Size	15 mm
Mesh Quality Criteria	Skewness < 0.85, Aspect Ratio < 5
Adaptive Meshing	Yes (based on velocity & pressure gradients)
Advanced Refinement Zones	Yes, near the valve seat and fluid contacts
Number of Elements	2.5 million elements
Number of Nodes	1.8 million nodes
Automatic Mesh Sizing	Automatic with manual refinements
Solver Settings	Steady-State, Incompressible flow
Computational Domain	Full 3D domain around the valve
Mesh Symmetry	Full valve simulated
Valve Seat Mesh Density	High density (Fine mesh)
Inlet & Outlet Mesh Refinement	2-5 mm
Pressure Convergence Criteria	100 kPa
Velocity Convergence Criteria	1 m/s
Turbulence Model	k-ε (k-epsilon)
Gravity Settings	Enabled (based on vertical flow)
Fluid Material	Water
Solid Material	Ductile Iron
Mesh Independence Study	Yes, after convergence
Post-Processing Output	Velocity contours, pressure gradients
Wall Treatment	Wall functions or full resolution
Wall functions or full resolution	Full resolution
Parallel Computing	Yes, enabled (multi-core processor)

 https://doi.org/10.1051/epjconf/202634702001

A.2. Door Positioning for Varying Valve Openings



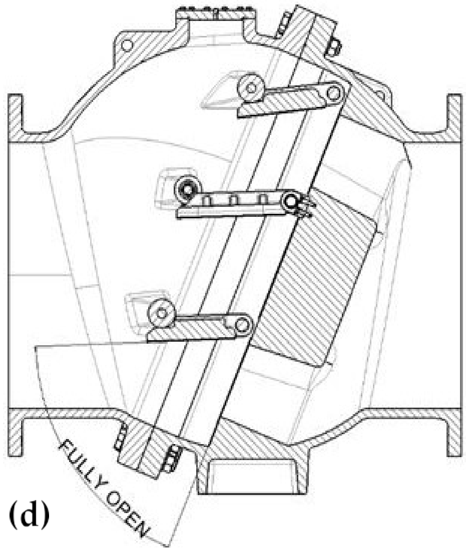


Fig. A.1. Valve door profiles at incremental opening angles (a) fully closed; (b) 10° open; (c) 50° open; (d) fully open.