



water

Special Issue Reprint

Hydrodynamics in Pressurized Pipe Systems

Edited by
Kamil Urbanowicz

mdpi.com/journal/water



Hydrodynamics in Pressurized Pipe Systems

Hydrodynamics in Pressurized Pipe Systems

Guest Editor

Kamil Urbanowicz



Basel • Beijing • Wuhan • Barcelona • Belgrade • Novi Sad • Cluj • Manchester

Guest Editor

Kamil Urbanowicz
Department of Mechanical
Engineering and Mechatronics
West Pomeranian University
of Technology in Szczecin
Szczecin
Poland

Editorial Office

MDPI AG
Grosspeteranlage 5
4052 Basel, Switzerland

This is a reprint of the Special Issue, published open access by the journal *Water* (ISSN 2073-4441), freely accessible at: https://www.mdpi.com/journal/water/special_issues/4TT8544L11.

For citation purposes, cite each article independently as indicated on the article page online and as indicated below:

Lastname, A.A.; Lastname, B.B. Article Title. <i>Journal Name</i> Year , Volume Number, Page Range.
--

ISBN 978-3-7258-8271-7 (Hbk)

ISBN 978-3-7258-8272-4 (PDF)

<https://doi.org/10.3390/books978-3-7258-8272-4>

Cover image courtesy of Kamil Urbanowicz
Pilchowice Dam, Poland

© 2026 by the authors. Articles in this reprint are Open Access and distributed under the Creative Commons Attribution (CC BY) license. The reprint as a whole is distributed by MDPI under the terms and conditions of the Creative Commons Attribution-NonCommercial-NoDerivs (CC BY-NC-ND) license (<https://creativecommons.org/licenses/by-nc-nd/4.0/>).

Contents

Preface	vii
Kamil Urbanowicz	
Hydrodynamics in Pressurized Pipe Systems	
Reprinted from: <i>Water</i> 2026 , 18, 1340, https://doi.org/10.3390/w18111340	1
Arris S. Tijsseling, Alan E. Vardy and C. J. Bruce Cartwright	
Do Pipe Bends Affect Waterhammer Waves?	
Reprinted from: <i>Water</i> 2026 , 18, 657, https://doi.org/10.3390/w18060657	7
Pedro Ferreira and Dídía Covas	
Quasi-2D Axisymmetric Modelling of Smooth–Turbulent Transient Flow: Comparison with 1D and 3D-CFD Models	
Reprinted from: <i>Water</i> 2026 , 18, 141, https://doi.org/10.3390/w18020141	24
Shuaihui Sun, Jinyang Ma, Bo Zhang, Jingwen Jia and Jiuwang Li	
Characterization of Non-Constant Flow in the Recession Process of Pressurized Pipelines with Air Valves	
Reprinted from: <i>Water</i> 2025 , 17, 3022, https://doi.org/10.3390/w17203022	46
Alan E. Vardy and Arris S. Tijsseling	
What Is a (Pressure) Wavefront?	
Reprinted from: <i>Water</i> 2025 , 17, 1907, https://doi.org/10.3390/w17131907	63
Samane Aghaei, Mehdi Hamidi, Ahmad Malekpour and Mohsen Besharat	
A Numerical Investigation into the Performance of Bypass Systems During Filling and Air Removal in Partially Drained Pipelines	
Reprinted from: <i>Water</i> 2025 , 17, 1544, https://doi.org/10.3390/w17101544	84
Mariangel Puerta-Zurita, Juan J. Villero-Guerra, Oscar E. Coronado-Hernández, Vicente S. Fuertes-Miquel and Manuel Saba	
A Single Water Pipeline Design Considering Pressure-Dependent Consumptions: A New Perspective	
Reprinted from: <i>Water</i> 2025 , 17, 307, https://doi.org/10.3390/w17030307	105
Dalia M. Bonilla-Correa, Oscar E. Coronado-Hernández, Alfonso Arrieta-Pastrana, Vicente S. Fuertes-Miquel, Modesto Pérez-Sánchez and Helena M. Ramos	
Mathematical Approach for Directly Solving Air–Water Interfaces in Water Emptying Processes	
Reprinted from: <i>Water</i> 2024 , 16, 3203, https://doi.org/10.3390/w16223203	121
Duban A. Paternina-Verona, Oscar E. Coronado-Hernández, Modesto Pérez-Sánchez and Helena M. Ramos	
Real-Time Analysis and Digital Twin Modeling for CFD-Based Air Valve Control During Filling Procedures	
Reprinted from: <i>Water</i> 2024 , 16, 3015, https://doi.org/10.3390/w16213015	137
Emil Smyk, Michał Stopel and Mikołaj Szyc	
Simulation of Flow and Pressure Loss in the Example of the Elbow	
Reprinted from: <i>Water</i> 2024 , 16, 1875, https://doi.org/10.3390/w16131875	156

Preface

Pressurized water pipeline systems are among the most critical components of modern water infrastructure. Their reliable operation depends on a thorough understanding of hydraulic behavior under both normal and transient conditions. As water distribution and conveyance networks become increasingly complex, the need for accurate modeling, efficient operation, and effective management of hydraulic phenomena continues to grow.

This Reprint was prepared to bring together recent research advances in the field of hydrodynamics in pressurized pipe systems. The collected contributions address a broad range of topics, including water hammers and pressure transients, fluid-structure interactions, turbulent flow modeling, pipeline filling and emptying processes, air management, leakage assessment, hydraulic losses, and emerging Digital Twin technologies. Together, these studies demonstrate how theoretical developments, numerical simulations, laboratory investigations, and real-world applications can be integrated to improve the safety, efficiency, and resilience of water infrastructure.

One of the motivations for assembling this Reprint was to provide a consolidated reference that reflects current trends in hydraulic research and highlights the growing role of advanced computational tools, real-time monitoring, and data driven approaches in pipeline engineering. The contributions also illustrate the importance of balancing model accuracy with practical applicability, an issue that remains central to both research and engineering practice.

This Reprint is intended for researchers, engineers, water utility professionals, consultants, and graduate students interested in hydraulic transients, pipeline design, water distribution systems, and infrastructure management. We hope that the knowledge presented here will support future research, encourage interdisciplinary collaboration, and contribute to the continued development of more reliable and sustainable water systems.

Kamil Urbanowicz

Guest Editor

Hydrodynamics in Pressurized Pipe Systems

Kamil Urbanowicz

Faculty of Mechanical Engineering and Mechatronics, West Pomeranian University of Technology in Szczecin,
70-310 Szczecin, Poland; kamil.urbanowicz@zut.edu.pl

1. Introduction

Hydrodynamics, a central branch of fluid mechanics, concerns the study of fluids in motion and the forces and energies associated with them. It provides the theoretical and practical framework for describing key physical quantities such as fluid density, flow velocity, and energy distribution within fluid systems [1–3]. In engineering practice, these principles are fundamental to the analysis and design of pressurized pipe systems, where fluid transport, pressure regulation, and operational safety are of critical importance.

At the core of hydrodynamic modeling lies the continuum hypothesis, which assumes that fluid properties vary smoothly in space and time, enabling their description through differential equations [2,4]. This approximation has proven highly effective for macroscopic engineering systems, where the characteristic length scales are significantly larger than molecular dimensions. As a result, the governing conservation laws of mass, momentum, and energy can be successfully applied to analyze complex flow behavior in water distribution networks and industrial pipeline systems.

In recent decades, the field has undergone a significant transformation driven by advances in computational methods and experimental techniques. High-resolution numerical simulations, particularly within the framework of computational fluid dynamics (CFD), have enabled increasingly accurate predictions of complex and unsteady flow phenomena [5,6]. The integration of theoretical modeling, numerical analysis, and experimental validation now forms the foundation of modern hydrodynamic research.

This Special Issue focuses on one of the most challenging and practically significant areas within hydrodynamics: transient flow phenomena in pressurized pipe systems. Transient flows are characterized by rapid temporal variations in flow conditions, typically induced by operational events such as valve closures, pump start-ups or shutdowns, and sudden changes in discharge or pressure. These unsteady processes can generate severe hydraulic effects, most notably water hammer, in which pressure waves propagate through the system and may lead to structural damage, system failure, or reduced operational reliability [7,8]. Accurate prediction and control of such phenomena require advanced modeling approaches capable of capturing wave propagation, energy dissipation, and fluid–structure interactions.

Beyond transient effects, hydrodynamics plays a crucial role in addressing broader engineering challenges associated with fluid transport systems. These include improving energy efficiency, minimizing hydraulic losses, and ensuring the long-term reliability and resilience of infrastructure. The study of turbulent flow behavior and pressure-dependent losses remains essential across a wide range of applications, from water distribution networks to industrial pipeline systems [9,10]. In addition, increasing attention is being given to air–water interactions, particularly during pipeline filling and emptying processes, where improper air management can lead to sub-atmospheric pressures or damaging pressure surges.

Recent developments in computational modeling, real-time monitoring, and Digital Twin technologies have further enhanced the ability to analyze and manage complex hydraulic systems. These tools enable more accurate representation of transient processes, support predictive maintenance strategies, and facilitate data-driven decision-making in modern water infrastructure [11,12]. At the same time, the growing complexity of these systems highlights the need for interdisciplinary approaches that integrate fluid mechanics, computational modeling, and system engineering.

Addressing the multifaceted challenges explored in this Special Issue requires continued collaboration between researchers and practitioners across disciplines. By combining theoretical developments, numerical modeling, and experimental validation, the hydrodynamics community is well positioned to advance the understanding of transient flow phenomena and to develop innovative solutions for safer, more efficient, and more resilient pressurized pipe systems.

We are therefore pleased to present this Special Issue dedicated to transient flow phenomena and contemporary challenges in hydrodynamics.

2. Review of Relevant Studies

Tijsseling et al. [C1] examine how piping systems respond to extreme pressure changes caused by water hammer waves. The study highlights that, although pipes are typically designed for steady pressures, predicting unsteady conditions and fatigue is more challenging. A key issue is whether structural movements, particularly at bends, influence the pressures used in design, especially given misleading conclusions in earlier studies. The analysis shows that immovable bends in uniform pipes have a negligible impact on passing waves, aside from minor reflections. When bends are free to move, pressure waves can induce structural motion that generates detectable reflections. To focus on the dominant behavior, the study simplifies the system by neglecting secondary effects such as damping and considers primarily early-time responses. Both one-way and two-way fluid–structure interaction models are applied to examine the relationship between fluid and structural dynamics. The results indicate that structural motion can influence pressure transients and that pressure reflections can be used to estimate bend displacement. The study concludes that while simplified models are useful in practice, fully coupled effects eliminate certain resonance behaviors predicted by less comprehensive approaches.

Ferreira and Covas [C2] compare different modeling approaches for transient turbulent flows generated by rapid valve closure. The study evaluates a quasi two-dimensional (Q2D) model against high-accuracy three-dimensional computational fluid dynamics (3D-CFD) simulations and several one-dimensional (1D) models, focusing on each approach's ability to capture transient flow features and the trade-off between accuracy and computational cost. The 3D-CFD model provides the most detailed representation, particularly near the valve, where complex turbulence and non-uniform velocity profiles occur. The Q2D model predicts unsteady energy dissipation accurately under fully developed flow conditions while requiring significantly less computational effort. However, it is less accurate near the valve during instantaneous closure due to simplifying assumptions about pressure distribution and velocity components, and it does not fully reproduce wave symmetry or initial flow behavior immediately after closure. Full convolution-based 1D models produce results similar to the Q2D model but at a higher computational cost, whereas simplified 1D formulations reduce computational effort but fail to capture important transient features. Experimental data confirm that the Q2D model performs well for realistic valve closures, although some wave characteristics are captured only by the 3D-CFD approach. Overall, the study concludes that the Q2D model is an efficient compromise between accuracy and

computational efficiency, with potential for improvement in boundary condition modeling and computational optimization.

Sun et al. [C3] investigate the emptying process of pressurized pipelines equipped with air valves under unsteady flow conditions. The authors develop a mathematical model based on the rigid water column theory to describe drainage behavior. The study analyzes how valve opening strategies, opening degree, and air valve diameter influence transient flow characteristics. Results show that a nonlinear valve opening pattern improves performance by increasing the minimum air pocket pressure and delaying its occurrence. Increasing the valve opening degree raises flow velocity and reduces emptying time, though it can decrease system stability. Larger air valve diameters reduce pressure fluctuations and accelerate pressure recovery. Comparisons with experimental data confirm that the model provides accurate predictions with low error. The study further demonstrates that using multiple air valves in an optimized arrangement shortens the duration of negative pressure conditions. A case study from the Hanjiang-to-Weihe River Diversion Project supports the effectiveness of these strategies. While the model simplifies the system by neglecting fluid elasticity and complex two-phase flow effects, the study concludes that improved valve design and modeling approaches can enhance both the efficiency and stability of pipeline emptying processes.

Vardy and Tijsseling [C4] address the ambiguity in the use of the term “wavefront” in one-dimensional fluid flow systems. While the concept has a clear meaning in two- and three-dimensional wave propagation, its interpretation in one dimension is inconsistent and lacks a formal definition, leading to potential miscommunication in engineering contexts. The study explores a wide range of one-dimensional unsteady flow phenomena to illustrate how waves can behave differently even under simple initial conditions. It shows that key parameters such as pressure and velocity may evolve in fundamentally different ways, making a universal definition impractical. Instead of proposing a strict quantitative definition, the authors develop a qualitative description that captures the most common uses of the term, applicable to both ideal and non-ideal flow conditions, including dissipative systems. This definition does not require disturbances to maintain their shape or for all wave components to travel in the same direction. The study also highlights similar past confusion regarding related concepts like wavespeed and emphasizes the importance of clear terminology. While the proposed definition improves clarity, challenges remain in analyzing complex flows and developing reliable interpretation methods, including those based on modern computational techniques.

Aghaei et al. [C5] present a one-dimensional elastic numerical model for simulating the filling process of partially drained pipelines with complex profiles. The model applies the Method of Characteristics to solve water hammer equations and incorporates the Discrete Gas Cavity Model to account for column separation. Validation against experimental data and other numerical models demonstrates good accuracy in predicting transient pressure behavior. The study identifies that oversized bypass systems can generate severe water hammer pressures during pipeline filling, while effective air management enables higher filling flow rates and reduces overall filling time. The analysis highlights the influence of air valve nozzle size: larger nozzles allow rapid air release but can lead to strong pressure surges, whereas smaller nozzles moderate surges through compression effects but may introduce early-stage pressure fluctuations. The introduction of bypass lines reduces flow velocity and dampens pressure oscillations, though excessively small bypass diameters significantly increase filling time. The study concludes that the proposed model provides a reliable basis for optimizing filling strategies and suggests future development of hybrid one- and three-dimensional modeling approaches for improved accuracy.

Puerta-Zurita et al. [C6] conclude that traditional water pipeline design methods, which treat water losses as independent of pressure, do not accurately represent real conditions. Water leakages actually depend on pressure and should be modeled accordingly during design. The proposed methodology provides a realistic way to size pipelines by accounting for pressure-dependent losses and ensuring minimum service pressure. Results indicate that at least two internal pipe diameters are typically required to meet service pressure, and the emitter coefficient strongly influences sizing. In contrast, pipe roughness and valve resistance have little effect because leakages occur downstream. Current design practices, which estimate losses as a fixed percentage of consumption, can differ by up to 28 percent compared to this new method, highlighting limitations in existing standards. Accurate pipeline sizing is also critical for developing Digital Twin models. Overall, the study presents a more reliable design approach that reflects real system behavior and recommends further work to extend this method to larger and more complex water distribution networks.

Bonilla-Correa et al. [C7] develop a new mathematical approach for modeling emptying processes in water pipelines, capturing air–water interactions during transient events. Unlike traditional one-dimensional models or three-dimensional CFD simulations, this method directly solves air–water interactions without extensive computational resources or complex differential algebraic equation systems. Demonstrated on a 350-meter-long pipe and validated experimentally, the method reliably predicts transient behavior. It computes minimum sub-atmospheric air pocket pressures with a root mean square error of 1.13 percent, slightly higher than traditional one-dimensional models due to consideration of instantaneous valve operations. Sensitivity analysis shows that air pocket size, friction factor, pipe diameter, resistance coefficient, polytropic coefficient, and pipe slope influence the emptying process. The model enables water utilities to anticipate pressure pulses that could lead to pipeline collapse and supports informed decisions on pipe stiffness selection based on soil conditions. This simpler and faster computational tool also provides a foundation for integration into Digital Twin models, improving safety and efficiency in hydraulic pipeline management.

Paternina-Verona et al. [C8] investigate air management during the filling of irregular-profile water pipelines, combining real-time data, three-dimensional CFD simulations, and Digital Twin technology. The study examines the complex behavior of trapped air pockets, air outflow through valves, and unsteady water flow evolution. CFD models calibrated with experimental data provide a reliable tool for predicting air-release dynamics and optimizing air valve performance. Key findings demonstrate that integrating CFD simulations with Digital Twin platforms allows detailed spatial and temporal tracking of pressure, velocity, and temperature, improving energy efficiency, reliability, and safety. The approach surpasses two-dimensional simulations by capturing three-dimensional air–water interactions more accurately. Machine learning techniques are explored to enhance multiphase flow modeling, air bubble reconstruction, and predictive modeling for more efficient simulations. Limitations include data quality, model robustness, interpretability, and complexity. Future research should focus on extending trapped air management to complex network branches, examining air valve mechanics, integrating GIS and AI for anomaly detection, and improving real-time sensor acquisition. Overall, the study provides a robust framework for optimizing filling procedures and managing transient events, supporting safer and more efficient pressurized water distribution systems.

Smyk et al. [C9] investigate flow behavior in pipe elbows using numerical simulations with ANSYS Fluent (2019), applying the turbulent k- ω model and SIMPLE method across Reynolds numbers from 500 to 1,000,000. The study focuses on pressure drops, minor losses, secondary flows, and vortex structures, which are critical in pipeline design. Results

show that the k- ω model accurately predicts pressure losses at high Reynolds numbers, while the Darcy friction factor aligns with theoretical values only for Re greater than or equal to 2300. Minor loss coefficients depend strongly on Reynolds number, matching literature values for Re above 20,000 but requiring further study at lower values. Secondary flows are observed in all cases, with velocity profiles revealing flow dead zones behind the elbow, more pronounced at lower Reynolds numbers. The separation point stabilizes at higher Reynolds numbers, mirroring the behavior of minor loss coefficients. Q-criterion vortex analysis identifies two vortex structures resulting from interactions between main and secondary flows, with the impact of secondary flow decreasing as the Reynolds number increases. Overall, the study provides a detailed numerical assessment of elbow flow phenomena, emphasizing Reynolds number effects on pressure losses, secondary flow patterns, and vortex formation.

In summary, the reviewed studies illustrate the multifaceted nature of water pipeline analysis and design. The transient pressure and water hammer studies [C1,C2,C4] demonstrate the importance of accurately capturing wave propagation, valve closure effects, and fluid–structure interactions, highlighting the trade-offs between model complexity and computational efficiency. The pipeline filling and emptying research [C3,C5,C7,C8] emphasizes the critical role of air–water interactions, valve operation strategies, and air management in mitigating sub-atmospheric pressures and pressure surges, while also showing the potential of Digital Twin and real-time monitoring tools for predictive control. The design-oriented study [C6] underscores that conventional pipeline sizing methods, which treat water losses as pressure-independent, can significantly misrepresent actual system behavior, advocating for pressure-dependent approaches that better inform both design and digital modeling. Finally, the steady-flow analysis in elbows [C9] provides foundational insights into hydraulic losses, secondary flows, and vortex formation, essential for normal operation and accurate loss estimation. Collectively, these studies highlight that integrating advanced modeling, experimental validation, and real-time monitoring is crucial for improving safety, efficiency, and reliability across both design and operational stages of water distribution systems.

3. Conclusions

The reviewed studies collectively demonstrate the complexity of transient phenomena in water distribution systems, including pressure surges, pipeline filling and emptying, air–water interactions, and flow behavior in bends and elbows. Accurate prediction and management of these phenomena require careful consideration of pressure-dependent losses, air pocket dynamics, valve operation, and pipeline geometry. Modeling approaches ranging from one-dimensional to three-dimensional simulations, often integrated with Digital Twin platforms and machine learning, provide valuable tools for balancing computational efficiency with predictive accuracy. The findings highlight that real-time monitoring, experimental validation, and advanced computational methods are critical for improving the safety, efficiency, and reliability of water networks. Despite significant progress, challenges remain in fully capturing multiphase interactions, complex network behavior, and extreme transient events. Future research should focus on hybrid modeling strategies, enhanced data acquisition, and integration of AI and GIS technologies to support proactive management and resilient design of water infrastructure.

Funding: This research received no external funding.

Conflicts of Interest: The author declares no conflicts of interest.

List of Contributions

1. Tijsseling, A.S.; Vardy, A.E.; Cartwright, C.J.B. Do Pipe Bends Affect Waterhammer Waves? *Water* **2026**, *18*, 657. <https://doi.org/10.3390/w18060657>.
2. Ferreira, P.; Covas, D. Quasi-2D Axisymmetric Modelling of Smooth–Turbulent Transient Flow: Comparison with 1D and 3D-CFD Models. *Water* **2026**, *18*, 141. <https://doi.org/10.3390/w18020141>.
3. Sun, S.; Ma, J.; Zhang, B.; Jia, J.; Li, J. Characterization of Non-Constant Flow in the Recession Process of Pressurized Pipelines with Air Valves. *Water* **2025**, *17*, 3022. <https://doi.org/10.3390/w17203022>.
4. Vardy, A.E.; Tijsseling, A.S. What Is a (Pressure) Wavefront? *Water* **2025**, *17*, 1907. <https://doi.org/10.3390/w17131907>.
5. Aghaei, S.; Hamidi, M.; Malekpour, A.; Besharat, M. A Numerical Investigation into the Performance of Bypass Systems During Filling and Air Removal in Partially Drained Pipelines. *Water* **2025**, *17*, 1544. <https://doi.org/10.3390/w17101544>.
6. Puerta-Zurita, M.; Villero-Guerra, J.J.; Coronado-Hernández, O.E.; Fuertes-Miquel, V.S.; Saba, M. A Single Water Pipeline Design Considering Pressure-Dependent Consumptions: A New Perspective. *Water* **2025**, *17*, 307. <https://doi.org/10.3390/w17030307>.
7. Bonilla-Correa, D.M.; Coronado-Hernández, O.E.; Arrieta-Pastrana, A.; Fuertes-Miquel, V.S.; Pérez-Sánchez, M.; Ramos, H.M. Mathematical Approach for Directly Solving Air–Water Interfaces in Water Emptying Processes. *Water* **2024**, *16*, 3203. <https://doi.org/10.3390/w16223203>.
8. Paternina-Verona, D.A.; Coronado-Hernández, O.E.; Pérez-Sánchez, M.; Ramos, H.M. Real-Time Analysis and Digital Twin Modeling for CFD-Based Air Valve Control During Filling Procedures. *Water* **2024**, *16*, 3015. <https://doi.org/10.3390/w16213015>.
9. Smyk, E.; Stopel, M.; Szyca, M. Simulation of Flow and Pressure Loss in the Example of the Elbow. *Water* **2024**, *16*, 1875. <https://doi.org/10.3390/w16131875>.

References

1. White, F.M. *Fluid Mechanics*, 7th ed.; McGraw-Hill: New York, NY, USA, 2011.
2. Batchelor, G.K. *An Introduction to Fluid Dynamics*; Cambridge University Press: Cambridge, UK, 1967.
3. Chanson, H. *Applied Hydrodynamics: An Introduction*; CRC Press: London, UK, 2014.
4. Kundu, P.K.; Cohen, I.M.; Dowling, D.R. *Fluid Mechanics*, 6th ed.; Academic Press Elsevier Science Publishing Co., Inc.: Cambridge, MA, USA, 2016.
5. Anderson, J.D. *Computational Fluid Dynamics: The Basics with Applications*; McGraw-Hill: New York, NY, USA, 1995.
6. Ferziger, J.H.; Perić, M.; Street, R.L. *Computational Methods for Fluid Dynamics*, 4th ed.; Springer Nature: Cham, Switzerland, 2020.
7. Streeter, V.L.; Wylie, E.B. *Fluid Transients in Systems*; Prentice Hall: Englewood Cliffs, NJ, USA, 1993.
8. Chaudhry, M.H. *Applied Hydraulic Transients*, 3rd ed.; Springer: New York, NY, USA, 2014.
9. Pope, S.B. *Turbulent Flows*; Cambridge University Press: Cambridge, UK, 2000.
10. Dey, S. *Fluvial Hydrodynamics*; Springer: Berlin, Germany, 2014.
11. Bam, P.G.; Rezaei, N.; Roubanis, A.; Austin, D.; Austin, E.; Tarroja, B.; Takacs, I.; Villez, K.; Rosso, D. Digital Twin Applications in the Water Sector: A Review. *Water* **2025**, *17*, 2957. [CrossRef]
12. Fuller, A.; Fan, Z.; Day, C.; Barlow, C. Digital Twin: Enabling Technologies, Challenges and Open Research. *IEEE Access* **2020**, *8*, 108952–108971. [CrossRef]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

Do Pipe Bends Affect Waterhammer Waves?

Arris S. Tijsseling ^{1,*}, Alan E. Vardy ² and C. J. Bruce Cartwright ³

¹ Mathematics and Computer Science, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands

² Civil Engineering, University of Dundee and Dundee Tunnel Research, Dundee DD1 4HN, UK; a.e.vardy@dundee.ac.uk

³ Formerly: Civil Engineering, University of Leeds, Leeds LS2 9JT, UK

* Correspondence: a.s.tijsseling@tue.nl

Abstract

Piping systems must cope with the internal pressure of the fluid they carry. They are almost always well-designed for withstanding steady-flow pressures, but allowing for unsteady-flow pressures and for fatigue can be more challenging. Positive and negative gauge pressures induced by waterhammer waves are possibly the most extreme that piping is likely to face during its lifetime. It is widely accepted that this should be addressed by analyses during the design phase, but this is usually done under the assumption that consequential (non-hoop) structural movements do not affect the calculated pressures. However, the calculated pressures are used as input to the structural design. Commonly, attention focusses on static predictions of induced hoop stresses and on the risk of buckling, but attention sometimes has to be paid to dynamic responses. In these cases, the complexity of the structural analysis depends on the assumed degrees of freedom of possible movement, so it is desirable to avoid including unnecessary detail. The title of this paper poses one question that is frequently asked. However, the correct answer is not always obtained, partly because highly misleading answers were published in one early paper, the rebuttals to which were much less widely reported. The current contribution attempts to answer the question for both fixed and movable bends. Attention is paid to pressure transients arriving at bends from remote locations and potentially inducing pipe movement. Then, the opposite effect is considered, namely the generation of pressure transients by structural movements. To avoid distorting the picture by combining this with nominally unrelated causes, strong simplifications are made—e.g., disregarding all forms of energy dissipation.

Keywords: waterhammer; fluid transient; pressure surge; pipe elbow; fluid–structure interaction

1. Introduction

Do waterhammer waves pass pipe bends without noticing them? That is, are they (the waves) unaffected by them (the bends)? This question has been asked (to the authors and in general) and discussed many times, but being justifiably confident about answering it correctly can be difficult. The issue is reconsidered, including references to two not-to-be-forgotten papers from the 1960s, [1] Contractor (1965) and [2] Swaffield (1968–1969), that were instructive in many ways, noting that the latter has misled researchers ever since. Later insights with respect to fluid–structure interaction at elbows as reviewed by [3] Tijsseling (2019) should be considered. The primary purpose of this paper is to discuss and, at least partially, answer the question posed in the title.

The discussion begins in Section 2 with consideration of pressure waves in a pipe with a bend that is restrained to prevent any structural movement. This is followed in Section 3 by an assessment of pressure waves in pipes with bends that are able to move in at least one axial direction. Section 4 focusses on structural motion induced by prescribed pressure changes and it allows for the influence of those movements on subsequent pressures. This interactive behaviour is an example of two-way fluid–structure interaction (FSI). Section 5 considers FSI caused by externally imposed structural movement. In this instance, the presented analysis considers resulting pressure waves in the fluid. However, no allowance is made for the corresponding influence of the pressure waves on the structural motion. This is an example of one-way FSI (structure to fluid). In engineering practice, use is also made of one-way FSI (fluid to structure) in which the phenomenon is simulated in two steps, namely (i) a pressure surge analysis assuming stationary pipes (except for pressure-induced hoop-deformation) followed by (ii) a structural analysis of the response to the simulated pressures. All cases considered are examples of so-called ‘junction-coupling’ at the elbow, a general FSI term that also embraces coupling at joints, branches, valves, etc.

Simplifications

The whole paper is targeted at clarifying the influence of pipe bends on waterhammer waves. This is achieved by disentangling the targeted phenomena from as many potential complications as possible. For example, another form of FSI known as Poisson-coupling also exists but is disregarded herein. In pipework typically used in engineering applications, it is usually a weaker effect than junction coupling [4] Wiggert (1986). All possible causes of fluid or structural damping are also excluded, even though many such causes need to be considered when undertaking simulations for practical purposes or when attempting to deduce causes of unexpected behaviour, whether physical, numerical or analytical [5] Leslie & Tijsseling (1999); [6] Vardy (2023). Other simplifications include (i) assuming that the waves in the fluid and in the pipe wall travel at constant speeds, regardless of variations in pressures, stresses and velocities, and (ii) limiting attention to 90° bends.

In addition to simplifying, ensuring a clear focus on the physical phenomena under investigation, these simplifications enable the interactions to be expressed clearly in the equations presented below. They also make possible the development of symbolic solutions that express the behaviour in analytical forms that can be interpreted without the need for numerical methods. Useful generic information can also be inferred by means of dimensional analysis [1] Contractor (1965); [2] Swaffield (1968–1969); [7] Wylie (1996); however, that approach is not followed herein.

2. Fixed Bends: Do They Exist in Practice?

It is useful to begin with a discussion of pressure waves in pipes with fixed bends. In *steady* flows, account is usually taken of pressure losses at bends and junctions and other interruptions to simple flow in a straight pipe. These result from secondary flows, flow separation, vortex shedding and enhanced turbulence. Examples for simple bends are described by [8] Itō (1960) and [9] Smyk et al. (2024). Such losses are neglected herein, partly for simplicity, but mostly because they are usually far smaller than pressure changes caused by waterhammer-like waves. For simple bends, the losses are typically less than $\frac{1}{2} \rho V^2$, where ρ and V are the fluid density and axial velocity. When the passage of a pressure wave changes the velocity, the change in this pressure loss is less than $\frac{1}{2} \rho \Delta(V^2)$. In contrast, the pressure change caused by the wave is in the order of $\rho a \Delta V$ where a is the wave speed. The ratio of these two changes is in the order of the flow Mach number, V/a , which is almost always very small in liquid flows and is commonly also small in gas flows. For completeness, however, it is noted that these so-called ‘minor’ losses can be large in other

contexts, notably partially closed valves or orifices such as those used by [1] Contractor (1965) to generate losses of up to $1150 \times \frac{1}{2} \rho V^2$. These cases are not considered herein, and neither are other causes such as entrapped gas in an elevated bend [10] Bergant et al. (2008). Similarly, no account is taken of specialized phenomena such as extreme cases of plastic deformation [11] Florence (1975); [12] Youngdahl et al. (1981). Also, Cartwright (1977) [13] showed that locally increased Boussinesq momentum correction coefficients [14] Vardy & Tijsseling (2022) have negligible consequences.

Impossibility of Reflections from Fixed Bends

We begin by demonstrating that, with the above simplifications, reflections from structurally fixed bends in uniform pipes are impossible. The following assessment applies equally well to a case where few, if any, persons would imagine a reflection to be possible, namely a simple join between two pipes of equal bore (disregarding the fact that flanges and welds give tiny reflections because of the increased local stiffness and mass of the pipe's wall). A simple proof is possible by assuming that a sustained reflection *is* possible and showing that this leads to a contradiction.

Figure 1 shows a step wavefront approaching such a bend/join. The initial steady flow pressure and velocity in limbs 1 and 2 are P_0 and V_0 and the approaching pressure wave increases these to P_A and V_A . The figure also shows (i) an *assumed* reflection in the upstream limb 1 and (ii) the ongoing wavefront in the downstream limb 2. In this illustration, both the incident wavefront and the wavefront in limb 2 are deemed to cause increases in pressure, so $P_B > P_A$. Continuity of pressure requires that the amplitude of the wavefront transmitted into limb 2 is also equal to P_B .

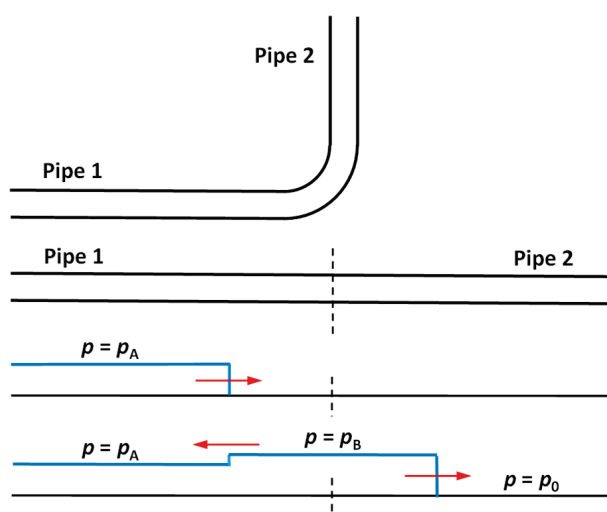


Figure 1. Indicative pressure distributions implied by a sustained reflection of a wavefront at an internal boundary in a uniform pipe.

Now consider the implied velocities:

- In limb 1, the positive reflection of pressure reduces the velocity so $V_{B1} < V_A$
- In limb 2, the pressure exceeds P_A and so $V_{B2} > V_A$

That is, for the assumed conditions, V_{B2} is *not* equal to V_{B1} . Clearly, this is inconsistent with the requirements of conservation of mass, and it follows that the original assumption violates elementary laws of physics. Therefore, the hypothesised reflection in limb 1 is impossible. Strictly, this has been demonstrated only for the case of a compressive wavefront and a compressive reflection. However, the same conclusion is reached if either (or both) of these is deemed to be negative. Accordingly, no further attention is paid in this

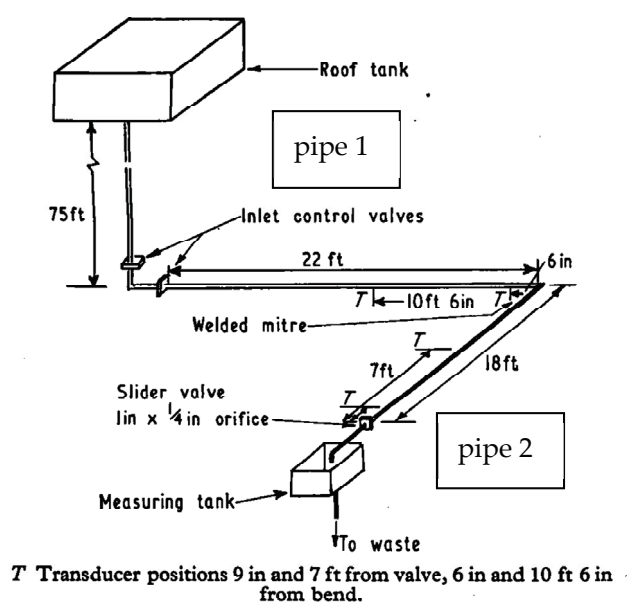
paper to fixed, loss-free bends in uniform pipes. Instead, the focus is on the influence of bends that do not remain stationary.

One critical note: wave reflection due to pure scattering (because of the 2D or 3D geometry of the bend) occurs but is negligible for long (relative to pipe diameter) wavelengths [15] Martin (2023).

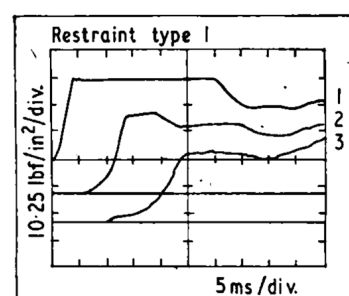
3. Bends That Are Free to Move: Overview

3.1. Swaffield

Swaffield (1968–1969) [2] investigated experimentally the influence of bends with a range of axial curvature (maximum radius of curvature of $5D$) and angles (30° , 60° , 90° and 135°) on the reflections of pressure waves. He buttressed the bends in a manner that he considered could be regarded as perfectly fixed, and obtained measurements that clearly showed reflections (e.g., Figure 2b). Furthermore, they showed a dependence on the bend curvature and the bend angle. He interpreted the measurements as waterhammer reflections from bends and reported his conclusions accordingly. We now realise, of course, that the bends cannot have been adequately fixed and, indeed, Swaffield himself reached this conclusion after discussion with others. Evidence for this is cited by the following authors:



(a) Schematic layout of test rig



$V_0 = 3.1$ ft/s. Pipe period = 32 ms.
 $T_w = 63^\circ\text{F}$. Trace 1, 9 in from valve, Trace 2, 8 ft 9 in from valve and Trace 3, 4 ft upstream of bend.

(b) Pressure variation at 3 points along a 2 in bore polythene pipe including a 90° mitre bend

Figure 2. Negative reflection of pressure wave from bend as observed by Swaffield [2], (1968–1969).

Cartwright, 1977 [13]:

The work described in the bulk of this (Cartwright's) thesis assumed that no bend motion took place and that the bend was rigidly held. This assumption was based on the work of Swaffield (1968–1969) who, as already stated, attempted to eliminate bend motion. Swaffield (141) pointed out that in view of the arguments repeated in the first section of this chapter, it is unlikely that this was so." N.B. Cartwright's Ref. 141 cited as: Swaffield. Private communication.

Wilkinson, 1980 [16]:

On the effect of restraints, Swaffield found little or none but decided later, Ref. 5, that the restraints he had used had not been sufficient to reduce the very small displacement

that occurred. His results are effectively unrestrained.” N.B. Wilkinson’s Ref. 5 cited as Swaffield, J. A. Private communication. (1976).

Unfortunately, these rebuttals were not widely seen and, for many years, Swaffield’s published paper was accepted at face value by a significant number of researchers. Indeed, as recently as two years ago, answers generated by AI searches were seriously misleading (although they have improved noticeably since then). For the avoidance of doubt, therefore, we declare unequivocally that the negative reflection seen in Figure 2b herein were a consequence of bend motion. An assessment of the movement needed to cause the measured reflections is given in Section 4.6.

3.2. Wylie & Streeter

The pumping mechanism of vibrating bends and panting (breathing) of pipes is well known. For example, the classic textbook by Wylie & Streeter (1993) [17] includes the following statements:

“There are two basic types of exciters that may perturb a system. They may act independently, in unison, or in combination with other elements. The two are:

1. *An element that provides an oscillatory flow across it but maintains a common head. This may be classified as a capacitance element, as it has the capability of storage. It may be a moving pipe bend as shown in Figure 3, left, a “panting” penstock as shown in Figure 3, right, or any other device with an elastic behaviour.*

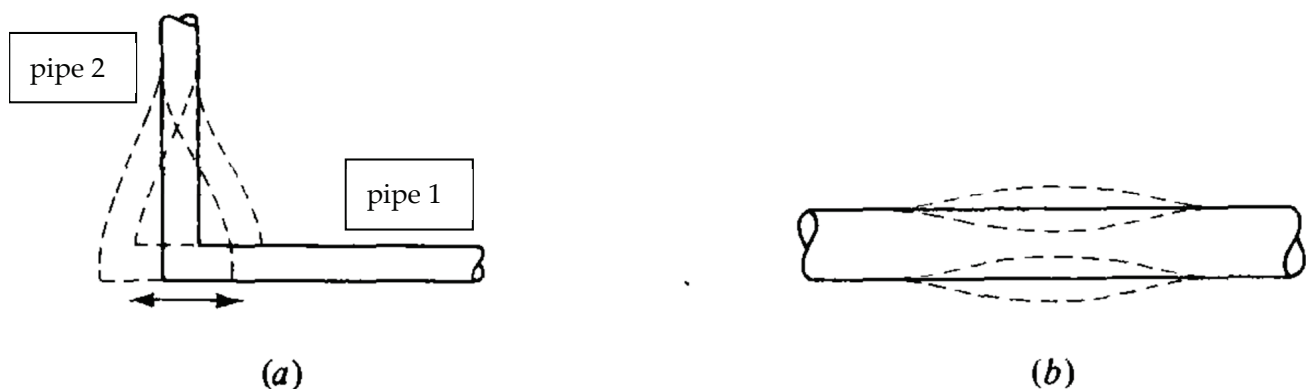


Figure 3. Capacitance elements (in [17], Section 13-8): (a) vibrating elbow; (b) breathing pipe.

2. *An element that provides an oscillatory head across it but maintains a common discharge. This is a resistance element or an inertia element. A turbomachine or variable-loss element may fall into this category. A variable loss may arise due to eddy shedding or hydroelastic interaction between fluid and structural components.”*

As illustrated in Figure 3a, the axial vibration of pipe 1 causes bending in pipe 2. For a rigid elbow, to keep its angle right, this implies some bending in pipe 1 too.

3.3. Cartwright

Whereas the first reflection of a positive pressure wave from a minor loss (if detectable) from a fixed bend would be positive [1], it is negative from a bend that is free to move because this creates additional storage (e.g., Figure 3a). Cartwright’s (1977) [13] explanation of this mechanism is illustrated in Figure 4. The reflection (when observed in measured pressure signals) tells something about the “stiffness” of external supports at the bend. Related work on elbows worth reading is by Belytschko et al. (1982, 1986) [18,19] and Tijsseling et al. (1996) [20]. Further reference to Cartwright’s work is made in Sections 4.3 and 5.2.

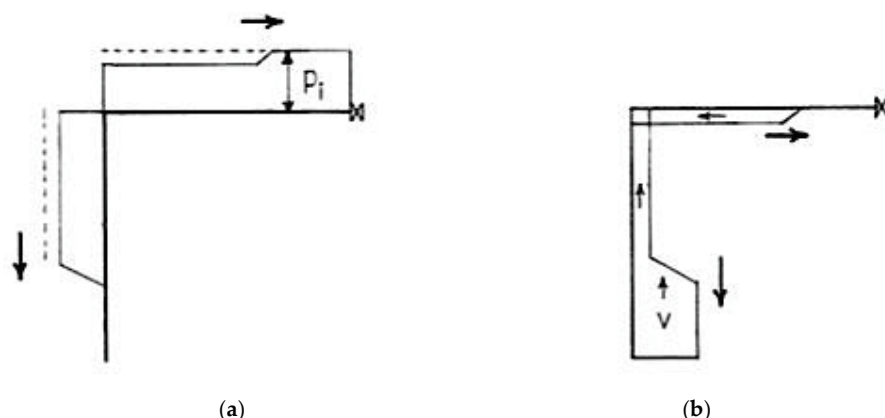


Figure 4. Wave reflection at a moving bend according to Cartwright 1977 ([13], Figure 10.2). (a) Pressures after transmission and reflection of a positive incident pressure wave p_i caused by valve closure on the right end (as bottom graphs in Figure 1). (b) Fluid velocities after transmission and reflection; the initial steady fluid velocity before valve closure is V .

4. Analysis: Fluid-Induced Disturbances in the Structure

4.1. Fully Fixed Bends

Extensive literature exists regarding pressure surges in pipework that is deemed to be immovable. Therefore, no useful purpose would be served by presenting further analysis of it herein. Instead, we simply cite the well-known Joukowski relationship between changes (indicated by Δ) in pressure and velocity induced by isolated wavefronts in pipes, namely, the following:

$$\Delta H = \pm \frac{a}{g} \Delta V \quad (1)$$

where $H = P/(\rho g)$ is the pressure head, ρ is the fluid density, a is the speed of wave propagation, g is gravitational acceleration, and V is the fluid velocity. This is used below as a reference condition.

4.2. Bends That Are Moving

Figure 5 depicts a wavefront propagating along pipe 1 and reaching a bend (J) at which it partially reflects and partially transmits into pipe 2. In pipe 1, three regions of z - t space are identified, namely $1(0)$ = the initial (steady-flow) conditions, $1A$ = after the wavefront, but before its reflection from the bend arrives, and $1J$ = after the reflection. In pipe 2, only two regions exist, namely, $2B = 2(0) = 1(0)$ before the transmitted wavefront, and $2J$ after the transmitted wavefront. The figure also shows typical wave paths in both pipes before and after the incident wavefront reaches the bend.

For completeness, it is useful to note that, as described in Section 2 in the context of *fixed* bends, in pipe 1, $H_{1J} < H_{1A}$ implies that $V_{1J} > V_{1A}$, whereas in pipe 2, it implies that $V_{2J} < V_{1A}$. For the fixed-bend case, this violated mass conservation. For the moving-bend case, however, it acts as a measure of the storage effect of the movement, see Figure 3a.

Using the Method of Characteristics (MOC), the following equations describe the changes across the reflected and transmitted wavefronts.

$$\begin{cases} (H_{1J} - H_{1A}) + \frac{a}{g} (V_{1J} - V_{1A}) = 0 \\ (H_{2J} - H_{2B}) - \frac{a}{g} (V_{2J} - V_{2B}) = 0 \end{cases} \quad (2)$$

In the idealized fixed-bend example, the flow states in the regions $1J$ and $2J$ are equal. When the bend is free to move, however, these states are not equal. Herein, they are deemed

to satisfy the following boundary conditions, in which $\dot{u}$ denotes the axial velocity of a pipe wall.

$$\begin{cases} H_{1J} = H_{2J} := H_J \\ V_{1J} - \dot{u}_{1J} = V_{2J} - \dot{u}_{2J} \end{cases} \quad (3)$$

Equations (2) and (3) can be combined to give the following expressions for the velocities and pressure head at the bend after the reflection/transmission process:

$$\begin{cases} V_{1J} = \frac{g}{2a}(H_{1A} - H_{2B}) + \frac{1}{2}[(V_{1A} + V_{2B}) + (\dot{u}_{1J} - \dot{u}_{2J})] \\ V_{2J} = \frac{g}{2a}(H_{1A} - H_{2B}) + \frac{1}{2}[(V_{1A} + V_{2B}) - (\dot{u}_{1J} - \dot{u}_{2J})] \\ H_J = \frac{1}{2}(H_{1A} + H_{2B}) + \frac{a}{2g}[(V_{1A} - V_{2B}) - (\dot{u}_{1J} - \dot{u}_{2J})] \end{cases} \quad (4)$$

Assuming zero initial conditions in both pipes (i.e., $H = 0$ and $V = 0$) and noting that Equation (3, bottom) gives $V_{2J} = V_{1J} - (\dot{u}_{1J} - \dot{u}_{2J})$, Equation (4) reduce to ($t > L_1/a$):

$$\begin{cases} V_{1J} = \frac{g}{2a}H_{1A} + \frac{1}{2}[V_{1A} + (\dot{u}_{1J} - \dot{u}_{2J})] \\ V_{2J} = \frac{g}{2a}H_{1A} + \frac{1}{2}[V_{1A} - (\dot{u}_{1J} - \dot{u}_{2J})] \\ H_J = \frac{1}{2}H_{1A} + \frac{a}{2g}[V_{1A} + (\dot{u}_{1J} - \dot{u}_{2J})] \end{cases} \quad (5)$$

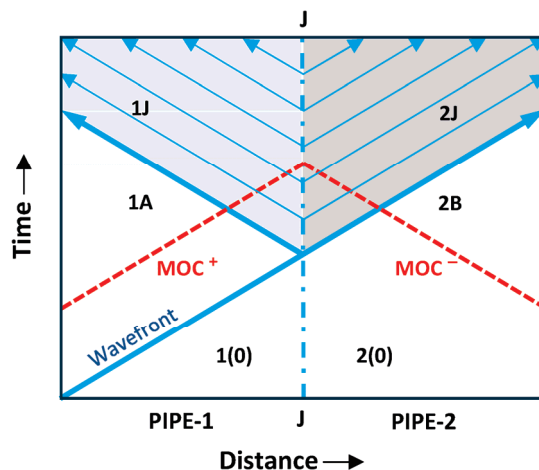


Figure 5. Wave paths depicting reflection and transmission at an internal boundary (J). Wave generated at $t = 0$ at the far end of pipe 1.

4.3. Bend Allowed to Move in Only One Axial Direction: $\dot{u}_{1J} \neq 0; \dot{u}_{2J} = 0$

Equation (5) relate the four unknowns V_{1J} , V_{2J} , H_J and $\dot{u}_{1J}$. To proceed further, it is necessary to infer information about the structural velocities, and this necessitates consideration of external structural restraints as well as internal forces. At one extreme, namely the fixed bend considered above, the external (anchor) forces are deemed to be so great that the bend cannot move. We now consider the opposite extreme, namely a bend with no nearby external support [3,20]. This is a good approximation to reality in many applications with suspended pipework. In this section, the bend is free to move in the axial direction of pipe 1 but is prevented from moving in the axial direction of pipe 2. That is, $\dot{u}_1 \neq 0$; $\dot{u}_2 = 0$, as illustrated in Figure 3a. In Section 4.5, it is free to move in both directions.

The control volume shown in Figure 6 includes a 90° bend that is deemed to be rigid, so that no account needs to be taken of its flexibility. The control volume is stationary relative to the coordinate axes, so the momentum equation in the axial z_1 direction after the wavefront has passed through the bend is

$$P_J A - \sigma_{1J} A_w - F_1 = \frac{\partial}{\partial t} (m V_{1J} + m_w \dot{u}_{1J}) + \rho A V_{1J} (\dot{u}_{1J} - V_{1J}) \quad (6)$$

in which A_w is the cross-sectional area of the pipe wall, σ_{1J} is the axial stress, m is the mass of fluid in the control volume and m_w the mass of the elbow. No account is taken of shear or inertial forces associated with bending in pipe 2. With (temporarily) constant P_J and σ_{1J} , the conditions are steady so that the first term on the right-hand-side of Equation (6) is zero. The second term on the right is a convective term which normally is disregarded in waterhammer analysis. The reaction force F_{1J} is 0 (unrestrained bend), so the axial stress in the wall satisfies

$$\sigma_{1J} = \frac{A}{A_w} [\rho g H_J] \quad (7)$$

and the associated axial strain ε_{1J} is

$$\varepsilon_{1J} = \frac{A}{EA_w} [\rho g H_J] \quad (8)$$

where E is the Young's modulus of the wall material. This strain exists along the whole length of the region between the bend and the stress-wavefront induced in the wall by the reflection and transmission of the incident pressure wavefront. The length of that region is increasing at approximately the speed of sound in the wall, namely, $a_w = \sqrt{(E/\rho_w)}$, where ρ_w is the density of the wall material, so the induced bend velocity (derived via "Joukowski": $\sigma_{1J} = \rho_w a_w \dot{u}_{1J}$) is

$$\dot{u}_{1J} = a_w \varepsilon_{1J} = \frac{1}{E} a_w \sigma_{1J} = \frac{a_w A}{EA_w} [\rho g H_J] \quad (9)$$

Since $\dot{u}_{2J}$ is deemed to be zero, the values of H_J , V_{1J} and $\dot{u}_{1J}$ can now be obtained by solving Equations (5, top and bottom) and (9) simultaneously, and the value of V_{2J} then follows from Equation (5, middle).

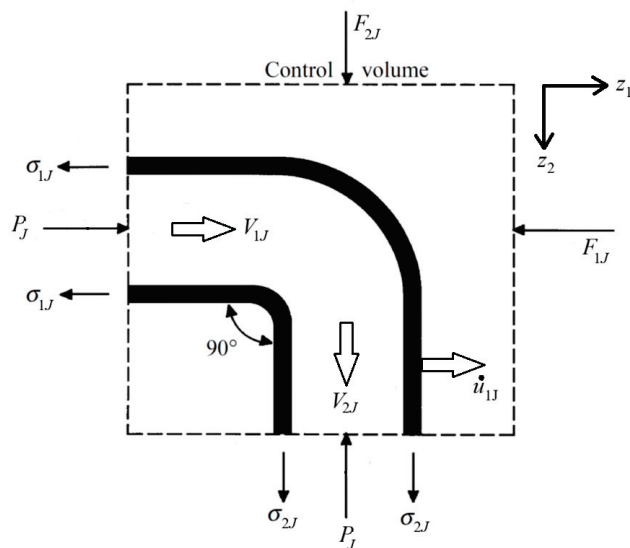


Figure 6. Stationary control-volume surrounding the bend.

It is acknowledged that, although there are no externally imposed forces in the z_1 direction in this example, the assumption of zero structural resistance to the movement of pipe 1 is not exactly correct. The axial movement of pipe 1 is resisted by bending in pipe 2. The shear force that is needed in this pipe to cause this corresponds to an axial force in pipe 1, and a further axial force is required to overcome associated inertial forces. Although a shear force in pipe 2 will reduce the axial velocity $\dot{u}_{1J}$, the bend is mostly held in position by the nearest rigid support of pipe 1. The stress-wave reflected from this support will pull the bend back.

4.4. Cartwright's Test Problem with Fluid-to-Structure Coupling

The following test problem is a variation of one discussed by Cartwright (1977) [13]. His two-way FSI model explored waterhammer waves interacting with a 'lumped' pipe system (i.e., the bend and connected piping were represented by a spring-mass k - m system, see Figure 7) as applied by Wood & Chao (1971) [21], Wiggert et al. (1985) [22], Khalighi et al. (2024) [23]. In such models, the moving bend's mass and elastic support are estimated values; there are no waves propagating in the pipe walls. In this instance, the interaction is one-way FSI (fluid-to-structure) and the bend is free to move only in the direction of the axis of pipe 1. In Section 5.2, one-way FSI (structure-to-fluid) is considered for the same pipework. The pipe data are as follows: length $L_1 = L_2 = 10$ m and internal diameter $D = 0.0762$ m (i.e., 3 inch), giving a flow area of $A = 4.56 \cdot 10^{-3}$ m². The selected lowest mass (in Cartwright's 24 test cases) is $m = 40$ kg; so, for a density of $\rho_w = 7850$ kg/m³ (present authors' estimate) and rigid-body pipe 1 motion, the wall area is $A_w = m/(\rho_w L_1) = 5.1 \cdot 10^{-4}$ m², implying a wall thickness of approximately $e = A_w/(\pi D) = 2.1$ mm. The selected lowest stiffness is $k = 10^5$ kg/s² (of importance in Section 5.2). The waterhammer wave speed was estimated as approximately $a = 1280$ m/s.

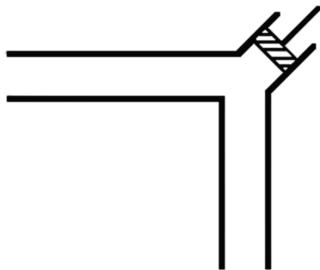


Figure 7. Spring-mass system according to Cartwright 1977 ([13], Figure 10.8).

In Cartwright's example, the initial condition is stationary pipework conveying a steady water flow with a high velocity V_0 of 5 m/s. This requires the pipe system to be anchored in some manner to resist the fluid-momentum forces at the bend. Inviscid flow is assumed and all subsequent values of the liquid and structural parameters are expressed relative to these initial conditions. For reasons that will become clear in the next paragraph, the prescribed initial flow is in the negative axial directions, i.e., from inlet at the $+z_2$ end of pipe 2 to outlet through a valve at the $-z_1$ end of pipe 1; this is compliant with the direction of the pressure wave depicted in Figure 1.

A valve at the outlet from pipe 1 is now closed instantaneously, causing a Joukowsky pressure head of $H = a V_0/g = 652$ m (with $g = 9.81$ m/s²). This implies a huge pressure force on the valve, namely, $F = P A = \rho g A H = 29.2$ kN (with $\rho = 1000$ kg/m³). Nevertheless, neglecting Poisson's ratio effects, the stipulation that the valve does not move ensures that no axial stresses are induced in the pipe wall. Therefore, the resulting behaviour (*relative to the initial conditions*), is as described by Equations (5) and (9) for the special case of initially zero flow. Using Equation (7), the axial stress induced in the pipe wall (by pressure-wave impact at the bend) is $\sigma_{1J} = F/A_w = 57$ MPa and, using a Young's modulus of $E = 200$ GPa (present authors' estimate), the corresponding strain is $\varepsilon = \sigma_{1J}/E = 0.00029$. The implied speed of sound in the pipe wall is $a_w = \sqrt{E/\rho_w} = 5050$ m/s, so the velocity of the bend given by Equation (9) is $\dot{u}_{1J} = 5050 \cdot 0.00029 = 1.5$ m/s (derived via "Joukowsky": $\sigma_{1J} = \rho_w a_w \dot{u}_{1J}$). This velocity remains constant for a period of $\Delta t = 2 L_1/a_w = 4.0$ ms, during which, the pipe extends by $\dot{u}_{1J} \Delta t = 2 L_1 \varepsilon = 5.8$ mm. Note that the displacement is twice its static value $L_1 \varepsilon$, which is one argument for the common use of dynamic load factors of 2. The value of 5.8 mm is used in Section 5.2 to assist in the choice of a suitable initial displacement in forced-vibration and snap-back tests. In mechanics, a *snap-back test* is

a dynamic testing method where a pre-loaded structure is suddenly released to induce free vibration, to study stiffness, damping, and unexpected behaviour (due to cracks and other faults).

4.5. Bend Allowed to Move in Both Axial Directions: $\dot{u}_{1J} \neq 0; \dot{u}_{2J} \neq 0$

We now consider the more probable case of a bend that is free to move in both directions, z_1 and z_2 , as sketched in Figure 8.

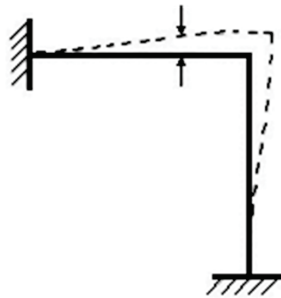


Figure 8. Deflected bend according to Cartwright 1977 ([13], Figure 10.13).

Equations (7)–(9) are valid for this case too, and the corresponding equations for pipe 2 are as follows:

$$\begin{cases} \sigma_{2J} = \frac{A}{A_w} \rho g H_J \\ \varepsilon_{2J} = \frac{\sigma_{2J}}{E} \\ -\dot{u}_{2J} = a_w \varepsilon_{2J} = \frac{1}{E} a_w \sigma_{2J} = \frac{a_w A}{E A_w} \rho g H_J \end{cases} \quad (10)$$

The values of H_J , $\dot{u}_{1J}$ and $\dot{u}_{2J}$ can be obtained by solving Equations (5, bottom), (9) and (10, bottom) simultaneously, and the values of V_{1J} and V_{2J} then follow from Equation (5, top, middle).

The different signs of $\dot{u}_{1J}$ and $\dot{u}_{2J}$ in Equations (9) and (10, bottom) arise because the positive directions of both axes (z_1 and z_2) correspond with the direction of flow whereas the directions of bend movement are determined by the forces in the pipe walls that cause the pipes to extend (see Figure 6).

It is noted in passing that, for the particular case under discussion, the geometrical and mechanical properties of the two pipes are identical, so the absolute values of $\dot{u}_{1J}$ and $\dot{u}_{2J}$ are equal. Therefore, $\dot{u}_{1J} - \dot{u}_{2J} = 2\dot{u}_{1J}$ and so the only adjustment that would need to be made to Equation (5) to make them suitable for this more general case is the replacement of $\dot{u}_{1J}$ by $2\dot{u}_{1J}$. Naturally, this simple substitution is appropriate only for the particular case of a 90° bend in a uniform pipe.

4.6. Swaffield's Experiments

Swaffield's test rig and basic data are shown in Figure 2 above. The geometrical data for the polythene (PE) pipes are as follows: $L_1 = 22 \text{ ft} = 6.7 \text{ m}$, $L_2 = 18 \text{ ft} = 5.5 \text{ m}$ (18 feet in Figure 2a, but 19 feet on page 606 in [2]), and $D = 2 \text{ inch} = 0.051 \text{ m}$. The wall thickness is not stated. The pipe-wall properties are $\rho_w = 930 \text{ kg/m}^3$ and $E = 0.5 \text{ GPa}$, so, neglecting Poisson coupling, the speed a_w of sound in the walls would be 733 m/s.

The water properties are $\rho = 1000 \text{ kg/m}^3$ and $K = 2.1 \text{ GPa}$, implying a speed of sound in an unconfined medium of $a = \sqrt{(K/\rho)} = 1449 \text{ m/s}$. It is difficult to infer the actual wave speed, allowing for the pipe's radial flexibility, because some conflicts exist in Swaffield's paper [2]. For example, Figure 2 in that paper gives $2L_2/a = 32 \text{ ms}$, which implies $a = 343 \text{ m/s}$, whereas his Figure 6 gives $2L_2/a = 46 \text{ ms}$, which implies $a = 239 \text{ m/s}$.

Some clues to the true speed can be obtained by considering measured flow parameters. Based on Figure 2 in [2], the initial flow velocity was $V_0 = 3.1 \text{ ft/s} = 0.95 \text{ m/s}$ and the resulting pressure rise at the downstream valve in pipe 2 was $\Delta P = 29.3 \text{ lbf/in}^2 = 202 \text{ kPa}$. Using these values

and an estimated wall thickness $e = D/10$, the Joukowsky equation gives $a = \Delta P / (\rho V_0) = 214 \text{ m/s}$ where Korteweg's expression gives $a = \sqrt{(K/\rho) / \sqrt{(1 + (D/e) K/E)}} = 221 \text{ m/s}$. The numerical values in the following paragraph are based on a nominal wave speed of $a = 214 \text{ m/s}$.

Swaffield observed pressure reflections of up to 16% of ΔP from 90° bends in both polythene and steel pipes. To assess the implications of this for inferred bend motion, consider an initial condition of steady, frictionless flow with a velocity of V_0 and a gauge pressure of zero. Sudden closure of an immovable valve (at the end of pipe 2 as in Figure 2a) would cause a pressure wave that reduces the velocity to zero and increases the pressure by $\Delta P = \rho a V_0$. If a reflected wave from the bend then reduces the pressure by 16%, it must also cause a velocity increase of $0.16 \Delta P / (\rho a)$ in the original direction of the wave—i.e., in pipe 2 towards the bend. To infer what this implies about the bend motion, we must first consider the wave transmitted into pipe 1. Assuming continuity of pressure at the bend, the amplitude of this transmitted wave is $0.84 \Delta P$ so the induced velocity will be $0.84 V_0$. That is, the fluid velocity in pipe 1 towards the bend exceeds the fluid velocity in pipe 2 from the bend by $0.32 V_0$. Continuity (Equation (3, bottom)) requires (for positive velocity from pipe 1 to pipe 2) the following: $-0.32 V_0 = V_{2J} - V_{1J} = -(\dot{u}_{1J} - \dot{u}_{2J})$.

From symmetry, expressed by Equations (9) and (10, bottom), it follows that $\dot{u}_{2J} = -\dot{u}_{1J}$, implying net movement at 45° to both axes. Consequently,

$$\begin{cases} \dot{u}_{1J} = -\dot{u}_{2J} = 0.16 V_0 = 0.15 \text{ m/s} \\ \dot{u}_{\text{bend}} = \dot{u}_{1J} \sqrt{2} = 0.021 \text{ m/s} = 21 \text{ mm/s} \end{cases} \quad (11)$$

It is re-emphasised that this estimate is subject to considerable uncertainty, partly because it is based on uncertain data and partly because the process that has led to it includes a number of assumptions. Nevertheless, unless the result is greatly in error, useful qualitative conclusions can be made. In particular, even if this velocity were maintained for the whole of the time $2L_1/a_w$ required for stress waves in the pipe wall to travel back and forth, *namely*, approximately 18 ms, the total movement would be only about 0.4 mm (18 ms times 21 mm/s). Thus, the displacements implied by the measured deflections are so small (and so short-lived) that there is no wonder that Swaffield originally believed it to be reasonable to regard the bend as fixed. An important corollary of this inference is that it will be easy for engineers who are not fully aware of this aspect of FSI to repeat Swaffield's mistake even when they actively try not to do so.

5. Analysis: Structure-Induced Disturbances in the Fluid

Attention now turns to one-way 'pipe to fluid' FSI, in which a prescribed external vibration of a bend causes pressure waves in the contained liquid. To enable these to be seen clearly, all other possible causes of pressure waves are excluded. Likewise, in common with the fluid-to-structure cases, all possible causes of damping are excluded.

5.1. Bend Allowed to Move in One Axial Direction (z_1)

The assessment begins with the particular case of a bend that is prevented from moving in the z_2 direction and is acted on by structural forces that induce a harmonic vibration on the z_1 axis at the bend (Figure 3a):

$$u_{1J} = U \cos(\omega t) \quad \text{so that} \quad \dot{u}_{1J} = -U\omega \sin(\omega t) \quad (12)$$

in which ω denotes the angular frequency and U the amplitude of displacement. Note that the displacement and velocity are 90° out of phase and that $\omega = 0$ describes a rigid (fully anchored) bend. In the following development, both U and ω are regarded as prescribed values. This causes—if the frequency is high enough—pressure waves to propagate along

both pipes. They will do so without change of form (d'Alembert's principle) until reaching the remote ends of the pipes.

In the following development, the liquid is initially at rest ($V_0 = 0$) and the pressure is uniform ($H_0 = 0$). At some instant, the external structural excitation is suddenly switched on and begins to induce pressure disturbances in both pipes. The pipes are deemed to be infinitely long (i.e., as in the experimental results presented by Swaffield [2], no reflections occur in the time window considered), so all wave propagation in them is exclusively in the direction away from the bend. Figure 9 depicts the wave propagation in (z - t) space.

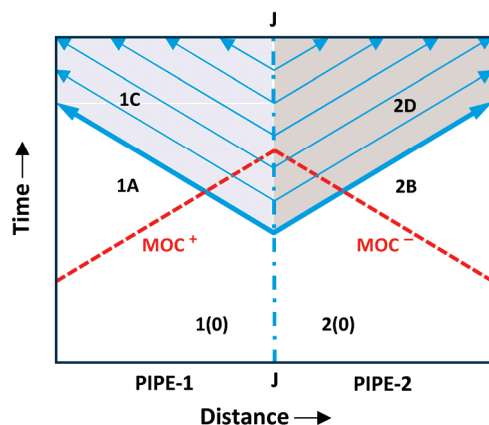


Figure 9. Wave paths depicting symmetric propagation in pipes 1 and 2. Snap-back starts at $t = 0$ at the bend (J).

5.1.1. Time-Domain Analysis

With $\dot{u}_{1J}$ as defined in Equation (12) and $\dot{u}_{2J} = 0$, the MOC Equation (4) become

$$\begin{cases} V_{1J} = \frac{g}{2a}(H_{1A} - H_{2B}) + \frac{1}{2}[(V_{1A} + V_{2B}) - \omega U \sin(\omega t)] \\ V_{2J} = \frac{g}{2a}(H_{1A} - H_{2B}) + \frac{1}{2}[(V_{1A} + V_{2B}) + \omega U \sin(\omega t)] \\ H_J = \frac{1}{2}(H_{1A} + H_{2B}) + \frac{a}{2g}[(V_{1A} - V_{2B}) + \omega U \sin(\omega t)] \end{cases} \quad (13)$$

For the prescribed harmonic disturbance, the conditions in the regions 1C and 2D vary continually, but in both pipes, characteristic paths meeting at the bend (J) can be projected back to the period of zero flow (N.B. compare with Figure 5, in which this is the case only in pipe 2). Then, all of H_{1A} , H_{1B} , V_{1A} and V_{1B} are zero (regions 1A = 1(0) and 1B = 2(0)) so that Equation (13) become the following:

$$\begin{cases} V_{1J} = -\frac{1}{2}\omega U \sin(\omega t) \\ V_{2J} = +\frac{1}{2}\omega U \sin(\omega t) \\ H_J = +\frac{a}{2g}\omega U \sin(\omega t) \end{cases} \quad (14)$$

5.1.2. Symbolic Solution for Steady Oscillations in Pipes of Finite Length

Although Equation (14) can be evaluated analytically, they are valid only when all wave propagation is in the directions away from the bend. Then, the characteristic paths can be projected back as far as the time $t = 0$, at which ambient conditions prevail. This is always possible for pipes of infinite length, but for finite-length pipes, it is possible only during $t < 2L/a$, where L is the smaller of L_1 and L_2 . Thereafter, the true behaviour at the bend is influenced by the remote boundaries. Solutions can still be obtained using Equation (13) together with the appropriate boundary conditions at the remote ends, but this will usually require the use of a computer.

It is nevertheless possible to obtain information about the bend-induced waves at some later times analytically. In particular, solutions can be developed for the large-time

condition after the initial transient behaviour has gradually died away and the overall behaviour has settled down to a steady-oscillating state. This behaviour occurs when the forcing parameter (i.e., the prescribed external harmonic force) is maintained indefinitely. Tijsseling et al. [24,25] obtained a solution for steady oscillations in a single pipe [24] or two pipes in series [25] with constant pressure at one boundary and harmonic pressure or velocity excitation at the other boundary. That is,

$$\begin{aligned} \text{Initial condition : } & \text{not influencing steady oscillatory state} \\ \text{Remote boundaries : } & H_1(z_1 = 0, t) = 0 \text{ and } H_2(z_2 = L_2, t) = 0 \end{aligned} \quad (15)$$

Reflected and transmitted pressures and flows in pipes 1 and 2 interact at the bend where $z_1 = L_1$ and $z_2 = 0$. If $L_1 = L_2 = L$, and the remote-end conditions are the same (Equation (15)), one can use symmetry: $V_{1J} = -V_{2J}$. The pipe 1 solution for this case is as follows:

$$V_1(z_1, t) = -\frac{1}{2}\omega U \sin(\omega t) \frac{\cos\left(\frac{\omega z_1}{a}\right)}{\cos\left(\frac{\omega L}{a}\right)} \quad (16)$$

$$H_1(z_1, t) = +\frac{a}{2g}\omega U \cos(\omega t) \frac{\sin\left(\frac{\omega z_1}{a}\right)}{\cos\left(\frac{\omega L}{a}\right)} \quad (17)$$

By inspection, at the bend ($z_1 = L$, $z_2 = 0$), the fluid velocities (Equation (16)) are the same as those in the example without reflected waves (Equation (14, top)). However, the pressures in the two cases are different (Equations (14, bottom) and (17)). In the steady-oscillating state, they are modulated by $\tan(\omega L/a)$. The amplitude of the steady oscillations depends on the driving frequency. In the special case when $\omega L/a = \pi/2$, the denominator in Equation (17) is zero and resonance occurs. The absolute frequency corresponding to this case is $f = \omega/(2\pi) = a/(4L)$.

5.2. Cartwright's Test Problem with Structure-to-Fluid Coupling

The basic data for the following case are the same as those used for the corresponding valve-closure simulation in Section 4.4. The bend has a prescribed harmonic motion due to a small initial displacement in the positive z_1 direction.

To ensure realistic behaviour, the chosen amplitude of the maintained forced harmonic vibration is $U = 0.3$ mm, which is approximately 5% of the one-off displacement estimated in the valve-closure case in Section 4.4. The angular frequency, estimated from Cartwright's fundamental spring-mass system, is $\omega = \sqrt{k/m} \approx 50$ Hz, so the absolute frequency $f = \omega/(2\pi) \approx 8$ Hz. Using Equation (14, bottom), the amplitude of the bend-motion-induced pressure head *without* reflections is $a \omega U/(2g) \approx 1$ m. With $U = 5.8$ mm, the amplitude of the pressure head is about 20 m, which is small in comparison with the Joukowsky head of 642 m obtained in the valve-closure case (without valve motion).

Figure 10 compares pressure fluctuations obtained for this case. The continuous line shows early time behaviour given by the MOC solution—Equation (14, bottom)—and is applicable only before reflections begin to arrive from the remote ends of the pipes. The broken line shows the large-time behaviour given by the symbolic solution, allowing for the finite lengths of the pipes—Equation (17) for $z_1 = L$. For this particular case, the large-time amplitudes (with reflections from far ends) are more than twice smaller than the small-time ones (without any reflections).

This behaviour is broadly representative of that corresponding to many frequencies of excitation, but it will clearly not be so when the frequency of the forced structural excitation is close to a resonance frequency. This is illustrated in Figure 11, which shows the dependency of the maximum pressure head on the imposed frequency. In this example, the resonance frequency $f = a/(4L) = 1280/40 = 32$ Hz and the angular frequency is $\omega = 2\pi f = 201$ rad/s.

In principle, symbolic steady-oscillatory solutions can be obtained for the $L_1 \neq L_2$ case (Swaffield's system) as well, but this is outside the scope of this paper.

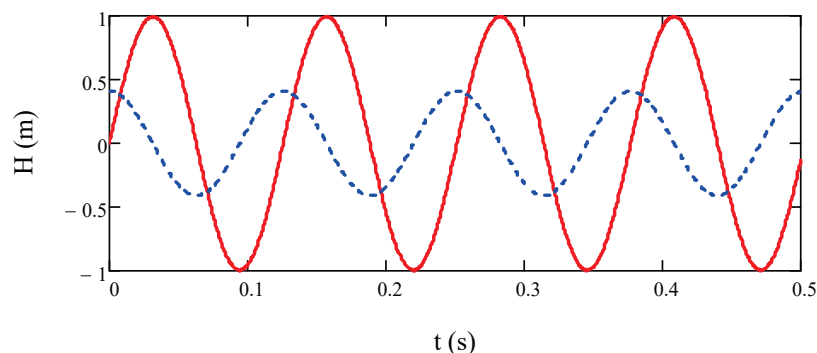


Figure 10. Pressure head oscillations at the bend induced by structural excitation with an amplitude of 0.3 mm and a frequency of 50 Hz (steel pipe, $L_1 = L_2 = L = 10$ m). Continuous line = small time: before reflections from remote ends. Broken line = large times: steady oscillating flow.

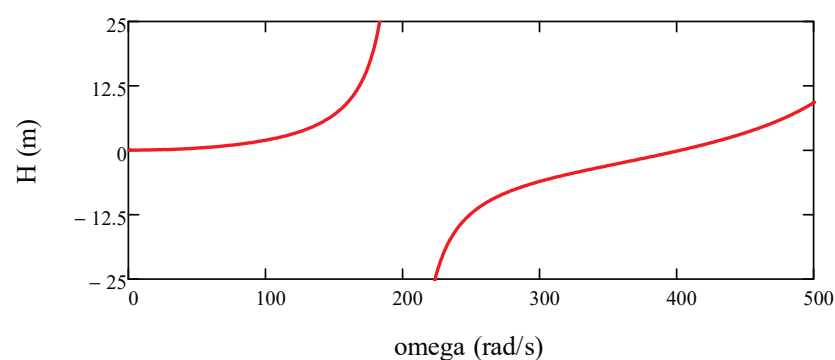


Figure 11. Dependence of the amplitude of the pressure head on the forcing's angular frequency; large-time behaviour.

6. Conclusions

To a close approximation, waterhammer waves do not 'see' immovable bends in uniform pipes: they simply pass through them. Tiny reflections due to minor losses can exist but are unlikely to be detectable in practical applications. However, waterhammer waves are commonly strong enough to make a bend move, and in the case of positive waves, this causes negative reflections that *will* be detectable. The resulting behaviour has been discussed and has been analysed in a simple manner by disregarding potential influences of secondary effects such as damping and rotation. The analysis is of practical use because the early time responses of the pipework when pressure waves pass through a bend are not strongly influenced by secondary effects. The analysis of this case has been undertaken allowing for two-way FSI.

The influence of externally forced vibration of bends on induced pressure waves has also been assessed, in this case by simulating the behaviour as only one-way FSI (structure to fluid). Again, secondary effects have been neglected because they have only a small influence on the initial responses of the fluid, although they can be important if the vibrations induce resonance.

The motion of bends can be deduced from reflections observed in measured pressure signals, by a sort of inverse analysis (assessing structural stiffness). In this way, the transient bend displacement in Swaffield's classical experiments [2] has been estimated to have been no greater than 0.4 mm in a 6 m long pipe, something barely visible to the human eye.

In closing, we note that the type of resonance predicted by one-way FSI models (fluid-to-structure) will not exist in practice because of two-way effects that exist because of stress waves

that are induced simultaneously with the pressure waves [26] Wiggert & Tijsseling (2001). However, one-way coupling at bends and other junctions is nevertheless informative and it is easily implemented as a capacitance element (Figure 3a) in professional waterhammer software such as AFT Impulse, ALLIEVI, ARIETE, ARTS, Bentley HAMMER, BOSFluids, CEBELMAIL, DRAKO, DSHplus, DYAGATS, EPA SURGE, FLOW 3-D, FLOWMASTER 2, H₂O SURGE, HYPRESS, HYTRAN, KYPIPE, LIQT, MIKE URBAN, PIPENET, SENARIET, SIR 3S, SURGE, Synergi Pipeline Simulator, TRANSAM, TSNet, uSLAM, WANDA, WH, and WHAMO.

Author Contributions: Conceptualization, A.S.T.; Investigation, C.J.B.C.; Writing, review and editing, A.E.V. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Data Availability Statement: No new data were created in this study. The paper is based on a re-interpretation of results obtained by other authors, supplemented by some new simulations. For the data shown in Figures 10 and 11, refer to the corresponding author.

Acknowledgments: This Technical Note is based on a presentation held by the first author at the 40th IAHR World Congress, Vienna (2023), session IAHR_6_S1: Transient Flows. Kanayochukwu Ibiam of Magdeburg-Stendal University of Applied Sciences is thanked for preliminary numerical simulations of bend motion within waterhammer MOC.

Conflicts of Interest: The authors declare no conflicts of interest.

Nomenclature

A	area (m ²)
a	wave speed (m/s)
D	pipe inner diameter (m)
E	Young modulus (Pa)
e	pipe wall thickness (m)
F	force (N)
f	frequency (Hz)
g	gravitational acceleration (m/s ²)
H	pressure head (m)
K	bulk modulus (Pa)
k	stiffness (N/m)
L	pipe length (m)
m	mass (kg)
P	pressure (P)
t	time (s)
U	maximum displacement of bend (m)
u	displacement of bend (m)
$\dot{u}$	velocity of bend (m/s)
V	velocity of fluid (m/s)
z	axial distance (m)
Δ	change, finite difference
ε	axial strain
ρ	mass density (kg/m ³)

σ	axial stress (Pa)
ω	angular frequency (rad/s)
Subscripts	
A, B	earlier positions along characteristic lines in the distance–time plane
J	junction
w	wall
0	initial
1	connection of pipe 1 and bend
2	connection of pipe 2 and bend

References

- Contractor, D.N. The reflection of waterhammer pressure waves from minor losses. *ASME J. Basic Eng.* **1965**, *87*, 445–451; discussion 451–452. [CrossRef]
- Swaffield, J.A. The influence of bends on fluid transients propagated in incompressible pipe flow. *Proc. Inst. Mech. Eng.* **1968–1969**, *183*, 603–609; discussion 609–614. [CrossRef]
- Tijsseling, A.S. An overview of fluid-structure interaction experiments in single-elbow pipe systems. *J. Zhejiang Univ. Sci. A* **2019**, *20*, 233–242. [CrossRef]
- Wiggert, D.C. Coupled transient flow and structural motion in liquid-filled piping systems: A survey. In *Proceedings of the ASME Pressure Vessels and Piping Conference, Chicago, IL, USA, 20–24 July 1986*; Paper 86-PVP-4; ASME: New York, NY, USA, 1986.
- Leslie, D.J.; Tijsseling, A.S. A review of modelling damping mechanisms in coupled liquid-pipe vibrations. In *Proceedings of the 3rd ASME & JSME Joint Fluids Engineering Conference, Symposium S-290 Water Hammer, San Francisco, CA, USA, 18–23 July 1999*; Liou, J.C.P., Ed.; Paper FEDSM99-6878; ASME-FED: New York, NY, USA, 1999; Volume 248. Available online: https://www.researchgate.net/publication/293235462_A_review_of_modelling_damping_mechanisms_in_coupled_liquid-pipe_vibrations (accessed on 17 February 2026).
- Vardy, A.E. On sources of damping in water-hammer. *Water* **2023**, *15*, 385. [CrossRef]
- Wylie, E.B. Unsteady internal flows—Dimensionless numbers & time constants. In *Proceedings of the 7th International Conference on Pressure, Harrogate, UK*; Boldy, A., Ed.; BHR Group: Harrogate, UK, 1996; pp. 283–288.
- Itō, H. Pressure losses in smooth pipe bends. *ASME J. Basic Eng.* **1960**, *82*, 131–140, discussion 140–143. [CrossRef]
- Smyk, E.; Stopel, M.; Szyca, M. Simulation of flow and pressure loss in the example of the elbow. *Water* **2024**, *16*, 1875. [CrossRef]
- Bergant, A.; Tijsseling, A.S.; Vítkovský, J.P.; Covas, D.I.C.; Simpson, A.R.; Lambert, M.F. Parameters affecting water-hammer wave attenuation, shape and timing. Part 2: Case studies. *IAHR J. Hydraul. Res.* **2008**, *46*, 382–391. [CrossRef]
- Florence, A.L. Pulse propagation in fluid-filled pipes. In *Proceedings of the Third International Conference on Structural Mechanics in Reactor Technology, SMiRT3, London, UK, 1–5 September 1975*; Paper F 4/8. Available online: <https://repository.lib.ncsu.edu/server/api/core/bitstreams/6671d48f-26a2-4ca3-b1ee-5a2206e88798/content> (accessed on 17 February 2026).
- Youngdahl, C.K.; Kot, C.A.; Valentin, R.A. Pressure transient analysis of elbow-pipe experiments using the PTA-2 computer code. *ASME J. Press. Vessel. Technol.* **1981**, *103*, 33–42. [CrossRef]
- Cartwright, C.J.B. Pressure Transient Reflections at Pipe Bends. Ph.D. Thesis, Department of Civil Engineering, The University of Leeds, Leeds, UK, 1977.
- Vardy, A.E.; Tijsseling, A.S. On Boussinesq and Coriolis coefficients and their derivatives in transient pipe flows. *Appl. Math. Model.* **2022**, *111*, 454–470. [CrossRef]
- Martin, P.A. Going round the bend: Reflection and transmission of long waves by waveguide corners and labyrinths. *Proc. R. Soc. A Math. Phys. Eng. Sci.* **2023**, *479*, 20230635. [CrossRef]
- Wilkinson, D.H. Dynamic response of pipework systems to water hammer. In *Proceedings of the 3rd International Conference on Pressure Surges, Canterbury, UK, 25–27 March 1980*; BHRA: Cranfield, UK, 1980; pp. 185–202.
- Wylie, E.B.; Streeter, V.L. *Fluid Transients in Systems*; Prentice Hall: Englewood Cliffs, NJ, USA, 1993.
- Belytschko, T.; Karabin, M.; Lin, J.I. Fluid-structure interaction in waterhammer response of flexible piping. *ASME J. Press. Vessel. Technol.* **1986**, *108*, 249–255. [CrossRef]
- Belytschko, T.; Lin, J.I.; Csay, C.-S. Modal recovery methods for solution of fluid-structure problems with rigid wall loads. *Nucl. Eng. Des.* **1982**, *71*, 67–78. [CrossRef]
- Tijsseling, A.S.; Vardy, A.E.; Fan, D. Fluid-structure interaction and cavitation in a single-elbow pipe system. *J. Fluids Struct.* **1996**, *10*, 395–420. [CrossRef]
- Wood, D.J.; Chao, S.P. Effect of pipeline junctions on water hammer surges. *ASCE Transp. Eng. J.* **1971**, *97*, 441–456. [CrossRef]

22. Wiggert, D.C.; Otwell, R.S.; Hatfield, F.J. The effect of elbow restraint on pressure transients. *ASME J. Fluids Eng.* **1985**, *107*, 402–406, discussion 108, 121–122. [CrossRef]
23. Khalighi, F.; Ahmadi, A.; Keramat, A.; Tijsseling, A.S.; Zecchin, A.C. An efficient estimation of fluid–structure interaction in blocked L-shaped pipelines. *Iran. J. Sci. Technol. Trans. Mech. Eng.* **2024**, *48*, 1777–1795. [CrossRef]
24. Tijsseling, A.S.; Hou, Q.; Svingen, B.; Bergant, A. Acoustic resonance in a reservoir-pipeline-orifice system. In *Proceedings of the ASME 2010 Pressure Vessels and Piping Division Conference, Bellevue, WA, USA, 18–22 July 2010*; Paper PVP2010-25083; ASME: New York, NY, USA, 2010. [CrossRef]
25. Tijsseling, A.S.; Hou, Q.; Svingen, B.; Bergant, A. Acoustic resonance in a reservoir—double pipe—orifice system. In *Proceedings of the ASME 2012 Pressure Vessels and Piping Division Conference, Toronto, ON, Canada, 15–19 July 2012*; Paper PVP2012-78085; ASME: New York, NY, USA, 2012. [CrossRef]
26. Wiggert, D.C.; Tijsseling, A.S. Fluid transients and fluid-structure interaction in flexible liquid-filled piping. *ASME Appl. Mech. Rev.* **2001**, *54*, 455–481. [CrossRef]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

Quasi-2D Axisymmetric Modelling of Smooth–Turbulent Transient Flow: Comparison with 1D and 3D-CFD Models

Pedro Ferreira ^{1,*} and Dídia Covas ²¹ Department of Civil Engineering, Polytechnic of Porto, 4200-465 Porto, Portugal² Civil Engineering Research and Innovation for Sustainability, Instituto Superior Técnico, Universidade de Lisboa, 1649-004 Lisboa, Portugal; didia.covas@tecnico.ulisboa.pt

* Correspondence: plf@isep.ipp.pt

Abstract

This study compares the numerical results obtained with an axisymmetric quasi two-dimensional (Q2D) model with those from two different types of dimensional models for smooth–turbulent transient flows generated by an instantaneous valve closure. The first is a high-accuracy three-dimensional computational fluid dynamics (3D-CFD) model. The second type is the one-dimensional (1D) model, analysing results from five unsteady friction formulations, namely four convolution integral-based (CIB) formulations and one instantaneous acceleration-based (IAB) formulation. The Q2D and 1D models are also compared with experimental data collected under laboratory conditions for a fast but non-instantaneous valve closure. The differences between the 1D, Q2D and 3D-CFD modelling approaches are quantified and discussed, focusing on the ability of each model to reproduce the different features of the transient flow phenomenon and the associated computation–accuracy trade-offs. The 3D-CFD model exhibits a more pronounced front-wave rounding, as it more accurately represents the valve conditions, where its closure induces highly turbulent flow with a strongly heterogeneous velocity profile. The Q2D model provides an accurate estimation of unsteady energy dissipation, when fully developed flow conditions are ensured with the advantage of requiring less computational effort than the 3D-CFD. These conclusions also apply to full convolution-based 1D models, whereas approximate 1D formulations can significantly reduce accuracy and limit their range of applicability. The comparison with experimental data corroborates the excellent results of the Q2D model. The Q2D model demonstrates to be an efficient alternative in terms of accuracy and computational time to 3D-CFD and 1D full convolution models.

Keywords: hydraulic transient; 1D unsteady friction models; quasi-2D model; 3D-CFD model; experimental tests

1. Introduction

The development of accurate, simple and computationally efficient numerical models for transient flow simulations has been a major challenge over the past decades. Transient flow modelling is inherently complex. An ideal representation should account for secondary phenomena not captured by the classical Joukowski formulation, including fluid structure interaction, cavitation, pipe-wall viscoelasticity and frequency-dependent energy dissipation [1–5].

The investigation effort to accurately calculate unsteady energy dissipation has followed different paths [6]. The instantaneous acceleration-based (IAB) one-dimensional (1D)

formulations calculate friction losses based on the local and convective instantaneous accelerations, derived directly from mean velocity values without accounting for cross-sectional velocity distribution [7–9]. The convolution integral-based (CIB) 1D models incorporate past local accelerations through weighting functions to account for the velocity profile dynamics. These methods assume an axisymmetric velocity distribution and radially uniform pressure, resulting in higher accuracy compared with IAB formulations [10,11]. The quasi two-dimensional (Q2D) axisymmetric models adopt nearly the same assumptions as the CIB methods and, therefore, tend to have comparable results. However, Q2D models numerically discretise the pipe cross-section into concentric hollow cylinders and compute the axial and radial velocities within each cylinder independently, thereby enabling the incorporation of realistic turbulence models [12,13]. The three-dimensional computational fluid dynamics (3D-CFD) models solve all three velocity components without relying on axisymmetric or uniform-pressure assumptions and allow the use of advanced turbulence models [14–16].

No exact and completely satisfying solution exists for the calculation of unsteady friction in fast pressurised transient flows when using 1D models. The IAB formulations offer the simplest representation of the phenomenon with the fastest computational time, but they lack physical consistency [17–19]. The CIB models involve a high computation effort, and their simplified versions lack accuracy and generality [20,21]. The 3D-CFD models provide the highest level of accuracy in describing the 3D nature of the transient flow, which is particularly important near singularities, yet these models remain computationally prohibitive for the application in current engineering practice [20]. The Q2D models are a promising modelling alternative as they capture solely the most essential features of the fully developed flow during the transient event [22–24]. As a result, the Q2D models are a highly accurate and efficient numerical scheme, enabling the calculation of the wall shear stress according to the velocity profile, coupled with a turbulent model, which is not ensured by unsteady 1D models, while substantially reducing CPU time compared to 3D-CFD and CIB models. Nevertheless, the previous assumptions are not justified in all situations [20,25–28].

From an overly simplistic perspective, a better description of the transient phenomena presupposes more computational effort. Therefore, it is essential to identify the relevant features for a specific flow simulation and to understand the capabilities of each model to correctly select the most appropriate formulation. This paper aims to assess the accuracy and limitations of the Q2D model in comparison with traditional modelling approaches, from simplified 1D formulations to high-fidelity 3D-CFD simulations. First, a detailed comparison is carried out between Q2D and 3D-CFD results for an instantaneous valve closure event. This procedure provides the benchmark for analysing 1D models, enabling the identification of their accuracy and applicability. Additionally, experimental data from a real S-shaped valve closure manoeuvre is used to assess the accuracy of Q2D and 1D models under realistic valve closure conditions.

The current paper innovatively presents the following: (i) a comprehensive analysis of the main modelling approaches for an instantaneous valve closure event in a well-documented system configuration; (ii) an evaluation of the implications of assuming an axisymmetric velocity profile and radially uniform pressure; (iii) experimental data are used to examine the influence of realistic valve closure on the predictions of 1D and Q2D models; and (iv) the identification of future improvements required for the Q2D model to accurately simulate transient events in pressurised pipes while preserving its lower computational effort.

2. Experimental Facility

The experimental data were collected from an experimental pipe facility at the Laboratory of Hydraulics, Water Resources and Environment at the Instituto Superior Técnico (IST). The system has a reservoir–pipe–valve configuration, comprising a copper pipe with an inner diameter (D) of 0.02 m and a total length (L) of 15.2 m (Figure 1).

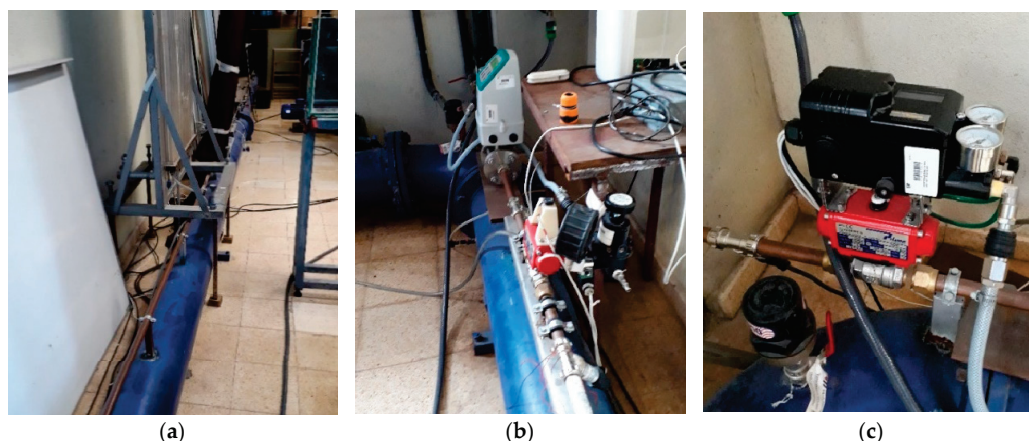


Figure 1. Experimental facility: (a) copper pipe; (b) downstream pipe section; and (c) pneumatically actuated valve.

A hydropneumatic vessel with a pump is installed at the upstream end of the pipeline to simulate a constant-level reservoir. At the downstream end, there are two ball valves: the first is manually operated and is used to control the initial flow rate; and the second is used to generate transient events. This second valve is pneumatically actuated allowing to have different closure times by changing the operating pressure of the valve actuator. The data acquisition system synchronises the valve closure and the acquisition interval of the measuring equipment.

The electromagnetic flowmeter has an internal diameter of 20 mm and an accuracy of 0.4% for a minimum flow rate of 15 L/h. The system is equipped with three pressure transducers with a full-range 0.25% accuracy and a nominal pressure of 25 bar. A detailed description of the experimental facility, hydraulic conditions, test results and valve closure law are detailed in Ferreira and Covas [29] and Ferreira [30].

The pressure data for smooth–turbulent flow was originally collected for three steady-state flow rate values: $Q_0 = 71 \times 10^{-3}$, 96×10^{-3} and 120×10^{-3} L/s. Only the measurements for the highest initial Reynolds number ($Re_0 = 7638$) are presented and analysed herein because the results for the three tests overlapped in the dimensionless form, and no distinction is recognised regarding the wave damping and shape. The fastest valve closure, with an effective closing time of 0.01 s, is evaluated.

Piezometric time histories at the downstream and middle sections of the pipe for the initial flow rate $Q_0 = 120$ L/s ($Re_0 = 7638$) at the downstream end and mid-section transducers are presented in Figure 2.

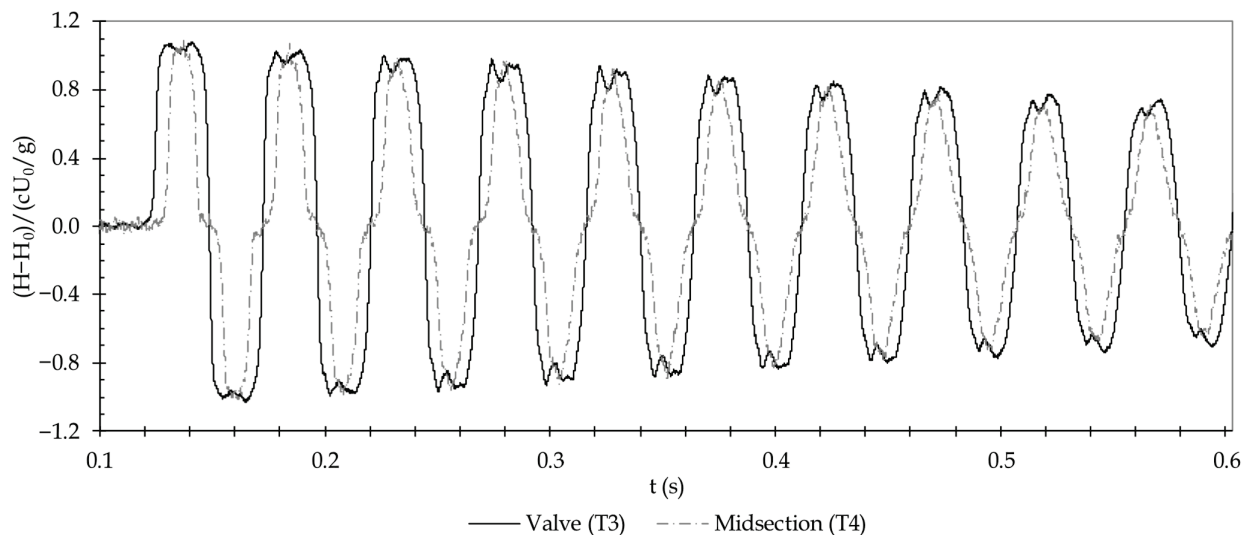


Figure 2. Piezometric time histories at downstream near the valve (Valve—T3) and at the pipe middle section (Midsection—T4) for smooth turbulent flow ($Re_0 = 7638$).

3. Numerical Models

The 1D and Q2D transient solvers and the respective numerical simulations were developed by the authors, whereas the 3D-CFD results were obtained from the previous research of Martins et al. [15], generated using a commercially available CFD software.

3.1. The Quasi-2D Axisymmetric Model

The Q2D model follows the standard 1D assumptions for the fluid and the pipe [12]. By assuming axisymmetric flow, radial velocity and non-axial viscous terms are neglected, which simplifies the axial momentum equation and eliminates the radial momentum equation. This results in a system of two partial differential equations with three unknown parameters (p, u, v) with the following form [13]:

$$\frac{\partial H}{\partial t} + \frac{c^2}{g} \frac{\partial u}{\partial x} = -\frac{c^2}{gr} \frac{\partial (rv)}{\partial r} \quad (1)$$

$$\frac{1}{g} \frac{\partial u}{\partial t} + \frac{\partial H}{\partial x} = \frac{1}{r\rho} \frac{\partial r\tau}{\partial r} \quad (2)$$

$$\tau = \rho\nu \frac{\partial u}{\partial r} - \rho \overline{u'v'} \quad (3)$$

where x is the distance along the pipe; r is the distance from the axis in the radial direction; t is time; H is the piezometric head; $u(x, r, t)$ is the local axial velocity; $v(x, r, t)$ is the local radial velocity; g is the gravitational acceleration; c is the wave speed; ρ is the liquid density; τ is the shear stress; ν is the liquid kinematic viscosity; u' and v' are turbulence fluctuations corresponding to longitudinal velocity u and v , respectively. A five-region turbulence model is used herein to calculate the Reynolds stress term $-\rho \overline{u'v'}$ [31,32].

A geometric sequence (GS) radial mesh is adopted, defined by prescribing the thickness of the outermost cylinder adjacent to the pipe wall, $\Delta r_{j=N_C-1}$, together with a constant common ratio, C_R . The thickness of each cylinder j is given by the following:

$$\Delta r_j = \Delta r_{j=N_C-1} \cdot (1 + C_R)^{N_C-j} \quad (4)$$

where N_C denotes the total number of concentric hollow cylinders, indexed from the centreline ($j = 0$) to the pipe wall ($j = N_C - 1$). For each cylinder, the radial thickness is $\Delta r_j = r_j - r_{j-1}$ and the external radius is $r_j = \sum_{i=0}^j \Delta r_i$.

A GS_{9%}60 radial mesh is used herein, consisting of a total of 60 cylinders ($N_C = 60$), with a 9% increase in cylinder thickness from the wall to the centre of the pipe ($C_R = 9\%$). This configuration was identified as the most suitable mesh for the flow characteristics under analysis. The methodology used to optimise the radial mesh, together with a comprehensive description of numerical implementation, radial and axial grid generation, recommended discretisation strategies and the influence of alternative numerical schemes for the Q2D model are detailed in Ferreira and Covas [33].

3.2. The 1D Unsteady Friction Models

Five 1D unsteady friction models were implemented, comprising four convolution integral-based (CIB) formulations and one instantaneous acceleration-based (IAB) model. For laminar flow, two CIB models were considered: the full Zielke model [10] and the approximate Trikha model [20]. For smooth-turbulent flow, two additional CIB formulations were used: the *full* Vardy and Brown model [11] and the approximate Vardy and Brown [19] model. Finally, the Vitkovsky et al. IAB model [9] was used.

3.2.1. Convolution Integral-Based (CIB) Models

Zielke's analytical solution for laminar unsteady friction is used. In this formulation, the unsteady wall shear stress, τ_{wu} , is defined as the difference between the total wall shear stress, τ_w , and the corresponding steady-state shear stress for the same mean velocity, τ_{ws} [10,34,35]. At a given time, τ_{wu} is expressed as the convolution of a weighting function and past fluid accelerations, depending on the entire velocity history from the start of the transient event until the time being evaluated:

$$\tau_{wu}(t) = \frac{2\mu}{R} \int_0^t \frac{\partial U}{\partial t}(t^*) W(\psi) dt^* \quad (5)$$

in which μ is the fluid viscosity, R is the pipe radius, t is the current time, t^* is the convolution integral time ($t^* = t - t_s$) and is measured backwards from the current instant to the unsteady start, t_s is the time at which the unsteady flow starts, W is the weighting function of the dimensionless time parameter ($\psi = 4t^* \nu / D^2$). The weighting function is defined as follows:

$$W(\psi) = 0.282095\psi^{-\frac{1}{2}} - 1.25 + 1.057855\psi^{\frac{1}{2}} + 0.9375\psi + 0.3966960\psi^{\frac{3}{2}} - 0.351563\psi^2 \quad (6)$$

$$W(\psi) = e^{-26.3744\psi} + e^{-70.8493\psi} + e^{-135.0198\psi} + e^{-218.9216\psi} + e^{-322.5544\psi} \quad (7)$$

for $\psi < 0.02$ and $\psi > 0.02$.

Building on this formulation, Vardy and Brown [1] extended the approach to capture smooth turbulent flow and the new weighting function is expressed as follows:

$$W_a(t) = \frac{\sqrt{(v_w/v_l)} e^{-\psi/C^*}}{2\sqrt{\pi\psi}} \quad (8)$$

where v_l is the laminar kinematic viscosity, v_w is the kinematic viscosity at the wall and C^* is the shear decay coefficient that is also a function of the Reynolds number. The parameter κ denotes the von Kármán constant.

$$C^* = 12.86/Re^\kappa \text{ and } \kappa = \log_{10}\left(15.29/Re^{0.0567}\right) \quad (9)$$

A computationally efficient alternative was introduced by Trikha [20] for laminar flow in which the cumulative effect of the full acceleration history is represented by three exponential terms updated at each time step. The unsteady wall shear stress at the instant $t + \Delta t$ is obtained using the variables y_1 , y_2 and y_3 stored from the previous time step, as follows:

$$\tau_{wu}(t) = \frac{2\mu}{R} (Y_1 + Y_2 + Y_3) \quad (10)$$

$$Y_i(t) = Y_i(t - \Delta t)e^{-n_i \frac{4\nu}{D^2} \Delta t} + m_i[U(t) - U(t - \Delta t)] \quad (11)$$

where m_i and n_i are coefficients that must be fitted such that the approximate weighting function resembles the true weighting function.

In the same direction, a general one-step integration method for smooth–turbulent flow was proposed by Vardy and Brown [9] and is given as follows:

$$\tau_{wu}(t + \Delta t) = \frac{2\rho\nu}{R} \sum_{i=1}^N \left[Y_{ai}(t)e^{-n_i\nu/R^2\Delta t} + \frac{m_i U R^2}{n_i \nu} (1 - e^{-n_i\nu/R^2\Delta t}) \right] \quad (12)$$

in which $\dot{U}$ is the average acceleration of the mean flow during the last time step, m_i and n_i are coefficients defined by the authors according to the selected approximation order and Y_a represents the cumulative contribution of the previous terms.

3.2.2. Instantaneous Acceleration-Based (IAB) Models

The formulation proposed by Vítkovský et al. [9] is adopted and it distinguishes between positive or negative wave propagation, thereby providing broader applicability compared with earlier IAB models. In this approach, the unsteady wall shear stress is expressed as follows:

$$\tau_{wu} = \frac{\rho D}{4} k_3 \cdot \left(\frac{\partial U}{\partial t} - c \cdot \text{sign}(U) \left| \frac{\partial U}{\partial x} \right| \right) \quad (13)$$

where k_3 is a decay coefficient that must be experimentally calibrated and $\text{sign}(U)$ takes the value +1 for $U \geq 1$ and -1 for $U \leq 1$.

3.3. The 3D-CFD Model

The 3D-CFD model was developed by Martins et al. [14–16] using the commercial software ANSYS Fluent to describe the pipe system and the transient phenomena in three dimensions. The model employs a finite-volume numerical discretisation together with a semi-implicit algorithm to solve the mass and all momentum conservation equations. Turbulence was represented using a *realisable* k - ϵ turbulence model. Convergence was assessed through residual monitoring, with solutions considered converged when the relative residuals of the governing equations fell below 1×10^{-6} .

4. Q2D and 3D-CFD Model Results

4.1. Considerations on the Computational Effort

Before comparing the Q2D and 3D-CFD results, it is important to note that the 3D-CFD model demands substantially higher computational time for three main reasons.

Firstly, the 3D-CFD model used a mesh with 1.7×10^6 cells. This mesh was studied by Martins et al. [15] to ensure the best compromise between accuracy and computation effort. It has a uniform circumferential spacing of 0.016 m, a radial discretisation defined by a geometric ratio of 1.24 with a minimum spacing of 10^{-7} m at the wall, and a conduit discretisation of 0.026 m. In contrast, the Q2D model uses a GS_{9%}60 radial mesh (with a conduit interval of 0.05 m), has no circumferential discretisation and requires only

1.8×10^3 cells. Overall, the Q2D grid contains approximately one thousandth of the nodes used in the 3D-CFD model.

Secondly, the 3D-CFD model calculates the three velocity components and assesses the pressure value at each radial node, requiring the solution of four conservation equations for each grid point (continuity, radial and circumferential momentum and energy). Conversely, the Q2D model solves only two conservation equations (mass and axial momentum), as it neglects radial pressure variation and circumferential mass transfer.

Thirdly, the Q2D model uses a frozen-viscosity five-region turbulence model [34], where the eddy viscosity is assumed to be constant over time and equal to its steady-state radial distribution. In the 3D-CFD model, however, the $k-\varepsilon$ turbulence model implies an iterative calculation process until convergence is reached for every time step and node.

Consequently, the computational effort associated with the Q2D model is estimated to be approximately 16,000 times lower than that of the 3D-CFD model, assuming two turbulence model iterations per grid point in the latter.

4.2. Instantaneous Valve Manoeuvre

The piezometric head time history obtained for the 3D-CFD and the Q2D models for the first five periods are presented in Figure 3. The instantaneous valve closure occurs at $t = 0$ and both models exhibit a pressure wave front becoming smoother with time, which is significantly different from the classical Joukowski square-shaped pressure wave.

After the first pressure wave, the 3D-CFD results exhibit a more pronounced front wave rounding than those obtained with the Q2D model (Figure 3a,b). This can be the result of a higher unsteady energy dissipation or a more realistic representation of the valve closure manoeuvre. Additionally, the 3D-CFD model does not exhibit the square-shaped geometry observed in the Q2D immediately before the pressure wave arrival (see, for example, the 3D-CFD and Q2D results in Figure 3d and compare both models' results at $t/T = 0.375$); instead, this model tends to display a more symmetric wave shape.

At $t = 0$, the Q2D model predicts an instantaneous piezometric head surge, with the pressure change occurring within a single time step, as expected for a numerical model that assumes a uniform radial pressure and neglects the circumferential velocity following an instantaneous valve closure. In contrast, the 3D-CFD model produces a very fast (but not instantaneous) S-shape variation (Figure 3c). When the pressure wave reaches the pipe midsection ($t/T = 0.125$; Figure 3d), the discrepancy between the two models becomes more evident, the 3D-CFD results show a more rounded pressure wave, progressively dissipating the rectangular wave geometry that remains visible in the Q2D model prior to the arrival of the pressure surge.

Figure 4a,b shows the wall shear stress, τ_w , time variation at the pipe midsection obtained by both models. The 3D-CFD model results have an immediate reduction of τ_w value and an S-shaped variation (see Figure 3b at $t/T = 0.125$). This behaviour reflects the rapid transformation of the nearly instantaneous velocity change imposed at the valve into a well-defined S-shaped velocity profile at the pipe midsection (Figure 3c). On the contrary, at the same instant ($t/T = 0.125$), the Q2D model highlights a relevant τ_w peak, consistent with the instantaneous pressure variation at the valve and the lower front-wave rounding computed by the Q2D model. The mean velocity variation predicted by each model is coherent with the corresponding wall shear stress response (Figure 3b,c).

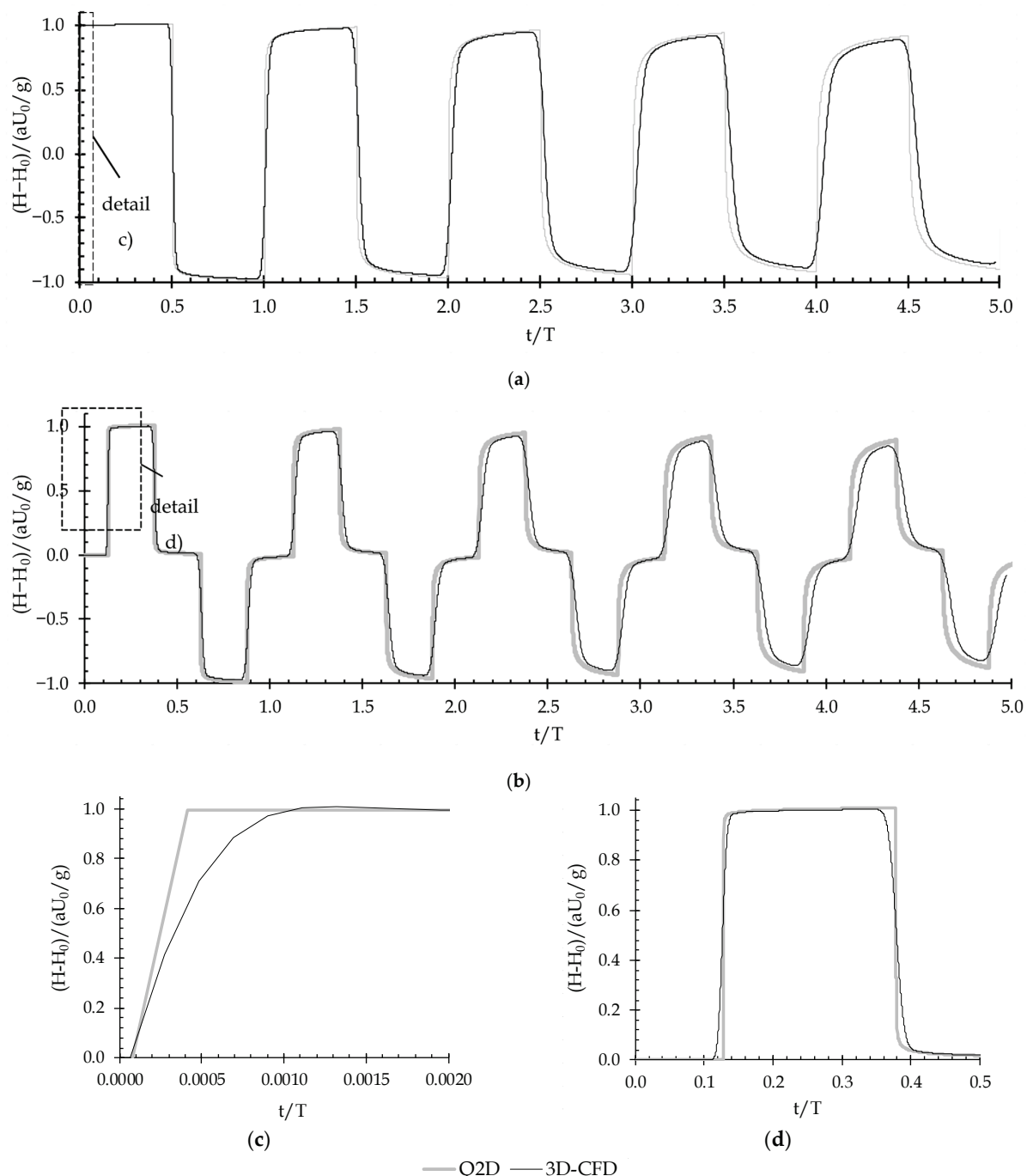


Figure 3. 3D-CFD versus Q2D piezometric head time history: (a) at the pipe end section ($0.0 \leq t/T \leq 5.0$); (b) at the pipe midsection ($0.0 \leq t/T \leq 5.0$); (c) at the pipe end section ($0.0 \leq t/T \leq 0.002$); and (d) at the pipe midsection ($0.0 \leq t/T \leq 0.5$).

Please note that the 3D-CFD model wall shear stress simulation was only run until the second wave period, since it required a high computational time. After the passage of the wavefront, both models converge to nearly identical τ_w variations. The rounder and more symmetric pressure wave predicted by the 3D-CFD model (Figure 3c,d) results primarily from the detailed modelling of the fluid behaviour at the valve during the instantaneous closure, which directly influences the unsteady energy dissipation calculation (especially the peak values), instead of being a consequence of it.

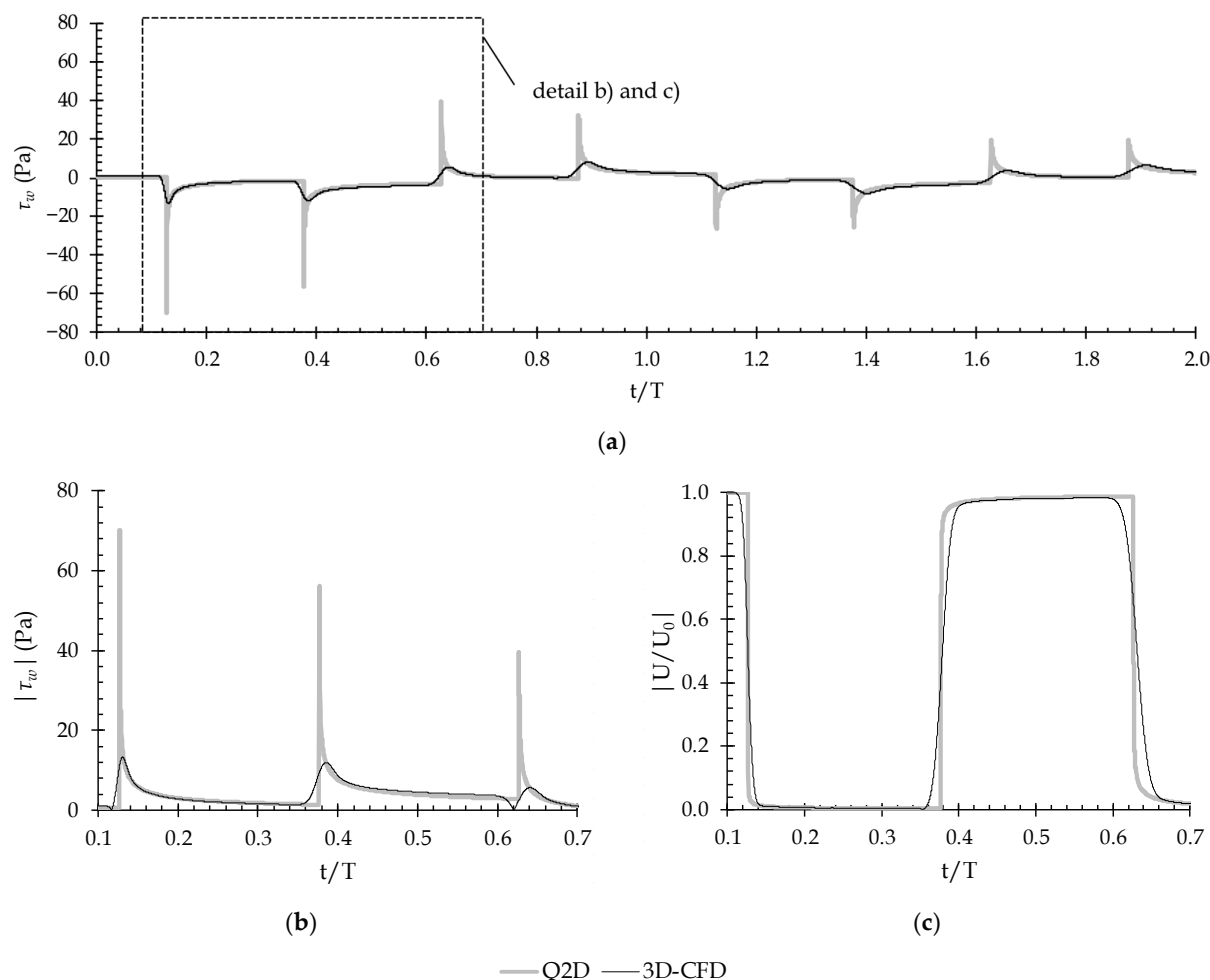


Figure 4. 3D-CFD versus Q2D time history at the pipe midsection: (a) wall shear stress during the complete two wave periods; (b) wall shear stress between $0.1 \leq t \leq 0.7$; and (c) absolute mean velocity variation between $0.1 \leq T \leq 0.7$.

4.3. Valve Closure Calibration

A sigmoidal law valve closure function was implemented in the Q2D model to more accurately reproduce the pressure variation obtained with the 3D-CFD model (Figure 3c). This procedure ensures that both models share identical initial boundary conditions at the valve section. Figure 5 shows the dimensionless flow variation predicted by the 3D-CFD model together with the calibrated sigmoidal law response adopted in the Q2D model for an effective closure time of 0.0014 s. In this formulation, τ denotes the valve closure percentage, varying between 0 (fully open) and 100 (fully closed).

This valve calibration ensures a better agreement between the piezometric head time histories predicted by the two models, particularly during the first two wave periods (compare Figures 3b and 6). An excellent match is also obtained for the wall shear stress during the first pressure wave (Figure 7). These results demonstrate that, when an appropriate valve closure law is applied (calibrated for the Q2D model and partially eliminating the raw representation of valve closure inherent to the Q2D formulation), both models exhibit comparable unsteady energy dissipation, as well as closely aligned mean-velocity and pressure variations. This confirms that the Q2D model can achieve the same unsteady dissipation accuracy as the 3D-CFD model, provided that a fully developed flow is established.

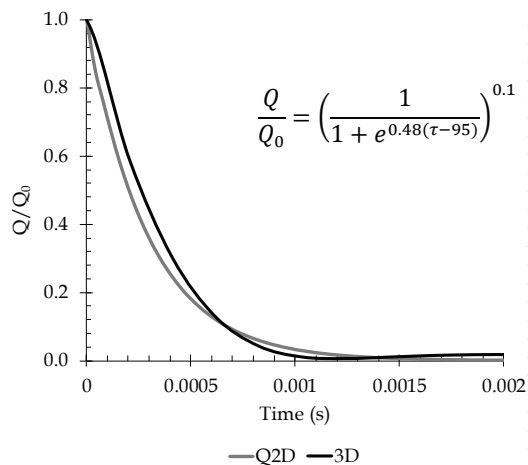


Figure 5. Flow rate at pipe midsection with the 3D model and the calibrated valve manoeuvre with the Q2D model.

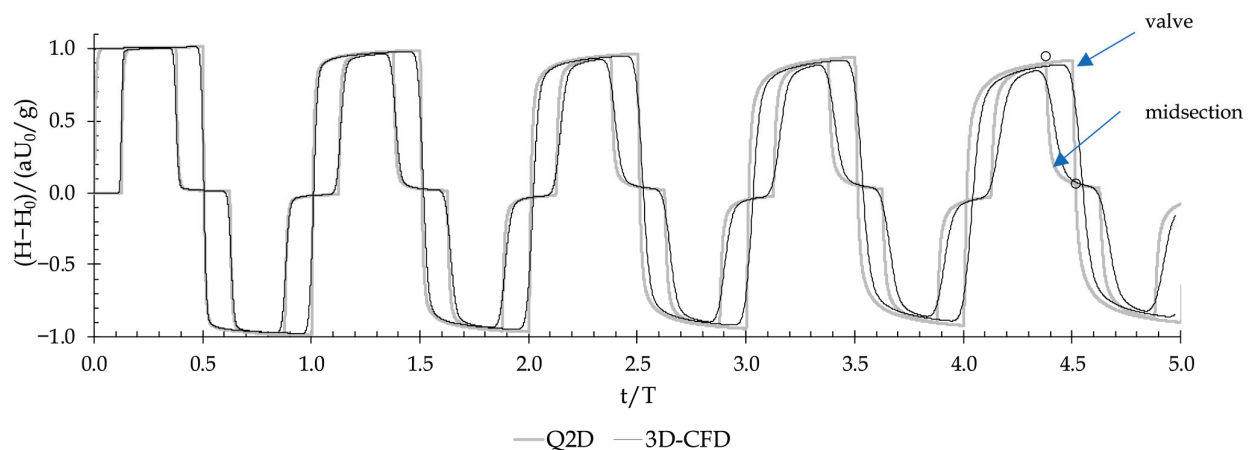


Figure 6. 3D-CFD versus Q2D (calibrated) piezometric time history at the pipe valve and midsection.

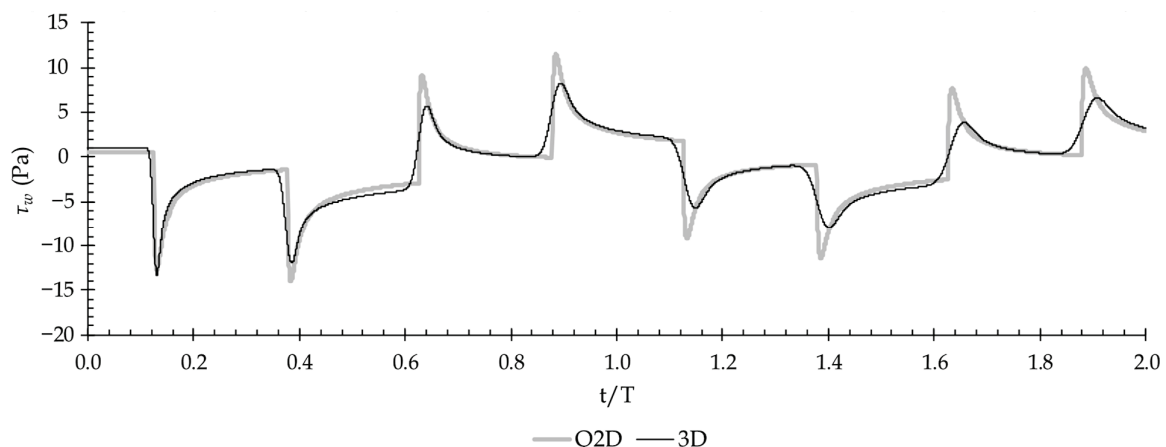


Figure 7. 3D-CFD versus Q2D (calibrated) wall shear stress time history at the pipe midsection.

After the first pressure surge, the 3D-CFD model progressively exhibits lower peak values than the Q2D model, reflecting the latter's slower dissipation of the pressure wave shape and the gradual loss of the valve closure calibration effect. This behaviour suggests that the discrepancies between the Q2D and 3D-CFD models are not limited to the representation of the valve manoeuvre, but also arise from additional factors, such as the wave reflections at the closed valve and at the reservoir.

4.4. Discussion of Differences in Results from the Q2D and the 3D-CFD Models

Differences between the two models' results can be explained by the following 3D-CFD model features not described by the Q2D model: (i) the use of a time-dependent turbulence model; (ii) the circumferential mesh discretisation and the calculation of the respective velocity component; and (iii) the non-uniform radial pressure distribution.

The applicability of frozen-viscosity models, such as the one used in the Q2D model, has been extensively studied [30]. He and Jackson [36] reported that the response of turbulence to the flowrate variations is characterised by three phases (turbulence production, turbulence energy redistribution and radial diffusion) and a time delay is observed. Similarly, Zhao and Ghiaoui [37] identified two important time scales, the radial diffusion time scale and wave time scale. Using these two scales, the authors introduced a dimensionless parameter, $P = 2Da / \sqrt{f}UL$, that quantifies the number of wave periods required for the wall shear stress vortex generated by the first wave to influence the pre-existing turbulent characteristics in the whole pipe cross-section. Applying this parameter to this system (i.e., experimental facility) results in $P = 45$, indicating that the frozen turbulence assumption can be used during the first five wave periods considered.

Zhao and Ghiaoui [11] compared the Q2D model with both the five-region model and $k-\epsilon$ turbulence model for several systems, concluding that the differences are negligible. In high-frequency transient simulations, the largest velocity gradients occur near the wall; however, in smooth-turbulent flows, the eddy viscosity in this region is nearly zero. This is why the quasi-laminar model, which assumes zero eddy viscosity throughout the entire cross-section, is adequate and the Zielke's model presents good results in turbulent flow conditions. This also explains why existing unsteady-friction convolution formulations, which rely on simplified eddy viscosity representations in the core region, remain valid. Moreover, even though the turbulent radial distribution is significant in the pipe core, an accurate specification of the eddy viscosity in this region is not essential, because the velocity gradient in that region is comparatively small.

Additionally, the 3D-CFD model ensures a non-uniform radial pressure distribution and a circumferential velocity component. On the other hand, the Q2D model assumes a uniform cross-section pressure and neglects the circumferential velocities, since axial variations are considerably higher. Mitra e Rouleau [38] noted that this approximation is less accurate close to a rapidly closing valve and suggested that several pipe diameters are required downstream of the disturbance before an axisymmetric model can provide meaningful results. Consistent with their observations, the 3D-CFD velocity fields at different valve closure percentages reported by Ferreira et al. [22] for this particular system clearly shows a non-axisymmetric velocity profile close to the valve and a high radial velocity component.

The 3D-CFD model provides a more realistic representation of the transient event generated by the instantaneous valve closure, as the Q2D model neglects key flow features near the valve and boundary conditions. Nevertheless, once 3D boundary effects become negligible, the Q2D model correctly describes the unsteady energy dissipation.

5. Q2D and 1D Model Results

Five unsteady friction models were implemented in the 1D solver, namely four CIB models and one IAB model. The results are presented in the following sections. First, the 1D model results obtained for an instantaneous valve manoeuvre are compared with those obtained using the Q2D model (Figures 8 and 9). Second, the 1D model results with the calibrated valve manoeuvre are compared with Q2D and 3D-CFD models (Figure 10). For clarity in the results analysis, the 3D-CFD results are only included in the second comparison.

5.1. Instantaneous Valve Manoeuvre

The Q2D and the 1D models' piezometric head time history are presented in Figure 8: the quasi-steady (classical) approach (Figure 8a); the full Zielke [10] and the approximate Trikha [20] CIB models for laminar flow (Figure 8b); the full Vardy and Brown [1] and the approximate Vardy and Brown [11] CIB models for smooth–turbulent flow (Figure 8c); and the Vitkosky et al. [9] IAB model (Figure 8d).

The classic water hammer model with the quasi-steady friction formulation cannot describe the actual damping induced by the unsteady wall shear stress (Figure 8a). Consequently, its applicability is limited to the first transient wave cycle, or, in theory, to situations involving slow variations of the mean velocity, which is not the case. This limitation arises from the fact that this model does not account for the frequency-dependent radial velocity gradient, neglecting the high-velocity gradient after the pressure surge arrival. Thus, the damping observed over the five periods is almost non-existent, and the predicted pressure wave retains an unrealistically square-shaped profile, instead of the smooth and rounded S-shape behaviour exhibited by real systems and accurately captured by the Q2D and 3D-CFD models.

Zielke's model can closely describe the transient pressure wave amplitude, phase and shape, matching the accuracy of the Q2D model (Figure 8b), despite the high computation effort required to compute all terms of the convolution integral. Both models rely on the same axisymmetric flow and frozen viscosity assumptions but differ on their steady-state viscosity profiles. Zielke's model adopts a uniform molecular viscosity profile, whereas the Q2D model implements a five-layer turbulent viscosity profile, which is more realistic. Results are comparable over the five-wave pressure simulation periods because the viscous layer dominates during the wall shear stress computation.

Trikha's approximate model, although requiring significantly less computational time, does not ensure the same simulation accuracy as Zielke's model (Figure 8b). While it reproduces a comparable overall wave damping, it fails to capture energy dissipation immediately after the wavefront arrival, resulting in insufficient smoothing of the pressure wave. This behaviour reflects the well-documented limitations of the original Trikha implementation, as noted by Kagawa et al. [39], School [40], Vardy and Brown [21] and Vitkovsky et al. [9]. Nevertheless, it provides a major reduction in the simulation time in comparison with the direct implementation of the full weighting functions.

For smooth–turbulent flows (Figure 8c), both the full CIB model and the nine-term approximate CIB model closely match the Q2D model results. The five-term approximate model exhibits a slight phase shift during the high dissipation periods after the third wave. The reduction in the number of exponential terms diminishes the frequency range over which the true weighting function is accurately represented, which leads to a loss of accuracy during intervals of high dissipation, where the wall shear stress peaks are most pronounced.

Vitkovsky's model maintains the square shape of the pressure wave observed in the quasi-steady friction approach (Figure 8a,d). Nevertheless, by using a calibrated decay coefficient, k_3 , the model can reproduce the overall wave damping. Even so, all previous weighting models ensure a more realistic description.

The previous pressure histories are not a direct measurement of the frequency-dependent energy dissipation; rather, they reflect each model wall shear stress simulation, assuming the same steady-state conditions, as referred by Abreu and Almeida [24]. In this context, wall shear stress offers the most reliable basis for evaluating model performance, particularly when the piezometric head predictions are very similar, as is the case here. Figure 9 presents the absolute wall shear stress time histories for the unsteady friction mod-

els previously analysed. Figure 9a shows the Q2D reference solution, and the subsequent figures compare it with the predictions from the other UF formulations.

The piezometric time history results obtained by the Q2D and full CIB formulations for both laminar and smooth–turbulent flows are very similar. This is justified by identical wall shear stress simulations (Figure 9b,c). The differences are limited to the initial peak, and after four time periods, the results of all three models converge. The three models also predicted identical residual wall shear stress values (i.e., the values immediately prior to the arrival of the next pressure wave).

The incorrect pressure wave shape produced by Trikha’s model (Figure 9b) is a direct consequence of its inability to describe the wall shear stress peak and how it decays following each pressure surge (Figure 9b). Instead of exhibiting the exponential decay characteristic of more accurate unsteady-friction models, Trikha’s formulation produces an almost linear decline of the wall shear stress value.

The approximate CIB model for smooth turbulent flow correctly represents the wall shear stress decay (Figure 9c). However, it underestimates the peak value compared with the full CIB and the Q2D models. The number of exponential terms considered in the approximating of the weighting function strongly affects the correct description immediately after each wave arrival. For the nine- and five-term approximations, the wall shear stress first peak ($t/T = 0.12$) is one-half and one-fifth of the Q2D value, respectively. These differences diminish after the second wave cycle. Crucially, unlike Trikha’s model, these approximate CIB formulations still capture the correct exponential decay following each pressure surge.

Vitkovsky’s model (Figure 9d), after the calibration of the decay coefficient ($k_3 = 0.08$), shows a sequence of high and non-continuous wall shear stress values associated with the instantaneous mean velocity changes, decreasing as the pressure wavefront becomes rounder. However, it does not represent the wall shear stress gradual decay observed in the Q2D and 1D CIB models. This result is the direct consequence of the model phenomenon simplification, particularly inadequate for an instantaneous valve closure. In this formulation, the correct overall pressure wave damping is ensured solely by the k_3 calibration and not by the model correct description of the phenomenon.

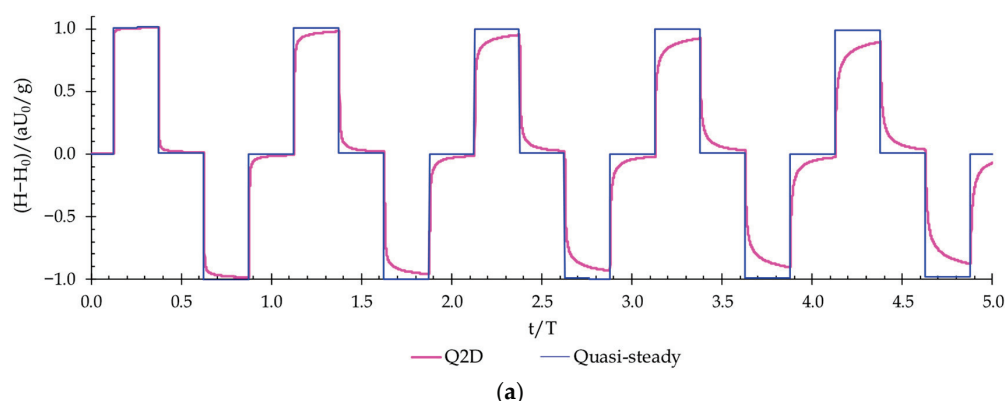


Figure 8. *Cont.*

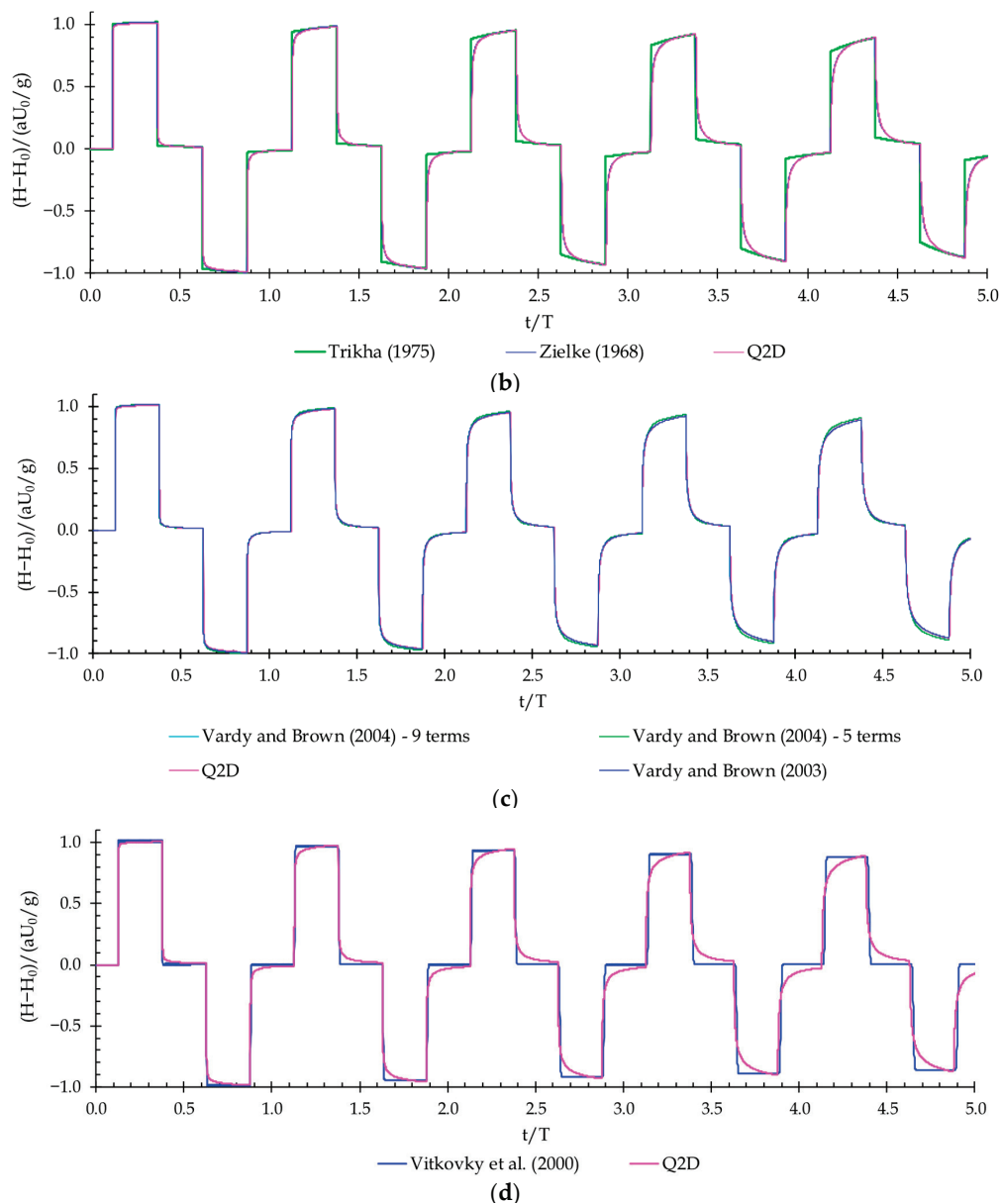


Figure 8. Q2D versus 1D model results in terms of adimensional piezometric head at the pipe mid-section: (a) Q2D; (b) Q2D versus laminar flow CIB model [10,20]; (c) Q2D versus smooth flow CIB model [1,11]; and (d) Q2D versus IAB model [9].

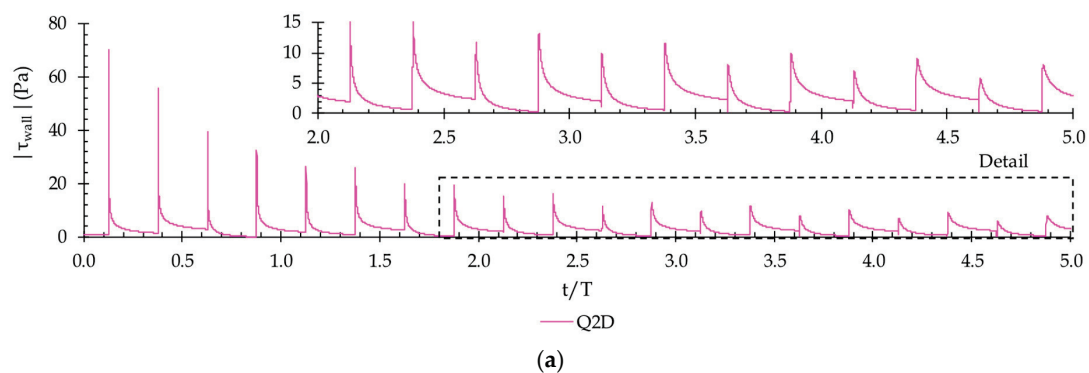


Figure 9. Cont.

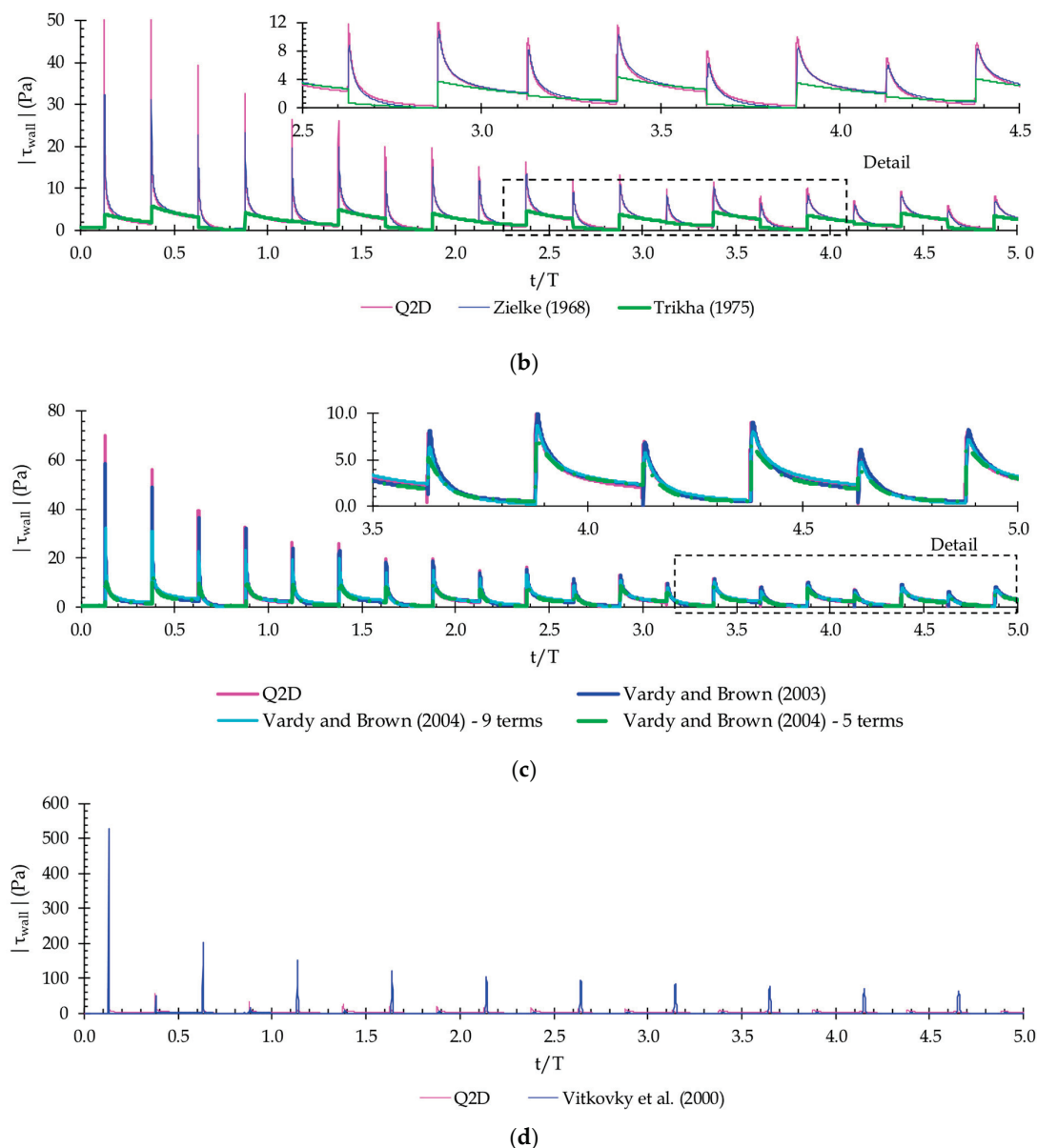


Figure 9. Q2D versus 1D model results wall shear stress at the pipe mid-section: (a) Q2D; (b) Q2D versus laminar flow CIB model [10,20]; (c) Q2D versus smooth–turbulent flow CIB model [1,11]; and (d) Q2D versus IAB model [9].

5.2. Calibrated Valve Manoeuvre

The results obtained using the calibrated valve manoeuvre with the 1D and Q2D models are compared with those obtained from the 3D-CFD simulation (Figure 5). Figure 10 shows the absolute wall shear stress simulation for all models during the first two wave periods since the beginning of the transient, due to the high computational effort associated with the 3D-CFD model, as previously explained. Figure 10a shows the Q2D and the 3D-CFD models' simulations for the calibrated valve manoeuvre, whilst Figure 10b shows the laminar CIB model predictions together with those from the 3D-CFD model. As previously observed for the instantaneous valve closure, the full laminar CIB model produces results indistinguishable from those of the Q2D model. The same behaviour is found for the full turbulent CIB formulation, which again closely matches the Q2D predictions and, consequently, the 3D-CFD results (Figure 10c).

Trikha's model (the laminar approximate formulation) continues to produce a nearly linear decay of wall shear stress (Figure 10b). However, Trikha's model results closely approach the Q2D and 3D-CFD solutions because the calibrated valve manoeuvre imposes a smoother pressure variation, reducing both the peak magnitude and the subsequent exponential decay. This feature also explains the improvement obtained with the approximate smooth–turbulent flow models. With five and nine exponential terms, these approximations yield almost identical results and follow the reference solutions closely.

Vitkovsky's model (Figure 10d) considerably reduces the peak values compared with the instantaneous valve closure outcome (Figure 9d) due to the gradual pressure variation at the valve. Although fitting the decay coefficient, k_3 , allows the model to reproduce the decay of the pressure wave amplitude, it continues to fail to capture the pressure-wave shape, which is accurately described by the 1D CIB, Q2D and 3D-CFD models.

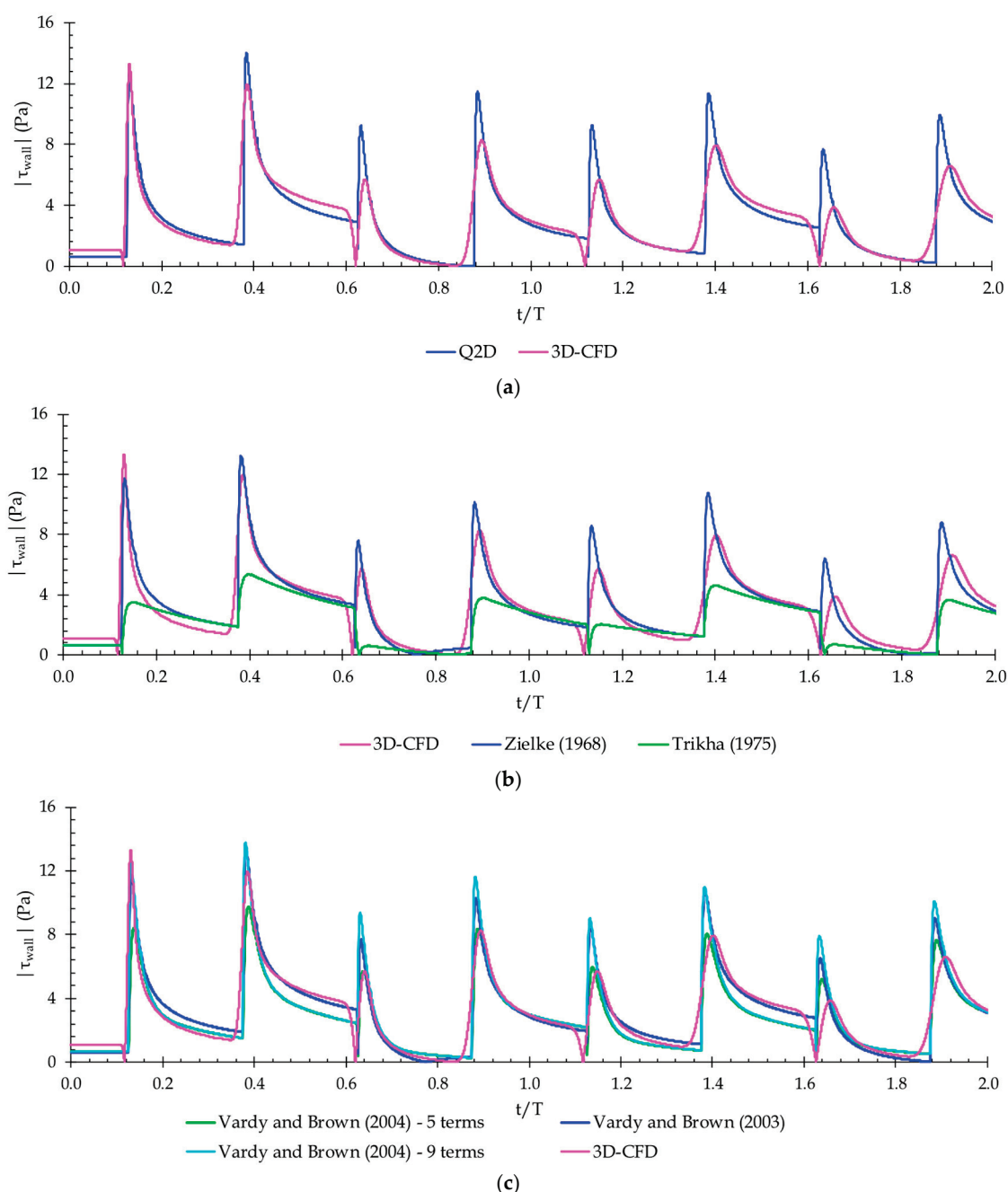


Figure 10. Cont.

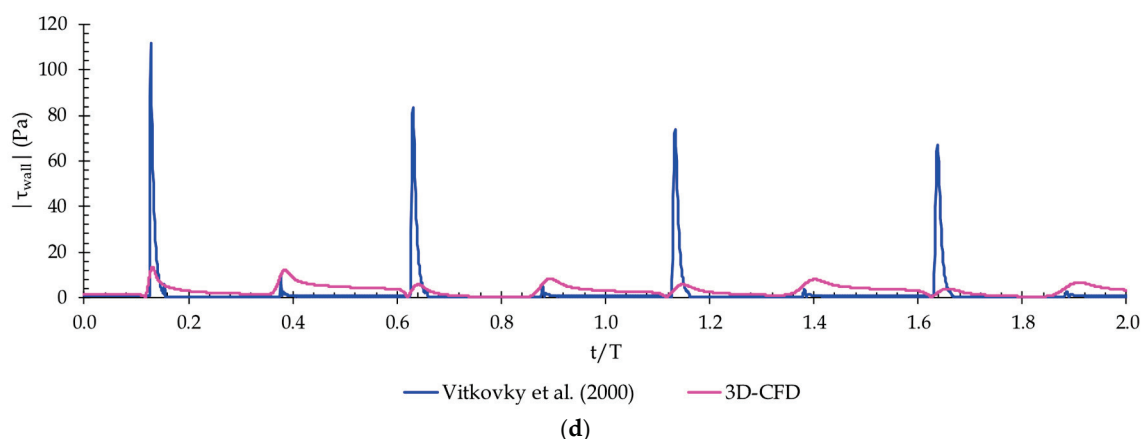


Figure 10. Wall shear stress time history at the pipe midsection for the calibrated valve manoeuvre: (a) Q2D versus 3D model; (b) Q2D versus laminar CIB models [10,20]; (c) Q2D versus turbulent CIB models [1,11]; and (d) Q2D versus IAB model [9].

6. Q2D Model Results vs. Experimental Data

Experimental data for an effective valve closing time of 0.01 s is used herein for comparison with 1D and Q2D models' results. The experimental valve closure manoeuvre is calibrated for all models and is ten times slower than the valve manoeuvre used in the previous comparison (Figure 5).

Figure 11 presents the collected piezometric-head data and the Q2D model simulation results. The CIB models' piezometric head results are not represented, as all models closely match the Q2D results and no significant differences are observed. Increasing the valve closure time reduces wall shear stress peaks and improves the agreement between the full and approximate CIB models. Trikha's model provides a poorer description during the high dissipation period, although the piezometric head results remain very close. The IAB model continues to reproduce wave damping but with a rectangular pressure-wave shape.

The experimental data corroborates the almost symmetric pressure wave shape previously represented by the 3D-CFD model (Figure 11a,b). The extra wave rounding is immediately observed after the first wave reflection in the reservoir (Figure 11c at $t/T = 0.65$). This feature is not described by the Q2D model that closely maintains the valve closure S-shape. The difference between experimental data and Q2D model results increases with the simulation time (compare Figure 11c at $t/T = 0.65$ and Figure 11d at $t/T = 13.4$), as also observed in the comparison with the 3D-CFD results. The Q2D valve calibration improves the initial agreement with the experimental data, but deviations persist due to simplified boundary condition representation.

The wall shear stress for the approximate CIB models and the Q2D model are presented in Figure 12. The full CIB models' results perfectly overlap with those from the Q2D model and, for simplification reasons, are not presented herein. The valve slower closure considerably reduces the τ_w peak values. The Q2D first pressure peak decreases from 72 Pa in the theoretical instantaneous manoeuvre calculations (Figure 9a), to 13.5 Pa in the 3D-CFD calibrated manoeuvre (Figure 10a) and to 5.2 Pa in the experimental data (Figure 12). This peak reduction substantially lowers the required numerical accuracy and brings the approximate CIB model results closer to those of the Q2D model, as evidenced by the good agreement among all models in Figure 12.

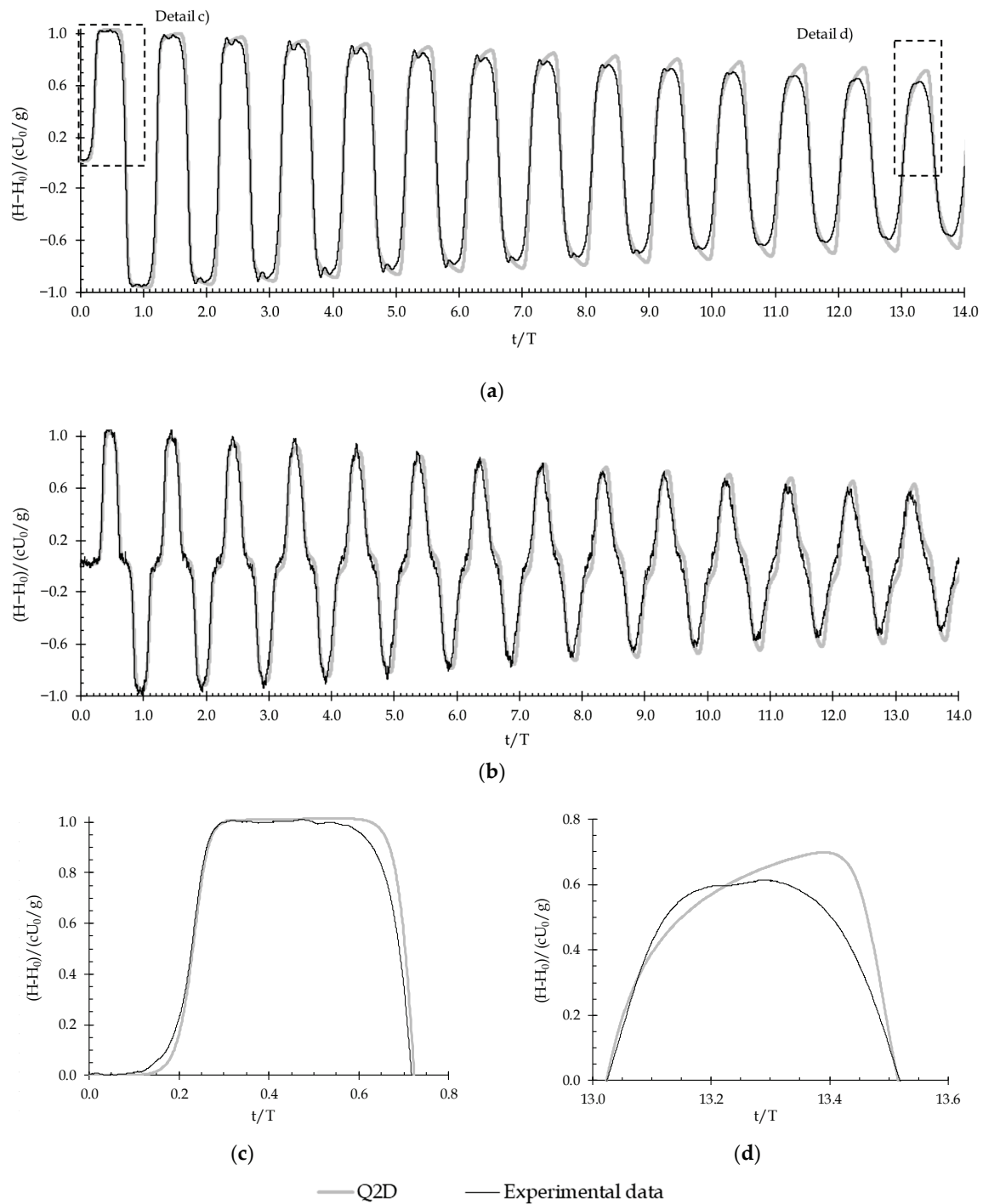


Figure 11. Piezometric head time history for the experimental data and Q2D results. (a) Valve section, $0 \leq t/T \leq 14$; (b) pipe midsection section, $0 \leq t/T \leq 14$; (c) valve section, $0 \leq t/T \leq 0.8$; (d) valve section, $13.0 \leq t/T \leq 13.6$.

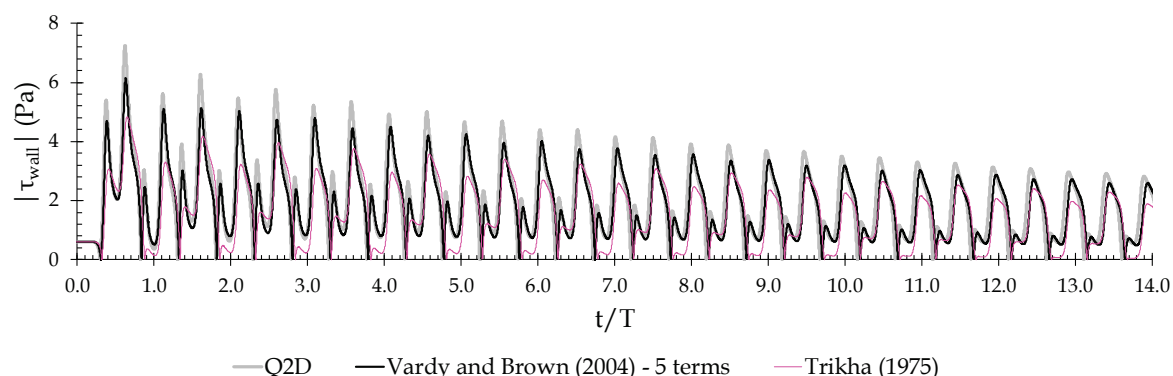


Figure 12. Wall shear stress time history at the pipe midsection for the experimental valve manoeuvre with the Q2D [11,20].

7. Conclusions

The Q2D model results are compared with those obtained from the 3D-CFD and 1D numerical models for an instantaneous valve closure, while the 1D and Q2D predictions are also evaluated with the valve manoeuvre observed in the laboratory tests.

The Q2D model accurately describes unsteady energy dissipation when fully developed flow is ensured. Its results are comparable to those of the 3D-CFD model, while requiring significantly less computational effort. However, it fails to correctly simulate instantaneous valve closure, as captured by the 3D-CFD model. This limitation directly results from the Q2D model assumptions of radially uniform pressure and negligible circumferential velocities, which are not valid near the valve or immediately after its closure. Consequently, the Q2D model does not predict a symmetric pressure wave and emphasises the high energy dissipation periods after each pressure surge arrival. This comparison shows the importance of developing specific boundary conditions to improve the Q2D model accuracy. Nevertheless, it emphasises that the 3D-CFD benefit is associated with the simulations near the boundary and during the valve closure, rather than with unsteady energy calculation.

The same assumptions are made in the convolutional 1D unsteady models, which explain why the results for wall shear stress and piezometric head obtained by the Q2D model and the full CIB 1D models are almost identical. The latter requires more computing power. Nevertheless, the approximate CIB 1D models cannot provide an accurate description of the transient event following the arrival of the pressure wave.

The laboratory valve closure represents a fast yet realistic closure of the valve, and no significant differences are observed between the Q2D and CIB models. Higher valve closure times reduce the wall shear stress peak value and approximate the results of the Q2D models (and full CIB models) with the approximate CIB models. The experimental data corroborates the almost symmetric pressure wave shape (previously represented by the 3D-CFD results) and the extra wave rounding observed in the first wave, not described by 1D and Q2D models.

Future efforts should focus on conducting further sensitivity analyses on the effect of key parameters on the Q2D model (e.g., pipe diameter, length, steady-state mean velocity) as well as on reducing the computational time of the Q2D model to make it more competitive with the 1D models. One possibility to explore for reducing computational effect is adapting the mesh geometry during the transient event according to the axial velocity gradient. A dynamic mesh that reconfigures as the axial velocity gradient decreases would reduce computational time. Secondly, the existing unsteady model accurately simulates unsteady friction for real valve closure manoeuvres, so efforts should focus on improving these models' boundary description.

Author Contributions: Conceptualisation, P.F. and D.C.; methodology, P.F. and D.C.; software, P.F.; validation, P.F.; formal analysis, P.F.; investigation, P.F. and D.C.; resources, P.F. and D.C.; data curation, D.C.; writing—original draft preparation, P.F.; writing—review and editing, P.F. and D.C.; visualisation, P.F.; supervision, D.C.; project administration, D.C.; funding acquisition, D.C. All authors have read and agreed to the published version of the manuscript.

Funding: This research was partially funded by the Fundação para a Ciência e a Tecnologia, I.P. (FCT, <https://ror.org/00snfqh58>, accessed on 3 January 2026) under Grant UID/6438/2025 (<https://doi.org/10.54499/UID/06438/2025>) of the research unit CERIS. For the purpose of Open Access, the author has applied a CC-BY public copyright license to any Author's Accepted Manuscript (AAM) version arising from this submission.

Data Availability Statement: The original contributions presented in this study are included in the article. Further inquiries can be directed to the corresponding author.

Acknowledgments: The authors would like to acknowledge João Paulo Ferreira and Nuno Martins for assisting the data collection in the experimental facility and for providing the 3D-CFD data.

Conflicts of Interest: The authors declare no conflicts of interest.

Abbreviations and Nomenclature

The following abbreviations and nomenclatures are used in this manuscript:

1D	One-dimensional
2D	Two-dimensional
3D	Three-dimensional
CFD	Computation fluid dynamics
CIB	Convolution integral-based method
Q2D	Quasi-2D axisymmetric model
GS	Geometric sequence cylinder mesh
IAB	Instantaneous acceleration-based model
UF	Unsteady friction
C_R	geometric sequence common ratio (-)
c	pressure wave speed (m/s)
D	pipe inner diameter (m)
H	piezometric head (m)
g	acceleration due to gravity (m/s^2)
L	pipe length (m)
Q	flow rate or discharge (m^3/s)
N_C	total number of cylinders of the radial mesh (-)
Re	Reynolds number (-)
r	distance from the axis in the radial direction (m)
t	time (s)
T	pressure wave period ($T = 4L/c$) (s)
U	mean velocity of the fluid in the pipe cross-section (m/s)
W	weighting function (-)
Δr_j	wall thickness of the j th cylinder ($\Delta r_j = r_j - r_{j-1}$)
u	axial velocity (m/s)
u'	turbulence velocity fluctuation on the axial direction (m/s)
v	radial velocity (m/s)
v'	turbulence velocity fluctuation on the radial direction (m/s)
x	distance along the pipe (m)
ν	kinematic viscosity of liquid (m^2/s)

y	distance from the pipe wall, measured in the radial direction (m)
Δt	time interval (s)
κ	Von Kármán constant (-)
λ	bulk viscosity ($\lambda \approx 2/3\mu$) (Pa.s)
ν	kinematic viscosity (m^2/s)
ε	eddy viscosity (m^2/s)
μ	dynamic viscosity (Pa.s)
τ_w	total wall shear stress (Pa)
τ_{wu}	unsteady wall shear stress (Pa)
τ_{ws}	steady wall shear stress (Pa)
ρ	liquid density (kg/m^3)

References

1. Vardy, A.E.; Brown, J.M.B. Transient turbulent friction in smooth pipe flows. *J. Sound Vib.* **2003**, *259*, 1011–1036. [CrossRef]
2. Covas, D.; Stoianov, I.; Ramos, H.; Graham, N.; Maksimovic, C. The dynamic effect of pipe-wall viscoelasticity in hydraulic transients. Part 1: Experimental analysis and creep characterization. *J. Hydraul. Res.* **2004**, *42*, 517–532. [CrossRef]
3. Ferras, F.; Manso, A.; Schleiss, A.; Covas, I.C.D. Experimental distinction of damping mechanisms during hydraulic transients in pipe flow. *J. Fluids Struct.* **2016**, *66*, 424–446. [CrossRef]
4. Urbanowicz, K.; Bergant, A.; Stosiak, M.; Karpenko, M.; Bogdevičius, M. Developments in analytical wall shear stress modelling for water hammer phenomena. *J. Sound Vib.* **2023**, *562*, 117848. [CrossRef]
5. Urbanowicz, K.; Jing, H.; Bergant, A.; Stosiak, M.; Lubecki, M. Progress in analytical modeling of water hammer. *J. Fluids Eng.* **2023**, *145*, 081203. [CrossRef]
6. Ghidaoui, M.S.; Zhao, M.; McInnis, D.A.; Axworthy, D.H. A review of water hammer theory and practice. *Appl. Mech. Rev.* **2005**, *58*, 49–76. [CrossRef]
7. Brunone, B.; Golia, U.; Greco, M. Modelling of fast transients by numerical methods. In *International Conference on Hydraulic Transients with Water Column Separation, Proceedings of the 9th Round Table of the IAHR Group, Valencia, Spain, 4–6 September 1991*; IAHR: Madrid, Spain, 1991; pp. 273–280.
8. Brunone, B.; Golia, U.; Greco, M. Some remarks on the momentum equation for fast transients. In *International Conference on Hydraulic Transients with Water Column Separation, Proceedings of the 9th Round Table of the IAHR Group, Valencia, Spain, 4–6 September 1991*; IAHR: Madrid, Spain, 1991; pp. 201–209.
9. Vitkovsky, J.; Stephens, M.; Bergant, A.; Lambert, M.; Simpson, A. Efficient and accurate calculation of Vardy–Brown unsteady friction in pipe transients. In *Proceedings of the 9th International Conference on Pressure Surges*, Chester, UK, 24–26 March 2004.
10. Zielke, W. Frequency-dependent friction in transient pipe flow. *J. Basic Eng.* **1968**, *90*, 109–115. [CrossRef]
11. Vardy, A.E.; Brown, J.M.B. Transient turbulent friction in fully rough pipe flows. *J. Sound Vib.* **2004**, *270*, 233–257. [CrossRef]
12. Vardy, A.E.; Hwang, K.-L. A characteristic model of transients in pipes. *J. Hydraul. Res.* **1991**, *29*, 669–684. [CrossRef]
13. Zhao, M.; Ghidaoui, M. Investigation of turbulence behavior in pipe transients using a k – ε model. *J. Hydraul. Res.* **2006**, *44*, 682–692. [CrossRef]
14. Martins, N.M.C.; Carriço, N.J.G.; Ramos, H.M.; Covas, I.C.C. Velocity distribution in pressurized pipe flow using CFD: Accuracy and mesh analysis. *Comput. Fluids* **2014**, *105*, 218–230. [CrossRef]
15. Martins, N.; Brunone, B.; Meniconi, S.; Ramos, H.M.; Covas, D.I.C. CFD and 1D approaches for the unsteady friction analysis of low Reynolds number turbulent flows. *J. Hydraul. Eng.* **2017**, *143*, 04017050. [CrossRef]
16. Martins, N.; Brunone, B.; Meniconi, S.; Ramos, H.M.; Covas, D.I.C. Efficient computational fluid dynamics model for transient laminar flow modelling: Pressure wave propagation and velocity profile changes. *J. Fluids Eng.* **2018**, *140*, 011002. [CrossRef]
17. Vardy, A.E. Acceleration-dependent unsteady friction revisited. In *Proceedings of the 13th International Conference on Pressure Surges*, Bordeaux, France, 14–16 November 2018; BHR Group: Brittany, France, 2018; pp. 265–282.
18. Adamkowski, A.; Lewandowski, M. Experimental examination of unsteady friction models for transient pipe flow simulation. *J. Fluids Eng.* **2006**, *128*, 1351–1363. [CrossRef]
19. Ariyaratne, C.; He, S.; Vardy, A.E. Wall friction and turbulence dynamics in decelerating pipe flows. *J. Hydraul. Res.* **2010**, *48*, 810–821. [CrossRef]
20. Trikha, A. An efficient method for simulating frequency-dependent friction in transient liquid flow. *J. Fluids Eng.* **1975**, *97*, 97–105. [CrossRef]
21. Vardy, A.E.; Brown, J.M.B. Efficient approximation of unsteady friction weighting functions. *J. Hydraul. Eng.* **2004**, *130*, 1097–1107. [CrossRef]

22. Ferreira, J.; Martins, N. Ball valve behaviour under steady and unsteady conditions. *J. Hydraul. Eng.* **2018**, *144*, 04018005. [CrossRef]
23. Cebeci, T. *Analysis of Turbulent Flows with Computer Programs*, 3rd ed.; Elsevier: Oxford, UK, 2013.
24. Abreu, J.; Betâmio de Almeida, A. Timescale behavior of the wall shear stress in unsteady laminar pipe flows. *J. Hydraul. Eng.* **2009**, *135*, 415–424. [CrossRef]
25. Che, T.; Duan, H.; Lee, P.J.; Meniconi, S.; Pan, B.; Brunone, B. Radial pressure wave behavior in transient laminar pipe flows under different flow perturbations. *J. Fluids Eng.* **2018**, *140*, 101203. [CrossRef]
26. Zidouh, H. Velocity profiles and wall shear stress in turbulent transient pipe flow. *Int. J. Dyn. Fluids* **2009**, *5*, 61–83.
27. Martins, N.M.C.; Wahba, E.M. On the hierarchy of models for pipe transients: From quasi-two-dimensional water hammer models to full three-dimensional computational fluid dynamics models. *ASME J. Press. Vessel Technol.* **2022**, *144*, 021402. [CrossRef]
28. Urbanowicz, K.; Andrade, D.M.; Kubrak, M.; Kodura, A. Accuracy and limitations of 1D and Q2D water hammer models for laminar and turbulent flows. *J. Braz. Soc. Mech. Sci. Eng.* **2025**, *47*, 666. [CrossRef]
29. Ferreira, P.L.; Covas, D.I.C. New Optimized Equal-Area Mesh Used in Axisymmetric Models for Laminar Transient Flows. *Water* **2023**, *15*, 1402. [CrossRef]
30. Ferreira, P.M.L. *Axisymmetric Model for Pressurized Transient Flow: Efficient Meshes' Analysis*; Instituto Superior Técnico: Lisbon, Portugal, 2022.
31. Kita, Y.; Adachi, Y.; Hirose, K. Periodically oscillating turbulent flow in a pipe. *Bull. JSME* **1980**, *23*, 656–664. [CrossRef]
32. Bratland, O. Frequency-dependent friction and radial kinetic energy variation in transient pipe flow. In Proceedings of the 5th International Conference on Pressure Surges, Hanover, Germany, 22–24 September 1986; BHRA, The Fluid Engineering Centre: Bedford, UK, 1986.
33. Ferreira, P.L.; Covas, D.I.C. Mesh Sensitivity Analysis of Axisymmetric Models for Smooth–Turbulent Transient Flows. *Fluids* **2024**, *9*, 268. [CrossRef]
34. Vardy, A.E.; Brown, J.M.B. Transient, turbulent, smooth pipe friction. *J. Hydraul. Res.* **1995**, *33*, 435–456. [CrossRef]
35. Vardy, A.; Brown, J.; He, S.; Ariyaratne, C.; Gorji, S. Applicability of frozen-viscosity models of unsteady wall shear stress. *J. Hydraul. Eng.* **2015**, *141*, 04014073. [CrossRef]
36. He, S.; Jackson, J.D. A study of turbulence. A study of turbulence under conditions of transient flow in a pipe. *J. Fluid Mech.* **2000**, *408*, 1–38. [CrossRef]
37. Zhao, M.; Ghidaoui, M. Efficient quasi-two-dimensional model for water hammer problems. *J. Hydraul. Eng.* **2003**, *129*, 1007–1013. [CrossRef]
38. Mitra, A.; Rouleau, W. Radial and axial variations in transient pressure waves transmitted through liquid transmission lines. *J. Fluids Eng.* **1985**, *107*, 105–111. [CrossRef]
39. Kagawa, T.; Lee, I.; Kitagawa, A.; Takenaka, T. High-speed and accurate computing method of frequency-dependent friction in laminar pipe flow for the method of characteristics. *Trans. Jpn. Soc. Mech. Eng.* **1983**, *49*, 2638–2644. [CrossRef]
40. Schohl, G.A. Improved approximate method for simulating frequency-dependent friction in transient laminar flow. *J. Fluids Eng.* **1993**, *115*, 420–424. [CrossRef]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

Characterization of Non-Constant Flow in the Recession Process of Pressurized Pipelines with Air Valves

Shuaihui Sun ^{1,*}, Jinyang Ma ², Bo Zhang ³, Jingwen Jia ¹ and Jiuwang Li ¹

¹ School of Water Resources and Hydroelectric Engineering, Xi'an University of Technology, Xi'an 710048, China

² Shaanxi Hanjiang-to-Weihe River Water Diversion Project Construction Co., Ltd., Xi'an 710010, China

³ Powerchina Northwest Engineering Co., Ltd., Xi'an 710100, China

* Correspondence: shs@xaut.edu.cn

Abstract: Emptying pressure pipelines is a routine operation during pipeline maintenance. This study investigates the emptying characteristics of pressurized pipelines with air valves under unsteady flow conditions. A mathematical model for the emptying process is developed using the rigid water column theory, exploring the influence of drain valve opening, initial air pocket length, and valve opening patterns on the transient flow behavior. The results indicate that, compared with the linear valve opening pattern, a nonlinear power function opening increases the minimum air pocket pressure head by 0.1014 m and delays its occurrence by 0.655 s. The maximum emptying velocity rises by 0.48 m/s when the opening is increased from 10% to 30%, thereby shortening the emptying time by 65.4%. However, the pressure head inside the air pocket decreases accordingly. When the air valve diameter is enlarged from 0.003 mm to 0.008 mm, the pressure recovery time is markedly reduced and the initial pressure fluctuations are attenuated. Numerical simulations based on the Heihe emptying case demonstrate that a well-planned layout of multiple air valves effectively shortens the duration of negative pressure heads. Replacing the first air valve with a 50 cm diameter circular orifice significantly raises the minimum pressure head of the pipeline and dramatically enhances the stability of emptying pressurized pipeline.

Keywords: pressurized pipeline; drainage process; air valve; rigid water column model; non-constant flow

1. Introduction

Long-distance water transfer projects constitute a direct and effective means of mitigating the spatio-temporal imbalance of water resources and alleviating urban water shortages. Thanks to their high efficiency and low leakage rates, pressurized pipelines have been widely deployed in such projects as well as in municipal distribution networks. Accelerated urbanization and growing worldwide demand for hydraulic infrastructure have driven these conveyance systems towards ever-larger scales and increasingly complex configurations, rendering the transient phenomena encountered during pipeline emptying operations more intricate and variable. During emptying, the failure of air valves, overly rapid opening of drain valves, or excessive emptying velocities can readily induce sub-atmospheric pressures and severe pressure surges, potentially triggering catastrophic events such as pipeline collapse [1]. Consequently, an in-depth investigation of transient flow in the emptying of pressurized pipelines is of paramount importance.

Compared with the emptying process, the transient behavior during the filling of water transmission pipelines has hitherto received more extensive attention in the literature [2,3]. Currently, studies on the draining process of pressurized pipelines are primarily based on the filling process [4], and the specification of parameters such as draining velocity and the degree of drain valve opening during draining also draws from pipeline filling procedures. Zhang et al. [5] compared several rapid empty-pipe filling schemes, analyzing the impact of filling velocity on the hydraulic characteristics of hump pipes with air valves, and concluded that filling schemes for long-distance pipeline systems with velocities not exceeding 0.3 m/s are overly conservative and inefficient. Zhou et al. [6] developed a numerical simulation model for rapid pipeline filling with multiple entrapped air pockets to investigate the hydraulics-pneumatics transient flow during water conveyance system filling. Liang et al. [7] investigated the filling process under different pipeline inclination angles, analyzing variations in flow velocity, pressure, and gas-liquid two-phase distribution. Their findings indicate that when the pipeline inclination is less than 30° , appropriately increasing the filling velocity can effectively shorten the filling cycle. Ling et al. [8] conducted a numerical analysis of the rapid filling process in bypass pipelines based on an elastic model combined with the polytropic gas law, identifying the rate of air expulsion as the key factor affecting filling efficiency. Yan et al. [9] utilized Fluent for three-dimensional numerical simulation of the filling process in pressurized pipelines, finding that high-flow-rate filling causes intense pressure variations in trapped gas and recommending the adoption of a “low-flow-rate filling principle”. Lin et al. [10] employed a three-dimensional computational fluid dynamics model to study the evolution of the air-water interface in horizontal pipes during filling, observing that residual air can induce severe pressure oscillations leading to violent fluctuations at the interface. Collectively, research on the filling process of pressurized pipelines has revealed the influence of pipeline characteristics, trapped air pockets, filling velocity, air valves, and other factors on unsteady flow during filling, providing a theoretical foundation for studying the draining process.

The draining of pressurized pipelines generally falls into two scenarios: one involves emptying the fluid for pipeline maintenance and repair, and the other requires diverting excess flow through a bypass pipeline or drain outlet when the pipeline flow rate is too high. Coronado-Hernández et al. [11] designed an experimental setup to study two typical draining scenarios, with air valves either failing or functioning normally. They complemented this with computational fluid dynamics (CFD) simulations to investigate the influence of trapped air pocket size, valve opening rate, and valve opening time on pressure variations within the pipe. Their results identified the amount of reverse airflow as the primary factor affecting draining stability. Paternina-Verona et al. [12] proposed a 3D model to simulate the emptying process in a single pipeline, incorporating a PVOF model and a two-equation turbulence model. The model is used to capture more detailed flow characteristics—such as pressure and velocity patterns of discrete air pockets, contours of air and water velocity distributions, and temperature variations during air pocket expansion. Laanearu et al. [13] analyzed the movement of the air-water interface within pipes through experiments and simple numerical modeling. Experimental measurements of flow rate, water level changes, and pressure variations during draining were compared with results from a control volume model, revealing that the draining velocity is determined collectively by cavity pressure, siphon outflow, pipe friction resistance, and leakage at the pipe tail. Besharat et al. [14] proposed a computational model based on the rigid water column model to simulate transient phenomena in single pipes, validating it with experimental results. They concluded that the model could accurately predict experimental outcomes such as pressure variations and negative pressure values. Coronado-Hernández et al. [15]

developed a mathematical model to simulate the emptying process in water supply networks. They applied it to a case study, highlighting negative pressure as a critical cause of pipeline collapse. Tijsseling et al. [16] proposed an improved one-dimensional model for rapid pipe draining and filling. This model considered pressure, gravity, friction, the air content at the tail of the water column, and aeration at the front, enabling predictions of the water column's velocity, length, and position during rapid operations; however, experimental validation was not performed. Fuertes-Miquel et al. [17] reviewed various mathematical models for the water phase, air-water interface, and gas phase during the filling and draining of water pipelines, proposing that computational fluid mechanics offers a better explanation for cavitation dynamics during draining. Tasca et al. [18] developed a one-dimensional Method of Characteristics (MOC) transient flow model incorporating dynamic boundary conditions for air valves and evaluated the model with test data. The model's key control parameters include the air valve orifice area, the discharge coefficient, and the pipe friction factor. Bian et al. [19] applied a three-action air valve in tandem with a two-stage valve closure to a highly undulating gravity-flow pipeline, raising the minimum pressure head from -10 m to -1.19 m and thereby eliminating the risk of column-separation water hammer. Wu et al. [20] developed a novel staged-exhaust air valve that reduced the pressure peak of water hammer by 29.72% compared with an unprotected case, effectively damping pressure surges. Li et al. [21] provided a combined air-vessel and over-pressure relief-valve strategy that lowered the maximum transient pressure from 1.343 MPa to 1.087 MPa, while simultaneously reducing the air-vessel volume by 20% relative to stand-alone air-vessel protection.

In summary, current research on unsteady flow during pressurized pipeline emptying has primarily focused on analyzing the pressure within the air pocket, the air–water interface, and the evolution of the water level, as shown in Table 1. However, a systematic and quantitative investigation of how key boundary conditions—such as air-valve characteristics (diameter and number) and drain-valve opening—govern critical hydraulic parameters during emptying (e.g., transient discharge, negative-pressure extremes, and emptying efficiency) is still lacking. Consequently, this study develops a two-phase mathematical model of the emptying process for a pressurized pipeline, grounded in an actual engineering case. The study was conducted to quantitatively investigate the effects of the valve operation scheme, valve opening, initial empty segment length, air valve diameter, and the number of air valves on key performance metrics, including drainage velocity, cavity pressure head, and flow rate.

Table 1. Summary of relevant studies.

Research Source	Mathematical Model	Key Control Parameters	Valve Opening Law	Air Valve Arrangement
Coronado-Hernández and Fuertes-Miquel et al. [11,15,17]	Rigid water column model for single-pipe systems	Valve opening time, air valve orifice size, pipe slope	Linear opening	Single air valve
Tasca and Bian et al. [18,19]	One-dimensional transient flow model	Break-point opening, air valve orifice size	Two-stage linear closing	Ordinary, three-action or improved dual-orifice air valve
Wu et al. [20]	Coupled 3D CFD & MOC transient flow model	Buffer plate orifice size	Staged exhaust triggered by pressure difference	Staged exhaust air valve
Li et al. [21]	Combined air vessel–relief valve transient model (MOC)	Valve opening/closing law	Two-stage linear closing	Integrated air vessel & over-pressure relief valve

Unlike previous work that relied on piece-wise linear valve opening laws and suffered from abrupt velocity changes, the present study introduces a nonlinear power-law opening schedule. Furthermore, by innovatively coupling the optimisation of air-valve quantity and diameter, a “circular orifice + multi-valve coordination” layout is proposed. This approach enables the quantitative, system-level elucidation of how these specific boundary conditions affect the core hydraulic parameters of the emptying process, thereby providing more accurate theoretical guidance for optimizing emptying operations and ensuring both safety and efficiency.

2. Mathematical Model

2.1. Schematic Diagram of Emptying in Pressurized Pipelines

To analyze the emptying process of a pressurized pipeline, a geometric model of a single-pipe system equipped with air valves is established, as illustrated in Figure 1, for simulating the pipeline emptying procedure.

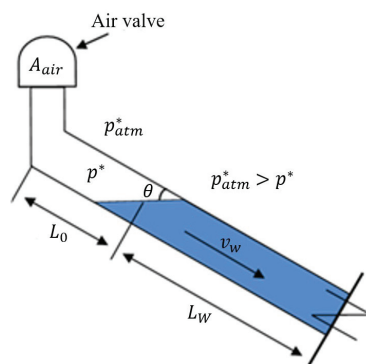


Figure 1. Schematic Diagram of Water Drainage in a Pressurized Pipeline.

2.2. Mathematical Model of the Emptying Process

In engineering practice, heat exchange between the fluid and the surroundings is generally neglected; hence, the model considers only the conservation of momentum and mass of the fluid motion. To simplify the analysis, the following basic assumptions are adopted [22]:

- (1) The wall shear stress acting on the water column is assumed constant and is evaluated using the Darcy–Weisbach equation;
- (2) The air–water interface is well-defined and remains parallel to the horizontal plane;
- (3) The characteristic time of the thermodynamic process within the air pocket during emptying is much shorter than the heat-exchange time scale, so the process is treated as adiabatic;
- (4) The pipeline possesses sufficient structural strength to withstand the sub-atmospheric pressures developed during emptying without significant deformation or collapse.

2.2.1. Liquid Phase Model

Based on the principles of momentum and mass conservation, the governing equation for the liquid phase during the emptying process is derived. Under the rigid water column assumption, the entire water body within the pipeline is considered the control volume, indicated by the blue color, with the x-coordinate aligned in the direction of flow. The corresponding free-body diagram is shown in Figure 1.

The force at the upstream end of the control volume is $p^* A \sin \theta$; the force at the downstream valve side is $-p_{atm}^* A \sin \theta$; the component of the gravity of the water body

along the x-axis is $L_w A \sin \theta^2 \rho_w g$; the force between the water body and the pipe wall is $-\tau \pi D L_w \sin \theta$; the local loss of fluid at the valve is $-h_f$. Applying the momentum conservation principle [23]:

$$p^* A \sin \theta - p_{atm}^* A \sin \theta + L_w A \sin \theta^2 \rho_w g - \tau \pi D L_w \sin \theta - h_f = A \sin \theta L_w \rho_w \frac{dv_w}{dt} \quad (1)$$

In the equation, p^* represents the absolute pressure of the gas in the cavity (Pa); p_{atm}^* represents the standard atmospheric pressure (Pa); A is the interfacial area (m^2); θ is the angle between the pipeline and the horizontal plane (rad); L_w is the length of the water column (m); ρ_w is the density of the fluid (kg/m^3); g is the acceleration due to gravity (m/s^2); D is the diameter of the pipeline (m); h_f is the local head loss of the fluid at the valve (m); v_w is the instantaneous average velocity of the fluid in the pipeline (m/s); τ is the shear stress (Pa), which is typically calculated using the Darcy-Weisbach equation [24]:

$$\tau = \frac{f \rho_w v_w |v_w|}{8} \quad (2)$$

$$\frac{1}{\sqrt{f}} = -2.0 \lg \left(\frac{\varepsilon}{3.7D} + \frac{2.51}{Re \sqrt{f}} \right) \quad (3)$$

In the equation, f represents the friction coefficient of the pipe wall, calculated using the Colebrook-White formula [25]; ε is the equivalent wall roughness (mm); D is the pipe diameter (mm); and Re is the Reynolds number.

By substituting Equation (2) into Equation (1), and dividing both sides of the equation by $A \sin \theta L_w \rho_w$, it can be simplified to Equation (4):

$$\frac{p^* - p_{atm}^*}{\rho_w L_w} + g \sin \theta - f \frac{v_w |v_w|}{2D} - \frac{h_f}{A \sin \theta L_w \rho_w} = \frac{dv_w}{dt} \quad (4)$$

From mass conservation, the water-column length satisfies [26]:

$$L_w = L - L_0 - \int v_w dt \quad (5)$$

where L represents the total length of the pipeline (m); L_0 represents the initial length of the air pocket (m).

Differentiating Equation (5) with respect to time gives the continuity differential equation:

$$\frac{dL_w}{dt} = -v_w \quad (6)$$

2.2.2. Gas Phase Model

During emptying, water is treated as an incompressible fluid; hence, over Δt the increase in the empty pipe-section volume equals the volume of water discharged. The air-pocket mass-conservation equation reads [12]:

$$\frac{dm_a}{dt} = \rho_{a0} v_a A_{a0} \quad (7)$$

where ρ_{a0} is the atmospheric density (kg/m^3), v_a is the velocity of the gas entering the air valve (m/s), A_{a0} is the inlet area of the air valve (m^2) and m_a is the mass of the gas in the empty section of the pipe (kg), which can be expressed as:

$$m_a = \rho_a A \sin \theta (L - L_w) \quad (8)$$

where ρ_a is the air-pocket density (kg/m^3).

Combining Equations (7) and (8) yield the rate of change of the air-pocket density:

$$\frac{d\rho_a}{dt} = \frac{\rho_{a0} v_a A_{a0} - \rho_a v_w A \sin \theta}{A \sin \theta (L - L_w)} \quad (9)$$

The air-pocket pressure evolution follows the ideal-gas law together with the expansion–compression equation [27–29]:

$$\frac{dp^*}{dt} = -k \frac{p^*}{V_a} \frac{dV_a}{dt} + k \frac{p^*}{m_a} \frac{dm_a}{dt} \quad (10)$$

where V_a is the air-pocket volume (m^3), and k is the polytropic exponent; for an adiabatic process, $k = 1.4$.

2.2.3. Air Valve Model

To describe the air inflow through the air valve, a dynamic air valve model is established under the following assumptions:

- (1) The air inflow is isentropic;
- (2) The air temperature inside the pipeline is assumed equal to the water temperature;
- (3) Variations in the air volume within the pipeline depend solely on the water outflow.

During emptying, the air pocket expands, and the absolute pressure inside the pipe falls below the ambient atmospheric pressure. Two flow regimes must therefore be considered [30,31]:

Sub-critical flow ($0.53p_{atm}^* \leq p^* \leq p_{atm}^*$):

$$\frac{dm_a}{dt} = C_{in} A_{a0} \sqrt{7p_{atm}^* \rho_{a0} \left[\left(\frac{p^*}{p_{atm}^*} \right)^{1.4286} - \left(\frac{p^*}{p_{atm}^*} \right)^{1.714} \right]} \quad (11)$$

Choked flow ($p^* < 0.53p_{atm}^*$):

$$\frac{dm_a}{dt} = C_{in} A_{a0} \frac{0.686}{\sqrt{RT}} p^* \quad (12)$$

where C_{in} is the air-inflow discharge coefficient; R is the gas constant ($\text{J}/(\text{kg} \cdot \text{K})$); T is the absolute temperature (K).

2.2.4. Drain Valve Operation Model

Linear opening: The valve opening varies piece-wise linearly with time t ,

$$a = \begin{cases} k_1 t, 0 \leq a < n \\ \dots\dots\dots \\ k_i t, n < a \leq m \end{cases} \quad (13)$$

where a is the valve opening (%); k_i is the slope of segment i ; t is the actuation time (s); n is the intermediate opening degree (%); m is the preset opening degree (%).

Nonlinear power-law opening: Based on the valve's head-loss characteristics, a 10 s power-law schedule is designed that opens slowly at the beginning and accelerates towards the end,

$$a = 0.3t^2 \quad (14)$$

From the manufacturer-supplied valve-opening–flow–head-loss curve, the local loss coefficient ζ at any opening is defined as [32]:

$$\zeta = \frac{2h_f g}{v^2} \quad (15)$$

where v is the fluid velocity at the valve (m/s).

2.3. Initial and Boundary Conditions

Initial Conditions: Before the commencement of water drainage, the initial drainage velocity is $v_{w(0)} = 0$, the initial water column length is $L_{W(0)} = L - L_0$ (where L_0 denotes the length of the vented pipe section), the initial absolute pressure in the vented section is $p^*(0) = 101,325$ Pa, the initial air density in the vented section is $\rho_{a(0)} = 1.205$ kg/m³, and the initial air intake velocity at the air valve is $v_{a(0)} = 0$.

Boundary Conditions: The upstream boundary condition is governed by the absolute pressure P^* within the vented pipe section. At the air valve and the outlet of the downstream water drainage valve, the flow is a free outflow to the atmosphere.

Using the specified initial and boundary conditions, the fourth-order Runge–Kutta method is employed to solve Equations (1)–(15) simultaneously for the time-dependent variations of the pressure in the vented pipe section, the water drainage velocity, the air intake mass flow rate at the air valve, and the water column length.

2.4. Model Verification

The accuracy of the present emptying model is evaluated against the experimental dataset reported by Coronado-Hernández et al. [11,15]. The test pipe is constructed of Methacrylate (Aminnth Pipe Systems, Dammam, Saudi Arabia). The pipeline geometry is characterized as follows: length 4.36 m, internal diameter 0.042 m, relative roughness 0.018, and inclination to the horizontal 30°. The ambient conditions were: local absolute atmospheric pressure 101.325 kPa, and absolute temperature 293.15 K.

The mathematical model presented in Section 2.2 is employed to simulate the emptying sequence of this pipeline, and the computed results are compared with the experimental measurements. The comparison, illustrated in Figure 2, shows that the predicted pressure oscillations within the air pocket closely agree with the experimental traces, with a maximum discrepancy of 3.59%. This confirms that the proposed model is capable of accurately reproducing the hydraulic behavior during the emptying process.

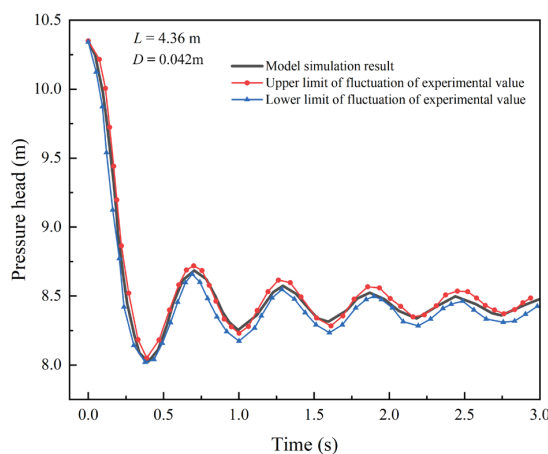


Figure 2. The comparison results between the simulation results and the experiments.

3. Analysis of Transient Flow in Pressurized Pipelines During Water Drainage

For the water drainage process under normal conditions, rapid air admission through the air valve is particularly critical. Efficient air replenishment enables faster balancing of the negative pressure generated during drainage, thereby ensuring safe operation. Table 2 presents the calculation cases for water drainage under regular air valve operation, investigating the influence of the initial vented pipe section length, valve opening degree, and air valve inlet orifice diameter on the drainage process, respectively.

Table 2. Calculation Cases for Water Drainage with an Air Valve.

Working Condition Number	Initial Length of the Emptying Column	Maximum Valve Opening	Diameter of the Air Inlet Hole of the Air Valve
Test 1	0.3 m	20%	0.005 m
Test 2	0.5 m	20%	0.005 m
Test 3	1.0 m	20%	0.005 m
Test 4	0.5 m	10%	0.005 m
Test 5	0.5 m	20%	0.005 m
Test 6	0.5 m	30%	0.005 m
Test 7	0.5 m	20%	0.003 m
Test 8	0.5 m	20%	0.005 m
Test 9	0.5 m	20%	0.008 m

3.1. Impact of the Water Drainage Valve Opening Strategy on the Drainage Process

The opening strategy of the water drainage valve plays a crucial role in ensuring the stable operation of the fluid within the pipeline. Figure 3 presents a comparison of the pressure head variations in the vented cavity under different opening strategies. As evident from the figure, compared to the instantaneous opening (0-s strategy), both the linear and nonlinear power-function opening strategies mitigate the sudden drop in pressure head observed in the no-strategy scenario. These strategies increase the minimum pressure head in the vented cavity during drainage by 0.5369 m and 0.6383 m, respectively, and delay the time of its occurrence. Furthermore, the partially enlarged view reveals that the nonlinear power-function strategy increases the minimum cavity pressure head by 0.1014 m and delays its occurrence time by 0.655 s compared to the linear strategy. However, it also increases the water drainage completion time by 3.371 s.

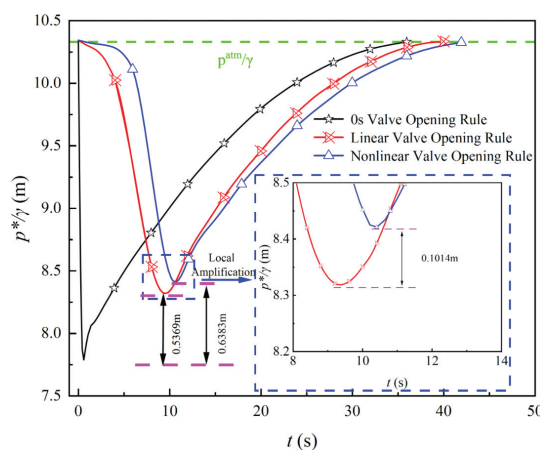


Figure 3. Influence of Different Valve Opening Strategies on Vented Cavity Pressure Head.

Figure 4 further compares the variation of the valve resistance coefficient versus drainage time for the linear and nonlinear power-function opening strategies. During the initial drainage phase, the resistance coefficient corresponding to the nonlinear power-function strategy at any given time is significantly higher than that of the linear approach. Consequently, the valve resistance is greater at corresponding time points, effectively restraining the excessive increase in drainage velocity during the early stage and enhancing the safety of the drainage process. Therefore, the nonlinear power-function opening strategy is adopted for all water drainage calculations in pressurized pipelines presented in this study.

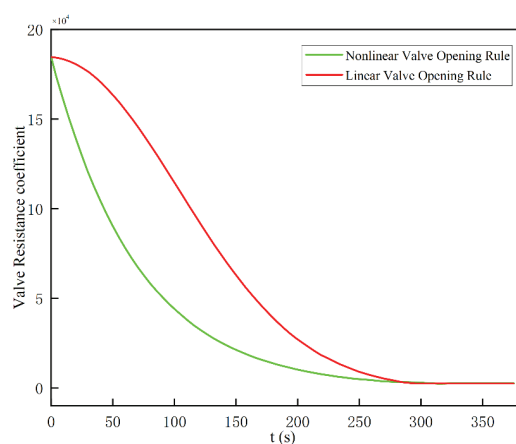


Figure 4. Changes in Resistance Coefficient under Various Valve Opening Strategies.

3.2. Impact of Different Boundary Conditions on Water Drainage Velocity

Figure 5 shows that the drainage flow velocity increases sharply during the initial drainage phase. This is attributed to unstable air intake through air valves and the gradual opening of the drainage valve at this stage. As the drainage process progresses, the drainage valve opening remains constant, and air intake stabilizes, resulting in a gradual decrease in flow velocity. The partially enlarged view indicates that a smaller initial cavity length corresponds to a higher peak drainage flow velocity, although the magnitude of change is relatively minor. Under ideal fluid conditions, where energy is conserved along streamlines, a longer water column results in a higher hydraulic head and, consequently, greater flow velocity.

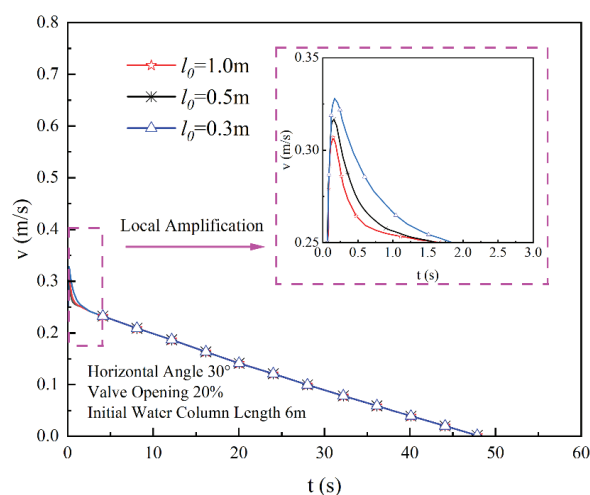


Figure 5. Effect of Cavity Length on Drainage Flow Velocity.

Figure 6 shows the influence of the opening degree of the backwater valve on the flow velocity of the backwater inside the pipeline. During the water withdrawal process, the maximum water withdrawal flow velocity corresponding to a 10% opening degree was 0.11 m/s. With the increase in the opening degree, the maximum water withdrawal flow velocities increased by 0.2 m/s and 0.48 m/s, respectively. Meanwhile, the water withdrawal times for a 20% opening degree and a 30% opening degree were shortened by 52 s and 72 s, respectively, compared to a 10% opening degree. However, the corresponding initial velocity fluctuations during water withdrawal increased accordingly. This is because when the valve opening increases, due to the relatively high water withdrawal speed, the air intake volume of the air valve is insufficient.

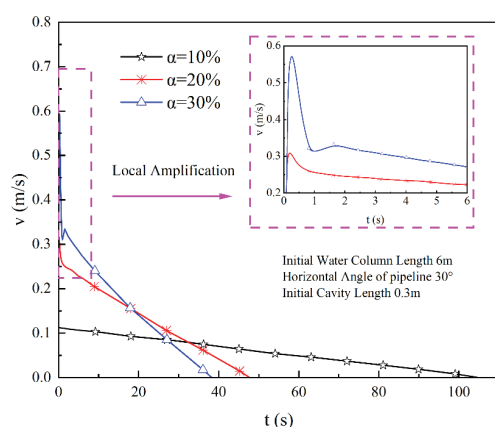


Figure 6. Effect of Drainage Valve Opening on Drainage Flow Velocity.

Figure 7 illustrates variations in drainage flow velocity with different air valve diameters. At an air valve diameter of 0.003 m, the required drainage time was 104.7 s. As the air intake orifice diameter increased, drainage time decreased by 54% and 63.3%, respectively. When air valve diameters measured 0.003 m and 0.005 m, significant flow velocity fluctuations were observed during the initial drainage phase, with more pronounced oscillations occurring at smaller intake areas. These fluctuations primarily resulted from limited air inflow through smaller orifices at the onset of drainage. Insufficient air replenishment reduced the minimum pressure head in the pipeline, consequently decreasing drainage velocity. As air intake gradually increased over time, the pressure head rose and drainage velocity increased again. This cyclical pattern persisted until the required air replenishment diminished, after which the drainage process progressively stabilized.

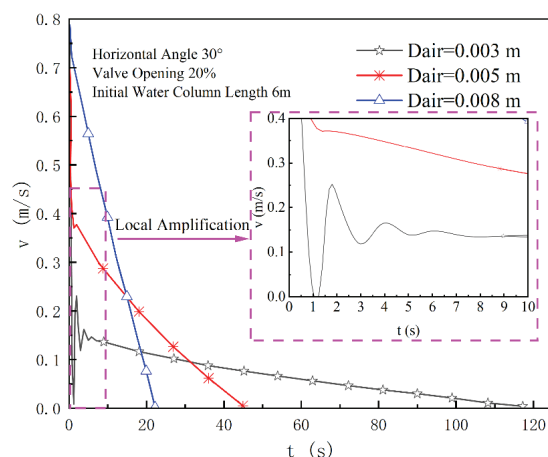


Figure 7. Effect of Air Valve Diameter on Drainage Flow Velocity.

3.3. The Influence of Different Boundary Conditions on Cavity Pressure

Figure 8 illustrates the temporal variation of pressure head under different initial cavity lengths. During the initial drainage phase, the cavity pressure head plunges to its minimum before gradually recovering as drainage progresses. Upon completion of drainage, the cavity pressure equilibrates with the local atmospheric pressure. As the cavity length increases, the minimum pipe pressure rises from 8.994 m to 9.065 m. This occurs because longer initial cavity lengths provide greater cavity volume available for expansion at drainage onset, thereby elevating the minimum internal pressure.

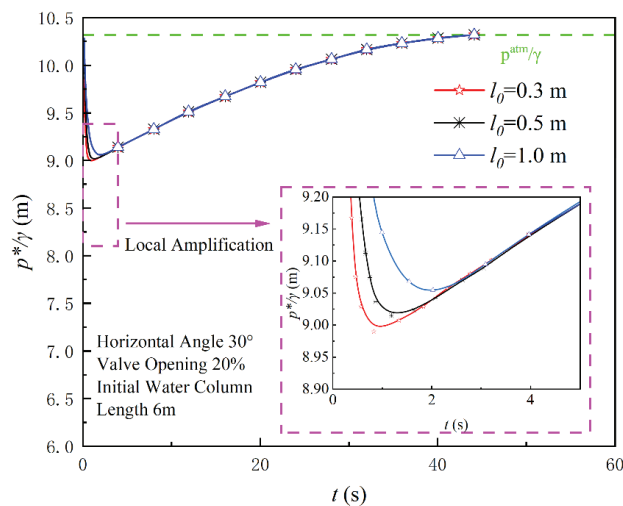


Figure 8. The influence of the initial length of the cavity on the pressure head.

Figure 9 reveals similar trends in cavity pressure head variation across three valve openings. At the onset of drainage, the pressure head plunges rapidly before gradually recovering to its initial state. The minimum pressure heads corresponding to 10%, 20%, and 30% openings measure 10.069 m, 9.019 m, and 7.783 m, respectively. As valve opening increases, the minimum pressure head during initial drainage decreases by 1.05 m and 2.286 m incrementally. However, at the 30% opening, the cavity pressure head recovers to its initial conditions 62.85% faster than at the 10% opening, despite a longer drainage duration.

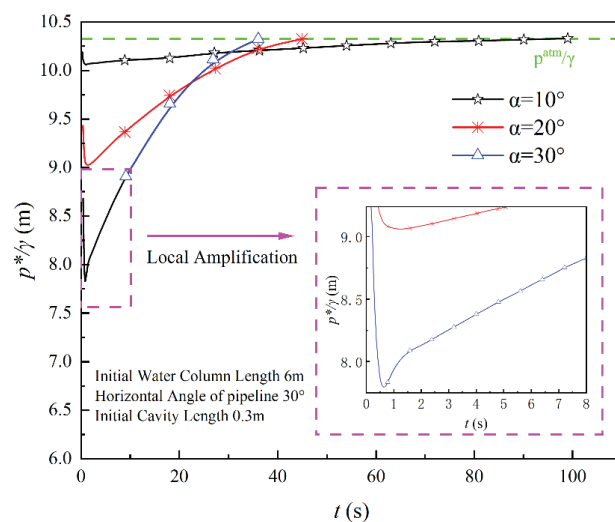


Figure 9. Effect of Drainage Valve Opening on Pressure Head.

Figure 10 demonstrates that the minimum cavity pressure heads for three air valve diameters measure 6.13 m, 7.05 m, and 8.61 m, respectively. Increasing the air valve diameter reduces the pressure recovery time to initial values by 69.3 s and 98.5 s incrementally. Notably, at smaller diameters (particularly 0.003 m), significant pressure head fluctuations occur during initial drainage due to insufficient air intake. These oscillations are substantially mitigated with larger air intake orifice diameters. Consequently, increasing the air valve diameter significantly enhances drainage stability in pressurized pipelines.

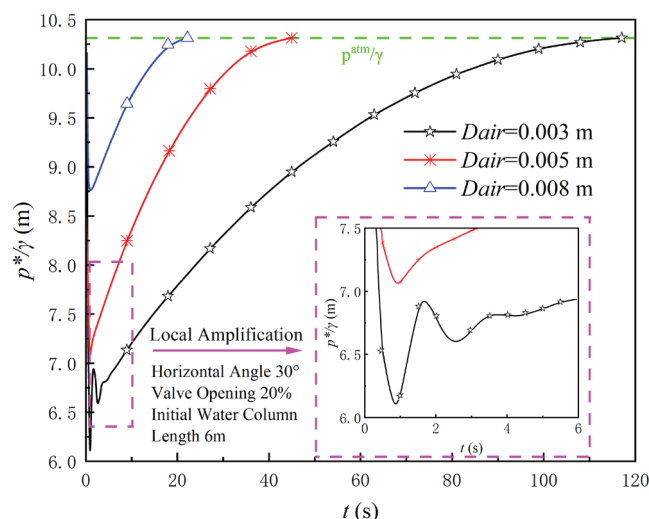


Figure 10. The influence of air valve diameter on the pressure water head.

In summary, during pressurized pipeline drainage, the nonlinear power-law valve opening elevates the minimum cavity pressure head and delays its occurrence compared to linear valve operation, while simultaneously suppressing excessive growth in flow velocity during the initial drainage phase. Altering the initial cavity length exerts a limited influence on hydraulic parameters throughout the drainage process. Increasing the drainage valve opening significantly enhances flow velocity and reduces drainage duration—for instance, at a 20% opening, velocity increases by 0.2 m/s, and drainage time decreases by 55.7% compared to a 10% opening. Furthermore, appropriately enlarging the air valve diameter (e.g., from 0.003 m to 0.005 m and 0.008 m) substantially improves drainage stability by mitigating pressure fluctuations induced by air deficiency. Collectively, these strategies effectively optimize the drainage process and accelerate the recovery of pressure head within the pipeline.

4. Case Study: Heihe River Drainage Section of the Hanjiang-to-Weihe River Diversion Project

The second-phase project of the Han-to-Wei Water Transfer Scheme constitutes a key component of the overall conveyance system. Located in Zhouzhi County, Xi'an City, Shaanxi Province, the project comprises a southern and a northern main line. The north line utilises pressurised pipelines; during routine inspection and maintenance, emptying operations are required. This case study focuses on the Heihe emptying reach, which is designed as a pair of parallel pipelines. Each pipeline has an internal diameter of 3.4 m and a total length of 3168.43 m, with a maximum elevation difference of 51.94 m. The longitudinal profile of the pipeline is illustrated in Figure 11. The pipe length, elevation difference, and $\sin\theta$ with respect to the horizontal for each pipe segment (I–V) are calculated and presented in Table 3.

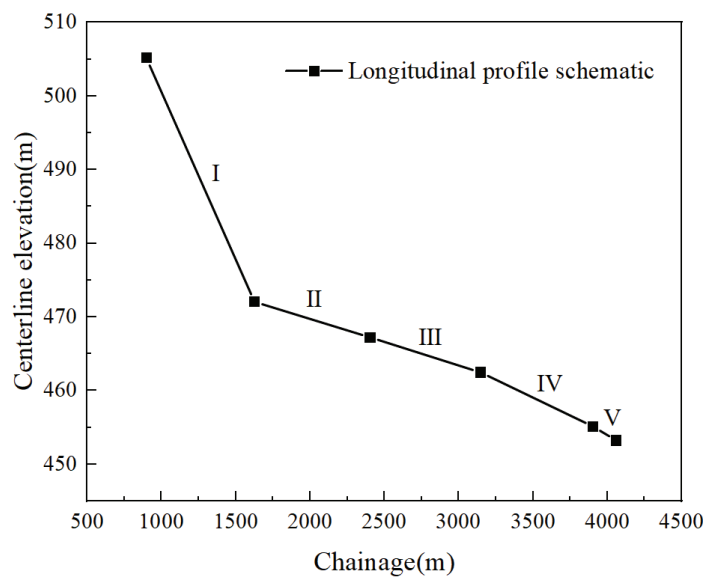


Figure 11. Longitudinal profile of the Heihe emptying pipeline.

Table 3. Pipeline parameters.

Pipe Section Number	I	II	III	IV	V
Pipe length (m)	732.12	777.07	749.98	750.00	159.26
Elevation difference (m)	33.11	4.83	4.75	7.34	1.91
$\sin\theta$	0.0452	0.0062	0.0063	0.0098	0.0120

The drainage pipeline was subdivided into five segments based on elevation differentials. As illustrated in Figure 12, four operational scenarios were designed for comparative analysis. Scenario 1 incorporates a single air valve at the pipeline inlet. Scenario 2 deploys air valves at chainages 0 + 862.12 and 1 + 468.69. Scenario 3 installs five air valves positioned at chainages 0 + 862.12, 1 + 468.89, 2 + 389.19, 3 + 139.17, and 3 + 889.17. Building on the findings in Section 3, which demonstrate that increasing the air valve diameter enhances flow velocity and minimizes cavity pressure, Scenario 4 modifies Scenario 3 by replacing the first air valve with a 0.5 m diameter circular orifice. All scenarios employ DN300 air valves (Bensv Valve Stock Co., Ltd., Tianjin, China) configured for gas exchange exclusively when the water–air interface descends below valve elevation.

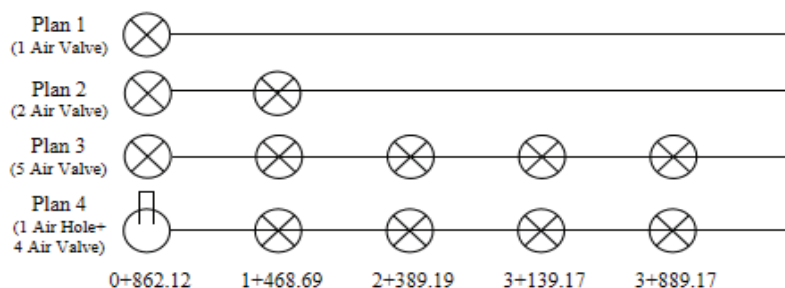


Figure 12. Research plan.

Initial Conditions: At time $t = 0$, the cavity pressure $p^*_{(0)} = 101,325$ Pa, the drainage water velocity $v_{w(0)} = 0$ m/s, the air density within the cavity $\rho_{w(0)} = 1.205$ kg/m³, and the air valve intake velocity $v_{air(0)} = 0$ m/s. The maximum preset opening of the drainage valve is specified as 12%, achieved through a linear valve opening process over 5 min. The

drainage flow rate must not exceed $3.63 \text{ m}^3/\text{s}$, and the air valve intake velocity must not exceed the critical velocity. Since the mathematical drainage model established in this study requires the cavity in the emptied pipe section to possess a certain initial length at time $t = 0$ to satisfy the computational prerequisites, an initial cavity length of 5 m is assumed at $t = 0$.

The variation of the cavity pressure water head of the Black River water return pipe section under different calculation schemes is shown in Figure 13. It can be seen from the figure that the models of the hollow air valves in Schemes 1 to 3 are the same, and the minimum cavity pressure water head is 9.97 m. The time at which the minimum value occurs is 299.1 s, when the water return valve is fully open. As the water return process proceeds, the cavity pressure water head gradually increases. Scheme 1 features a single air valve for air replenishment throughout the process. When the water withdrawal process reaches 1131.56 s, due to insufficient air replenishment, the rate of increase of the cavity pressure water head gradually slows down. At the same time, in Schemes 2 and 3, the second air valve begins to supply air, and the cavity pressure water head in the pipe section increases by 0.06 m over the next 40.3 s. Compared with Schemes 1 and 2, at 12,992.81 s, the pressure water head inside the pipe of Scheme 3 reaches 10.326 m, which is essentially close to the standard atmospheric pressure water head. The part of the empty pipe pressure water head that is lower than the standard atmospheric pressure water head is defined as the negative pressure water head. By comparing schemes 1, 2 and 3, it can be observed that the reasonable arrangement of multiple air valves along the drainage pipeline can significantly reduce the duration of negative pressure water head in the pipe; however, the minimum pressure water head in the pipe remains unchanged at the beginning of drainage. By comparing Scheme 4 with Scheme 3, the minimum pressure water head in the hollow cavity of Scheme 4 is 10.305 m, and the occurrence time is 345.2 s. Compared with the previous three schemes, it increases by 0.335 m, the occurrence time is 345.2 s, and the occurrence is delayed by 46.1 s. Meanwhile, the cavity pressure water head in Scheme 4 has recovered to the atmospheric pressure water head at 10,786.32 s.

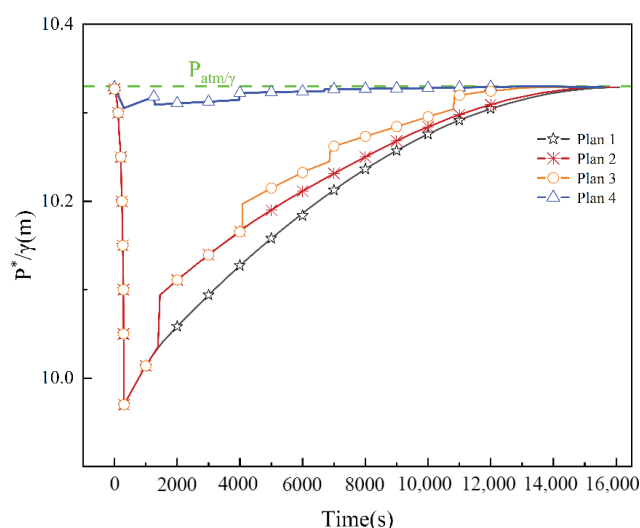


Figure 13. The variation of cavity pressure water head in different drainage schemes.

Figure 14 shows a comparative schematic of the drainage flow rates under different schemes. As the drainage time increases, the flow rates in all schemes exhibit a trend of initially increasing and then decreasing. Comparing the four schemes, the overall drainage process is relatively stable. The maximum drainage flow rate observed in Schemes 1–3 was

3.55 m³/s, which is 0.06 m³/s lower than that in Scheme 4. The drainage flow rates did not exceed the preset flow rate limit; however, oscillations in the flow rate occurred upon the opening of the air valves. Upon completion of the drainage process, Scheme 1 required 16,098.85 s (approximately 4.47 h), while Schemes 2–4 achieved completion 105.71 s, 273.5 s, and 498.62 s faster, respectively.

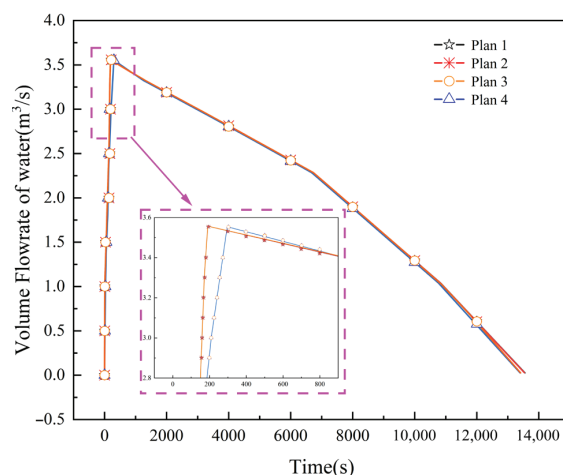


Figure 14. Variation of Drainage Flow Rate under Different Drainage Schemes.

5. Conclusions

This paper establishes a mathematical model for the drainage process in a pressurized pipeline with a single air valve. The model was validated using existing experimental data. The study focused on the relationships between the opening strategy, valve opening degree, and air valve diameter, as well as key hydraulic parameters such as pipeline drainage velocity and cavity pressure head. The drainage model with multiple air valves was also applied to the Hehe drainage section pipeline of the North Main Line in the “Hanjiang-to-Weihe River Diversion Project” Phase II. Drainage calculations and analyses for this section led to the following main conclusions:

- (1) A mathematical model for the drainage process was developed using the rigid water column assumption and validated by experiments. The results show that the model matches experimental data well, with a maximum error of 3.59%.
- (2) During drainage in pressurized pipelines with air valves, both linear and nonlinear power-function valve opening strategies improve the process. The nonlinear opening strategy increased the minimum cavity pressure head by 0.1014 m and delayed its occurrence by 0.655 s compared to the linear strategy. It also effectively reduced the rapid increase in drainage water velocity at the start of drainage.
- (3) As the valve opening degree increases, drainage water velocity rises significantly and drainage time shortens. At 20% opening, velocity increased by 0.2 m/s and drainage time dropped by 55.7%. At a larger opening, velocity increased by 0.48 m/s and drainage time fell by 65.4%. However, the minimum cavity pressure head fell by 1.05 m and 2.286 m, reducing drainage stability. Increasing the valve opening degree speeds up drainage but may reduce process stability.
- (4) Increasing the air valve diameter improves drainage stability in pressurized pipelines. Small air valves (0.003 m) caused pressure fluctuations due to low air intake, but these weakened when the diameter increased to 0.005 m or more. At 0.005 m and 0.008 m, the drainage time was reduced by 54% and 63.3%, respectively, compared to the small valves.

- (5) Well arrangement of multiple air valves can reduce negative pressure head duration without changing the minimum pressure value. All schemes showed drainage flow rates that first rose then fell, reaching a maximum of 3.61 m³/s. Compared to Scheme 1, drainage times for Schemes 2, 3, and 4 were reduced by 105.71 s, 273.5 s, and 498.62 s.

The rigid water column model adopted in this study simplifies the actual drainage process by neglecting water elasticity and spatial variations, and assumes that submerged air valves remain closed. To more accurately unravel the complex dynamics of pipeline drainage, future work should incorporate more sophisticated hydraulic models, such as elastic models, and rigorously investigate the complex gas-liquid two-phase flows triggered by the intrusion of air through submerged air valves.

Author Contributions: Data curation, S.S.; software, validation, J.M.; formal analysis, investigation, resources, B.Z.; writing—original draft preparation, J.M.; writing—review and editing, J.J. and J.L. All authors have read and agreed to the published version of the manuscript.

Funding: This work was supported by the research project of Shaanxi Natural Science Basic Research Program—Yin Han Ji Wei Joint Fund (No. 2021JLM-55), the Young Innovation Team Project of the Education Department of Shaanxi Province (24JP118), and the Innovation Capability Support Program of Shaanxi (No. 2024RS-CXTD-31). The support is gratefully acknowledged.

Data Availability Statement: The original contributions presented in this study are included in the article. Further inquiries can be directed to the corresponding author.

Acknowledgments: During the preparation of this manuscript, we have utilized DeepSeek-R1 Chat to verify grammatical accuracy and refine academic expressions. All AI-generated content has been rigorously reviewed and manually edited by us, and we take full responsibility for the publication's originality, validity, and integrity. We confirm that all our academic viewpoints, experimental data, and conclusions originate exclusively from our independent research, in full compliance with MDPI's publication ethics policies.

Conflicts of Interest: Author Jinyang Ma was employed by the company Shaanxi Hanjiang-to-Weihe River Water Diversion Project Construction Co., Ltd. Author Bo Zhang was employed by the company Powerchina Northwest Engineering Co., Ltd. The remaining authors declare that the research was conducted in the absence of any commercial or financial relationships that could be construed as a potential conflict of interest.

References

1. Dong, H.K.; Tian, R.; Wang, Q.G. Calculation of Hydraulic Transient Process in Long-Distance Water Conveyance Pipelines. *Water Resour. Dev. Manag.* **2022**, *42*, 32–38.
2. Wang, Q. Study on Filling Test of Long-Distance, Large-Section, Pressurized Water Conveyance Tunnel. *Water Resour. Dev. Manag.* **2021**, *41*, 41–45.
3. Tang, X.L.; Liu, Z.K.; Zhang, L.; Li, X.Q. Experimental investigations on characteristics of water-air two-phase flows during water-fillings in undulation pipelines. *J. Hydrodyn.* **2024**, *12*, 706–724. [CrossRef]
4. RolandoYezid, P.; Arman, R.; Musandji, F. Experimental Investigation of the Emptying Process and Air Cavity Dynamic in Pipelines. *J. Irrig. Drain. Eng.* **2022**, *148*, 04022040. [CrossRef]
5. Zhang, H.Z.; Li, Y.H.; Jiang, J.; Wang, Y.C.; Luo, S. Influence of filling velocity on the hydraulic characteristics of the air valve hump section. *China Rural Water Hydropower* **2019**, *11*, 172–187.
6. Zhou, L.; Cai, B.H.; Hu, C.Z.; Lu, Y. Simulation of Hydraulic Transients during Pipeline Filling with Multiple Air Pockets. *J. Huazhong Univ. Sci. Technol.* **2025**, *53*, 158–165.
7. Liang, X.; Wang, Y.L.; Cui, S.J.; Zhou, B.; Wang, L.; Cao, W. Analysis of Pipeline Filling Characteristics under Different Inclination Angles. *Water Resour. Power* **2024**, *42*, 60–62.
8. Ling, W.F.; Bryan, K.; Ahmad, M. Numerical investigation of filling transients in small-scale pipelines with submerged outlet. *J. Hydraul. Res.* **2016**, *55*, 647–656.

9. Yan, J.S.; Liao, G.L. Three-dimensional numerical simulation of hydraulic characteristics during the filling process of pressurized pipelines. *Water Resour. Hydropower Eng.* **2015**, *46*, 110–114.
10. Lin, L.; David, Z.; Biao, H. Analysis of pressure transient following rapid filling of a vented horizontal pipe. *Water* **2018**, *10*, 1698. [CrossRef]
11. Coronado-Hernández, E.O.; Fuertes-Miquel, V.S.; Iglesias-Rey, L.P.; Martínez-Solano, F.J. Rigid water column model for simulating the emptying process in a pipeline using pressurized air. *J. Hydraul. Eng.* **2018**, *144*, 06018004. [CrossRef]
12. Paternina-Verona, A.D.; Coronado-Hernández, E.O.; Aguirre-Mendoza, M.A.; Espinoza-Román, H.G.; Fuertes-Miquel, V.S. Three-Dimensional Simulation of Transient Flows during the Emptying of Pipes with Entrapped Air. *J. Hydraul. Eng.* **2023**, *149*, 13302. [CrossRef]
13. Laanearu, J.; Annus, I.; Koppel, T.; Bergant, A.; Vučković, S.; Hou, Q.; Tijsseling, A.S.; Anderson, A.; van't Westende, J.M. Emptying of large-scale pipeline by pressurized air. *J. Hydraul. Eng.* **2012**, *138*, 1090–1100. [CrossRef]
14. Besharat, M.; Coronado-Hernández, O.E.; Fuertes-Miquel, V.S.; Viseu, M.T.; Ramos, H.M. Backflow air and pressure analysis in emptying a pipeline containing an entrapped air pocket. *Urban Water J.* **2018**, *15*, 769–779. [CrossRef]
15. Coronado-Hernández, O.E.; Fuertes-Miquel, V.S.; Angulo-Hernández, F.N. Emptying operation of water supply networks. *Water* **2018**, *10*, 22. [CrossRef]
16. Tijsseling, A.S.; Hou, Q.; Bozkuş, Z.; Laanearu, J. Improved one-dimensional models for rapid emptying and filling of pipelines. *J. Press. Vessel Technol. Trans. ASME* **2016**, *138*, 031301. [CrossRef]
17. Fuertes-Miquel, V.S.; Coronado-Hernández, E.O.; Iglesias-Rey, L.P.; Mora-Meliá, D. Transient phenomena during the emptying process of a single pipe with water–air interaction. *J. Hydraul. Res.* **2019**, *57*, 318–326. [CrossRef]
18. Tasca, E.S.A.; Karney, B. Improved Air Valve Selection through Better Device Characterization and Modeling. *J. Hydraul. Eng.* **2023**, *149*, 142–156. [CrossRef]
19. Bian, S.K.; Zhang, X.Y.; Li, G.; Li, L. Study on valve-closure law and air-valve optimization in gravity flow pipelines. *Adv. Sci. Technol. Water Resour.* **2025**, *45*, 55–61.
20. Wu, F.; Liu, W.Y.; Zai, S.M.; He, R.; Gao, X.J.; Huang, J.; Nie, M.M. Airflow characteristics and water hammer elimination effect of the staged exhaust air valve. *J. Drain. Irrig. Mach. Eng.* **2025**, 1–9. Available online: <https://link.cnki.net/urlid/32.1814.th.20250620.1730.004> (accessed on 17 June 2025).
21. Li, N.; Zhang, J.; Shi, L.; Chen, X.Y.; Zhang, X. Water Hammer Protection Characteristics of a Combined Air Vessel and Overpressure Relief Valve. *J. Drain. Irrig. Mach. Eng.* **2020**, *38*, 254–260.
22. Bashiri-Atrabi, H.; Hosoda, T. The motion of entrapped air cavities in inclined ducts. *J. Hydraul. Res.* **2015**, *53*, 814–819. [CrossRef]
23. Izquierdo, J.; Fuertes, V.S.; Cabrera, E.; Iglesias, P.L.; Garcia-Serra, J. Pipeline start-up with entrapped air. *J. Hydraul. Res.* **1999**, *37*, 579–590. [CrossRef]
24. Brunone, B.; Karney, B.W.; Mecarelli, M.; Ferrante, M. Velocity profiles and unsteady pipe friction in transient flow. *J. Water Resour. Plan. Manag.* **2000**, *126*, 236–244. [CrossRef]
25. Huang, H.; Guo, P.C.; Yan, J.G.; Li, B. Progress in the study of friction models for unsteady flow in pressurized pipelines. *J. Hydrodyn.* **2025**, *40*, 274–296.
26. Fuertes-Miquel, V.S. Hydraulic Transients with Entrapped Air Pockets. Ph.D. Thesis, Polytechnic University of Valencia, Valencia, Spain, 2001.
27. Lee, N. Effect of Pressurization and Expulsion of Entrapped Air in Pipelines. Ph.D. Thesis, School of Civil and Environmental Engineering, Georgia Institute of Technology, Atlanta, GA, USA, 2005.
28. Leon, A.S.; Ghidaoui, M.S.; Schmidt, A.R. A robust two-equation model for transient-mixed flows. *J. Hydraul. Res.* **2010**, *48*, 44–56. [CrossRef]
29. Martin, C. Entrapped air in pipelines. In Proceedings of the Second International Conference on Pressure Surges, London, UK, 22–24 September 1976.
30. Iglesias-Rey, L.P.; Fuertes-Miquel, V.S.; García-Mares, F.J. Comparative study of intake and exhaust air flows of different commercial air valves. In Proceedings of the 16th Conference on Water Distribution System Analysis, Bari, Italy, 13–16 July 2014.
31. Wylie, E.; Streeter, V. *Fluid Transients in Systems*; Prentice Hall: Hoboken, NJ, USA, 1993.
32. Liu, X.M.; Xie, Y.B. Study on calculation methods for the friction resistance coefficient in long-distance district heating networks. *Fluid. Mach.* **2022**, *50*, 100–104.

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

What Is a (Pressure) Wavefront?

Alan E. Vardy ^{1,*} and Arris S. Tijsseling ²¹ Civil Engineering, University of Dundee and Dundee Tunnel Research, Dundee DD1 4HN, UK² Mathematics and Computer Science, Eindhoven University of Technology, Eindhoven, P.O. Box 513, 5600 MB Eindhoven, The Netherlands; a.s.tijsseling@tue.nl

* Correspondence: a.e.vardy@dundee.ac.uk

Abstract: In the context of two- and three-dimensional wave propagation, the word ‘wavefront’ is used to describe a surface that is expanding in time, and in the corresponding two-dimensional case, it describes a line that is also expanding in time. The equivalent definition in one-dimensional applications is simply a point, but the word ‘wavefront’ is nevertheless commonly used in the context of fluid waves in pipes, albeit for a very different purpose. However, no formal definition exists in the 1D context and the word is not interpreted consistently by all authors, thereby creating a potential for miscommunication that is clearly undesirable in scientific or engineering contexts. This paper uses a wide range of one-dimensional, wave-like phenomena to illustrate how varied wave propagation can be, even when the initial conditions are undisturbed and the trigger of the disturbance is simple. Despite these simplifications, no universally meaningful, formal definition of a wavefront seems to be achievable, partly because, in some instances, key flow parameters such as pressure and velocity exhibit completely different behaviour. Accordingly, instead of seeking a universal, unambiguous definition, a qualitative definition is proposed that is consistent with most of the presented examples and with the most common uses of the word in the general literature.

Keywords: wavefront; dispersive waves; wave steepening; wavespeed; water-hammer

1. Introduction

In studies of waves propagating in two or three dimensions, the word ‘wavefront’ has a formal definition. One formulation of this definition is “*the wavefront of a time-varying wave field is the set (locus) of all points having the same phase*” (Whelan & Hodgeson [1]). This is used in applications such as optics and electromagnetics as well as in fluid flows, but attention herein is focused exclusively on waves in fluids. To set the scene, consider waves propagating radially outwards from a point at which a pebble has been dropped into a lake with an otherwise undisturbed surface. At any instant, the resulting disturbance resembles a succession of waves following one after the other in a continuous manner, each decreasing in amplitude with increasing radius. For this simple case, the overall pattern comprises a series of crests and troughs, all of which are circular in plan. Using the above definition, the circle of points on the crest of any particular wave is an example of a wavefront. So are corresponding circles of points in a trough or at intermediate locations between crests and troughs. Importantly, although the fluid particles are moving only radially and vertically, the wavefront itself is circumferential. That is, it is in a direction normal to that of the particle motion and wave propagation. Also importantly, individual circles

(wavefronts) give no information about *radial* distributions of flow parameters. To obtain that information, it is necessary to compare conditions on successive circles (wavefronts).

In analyses of flows along straight, uniform pipes, it is usually assumed that, at least for pressure, plane waves remain plane. Then, the whole flow behaviour can be represented one-dimensionally by axial variations at any randomly chosen point in the cross-section. Based on the above definitions, if the flow is regarded as three-dimensional, a wavefront is the plane of a cross-section. Likewise, if it is regarded as two-dimensional, a wavefront is a line in the cross-section. Similarly, from a one-dimensional perspective, a wavefront would simply be a point in the cross-section. Clearly, however, none of these interpretations is of practical use for describing wave propagation in pipes. Nevertheless, the term ‘wavefront’ is frequently used, but with a completely different meaning. Instead of describing a feature normal to the direction of wave propagation, it is used to describe the character of *axial* variations—i.e., in the same direction as the wave propagation itself. Commonly, but not always, attention is focused on relatively steep-fronted disturbances, namely with strong axial variations in pressure p and velocity U , etc. In practice, different parameters can have different axial shapes—e.g., $p\{x\}$ and $U\{x\}$ —so strictly, it would be proper to refer to, say, pressure wavefronts and velocity wavefronts. In practice, this is rarely done because attention is usually focused on pressure (although both parameters are discussed herein). A classical example of such wavefronts is so-called water-hammer in liquid-filled pipes (Wylie & Streeter [2]), so named because induced movements of pipes or valve stems, etc., can cause loud knocking sounds. Other examples of steep-fronted waves are encountered in many contexts—e.g., pipe filling (Ferreira [3]) or emptying (Hou et al. [4]), arterial blood flow (Alastruey et al. [5]), leak detection (Xu et al. [6]), railway tunnels (Herb et al. [7]; Wang et al. [8]), fluid–structure interactions (Ferrás et al. [9]) and shock tubes (Balan & Raj [10]).

Notwithstanding this use of the word ‘wavefront’, the authors have never seen any attempt to define its meaning in the context of one-dimensional, transient flows. The purpose of this paper is to illustrate a wide range of different behaviours and to propose a tentative definition that could be used as a focus. Naturally, any such definition needs to be suitable for identifying examples of the phenomenon even in flows in which other disturbances also exist. Nevertheless, for clarity, each example presented below illustrates only an isolated wavefront, thereby avoiding complications arising from interactions that would mask the individual behaviour.

Indicative Wavefronts

In each example, the initial conditions are uniform pressure and zero velocity. A disturbance is then initiated by a prescribed, rapid increase in pressure at an upstream boundary, rising from ambient pressure to a greater value that is then held constant. This is just one of an infinity of possible ways of creating suitable disturbances, but it has the advantage of enabling many different types of wavefront to be compared in a relatively straightforward manner. Nevertheless, to avoid confusion in the discussions of the various examples, the use of the word ‘wavefront’ is avoided when describing the physical behaviour. Instead, as earlier in this paragraph, the key events are referred to simply as ‘disturbances’. This word would be far too vague to be used in general contexts, but it is unambiguous herein because each example has only one strong disturbance, albeit usually evolving in magnitude and form as it propagates.

Attention is focused on disturbances propagating along uniform pipes and ducts and, *except as otherwise stated*, the following assumptions are made to provide a clear basis for comparisons:

- The pipe is horizontal, stationary, of uniform bore and elastic-walled;
- The fluid is inviscid;
- In the examples with liquid flows, the fluid density is treated as a constant except in the determination of the speed of sound, which itself is treated as a constant;
- The initial conditions along the whole pipe are uniform-pressure and zero-flow;
- A disturbance of finite magnitude is initiated by a prescribed time-dependent *increase* in the stagnation pressure (sometimes called total pressure) at the origin of the x axis. After the initial increase, the pressure is held constant;
- All changes are treated as occurring in a homentropic manner.

This method of inducing a disturbance could be mimicked in practice by the rapid opening of a valve at the upstream end of a pipe connected to a reservoir of constant pressure. For present purposes, it has the advantage of ensuring that the induced flow is in the same direction as the wave-like propagation. An alternative method that is preferred in many textbook descriptions of water-hammer, is to begin with a steady flow and to close a valve rapidly at the downstream end of the pipe. However, this can complicate the interpretation of outcomes, especially when the initial pressure distributions include variations in pressure due to, say, skin friction, gravity, finite Mach numbers or unsteadiness. Herein, such complications would bring no benefit to offset these disadvantages.

Each example illustrates the influence of a particular phenomenon—inertia, friction, Mach number, etc. All examples in Part 1 are for liquid flows and those in Part 2 are for gas flows. The latter are inherently less simple than the former because changes in pressure necessarily cause changes in density. Also, both the density ρ and the speed of sound c vary with temperature T . This is important because, except in idealised cases, all three of these parameters vary along propagating disturbances.

The focus of the whole paper is on qualitative behaviour, not on quantitative details. In each case, the same qualitative behaviour would be predicted with any reasonable method of analysis. Accordingly, presenting details of any particular method would deflect attention from the primary focus without adding meaningful value for readers. For completeness, however, it is declared that all of the analyses have been undertaken using the software ThermoTun [11] which has been used extensively in academic research and in practical engineering design, primarily for simulating flows of gases, notably air, but also in a range of other applications—e.g., subsea oil pipelines. The software uses the Method of Characteristics (MoC) to solve one-dimensional (1D) equations of flow and it has been validated by many measurements at model scale and full scale as well as by detailed comparisons with analytical solutions. In contrast with most textbooks and papers on water-hammer in pipelines, convective terms are retained in the basic equations, thereby enabling behaviour such as the inertial shortening and lengthening of localised phenomena to be simulated. This is rarely important in the case of liquid flows (exceptions include regions of distributed cavitation, where the pressure gradient is zero), but, as illustrated in Section 7 below, it can be of high importance in some applications of waves in gas flows.

Part 1. Liquid Flows

2. Example 1: Base Case

The first example is an idealised base-case against which subsequent cases may be compared.

- The diameter and wall-thickness of the semi-infinite, horizontal, uniform-bore pipe are $D = 100$ mm and $e = 4$ mm, respectively;
- The elastic pipe-wall has a density of 7850 kg/m³ and a Young's modulus of 200 GPa;

- The fluid is an inviscid liquid with a density of 1000 kg/m^3 and a bulk modulus of 2.20 GPa ;
- With these data, the speed of sound in the liquid is $c = 1483 \text{ m/s}$ and the effective speed of sound, allowing for radial flexibility of the pipe, is $c_{eff} = 1326 \text{ m/s}$;
- At $t = 0$, the absolute pressure throughout the pipe and in the surrounding atmosphere is 100 kPa . All pressures presented in the following graphical output are expressed relative to this ambient value;
- The prescribed rapid change in pressure at the upstream boundary is a linear increase in 0.5 s from the ambient value to a maximum value that is then held constant. The chosen increase causes a velocity increase of 2 m/s .

The lower row in Figure 1 shows pressure and velocity distributions along the pipe at successive instants after valve opening. The first of these, at $t = 0.5 \text{ s}$, is at the instant when the prescribed change at the boundary ceases and the remainder are at intervals of 2 s . By inspection, the only obvious difference between the various distributions is the distance from the boundary to the ramp-like disturbance. The validity of this interpretation is confirmed by the upper row in the figure, which compares pressure and velocity histories at five locations, namely $x = 0, 2, 4, 6$ and 8 km . To enable these comparisons to be made in detail, the graphs are offset so that the intervals between them are approximately 0.05 s instead of the true intervals of slightly more than 1.5 s .

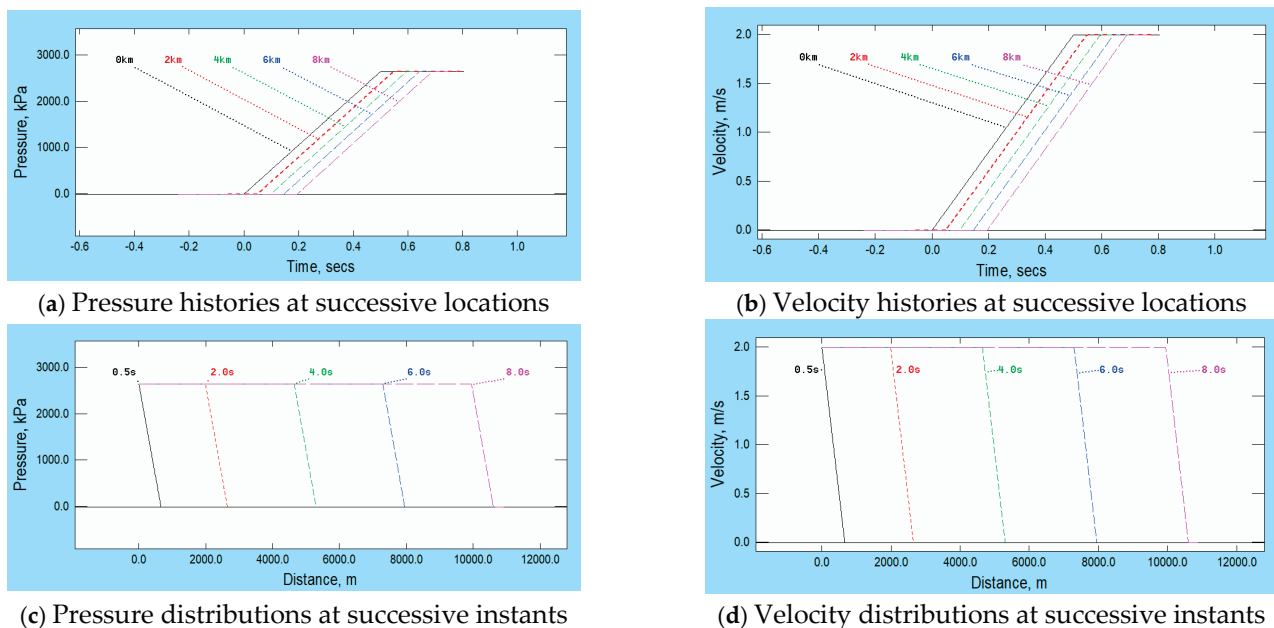


Figure 1. Idealised wave propagation—inviscid liquid flow. Temporal histories at fixed locations and spatial distributions at fixed instants in time. The temporal histories are offset in time to highlight the almost unchanging shape of the disturbance. NB: Graphs with a blue background denote liquid flows.

Although general behaviour cannot be inferred from individual numerical simulations such as this, the evidence in the figure suggests that the leading ramp is propagating at an almost constant speed and that it is doing so without any change in form. However, as demonstrated below (in Example 6), it is not *exactly* true to state that the ramp is propagating without any change whatsoever. Strictly, this could be possible only for infinitesimal disturbances, just as, in the frequency domain, only acoustic amplitude waves approximate almost exactly to propagation in a linear manner. In both cases, waves travel at the local speed of sound relative to fluid particles—i.e., at $U + c$, not just c . Thus, the

propagation speed in regions of high pressure and velocity is greater than that in regions of low pressure and velocity. It follows that the tentative definition given above excludes regions where variations in $U + c$ cannot reasonably be regarded as negligible. This issue is revisited in Section 7 (Example 6) in the context of fluids that are more compressible than liquids.

As an aside, it is worth noting that the fact that the results presented in Figure 1 are so similar at successive locations up to $x/D = 80,000$ is an indication of the ability of the numerical method to propagate disturbances without causing significant non-physical dispersion. It follows that strong dispersive behaviour predicted in subsequent examples can reasonably be assumed to be primarily attributable to true physical causes, not to numerical inadequacies.

3. Example 2: Quasi-Steady Friction

The second example differs from the first in only one respect, namely removing the idealisation of inviscid conditions. Instead, account is taken of an inevitable consequence of viscosity and turbulence, namely skin frictional resistance at the pipe wall surface. In this example, the resistance is approximated by quasi-steady relationships suitable for turbulent flows. That is, at any particular location, it is treated as being dependent only on the instantaneous Reynolds number, not on rates of change thereof. Consideration of dependence on the latter is deferred until a later example (in Section 8). However, account is taken of the variation of the quasi-steady skin friction coefficient with the Reynolds number. To highlight the qualitative behaviour, the implied roughness of the pipe wall is relatively large, corresponding to Fanning (and Darcy–Weisbach) friction coefficients of $f = 0.01$ (and $\lambda = 0.04$).

Figure 2 shows pressure and velocity distributions along the pipe when the prescribed upstream pressure is identical to that used in the first example. However, some of the pressure difference is needed to overcome skin friction, and this increases as the length of the disturbance increases. Therefore, the pressure difference available for overcoming inertial resistance reduces, resulting in a continual reduction in the amplitude of the leading ramp as it propagates along the pipe. Furthermore, although the magnitude of the pressure at any particular location along the pipe consistently *increases*, the magnitude of the corresponding velocity *decreases*. Indeed, it does so even at the upstream boundary where the pressure is held constant. This behaviour was studied in depth by Leslie & Tijsseling [12] and Liou [13] who provided simple analytical expressions describing the rate of decay of the step's amplitude with distance together with corresponding expressions for related phenomena. This extended early work by Ludwig & Johnson [14]; Binnie [15] and Burnett [16] that focussed on specific aspects of the topic.

The preceding qualitative description does not, in itself, explain why the pressure distribution along the pipe is not almost linear—as it would be in a steady flow with uniform velocity. The principal reason for this relates to the time taken for waves to travel. As the leading jump moves along the pipe, the length of the region of wall resistance increases and local flow dynamics just behind the jump require its amplitude to decrease. This is true for the velocity as well as for the pressure, and this information has to be transmitted back to the upstream boundary, implying the existence of continuous reflections travelling in the negative x -direction. Each of these waves itself influences the local flow states during its travel and that, in turn, causes further reflections in the positive x -direction, and so on. Thus, the seemingly simple act of relaxing the assumption of inviscid flow leads to complexities that, although qualitatively explicable, could not be deduced in detail by physical reasoning alone. For completeness, it is acknowledged that, although the

principal reason for the non-linear shapes of the depicted pressure distributions is as stated above, the detailed shapes will also be influenced by non-linear aspects of skin friction in turbulent flows.

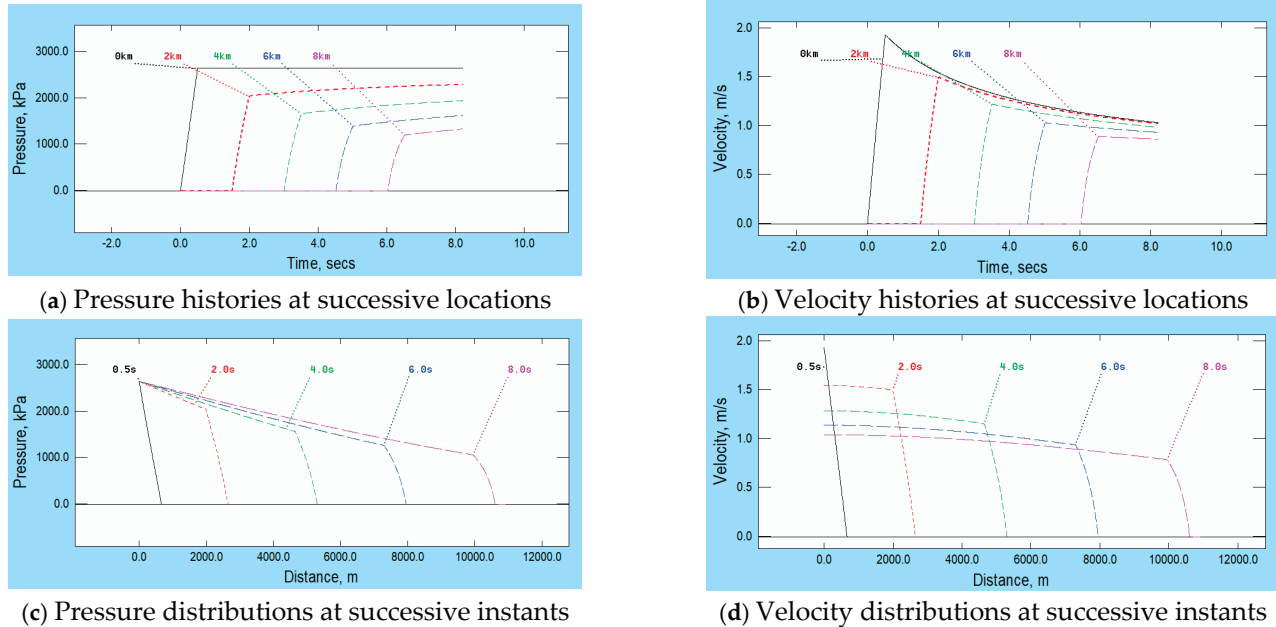


Figure 2. Influence of quasi-steady friction on wave propagation (NB: consideration of unsteady friction is deferred to Section 8, Example 7).

Even the qualitative nature of the particular outcome illustrated in Figure 2 is valid only for upstream boundary conditions that imply constant pressure after the initial increase. In these cases, after the initial increase, the corresponding velocity at the boundary reduces indefinitely with increasing time. In contrast, if the prescribed boundary condition were an increase in *velocity* to a constant value, the corresponding pressure at the boundary would increase indefinitely. More general upstream boundary conditions include a prescribed relationship between pressure and flowrate (e.g., representing a pump or a valve). The consideration of such cases would lead to less simple outcomes, but it would not add meaningfully to the underlying purposes of the present paper. In most of the following examples, the amplitudes of the velocity and pressure at any particular distance behind the leading zone of the disturbance vary continuously, albeit perhaps slowly. That is, although the disturbances have characteristics in common with those in Figure 1, the disturbances change as they propagate. Furthermore, they would do so even if additional complications arising from their finite amplitudes did not occur.

One particular detail of the graphs merits further attention, namely the fact that the leading ramp does not maintain its original linear form. This is primarily because, for turbulent flows, the quasi-steady resistance varies approximately with the square of the velocity. Very close to the leading tip, the velocity is small and only a small proportion of any available pressure difference is needed to overcome skin friction. Thus, here, the behaviour will resemble the inviscid equivalent quite closely. However, still focussing on conditions close to the leading jump, the required proportion necessarily increases with increasing distance from the tip. To illustrate this numerically, consider the pressure changes needed to induce velocity changes (a) from zero to 0.2 m/s and (b) from 0.2 m/s to 0.4 m/s. The inertial changes in these two cases are identical so the corresponding pressure differences are also identical. However, the increase in U^2 for the first step, namely

$0.04 \text{ m}^2/\text{s}^2$, is much smaller than that for the second step, namely $0.4^2 - 0.2^2 = 0.12 \text{ m}^2/\text{s}^2$. This behaviour persists all the way back to the upstream boundary.

4. Example 3: Tapered Pipe

Although it is rare for water-hammer simulations to be necessary for liquid flows in stiff, tapered pipes, the topic has occasionally attracted interest—e.g., Tarantine & Rouleau [17], Yoshizawa & Ando [18] and, more recently, Jarachi et al. [19]. Tapering is also important in the context of arterial blood flows (Alastruey et al. [5]). In the third example, the diameter of a 1 km long tapered pipe varies from 100 mm at the upstream boundary to 400 mm at the remote boundary in such a manner that the area varies linearly. The authors do not suggest any practical application for which this geometry would be suitable, but the resulting behaviour is nevertheless informative and is indicative of the influence of tapers. The prescribed pressure history at the upstream boundary is identical to that used in Figure 1 and inviscid conditions are assumed, thereby enabling the influence of the taper to be inferred unambiguously. Each individual graph of pressure history and distribution in Figure 3a,c broadly resembles the distributions obtained with quasi-steady friction in a uniform pipe. However, the similarities exist only for the pressures. The trends exhibited in the graphs of velocity are the *opposite* of those found in the example with quasi-steady friction. Although the pronounced change in velocity at the front of the disturbance reduces as it propagates, its product with the cross-sectional area—i.e., the volumetric flowrate—nevertheless increases strongly. This information has to be communicated back to the upstream boundary, ‘telling it’ that the prescribed constant pressure can sustain an ever-increasing flow rate and hence an ever-increasing velocity. As a result, the velocity at any particular location along the pipe continually *increases* in time, not decreases.

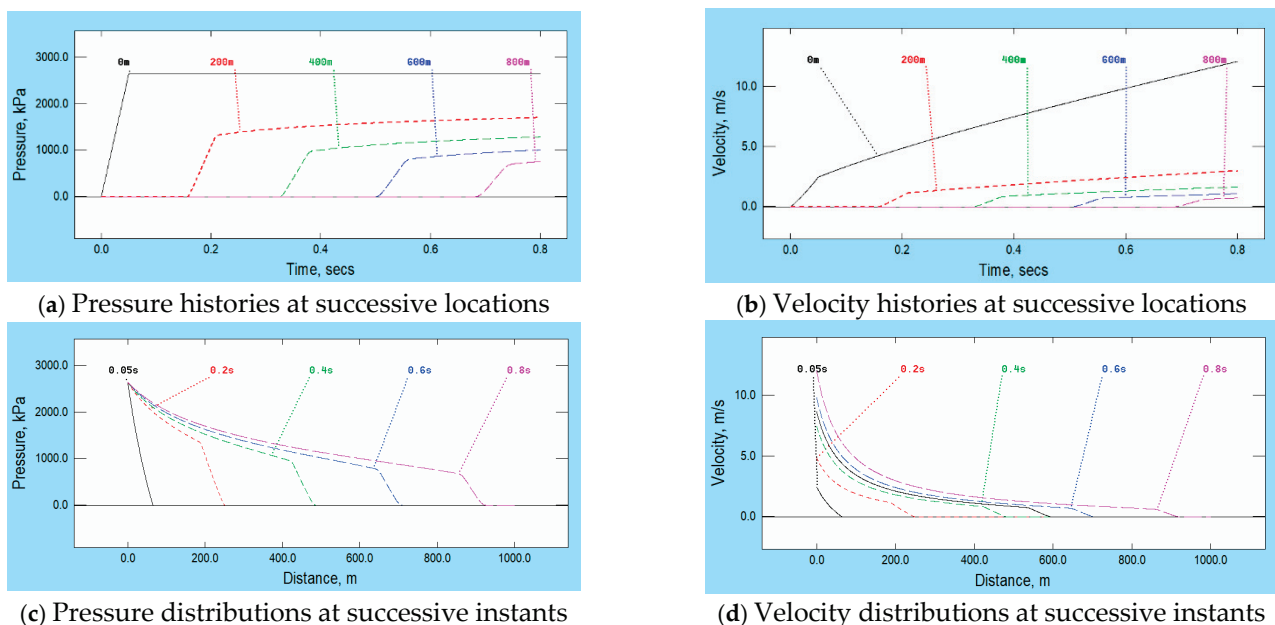


Figure 3. Wave propagation in a tapered pipe—linearly increasing $A\{x\}$.

Before moving on to the next topic, it is instructive to consider briefly an alternative variation in area along the pipe, namely a linear increase in *diameter*, again from 100 mm to 400 mm. Inevitably, there is a close resemblance between the qualitative behaviours in the two cases (Figures 3 and 4). However, at any particular location, the pressure in the

linear-diameter case has the striking result of remaining constant after the leading step has passed even though the velocity continues to increase.

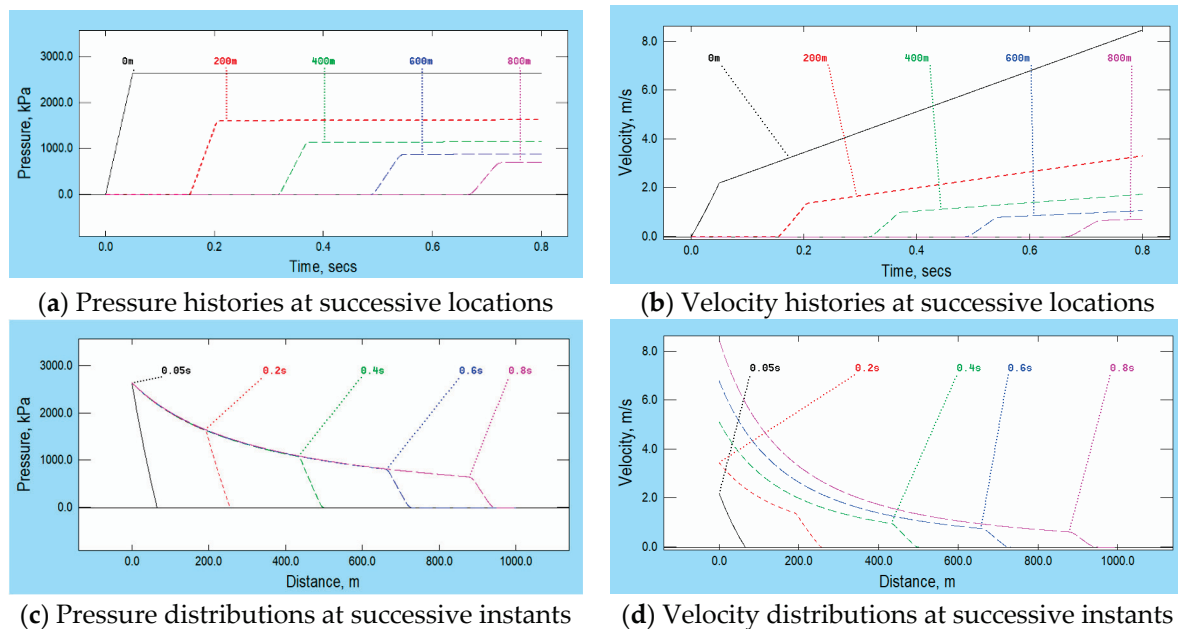


Figure 4. Wave propagation in a tapered pipe—linearly increasing $D\{x\}$.

Although this might be surprising at first sight, it is actually a special case of a well-known result in a different context, namely the spherical propagation of sound from an isolated point source in an unbounded medium. When a disturbance such as the leading step herein propagates radially outwards, its pressure amplitude varies inversely with the radial distance from the source—see, for instance, Lighthill [20]. After the disturbance at the origin ceases, no further changes propagate outwards so the radial extent of the zone of unchanging pressures gradually increases—as in the left-hand column of Figure 4.

5. Example 4: Lateral Mass Flow

Now, the propagation of a disturbance along a leaky pipe is considered (Wylie & Streeter [2]). The internal diameter of the pipe is 100 mm and its walls are uniformly perforated. The total flow area through them in a 1 km length is equal to twice the cross-sectional area of the pipe. The prescribed pressure history at the upstream boundary is the same as in the preceding examples. Figure 5 shows that the qualitative features of the pressure and velocity histories and distributions are similar to that obtained above for tapered pipes. However, in stark contrast with those cases, the continual reduction in velocity in the leading zone in this instance is associated with ever-reducing flow rates. This is necessarily accompanied by a continual reduction in the associated pressure change and, as this information is received at the upstream boundary, the rate of flow into the pipe at that boundary increases.

The loose, qualitative resemblance of the behaviours in the tapered pipe and leaky pipe cases would be even more pronounced if the leakage area were reduced by, say, a half. However, the use of the larger area has the advantage of highlighting a particular characteristic that is of special relevance in the present context, namely the vanishing amplitude of the leading step at larger times. Further reference is made to this later, when attempting to provide a realistic definition of a wavefront.

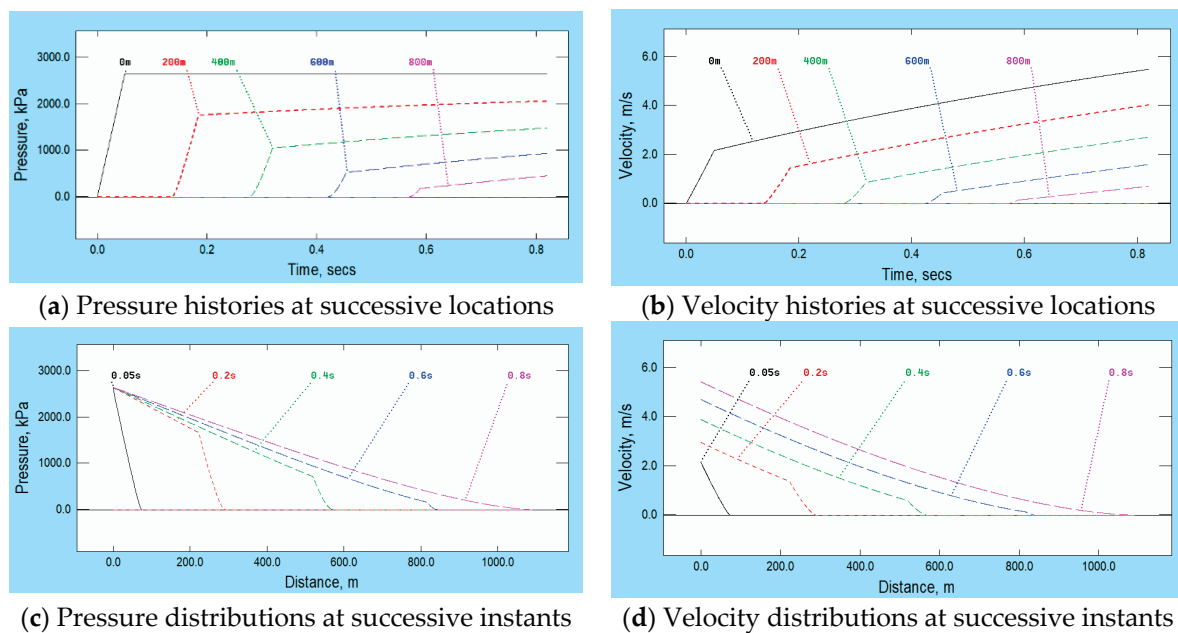


Figure 5. Wave propagation in a pipe with uniformly distributed leakage holes.

6. Example 5: Fluid–Structure Interaction

In all the above examples, account is taken of the influence of the pipe's radial flexibility on the fluid wavespeed along the pipe. However, no account is taken of consequential *axial* responses in the pipe wall. Instead, the direct influence of the radial flexibility (albeit not also its radial inertia) is treated in a conventional manner that utilises an effective bulk modulus. That is, the pipe is treated as infinitely rigid and the bulk modulus of the fluid is reduced to mimic the influence of the radial flexibility. The adjustment ensures that the resulting speed of sound in the imaginary fluid in a rigid pipe is a good approximation to the speed of propagation of small disturbances in the true fluid in the true pipe.

Although this adjustment is adequate for most engineering purposes, the neglecting of axial disturbances in the pipe wall itself makes the analysis unsuitable for applications in which such disturbances play a significant role. Reviews of papers that allow for the effect are given by Li et al. [21] and Ferras et al. [9]. Notable examples include pipework that has little axial restraint, especially when the disturbances are structure-induced, not fluid-induced. In suspended pipework, for instance, pressure forces caused by the rapid closure of a valve can cause the valve itself to move. This causes axial stress waves in the pipe wall and, when either type of wave reaches a bend or junction elsewhere along the pipe, it triggers new disturbances. In general, these can include bending and shear waves as well as axial waves, but attention herein is on single, straight pipes, so only axial waves are considered. This coupling behaviour is known as junction coupling. In most engineering applications, waves in solids travel faster than those in fluids so they can be tracked independently. Importantly, in most engineering applications, notably with flows in metal pipes, these stress waves propagate much faster than the corresponding waves in the fluid itself.

In practice, waves travelling along the pipe in one medium (e.g., the wall) will be detectable in the other medium (i.e., the fluid). For example, stress waves propagating axially in a pipe wall cause circumferential strain as well as axial strain—because of the Poisson ratio effect. The resulting changes in the cross-sectional area of the pipe cause changes in fluid pressure. Likewise, pressure waves in the fluid induce circumferential

strains in the pipe wall and the associated axial strains influence axial stress waves in the wall. This behaviour is known as Poisson coupling.

Poisson coupling is usually much smaller than junction coupling, and it is commonly neglected in analyses. That is, stress waves and pressure waves are assumed to propagate between boundaries independently, regardless of whether account is taken of interactions at boundaries. However, Poisson coupling is allowed for in the example shown in Figure 6. Indeed, to highlight the effect clearly, its importance is intentionally exaggerated in this example. Firstly, the chosen ratio of the pipe wall thickness and internal diameter, namely 1/50, is smaller than would be likely in practical applications in which water-hammer is expected. Secondly, in contrast with the ramp-like pressure changes considered in the preceding examples, the prescribed pressure increase at the upstream boundary is effectively instantaneous.

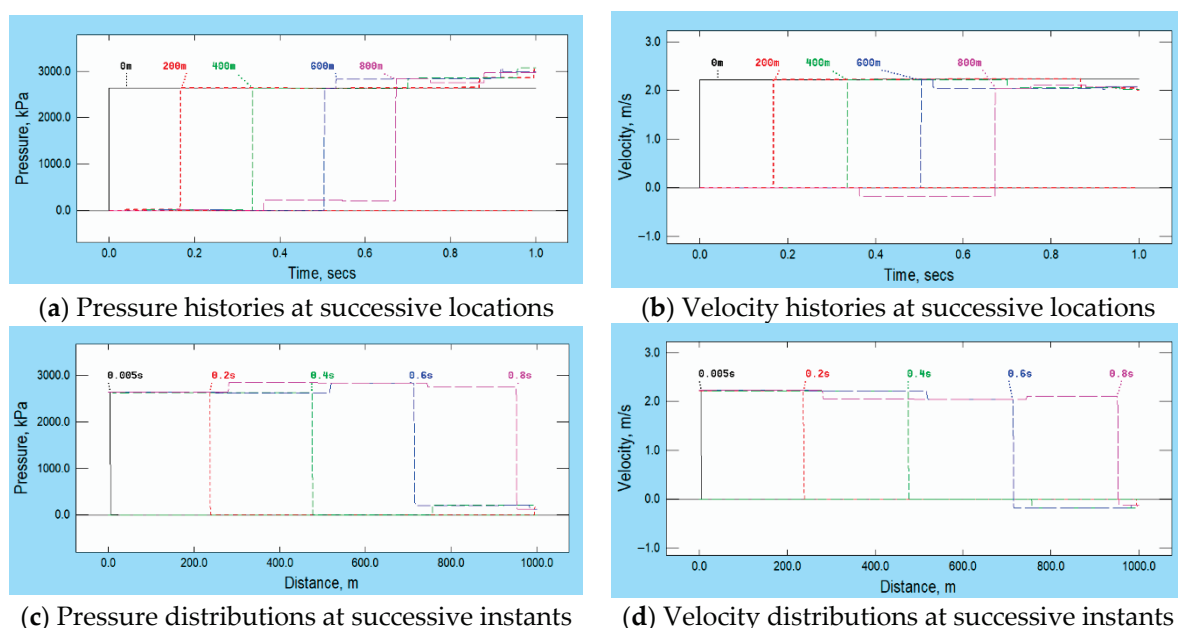


Figure 6. Wave propagation allowing for waves in the pipe wall as well as in the liquid.

Axial motion of the pipe wall at the upstream boundary is prevented and, as a consequence, the circumferential strain caused by the increased pressure causes a small, tensile, axial stress wave in the pipe wall. This propagates along the pipe ahead of the main pressure change and, as described above, it will be accompanied by a small increase in pressure also travelling at the speed of the stress wave. However, this change is so small that it cannot be seen directly in the figure. Nevertheless, its reflection from the closed end of the pipe can be seen easily—e.g., at approximately 0.38 s in the pressure history at 800 m and approximately 0.52 s at 600 m. This illustrates the earlier statement that structure-induced changes can be significantly stronger than fluid-induced ones. In this particular case, when the small tensile wave reaches the closed end, it pulls the end back upstream, thereby inducing a compression wave in the fluid.

The influence of Poisson coupling is barely visible in this Figure 6, but it is seen easily in Figure 7, which focusses on the periods shortly before and after the reflections of the first stress wave and the first pressure wave at the closed end. The left-hand box shows pressure histories at five locations approximately 100 m upstream of the end. The points labelled A–F are associated with the following events:

- A. The tensile stress wave shortly before it reaches the closed end;
- B. A smaller, compressive stress wave reflected from the end;
- C. A strong compressive pressure wave induced simultaneously with 'B';
- D. A small compressive stress wave initiated when 'B' reflected at the upstream end;
- E. A smaller, tensile stress wave reflected from the closed end;
- F. A rarefaction pressure wave initiated simultaneously with 'E'.

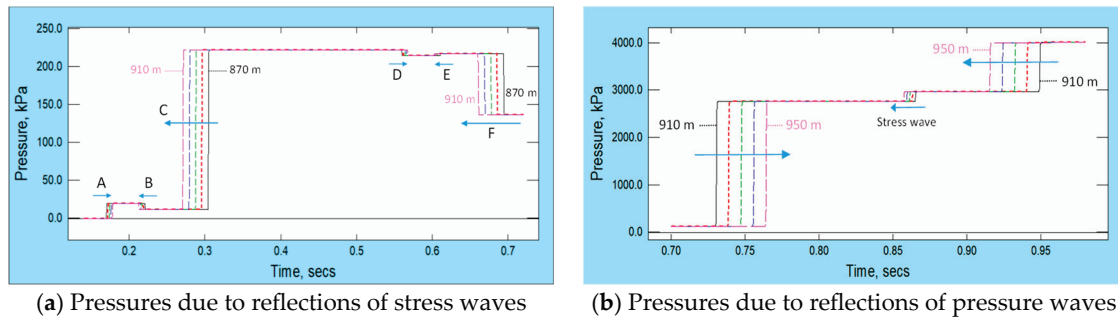


Figure 7. Reflections of waves at an unrestrained, closed end of a pipe.

The events A, B, D and E illustrate Poisson coupling and the events C and F illustrate junction coupling.

The right-hand box shows corresponding pressure histories during the subsequent reflection of the initial *pressure* wave at the closed end. The approaching wave passes the chosen locations shown in the graphs during approximately 0.76 s to 0.80 s. When it reaches the end, it forces the pipe to move in the positive *x*-direction, thereby causing a tensile stress wave to travel back upstream. This passes the graph locations at approximately 0.85 s and is seen in the pressure traces because of Poisson coupling. The primary reflected pressure wave passes during approximately 0.88 s to 0.92 s. The different durations of the stress wave and pressure wave events are caused only by the different speeds of propagation, as also are the relative timings of the events.

Part 2: Gas Flows

In all of the above examples, the fluid is a liquid and, for most purposes, its behaviour can be described with good accuracy without considering its compressibility, except when calculating the speed of sound. However, as illustrated by Zhao et al. [22] in the context of leak detection, this simplification is not acceptable when analysing pressure waves in gas pipelines. In each example in the remainder of the paper, the fluid is a gas which, for simplicity, is deemed to have the properties of air. In particular, it satisfies an equation of state, namely

$$p = \rho RT$$

where *p* is pressure, *ρ* is density, *R* = 287 J/(kg.K) is the specific gas constant and *T* is the absolute temperature. Thus, whereas the density of the liquid in the above examples was deemed to be a constant (except for its influence on the speed of sound), the gas density depends upon both the pressure and the temperature. This has a marked influence in all but one of the following examples. Likewise, the speed of sound satisfies

$$c^2 = \gamma RT$$

where $\gamma = 1.401$ is the ratio of the principal specific heats of dry air. No allowance is made for pipe flexibility because its influence is usually tiny in comparison with gas compressibility.

7. Example 6: Mach Number

First, consider the influence of the magnitude of the prescribed pressure change and, consequently, of the induced velocity and Mach number. Figure 8 shows two independent cases with different magnitudes of the prescribed change in the upstream pressure. In both cases, external complications such as skin friction and heat transfer are absent and so, in common with the example given in Figure 1 for wave propagation in a liquid, the maximum pressures do not vary during propagation. However, the steepening process referred to in the closing paragraph of Section 2 can now be seen clearly. Along the leading ramp, both the velocity U and the speed of sound c increase with pressure. Therefore, so does the wavespeed relative to the pipe, namely $dx/dt = U + c$, where x and t are the distance and time coordinates. In the example with larger pressures, dx/dt ranges from approximately 340 m/s at the tip to approximately 429 m/s at the maximum pressure. That is, the heel of the ramp travels approximately 89 m/s faster than the toe. The initial length of the ramp is approximately 170 m, so the heel should catch the toe after approximately 2 s, thereby creating a normal shock. Thereafter, the shock should propagate as a step change travelling at approximately 380 m/s. This process is not reproduced exactly in the figure because the numerical algorithms in the software package used for all examples in this paper do not include shock-capturing capabilities. Nevertheless, by the time that the ramp has reached 1000 m, it is almost a step and it is travelling at approximately the correct speed. The steepening process for the smaller-amplitude wave is similar, but the simulation ends before the time required for it to evolve into a shock (approximately 4 s). This phenomenon is of special interest in shock tube applications, but as demonstrated by pressure measurements reported by Herb et al. [7], it can also be important in some large-scale infrastructure.

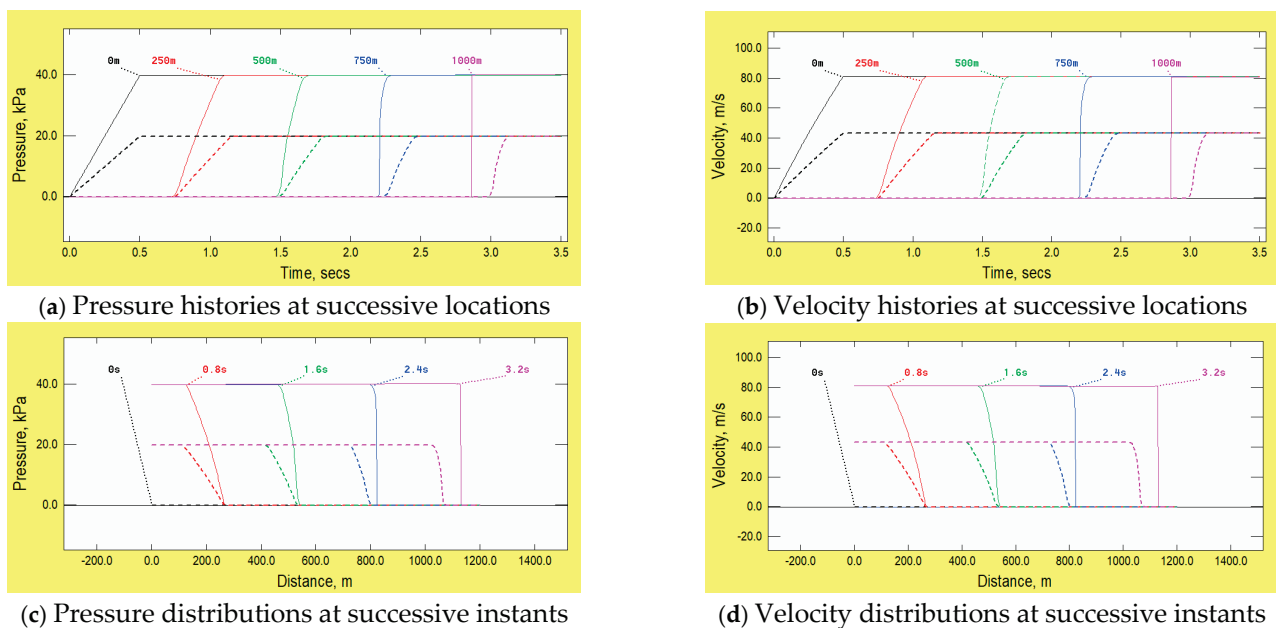


Figure 8. Influence of Mach number on wave propagation—continuous lines: $Ma = 0.234$; broken lines: $Ma = 0.128$. NB: Graphs with a yellow background are for gas flows.

In passing, it is worth noting that the failure of the software to predict the steepening process exactly is attributable primarily to the use of a fixed numerical grid. The spatial grid size Δx remains constant throughout the simulation, and, although the time step Δt is chosen independently at the beginning of each time step, the same value is used at all locations. Physically, at any particular (x, t) , the local disturbance propagates at $dx/dt = U + c$, where U and c are the local velocity and the speed of sound (in a stationary, unconfined fluid). Both of these vary along the ramp so the distances travelled during fixed intervals also vary, causing the overall shape of the ramp to change during propagation. An approximate allowance is made for this by choosing the time step at each instant so that $\Delta x/\Delta t$ is equal to the mean of the maximum and minimum values of $(U + c)$ in the previous time step. Inevitably, this causes the tip of the disturbance to travel too rapidly and the heel to travel too slowly, but it enables the general behaviour to be seen and it is a close approximation to the true behaviour after the ramp has become a step. Alternative 1D methods of analysis based on, say, control volume or finite element methods can reduce this effect, but have other downsides such as non-physical oscillations that require specialised numerical counter-measures. Two- and three-dimensional methods can be much less sensitive to this problem, but they, too, can cause numerical oscillations and they require far greater computing resources.

8. Example 7: Unsteady Friction

In Section 3, the influence of skin friction was illustrated, but, to provide a clear focus, consideration was restricted to a quasi-steady representation thereof. That is, although the simulated skin friction forces depended on the flow velocity, they did not depend upon acceleration or higher derivatives of velocity. Its influence is represented by a convolution-integral model of the generic type first proposed by Zielke [23]. Convolution-based models (usually designated ‘CVB’) are widely recognized as being more accurate than instant-acceleration-based models (IAB) that do not take explicit account of historical influences of previous flow changes, but the former typically require more complex coding and, until recently, have required significantly greater CPU resources. Strong recent advances made towards overcoming these drawbacks are exemplified by Urbanowicz et al. [24] who, in addition to greatly increasing efficiency, give an excellent, concise account of key work on this topic.

By inspection, the consequences of the unsteady contribution differ from those of the quasi-steady contribution. They are strongest in regions where the greatest rates of change of acceleration occur, namely near the toe and heel of the leading ramp. As time progresses, the cumulative effect is to reduce the greatest rates of change and, simultaneously, to extend the duration of the period when the influence of the phenomenon is observed. This type of behaviour is commonly seen in measured pressure histories in water-hammer-like flows, especially in laminar flows (e.g., Zielke [23]), but also in turbulent flows. However, it is less commonly commented upon. Indeed, in the authors’ experience, it is often overlooked by authors claiming ‘good’ agreement with measured pressure histories even if their theoretical predictions do not show the effect. Attention is drawn to this because measured pressure histories are often made close to boundaries such as valves, and pressure histories during the reflections of waves at such boundaries can be strongly influenced by their detailed shapes. *[Example: The initiating disturbance in Figure 9 is a ramp change in pressure in 0.5 s followed by constant pressure. The simulated speed of sound is approximately 340 m/s, so the length of the linear region is approximately 170 m. If such a disturbance were to reflect at a boundary, any measuring device within 85 m of the boundary would detect the tip of the reflection before the arrival of the heel of the ramp.]*

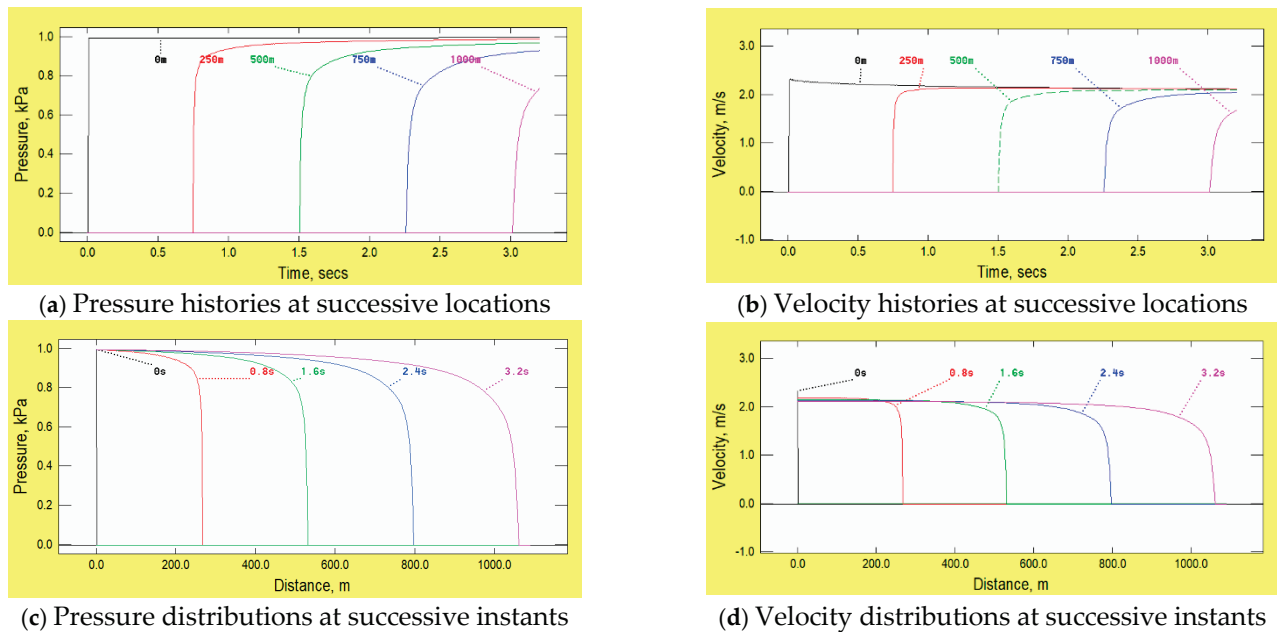


Figure 9. Influence of unsteady components of friction on wave propagation.

9. Example 8: Heated Pipe

In gas flows, changes in pressure are usually accompanied by changes in density and temperature. This leads to temperature differences between the gas and the duct that induce heat transfers that further complicate the behaviour. Banister [25] pays special attention to heat transfers that arise passively from pressure change due to friction, but points out that greater effects are to be expected when heat transfers are imposed actively. Figure 10 illustrates a disturbance propagating along a duct that is actively heated at a uniform rate along its whole length. At first sight, the figure appears to be rather complicated, so it is useful to begin the discussion by considering the influence of heat in the absence of the disturbance. If both ends of the duct were closed, the consequential increases in temperature would cause the pressure to increase uniformly along its length. In contrast, if its ends were open, gas would flow out of them and the pressure would decrease from a maximum at mid-pipe-length to a minimum at the ends. The wave propagation seen in the figure superimposes on this background effect.

The chosen initial condition is zero flow, uniform pressure and uniform temperature. Thereafter, the prescribed pressure at the end $x = 0$ is held constant at an increased value and the rate of heat transfer to the gas is uniform and constant. As the abrupt change in pressure propagates along the duct, it superimposes on the background effect described above. Its magnitude varies slightly because of the influence of the heat on the overall pressure distribution, but its propagation along the duct is qualitatively similar to that in other examples above. The same is not true, however, for the velocity. The initial step change at the prescribed-pressure boundary is essentially the same as if there were no heat. In contrast, the subsequent history is affected strongly by the outflow influence of the heat as well as by the inflow influence of the prescribed pressure. Initially, the latter dominates, but the former gradually increases in importance and eventually causes the direction of flow at $x = 0$ to reverse. In contrast, throughout the period illustrated, the conditions at the other end of the duct are uninfluenced by the imposed pressure-disturbance. The pressure at that end remains constant and the velocity continually increases. Later, the pressure-induced disturbance will cause additional outflow through this boundary.

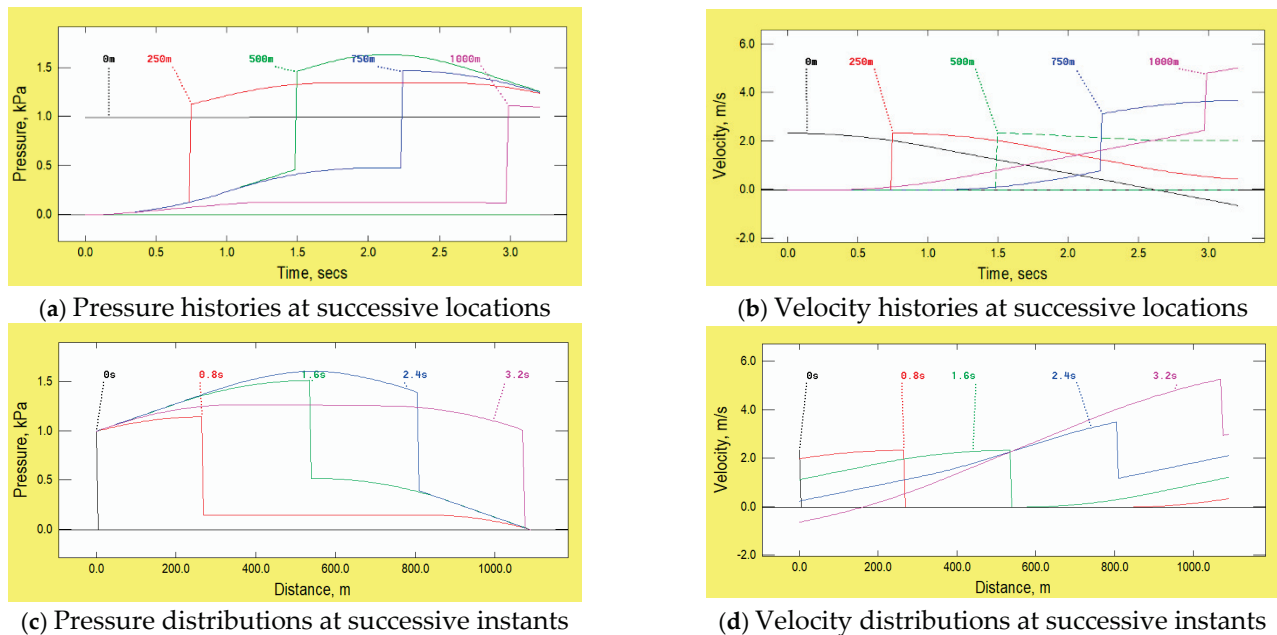


Figure 10. Wave propagation along an externally heated pipe.

10. Example 9: Duct with Chambers

In the example discussed in Section 4 (Figure 5), flow could pass through holes in the pipe wall. This is again the case in the present example, but in this instance, the holes are small orifices leading to enclosed chambers distributed axially along the duct. To trigger a strong effect in a short length of duct, the total volume of the chambers is equivalent to 12% of the duct volume. The prescribed pressure history at the upstream boundary is shown as a continuous, black line in the upper-left box in Figure 11. Although the chambers are quite large, the mass of fluid that can enter them from the pipe is small because it is determined solely by the increase in density that the inflow induces. This, in turn, is determined by the maximum possible increase in pressure inside the chambers. In this example, the geometry of the openings is chosen to ensure that the chambers behave in a slightly over-damped manner, thereby avoiding oscillatory behaviour typical of Helmholtz chambers. As a consequence, the maximum increase in pressure inside any chamber is the same as that in the wave propagating along the duct.

Although, in this example, chambers exist along the whole length of the duct, the length in which their influence is strong is quite short. This is because chambers at successively increased distances behind the leading parts of the disturbance have been exposed to elevated tunnel pressures for successively greater times. As a consequence, their internal pressures have become increasingly close to the maximum imposed pressure and their ability to accept further increases in mass has reduced accordingly. This characteristic behaviour has been recognised for many years (e.g., Vardy & Brown [26]). A potential practical application of it is given by Liu et al. [27].

The evolution of the disturbance as it propagates along the duct has some features in common with those caused by unsteady components of skin friction (Figure 9). In both cases, the influence is strongest in the leading parts of the disturbance and decays asymptotically to zero with increasing distance behind that region. This is to be expected because they are driven by *rates of change* of pressure and velocity, not by quasi-steady phenomena. Also in both cases, the phenomenon under consideration is dispersive, although the causes of this are different. In the case of unsteady friction, the difference from ideal behaviour

is caused by forces that influence momentum, but not mass-continuity. In the present example, however, it is caused by leakage from the main duct into the chambers, so it influences mass-continuity as well as momentum. In common with all other examples herein, the pressure and velocity histories after the localised regions of relatively abrupt change are particular to the prescribed upstream pressure history. However, the spatial extent of the region that is strongly influenced by unsteady friction will normally be much shorter than that which depends strongly on arrays of chambers.

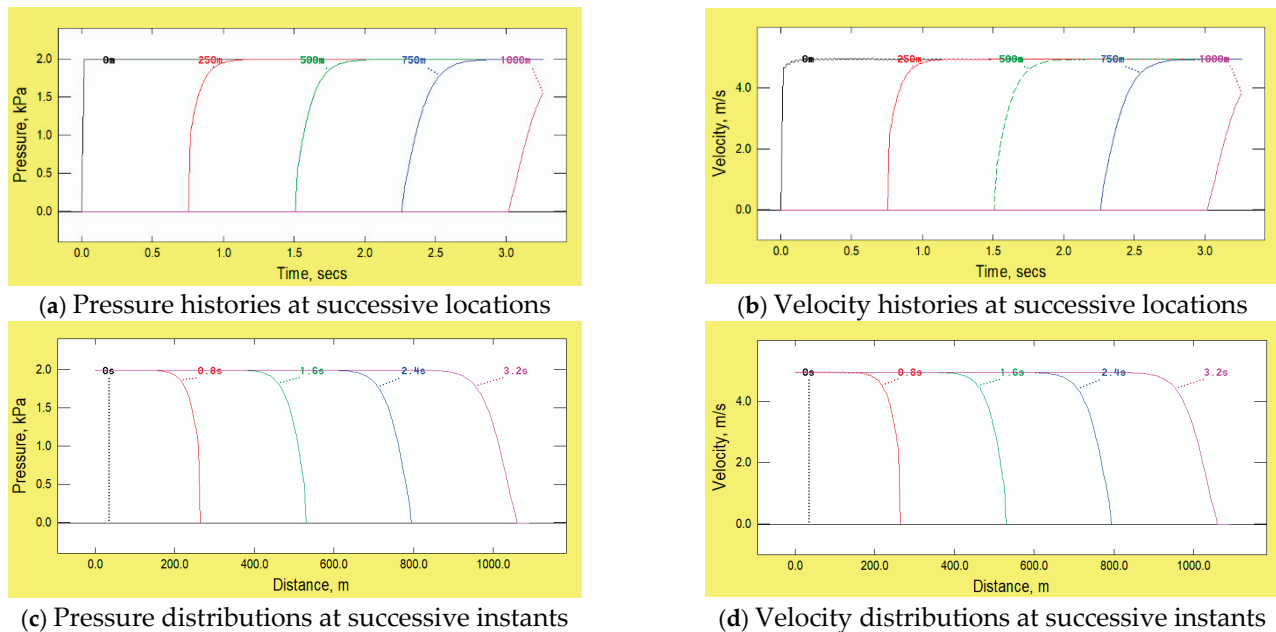


Figure 11. Wave propagation along a duct with lateral (closed) chambers.

11. Example 10: Mach Number and Quasi-Steady Friction

As far as is practicable, each of the above examples illustrates the influence of a single variation from an ideal condition. This is helpful for the purposes of illustrating the different behaviours, but it masks the consequences of interactions between multiple influences that are present simultaneously. Such consequences are illustrated in this final example, in which both the Mach number and quasi-steady skin friction exert significant influence (Figure 12). The only prescribed difference from the example shown in Figure 8 is the existence of skin friction, but the outcomes of the two cases are very different. Firstly, the magnitude of the disturbance reduces strongly as it propagates—for reasons given in the above discussion of the quasi-steady friction simulation for a liquid flow (Figure 2). The decay in the present case is much stronger, but this is to be expected with the larger velocities associated with the higher Mach number.

A feature that might be less expected (at least, it surprised the authors) is the gradual convergence of the conditions in the two cases in the leading parts of the disturbance. This is seen especially clearly in the graphs of velocity. If the corresponding example based on inviscid flow had not been undertaken previously, it would have been difficult to infer the causes of the convergence.

Although the physical behaviour observed in this example is of interest in its own right, the more general deduction is that interactions between independent influences can cause effects that are as strong as the individual influences themselves. Nevertheless, the overall outcome—namely a disturbance that decays as it propagates—is of the same

general nature as several of the examples given above, in which only one parameter had a strong influence on the nature of the propagating disturbance.

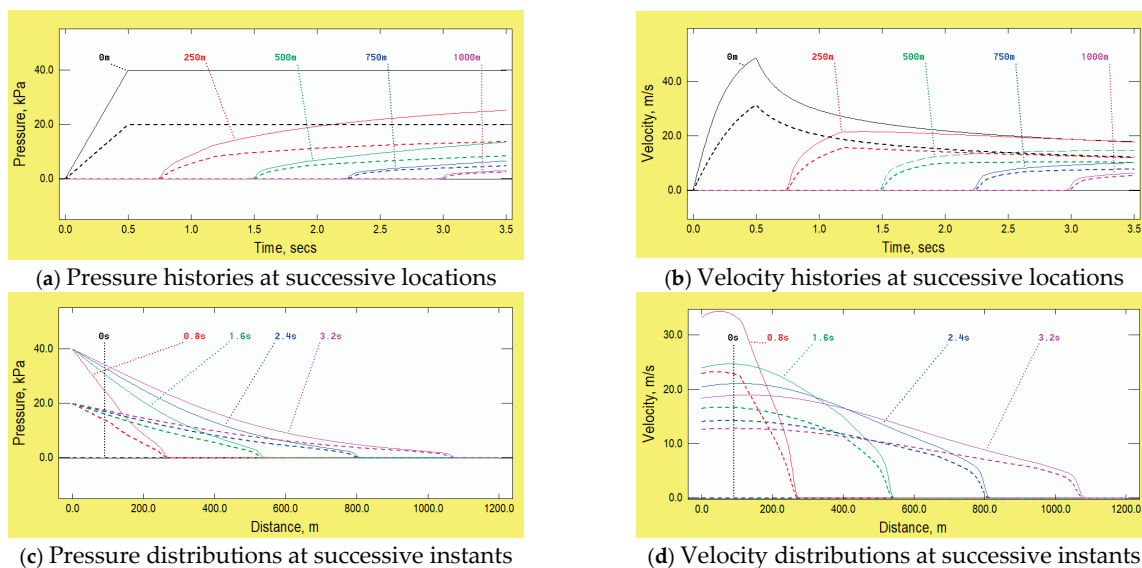


Figure 12. Combined influence of Mach number and quasi-steady friction.

12. Other Phenomena

Although the above examples examine wavefront propagation in a wide variety of contexts, the list is far from exhaustive. Other phenomena that can influence the propagation include, for instance, cavitation in liquids (Bergant et al. [28]), visco-elastic pipes, elevation, non-uniform fluid properties, condensation and evaporation. Each such case has its own specific characteristics that influence how disturbances evolve during propagation, but it will be common for their leading zones to have similar qualitative forms to those presented above.

As indicated in the opening paragraph of Section 6, the influence of radial inertia is neglected in the subsequent discussion. In practice, inertial influences can cause trailing oscillations of wavefronts as illustrated in Figure 13 (Tijsseling et al. [29]) and can lead to dynamic safety factors up to four (Beltman et al. [30]). The first are theoretical (never observed) and the second have led to provocative discussion (Leishear [31]).

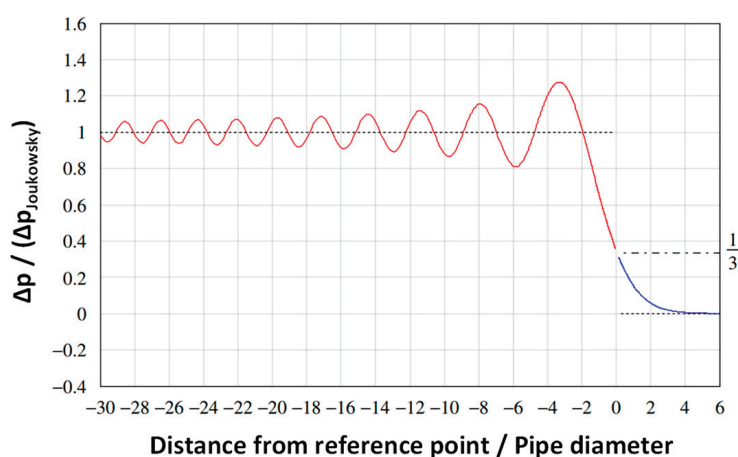


Figure 13. Trailing oscillation due to radial inertia of a pipe wall. The reference point ($1/3$) propagates at constant speed and amplitude.

13. Possible Definitions of ‘Wavefront’

A common characteristic of all the disturbances considered above is that their leading parts propagate in a wave-like manner, albeit often with significant dispersion. In most cases, the extents of their ever-increasing zones of influence are clearly visible in the graphs. That is, the loci of their leading tips are well defined, at least during the early stages of propagation. In some cases, however, the magnitude of the leading part of the disturbance gradually decays so much that the remaining behaviour no longer merits description by a term implying the existence of a defined ‘front’. The following discussion applies only before such conditions are reached.

In the case of Example 1, the magnitude of the disturbance never changes. Also, the rate of change of pressure in the leading ramp increases so slowly that it is not distinguishable in Figure 1. In the corresponding case for a gas (Example 6), it is again true that the *magnitude* does not change, but, in this case, the leading part steepens and has almost become a normal shock in the period simulated. In both cases, it is natural for attention to focus primarily on the evolution of the short ramp-like region even though the length of the region in which flow has changed grows continually. Of course, this is a special case that exists only in idealised simulations that neglect physical influences such as friction and heat transfer. In all cases encountered in conventional practical applications, such influences do exist and changes typically occur continuously along the whole of the region behind the leading part of the disturbance. Nevertheless, any proposed general interpretation of the word ‘wavefront’ will need to be applicable to Examples 1 and 6, as well as to more general cases, so the definition will need to focus on the leading region.

Another aspect of these two examples that is specific to the idealised inviscid constraint is the one-to-one correspondence between the pressure and velocity histories. In more general cases, if the pressure at the boundary is held constant after the initial rise, the velocity at that location will vary in response to whatever happens along the pipe. Such behaviour is seen in all of the other examples above. Likewise, if the velocity were held constant, the pressure would inevitably vary. This distinction is implicitly recognised in the literature insofar as discussion is almost always of *pressure* waves or wavefronts, not of *velocity* or *flow* waves or wavefronts. One reason for focussing on pressure instead of flow for this purpose is that it is usually much easier to obtain good measurements of pressures than of flows. Another reason is that maximum and minimum pressures determine the required pipe strength/class. Nevertheless, a comparison of Examples 3 and 4 (non-uniform area and leaky pipes) demonstrates unequivocally that a knowledge of the pressure history is far from sufficient for inferring flow histories unless adequate supplementary information is also available.

Notwithstanding this potential deficiency, it would be impracticable to require a generic terminology to be dependent on the availability of detailed information of both pressure and flow. Accordingly, the following summary follows the usual practice of describing only pressures. With this proviso, the generic features of disturbances that can be classified as wavefronts are provisionally summarised as:

- The overall shape of the pressure distribution at any instant comprises a leading zone of relatively rapidly varied pressure followed by a longer period of more slowly varied pressure;
- The speed of propagation of the leading part of the disturbance is comparable with the effective speed of sound superimposed on the local state of the flow;
- Notwithstanding the continuously increasing lengths of the disturbed regions, the pressure distributions at successive instants in time share a close family resemblance.

The clarity of these characteristics in the above examples is facilitated by the choice of simple initial conditions and the absence of secondary, externally induced changes. In more general cases, this ideal behaviour would, at best, merely superimpose on the consequences of the other causes of change, and, more likely, would change because of them. Even so, it would not usually prevent the detection of at least the leading part of the propagating disturbance.

Tentative Qualitative Definition

In view of the various factors considered above, it is proposed that the term ‘wavefront’ should normally be reserved to imply *“a localised region of relatively rapid change (i) that is readily seen to be moving at approximately the effective speed of sound relative to the local flow and (ii) that is followed by a less localised region of more slowly varying flow.”*

This proposed definition focuses on the most consistent character of the above examples, whilst also being uninfluenced by details of the initiating event after the generation of the leading parts of the disturbance. It has the advantage of being consistent with the natural instinct of observers to focus on the strongest features of propagating phenomena. More important, by focussing on localised features, it restricts the use of the term to features that are propagating primarily in an inertial manner. Thus, for instance, whereas the definition is meaningful in the early stages of the leaky pipe example in Section 5, it excludes the dominantly dispersive behaviour that characterises later stages.

14. Summary and Conclusions

A lack of clarity in the meaning of the term ‘wavefront’ in the context of unsteady fluid flows in pipes and ducts has triggered a search for a suitable definition. Ideally, a rigorous, quantitative definition would be found, but this is impracticable except in almost trivial cases. Instead, the practical search is for a qualitative definition that is specific and as free from ambiguities as practicable. The search process has been informed by illustrating a wide-range of unsteady flows in which the principal behaviour would typically be described with the aid of the word ‘wavefront’. This has led to a proposed generic definition that encapsulates the observed behaviours without reference to any specific application.

The proposed definition is not restricted to idealised flow conditions. Furthermore, it does not require that the disturbance to which it relates must propagate in unchanged form. Also, it does not require all wave components of the disturbance to propagate in the same direction, provided that the overall disturbance does so. Indeed, it is capable of use in the description of flows that are strongly dissipative and even in flows containing sinks that cause loss of mass.

A decade ago, the authors recognized that a similar lack of clarity in the intended meaning of the word ‘wavespeed’ had caused significant confusion in communications, even between persons with considerable experience in theoretical and practical studies of unsteady flows. Before the confusion was recognised, the ambiguity was a strong barrier to the development of proper understanding, and it triggered the preparation of a paper targeted at minimising the risk of similar confusion amongst other experts and, especially, non-experts (Tijsseling & Vardy [32]). It is hoped that the present paper will have similar benefits regardless of whether the reader has previously detected evidence of miscommunication and its consequences.

In closing, it is emphasised again that a wide range of types of overall flow behaviour has been illustrated above even though attention has been restricted almost exclusively to disturbances resulting from a single triggering event. The number of possible outcomes

will increase hugely when additional factors are present, as will nearly always be the case in practical applications. This is one reason why developing algorithms capable of disentangling multiple contributions to measured signals has been so difficult in the past. Current rapid developments in the design of AI methods offer hope for major advances in this respect. Nevertheless, it is clear that training such software to distinguish between different possible causes of particular measurements will be far from straightforward. In the short term, achieving a reliable identification of isolated wavefronts might be a sufficient challenge. Indeed, even then, it might be necessary to restrict the scope to specific systems instead of seeking methods that are effective in all applications. Either way, developers will need to consider issues such as those addressed herein, regardless of how they choose to define words like ‘wavefront’.

Author Contributions: Conceptualization, A.E.V. and A.S.T.; Methodology, A.E.V. and A.S.T.; Validation, A.E.V.; Formal analysis, A.E.V. and A.S.T.; Investigation, A.E.V. and A.S.T. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Data Availability Statement: The simulations presented in the paper are designed solely to highlight the existence of large qualitative differences between wave propagation in a wide range of cases. In a small number of cases identified in the paper, this purpose has been assisted by using unusual values of particular parameters. Nevertheless, if a reader wishes to obtain digital versions of the graphs in a figure, they should contact the corresponding author.

Conflicts of Interest: The authors declare no conflicts of interest.

Nomenclature

c	speed of sound in stationary, unconfined fluid
c_{eff}	effective speed of sound, allowing for pipe flexibility
D	internal diameter
f	Fanning friction coefficient
Ma	Mach number (U/c)
p	pressure
R	specific gas constant
T	absolute temperature
t	time coordinate
U	fluid velocity
x	axial coordinate
<i>Greek characters</i>	
γ	ratio of principal specific heat capacities
λ	Darcy–Weisbach friction coefficient
ρ	fluid density

References

1. Whelan, P.M.; Hodgson, M.J. *Essential Principles of Physics*, 2nd ed.; John Murray: London, UK, 1978; ISBN 0-7195-3382-1.
2. Wylie, E.B.; Streeter, V.L. *Fluid Transients in Systems*; Prentice-Hall: Englewood Cliffs, NJ, USA, 1993; pp. 400–403.
3. Ferreira, J.P.; Ferras, D.; Covas, D.I.C.; Kapelan, Z. Modelling of pipe filling events in intermittent water supply systems. In Proceedings of the 14th International Conference on Pressure Surges, Eindhoven, The Netherlands, 12–14 April 2023; Technical University of Eindhoven: Eindhoven, The Netherlands, 2023; pp. 311–320.
4. Hou, Q.; Tijsseling, A.S.; Khu, S.T. An SPH-based elastic column model for transients in rapid emptying of pipelines. In Proceedings of the 14th International Conference on Pressure Surges, Eindhoven, The Netherlands, 12–14 April 2023; Technical University of Eindhoven: Eindhoven, The Netherlands, 2023; pp. 283–293.

5. Alastruey, J.; Parker, K.H.; Sherwin, S.J. Arterial pulse wave haemodynamics. In Proceedings of the 11th International Conference on Pressure Surges, Lisbon, Portugal, 24–26 October 2012; Anderson, S., Ed.; BHR Group: Bedfordshire, UK, 2012; pp. 401–442.
6. Xu, C.; Waqar, M.; Louati, M.; Ghidaoui, M.S. High-resolution pipeline leak localization using the side-lobes of the defect detection functional of matched-field processing. *Mech. Syst. Signal Process.* **2024**, *217*, 111529. [CrossRef]
7. Herb, J.; Deeg, P.; Tielkes, T. Assessment of possible sonic boom effects in German high-speed railway tunnels—experimental and numerical data for the wave steepening process. In Proceedings of the 11th International Symposium on Aerodynamics and Ventilation of Vehicle Tunnels, Luzern, Switzerland, 7–9 July 2003; BHR Group: Bedfordshire, UK, 2003; pp. 775–782.
8. Wang, H.; Lei, B.; Bi, H.; Yu, T. Wavefront evolution of compression waves propagating in high speed railway tunnels. *J. Sound Vib.* **2018**, *431*, 105–121. [CrossRef]
9. Ferras, D.; Manso, P.A.; Schleiss, A.J. One-dimensional fluid-structure interaction models in pressurized fluid-filled pipes: A review. *Appl. Sci.* **2018**, *8*, 1844. [CrossRef]
10. Balan, G.S.; Raj, S.A. A review of shock tubes with multitudinous applications. *Int. J. Impact Eng.* **2023**, *172*, 104406. [CrossRef]
11. ThermoTun v8.2. 2025. Available online: www.ThermoTun.com (accessed on 12 March 2025).
12. Leslie, D.J.; Tijsseling, A.S. Travelling discontinuities in waterhammer theory—Attenuation due to friction. In Proceedings of the 8th International Conference on Pressure Surges, The Hague, The Netherlands, 12–14 April 2000; Professional Engineering Publishing: London, UK, 2000; pp. 323–335.
13. Liou, J.C.P. Understanding line packing in frictional water hammer. *ASME J. Fluids Eng.* **2016**, *138*, 081303. [CrossRef]
14. Ludwig, M.; Johnson, S.P. Prediction of surge pressures in long oil transmission lines. *Proc. API Div. Transp.* **1950**, *30*, 62–70.
15. Binnie, A.M. The effect of friction on surges in long pipelines. *Q. J. Mech. Appl. Math.* **1951**, *4*, 330–343. [CrossRef]
16. Burnett, R.B. Predicting and controlling transient pressures in long pipelines. *Oil Gas J.* **1960**, *58*, 153–160.
17. Tarantine, F.J.; Rouleau, W.T. Water-hammer attenuation with a tapered line. *ASME J. Basic Eng.* **1969**, *91*, 341–351. [CrossRef]
18. Yoshizawa, M.; Ando, T. Water-hammer in a tapered pipeline. *Keio Eng. Rep.* **1974**, *27*, 127–140.
19. Jarachi, M.; Zhang, L.; Smith, A.; Whittington, W. Pressure Waves Propagation and Amplification in Tapered Pipes. Available online: <https://www.ssrn.com/abstract=4727372> (accessed on 30 December 2024).
20. Lighthill, M.J. *Waves in Fluids*; Cambridge University Press: Cambridge, UK, 1978.
21. Li, S.J.; Karney, B.W.; Liu, G.M. FSI research in pipeline systems—A review of the literature. *J. Fluids Struct.* **2015**, *57*, 277–297. [CrossRef]
22. Zhao, L.; Cao, Z.; Deng, J. A review of leak detection methods based on pressure waves in gas pipelines. *Measurement* **2024**, *236*, 115062. [CrossRef]
23. Zielke, W. Frequency-dependent friction in transient pipe flow. *J. Basic Eng. ASME* **1968**, *90*, 109–115. [CrossRef]
24. Urbanowicz, K.; Bergant, A.; Stosiak, M.; Deptula, A.; Karpenko, M.; Kubrak, M.; Kodura, A. Water hammer simulation using simplified convolution-based unsteady friction model. *Water* **2022**, *14*, 3151. [CrossRef]
25. Bannister, F.K. Influence of Pipe Friction and Heat Transfer on Pressure Waves in Gases: Effects in a Shock Tube. *J. Mech. Eng. Sci.* **1964**, *6*, 278–292. [CrossRef]
26. Vardy, A.E.; Brown, J.M.B. Influence of ballast on wave steepening in tunnels. *J. Sound Vib.* **2000**, *238*, 595–615. [CrossRef]
27. Liu, F.; Lei, H.; Wei, M.; Sun, H.; Iqbal, M.Y.; Chen, D. Improving the effect of air chambers on micro-pressure waves from tunnel portals: Moderate underdamping. *Tunn. Undergr. Space Technol.* **2024**, *153*, 106016. [CrossRef]
28. Bergant, A.; Simpson, A.R.; Tijsseling, A.S. Water hammer with column separation: A historical review. *J. Fluids Struct.* **2006**, *22*, 135–171. [CrossRef]
29. Tijsseling, A.S.; Lambert, M.F.; Simpson, A.R.; Stephens, M.L.; Vítkovský, J.P.; Bergant, A. Skalák’s extended theory of water hammer. *J. Sound Vib.* **2008**, *310*, 718–728. [CrossRef]
30. Beltman, W.M.; Burcsu, E.N.; Shepherd, J.E.; Zuhail, L. The structural response of cylindrical shells to internal shock loading. *ASME J. Press. Vessel. Technol.* **1999**, *12*, 315–322. [CrossRef]
31. Leishear, R.A. *Fluid Mechanics, Water Hammer, Dynamic Stresses and Piping Design*; Asme Press: New York, NY, USA, 2013; ISBN 978-0791859964.
32. Tijsseling, A.S.; Vardy, A.E. What is wavespeed? In Proceedings of the 12th BHR Group International Conference on Pressure Surges, Dublin, Ireland, 18–20 November 2015; BHR Group: Bedfordshire, UK, 2015; pp. 343–360.

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

A Numerical Investigation into the Performance of Bypass Systems During Filling and Air Removal in Partially Drained Pipelines

Samane Aghaei ¹, Mehdi Hamidi ^{1,*}, Ahmad Malekpour ² and Mohsen Besharat ^{3,*}

¹ Faculty of Civil Engineering, Babol Noshirvani University of Technology, Babol 4714871167, Iran; aghaei.ne@gmail.com

² WSP Canada Inc., 150 Commerce Valley Dr. W, Thornhill, ON L3T 7Z3, Canada; ahmad.malekpour@wsp.com

³ School of Civil Engineering, University of Leeds, Leeds LS2 9JT, UK

* Correspondence: m.besharat@leeds.ac.uk (M.B.); hamidi@nit.ac.ir (M.H.)

Abstract: This study presents an elastic one-dimensional numerical model to simulate the filling process of a large-scale, partially drained pipeline with an undulating profile, incorporating bypass systems. The model uses the Method of Characteristics to solve water hammer equations and integrates the Discrete Gas Cavity Model to capture column separation effects. Validation is performed using two experimental test rigs and comparisons with existing numerical models, showing RMSE values between 1.06 and 7.95. The results highlight three key findings: (1) oversized bypasses generate severe transient pressures; (2) effective air management enables higher filling flow rates, significantly reducing filling time; and (3) bypass lines help dampen pressure fluctuations, with a notable drop in ΔH from 528 m to 6.8 m occurring in stage b, following the release of trapped air. Additionally, this study challenges the practicality of the AWWA's recommended pipeline filling velocity limit of 0.3 m/s, showing that strict adherence to this guideline is often unrealistic for large-scale systems. Overall, the findings emphasize the need for a balanced design approach that reduces transient risks while maintaining operational efficiency in large-scale pipelines.

Keywords: transient analysis; filling; Method of Characteristics; 1D elastic model; air management

1. Introduction

Filling water and wastewater pipelines presents a significant engineering challenge due to the risk of transient pressure surges, which can cause pipe bursts, joint failures, and operational disruptions. These surges arise from the complex interactions between water and entrapped air during rapid filling. Such interactions may trigger severe pressure oscillations, particularly when air release mechanisms are insufficient [1]. Martin [2] was among the first to identify that when pipelines contain entrapped air pockets, rapid pressurization can result in severe pressure spikes—findings later confirmed through both experimental and numerical studies [3–5]. Compressed air pockets amplify pressure levels far beyond the upstream driving force, significantly increasing the likelihood of structural damage [5]. Additionally, incomplete air removal contributes to pressure losses, reduces transmission capacity, and can trigger sudden surges when air is abruptly expelled [6]. Due to these risks, accurate prediction of transient pressures and the optimization of filling

strategies—particularly in large-scale pipelines with undulating profiles, where hydraulic column separation influences flow behavior—remain critical for ensuring operational safety and long-term infrastructure reliability.

Air management is considered one of the most effective methods for controlling transient pressures during pipeline filling [7]. Researchers have developed numerical models to improve the prediction of pressure surges caused by air–water interactions. Zhou et al. [5] used the Rigid Column Model to analyze rapid pressurization in pipelines with air pockets, demonstrating reasonable agreement with experimental data. However, this model tends to overestimate pressure surges, particularly in long pipelines with small air volumes [8]. Introducing water column and pipe elasticity has been shown to reduce these overestimations, highlighting the importance of considering elastic effects in transient modeling. One-dimensional models have continued to evolve, with additional elasticity effects improving predictive accuracy [9,10], yet many still struggle to fully capture pressure dissipation over multiple transient cycles. Lee [11] tackled this issue by introducing a thermodynamic correction that accounts for heat exchange between air and water, significantly improving model performance. The Discrete Air Model (DAM), introduced by Feng et al. [8] improved transient pressure prediction, which enhances accuracy by discretizing air pockets and capturing localized pressure fluctuations.

Although 1D models have grown increasingly sophisticated, some researchers have turned to three-dimensional (3D) computational fluid dynamics (CFD) models to achieve more detailed simulations of air–water interactions [12–14]. CFD models can capture localized flow features, air pocket deformation, and high-gradient pressure zones with remarkable precision. García-Todolí et al. [15] used CFD techniques to evaluate the intake and expulsion capacity of commercial air valves, demonstrating that, with proper mesh resolution and turbulence modeling, air valve performance can be reasonably predicted under controlled conditions. Paternina-Verona et al. [16] developed a digital twin approach using a 3D OpenFOAM model linked with real-time data, showing that CFD tools can effectively simulate transient air release through valves during pipeline filling. However, despite these promising results, the computational cost of 3D simulations remains prohibitively high, particularly for large-scale transmission pipelines with long and undulating profiles. Moreover, accurately representing complex boundary conditions such as dynamic air valve behavior still presents challenges. Consequently, developing computationally efficient models remains essential for optimizing pipeline filling strategies, mitigating transient pressures, and ensuring the reliability of water infrastructure.

One of the most commonly used air management tools in pipeline systems is the air release valve, particularly during filling and drainage operations. However, improper sizing or placement of these valves may result in inadequate air expulsion, pressure imbalances, or operational failures [17,18]. Ramezani and Karney [18] analytically demonstrated that air cavity formation causes differential acceleration in adjacent water columns. When air is released, these columns may collide violently, inducing severe water hammer pressures. The magnitude of these surges depends on factors like air valve orifice size, valve placement, pipeline length, and profile shape. Zhou et al. [5] examined the influence of air release orifice size on transient pressures during rapid pipeline filling. Their study, which involved a horizontal pipeline with an upstream reservoir and a downstream air release orifice, revealed that larger orifices can intensify water hammer effects. The location of trapped air pockets also plays a critical role, with high-point accumulations exacerbating localized surges, particularly in long or undulating pipelines where transient modeling becomes increasingly complex [19]. Malekpour and Karney [20] further investigated rapid filling in undulating pipelines and found that entrapped air can lead to water column

separation at valve locations. Once the air is expelled, the sudden convergence of adjacent water columns generates secondary water hammer pressures, adding another layer of complexity to transient behavior. These findings have been validated through both experimental and numerical studies, reinforcing the challenges associated with air management in pipeline systems [7,21,22].

Another widely used approach to mitigating transient pressures is regulating the filling flow rate. However, no universally accepted computational framework currently determines the optimal rate for different pipeline configurations. In practice, engineers often rely on the American Water Works Association (AWWA) guideline, which suggests a very conservative filling velocity of 0.3 m/s [23]. While this recommendation is widely applied, it is based on simplified assumptions and lacks rigorous validation. Furthermore, strict adherence to this protocol is often impractical, particularly in manually operated pipelines where flow adjustments depend on throttling valves rather than automated control systems. Wang et al. [24] emphasized the importance of bypass systems in balancing pressure mitigation with operational efficiency, yet their effectiveness depends on proper design. Despite their potential, few studies have systematically examined how bypass lines and air valve sizing interact to influence transient pressure behavior.

The aim of this paper is to develop a one-dimensional elastic model that incorporates by-pass and air management strategies in order to simulate transient flow behavior during the filling of large-scale, partially drained pipelines. Unlike previous approaches that often isolate air management or bypass effects, the proposed model simultaneously evaluates their interactions under transient conditions. This integrated treatment enables a more realistic prediction of pressure surges and air release behavior, providing a practical framework for optimizing pipeline filling strategies. The governing equations are constructed by coupling the Method of Characteristics with the Discrete Gas Cavity Model to account for column separation effects. Bypass lines and air valves are incorporated as dynamic boundary conditions to reflect practical system responses. The model is then evaluated through comparisons with both laboratory-scale experiments and two established numerical models. Following validation, the influence of air valve nozzle diameter and bypass pipe size is analyzed to explore their role in pressure regulation. The paper concludes with a summary of key outcomes and implications for real-world pipeline filling strategies.

2. Theoretical Background

The aim of this paper is to develop a one-dimensional elastic model that incorporates bypass and air management strategies in order to simulate transient flow behavior during the filling of large-scale, partially drained pipelines. Unlike previous approaches that often isolate air management or bypass effects, the proposed model simultaneously evaluates their interactions under transient conditions. This integrated treatment enables a more realistic prediction of pressure surges and air release behavior, providing a practical framework for optimizing pipeline filling strategies. The governing equations are constructed by coupling the Method of Characteristics with the Discrete Gas Cavity Model to account for column separation effects. Bypass lines and air valves are incorporated as dynamic boundary conditions to reflect practical system responses. The model is then evaluated through comparisons with both laboratory-scale experiments and two established numerical models. Following validation, the influence of air valve nozzle diameter and bypass pipe size is analyzed to explore their role in pressure regulation. The paper concludes with a summary of key outcomes and implications for real-world pipeline filling strategies.

2.1. Governing Assumptions and Equations

The numerical model is based on the following assumptions [8,24,25]: (1) an elastic one-dimensional formulation is used that accounts for both water compressibility and pipe wall elasticity to enable realistic pressure wave simulation; (2) the friction coefficient is assumed to be a function of the pipe material and remains constant along the pipeline; (3) the air–water interface is assumed to remain perpendicular to the pipe centerline; (4) the compressible entrapped air is assumed to follow a polytropic process with a constant polytropic exponent, ensuring that the air pressure within the air pocket remains uniformly distributed.

The following partial differential equations govern transient flow in pressurized conduit system and are well known as water hammer equations [26,27].

$$\frac{\partial V}{\partial t} + g \frac{\partial H}{\partial x} + \frac{f}{2D} |V| V = 0 \quad (1)$$

$$\frac{\partial H}{\partial t} + \frac{a^2}{g} \frac{\partial V}{\partial x} = 0 \quad (2)$$

where V = velocity, g = gravitational acceleration, H = piezometric head, D = pipe diameter, a = elastic wave speed, f = Darcy–Weisbach friction factor, and x and t = distance and time independent variables respectively.

When pipeline pressure drops to vapor pressure, cavitating flow and column separation occur, complicating hydraulic modeling. In such cases, standard transient flow equations alone cannot fully capture system behavior. A widely used approach to address this is the Discrete Gas Cavity Model (DGCM) [28], which introduces small gas pockets at computational nodes to simulate both cavitation and water hammer effects. At high pressures, these gas pockets remain small and rigid, allowing pressure waves to propagate at the pipe's acoustic speed. However, when the pressure reaches vapor pressure, the gas pockets expand, stabilizing pressure and preventing further wave propagation. Due to its ease of implementation and accuracy, the DGCM is used in this study to effectively analyze column separation.

2.2. Numerical Implementation

The Method of Characteristics (MOC) is a widely used and efficient numerical technique for solving water hammer equations. In this approach, as shown in Figure 1, the computational domain is divided into a grid of small cells, each with equal spatial (Δx) and temporal (Δt) increments. Flow parameters, such as the flow rate and pressure at a given point (P) on the current time step, can be determined by simultaneously solving the positive and negative characteristic equations. This method provides a practical and accurate way to model transient flows in pressurized pipelines.

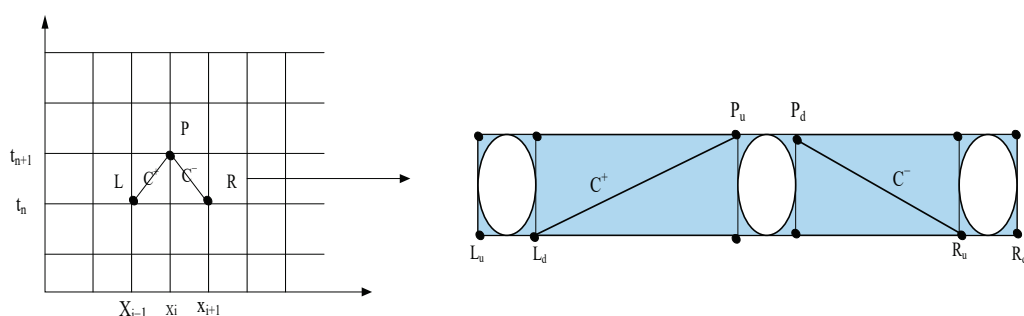


Figure 1. Computational grid in the Method of Characteristics.

To discretize the governing equations, refer to Figure 1, which illustrates a gas pocket within a computational cell. At the given point P on the current time step, four unknowns—the piezometric head, the gas cavity volume, and the flow rates on either side of the gas cavity—can be determined. These are calculated by simultaneously solving four equations: the positive and negative characteristic equations, the mass conservation equation for the gas cavity, and the gas equation of state.

$$C^+ : H_{Pu} = H_{Pd} = C_1 - C_2 Q_{Pu} \quad (3)$$

$$C^- : H_{Pd} = C_3 + C_4 Q_{Pd} \quad (4)$$

$$(\forall_P)_{t+\Delta t} = (\forall_P)_t + [\psi(Q_{Pd} - Q_{Pu})_{t+\Delta t} + (1 - \psi)(Q_{Pd} - Q_{Pu})_t] \times \Delta t \quad (5)$$

$$(\forall_P)_{t+\Delta t} = \frac{P_0^* \alpha_0 \forall_R}{\rho g (H_{Pd} - Z_{Pd} - H_V)} \quad (6)$$

With

$$C_1 = H_{Ld} + \frac{a}{gA} Q_{Ld}; \quad C_2 = \frac{a}{gA} + \frac{f\Delta x}{2gDA^2} |Q_{Ld}|$$

$$C_3 = H_{Ru} - \frac{a}{g} Q_{Ru}; \quad C_4 = \frac{a}{gA} + \frac{f\Delta x}{2gDA^2} |Q_{Ru}|$$

where Q_{Pu} and Q_{Pd} = flow rates at the upstream and downstream side of the gas cavity at point P, respectively, $H_{Pu} = H_{Pd}$ = piezometric head at the upstream and downstream side of the gas cavity at point P, respectively, $Z_{Pu} = Z_{Pd}$ = pipe elevation at the upstream and downstream side of the gas pocket at point P, respectively, Q_{Ld} = flowrate at the downstream side of the gas cavity at point L, H_{Ld} = piezometric head at the downstream side of the gas pocket at point L, Q_{Ru} = flowrate at the upstream side of the gas pocket at point R, H_{Ru} = piezometric head at the upstream side of the gas cavity at point R, A = cross-sectional area of the pipe, ρ = water density, $\forall_P$ = volume of the gas pocket at point P, H_V = gauge vapor pressure head, α_0 = initial gas void fraction, $\forall_R$ = computational cell volume, P_0^* = reference pressure, Δt = computational time step, and t , $t + \Delta t$ = indices referring to the previous and current time steps, respectively.

By substituting Equations (3), (4) and (6) into Equation (5) and performing some algebraic manipulations, the following equation is obtained:

$$(H_{Pd} - Z_{Pd} - H_V)^2 + 2B_1(H_{Pd} - Z_{Pd} - H_V) - B_4 = 0 \quad (7)$$

The analytical solution of the above equation is as follows:

$$H_{Pd} - Z_{Pd} - H_V = -B_1 \left(1 + \sqrt{1 + B_B} \right) \text{ if } B_1 \leq 0 \quad (8)$$

$$H_{Pd} - Z_{Pd} - H_V = -B_1 \left(1 - \sqrt{1 + B_B} \right) \text{ if } B_1 > 0 \quad (9)$$

where

$$B_1 = B_2 C_4 C_2 B_V + 0.5(Z_{Pd} + H_V) - B_2(C_2 C_3 + C_4 C_1); \quad B_B = \frac{B_4}{B_1^2}$$

$$B_2 = \frac{0.5}{C_2 + C_4}; \quad B_V = \frac{(\forall_P)_t}{\psi \Delta t} + \frac{1 - \psi}{\psi} (Q_{Pd} - Q_{Pu})_t; \quad B_4 = \frac{P_0^* \alpha_0 \forall_R B_2 C_4 C_2}{0.5 \rho g \psi \Delta t}$$

In scenarios where there are either large gas volumes at low pressure or small gas volumes at high pressure, Equations (8) and (9) may yield inaccurate results due to the

presence of the radical term, particularly when $|B_B| \ll 1$. To address this issue, a more reliable solution can be obtained by linearizing the original equations, leading to the following modified equations [26]:

$$H_{Pd} - Z_{Pd} - H_V = -2B_1 - \frac{B_4}{2B_1} \quad \text{if } B_1 \leq 0 \quad (10)$$

$$H_{Pd} - Z_{Pd} - H_V = \frac{B_4}{2B_1} \quad \text{if } B_1 > 0 \quad (11)$$

Equations (10) and (11) are applicable when B_B is relatively small, typically below 0.001. For larger values of B_B , Equations (8) and (9) should be used instead to ensure accuracy. Having calculated H_{Pd} , other unknowns, i.e., Q_{Pu} , Q_{Pd} , and $\forall_P$, can be calculated using Equations (3)–(5), respectively.

2.3. Bypass and Air Valve Boundary Condition

In the proposed model, the bypass system, the two-phase flow within the pipe, and the air valve used to expel air are treated as boundary conditions.

Considering Figure 2a, the following equations that are air equation of state, continuity equation in the pipe subject to filling, and energy equation at the bypass, respectively, can be written as follows [26]:

$$P_{air}(\forall_a)_{t+\Delta t} = [m_t + 0.5\Delta t(\dot{m}_t + \dot{m}_{t+\Delta t})]RT \quad (12)$$

$$(H_P - Z_B)_{t+\Delta t} = (1 + K_V) \frac{(Q_P)_{t+\Delta t}^2}{2gA_B^2} + \left(\frac{fL_f}{D} \right) \frac{(Q_P)_{t+\Delta t}^2}{2gA^2} + \left(\frac{P_{air} - P_{atm}}{\rho g} \right)_{t+\Delta t} \quad (13)$$

$$\psi(Q_P)_{t+\Delta t} + (1 - \psi)(Q_P)_t - \frac{\delta x}{\Delta t} = 0 \quad (14)$$

$$\begin{aligned} & \frac{1}{A} \left[(L_f + \delta x)(Q_P)_{t+\Delta t} - L_f(Q_P)_t \right] \\ &= \frac{1}{A^2} \left[\psi(Q_P)_{t+\Delta t}^2 + (1 - \psi)(Q_P)_t^2 \right] - gS_0(\psi\delta x + L_f) \\ & - g\psi \left(H_P - Z_B - \frac{P_{air} - P_{atm}}{\rho g} \right)_{t+\Delta t} \\ & - g(1 - \psi) \left(H_P - Z_B - \frac{P_{air} - P_{atm}}{\rho g} \right)_t + g\psi(L_f + \delta x)S_{t+\Delta t} \\ & + g(1 - \psi)L_fS_t \end{aligned} \quad (15)$$

where

$$(\forall_a)_{t+\Delta t} = (\forall_a)_t + 0.5\Delta t \left[(Q_P)_{t+\Delta t} + (Q_P)_t \right]; \quad S = \frac{Q_P^2}{2gA^2}$$

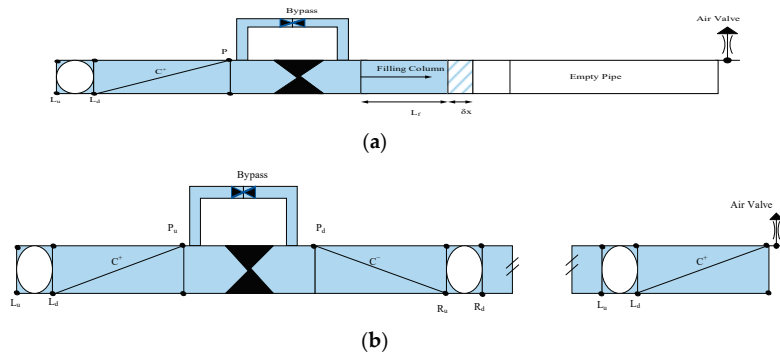


Figure 2. Boundary conditions at bypass location (a) during filling and (b) after filling. (The blue streak represents the filling region, the black triangle indicates the control valve, and the white ovals represent gas cavities).

The air mass flow can be calculated for sonic and subsonic air flows by the following equations [26]:

a. Subsonic air flow in

$$\dot{m} = C_{in} A_{in} \sqrt{7 P_0 \rho_0 \left[\left(\frac{P_{air}}{P_0} \right)^{1.4286} - \left(\frac{P_{air}}{P_0} \right)^{1.714} \right]} \quad P_0 > P_{air} > 0.53 P_0 \quad (16)$$

b. Critical air flow in

$$\dot{m} = C_{in} A_{in} \frac{0.686}{\sqrt{RT_0}} P_0 \quad P_{air} < 0.53 P_0 \quad (17)$$

c. Subsonic air flow out

$$\dot{m} = -C_{out} A_{out} P_{air} \sqrt{\frac{7}{RT} \left[\left(\frac{P_0}{P_{air}} \right)^{1.4286} - \left(\frac{P_0}{P_{air}} \right)^{1.714} \right]} \quad \frac{P_0}{0.53} > P_{air} > P_0 \quad (18)$$

d. Critical air flow out

$$\dot{m} = -C_{out} A_{out} \frac{0.686}{\sqrt{RT}} P_{air} \quad P_{air} > \frac{P_0}{0.53} \quad (19)$$

In the above equations, m_t = initial mass of the air pocket, $\dot{m}_t, \dot{m}_{t+\Delta t}$ = air mass flow rates at time t and $t + \Delta t$, respectively, V_a = air pocket volume, R = gas constant, T = absolute temperature, P_{air} = air absolute pressure, P_{atm} = atmospheric absolute pressure, Q_P = flow at the boundary, K_V = the bypass valve head loss coefficient, Z_B = pipe elevation at the bypass, ψ = time weighing factor, δx = water column advancement in current time step, L_f = filling water column in previous time step, S_0 = pipeline slope, and S = energy grade line slope, ρ_0 = atmospheric air mass density, C_{in} = inlet discharge coefficient, C_{out} = outlet discharge coefficient, A_{in} = air inlet opening area of the valve, and A_{out} = air outlet opening area of the valve.

Equations (12)–(15) can be solved simultaneously to determine the unknowns, Q_P , P_{air} , and δx . The Newton–Raphson method is employed to solve the equations. Once the air pressure and flow rate are calculated, the piezometric head at the boundary can be calculated using Equation (3).

After the air is completely expelled, the pipe is treated like other pipes in the system. A more detailed explanation of this process is provided in Appendix A.

This study follows a two-stage modeling approach. In the first stage, the proposed numerical model is validated using both experimental and numerical benchmarks. The experimental validation is based on data from two setups available in the literature: one involving a pressurized pipeline with an air valve and the other featuring a dead-end pipe configuration. These setups provide critical insight into the model's ability to capture transient behaviors. Additionally, the model is validated against two existing one-dimensional numerical models to assess its accuracy under different simulation conditions.

In the second stage, the validated model is applied to a hypothetical large-scale transmission pipeline with an undulating profile to investigate how bypass design and air valve sizing influence transient pressure behavior during filling.

2.4. Computational Detail

All simulations are carried out using a desktop computer equipped with an Intel® Core™ i7-4702MQ processor (2.2 GHz) and 8 GB of RAM. The numerical model was implemented in Fortran and executed under a Windows 10 operating system.

3. Model Verification

The proposed model is validated by comparing its numerical predictions with both experimental and numerical data. Two key experimental test rigs—Coronado-Hernández et al. [29] and Zhou et al. [30]—are used to evaluate accuracy. Coronado-Hernández et al. examined transient flow behavior in a 7.3 m long, 63 mm diameter pipeline equipped with an air valve at the downstream end. Their tests included three cases, each with an air pocket length (L_a) of 0.96 m and initial gauge pressures (P_0) of 0.5, 0.75, and 1.25 bar. The air valve's orifice size was 3.175 mm, with a discharge coefficient of 0.32, regulating the release of trapped air. Additionally, Zhou et al. [30] investigated transient pressure surges in an 8.862 m long, 40 mm diameter dead-end pipeline, testing four cases combining air pocket lengths (L_a) of 0.3 m and 0.4 m under reservoir pressures (P_r) of 0.08 and 0.12 MPa. These controlled experiments provide critical data for evaluating the model's ability to predict transient pressure variations under different operating conditions. The key characteristics of all experimental cases are summarized in Table 1.

Table 1. Characteristics of experimental cases.

Case No.	P_R (pa)	L_T (m)	L_a (m)	D (m)	D_{Orifice} (mm)
1	50,000	3.88	0.96	0.063	3.175
2	75,000	3.88	0.96	0.063	3.175
3	125,000	3.88	0.96	0.063	3.175
4	80,000	8.826	0.3	0.04	-
5	80,000	8.826	0.4	0.04	-
6	120,000	8.826	0.3	0.04	-
7	120,000	8.826	0.4	0.04	-

In addition to experimental validation, the model's accuracy is assessed against established one-dimensional numerical models, including the Rigid Column Model (RCM) [29] and the elastic Discrete Air Model (DAM) [8]. As illustrated in Figure 3, the proposed elastic model demonstrates higher accuracy compared to the RCM. This improvement is due to the fact that the RCM assumes an infinite wave propagation speed, which is not physically realistic, making the elastic model a more suitable choice for real-world undulating pipelines.

Further analysis in Figure 3 demonstrates that both models accurately estimate peak pressures during the first transient cycle. However, the proposed elastic model provides a more detailed representation of transient behavior under varying air–water interaction conditions. Since peak pressure in the first cycle is a key design factor, the model's ability to capture it with high precision reinforces its relevance for engineering applications. Comparisons with the DAM in Figure 4 further confirm this, as both models exhibit high accuracy in estimating the initial peak—an essential aspect in pipeline design and pressure control. This observation is further supported by the quantitative results presented in Table 2, where the root mean square error (RMSE) values, ranging from 1.06 to 7.95. As shown in Table 2, the difference between the predicted and experimental first-cycle peak pressures is small across both experimental cases, highlighting the model's strong agreement in capturing the most critical pressure response in transient conditions. While

the agreement in the first transient cycle is strong, the model tends to slightly overestimate pressure amplitudes in subsequent cycles. This overestimation can be attributed to the assumption of a constant friction factor, whereas in reality, transient friction effects vary over time [27]. Duan et al. [25] demonstrated that unsteady friction significantly affects laboratory-scale experiments, especially in small-diameter systems. However, in real-world large-scale pipelines, as considered in this study, unsteady friction effects are less pronounced. Another potential source of discrepancy is the uniform air model (UAM) assumption. Feng et al. [8] found that when the $\frac{L}{D}$ ratio exceeds 2000, the UAM and the DAM produce similar results. Given that real-world pipelines typically have large $\frac{L}{D}$ ratios, assuming uniform air distribution can be considered a reasonable approximation. Additionally, neglecting heat transfer effects in numerical models can introduce deviations beyond the first transient cycle, as complex thermodynamic interactions occur within the air pocket during compression and expansion [31]. Future improvements in transient modeling should incorporate these effects to enhance predictive accuracy across diverse operating conditions.

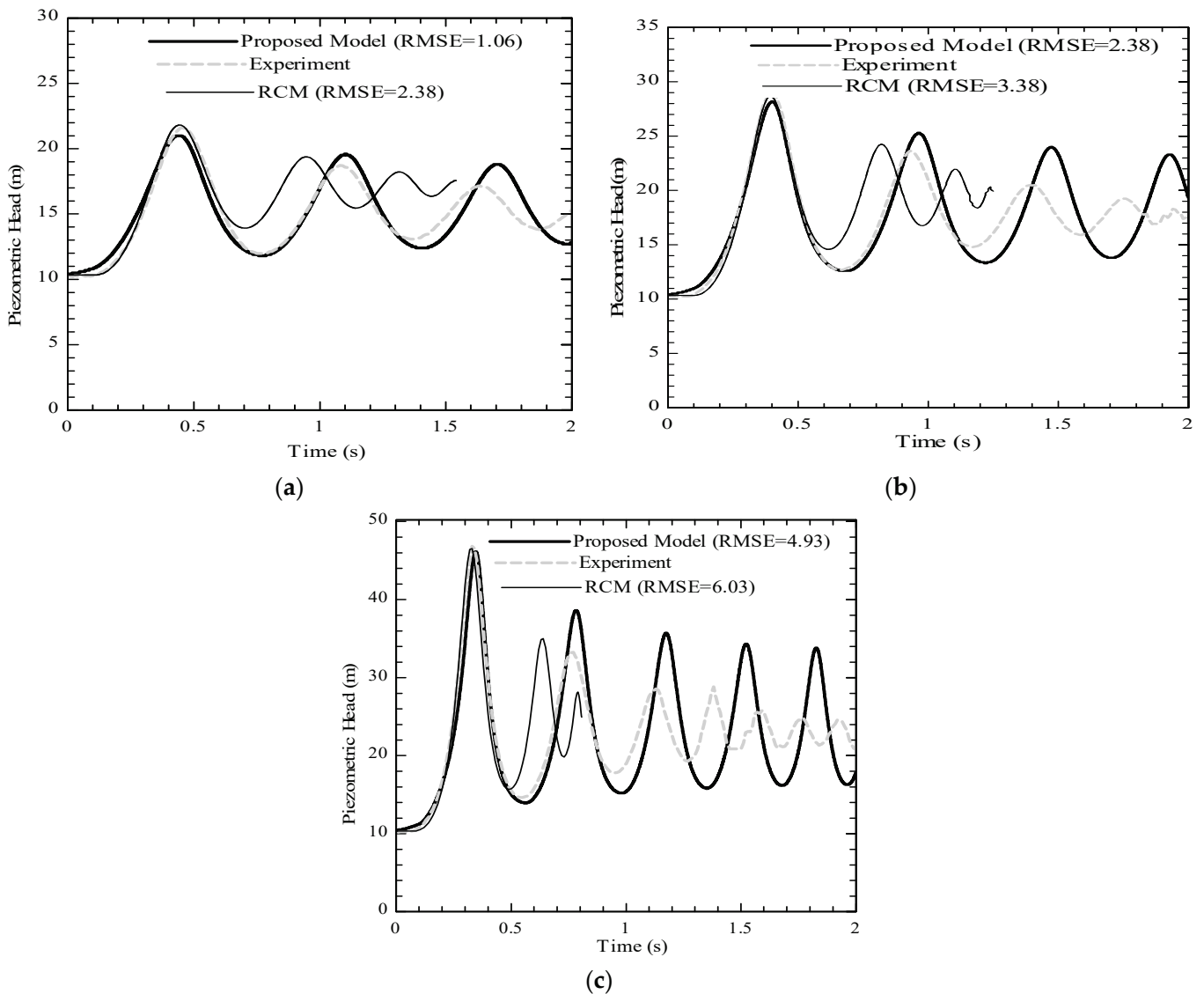


Figure 3. Comparison between calculated and experimental pressure pattern [29]; (a) case 1: $P_0 = 0.5$ bar; (b) case 2: $P_0 = 0.75$ bar; and (c) case 3: $P_0 = 1.25$ bar.

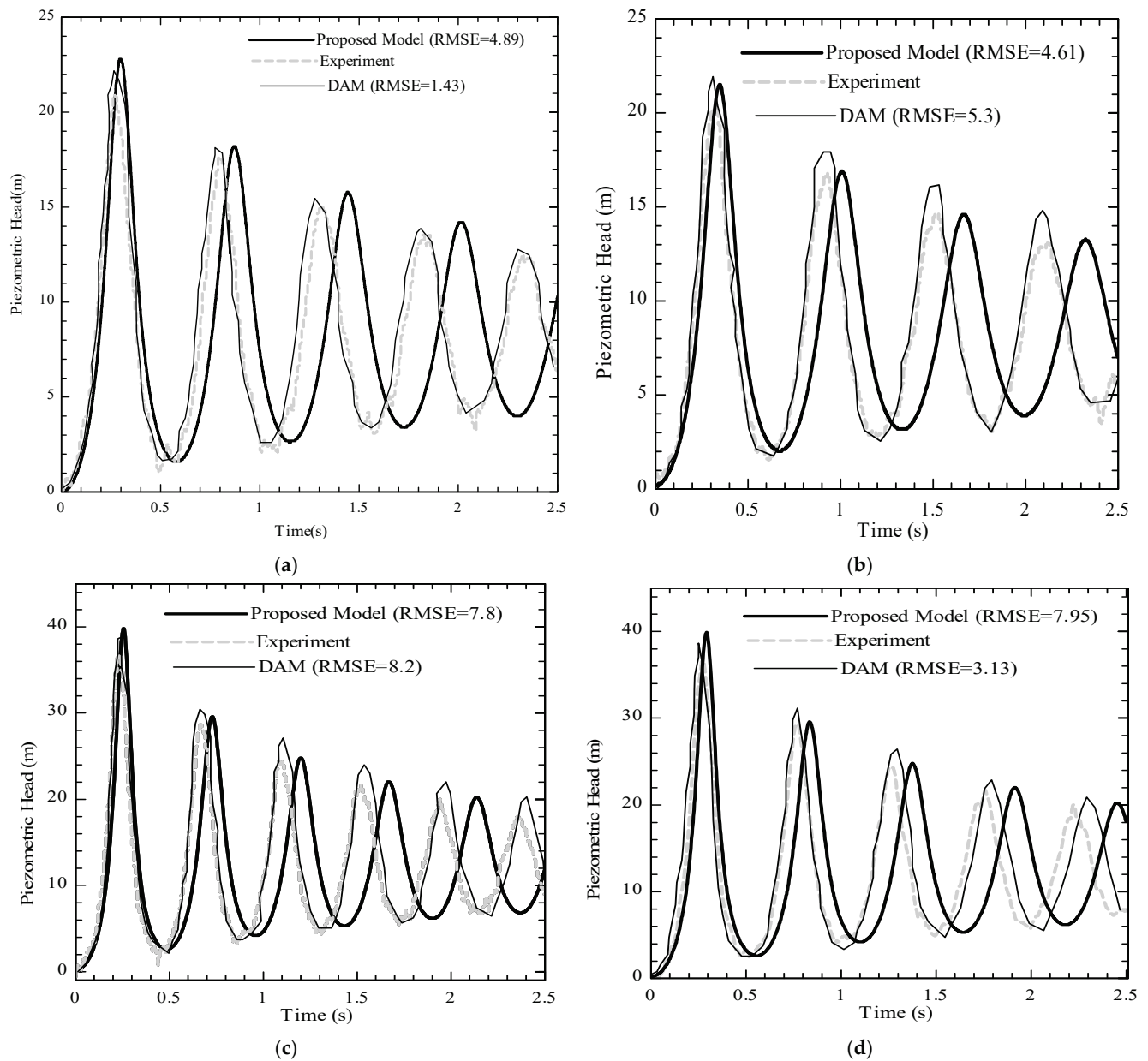


Figure 4. Comparison between calculated and experimental pressure pattern [30]; (a) case 4: $L_d = 0.3$ m $P_r = 0.08$ MPa; (b) case 5: $L_d = 0.4$ m $P_r = 0.08$ MPa; (c) case 6: $L_d = 0.3$ m, $P_r = 0.12$ MPa; and (d) case 7: $L_d = 0.4$ m, $P_r = 0.12$ MPa.

Table 2. Quantitative error metrics between model predictions and experimental data.

Case No.	Experimental Peak Pressure (m)	Proposed Model Peak Pressure (m)	RSME
1	21.64	21	1.06
2	29.06	28.2	2.38
3	46.82	46.1	4.93
4	20.79	22.8	4.89
5	21.5	20.24	4.61
6	39.8	36.63	7.8
7	36.63	39.9	7.95

As previously discussed, the highest transient pressure occurs during the first cycle, making its accurate prediction essential for pipeline design. Since one-dimensional models exhibit high accuracy in peak pressure estimation, they remain a reliable tool for transient flow analysis.

The length and diameter of experimental pipelines typically differ from those of real-world pipeline systems due to spatial and budgetary constraints in laboratory design. However, this study focuses on large-scale operational pipelines and proposes an optimal filling strategy for partially drained systems.

4. Numerical Results and Discussion

Numerical simulations are performed on a hypothetical pipeline (Figure 5), isolated from a larger system for maintenance. While hypothetical, this setup is designed to be broadly applicable to real-world pipeline operations. Most experimental studies focus on short, horizontal pipelines with upstream reservoirs and dead-end boundaries, limiting their relevance to large-scale, undulating systems. In this study, a long, undulating pipeline is considered, featuring an upstream reservoir at a constant elevation of 1100 m and a water level of 5 m. In this context, “large-scale” refers to pipelines with a length exceeding 10 km and a diameter above 0.8 m, which are commonly used in municipal and industrial water transmission systems.

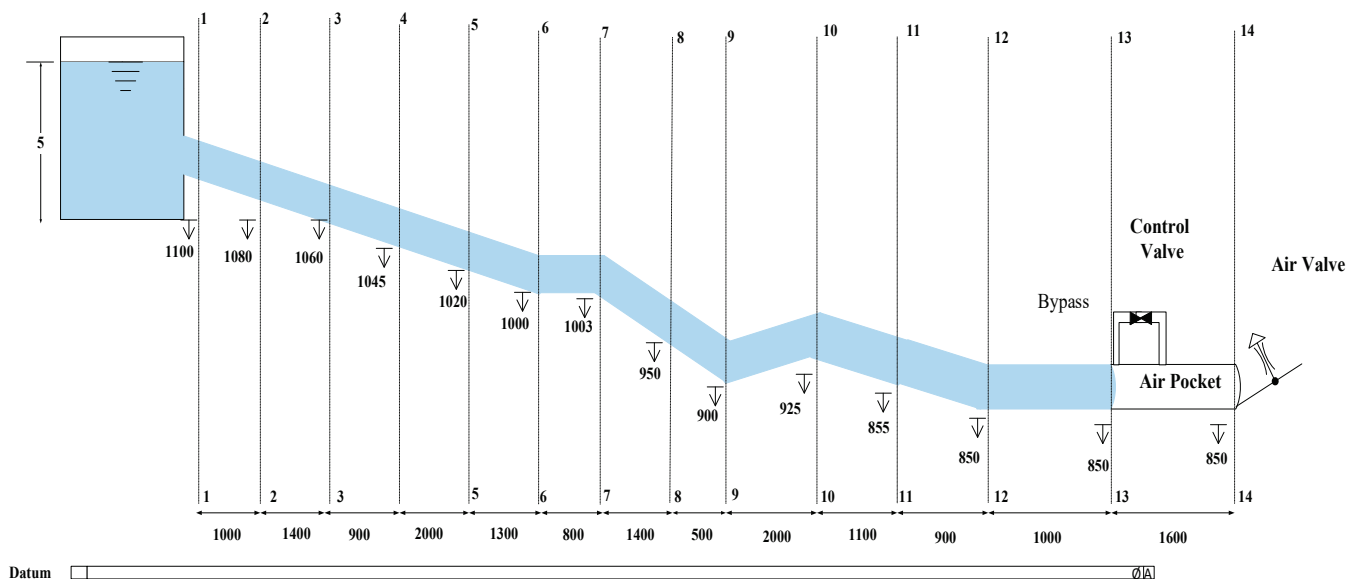


Figure 5. Schematic drawing of the hypothetical pipe system utilized in the numerical exploration.

The system consists of 13 pipes in series with a total length of 15,900 m, an internal diameter of 0.9 m, an acoustic wave speed of 1000 m/s, and a Darcy–Weisbach friction factor of 0.018. The last 1600 m of the pipeline is drained and isolated using a control valve, with a bypass system and air valve regulating air release during filling.

The control valve opening time plays a critical role in mitigating negative pressures during filling. As the control valve gradually opens, pressure waves propagate upstream, reflecting back as positive waves, reducing the intensity of negative pressures at the valve location. To prevent severe negative pressures, technical recommendations suggest that the valve opening time should be at least $2L/a$, allowing reflected waves to reach the valve before it fully opens. Given the 14,300 m water column and a wave speed of 1000 m/s, a 30 s valve opening duration is adopted, ensuring wave reflection reaches the valve before

full opening. This duration balances pressure control with manual operational feasibility. Extending this time beyond 30 s is not feasible due to practical constraints.

To systematically assess the transient response of the system, ten different cases are analyzed, each varying in bypass diameter (D_B) and air valve nozzle diameter (D_o), as presented in Table 3.

Table 3. Characteristic of cases for transient analysis.

Case No.	D_B (m)	D_o (mm)
1	-	150
2	-	50
3	-	15
4	0.3	150
5	0.3	50
6	0.3	15
7	0.1	150
8	0.1	50
9	0.1	30
10	0.1	15

4.1. Air Expulsion Mechanism and Pressure Surge Formation

The pipeline filling process follows a two-stage air expulsion mechanism, where interactions between the trapped air pocket and the advancing water column drive transient pressure fluctuations.

- **Stage a:** Compression and Expansion of Trapped Air

Before complete air expulsion, the trapped air pocket compresses and expands, influencing transient pressure dynamics. A gradual air release cushions the advancing water column, dampening pressure surges, whereas rapid air expulsion accelerates water flow, increasing the potential for intense pressure fluctuations.

- **Stage b:** Water Hammer and Pressure Wave Propagation

Once the air pocket is fully expelled, the high-speed water column abruptly decelerates at the pipeline's end, triggering a water hammer effect. This results in a sudden pressure surge, which propagates upstream as negative pressure reflections.

The severity of these pressure surges depends on air valve nozzle size and bypass line diameter, both of which regulate air release rates and control water column velocity. To accurately evaluate transient pressures during filling, this study investigates the effects of air valve nozzle size and bypass line diameter through numerical simulations, assessing their role in mitigating pressure surges at each stage.

4.2. Effect of Air Valve Nozzle Size on Transient Pressures

The air valve nozzle size plays a critical role in transient pressure control. A larger nozzle accelerates air expulsion, limiting compression–expansion cycles in Stage a. However, this rapid air release increases water column velocity, intensifying water hammer effects in Stage b. Figure 6b shows that for a 150 mm nozzle, the air pocket is fully expelled within 411 s, leading to an accelerated water column. Since air has a lower density than water, it escapes rapidly upon reaching the nozzle, intensifying water hammer effects [29]. In this scenario, air is quickly expelled from the system without experiencing significant compression–expansion cycles in Stage a. However, once the last air pocket exits, the high-speed water column impacts the pipe end, causing sudden deceleration and generating a

severe water hammer effect. This overpressure propagates upstream, reflecting as negative pressure surges and amplifying transient oscillations throughout the system. Conversely, reducing the nozzle size restricts air release, prolonging air retention and increasing localized pressure before expulsion. The entrapped air pocket acts as a cushion, gradually absorbing transient pressures through repeated compression–expansion cycles. As shown in Figure 7a, higher air pocket pressure reduces the kinetic energy of the water column, thereby mitigating water hammer surges in Stage b. However, before the air pocket exits in Stage a, excessive air compression can still generate high transient pressures, which propagate upstream as negative reflections, leading to significant pressure differentials across the system.

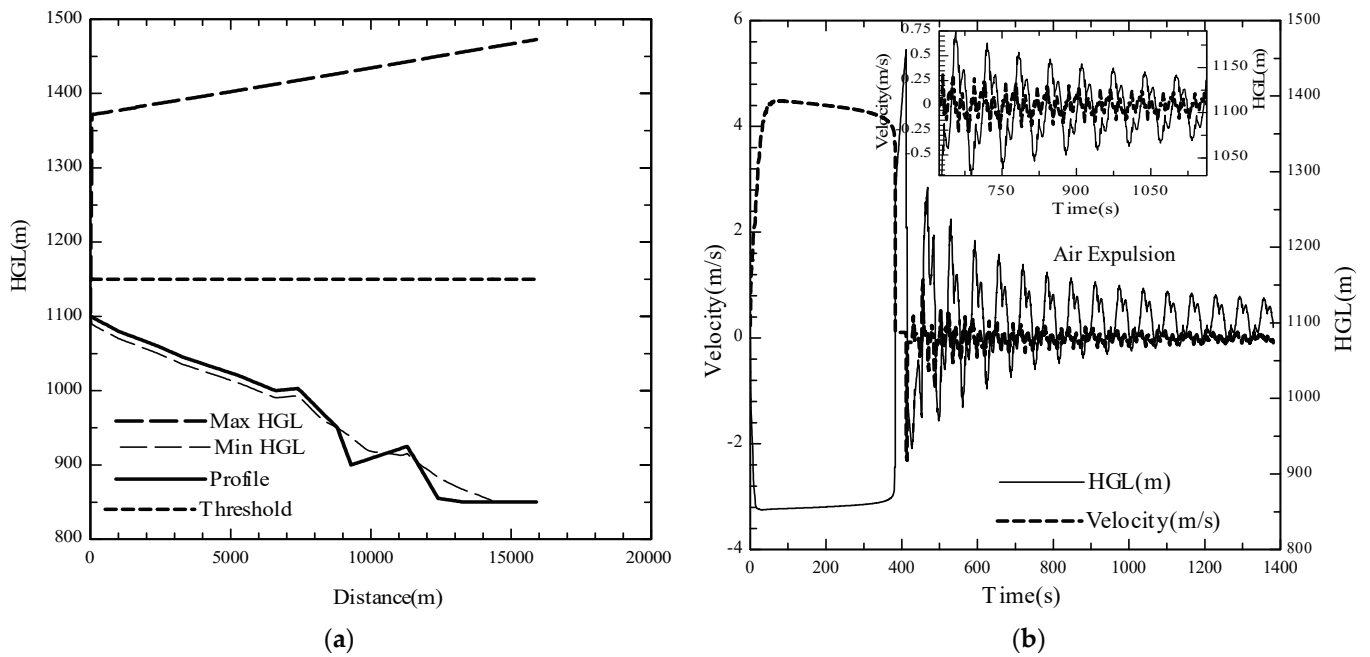


Figure 6. Transient response of the system for case 1 without bypass line and $D_o = 150$ mm: (a) velocity and piezometric head time history at the valve location and (b) Max. and Min. piezometric head envelopes developed across the pipe.

To quantify transient responses under varying air release conditions, this study evaluates key hydraulic parameters including air pocket exit time (T_{ex}), pressure increase rate (PIR), pressure reduction rate (PRR), and water hammer pressure (ΔH) in both stages of the filling process—indicated with indices 1 and 2—as defined in Equations (20) and (21) and Table 4. Table 4 presents these results for cases 1–3, where no bypass line is implemented. The findings indicate that while reducing the air valve nozzle diameter has minimal effect on negative pressure mitigation, it significantly lowers peak overpressures, reducing overall water hammer severity. However, despite this reduction in Stage b, peak pressures in Stage a remain considerably high due to excessive air compression before release.

$$PIR = \frac{H_{Max} - (Elevation)_{Bypass}}{H_{Static}} \quad (20)$$

$$PRR = \frac{H_{Min} - (Elevation)_{Bypass}}{H_{Static}} \quad (21)$$

Table 4. Hydraulic parameters without considering the bypass line in cases 1 to 3.

Case No.	$T_{ex}(s)$	PIR_1 (%)	PRR_1 (%)	$\Delta H_1(m)$	PIR_2 (%)	PRR_2 (%)	$\Delta H_2(m)$
1	411.08	92.52	0.88	229.1	244.68	33.48	528
2	568.05	165.04	0.96	410.2	122.56	83.96	96.5
3	2167.51	171.6	0.96	426.6	103.36	100.64	6.8

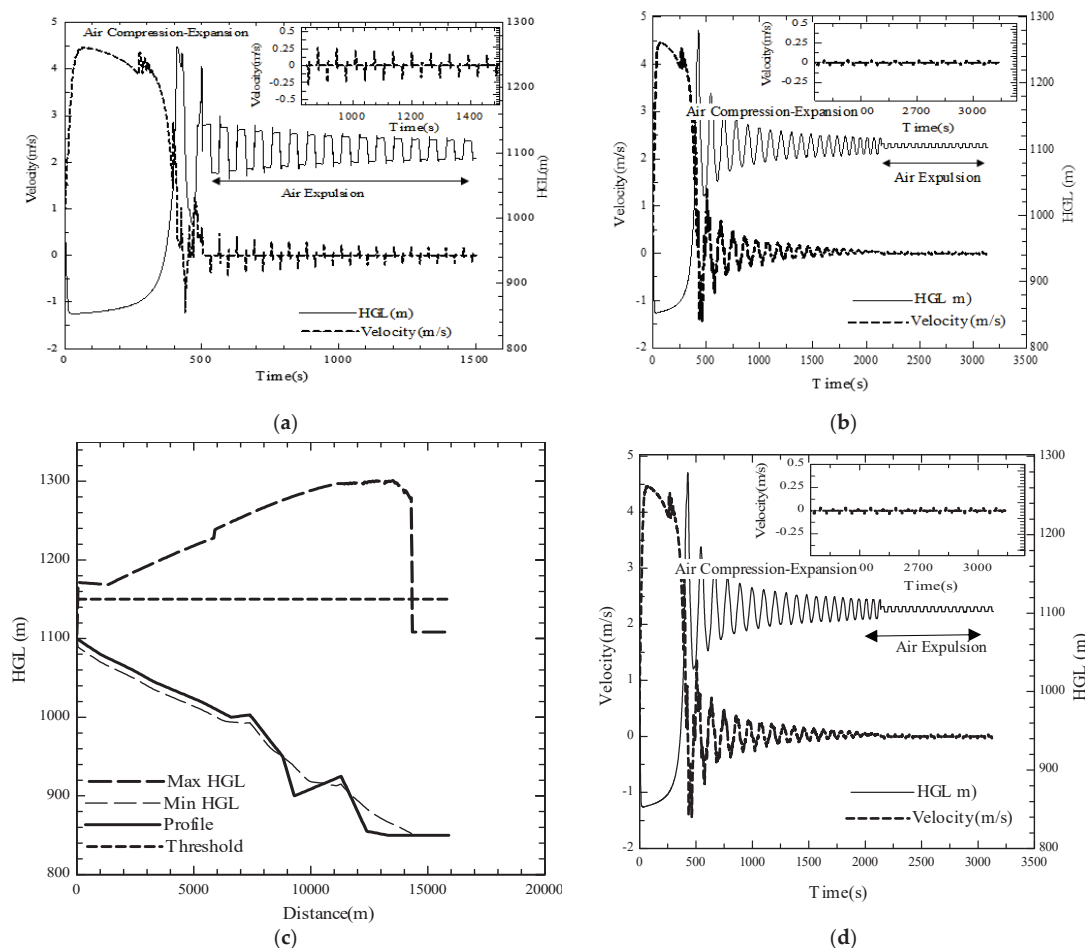


Figure 7. Transient response of the pipeline for cases 2 without bypass line, $D_o = 50$ mm and 3 without bypass line, $D_o = 15$ mm: (a) Max. and Min. piezometric head of the system case 2; (b) velocity and piezometric head time history at the valve location for case 2; (c) Max. and Min. piezometric head of the system for case 3; and (d) velocity and piezometric head time history at the valve location of the system for case 3.

Figures 6 and 7 further demonstrate that reducing the nozzle diameter slows the water column after air expulsion, preventing a sharp pressure rise. While prolonged air retention slightly increases the total filling time, Table 4 confirms that in all tested cases, the air pocket is fully expelled, supporting the feasibility of this approach. The use of smaller nozzles aligns with AWWA recommendations, as it prevents sudden, explosive air release. However, to effectively control both overpressures and negative pressures, a bypass system should be incorporated to provide balanced pressure regulation throughout the pipeline.

4.3. Combined Effect of Bypass Line and Air Valve Nozzle on Pressure Regulation

Although reducing the air valve nozzle size helps mitigate Stage b pressure surges, significant pressure differentials still develop along the pipeline due to excessive air compression

before expulsion. This occurs because a smaller nozzle restricts air outflow, leading to pressure rise and large variations before the air pocket is fully expelled. To further regulate transient pressures, an additional scenario is examined where a bypass line is integrated alongside air valve optimization to allow for better control of the valve diameter. A bypass line with a smaller diameter than the main pipeline is used to reduce water column velocity, thereby limiting its impact on the downstream boundary. This approach not only mitigates overpressures but also reduces pressure wave reflections. As shown in Figure 8, decreasing the bypass diameter reduces positive pressures at the downstream end, thereby minimizing their reflection back toward the reservoir. This dampening effect lowers the overall pressure differential between maximum and minimum system pressures, significantly attenuating water hammer surges. Additionally, as seen in Figure 8, incorporating a bypass line reduces water column velocity from approximately 4.4 m/s in a system without a bypass to about 0.54 m/s when a bypass with a 0.1 m diameter is used. Based on Joukowski's formulation, reducing water column velocity directly lowers pressure differentials, effectively reducing water hammer intensity.

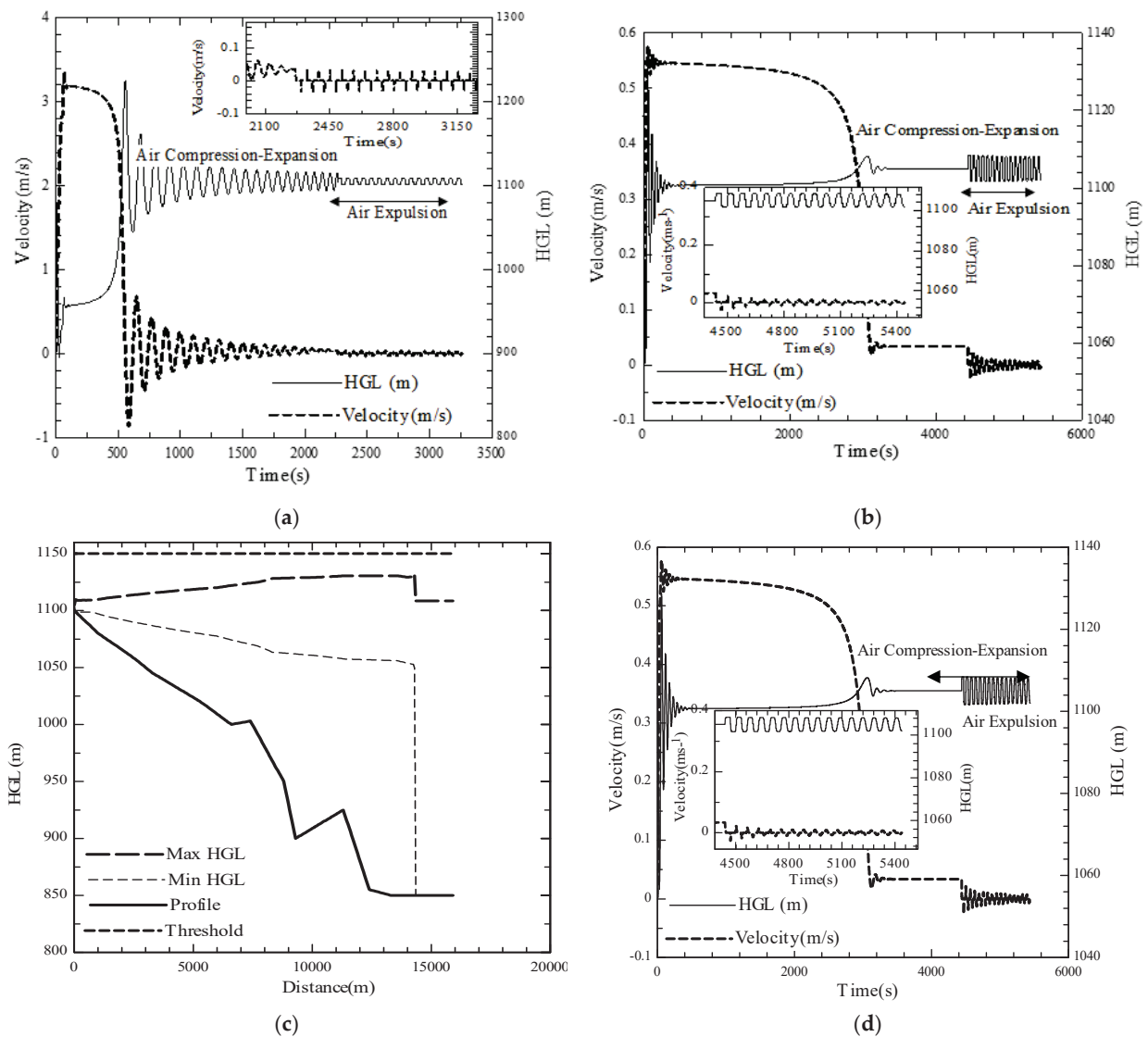


Figure 8. Transient response of the pipeline for cases 6 $D_B = 0.3$ m, $D_o = 15$ mm and 10 $D_B = 0.1$ m, $D_o = 15$ mm: (a) Max. and Min. piezometric head for case 6; (b) velocity and piezometric head time history at the valve location for case 6; (c) Max. and Min. piezometric head for case 10; and (d) velocity and piezometric head time history at the valve location for case 10.

To systematically compare different cases, the bypass diameter (D_B) and air valve nozzle diameter (D_o) were normalized using the main pipeline diameter (D), yielding the dimensionless parameters $\frac{D_B}{D}$ and $\frac{D_o}{D}$. As illustrated in Figure 9, the maximum and minimum piezometric heads are analyzed based on these parameters. The findings indicate that decreasing $\frac{D_o}{D}$ enhances the damping effect of the trapped air, reducing overpressures after air expulsion. However, excessively small air valve nozzles prolong air retention time, amplifying transient pressures in the early stages of filling. Similarly, decreasing $\frac{D_B}{D}$ reduces the velocity of the advancing water column, thereby reducing both maximum overpressures and negative pressure reflections near the upstream reservoir.

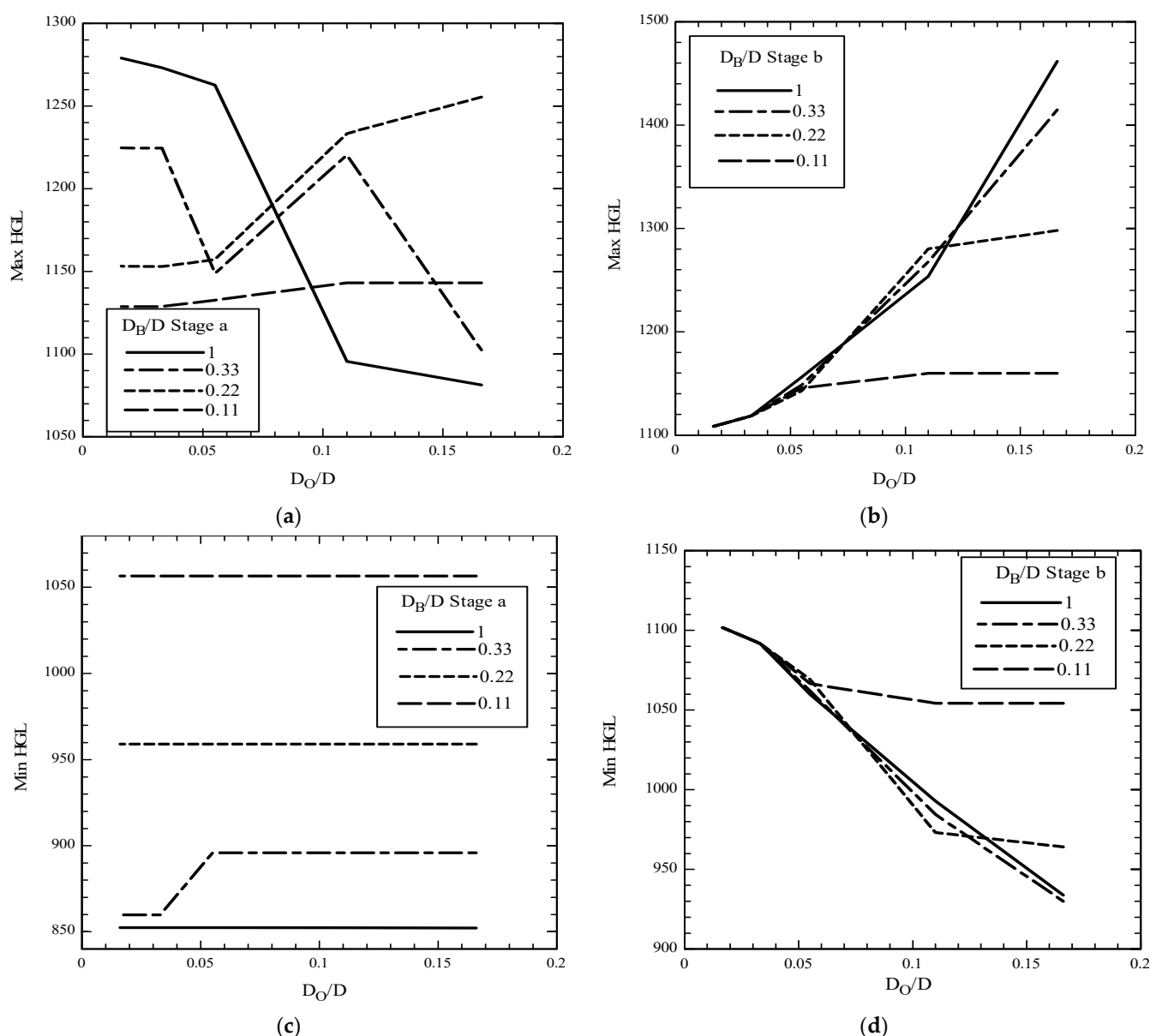


Figure 9. Piezometric head in the last pipe for nozzles with different diameters and bypass lines of various dimensions: (a) Maximum piezometric head in Stage a; (b) maximum piezometric head in Stage b; (c) minimum piezometric head in Stage a; and (d) minimum piezometric head in Stage b.

A quantitative comparison of these effects is provided in Table 5, demonstrating that combining an optimized bypass line with a reduced air valve nozzle size significantly improves transient pressure regulation. A comparison of cases 4–10 reveals that when

a properly sized bypass is implemented, both overpressures and negative pressures are substantially reduced. However, selecting an appropriate bypass diameter is crucial to balancing pressure mitigation with practical implementation constraints. While a smaller bypass diameter slightly increases overall filling time, this delay is negligible compared to the significant reduction in risks associated with entrapped air pockets.

Table 5. Hydraulic parameters for cases 4 to 10.

Case No.	T_{ex} (s)	PIR_1 (%)	PRR_1 (%)	ΔH_1 (m)	PIR_2 (%)	PRR_2 (%)	ΔH_2 (m)
4	556.045	101	18.36	518.9	225.92	31.96	484.9
5	660.52	149.92	18.36	328.9	119.48	84.8	86.7
6	2264.632	149.88	18.36	328.8	103.36	100.64	6.8
7	2976.299	117.24	82.64	86.5	123.92	81.64	105.7
8	3012.334	113.04	82.64	76	118.32	86.6	79.3
9	3202.519	111.48	82.64	72.1	107.44	96.68	26.9
10	4447.075	111.48	82.64	72.1	103.36	100.64	6.8

4.4. Evaluating AWWA Filling Recommendations

The AWWA recommends two primary strategies to minimize the risks associated with trapped air pockets:

- Maintaining a slow filling velocity of 0.3 m/s;
- Using properly sized air valves to facilitate gradual air release under high pressures.

However, implementing a strictly controlled 0.3 m/s filling velocity is often impractical due to human error and limitations in precise flow control mechanisms. As demonstrated in previous sections, incorporating a properly sized bypass effectively reduces water column velocity. However, even with a 0.1 m bypass diameter, the calculated filling velocity remains 0.54 m/s, exceeding the AWWA guideline. To meet the recommended velocity, the bypass diameter would need to be reduced further to 0.08 m, which introduces operational challenges.

As the simulation results demonstrate, utilizing a bypass pipe with a diameter of 0.1 m effectively eliminates all negative pressures along the pipeline, maintains maximum pressures within safe thresholds, and ensures the complete expulsion of entrapped air pockets. Therefore, reducing the bypass diameter further—such as to 0.08 m, though not explicitly modeled in this study—would extend the total filling time to approximately 5914.24 s and introduce additional implementation challenges due to the reduced cross-sectional flow area. While this scenario technically complies with the AWWA’s recommended filling velocity, it is not considered necessary from an engineering perspective, given that the 0.1 m bypass already achieves full air removal and significantly mitigates pressure surges. Hence, further reduction in bypass diameter offers limited additional benefit while imposing avoidable operational constraints.

5. Conclusions

Optimizing the pipeline filling process requires a careful balance between minimizing negative pressures, mitigating water hammer surges, and ensuring operational feasibility. Given the cost and complexity of physical testing, this study employs a high-precision one-dimensional numerical model to simulate a long, undulating pipeline, ensuring its applicability to real-world systems. The model’s accuracy is validated through comparisons with experimental data from two setups—one featuring an air valve and the other a dead-

end pipeline. Additionally, its reliability in predicting transient flow behavior is assessed against two numerical approaches—the Rigid Column Model and the Discrete Air Model—demonstrating its effectiveness in capturing key pressure dynamics.

This study analyzes the influence of air valve nozzle sizes and bypass pipe diameters on transient pressures. Numerical trials based on AWWA recommendations indicate that larger air valve nozzles allow for rapid air expulsion without inducing a significant pressure rise in Stage a. However, once air is fully released, the high-velocity water column impacts the pipeline's end, triggering severe water hammer pressures in Stage b. Conversely, reducing the nozzle diameter prolongs air retention, introducing compression–expansion cycles that help moderate surges after air expulsion. While this approach effectively reduces sudden pressure spikes, it also results in elevated peak pressures in the early filling stage, generating negative pressure waves propagating upstream.

To address these challenges, an additional scenario incorporates a bypass line. The results indicate that introducing a bypass with a smaller diameter than the main pipeline effectively reduces water column velocity, thereby lowering transient pressures throughout the system. This approach significantly dampens overpressures and mitigates negative pressure reflections near the upstream reservoir. However, while reducing the bypass diameter effectively controls transient pressures, excessive reduction extends the filling time, posing practical constraints for implementation.

Overall, this study establishes a reliable numerical foundation for optimizing pipeline filling strategies. The findings underscore the importance of balancing transient pressure control with operational feasibility, ensuring that proposed solutions remain both effective and applicable to real-world pipeline systems. Building on the validated one-dimensional model presented here, future efforts could explore the development of a coupled modeling framework, in which the main pipeline is represented through a 1D elastic model, while the downstream unfilled section—where air–water interactions dominate—is resolved using a localized 3D approach. This type of hybrid modeling would allow for a more detailed treatment of transient air pocket behavior, including non-uniform air distribution, complex interface dynamics, and thermodynamic effects such as heat transfer between compressed air, water, and pipe walls.

Author Contributions: Conceptualization, methodology, validation, and formal analysis, S.A., A.M. and M.B.; Writing—review and editing, S.A., M.H., A.M. and M.B.; Supervision, M.H., A.M. and M.B. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Data Availability Statement: Data used in this research can be shared with other researchers upon reasonable request.

Conflicts of Interest: Author Ahmad Malekpour was employed by the company WSP Canada Inc. The remaining authors declare that the research was conducted in the absence of any commercial or financial relationships that could be construed as a potential conflict of interest.

Abbreviations

V	velocity
g	gravitational acceleration
H	piezometric head
D	pipe diameter
f	Darcy–Weisbach friction factor
x	distance
t	time

Q_{Pu}	flow rates at the upstream side of the gas cavity
Q_{Pd}	flow rates at the downstream side of the gas cavity
H_{Pu}	piezometric head at the upstream side of the gas cavity
H_{Pd}	piezometric head at the downstream side of the gas cavity
H_V	gauge vapor pressure head
$\forall_R$	computational cell volume
P_0^*	reference pressure
m_t	initial mass of the air pocket
$\dot{m}$	air mass flow rate
$\forall_a$	air pocket volume
R	gas constant
T	absolute temperature
P_{air}	air absolute pressure
P_{atm}	atmospheric absolute pressure
K_V	the bypass valve head loss coefficient
Z_B	pipe elevation at the bypass
ψ	time weighing factor
δx	water column advancement in current time step
L_f	filling water column in previous time step
S_0	pipeline slope
S	energy grade line slope
ρ_0	atmospheric air mass density
C_{in}	inlet discharge coefficient
C_{out}	outlet discharge coefficient
A_{in}	air inlet opening area of the valve
A_{out}	air outlet opening area of the valve

Appendix A. Equations for the Last Pipe After Filling

After the air completely evacuates, the pipe would be treated like other pipes in the system. In the new condition, the boundary condition, as shown in Figure 2b, needs adjustment. As long as the pressure is higher than atmospheric pressure and the air volume is zero, the flow rate at the downstream end of the pipe would be zero, and the piezometric head (H_E) can be calculated by using the positive characteristic equation, as follows

$$H_E = \frac{C_1}{C_2} \quad (A1)$$

However, when the pressure falls below the atmospheric pressure, the air valve comes into action, and the flow and piezometric head needs to be calculated using the following equations:

$$P_{air} \{ (\forall_a)_t + 0.5\Delta t [(Q_E)_{t+\Delta t} + (Q_E)_t] \} = [m_t + \Delta t(\dot{m}_t + \dot{m}_{t+\Delta t})] RT \quad (A2)$$

Combining Equation (A2) with the positive characteristic equation results in the following non-linear equation that contains just one unknown, P_{air} .

$$P_{air} \left\{ (\forall_a)_t + 0.5\Delta t \left[\frac{C_1}{C_2} - \frac{P_{air}}{\rho g} - \frac{Z_E}{\rho g} + (Q_E)_t \right] \right\} = [m_t + \Delta t(\dot{m}_t + \dot{m}_{t+\Delta t})] RT \quad (A3)$$

where Q_E = flow rate at the downstream side of the pipe and should be calculated using the Newton–Raphson method.

As shown in Figure 2b, the bypass valve is treated like an internal boundary condition. Combining the energy equation on either side of the bypass valve with the positive and neg-

ative characteristic equation results in Equations (A5) and (A6), which can be analytically solved to calculate flow rate at the bypass valve.

$$H_{Pu} = H_{pd} + K_V \frac{|Q_{pu}|Q_{pu}}{A_B^2} \quad \text{Energy equation} \quad (\text{A4})$$

$$Q_{Pu}^2 + BBQ_{Pu} + CC = 0 \quad \text{For positive flow} \quad (\text{A5})$$

$$Q_{Pu}^2 - BBQ_{Pu} - CC = 0 \quad \text{For reverse flow} \quad (\text{A6})$$

where BB and CC can be calculated from the following equations

$$BB = \frac{2gA^2(C_2 + C_4)}{K_V}; \quad CC = \frac{2gA^2(C_1 - C_3)}{K_V}$$

References

1. Zhou, F.; Hicks, F.E.; Steffler, P.M. Observations of Air–Water Interaction in a Rapidly Filling Horizontal Pipe. *J. Hydraul. Eng.* **2002**, *128*, 635–639. [CrossRef]
2. Martin, C.S. Entrapped air in pipelines. In Proceedings of the 2nd International Conference on Pressures Surges, London, UK, 22–24 September 1976; Stephens, H.S., King, A.L., Stapleton, C.A., Eds.; BHRA Fluid Engineering, Cranfield: Bedford, UK, 1976.
3. Lee, N.H.; Martin, C.S. Experimental and Analytical Investigation of Entrapped Air in a Horizontal Pipe. In Proceedings of the 3rd ASME/JSME Joint Fluids Engineering Conference, San Francisco, CA, USA, 18–23 July 1999; pp. 1–8.
4. Nakamura, K.; Tomita, T. Water hammer caused by air in a pump started with the discharge valve closed. In Proceedings of the 3rd ASME/JSME Joint Fluids Engineering Conference, New York, NY, USA, 18–23 July 1999.
5. Zhou, F.; Hicks, F.E.; Steffler, P.M. Transient Flow in a Rapidly Filling Horizontal Pipe Containing Trapped Air. *J. Hydraul. Eng.* **2002**, *128*, 625–634. [CrossRef]
6. Falvey, H.T. *Air–Water Flow in Hydraulic Structures*; U.S. Department of the Interior, Water and Power Resources Service, Engineering and Research Center: Alexandria, VA, USA, 1980.
7. Lingireddy, S.; Wood, D.J.; Zloczower, N. Pressure surges in pipeline systems resulting from air releases. *J. Am. Water Works Assoc.* **2004**, *96*, 88–94. [CrossRef]
8. Feng, R.L.; Zhou, L.; Besharat, M.; Xue, Z.J.; Li, Y.J.; Chen, Q.X.; Hu, Y.Y.; Lu, Y.Q. Discrete air model for large scale rapid filling process contained entrapped air. *Eng. Appl. Comput. Fluid Mech.* **2024**, *18*, 2428423. [CrossRef]
9. Zhou, L.; Pan, T.; Wang, H.; Liu, D.; Wang, P. Rapid air expulsion through an orifice in a vertical water pipe. *J. Hydraul. Res.* **2019**, *57*, 307–317. [CrossRef]
10. Zhou, L.; Liu, D.; Karney, B. Investigation of Hydraulic Transients of Two Entrapped Air Pockets in a Water Pipeline. *J. Hydraul. Eng.* **2013**, *139*, 949–959. [CrossRef]
11. Lee, N.H. *Effect of Pressurization and Expulsion of Entrapped Air in Pipelines*; Georgia Institute of Technology: Atlanta, GA, USA, 2005.
12. Huang, B.; Fan, M.; Liu, J.; Zhu, D.Z. CFD Simulation of Air–Water Interactions in Rapidly Filling Horizontal Pipe with Entrapped Air. In Proceedings of the World Environmental and Water Resources Congress, Virtual, 1–7 July 2021; American Society of Civil Engineers: Reston, VA, USA, 2021; pp. 495–507.
13. Zhou, L.; Wang, H.; Karney, B.; Liu, D.; Wang, P.; Guo, S. Dynamic Behavior of Entrapped Air Pocket in a Water Filling Pipeline. *J. Hydraul. Eng.* **2018**, *144*, 04018045. [CrossRef]
14. Zhou, L.; Liu, D.; Ou, C. Simulation of Flow Transients in a Water Filling Pipe Containing Entrapped Air Pocket with VOF Model. *Eng. Appl. Comput. Fluid Mech.* **2011**, *5*, 127–140. [CrossRef]
15. García-Todolí, S.; Iglesias-Rey, P.L.; Mora-Meliá, D.; Martínez-Solano, F.J.; Fuertes-Miquel, V.S. Computational determination of air valves capacity using CFD techniques. *Water* **2018**, *10*, 1433. [CrossRef]
16. Paternina-Verona, D.A.; Coronado-Hernández, O.E.; Pérez-Sánchez, M.; Ramos, H.M. Real-Time Analysis and Digital Twin Modeling for CFD-Based Air Valve Control During Filling Procedures. *Water* **2024**, *16*, 3015. [CrossRef]
17. Escaramela, M.; Burrows, R.; Little, M.; Murray, S. *Air Problems in Pipelines*; HR Wallingford Ltd.: Wallingford, UK, 2005.
18. Ramezani, L.; Karney, B. Water Column Separation and Cavity Collapse for Pipelines Protected with Air Vacuum Valves: Understanding the Essential Wave Processes. *J. Hydraul. Eng.* **2017**, *143*, 04016083. [CrossRef]

19. Apollonio, C.; Balacco, G.; Fontana, N.; Giugni, M.; Marini, G.; Piccinni, A.F. Hydraulic transients caused by air expulsion during rapid filling of undulating pipelines. *Water* **2016**, *8*, 25. [CrossRef]
20. Malekpour, A.; Karney, B.W. Profile-Induced Column Separation and Rejoining during Rapid Pipeline Filling. *J. Hydraul. Eng.* **2014**, *140*, 04014054. [CrossRef]
21. Lee, T.S. Air Influence on Hydraulic Transients on Fluid System With Air Valves. *J. Fluids Eng.* **1999**, *121*, 646–650. [CrossRef]
22. Lee, T.S.; Leow, L.C. Numerical study on the effects of air valve characteristics on pressure surges during pump trip in pumping systems with air entrainment. *Int. J. Numer. Methods Fluids* **1999**, *29*, 645–655. [CrossRef]
23. AWWA. *Air Release, Air/Vacuum Valves and Combination Air Valves (M51)*; American Water Works Association: Denver, CO, USA, 2001; ISBN 1613001819.
24. Wang, L.; Wang, F.; Karney, B.; Malekpour, A. Numerical investigation of rapid filling in bypass pipelines. *J. Hydraul. Res.* **2017**, *55*, 647–656. [CrossRef]
25. Duan, H.-F.; Ghidaoui, M.S.; Lee, P.J.; Tung, Y.K. Relevance of Unsteady Friction to Pipe Size and Length in Pipe Fluid Transients. *J. Hydraul. Eng.* **2012**, *138*, 154–166. [CrossRef]
26. Streeter, V.L. *Fluid Transients*; University of Michigan: Ann Arbor, MI, USA, 1978; ISBN 0-07-072187-4.
27. Chaudhry, M.H. *Applied Hydraulic Transients*; Springer: New York, NY, USA, 1979; ISBN 0442215177.
28. Simpson, A.R.; Bergant, A. Numerical Comparison of Pipe-Column-Separation Models. *J. Hydraul. Eng.* **1994**, *120*, 361–377. [CrossRef]
29. Coronado-Hernández, Ó.E.; Besharat, M.; Fuertes-Miquel, V.S.; Ramos, H.M. Effect of a Commercial Air Valve on the Rapid Filling of a Single Pipeline: A Numerical and Experimental Analysis. *Water* **2019**, *11*, 1814. [CrossRef]
30. Zhou, L.; Cao, Y.; Karney, B.; Vasconcelos, J.G.; Liu, D.; Wang, P. Unsteady friction in transient vertical-pipe flow with trapped air. *J. Hydraul. Res.* **2020**, *59*, 820–834. [CrossRef]
31. Graze, H.R. A Rational Thermodynamic Equation for Air Chamber Design. In Proceedings of the 3rd Australasian Conference on Hydraulics and Fluid Mechanics, Sydney, Australia, 25–29 November 1968; pp. 57–61.

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

A Single Water Pipeline Design Considering Pressure-Dependent Consumptions: A New Perspective

Mariangel Puerta-Zurita ¹, Juan J. Villero-Guerra ¹, Oscar E. Coronado-Hernández ², Vicente S. Fuertes-Miquel ^{3,*} and Manuel Saba ¹

¹ Civil Engineering Program, Universidad de Cartagena, Cartagena 130001, Colombia; mpuertaz@unicartagena.edu.co (M.P.-Z.); jvillero1@unicartagena.edu.co (J.J.V.-G.); msaba@unicartagena.edu.co (M.S.)

² Instituto de Hidráulica y Saneamiento Ambiental, Universidad de Cartagena, Cartagena 130001, Colombia; ocoronadoh@unicartagena.edu.co

³ Departamento de Ingeniería Hidráulica y Medio Ambiente, Universitat Politècnica de València, 46022 Valencia, Spain

* Correspondence: vfuertes@upv.es

Abstract: Current design techniques for water installations are based on pressure-independent consumption. However, certain types of consumption depend on pressure, most notably water leakages, which occur in all water distribution systems. All water installations are prone to water leakages. To mitigate this issue during the design stage of water pipelines, manuals and local regulations recommend a percentage of water losses for sizing pipelines and using them with pressure-independent consumption. This research develops a practical procedure for sizing single water pipelines by considering the effect of pressure-dependent consumption, as in actual pipelines. The findings of this research offer water utilities a new perspective on pipeline design, which is essential for developing new tools to create digital twins. A new methodology for designing pipe diameters is developed in this research. A single pipeline, 2000 m long, is used for the analysis with an emitter coefficient of $0.00829 \text{ m}^3/\text{s}/\text{m}^{0.5}$. The proposed methodology focuses on calculating energy losses to maintain the required minimum pressure in the system. A sensitivity analysis is conducted on key parameters such as the emitter coefficient, absolute roughness, and the resistance coefficient of a regulating valve. The results of the proposed methodology are compared with the current design (using pressure-independent consumption), revealing differences of up to 28% in calculating water losses.

Keywords: leaks; leakage coefficient; digital twins; single pipe; sizing; water losses

1. Introduction

Water leakage is a volume of water escaping from hydraulic installations, which refers to seepage occurring throughout water supply systems, representing losses that depend on the average pressure within a conduit. Leakages in water distribution networks can be classified into several key factors: pipe failures, pressure fluctuations, cyclic loading (leading to fatigue), the age and condition of the infrastructure, environmental influences, and operational practises.

Understanding actual water losses in pipelines is vital for water utilities. The existence of two types of water losses is recognised, namely apparent and real [1–3], where the former refers to those reflected by inaccuracies in measurements or unauthorised actions, and the

real ones correspond to leaks found in the pipeline itself. The exponent coefficient can range from 0.5 to 2.5. A value of 0.5 has been used in several studies. Table 1 shows the results of the emitter coefficient, internal pipe diameter, pipe length, and water losses based on data from several publications. In many cases, water losses exceed 50%.

Table 1. Ranges of main parameters in several publications.

Authors	K_f ($\text{m}^3/\text{sm}^{0.5}$)		Diameter (mm)		Pipe Length (m)		Water Losses (%)	
	<0.01	≥ 0.01	<300	≥ 300	<10,000	$\geq 10,000$	<50%	$\geq 50\%$
Niebuhr et al. (2019) [4]	✓			✓	✓			✓
Sophocleous et al. (2017) [5]	✓			✓		✓		✓
Aksela et al. (2009) [6]		✓		✓		✓	✓	
Ulanicki et al. (2000) [7]	✓		✓		✓		✓	
AL-Washali et al. (2020) [8]		✓		✓	✓			✓
Saldarriaga and Salcedo (2015) [9]		✓		✓		✓	✓	
Deyi et al. (2014) [10]		✓	✓			✓		✓
Coronado-Hernández et al. (2024) [11]	✓			✓	✓			✓
Fuertes-Miquel et al. (2024) [12]	✓		✓		✓			✓
Puust et al. (2010) [13]	✓			✓		✓	✓	
Maskit and Ostfeld (2014) [14]	✓		✓		✓		✓	
Samir et al. (2017) [15]	✓			✓	✓		✓	
García-Ávila et al. (2019) [16]	✓		✓			✓	✓	
Pezzinga and Pititto (2005) [17]	✓			✓		✓	✓	

Ulanicki et al. (2000) [7] utilised pressure-reducing valves (PRVs) to decrease water leakage, demonstrating how leakage is influenced by emitter and exponent coefficients. In their analysis, water losses of up to 30% were observed. PRVs have proven to be an effective strategy for reducing water leakage; therefore, these devices should be considered in pipeline sizing and understanding the processes through which water leakage occurs [18]. The analysis and detection of water leakages in distribution systems have been thoroughly explored. Modelling using extended period simulation has been conducted and tested, considering both high- and low-demand periods. The trend of algorithms for the calibration of emitter and exponent coefficients according to different scenarios has been analysed, including using PRVs [19]. The reduction in water losses by applying control pressure has been assessed. The search for optimal configurations for PRVs for reducing water losses was analysed by studying case studies [10,16]. The modified orifice equation in combination with a variable speed pump was also considered in hydraulic installations. The BABE (bursts and background estimates) and NMF (minimum night flow) methods were employed to calculate and detect water leakages. These methods can be utilised to enhance hydraulic performance and achieve greater efficiency [8]. In addition, uncertainty and precision in water distribution networks have been assessed. Water losses can occur even in steel pipes, which have many applications due to their high resistance [20–22]. Improving the optimal performance of hydraulic devices is of utmost importance in water systems [23,24]. Computational Fluid Dynamics and mathematical models are techniques employed in complex scenarios to analyse the water phase [25–27]. The method of using pumps as turbines to manage low flow rates was introduced to avoid excessive pressure, generating energy and reducing leakage. This approach has been utilised as an optimisation tool in hydraulic systems [18]. Recently, the Rigid Water Model was applied to predict water leakages by considering

system inertia, demonstrating that this modelling approach provides a more accurate assessment of water leakages compared to extended period simulation [11,12].

The current literature in this field focuses on the development of digital twins for actual water installations [18,19,28,29], which consists of several components, as shown in Figure 1. A digital twin must comprise a robust platform and a reliable hydraulic model. The platform typically includes a Geographic Information System (GIS), pressure transducers, other sensors throughout the hydraulic network, and a SCADA system. The hydraulic model should be capable of performing extended-period simulations and account for system inertia during rapid valve manoeuvres. Water utilities aim to implement digital twins to collect real-time data, which can be used to detect both real and apparent water losses. These models provide a powerful tool for developing strategies to reduce water losses [30,31]. The hydraulic model accounts for water losses as pressure-dependent consumption to ensure a reliable tool. To optimise hydraulic systems, water utilities are increasingly adopting digital twins to explore and implement strategies for reducing these leakages.

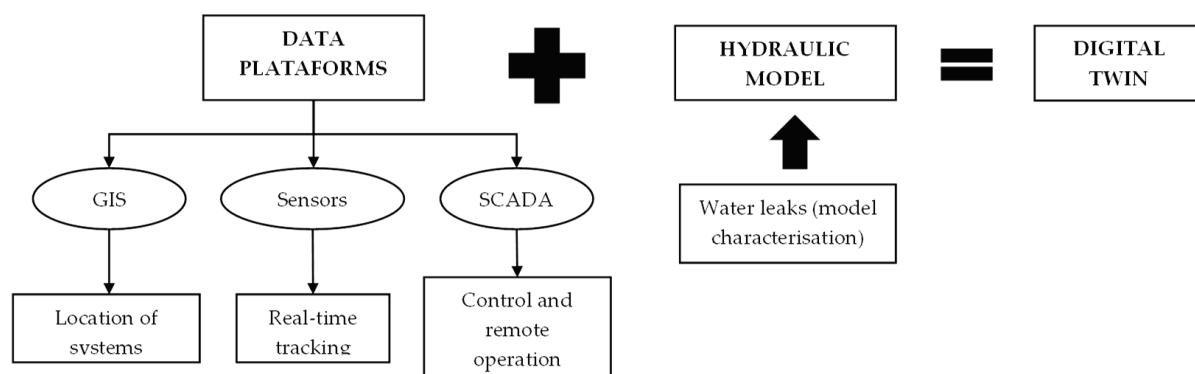


Figure 1. Current digital twin model.

The design of water supply systems depends on local regulations regarding the minimum required pressure at nodes, internal pipe diameter specifications, water demand calculations, the placement of hydraulic devices, and their characteristics, among other factors. However, when calculating water demands, the regulations recommend incorporating a percentage for water losses, usually included in domestic demand figures. For example, Colombia recommends a water loss percentage of 25% during the design phase of some components of water supply networks [32]. This approach, commonly applied during the design phase, does not account for calculating water losses as pressure-dependent consumption, overlooking the actual effect occurring in real pipelines that should be considered in pipeline design.

It is crucial to mention that water losses significantly impact the optimal design of these systems. In this context, this research proposes the design of single water pipelines that account for leakage as pressure-dependent consumption, a factor often overlooked in current pipeline strategies (traditional methods). The proposed methodology introduces a novel approach for achieving optimal sizing of single water pipelines based on prior knowledge of emitter and exponent coefficients specific to water installations. Various factors, including construction practises in water infrastructure, policies for pipeline rehabilitation, response times to leakages, and other operational and maintenance considerations, influence these coefficients. As explained in this research, the leakages must be evaluated based on calibration data by water utilities. This underscores the importance, as evidenced by this research, of accounting for water leaks during the design phase of distribution networks,

thereby enhancing the accuracy of current designs compared to traditional methods that overlook water losses as pressure-dependent consumption.

2. Materials and Methods

The following step-by-step process was developed to carry out this research, as described in the methodology shown in Figure 2. The proposed method consists of three main components as follows:

- Current design:** The single-pipeline design employs the continuity equation, energy loss calculations, and the Darcy–Weisbach equation to determine internal pipe diameters through optimisation. This process assumes a percentage of water losses based on local regulations or the expertise of engineers and designers.
- Design considering leakages:** This research incorporates the water balance framework proposed by the International Water Association (IWA) for sizing single pipelines. Additionally, the methodology involves calibrating the emitter coefficient using data from existing pipelines with similar characteristics, which can then be applied to design new water infrastructure (see Section 2.1).
- Optimisation process and numerical resolution:** Finally, the developed methodology introduces a flowchart to address the problem systematically (see Section 2.3).

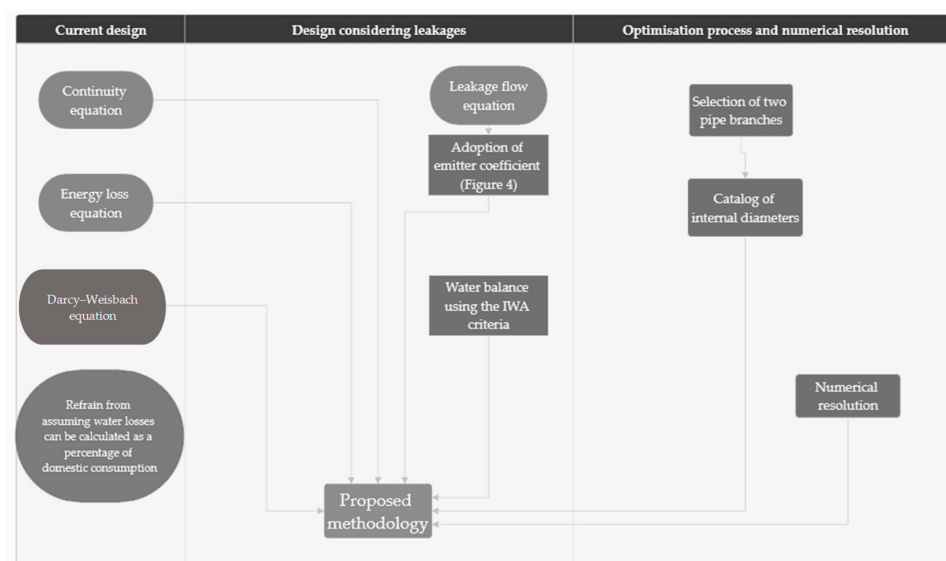


Figure 2. Methodology used in this research.

2.1. Proposed Methodology

A single water pipeline begins and terminates at two nodes separated by total length. Current methodologies for pipeline sizing typically compute two diameters to ensure a minimum pressure at the terminal node (see Figure 3). Water leakages are commonly accounted for as a percentage of total water losses [32], varying depending on local regulations and practises.

This section introduces the proposed methodology for pipeline design, which explicitly incorporates the effects of water leakages. The numerical resolution is carried out using an algorithm developed in this research. The methodology assumes that water utilities have calibrated emitter and exponent coefficients for existing pipeline systems. These coefficients are essential for applying the proposed method and must be implemented to ensure calculations.

Figure 4 illustrates the transition from an emitter coefficient calibrated for an existing pipeline (already constructed) to its adoption in designing a new pipeline using the proposed methodology. This diagram highlights how a calibrated emitter coefficient from an existing system can be used as information to create future infrastructure considering the same pipe material since water utilities in municipalities tend to build with similar patterns.

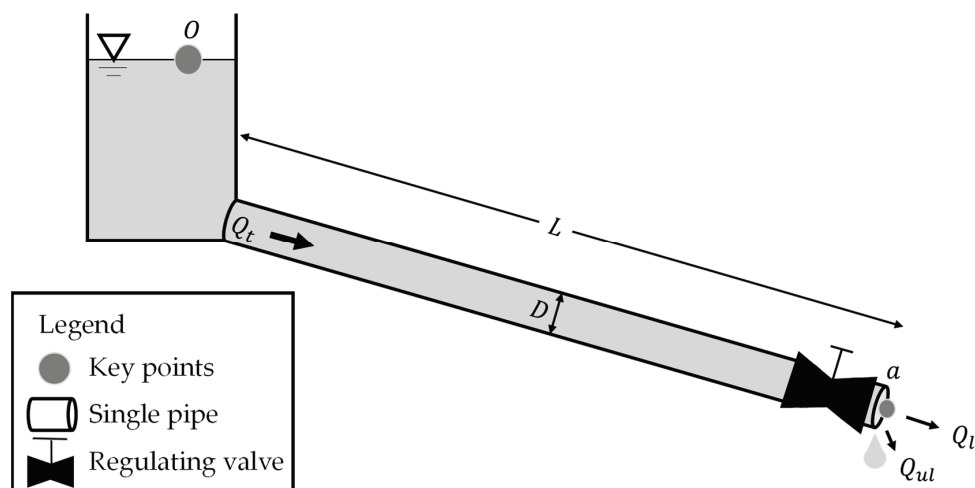


Figure 3. Scheme and variables for the proposed system.

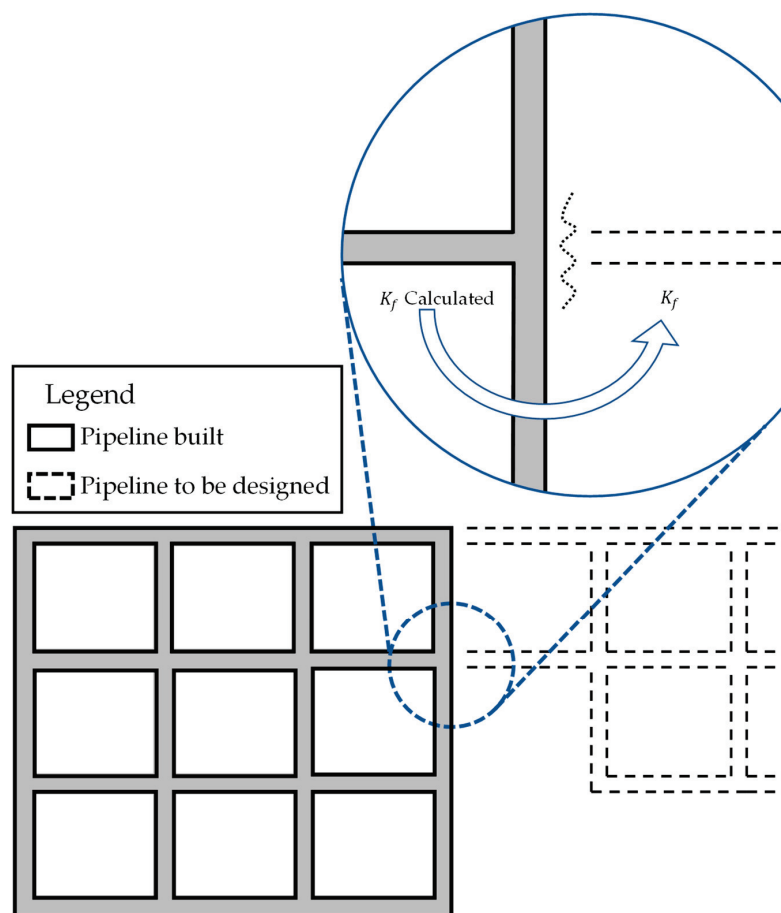


Figure 4. A scheme of the general assumption to solve the system.

The proposed methodology presents the following assumptions:

- Water movement is calculated using the Bernoulli equation.
- Friction accounts for energy losses are computed using the Darcy–Weisbach formula.
- Water leakages depend on the mean pressure head of a conduit, and they occur at the downstream end.

2.2. Governing Formulations

Appendix A presents the governing equations employed by the proposed methodology.

2.3. Optimisation Process and Numerical Resolution

A minimum pressure head (p_a/γ_w) is required at the consumption node a , as depicted in Figure 5a. Oversized diameters (represented by the blue and red lines) result in unnecessary construction costs that the system does not justify. Conversely, employing undersized diameters (purple and yellow lines) fails to satisfy the required pressure head constraint at the downstream node. Therefore, the optimal approach involves selecting a pair of internal pipe diameters from a catalogue that minimises construction costs while ensuring the required pressure head is achieved, as shown in Figure 5b. In this context, a combination of the purple and red lines represents the optimal solution.

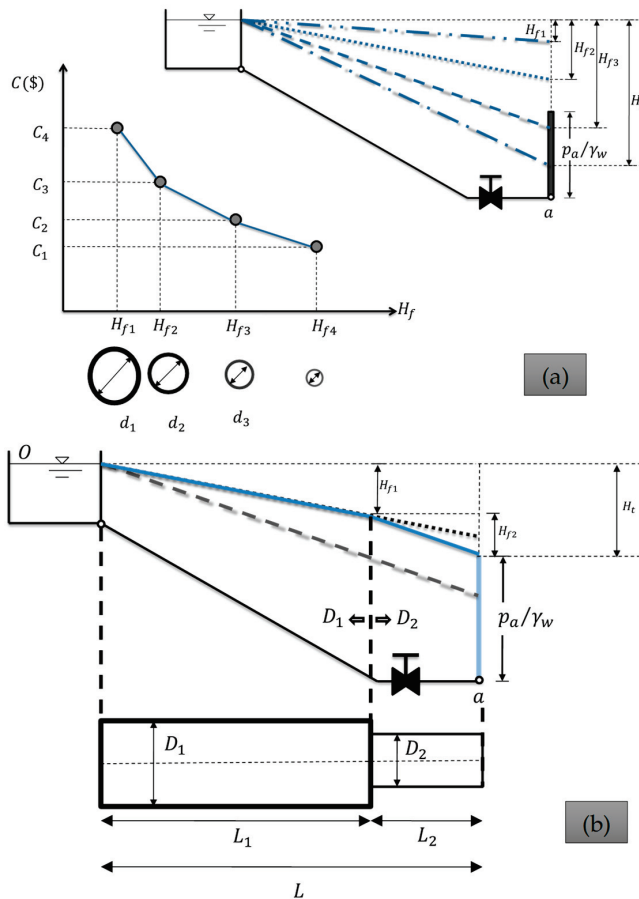


Figure 5. Considerations for optimal pipeline design: (a) relationship between cost and required service pressure; (b) selection of internal pipe diameters.

The friction losses can then be expressed as a function of a combination of two pipe diameter sizes as follows:

$$H_f = \frac{8(Q_{ul} + Q_l)^2}{\pi^2 g} \left(\frac{f_1 l_1}{D_1^5} + \frac{f_2 (L - l_1)}{D_2^5} \right) \quad (1)$$

Subscript 1 denotes the condition of the first pipe, while subscript 2 corresponds to the second. The relationship $L_2 = L - L_1$ serves as a constraint that must be satisfied.

Figure 6 presents a flowchart illustrating the proposed methodology for optimising pipe diameters while accounting for water leakages in single-pipeline installations. Two commercial diameters, one larger and one smaller, must be selected from a catalogue.

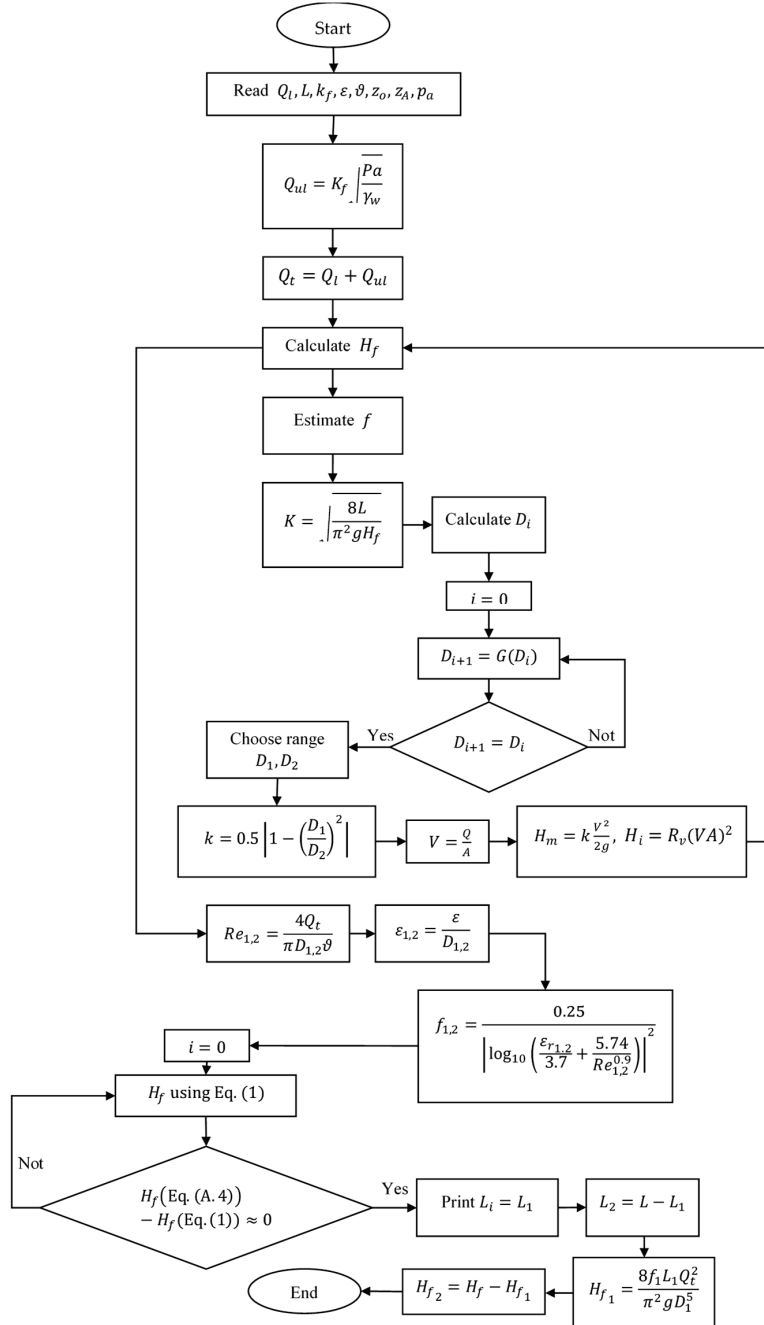


Figure 6. A flowchart of the proposed methodology.

3. Results

3.1. Dataset and Results

Considering the case of a single water pipeline, the following data were considered: $Q_l = 0.1 \text{ m}^3/\text{s}$, $\frac{p_a}{\gamma_w} = 30 \text{ m}$, $L = 2000 \text{ m}$, $\varepsilon = 0.00001 \text{ m}$, $\vartheta = 0.00000112 \text{ m}^2/\text{s}$, $Z_0 = 60 \text{ m}$,

and $Z_a = 5$ m. The effect of a regulating valve was considered, employing a resistance coefficient, $R_v = 190 \text{ s}^2/\text{m}^5$.

For this analysis, an emitter coefficient of $K_f = 0.00829 \text{ m}^3/\text{s}/\text{m}^{0.5}$ was used in combination with an exponent coefficient of 0.5. Based on these parameters, water leakages were calculated ($Q_{ul} = 45.41 \text{ L/s}$). Consequently, the total injected flow was determined, yielding a total pipe flow of $Q_t = 145.41 \text{ L/s}$ according to Equation (1).

Figure 7 shows the results obtained using the flowchart presented in Figure 5. The system exhibits a physical loss percentage of approximately 31%, calculated as the ratio of leaked flow to the total flow passing through the pipe. Furthermore, it is observed that most friction losses—around 96% of the total system—occur in the second pipe branch, since a $L_2 = 1921.51 \text{ m}$.

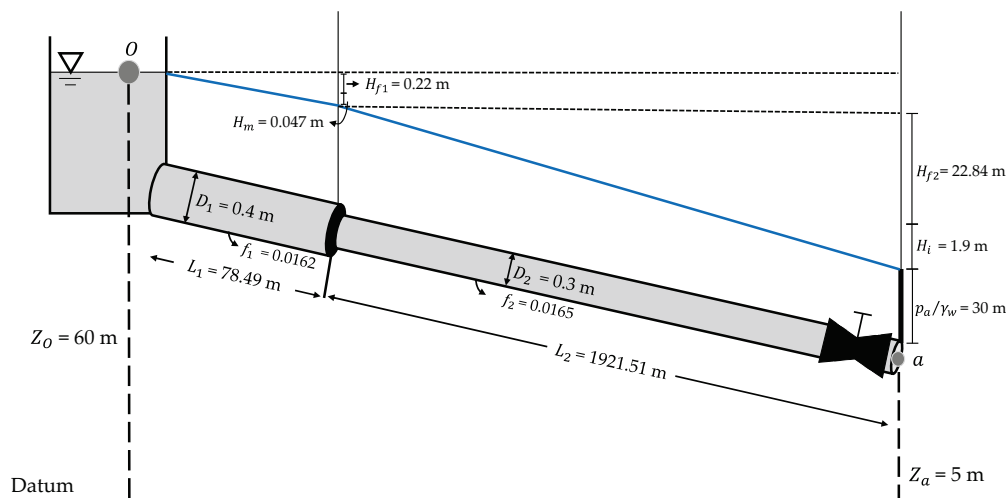


Figure 7. The solution employed for the proposed methodology.

3.2. Effect of Main Parameters

This section shows the effects of the main parameters in the proposed methodology. A change in the magnitude of leakage significantly impacts the optimal design (characterised mainly by the emitter coefficient), as this parameter plays a predominant role compared to other factors analysed (absolute roughness and resistance coefficient of a regulating valve). This highlights the importance of carefully accounting for leakage in the design process to ensure efficient and sustainable solutions. Furthermore, variations in leakage can affect the system's dimensions and hydraulic performance, emphasising the need for water utilities to use precise calibration and modelling strategies.

3.2.1. Emitter Coefficient

To investigate the effects of the emitter coefficient, a range spanning from 0.004 to $0.05 \text{ m}^3/\text{s}/\text{m}^{0.5}$ was employed in the calculations. These values correspond to the endpoints presented in Figure 8. The percentage of water losses under these conditions was evaluated. Furthermore, optimised solutions for pipe diameters and branch lengths were also determined to ensure efficient system performance.

Based on the results presented in Figure 8, the variation in the emitter coefficient significantly influences the percentage of water losses in the system as well as the lengths and diameters of the pipe branches. Specifically, when the emitter coefficient $K_f = 0.05 \text{ m}^3/\text{s}/\text{m}^{0.5}$ (Point 1) is considered, the system experiences the highest water losses, reaching 73%, alongside larger pipe diameters of 500 mm and 400 mm. Conversely,

for the minimum emitter coefficient value (Point 4), smaller pipe diameters of 300 mm and 200 mm are obtained.

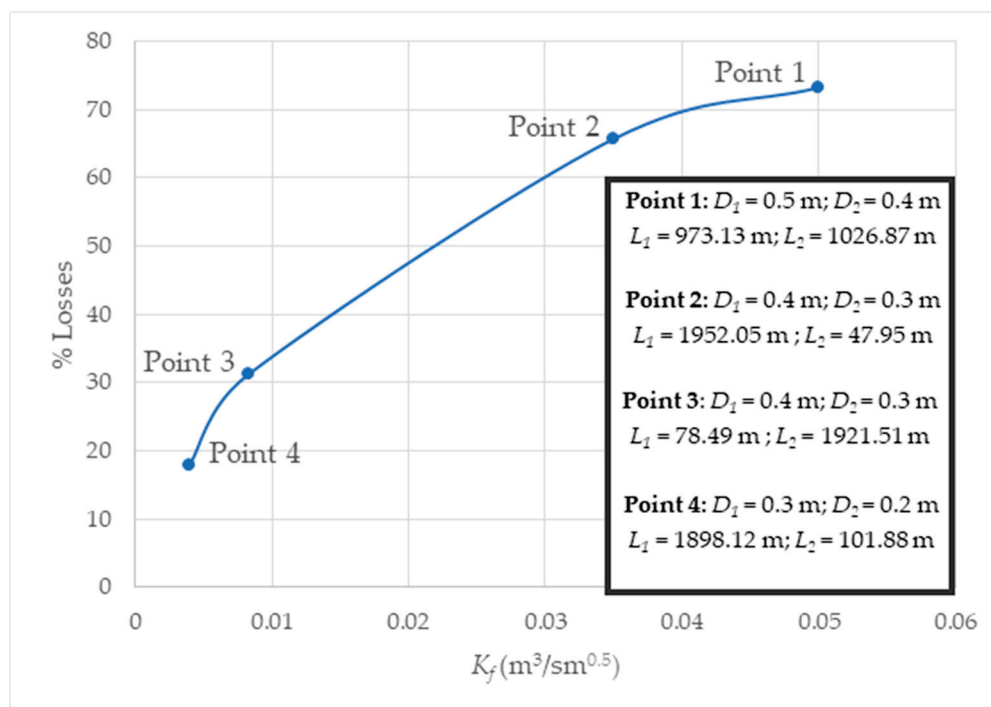


Figure 8. Effects in the variation in the emitter coefficient.

3.2.2. Variation in Absolute Roughness

The absolute roughness (ϵ) was analysed using values of 0.00015, 0.0001, and 0.005 m to evaluate its impact on the proposed methodology, as shown in Figure 9. The analysis considers emitter coefficient values of 0.004, 0.00829 (baseline), and 0.005 m³/s/m^{0.5}.

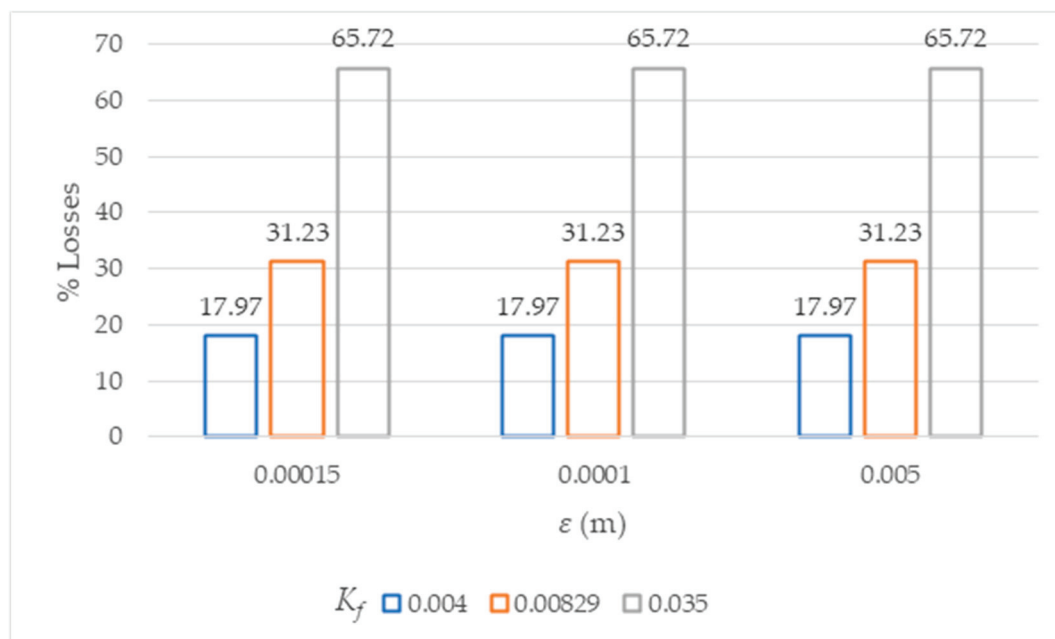


Figure 9. Analysis of absolute roughness versus emitter coefficient.

The variation in the roughness coefficient does not affect the percentage of water losses, as expected, since water losses depend on the pressure head in the pipes according to the assumptions, as shown in Figure 9. In this sense, the grey bars ($K_f = 0.035 \text{ m}^3/\text{s}/\text{m}^{0.5}$) show water losses around 65%, independent of the values selected for the roughness coefficient.

3.2.3. Resistance Coefficient

The analysis of the resistance coefficient (R_v) is presented to observe the changes generated by the effect of the regulating valves. This study considered the following data: 150, 190, and 240 s^2/m^5 . Figure 10 presents the results obtained. The results indicate that no changes were presented in the water losses, as observed in the bars with the same colour. For the blue bars ($K_f = 0.004 \text{ m}^3/\text{s}/\text{m}^{0.5}$), water losses are around 18% in the three cases, as water losses are located at the downstream end.

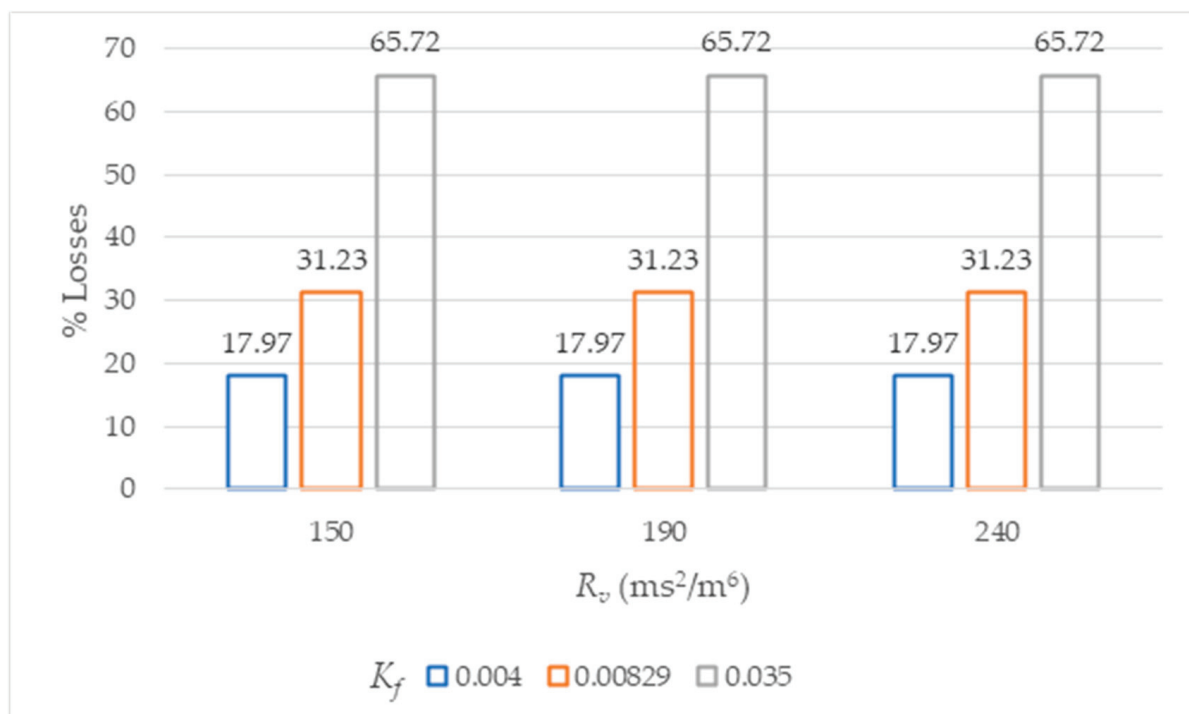


Figure 10. Analysis considering resistance coefficient.

4. Discussion

4.1. Analysis Considering Percentage of Water Losses

A typical practice in the current design is to select a percentage of water losses based on domestic consumption. In this sense, Colombian regulations suggest that a percentage of water losses must be considered in the design of water pipelines; typically, a value of 25% is accepted in this country. This research considered values of 0, 10, 25, and 45%. This assumption selects a pressure-independent consumption in the design phase of water distributions ($K_f = 0$). Figure 11 presents the results for these scenarios. The greater the water loss percentages, the higher internal pipe diameters are obtained, as demonstrated by comparing the values of Points 1 and 4.

An accurate computation of water losses is essential for water utilities to properly characterise these values. For example, an emitter coefficient of $K_f = 0.05 \text{ m}^3/\text{s}/\text{m}^{0.5}$ results in water losses of up to 73% (see Figure 8), significantly higher than the assumption

of 45% of water losses (considering pressure-independent consumption). A difference of 28% was found in this comparison.

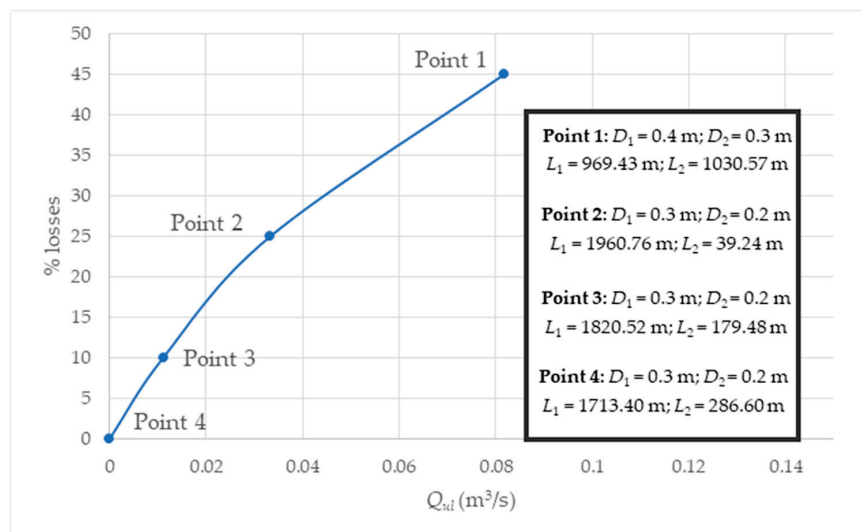


Figure 11. Analysis considering water losses as pressure-independent consumption.

4.2. A Comparison Between the Traditional and Proposed Methodologies

To highlight the advantages and disadvantages of the proposed method, a comparison was conducted between the proposed method and the traditional methodology, as presented in Table 2.

Table 2. Comparison between traditional and proposed methodologies.

Method	Advantages	Disadvantages
Traditional method (pressure-independent consumption)	- Utilises recommendations from local regulations to establish percentage of water losses - Algorithm for numerical resolution is more straightforward than proposed methodology - Numerous algorithms exist for designing pipelines in parallel, series, open, and closed networks	- The percentage of water losses considered does not reflect physical behaviour, as water losses occur in existing water installations. - It is not based on the modern concept of a digital twin model.
Proposed methodology (pressure-dependent consumption)	- Based on physical concept, as water losses occur in existing water installations - Facilitates preliminary establishment of digital twin model - Does not require selecting predefined percentage of water losses	- It requires a detailed analysis to compute parameters for quantifying water losses (pressure-dependent consumption) in existing infrastructure. - The numerical resolution is more complex than the traditional method. - No algorithms are currently proposed for parallel, series, open, or closed water distribution networks. - It requires the modification of local regulations to allow for the sizing of water installations.

4.3. Analysis of Formulas for Computing Water Leaks

Water leaks depend on pressure-dependent consumption, which have been computed employing Equation (A2). In addition, the orifice equation was used for simulating water leaks, which is yielded as follows:

$$Q_{ul} = C_d A \sqrt{2gh}^N \quad (2)$$

where C_d = the discharge coefficient, A = the cross-sectional area of a leakage, N = the exponent, and h = the mean pressure in a pipe (p_a / γ_w).

A comparative analysis was conducted between values of Q_{ul} obtained from applying Equations (A2) and (2) using an exponent value of 0.5. A cross-sectional area of 0.0012 m² and a discharge coefficient of 0.868 were employed for the orifice equation. Figure 12 presents the results for both equations. Equations show that the higher the pressure, the greater the water leaks obtained. The orifice formula (Equation (2)) yields lower values as it accounts for the influence of each constant separately rather than aggregating them into a single parameter (emitter coefficient), as seen in Equation (A2).

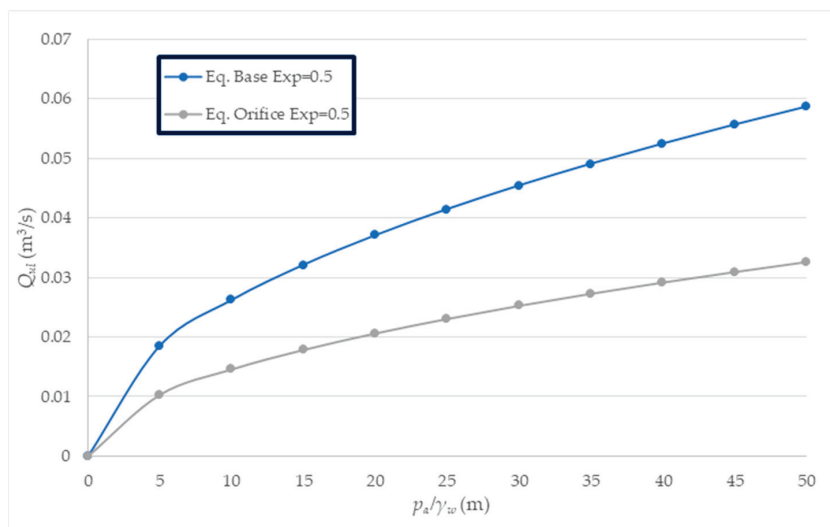


Figure 12. Effect of exponent coefficient in water leak formulas.

By applying the proposed methodology using Equation (2) (refer to Figure 13), internal pipe diameters of 0.3 m and 0.2 m, with corresponding pipe lengths of 1918.32 m and 81.68 m, are obtained, respectively. The pipe diameters determined using Equation (2) are smaller than those calculated with Equation (A2) as water leakage is reduced.

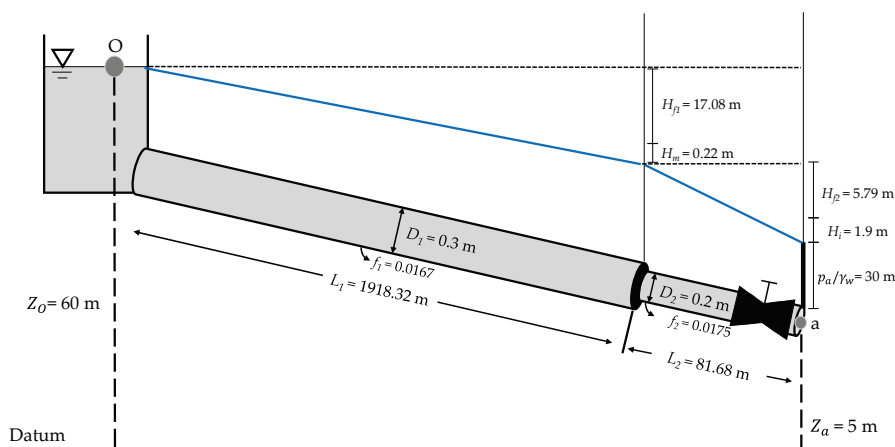


Figure 13. The solution obtained using the orifice equation.

5. Conclusions

The proposed methodology in this research offers a foundation for designing water distribution systems that treat water losses as pressure-dependent consumption. This

approach contrasts with current standards and regulations, which calculate water losses as a percentage of domestic consumption, assuming they are pressure independent. The proposed methodology can be employed for water utilities that have previously calibrated emitter coefficients in existing water installations.

This research presents a formulation for sizing single pipelines while accounting for water leaks as pressure-dependent consumption. A minimum of two internal pipe diameters is necessary to achieve the required service pressure. The sensitivity analysis reveals that the emitter coefficient significantly influences these diameters. Since water leakages are located downstream, absolute roughness and resistance coefficients do not impact the sizing of water installations. In addition, the current concept for sizing water installations considering water losses as a percentage of domestic consumption must be renewed as an essential difference can be achieved. The requirement of service pressure can be affected by considering traditional practises. The integration of digital twin models is closely linked to the appropriate sizing of water distribution networks from the design phase.

Future research should focus on developing general formulas for sizing long pipelines and water distribution networks (both open and closed) while considering water losses as pressure-dependent consumption.

Author Contributions: Conceptualization, M.P.-Z., J.J.V.-G. and O.E.C.-H.; methodology, M.S. and O.E.C.-H.; formal analysis, M.P.-Z., J.J.V.-G. and V.S.F.-M.; writing—original draft preparation, O.E.C.-H. and M.S.; writing—review and editing, V.S.F.-M. All authors have read and agreed to the published version of the manuscript.

Funding: This research project did not secure funding from external nor internal sources.

Data Availability Statement: Databases are available from the corresponding author.

Conflicts of Interest: The authors declare no conflicts of interest.

Appendix A. Governing Equations

This appendix shows the governing equations using the proposed methodology, which adopts the flow classification established by the International Water Association [3].

Appendix A.1. Total Flow

This equation presents the relationship between the injected total flow (Q_t) and water flow due to physical leakages (Q_{ul}) and water flow for pressure-independent consumption (Q_l).

$$Q_t = Q_l + Q_{ul} \quad (A1)$$

Pressure-independent consumption is usually divided into domestic, industrial, commercial, and official.

Appendix A.2. Leakage Flow

This equation represents the physical losses occurring within pipe systems. These losses depend on the pressure head of the pipeline, as well as the emitter and exponent coefficients. Typically, water losses in pipelines are calculated using the following formula [3,33]:

$$Q_{ul} = K_f \sqrt[n]{\frac{p_a}{\gamma_w}} \quad (A2)$$

where Q_{ul} = the leakage flow rate, K_f = the emitter coefficient, n = the exponent coefficient, p_a = the average pressure in a conduit, and γ_w = the water unit weight.

Appendix A.3. Energy Loss Equation

The Bernoulli equation establishes the relationship between the upstream and downstream nodes of a single pipeline, considering pipe elevations, friction losses along the pipeline, minor losses (caused by joints, changes in internal diameters, and inlet and outlet conditions, among others), valve losses, and the required pressure head at the downstream node [34].

$$H_f = Z_a - Z_o - H_m - \frac{P_a}{\gamma_w} - H_i \quad (A3)$$

where H_f = friction account losses, H_m = minor energy losses, H_i = energy loss in a regulating valve, k = the minor loss coefficient, V = water velocity, g = gravity acceleration (which takes a value of $9.81 \text{ m}^2/\text{s}$), Z_o = the elevation of the initial node, Z_a = the elevation of the final node, R_v = the resistance coefficient of a regulating valve, and A = the cross-sectional area. The minor energy losses and losses in a regulating valve are given by $H_m = k \frac{V^2}{2g}$ and $H_i = R_v(VA)^2$, respectively.

By considering the behaviour of water leakages through Equation (A2) in the term P_a/γ_w and organising terms, the following is obtained:

$$H_f = Z_a - Z_o - k \frac{V^2}{2g} - \left(\frac{Q_{ul}}{K_f} \right)^2 - R_v(VA)^2 \quad (A4)$$

Appendix A.4. Darcy–Weisbach Equation

This equation relates to the internal pipe diameter as a function of the injected total flow pipe length and friction account losses. The expression is yielded by the following [3,35]:

$$D = \left[\frac{8fL(Q_{ul} + Q_l)^2}{\pi^2 g H_f} \right]^{0.2} \quad (A5)$$

where L = the pipe length and f = the friction factor.

Equation (A5) can be expressed as a function of a K value as follows:

$$\frac{1}{\sqrt{f}} = \sqrt{\frac{8L}{\pi^2 g H_f}} \frac{(Q_{ul} + Q_l)}{D^{2.5}} = K D^{-2.5} \quad (A6)$$

Appendix A.5. Friction Factor Equation

Determining the friction factor is crucial for accurately accounting for friction losses. Due to typical water velocity ranges, the flow is turbulent in most water distribution systems. The Colebrook–White equation is widely recognised for its foundation in physical principles [35]. Calculating the pipe diameter requires an iterative process to solve the relationship expressed in the formula $D_{(i+1)} = G(D_i)$. The i value represents the current iteration, while the value $i + 1$ corresponds to the subsequent iteration. After simplifying some terms, the internal pipe diameter can be expressed as follows:

$$D = \left[-\frac{2}{K} \log_{10} \left(\frac{\varepsilon}{3.7D} + \frac{2.51 \sqrt{\frac{8L}{\pi^2 g H_f}} \pi \vartheta}{4D^{1.5}} \right) \right]^{-0.4} \quad (A7)$$

where ε = the pipe's absolute roughness and ϑ = the water kinematic viscosity.

Alternatively, the Swamee–Jain equation [36] can be employed to calculate the friction factor, offering an error of less than 1% compared to the Colebrook–White equation. Its

primary advantage lies in providing an explicit solution for the friction factor, eliminating the need for an iterative process. It is yielded as follows:

$$f = \frac{0.25}{\left[\log_{10} \left(\frac{\varepsilon_r}{3.7} + \frac{5.74}{\text{Re}^{0.9}} \right) \right]^2} \quad (\text{A8})$$

where Re is the Reynold number given by $\frac{4 \cdot Q_t}{\pi \cdot D \cdot v}$, and ε_r corresponds to the relative roughness $\left(\frac{\varepsilon}{D} \right)$.

References

1. Lambert, A. Accounting for Losses: The Bursts and Background Concept. *Water Environ. J.* **1994**, *8*, 205–214. [CrossRef]
2. Ramos, H.M.; Kuriqi, A.; Besharat, M.; Creaco, E.; Tasca, E.; Coronado-Hernández, O.E.; Pienika, R.; Iglesias-Rey, P. Smart Water Grids and Digital Twin for the Management of System Efficiency in Water Distribution Networks. *Water* **2023**, *15*, 1129. [CrossRef]
3. Hirner, W.; Lambert, A. *Losses from Water Supply Systems: Standard Terminology and Recommended Performance Measures*; IWA: London, UK, 2000.
4. Niebuhr, D.; Nsanzubuhoro, R.; van Zyl, J.E. Field Experience with Pressure-Based Leakage Characterisation of Bulk Water Pipelines. *Urban Water J.* **2019**, *16*, 709–717. [CrossRef]
5. Sophocleous, S.; Savić, D.A.; Kapelan, Z.; Giustolisi, O. A Two-Stage Calibration for Detection of Leakage Hotspots in a Real Water Distribution Network. *Procedia Eng.* **2017**, *186*, 168–176. [CrossRef]
6. Aksela, K.; Aksela, M.; Vahala, R. Leakage Detection in a Real Distribution Network Using a SOM. *Urban Water J.* **2009**, *6*, 279–289. [CrossRef]
7. Ulanicki, B.; Bounds, P.L.M.; Rance, J.P.; Reynolds, L. Open and Closed Loop Pressure Control for Leakage Reduction. *Urban Water* **2000**, *2*, 105–114. [CrossRef]
8. AL-Washali, T.; Sharma, S.; Lupoja, R.; AL-Nozaily, F.; Haidera, M.; Kennedy, M. Assessment of Water Losses in Distribution Networks: Methods, Applications, Uncertainties, and Implications in Intermittent Supply. *Resour. Conserv. Recycl.* **2020**, *152*, 104515. [CrossRef]
9. Saldarriaga, J.; Salcedo, C.A. Determination of Optimal Location and Settings of Pressure Reducing Valves in Water Distribution Networks for Minimizing Water Losses. *Procedia Eng.* **2015**, *119*, 973–983. [CrossRef]
10. Deyi, M.; van Zyl, J.; Shepherd, M. Applying the FAVAD Concept and Leakage Number to Real Networks: A Case Study in Kwadabeka, South Africa. *Procedia Eng.* **2014**, *89*, 1537–1544. [CrossRef]
11. Coronado-Hernández, O.E.; Pérez-Sánchez, M.; Arrieta-Pastrana, A.; Fuertes-Miquel, V.S.; Coronado-Hernández, J.R.; Quiñones-Bolaños, E.; Ramos, H.M. Dynamic Effects of a Regulating Valve in the Assessment of Water Leakages in Single Pipelines. *Water Resour. Manag.* **2024**, *38*, 2889–2903. [CrossRef]
12. Fuertes-Miquel, V.S.; Arrieta-Pastrana, A.; Coronado-Hernández, O.E. Analyzing Water Leakages in Parallel Pipe Systems with Rapid Regulating Valve Maneuvers. *Water* **2024**, *16*, 926. [CrossRef]
13. Puust, R.; Kapelan, Z.; Savić, D.A.; Koppel, T. A Review of Methods for Leakage Management in Pipe Networks. *Urban Water J.* **2010**, *7*, 25–45. [CrossRef]
14. Maskit, M.; Ostfeld, A. Leakage Calibration of Water Distribution Networks. *Procedia Eng.* **2014**, *89*, 664–671. [CrossRef]
15. Samir, N.; Kansoh, R.; Elbarki, W.; Fleifle, A. Pressure Control for Minimizing Leakage in Water Distribution Systems. *Alex. Eng. J.* **2017**, *56*, 601–612. [CrossRef]
16. García-Ávila, F.; Avilés-Añazco, A.; Ordoñez-Jara, J.; Guanuchi-Quezada, C.; Flores del Pino, L.; Ramos-Fernández, L. Pressure Management for Leakage Reduction Using Pressure Reducing Valves. Case Study in an Andean City. *Alex. Eng. J.* **2019**, *58*, 1313–1326. [CrossRef]
17. Pezzinga, G.; Pititto, G. Combined Optimization of Pipes and Control Valves in Water Distribution Networks. *J. Hydraul. Res.* **2005**, *43*, 668–677. [CrossRef]
18. Ramos, H.M.; Kuriqi, A.; Coronado-Hernández, O.E.; López-Jiménez, P.A.; Pérez-Sánchez, M. Are Digital Twins Improving Urban-Water Systems Efficiency and Sustainable Development Goals? *Urban Water J.* **2023**, *21*, 1164–1175. [CrossRef]
19. Ramos, H.M.; Morani, M.C.; Carravetta, A.; Fecarrotta, O.; Adeyeye, K.; López-Jiménez, P.A.; Pérez-Sánchez, M. New challenges towards smart systems' efficiency by digital twin in water distribution networks. *Water* **2022**, *14*, 1304. [CrossRef]
20. Lee, M.K.; Kim, D.; Kim, M.O. Experimental Study on the Chlorine-Induced Corrosion and Blister Formation of Steel Pipes Coated with Modified Polyethylene Powder. *Polymers* **2024**, *16*, 2415. [CrossRef]
21. Langelandsvik, L.I.; Kunkel, G.J.; Smits, A.J. Flow in a Commercial Steel Pipe. *J. Fluid. Mech.* **2008**, *595*, 323–339. [CrossRef]

22. Zhang, J.; Li, R.; Yang, S.; Liu, J.; Guo, Z.; Feng, L.; Shang, W.; Zhu, L.; Du, B. Field Load Testing and Numerical Analysis of Offshore Photovoltaic Steel Pipe Piles. *Soil Dyn. Earthq. Eng.* **2025**, *188*, 109034. [CrossRef]
23. Enany, P.; Shevchenko, O.; Drebenshtedt, C. Evaluation of Airlift Pump Performance for Vertical Conveying of Coal Particles. *Int. J. Coal Sci. Technol.* **2023**, *10*, 42. [CrossRef]
24. Enany, P.; Drebenshtedt, C. Performance Characteristics of the Airlift Pump under Vertical Solid–Water–Gas Flow Conditions for Conveying Centimetric-Sized Coal Particles. *Int. J. Coal Sci. Technol.* **2024**, *11*, 18. [CrossRef]
25. Li, Y.; Yang, R.; Wang, Y.; Fu, D. Influence of Water Coupling Coefficient on the Blasting Effect of Red Sandstone Specimens. *Int. J. Coal Sci. Technol.* **2024**, *11*, 8. [CrossRef]
26. Ma, D.; Duan, H.; Li, Q.; Wu, J.; Zhong, W.; Huang, Z. Water–Rock Two-Phase Flow Model for Water Inrush and Instability of Fault Rocks during Mine Tunnelling. *Int. J. Coal Sci. Technol.* **2023**, *10*, 77. [CrossRef]
27. Wang, D.; Jiao, D.; Cheng, Z.; Shi, Q.; Mischo, H. Multi-Criteria Comparative Analysis of the Pressure Drop on Coal Gangue Fly-Ash Slurry at Different Parts along an L-Shaped Pipeline. *Int. J. Coal Sci. Technol.* **2023**, *10*, 28. [CrossRef]
28. Bernard, M. Using Digital Twins to Improve Pumping and Distribution System Operations. *J. AWWA* **2024**, *116*, 6–15. [CrossRef]
29. Zechman, B.E.; Ehsan, S.M.; Lu, X.; Jason, W. Digital Twins for Water Distribution Systems. *J. Water Resour. Plan. Manag.* **2023**, *149*, 02523001. [CrossRef]
30. Ramos, H.M.; McNabola, A.; López-Jiménez, P.A.; Pérez-Sánchez, M. Smart water management towards future water sustainable networks. *Water* **2020**, *12*, 58. [CrossRef]
31. Alzamora, F.M.; Carot, M.H.; Carles, J.; Campos, A. Development and Use of a Digital Twin for the Water Supply and Distribution Network of Valencia (Spain). In Proceedings of the 17th International Computing & Control for the Water Industry Conference, Exeter, UK, 1–4 September 2019.
32. Vivienda, M.; Territorio, C. Ministry of Housing, C. and D. (MinVivienda). Resolution No. 0799/2021. Available online: https://minvivienda.gov.co/sites/default/files/normativa/0799_2021.pdf (accessed on 18 December 2024).
33. Almandoz, J.; Cabrera, E.; Arregui, F.; Cabrera, E.; Cobacho, R. Leakage Assessment through Water Distribution Network Simulation. *J. Water Resour. Plan. Manag.* **2005**, *131*, 458–466. [CrossRef]
34. Rubenstein, D.A.; Yin, W.; Frame, M.D. Chapter 3—Conservation Laws. In *Biofluid Mechanics*, 3rd ed.; Rubenstein, D.A., Yin, W., Frame, M.D., Eds.; Academic Press: Cambridge, MA, USA, 2022; pp. 71–131. [CrossRef]
35. Streeter, V.L.; Kestin, J. *Handbook of Fluid Dynamics*; McGraw-Hill: New York, NY, USA, 1961.
36. Swamee, P.K.; Jain, A.K. Explicit Equations for Pipe-Flow Problems. *J. Hydraul. Div.* **1976**, *102*, 657–664. [CrossRef]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

Mathematical Approach for Directly Solving Air–Water Interfaces in Water Emptying Processes

Dalia M. Bonilla-Correa ¹, Oscar E. Coronado-Hernández ^{2,*}, Alfonso Arrieta-Pastrana ²,
Vicente S. Fuertes-Miquel ^{3,*}, Modesto Pérez-Sánchez ³ and Helena M. Ramos ⁴

¹ Facultad de Ciencias Exactas y Naturales, Universidad de Cartagena, Cartagena 130001, Colombia; dbonillac@unicartagena.edu.co

² Instituto de Hidráulica y Saneamiento Ambiental, Universidad de Cartagena, Cartagena 130001, Colombia

³ Departamento de Ingeniería Hidráulica y Medio Ambiente, Universitat Politècnica de València, 46022 Valencia, Spain

⁴ Civil Engineering, Architecture and Environment Department (CERIS), Instituto Superior Técnico, University of Lisbon, 1049-001 Lisbon, Portugal; helena.ramos@tecnico.ulisboa.pt

* Correspondence: ocoronadoh@unicartagena.edu.co (O.E.C.-H.); vfuertes@upv.es (V.S.F.-M.)

Abstract: Emptying processes are operations frequently required in hydraulic installations by water utilities. These processes can result in drops to sub-atmospheric pressure pulses, which may lead to pipeline collapse depending on soil characteristics and the stiffness of a pipe class. One-dimensional mathematical models and 3D computational fluid dynamics (CFD) simulations have been employed to analyse the behaviour of the air–water interface during these events. The numerical resolution of these models is challenging, as 1D models necessitate solving a system of algebraic differential equations. At the same time, 3D CFD simulations can take months to complete depending on the characteristics of the pipeline. This presents a mathematical approach for directly solving air–water interactions in emptying processes involving entrapped air, providing a predictive tool for water utilities. The proposed mathematical approach enables water utilities to predict emptying operations in water pipelines without needing 2D/3D CFD simulations or the resolution of a differential algebraic equations system (1D model). A practical application is demonstrated in a case study of a 350 m long pipe with an internal diameter of 350 mm, investigating the influence of air pocket size, friction factor, polytropic coefficient, pipe diameter, resistance coefficient, and pipe slope. The mathematical approach is validated using an experimental facility that is 7.36 m long, comparing it with 1D mathematical models and 3D CFD simulations. The results confirm that the derived mathematical expression effectively predicts emptying operations in single water installations.

Keywords: emptying processes; mathematical approach; Simpson’s 1/3 rule; transient event; trapped air; water distribution networks

1. Introduction

The study of water emptying processes in pressurised pipelines has received limited attention from researchers worldwide, although water utilities routinely conduct this operation [1]. The accurate prediction of emptying processes is crucial to prevent pipeline collapses, as sub-atmospheric pressure pulses may develop during transient events depending on soil conditions and pipe stiffness. An efficient emptying process should ensure that the pipeline is free of air pockets, as large air pockets are recommended before initiating filling operations. This is because more oversized air pockets tend to lower air pocket pressures, which helps mitigate the risk of pipeline failure due to peak pressures. Moreover, these procedures are vital for expelling sediments and minimising issues related to pipe corrosion.

Water utilities should perform this operation to achieve appropriate emptying times and restore the regular operation of water installations as swiftly as possible. In developing

countries, frequent interruptions in water services often occur due to pipeline failures, leaving many communities without access to a reliable water supply.

The emptying process modelling involves multiphase flows (air and water), which should be simulated using thermodynamics and hydraulic formulations. Tijsseling et al. (2016) [2] proposed a 1D model for predicting emptying processes considering a stratified air–water interaction based on a holdup coefficient, which has been experimentally validated [3]. Coronado-Hernández et al. (2018) [4] and Fuertes-Miquel et al. (2019) [1] proposed a 1D mathematical model for analysing emptying operations using pressurised air in combination with a piston flow approach using experimental measurements during the validation model stage. These formulations are based on physical equations considering the rigid water model, a perpendicular air–water interface, and the polytropic law for the air phase, which can be applied for single and irregular profile water installations. When air valves are installed, the air valve characterisation and mass balance equations must be considered. The air balance is composed of air pocket density mixing with the air density in normal conditions. Chen et al. (2024) [5] presented a dataset of laboratory experiments in a large-scale PVC pipeline 275.2 m long with a diameter of 0.25 m, which presents inlet, bridge, and horizontal branches for studying emptying processes.

The 3D computational fluid dynamics (CFD) model allows for the numerical resolution of Navier–Stokes partial differential equations describing fluid behaviour. This technique predicts fluid movement within water installations, including two phases (water and air) [6]. In practice, CFD models are widely accepted by the scientific community as a reliable means of solving the Navier–Stokes equations, as well as the continuity and energy formulations [7–9]. Because of the computational power, one can use CFD quickly and effectively. Another essential feature of CFD models is that their solutions can be as precise as prototype models. Liu et al. (2019) [10] have investigated the emptying operation of an oil pipeline with a complex longitudinal profile, analysing air–water behaviour through 3D simulations using OLGA v7.0 software. Additionally, a 3D analysis of emptying processes was conducted to capture backflow air, employing OpenFOAM software in an experimental facility measuring 4.36 m long. The PVoF model was used with a two-equation turbulence model for the simulation [11].

Currently, the literature outlines two main approaches for solving emptying processes; (i) for 1D models, numerical methods such as Runge–Kutta or Rosenbrock can be implemented using solvers from computational packages like Matlab R2024b [12] and Octave 7.0 [13], among others, and (ii) for 3D CFD models, software packages such as Ansys 2024 R2 [14], OpenFOAM v2012 [15], OLGA 7.0 [10], Flow-3D v12.0 [16], and STAR-CCM+ 2402 [17] are commonly used. The numerical resolution of 1D models necessitates a system of equations that must be carefully integrated to solve the complex differential equations. Three-dimensional CFD models require significant computational times, with simulations of actual pipelines potentially taking several months to complete. In summary, the computation of the main hydraulic and thermodynamic behaviours of emptying processes have been addressed using 2D/3D CFD [11,18] models or applying a differential algebraic equations system [1,2].

This research presents a mathematical approach for directly solving air–water interaction in water emptying processes with entrapped air based on a one-dimensional formulation of the rigid water model, piston flow, and the polytropic law. The system, composed of differential algebraic equations proposed by the authors in previous publications, has been solved numerically and analytically to provide a solution applicable to water utilities for simulating these operations. The approach accounts for an instantaneous valve opening manoeuvre. A practical application to an extensive pipeline is demonstrated, offering a much faster solution than those produced by 3D CFD models. Furthermore, the proposed method is validated through experimental measurements and compared with 1D and 3D CFD models, demonstrating good accuracy in predicting this biphasic transient event.

2. Material and Methods

2.1. One-Dimensional Mathematical Model

The authors have investigated the transient behaviour of water emptying processes in pipelines in references [1,4]. These processes commence with assuming the pipes are initially filled with water. The entrapped air pocket expands upon opening the regulating valves, decreasing air pocket pressure. Typically, the initial air pocket is assumed to be at atmospheric pressure, and sub-atmospheric conditions are reached during these events. The current mathematical model, consisting of differential algebraic equations, was developed by the authors in earlier publications as follows:

$$\frac{dL}{dt} = -v \quad (1)$$

$$\frac{dv}{dt} = \frac{p_1^* - p_{atm}^*}{\rho L} + g \sin \theta - \frac{f}{2D} v|v| - \frac{R_v g A^2}{L} v|v| \quad (2)$$

$$p_1^* = \frac{p_{1,0}^* x_0^k}{(L_T - L)^k} \quad (3)$$

where p_1^* = air pocket pressure; v = water velocity; L = length of the water emptying column; p_{atm}^* = atmospheric pressure; ρ = water density; g = gravitational acceleration; f = friction factor; θ = pipe slope (rad); A = cross-sectional area of pipe; R_v = resistance coefficient; D = internal pipe diameter; L_T = total pipe length; and k = polytropic coefficient.

The initial conditions of the system are given by $v_0 = 0$, $L_0 = L_T - x_0$, and $p_{1,0}^* = p_{atm}^* = 101,325$ Pa. The subscript 0 indicates an initial condition of analysed variables.

Figure 1 illustrates the scheme of an emptying process in a single pipeline with entrapped air. The system starts at rest with zero water velocity (Figure 1a). Subsequently, when a regulating valve is opened, the emptying process begins, as depicted in Figure 1b. The system again reaches zero water velocity at the end of the biphasic transient event.

2.2. Proposed Mathematical Approach

This section shows the proposed model developed in this research. The model is based on the following assumptions: (i) water movement is described by the Euler equation; (ii) the air–water interface is considered perpendicular to the main direction of a pipeline; (iii) the friction factor is constant during the transient event; (iv) the air phase follows the polytropic law; and (v) the model applies to single water installations with a sufficient pipe slope.

The 1D mathematical model, consisting of Equations (1)–(3), can be rearranged to eliminate dependence on time t .

By substituting Equation (3) into Equation (2), the system of differential equations can be expressed by Equations (1) and (4).

$$\frac{dv}{dt} = \frac{p_{1,0}^* x_0^k}{\rho L (L_T - L)^k} - \frac{p_{atm}^*}{\rho L} + g \sin \theta - \frac{f}{2D} v|v| - \frac{R_v g A^2}{L} v|v| \quad (4)$$

The system comprising Equations (1) and (4) is autonomous [19]; thus, its numerical solution can be carried out in the vL plane by tracking its trajectory. The trajectory consists of points $(L(t), v(t))$, which satisfy Equations (1) and (4). The system can be simplified as follows:

$$\frac{dv}{dL} = \frac{\frac{dv}{dt}}{\frac{dL}{dt}} = \frac{\frac{p_{1,0}^* x_0^k}{\rho L (L_T - L)^k} - \frac{p_{atm}^*}{\rho L} + g \sin \theta - \frac{f}{2D} v|v| - \frac{R_v g A^2}{L} v|v|}{-v} \quad (5)$$

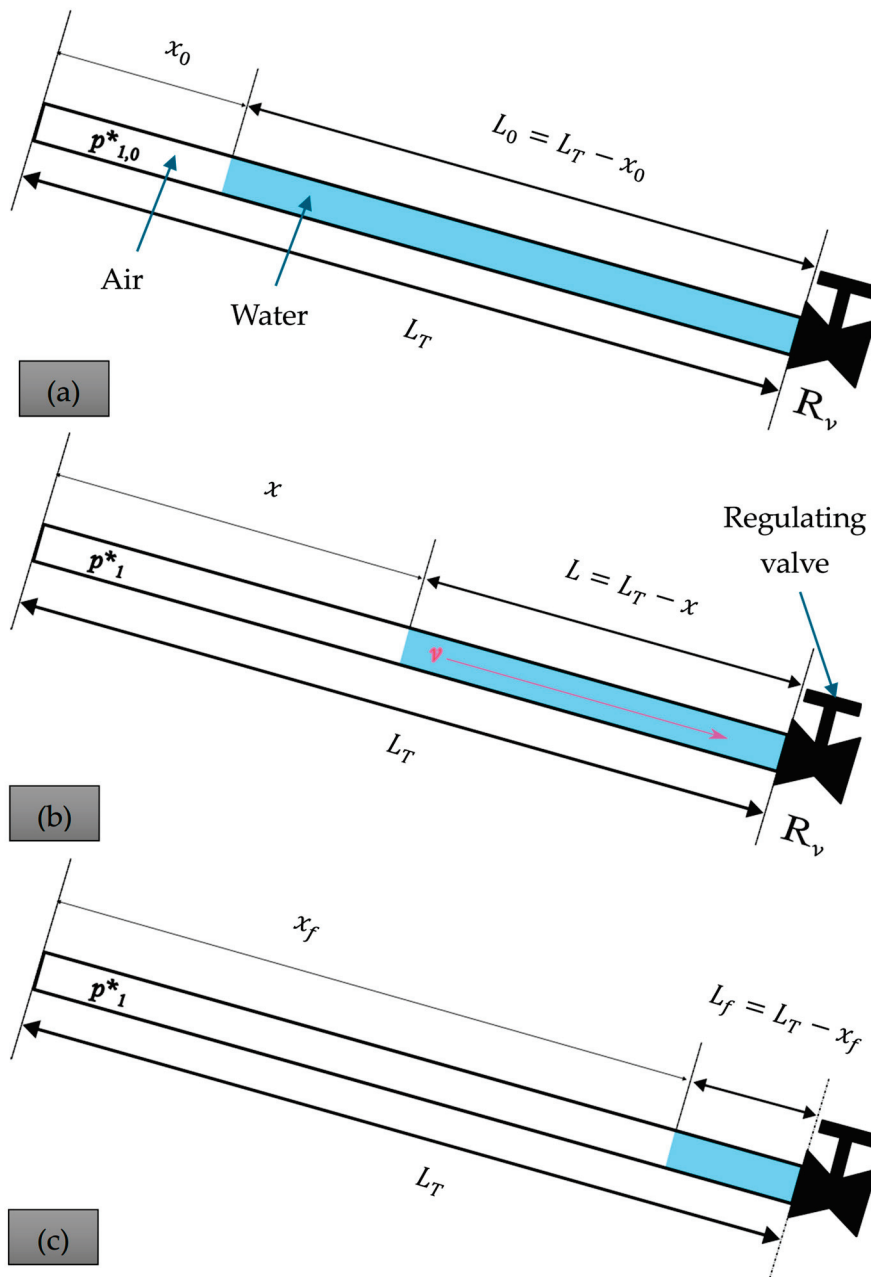


Figure 1. Scheme of water emptying process: (a) initial condition, (b) intermediate position, and (c) final position.

Considering Equation (5), it is possible to compute $\frac{dv}{dL}$. Equation (5) is a single first-order linear differential equation, where L is the independent variable and v is a function of L with no dependence on t . From Equation (5),

$$-v \frac{dv}{dL} = \frac{p_{1,0}^* x_0^k}{\rho L (L_T - L)^k} - \frac{p_{atm}^*}{\rho L} + g \sin \theta - \frac{f}{2D} v |v| - \frac{R_v g A^2}{L} v |v| \quad (6)$$

Multiplying Equation (6) by 2, then

$$2v \frac{dv}{dL} = \frac{-2p_{1,0}^* x_0^k}{\rho L (L_T - L)^k} + \frac{2p_{atm}^*}{\rho L} - 2g \sin \theta + \frac{f}{D} v |v| + \frac{2R_v g A^2}{L} v |v| \quad (7)$$

Since $\frac{d(v^2)}{dL} = 2v \frac{dv}{dL}$ and considering the region with the lowest values of sub-atmospheric pressure pulses (where $v \geq 0$), then

$$\frac{d(v^2)}{dL} = \frac{-2p_{1,0}^* x_0^k}{\rho L (L_T - L)^k} + \frac{2p_{atm}^*}{\rho L} - 2g \sin \theta + \frac{f}{D} v^2 + \frac{2R_v g A^2}{L} v^2 \quad (8)$$

By defining the auxiliary variable z as v^2 , Equation (8) can be expressed in terms of z as follows:

$$\frac{d(z)}{dL} - \left(\frac{f}{D} + \frac{2R_v g A^2}{L} \right) z = \frac{-2p_{1,0}^* x_0^k}{\rho L (L_T - L)^k} + \frac{2p_{atm}^*}{\rho L} - 2g \sin \theta \quad (9)$$

It is observed that Equation (9) is a first-order linear differential equation with variable coefficients, where z is the unknown variable. Additionally, since the equation is non-homogeneous, its solution involves multiplying both sides by an integrating factor $e^{h(L)}$, where $h(L) = -\int \left(\frac{f}{D} + \frac{2R_v g A^2}{L} \right) dL$ is the integral of the second coefficient. Therefore, the integrating factor becomes

$$e^{h(L)} = e^{-\int \left(\frac{f}{D} + \frac{2R_v g A^2}{L} \right) dL} = e^{-\left(\frac{f}{D} L + 2R_v g A^2 (\ln L) \right)} \quad (10)$$

Multiplying Equation (9) by the integrating factor in (10) yields the following result:

$$e^{h(L)} \frac{d(z)}{dL} - e^{h(L)} \left(\frac{f}{D} + \frac{2R_v g A^2}{L} \right) z = e^{h(L)} \left(\frac{-2p_{1,0}^* x_0^k}{\rho L (L_T - L)^k} + \frac{2p_{atm}^*}{\rho L} - 2g \sin \theta \right) \quad (11)$$

By the definition of $h(L)$ and calculating $h'(L) = -\left(\frac{f}{D} + \frac{2R_v g A^2}{L} \right)$, the chain rule gives $\frac{d(e^{h(L)})}{dL} = h'(L) e^{h(L)} = -\left(\frac{f}{D} + \frac{2R_v g A^2}{L} \right) e^{h(L)}$. This allows the left-hand side of Equation (11) to be rewritten using the product rule as follows:

$$\frac{d(z e^{h(L)})}{dL} = e^{h(L)} \left(\frac{-2p_{1,0}^* x_0^k}{\rho L (L_T - L)^k} + \frac{2p_{atm}^*}{\rho L} - 2g \sin \theta \right) \quad (12)$$

By integrating Equation (12) with limits from L_0 to L , where $v(L) > 0$, and $v(S) > 0$ for all S values, thus

$$z e^{h(L)} = \int_{L_0}^L \left(e^{-\left(\frac{f}{D} S \right)} (S)^{-2R_v g A^2} \right) \left(\frac{-2p_{1,0}^* x_0^k}{\rho S (L_T - S)^k} + \frac{2p_{atm}^*}{\rho S} - 2g \sin \theta \right) dS \quad (13)$$

Dividing Equation (12) by Equation (10) yields

$$z = \left(e^{\frac{f}{D} L} L^{2R_v g A^2} \right) \int_{L_0}^L \left(e^{\frac{f}{D} S} S^{2R_v g A^2} \right)^{-1} \left(\frac{-2p_{1,0}^* x_0^k}{\rho S (L_T - S)^k} + \frac{2p_{atm}^*}{\rho S} - 2g \sin \theta \right) dS \quad (14)$$

Considering that $z = v^2$, then the water velocity can be computed as

$$v(L) = \sqrt{\left(e^{\frac{f}{D} L} L^{2R_v g A^2} \right) \int_{L_0}^L \left(e^{\frac{f}{D} S} S^{2R_v g A^2} \right)^{-1} \left(\frac{-2p_{1,0}^* x_0^k}{\rho S (L_T - S)^k} + \frac{2p_{atm}^*}{\rho S} - 2g \sin \theta \right) dS} \quad (15)$$

Equation (14) is valid for L values between L_0 and L_f , where $v(L) \geq 0$.

The solution of Equation (14) yields the integral of Equation (15).

$$\int_{L_0}^L \left(e^{\frac{f}{D} S} (S)^{2R_v g A^2} \right)^{-1} \left(\frac{-2p_{1,0}^* x_0^k}{\rho S (L_T - S)^k} + \frac{2p_{atm}^*}{\rho S} - 2g \sin \theta \right) dS \quad (16)$$

Defining $\mathcal{M}(S)$ as the integrand of Equation (15), then

$$\mathcal{M}(S) = \left(e^{\frac{f}{D} S} (S)^{2R_v g A^2} \right)^{-1} \left(\frac{-2p_{1,0}^* x_0^k}{\rho S (L_T - S)^k} + \frac{2p_{atm}^*}{\rho S} - 2g \sin \theta \right) \quad (17)$$

The function $\mathcal{M}(S)$ results in very small values that can cause an unstable solution, as demonstrated in Section 3.

In this sense, the average of the term $\left(e^{\frac{f}{D} L} (L)^{2R_v g A^2} \right)$ is selected for the solution.

$$z = \int_{L_0}^L \left(e^{\frac{f}{D} (L-S)} \left(\frac{L}{S} \right)^{2R_v g A^2} \right) \left(\frac{-2p_{1,0}^* x_0^k}{\rho S (L_T - S)^k} + \frac{2p_{atm}^*}{\rho S} - 2g \sin \theta \right) dS \quad (18)$$

Denoting $\mathcal{N}(S)$ as the integrand of Equation (17) with $z = \int_{L_0}^L \mathcal{N}(S) dS$, then

$$\mathcal{N}(S) = \left(e^{\frac{f}{D} (L-S)} \left(\frac{L}{S} \right)^{2R_v g A^2} \right) \left(\frac{-2p_{1,0}^* x_0^k}{\rho S (L_T - S)^k} + \frac{2p_{atm}^*}{\rho S} - 2g \sin \theta \right) \quad (19)$$

The resulting values of $\mathcal{N}(S)$ are bigger compared to $\mathcal{M}(S)$, yielding a reliable solution for emptying processes.

For the numerical solution of z , the Simpson's 1/3 rule with a fifth-order error was employed [20,21]. To approximate the integral of a function $\mathcal{N}(S)$ over the interval $[a, b]$, it is divided into n equal sub-intervals, each of length $h = \frac{b-a}{n}$. It is important to note that in Simpson's 1/3 rule, n must be an even number to ensure a suitable solution. By dividing the interval into n parts, the integral of $\mathcal{N}(S)$ for the values s_i with $i = 0, \dots, n$, and considering that $z = \int_{L_0}^L \mathcal{N}(S) dS$ with $\mathcal{N}(S)$, then

$$\int_{L_0}^L \mathcal{N}(S) dS \approx (L - L_0) \frac{\mathcal{N}(L_0) + 4 \sum_{i=0}^{\frac{n-2}{2}} \mathcal{N}(S_{2i+1}) + 2 \sum_{i=1}^{\frac{n-2}{2}} \mathcal{N}(S_{2i}) + \mathcal{N}(L)}{3n} \quad (20)$$

The final solution is yielded by $v = \sqrt{-\int_{L_0}^L \mathcal{N}(S) dS}$.

3. Results

3.1. Dataset

This section shows the proposed approach's application. The model was developed to predict the minimum value of air pocket pressure pulses during an emptying procedure in a single installation. The proposed approach is applied to a water pipeline considering the dataset as follows: $L_T = 350$ m, $k = 1.2$, $x_0 = 50$ m, $D = 0.25$ m, $f = 0.017$, $\theta = 0.10$ rad, and $R_v = 0.15$ ms²/m⁶. The Supplementary Materials section provides the code created using the proposed approach to model the emptying process, with calculations performed in Octave v7.1.0.

3.2. Application of the Proposed Approach

For a numerical resolution, a value of $n = 30$ was considered for the proposed approach. Figure 2 shows the obtained results that relate to the three analysed variables of the problem (water velocity, air pocket pressure, and length of the water column). The water velocity

starts with a null value, reaching a maximum of 4.77 m/s. The proposed approach predicts only the behaviour until the minimum value of the sub-atmospheric pressure head is 1.34 m. At this point, the water velocity once again reaches a zero value.

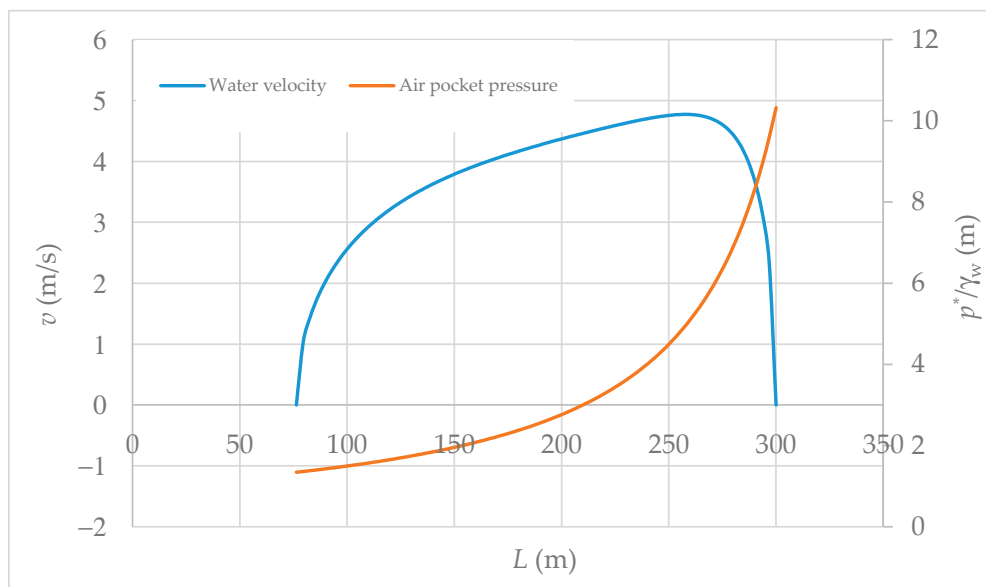


Figure 2. An analysis of hydraulic variables applying the proposed approach.

3.3. Effect of Main Parameters

The effects of the main parameters involved in this process are studied using the proposed approach. Figure 3 illustrates the relationship between the air pocket pressure head and the length of the water column, with variations in air pocket size. As expected, the smaller the air pocket size, the lower the values of sub-atmospheric pressure pulses achieved. In this instance, the air pocket size ranges from 10 to 150 m. An air pocket pressure of 0.205 m was found for an air pocket size of 10 m, while a value of 4.4 m was attained for a size of 150 m. It is paramount to remember that smaller air pocket sizes can lead to issues related to damaging pressure collapse depending on installation conditions and soil characteristics.

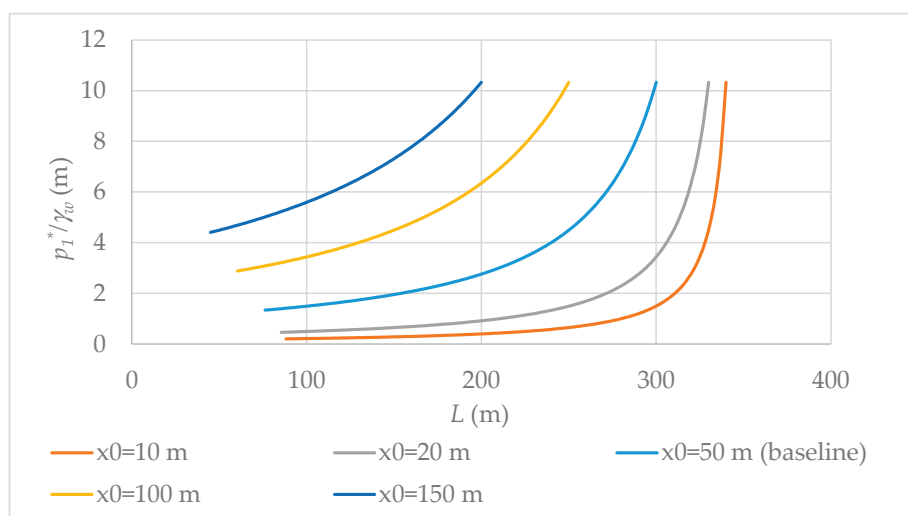


Figure 3. Relationship between air pocket pressure and water column length with varying air pocket sizes.

Another significant parameter is the selection of the polytropic coefficient. In practise, it is challenging to establish this coefficient a priori. During the design phase, engineers typically define an intermediate behaviour of the air phase ($k = 1.2$). The polytropic coefficient ranges between 1.0 (isothermal evolution) and 1.4 (adiabatic evolution). Figure 4 presents the minimum values of sub-atmospheric pressure heads against variations in the polytropic coefficient (k). Lower polytropic coefficients result in smaller sub-atmospheric pressure heads. However, there are no significant discrepancies across the entire range of the polytropic coefficient. In this sense, using $k = 1.0$, a minimum sub-atmospheric pressure of 1.31 m is obtained, while with $k = 1.4$, the air pressure reaches 1.37 m.

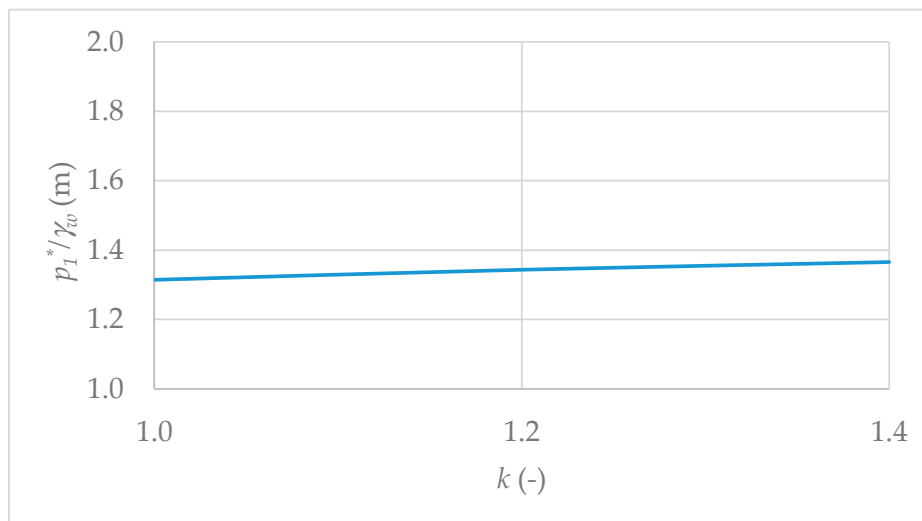


Figure 4. Minimum sub-atmospheric pressure heads as a function of the polytropic coefficient.

The internal pipe diameter was also varied to observe the behaviour of the proposed approach. Pipe diameters ranged between 0.15 and 0.40 m. No significant differences were detected when varying this parameter. For example, with a pipe diameter of 0.15 m, an air pocket pressure of 1.37 m is observed, whereas for a pipe diameter of 0.40 m, the air pocket pressure trends towards 1.31 m, as shown in Figure 5.

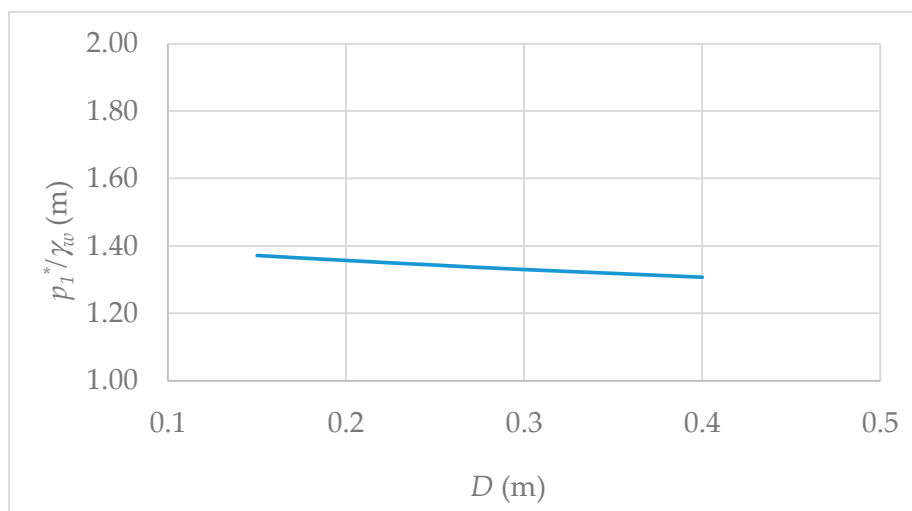


Figure 5. Variation in internal pipe diameter versus minimum sub-atmospheric pressure pulses.

Considering that the friction factor indicates how roughness can affect pipe resistance, various values ranging from 0.013 to 0.020 were analysed to observe the variations in

minimum sub-atmospheric pressure heads, as shown in Figure 6. The smaller the friction factor, the more significant the reduction in sub-atmospheric pressure. In this case, friction factor values of 0.013 and 0.020 resulted in minimum sub-atmospheric pressures of 1.32 and 1.35 m, respectively. This parameter does not significantly affect the phenomenon's behaviour during drainage events.

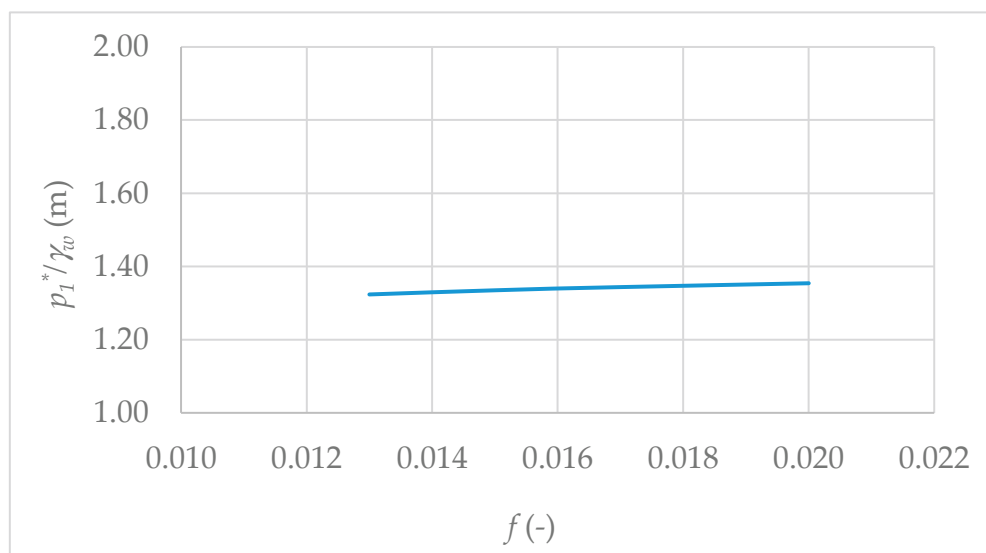


Figure 6. An analysis of the friction factor and its effect on the minimum sub-atmospheric pressure head.

Figure 7 analyses the variation in the resistance coefficient of a regulating valve, considering different aperture degrees ranging from 0.06 to 150 ms^2/m^6 . The higher the resistance coefficient, the greater the air pocket pressure achieved, with values ranging from 1.34 to 1.38 m for the full range of analysed resistance coefficients.

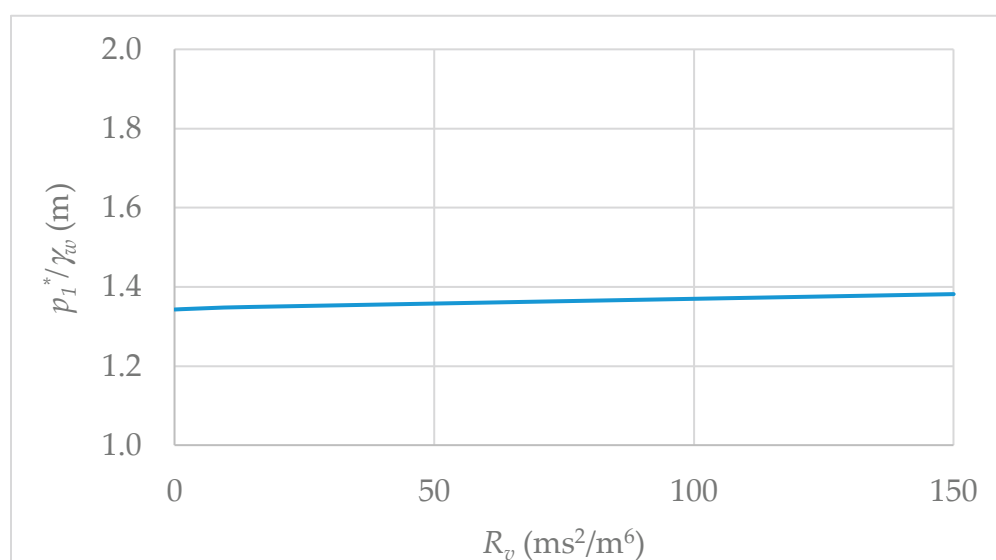


Figure 7. An analysis of the resistance coefficient of regulating valves on the minimum sub-atmospheric pressure heads.

As demonstrated in Figure 8, the bed slope significantly influences air pocket pressure pulses during emptying manoeuvres. To analyse these variations, pipe slopes from 0.05 to 0.20 radians were examined. The greater the bed slope, the lower the minimum sub-

atmospheric pressure heads attained. For instance, with a pipe slope (from 0.2 to 0.50 rad), a minimum pressure head of 1.96 m is reached, while a pipe slope of 0.20 rad results in a value of 1.13 m under the proposed approach.

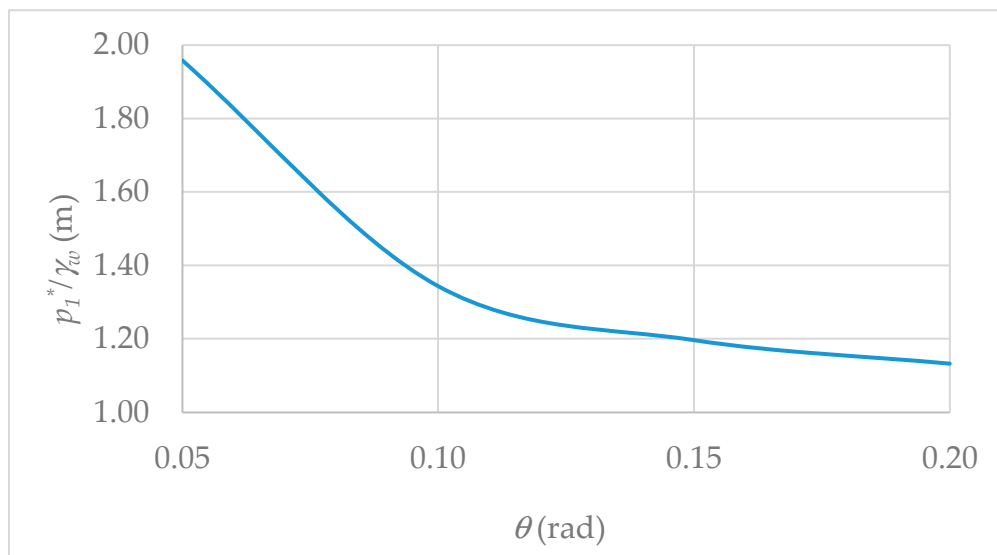


Figure 8. Influence of bed slope on minimum sub-atmospheric pressure heads.

4. Discussion

4.1. Numerical Resolution

A 1D mathematical model is compared to assess the proposed approach. The mathematical model is solved using Equations (1)–(3), which involve mass oscillation, piston flow, and polytropic formulations. These are based on physical formulations representing the emptying phenomena with entrapped air. The proposed approach uses these equations but only captures water movement until the minimum sub-atmospheric pressure occurs. As expected, both models exhibit the same behaviour, as illustrated in Figure 9. Indeed, the 1D mathematical model can replicate the transient event, but its numerical resolution is complex, as it involves a system of differential algebraic equations. The advantage of the proposed approach is that it offers a more straightforward numerical resolution.

Additionally, the results of Simpson's 1/3 rule approximation were compared with those of the trapezoidal rule, which has a third-order error, to evaluate the behaviour of the proposed approach. Both approximations yield reliable values for computing this process. For the interval $[s_i, s_{i+1}]$, the approximation of the trapezoidal rule for z is

$$\int_{L_0}^L \mathcal{N}(S) dS \approx \sum_{i=1}^n \left(\frac{L - L_0}{n} \right) \left(\frac{\mathcal{N}(S_{i-1}) + \mathcal{N}(S_i)}{2} \right) \quad (21)$$

Since the numerical resolution requires the number of intervals (n) to be specified, both even and odd numbers were utilised to assess their influence on the numerical approach. In this regard, the numerical resolution was performed using values of 2, 3, 7, 8, 30 (baseline solution), and 35, as shown in Figure 10. Even numbers provided a better fit compared to odd numbers. The greater the selected value of n , the more accurate the solution for the proposed approach. For example, using an odd number of $n = 3$, the water column length is 86.04 m for a null water velocity (the instant when the minimum air pocket pressure is reached), whereas using an even number of $n = 4$ results in a value of 82.8 m. The even number is more closely aligned with the final behaviour of the process, where a value of 76.33 m was obtained for the baseline solution ($n = 30$).

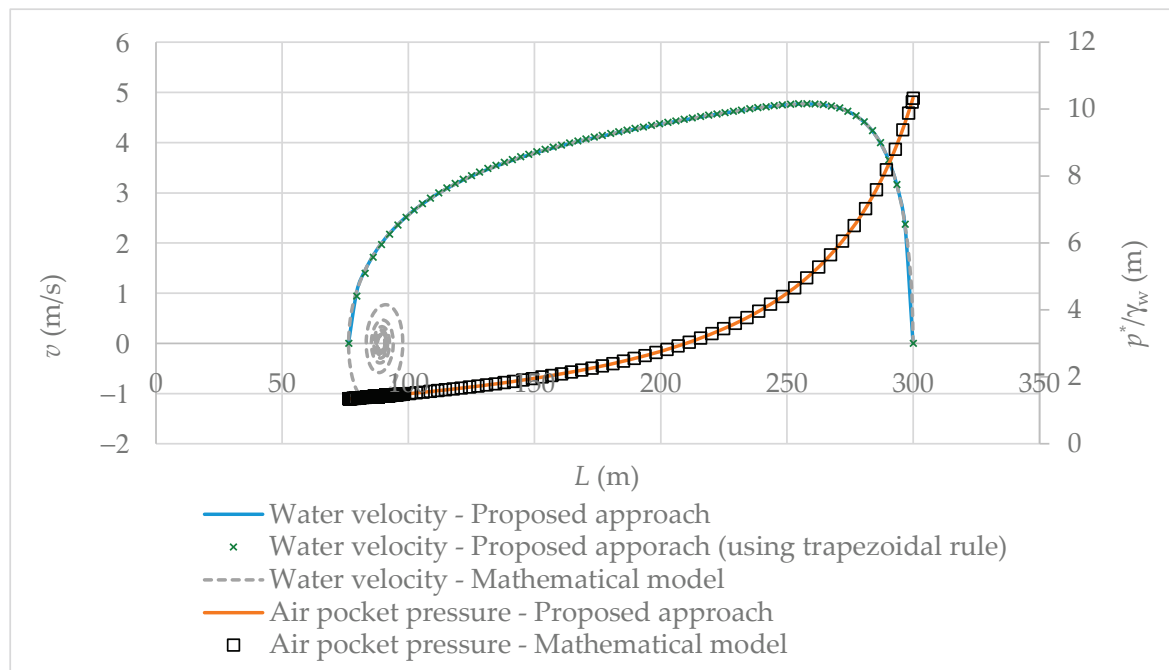


Figure 9. A comparison between the 1D mathematical model and the proposed approach.

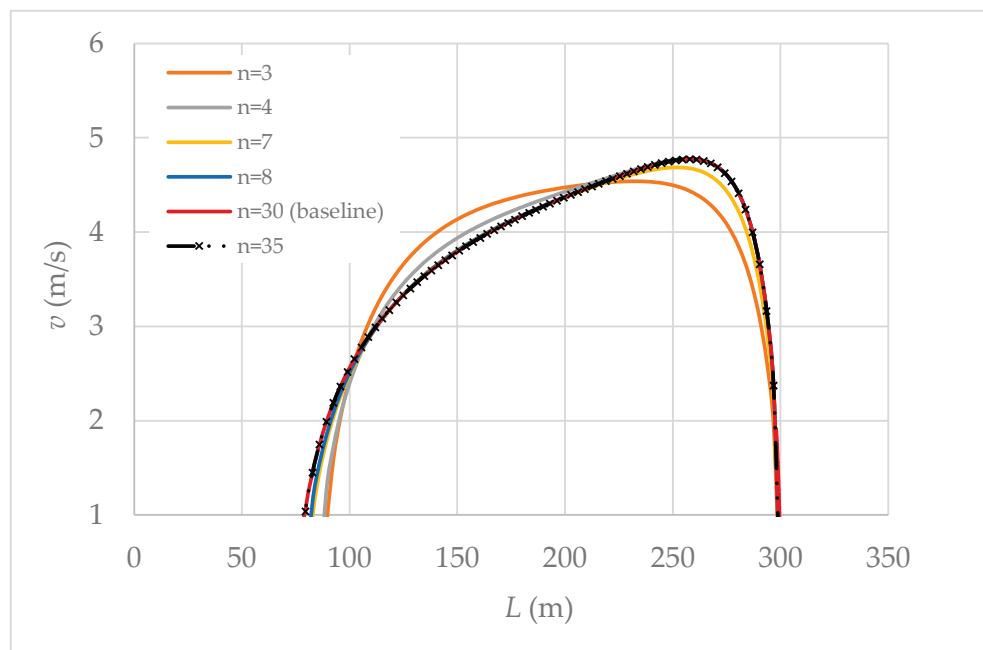


Figure 10. Variation in the number of intervals for numerical resolution.

The validation of the proposed approach also considers the assessment of $\mathcal{M}(S)$ and $\mathcal{N}(S)$, as shown in Figure 11. Figure 11a presents that the resulting $\mathcal{M}(S)$ values have a maximum in an order of magnitude around 10^{-4} , which is very small, and precision errors may arise during integration, causing the simulation to become unstable. In contrast, the $\mathcal{N}(S)$ values present a suitable value for integration purposes, as depicted in Figure 11b.

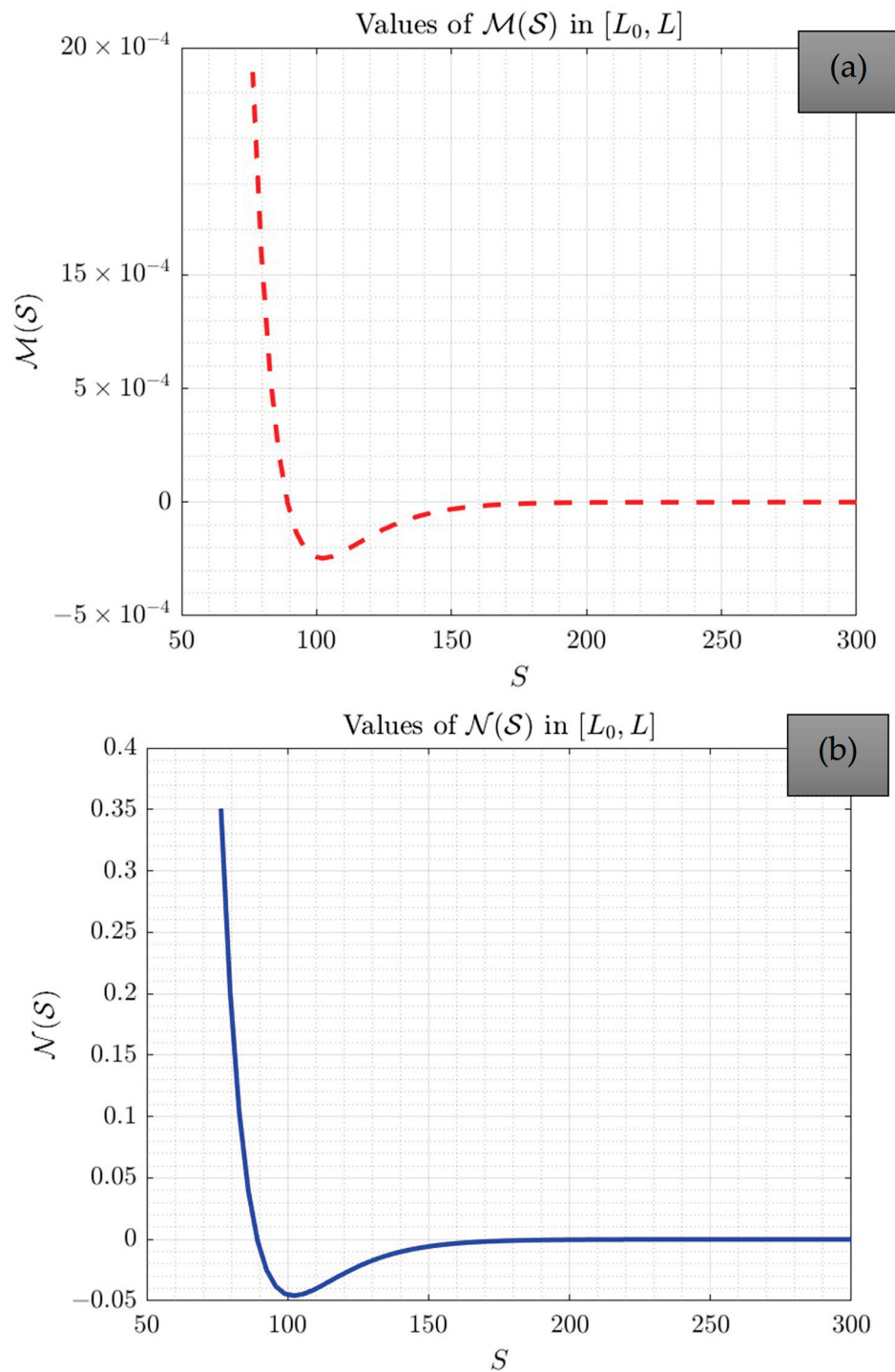


Figure 11. Computation of variables: (a) $\mathcal{M}(S)$; and (b) $\mathcal{N}(S)$.

4.2. Validation Employing Experimental Measurements

The proposed approach was validated using real-time measurements in a simple water pipeline, which the authors conducted. Data were obtained using a pressure transducer located at the highest point of the water installation, which consisted of a 4.36 m long pipe with an internal diameter of 42 mm. During the measurements, twelve (12) tests were carried out, varying the air pocket sizes from 0.205 to 0.450 m, with two slopes (0.457 and 0.515 rad) and employing opening manoeuvres in the regulating valve with resistance coefficients ranging from 11.89 to 138.41 ms^2/m^6 . During the experiments, the opening

times of the regulating valve ranged from 0.15 to 0.75 s. Details of the experiments can be found in the publication conducted by Fuertes-Miquel et al. (2018).

The proposed approach considers an instantaneous opening of regulating valves, expected to yield sub-atmospheric pressure values lower than those observed in real-time experiments. This is significant, as the approach provides values that water utilities can use to plan emptying operations in real-world water installations. The proposed approach was validated by comparing the minimum sub-atmospheric pressure values obtained through real-time measurements. Additionally, both a one-dimensional (1D) mathematical model (comprising the numerical resolution of Equations (1)–(3)) and a three-dimensional (3D) CFD model were employed to simulate the current emptying operation.

Notably, for the current problem, the following equations are required:

Continuity equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \quad (22)$$

X-momentum equation:

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} + \frac{\partial(\rho uw)}{\partial z} = -\frac{\partial p}{\partial x} + \frac{1}{Re_t} \left[\frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} \right] \quad (23)$$

Y-momentum equation:

$$\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho v^2)}{\partial y} + \frac{\partial(\rho vw)}{\partial z} = -\frac{\partial p}{\partial y} + \frac{1}{Re_t} \left[\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} \right] \quad (24)$$

Z-momentum equation:

$$\frac{\partial(\rho w)}{\partial t} + \frac{\partial(\rho uw)}{\partial x} + \frac{\partial(\rho vw)}{\partial y} + \frac{\partial(\rho w^2)}{\partial z} = -\frac{\partial p}{\partial z} + \frac{1}{Re_t} \left[\frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} \right] \quad (25)$$

Energy equation:

$$\begin{aligned} & \frac{\partial(E_T)}{\partial t} + \frac{\partial(uE_T)}{\partial x} + \frac{\partial(vE_T)}{\partial y} + \frac{\partial(wE_T)}{\partial z} \\ &= -\frac{\partial(u p)}{\partial x} - \frac{\partial(v p)}{\partial y} - \frac{\partial(w p)}{\partial z} - \frac{1}{Re_t Pr_t} \left[\frac{\partial q_x}{\partial x} + \frac{\partial q_y}{\partial y} + \frac{\partial q_z}{\partial z} \right] \\ &+ \frac{1}{Re_t} \left[\frac{\partial}{\partial x} (u \tau_{xx} + v \tau_{xy} + w \tau_{xz}) + \frac{\partial}{\partial y} (u \tau_{xy} + v \tau_{yy} + w \tau_{yz}) + \frac{\partial}{\partial z} (u \tau_{xz} + v \tau_{yz} + w \tau_{zz}) \right] \end{aligned} \quad (26)$$

where x , y , and z = tridimensional coordinate system; t = time; p = fluid pressure; ρ = fluid density; τ = fluid stress; q = heat flux; E_T = total energy of a fluid; Re_t = Reynolds number; u , v , and w = fluid velocity components in direction x , y , and z , respectively; and Pr_t = Prandtl number.

According to Paternina-Verona et al. (2023) [11], a 3D CFD model for emptying processes can be implemented based on the following assumptions: (i) non-isothermal and immiscible compressible flows; (ii) the air phase can be treated as an ideal gas; (iii) water density remains constant over time, as air elasticity is significantly higher compared to the water phase; and (iv) the pipe walls are modelled as rigid elements. The $k-\omega$ SST turbulence model was employed for the simulations, a model widely adopted by researchers to address air–water interface issues. Simulations were conducted using the OpenFOAM v2012 software with the compressibleInterFoam solver, which is well suited for modelling mixtures of air and water. For the calculations, the aspect ratio, non-orthogonality, and skewness were analysed across five meshes to validate the adopted 3D configuration.

The results of the 1D mathematical model and the 3D CFD model were compared to assess the proposed approach, as shown in Figure 12. As expected, the 1D mathematical model and 3D CFD model calibrated better than the proposed approach, since they are

based on physical formulations. However, applying the proposed approach is more accessible than these models. The 1D mathematical model requires a numerical resolution that involves ordinary differential equations, while the 3D CFD needs high computational times. The proposed approach is suitable for computing minimum values of air pocket pressure heads, being a practical tool that water utilities can use to predict rapidly emptying operations in actual water installations.

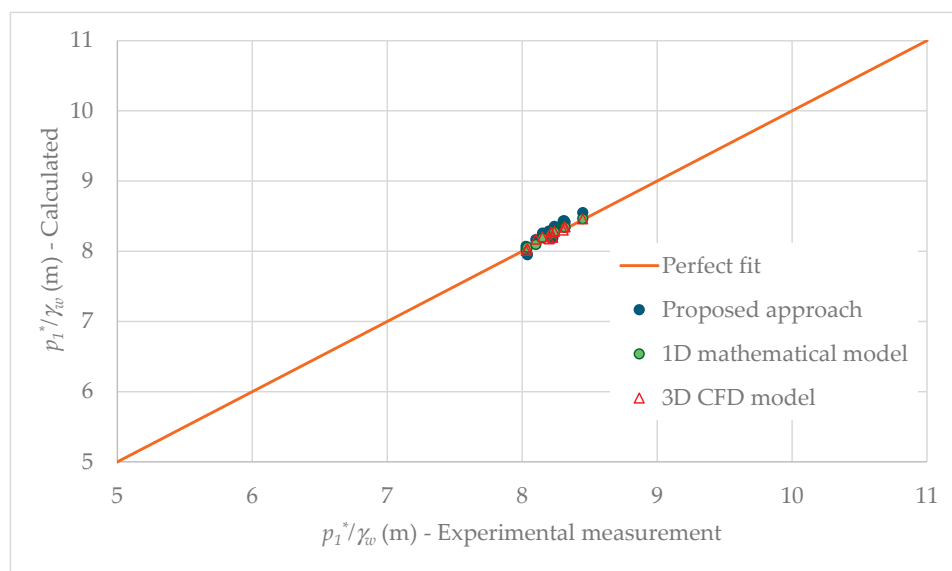


Figure 12. Minimum attained air pocket pressure heads for 1D and 3D models.

The root mean square error (*RMSE*) and the correlation coefficient were computed to assess the fit of the proposed approach numerically and its corresponding comparison with the 1D mathematical model and 3D CFD model, as presented in Table 1. It is essential to highlight that the proposed approach has good accuracy, with an *RMSE* of 1.13% and an R^2 of 0.95. The used formulations are described as follows:

$$RMSE = \sqrt{\frac{1}{N} \sum_{j=1}^N \left(\frac{p_{1,cal}^* - p_{1,exp}^*}{p_{1,exp}^*} \right)^2} \quad (27)$$

$$R^2 = \frac{\sum_{j=1}^N (p_{1,cal,j}^* - \overline{p_{1,cal}^*}) (p_{1,exp,j}^* - \overline{p_{1,exp}^*})}{\sqrt{\sum_{j=1}^N (p_{1,cal,j}^* - \overline{p_{1,cal}^*})^2} \sqrt{\sum_{j=1}^N (p_{1,exp,j}^* - \overline{p_{1,exp}^*})^2}} \quad (28)$$

where *RMSE* = root mean square error; R^2 = correlation coefficient; $p_{1,cal}^*$ = air pocket pressure calculated using the proposed approach, 1D mathematical model, or 3D CFD model; $p_{1,exp}^*$ = experimental measurement; and *N* = total analysed points.

Table 1. Computation of root mean square errors and correlation coefficients.

Model	<i>RMSE</i> (%)	R^2
Proposed approach	1.13	0.95
1D mathematical model	0.33	0.99
3D CFD model	0.47	0.96

5. Conclusions

In this study, a new mathematical expression was developed to directly model the transient events in emptying operations within single water installations, including biphasic

flow. The mathematical approach was derived by solving the system of algebraic differential equations from the 1D model proposed by the authors. This method offers a more straightforward computational approach compared to previous models. The proposed approach was validated using an experimental facility with a pipe length of 4.36 m and an internal diameter of 42 mm. The mathematical approach can compute the minimum values of the sub-atmospheric air pocket pressure head with a root mean square error of 1.13%, slightly higher than the 1D mathematical model (0.33%) due to the consideration of instantaneous valve opening manoeuvres.

This research provides a foundation for water utilities to estimate the minimum sub-atmospheric air pressure pulses that may occur during emptying processes. Incorporating such expressions in digital twin models should be considered for predicting these phenomena. The proposed mathematical approach enables water utilities to select an appropriate pipe stiffness class based on soil characteristics and the drop in sub-atmospheric pressure pulses calculated by the model.

Supplementary Materials: The following supporting information can be downloaded at: <https://www.mdpi.com/article/10.3390/w16223203/s1>. This section includes the code programmed in Octave v7.1.0, which solved the emptying process for the dataset presented in Section 3.1.

Author Contributions: Conceptualization, D.M.B.-C. and O.E.C.-H.; methodology, D.M.B.-C., O.E.C.-H. and M.P.-S.; formal analysis, D.M.B.-C., A.A.-P., O.E.C.-H. and H.M.R.; writing—original draft preparation, A.A.-P., V.S.F.-M. and O.E.C.-H.; supervision, A.A.-P., V.S.F.-M. and H.M.R. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Data Availability Statement: Data are contained within the article.

Conflicts of Interest: The authors declare no conflicts of interest.

References

1. Fuertes-Miquel, V.S.; Coronado-Hernández, O.E.; Iglesias-Rey, P.L.; Mora-Meliá, D. Transient Phenomena during the Emptying Process of a Single Pipe with Water–Air Interaction. *J. Hydraul. Res.* **2019**, *57*, 318–326. [CrossRef]
2. Tijsseling, A.S.; Hou, Q.; Bozkuş, Z.; Laanearu, J. Improved One-Dimensional Models for Rapid Emptying and Filling of Pipelines. *J. Press. Vessel. Technol.* **2015**, *138*, 31301. [CrossRef]
3. Laanearu, J.; Annus, I.; Koppel, T.; Bergant, A.; Vučković, S.; Hou, Q.; Tijsseling, A.S.; Anderson, A.; van’t Westende, J.M.C. Emptying of Large-Scale Pipeline by Pressurized Air. *J. Hydraul. Eng.* **2012**, *138*, 1090–1100. [CrossRef]
4. Coronado-Hernández, O.E.; Fuertes-Miquel, V.S.; Besharat, M.; Ramos, H.M. Subatmospheric Pressure in a Water Draining Pipeline with an Air Pocket. *Urban Water J.* **2018**, *15*, 346–352. [CrossRef]
5. Chen, X.; Hou, Q.; Laanearu, J.; Tijsseling, A.S. Experimental Data on Filling and Emptying of a Large-Scale Pipeline. *Sci. Data* **2024**, *11*, 603. [CrossRef] [PubMed]
6. Baranovskii, E.S.; Burmasheva, N.V.; Prosviryakov, E.Y. Exact Solutions to the Navier–Stokes Equations with Couple Stresses. *Symmetry* **2021**, *13*, 1355. [CrossRef]
7. Tiwari, G.; Kumar, J.; Prasad, V.; Patel, V.K. Utility of CFD in the Design and Performance Analysis of Hydraulic Turbines—A Review. *Energy Rep.* **2020**, *6*, 2410–2429. [CrossRef]
8. Olsen, N.R.B. Three-Dimensional CFD Modeling of Self-Forming Meandering Channel. *J. Hydraul. Eng.* **2003**, *129*, 366–372. [CrossRef]
9. He, J.; Hou, Q.; Lian, J.; Tijsseling, A.S.; Bozkus, Z.; Laanearu, J.; Lin, L. Three-Dimensional CFD Analysis of Liquid Slug Acceleration and Impact in a Voided Pipeline with End Orifice. *Eng. Appl. Comput. Fluid Mech.* **2022**, *16*, 1444–1463. [CrossRef]
10. Liu, E.; Ma, X.; Zhou, M. Analysis of Discharge Process of Oil Pipeline with Complex Topography. *Energy Rep.* **2019**, *5*, 678–687. [CrossRef]
11. Paternina-Verona, D.A.; Coronado-Hernández, O.E.; Aguirre-Mendoza, A.M.; Espinoza-Román, H.G.; Fuertes-Miquel, V.S. Three-Dimensional Simulation of Transient Flows during the Emptying of Pipes with Entrapped Air. *J. Hydraul. Eng.* **2023**, *149*, 4023007. [CrossRef]
12. Chen, Y. *System Simulation Techniques with MATLAB and Simulink*; John Wiley & Sons: Hoboken, NJ, USA, 2013.
13. Eaton, J.W. *GNU Octave 4.2 Reference Manual*; Samurai Media Limited: Surrey, UK, 2017.
14. Madenci, E.; Guven, I. Fundamentals of ANSYS. In *The Finite Element Method and Applications in Engineering Using ANSYS®*; Springer: New York, NY, USA, 2006; pp. 15–35.

15. Chen, G.; Xiong, Q.; Morris, P.J.; Paterson, E.G.; Sergeev, A.; Wang, Y. OpenFOAM for Computational Fluid Dynamics. *Not. AMS* **2014**, *61*, 354–363. [CrossRef]
16. Hu, H.; Zhang, J.; Li, T. Dam-Break Flows: Comparison between Flow-3d, MIKE 3 FM, and Analytical Solutions with Experimental Data. *Appl. Sci.* **2018**, *8*, 2456. [CrossRef]
17. Chang, H.-C.; Noorizadegan, A.; Liu, Y.-H.; Ma, K.-T. A Study on Offshore Anchor Selection with a Focus on Torpe-do Anchor Stability and Performance. *J. Mar. Sci. Eng.* **2024**, *12*, 1721. [CrossRef]
18. Besharat, M.; Coronado-Hernández, O.E.; Fuertes-Miquel, V.S.; Viseu, M.T.; Ramos, H.M. Backflow Air and Pressure Analysis in Emptying a Pipeline Containing an Entrapped Air Pocket. *Urban Water J.* **2018**, *15*, 769–779. [CrossRef]
19. Zill, D.G. *Differential Equations with Boundary-Value Problems*; Cengage Learning: Boston, MA, USA, 2016.
20. Hoffman, J.D.; Frankel, S. *Numerical Methods for Engineers and Scientists*; CRC Press: Boca Raton, FL, USA, 2018.
21. Stoer, J.; Bulirsch, R.; Bartels, R.; Gautschi, W.; Witzgall, C. *Introduction to Numerical Analysis*; Springer: Berlin/Heidelberg, Germany, 1980; Volume 1993.

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

Real-Time Analysis and Digital Twin Modeling for CFD-Based Air Valve Control During Filling Procedures

Duban A. Paternina-Verona ^{1,2,*}, Oscar E. Coronado-Hernández ³, Modesto Pérez-Sánchez ⁴ and Helena M. Ramos ^{5,*}

¹ School of Civil Engineering, Universidad del Sinú, Cartagena 13001, Colombia

² Facultad de Ingeniería, Universidad Tecnológica de Bolívar, Cartagena 131001, Colombia

³ Instituto de Hidráulica y Saneamiento Ambiental, Universidad de Cartagena, Cartagena 130001, Colombia; ocoronadoh@unicartagena.edu.co

⁴ Departamento de Ingeniería Hidráulica y Medio Ambiente, Universitat Politècnica de València, 46022 Valencia, Spain; mopesan1@upv.es

⁵ Civil Engineering, Architecture and Environment Department, CERIS, Instituto Superior Técnico, University of Lisbon, 1049-001 Lisbon, Portugal

* Correspondence: duban.paternina@unisnu.edu.co (D.A.P.-V.); helena.ramos@tecnico.ulisboa.pt (H.M.R.)

Abstract: Air exchange in pressurized water pipelines is an essential but complex aspect of pipeline modeling and operation. Implementing effective air management strategies can yield numerous benefits, enhancing the system's energy efficiency, reliability, and safety. This paper comprehensively evaluates an irregular profile pipeline filling procedure involving air-release through an air valve. The analysis includes real-time data tests and numerical simulations using Computational Fluid Dynamics (CFD). A Digital Twin model was proposed and applied to filling maneuvers in water installations. In particular, this research considers an often-overlooked aspect, such as filling a pipe with an irregular profile rather than a simple straight pipe. CFD simulations have proven to capture the main features of the transient event, which are suitable for tracking the air-water interface, the unsteady water flow, and the evolution of the trapped air pocket. Thus, they provide thorough and reliable information for real-time operational processes in the industry, focusing on the filling pressure and geometry of the air-valve hydraulic system. Additionally, this study provides details regarding the application of an efficient Digital Twin CFD approach, demonstrating its feasibility in optimizing the filling procedure in pipes with irregular profiles.

Keywords: air-release; air valves; air-water interaction; big data platform; computational fluid dynamics; Digital Twin; real-time data; trapped air

1. Introduction

Entrapped air in water pipelines encompasses a range of complex hydraulic and thermodynamic behaviors that must be studied. These interactions can affect the pipeline's structural integrity and hydraulic devices [1]. Sub-atmospheric pressures pose risks for unburied large-diameter and thin-walled pipes [2]. Pressure surges can increase the probability of pipe ruptures [3,4], especially in old pipes [5]. Effects resulting from the interaction between water and air within a pipeline include the following: (i) a reduction in pipeline water conveyance capacity due to entrapped air, which restricts the cross-sectional area available for the water phase [6]; (ii) alterations in the physical and chemical properties of the conveyed water [7]; (iii) movement of air pockets along the pipeline from low to high points; (iv) fluctuations in air pocket volume and pressure; and (v) temperature during transient events [8].

During pipeline filling maneuvers, pressure surges can be attributed to two primary reasons. First, for having undersized or absence of air valves in hydraulic installations,

where water columns rapidly compress entrapped air pockets. The second reason corresponds to when a water column closes the orifice of air valves by its high velocity, producing a second pressure surge that, in some cases, can deliver higher peaks of air pocket pressure compared to the first scenario. Consequently, filling procedures must be carefully performed with recommended water velocities and differential pressures [9]. In addition, these processes should be performed slowly and using well-sized air valves, which is crucial to mitigate dangerous transient oscillations [10].

Air/vacuum valves are used for filling and draining operations, while air-release valves are responsible for expelling small air bubbles during normal operation of water distribution systems [11]. For sizing purposes, it should be noted that air valve capacity depends on its orifice size and internal configuration. Air valve flow depends on the differential pressure, which can be modeled using the information provided by the manufacturer. It is also essential to select a reliable manufacturer with tested air valve capacity curves [12].

Different authors have developed mathematical models to understand the dynamic behavior of trapped air pockets in pipelines. Zhou et al. (2002) [13] studied air trapped in pipes with orifice sizes, identifying pressure oscillation patterns and the resulting pressure surges during rapid filling processes. De Martino et al. (2008) [14] developed a predictive equation derived from upstream conditions, orifice size, and pumping pressures of a water pipeline, establishing two phases of pressure oscillation: (i) an initial with low frequency during air-release and (ii) an overpressure caused by water hammer phenomena. Carlos et al. (2011) [15] studied the impact of trapped air on pipeline water transients using tests and a numerical model calibrated in experimental facilities, which can be used to predict the dynamic effects of air-release through different air valves and the overpressure that can cause damage in water distribution networks. Fuertes-Miquel et al. (2016) [16] analyzed the behavior of air when expelled through air valves through a mathematical model, focusing on the variation of isothermal and adiabatic conditions, impacting the effects of water hammers when water columns abruptly close air valves. Coronado-Hernández et al. (2019) [17] developed a rigid column mathematical model to simulate the filling of water pipelines with air valves, capturing hydraulic and thermodynamic variables and considering different physical equations such as the mass oscillation formula, air-water interface position, and the polytropic equation. Zhou et al. (2019) [18] investigated air expulsion in a vertical pipe through a mathematical model of elastic water columns, where they studied two stages of the filling processes: (i) pressurization of trapped air and (ii) water hammer impact. In addition, they explored different orifice sizes, observing air-damping effects in pipes with small orifices and dominant water hammer pressure in larger orifices. Romero et al. (2020) [19] presented a mathematical model to analyze pressure changes during the filling of hydraulic systems, focusing on large-scale pipelines. This mathematical model was initially validated by Coronado-Hernández et al. (2019) [17], and they were applied to pipelines with DN400 and DN600 [19]. The application to actual cases helped to understand transient events during filling processes. Zhou et al. (2019) [18] studied the expulsion of air through air valves in detail. In addition, they developed a mathematical model that considered the effects of trapped air and the water hammer phenomenon, which allowed a better understanding of the process and the factors that influence it.

A detailed understanding of the behavior of expelled air in pressurized pipelines.

Computational Fluid Dynamic (CFD) models have been complex, and information about this phenomenon is rarely available and focused on two essential aspects: expelled air in horizontal stormwater pipes with free surface flows and pressurized air using three-dimensional (3D) CFD models. For example, Li et al. (2018) [20] used a 3D CFD model to examine air-water interactions for expelled air in a horizontal pipe with trapped air during a rapid filling, revealing complex flow features and pressure patterns based on orifice position. Chan et al. (2018) [21] developed a 3D CFD model to study geysers in stormwater systems. They studied geyser formation through the dynamics of air pockets during the expulsion of air from horizontal hydraulic sections to vertical pipes. Wang and Vasconcelos

(2018) [22] developed 3D CFD simulations of a stormwater pipeline with pressurized air in which it was shown that large air pocket leaks generated pressure spikes that exceeded roof displacement thresholds, offering ideas for managing air at stormwater crossings and avoiding operational problems. Fang et al. (2022) [23] developed a 3D CFD model applying the VOF method to simulate air-water interaction in T-junction stormwater pipes. They appropriately replicated experimental pressure oscillations, identifying air movement and expulsion in vertical pipes without geysers and air expulsion and critical geyser formation.

On the other hand, the movement and expelled air flow in pipelines of irregular profiles have been scarcely studied through 2D CFD models [24], where 2D CFD models have been developed to study rapid filling processes in pressurized pipes with single entrapped air pockets. These models helped us understand the influence of air valve sizing on controlling water hammer phenomena and the effects of different commercial air valves.

The advent of extensive data analysis and digitalization is transforming water infrastructure, necessitating standardization and systems that ensure the security of drinking water pipelines. In this context, Digital Twin technology is being deployed in water distribution systems to manage water leakages and assist with design purposes. However, the current literature needs more specific documentation for predicting filling operations. Since previous research has focused on traditional 2D CFD models that predict only physical patterns of hydraulic and thermodynamic parameters as well as contour analysis, this work contributes to the presentation of a 3D CFD model that combines real-time data integration from big data analysis techniques—leveraging sensor technology linked to safety devices like air valve operations during the start-up of water pipelines. The proposed model can be used to have additional analysis as follows: (i) a comprehensive study of the hydraulic and thermodynamic behaviors of air-water interaction during rapid filling and air expelling processes in irregular pipes with different branches and different air pocket sizes; and (ii) a three-dimensional analysis of the effects of expelled air on the proper management of trapped air during filling maneuvers. Real-time data and numerical analysis are performed in this research through an experimental setup, which is carried out to represent different filling processes through a pipeline with horizontal and diagonal branches where different tests are performed with an air valve and different initial hydraulic conditions. A three-dimensional CFD model of the experimental setup is developed in OpenFOAM v2012 software using a multiphase model to understand better air-water interaction and relevant hydraulic-thermodynamic parameters resulting from the release of trapped air. The numerical model is validated with pressure patterns obtained from recorded data and calibrated through mesh sensitivity. Numerical and visual information is obtained to understand the behavior of the trapped air inside the pipes and during the expulsion processes.

2. Methodology

2.1. Digital Twin

This section details the development of the proposed Digital Twin applied to filling operations. Figure 1 illustrates the key components of the proposed model for filling maneuvers.

Filling operations can be carried out by the recommendations provided by the AWWA, which advise that these processes should be performed as slowly as possible [25]. The big data platform relates various components. The positioning of system elements pertains to pipes, junctions, valves, reservoirs, and hydraulic devices, which must be located using Geographical System Information (GIS). Sensors measure key hydraulic and thermodynamic parameters within water distribution systems. Pressure transducers (sensors) must be installed based on the behavior of transient events, with their placement determined by the required frequency. These sensors should be positioned near air valves. The design of maneuvers in regulating valves—whether manual, automatic, or electro-pneumatic—must be planned to prevent pipeline ruptures. Additionally, flow meters should be installed along water pipelines to monitor changes in water velocity over time. An electromagnetic flow meter or Ultrasonic Doppler Velocimetry can be selected depending on the required

frequency. An oscilloscope system should be configured to translate hydraulic and thermodynamic variables into electrical signals. An air-water interface can be captured using cameras for transparent pipes. A trigger can be used as an electrical strategy to synchronize the different devices to start all measurements simultaneously. In this sense, the Internet of Things (IoT) is essential during this stage. Subsequently, a 3D CFD model must be implemented to simulate critical scenarios, such as small air pockets trapped in water installations. Currently, computational times present a challenge during the simulation phase. Implementing such a model involves defining the water infrastructure geometry, analyzing the sensitivity of a selected mesh, choosing a turbulence model and its associated numerical resolution scheme, establishing initial and boundary conditions, and executing an appropriate visualization process. In this context, the parameters of a 3D CFD model must be calibrated using experimental measurements to ensure the model is suitable for predicting filling operations. Once a reliable 3D CFD model is established, various scenarios can be simulated to analyze the behavior of filling processes under different initial pressures and air pocket sizes.

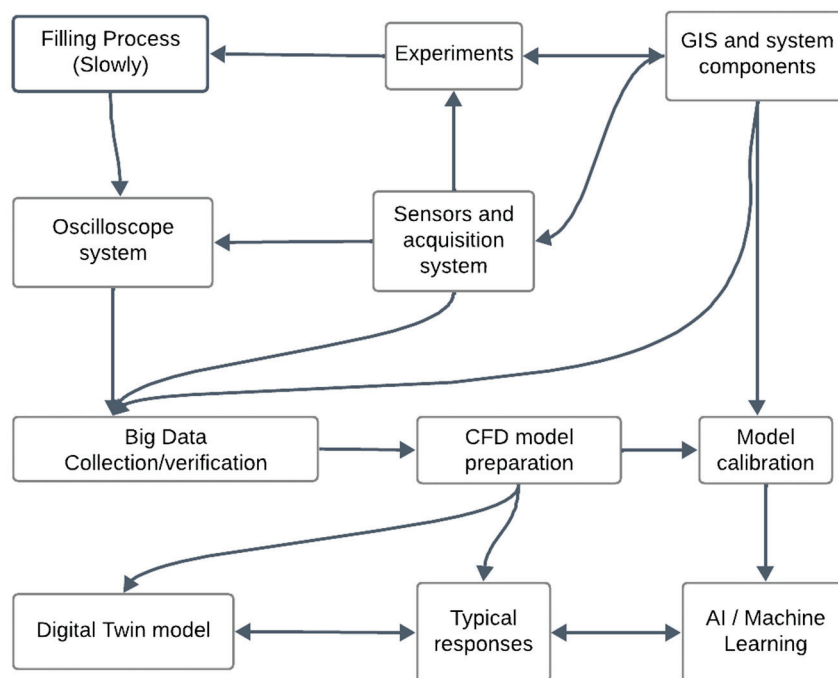


Figure 1. Components of the proposed Digital Twin for filling maneuvers.

The proposed Digital Twin integrates a big data platform, a 3D CFD model, and artificial intelligence techniques (see Figure 1).

2.2. Big Data Platform

This study developed the big data platform considering an experimental facility at the Hydraulic Laboratory of the Instituto Superior Técnico of the University of Lisbon. The facility consists of a pipeline with an inner diameter of 0.0514 m and a total length of 7.3 m. The pipeline has two branches: a left branch segmented by a horizontal and diagonal part ($L_{l,h}$ and $L_{l,d}$ with lengths of 2.05 m and 1.5 m, respectively) and a right branch segmented in branches $L_{r,d}$ and $L_{r,h}$ with lengths of 1.5 m and 1.85 m, respectively, as shown in Figure 2.

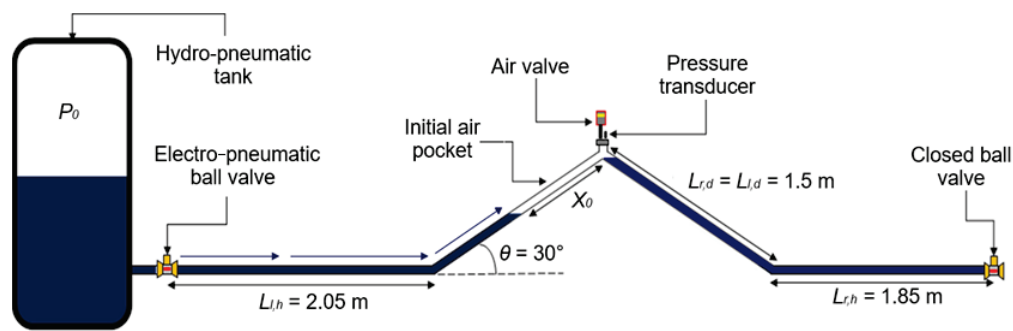


Figure 2. A schematic experimental setup with details on the accessories used.

The air pocket pressure pulses were recorded employing the pressure transducer with S-10 WIKA from Germany, which is suitable for values up to 25 bar. This sensor has an analog signal with an electric current of 4 to 20 mA.

Real-time data analysis considered a ball valve closed at the end of the $L_{r,h}$. An electro-pneumatic ball valve guarantees a water inflow from the hydro-pneumatic tank located at the end of $L_{l,h}$, and an air valve located at the upper end of the hydraulic system was used to ensure adequate air expulsion. The analysis defines an initial air pocket with length X_0 . A pressure transducer measures entrapped air's pressure oscillations at the hydraulic system's highest point. Air valve ARI S050 with an inner diameter (d_0) of 3.175 mm was used during the tests. This air valve can expel air at a rate of $80 \text{ m}^3/\text{s}$, assuming a differential pressure of 8 bar. The real-time data involves water movement from the hydro-pneumatic tank to the pipeline using an initial pressure (p_0).

An electro-pneumatic ball valve was configured during the real-time data analysis for all tests from the Georg Fischer (+GF) company (Schaffhausen, Switzerland). It works with an electrical signal to activate the opening maneuver. The model 546 DN63-DN50 was employed in this research, with a pressure range varying from 5.6 to 7.0 bar. The electro-pneumatic ball valve opens for 0.20 s to generate the inflow from the hydro-pneumatic tank and the pipe filling. Once amplified, the signals are captured with a PicoScope model 3424. This device processes the air pocket pressure signals, converts them into digital form, filters out any noise, and records the data on a computer.

During filling, the air valve guarantees the expelling of the entrapped air, so the water column subsequently occupies the hydraulic installation. To represent different hydraulic scenarios, six tests were performed using various values of hydro-pneumatic pressure (p_0) and X_0 . Hydro-pneumatic pressure varying from 20 to 50 kPa was used during the experimental stage. The hydraulic installation is at rest at the start of the transient event. The event is initiated by the opening of the electro-pneumatic ball valve, with six tests previously configured to generate different air pocket pressure patterns. Initial conditions were established considering pipeline conditions that must be conducted by water utilities, where the air pocket size needs to be filled by the water phase. This research is suitable for representing the assessment suggested by the AWWA in terms of water velocities and pressure variations [25]. Table 1 shows the initial conditions of the tests.

Table 1. Initial conditions of experimental tests.

Test No.	p_0 (kPa)	X_0 (m)
1	20	0.46
2	20	0.96
3	20	1.36
4	50	0.46
5	50	0.96
6	50	1.36

3. Digital Twin—CFD Model

A three-dimensional computational fluid dynamic (CFD) model based on the PVoF method is developed to understand the entrapment and expelling of air. The air-water interaction in the CFD model considers hydraulic and thermodynamic assumptions: (i) two fluids (liquid and gas) were considered compressible fluids with non-homogeneous temperatures and immiscible, and (ii) roughness and water surface tension effects (F_s) were considered due to the significant ratio between absolute roughness (K_s) and pipe diameter (D). Appendix A shows the governing equations of the 3D CFD model.

3.1. Numerical Approach

The numerical approach to rapid hydraulic transients in pipelines is solved through second-order numerical schemes based on the Gauss linear upwind scheme due to the quick convergence to the numerical solution and the guarantee of temporal stability, especially regarding pressure, flux velocity, and turbulence. On the other hand, temperature is solved using a first-order numerical scheme of Gauss Upwind to optimize the solution time and prioritize the numerical solution of flow parameters (pressure and velocity). On the other hand, the Pressure-Implicit with Splitting of Operators (PISO) algorithm is used to solve the equations. This iterative algorithm efficiently handles the coupling between the pressure and velocity fields by splitting the pressure and velocity updates into separate steps, which favors the reduction of computational costs, and is suitable for these scenarios associated with transient flows and is commonly used in the modeling of flows in complex geometries and with frequent turbulent phenomena.

3.2. Geometry and Mesh

The irregular pipeline and the air valve are represented using a three-dimensional geometry composed of a longitudinal pipe of 6.9 m in length from the electro-pneumatic ball valve to the closed ball valve. The pipe has a diameter of 51.4 mm. Certain assumptions were considered to represent the irregular pipe in this study:

- The electro-pneumatic valve opens from the instant $t = 0.0$ s, and the effect of the opening is negligible.
- The ball valve in the experimental setup's right sector is closed. Therefore, this element needs to be addressed in the geometry of the CFD model.
- The effect of the air valve is represented by a small pipe with a nominal diameter of one inch (1") and an orifice with a diameter of 3.175 mm, corresponding to the discharge orifice of the S050 air valve.

The geometry is discretized into unstructured cells, with a maximum cell size of 0.005 m, ensuring a detailed representation of the flow domain. In particular, the region around the air valve section is further refined, with a maximum cell size reduced to 0.001 m, as shown in Figure 3. This mesh configuration comprises 330,793 cells, all of which are tetrahedral. The cell number selection process was based on the mesh independence analysis performed by the authors in previous investigations [26]. The number of cells selected reflects an acceptable mesh resolution suitable for capturing the complexities of flow phenomena within the pipeline at the experimental scale. The sensitivity analysis was performed, showing that the number of cells is independent of pressure patterns, which favors using this number of cells [26]. Moreover, choosing a fine mesh facilitates the physical analysis that governs the expulsion of air through the air valve S050.

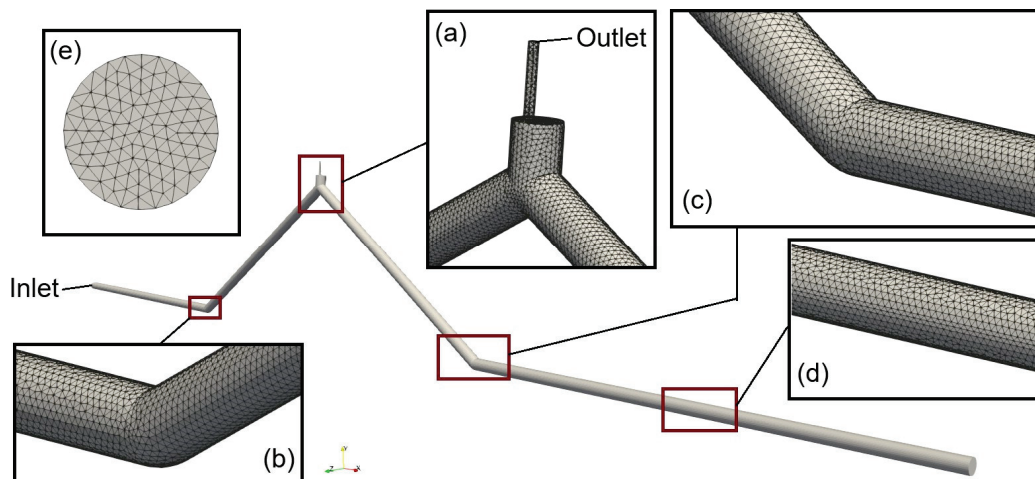


Figure 3. Mesh configuration and boundaries of CFD model: (a) air valve device; (b) walls of the elbow at the left branch; (c) walls of the elbow at the right branch; (d) mesh detail on the right horizontal branch of the pipeline; and (e) detail of pipe cross-section.

3.3. Boundary Conditions

In the three-dimensional CFD model, boundary conditions are applied to different regions within the computational domain. The domain walls were characterized by a fixed flux pressure condition, representing a specified pressure boundary condition to account for the known pressure conditions at this pipe surface. Additionally, a no-slip condition was implemented for the velocity at the walls, assuming that the fluid velocity at the walls is zero. For the outlet boundary (air-release zone), a total pressure condition was applied to ensure that the outflow conditions adequately accounted for the atmospheric pressure ($p_{atm} = 101,325$ Pa). In the same boundary, a pressure inlet-outlet velocity condition was enforced for velocity in the function of an atmospheric pressure condition. Moreover, the inlet boundary is defined with a total pressure determined by the absolute pressure ($p_{atm} + p_0$), reflecting the pressure of water inflow. On the other hand, a pressure inlet-outlet velocity condition for velocity dependent on the inlet pressure and the phase fraction condition at the inlet is defined by $\alpha_w = 1.0$, which indicates a water inflow. The temperature conditions were set with a zero gradient at the walls and inlet boundaries, maintaining a constant temperature gradient. In contrast, an inlet-outlet ($I - O$) condition was imposed at the outlet to facilitate the flow of thermal energy.

3.4. Computational Process

The computational process of the CFD model is conducted using the *compressibleInterFoam* solver within the OpenFOAM v2102. The simulation is initiated from the initial time $t_0 = 0$ until $t = 2.0$ s. To ensure temporal accuracy, a minimal time step of 1×10^{-10} s was adopted, while a maximum time step of 5×10^{-4} s was enforced to balance computational efficiency with solution stability. From the maximum and minimum time step values, the software automatically adjusts the time step according to the physical process occurring in the modeling, which depends on the maximum velocity presented in the model and is also controlled by a maximum Courant number of 0.9. During model processing, it was observed that the solution to the variables reached the numerical solution in 1 or 2 iterations.

4. Analysis of Results and Discussion

This section shows different methods of 3D CFD model validation. Air pocket pressure patterns provided reliable information, which is contained in the big data platform. These pulses are compared using numerical data from the 3D CFD model. In addition, the

dynamic behavior of the air-water interface obtained from the CFD model contours is compared with the movement captured on video frames during the laboratory procedure.

4.1. Numerical Validation of CFD Model

Six tests are performed in the laboratory with the initial conditions defined in Table 1. The pressure patterns obtained by the 3D CFD model are compared with the measurements obtained from the big data platform taken in the laboratory, as shown in Figure 4.

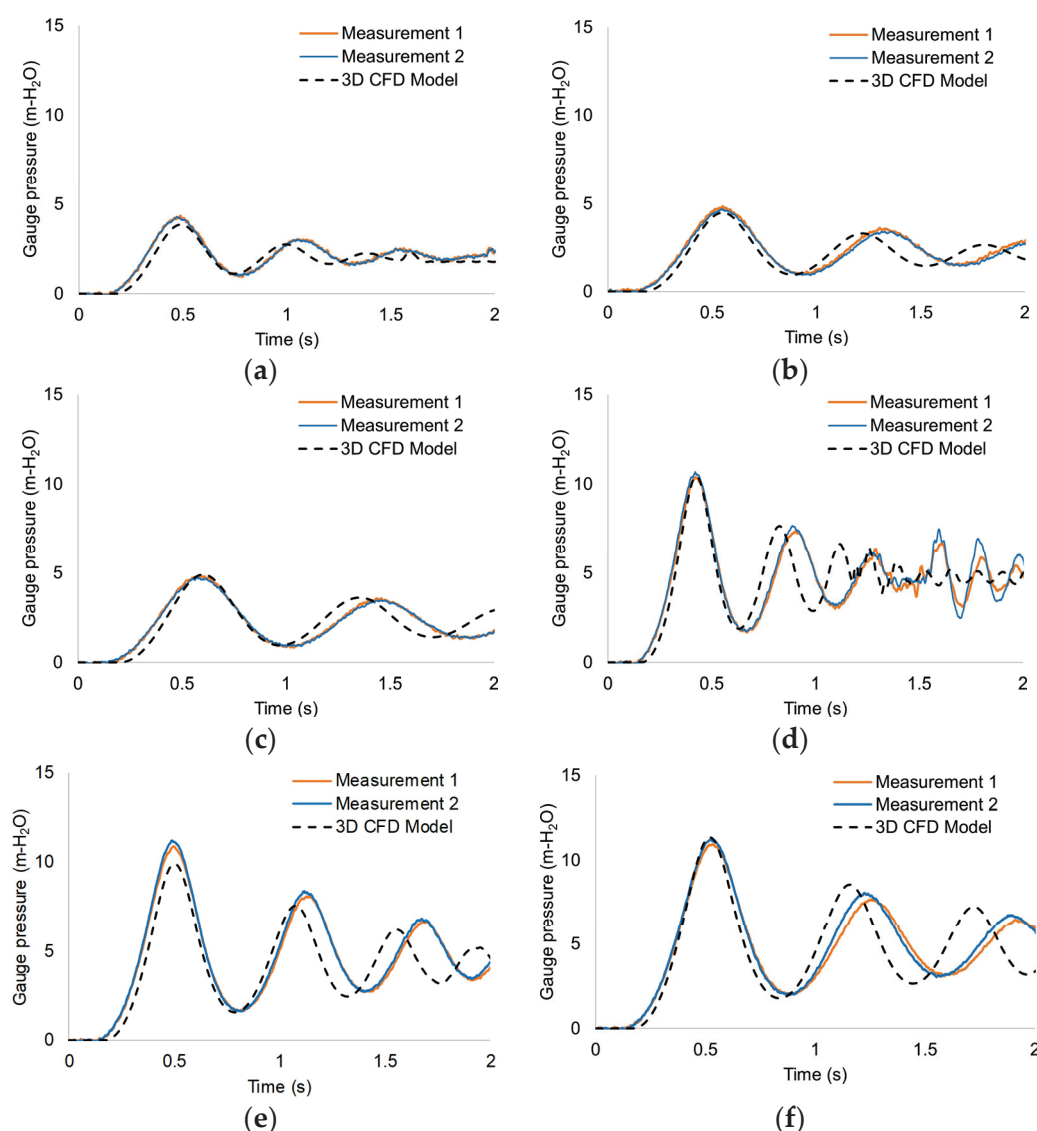


Figure 4. Comparison of the air pocket pressure of the 3D CFD model with experimental measurements: (a) Test 1; (b) Test 2; (c) Test 3; (d) Test 4; (e) Test 5; and (f) Test 6.

The CFD model oscillations generally present a similar trend to the pressure measurements regarding the number of peaks, adequately predicting the first pressure peak in all tests, which is usually the most critical peak in transient filling processes with trapped air.

Tests 1, 2, and 3 are performed at an inlet gauge pressure of 20 kPa. Thus, the maximum gauge pressure peaks ranged between 4.2 and 4.8 m-H₂O, while Tests 4, 5, and 6 were performed at a pressure of 50 kPa, thus generating gauge pressure peaks between 10.5 and 10.8 m-H₂O. In Tests 1 and 4, where the initial air pocket was 0.46 m long, it is possible to observe oscillations given by origin different from the compression and expansion processes of the air pocket. This is because the water column reaches the upper end of the pipe, thus generating pressure oscillations given as a product of the collision of the

water column at the upper end of the network before $t = 2.00$ s. This phenomenon is not shown in Tests 2, 3, 5, and 6 because the filling processes in these cases exceed the 2 s in the laboratory measurements.

In order to validate the accuracy of this model, the maximum pressure values are compared with those measured in the tests from the calculation of the relative error, using the formula $\epsilon = \left| \frac{p_{max,exp} - p_{max,CFD}}{p_{max,exp}} \right|$, where $p_{max,exp}$ = maximum pressure in an experimental test, and $p_{max,CFD}$ = maximum pressure in a CFD scenario. Table 2 shows the values of ϵ for each test, highlighting a good agreement of peak values of air pocket pressure of experimental and numerical conditions due to the values being less or equal to 10%, making this limit value appropriate for numerical resolution in CFD. Errors between CFD and experimental values may be related to the accuracy in capturing complex transient phenomena such as air-water interaction in these filling processes. In addition, minor variations in the initial experimental conditions may influence the observed differences.

Table 2. Root Mean Square Error for maximum pressure.

Test No.	$p_{max,exp}$ (m)	$p_{max,CFD}$ (m)	ϵ (%)
1	4.29	3.84	10.0%
2	4.67	4.48	4.1%
3	4.71	4.78	2.4%
4	10.53	10.42	1.0%
5	11.05	9.90	10.0%
6	11.34	11.10	2.2%

4.2. Validation of Air-Water Interaction

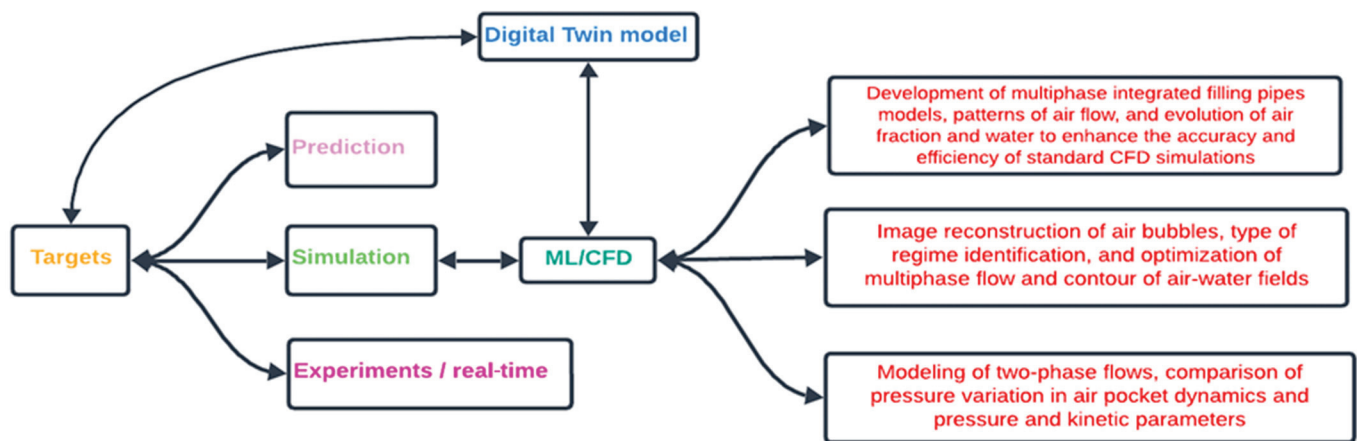
During the laboratory tests, a video was recorded of the filling process of Test 5, and images were processed using Machine Learning (ML) (Figure 5), which is used to validate the 3D CFD model. The frames are a recorded sample of observed data obtained from the six tests performed in the laboratory. ML and data science drive a transformative shift in hydraulic engineering, offering significant advantages in understanding and predicting complex hydraulic phenomena during the filling process. ML is a powerful tool that complements incomplete domain-specific knowledge typically used in conventional experimental and computational methods. ML provides flexible techniques for developing robust predictive models, eliminating values outside the pattern operating zone by identifying hidden patterns and mechanisms in data sets. This capability is particularly advantageous in hydraulic transients and filling processes, where traditional methods may fall short.

For instance, ML can enhance multiphase flow modeling by improving closure models for air expulsion, turbulence stresses, pulsation events, and backflow occurrence, thereby increasing the accuracy and efficiency of CFD simulations. Additionally, ML can assist in image reconstruction, regime identification, parameter prediction, and optimizing flow and air-dragging fields. Despite its many strengths, ML also presents challenges, such as the need for high-quality data and the risk of overfitting. Still, its application is to accelerate advancements in hydraulic systems, providing new opportunities for research and innovation in this field. Figure 6 represents five time instances of the filling process associated with Test 5. Frames from the post-processing of the 3D CFD model and video frames are used in this analysis.

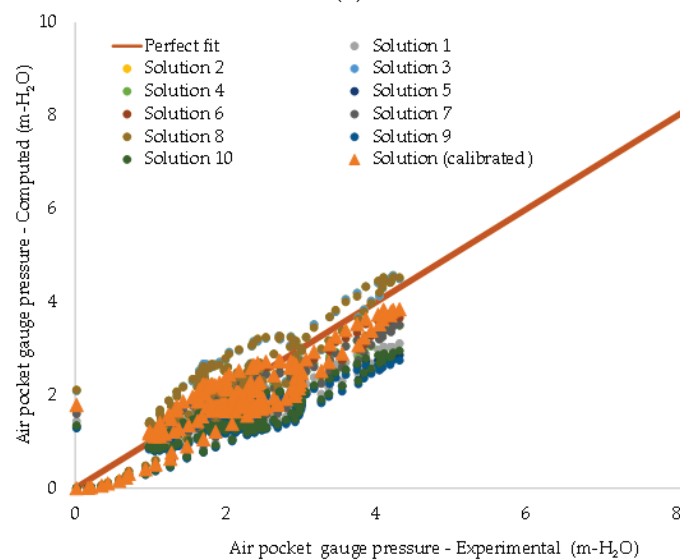
These CFD and experiment images are compared, and the sequence of events occurring in the filling process is analyzed, which is also described below:

- $t = 0.00$ s: At that instant, the pipe contains an initial air pocket at atmospheric conditions, and the water column is at rest since the filling process has not yet started.
- $t = 0.25$ s: The water column at the left branch gradually rises due to the hydro-pneumatic tank pumping, and the air-water interface starts to exhibit a different shape. In addition, the location of the air-water interface in the frame of the CFD model and the experimental one is similar.

- $t = 0.50$ s: The water column returns slightly in the opposite direction of flow, and the air-water interface collapses, generating a transition from pressurized flow to free surface flow, which occurs in the CFD model and the experiment facility.
- $t = 1.54$ s: The water column has reached the upper end of the pipe, thus displacing part of the air volume that the air valve S050 has expelled. At this stage, a moderate mixing of air bubbles with the water phase begins to be generated, evident in the CFD model frame and the experiment's video frame.
- $t = 2.00$ s: Much of the pipe's air volume is expelled at this experiment stage. In that sense, the pipe is almost entirely occupied by water. Small bubbles are observed due to the mixing of air and water at the high point of the pipe and are evident experimentally and numerically through the contours of the CFD model.



(a)



(b)

Figure 5. Validation of results: (a) DT model with categorization used for ML/CFD improvement, (b) analysis of results for Test 1.

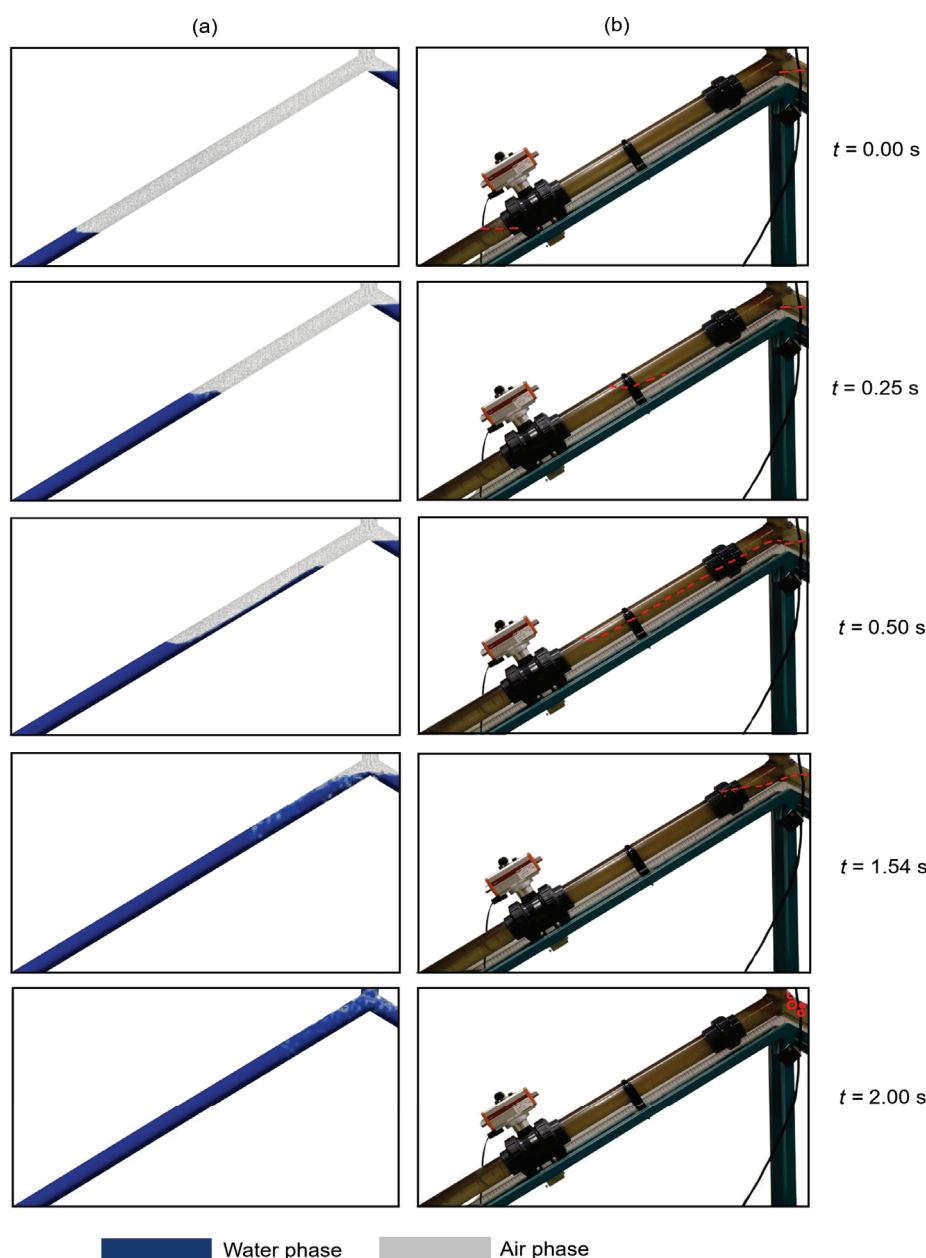


Figure 6. Dynamic of rapid filling in Test 5: (a) contours of air-water interaction from 3D CFD model, and (b) video frames from the observed experiment.

4.3. Dynamic of Air Expulsion from Pipelines

This section emphasizes analyzing the behavior of the air expelled through the air valve S050 from the volume of water inside the pipe. Figure 7 shows the evolution of the volume of water inside the pipe during the filling process, which is related to the amount of air expelled from the hydraulic system.

In Test 2 (Figure 7a), the initial air pocket is $X_{0,l} = 0.96$ m. i.e., 83.7% of the pipe volume is occupied by water. When the filling process starts, the water occupies 84.6% of the total volume of the pipe at $t = 0.25$ s. After that, the water volume fraction of the pipeline increases to 86.7% at $t = 0.50$ s. Subsequently, the pipe's water percentage amounts to 92.4% and 95.4% at $t = 1.50$ s and $t = 2.00$ s, respectively. At $t = 2.25$ s, the water filling process is still occurring at 96.9% of the water over the volume of the pipe, and finally, at $t = 2.50$ s, the water reaches 98.3% of the volume of the pipe. At this moment, the water column has reached the upper zone of the pipe, thus completing the filling process. In general, the amount of water in the pipeline gradually increased due to the controlled pressure

applied to the water column and the controlled operation of the air valve S050 to expel the trapped air.

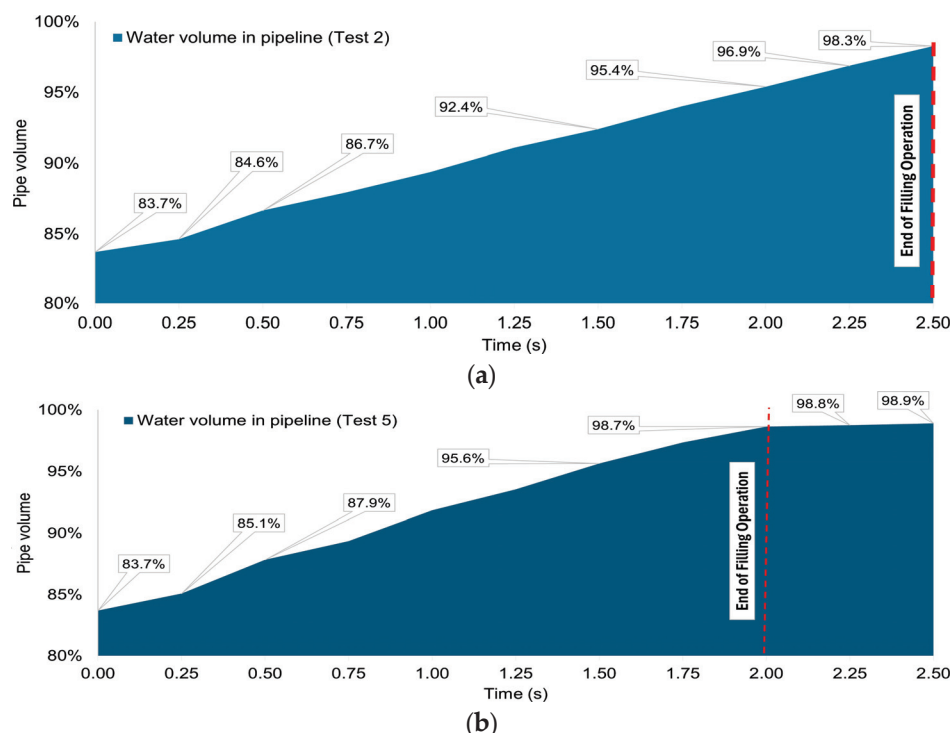


Figure 7. Evolution of the fraction of water available in the pipeline during the filling process: (a) Test 2; and (b) Test 5.

In the case of Test 5 (Figure 7b), the initial air pocket is also 0.96 m. At $t = 0.25$ s, the water column occupies 85.1% of the total volume of the pipe, which is noticeable when the water column rises without generating a collapse of the air–water interface (see Figure 5). At $t = 0.50$ s, the water column still influences the air outflow out of the pipe, although this undergoes a backflow (see Figure 6). In this instant, the water occupies 87.9% of the volumetric capacity of the pipe. At $t = 1.50$ s, the water column has expelled a significant percentage of the trapped air. At this instant, the volume of water available is 95.6%. At the same time, at $t = 2.00$ s, the pipe is almost entirely occupied by water, with a fraction of the total pipe volume equal to 98.7%, leaving 1.3% of air accumulated at the upper end of the pipe. This amount of air contained in the pipe at $t = 2.00$ s is demonstrated in experience, where it is stated that pressurized water flow in pipes occasionally has a certain amount of air equivalent to 2.7% on average, according to Lauchlan et al. (2005) [7]. At this instant, the filling process is completed. However, there is a slight oscillation of water volume between $t = 2.00$ s and $t = 2.50$ s, going from 98.7% to 98.9%, which is not a significant difference.

Figure 8 shows the evolution of air outflow over time through the S050 air valve from the 3D CFD model during the filling process in Tests 2 and 5. Figure 8b shows the air outflow behavior in Test 2. When the filling process starts due to the hydro-pneumatic tank pumping, the air is expelled, generating a first flow peak with the value of $4.47 \text{ m}^3/\text{h}$ at $t = 0.38$ s. Subsequently, the air outflow rate decreases gradually to $2.10 \text{ m}^3/\text{h}$ at $t = 0.72$ s. This decrease in air outflow occurs because the trapped air pocket accumulates compression energy, which is subsequently released and generates a backflow of water in the pipeline, thus leading to a transient dissipation of the airflow. Afterward, a new increase in the air outflow is generated, with a peak value of $3.85 \text{ m}^3/\text{h}$ at $t = 1.06$ s, and subsequently, a second valley value is generated at $t = 1.33$ s with a flow of $2.59 \text{ m}^3/\text{h}$. This oscillatory behavior of the air outflow is maintained over time, dissipating the amplitude of the oscillations, being that the air outflow converges to a rate of $2.92 \text{ m}^3/\text{h}$.

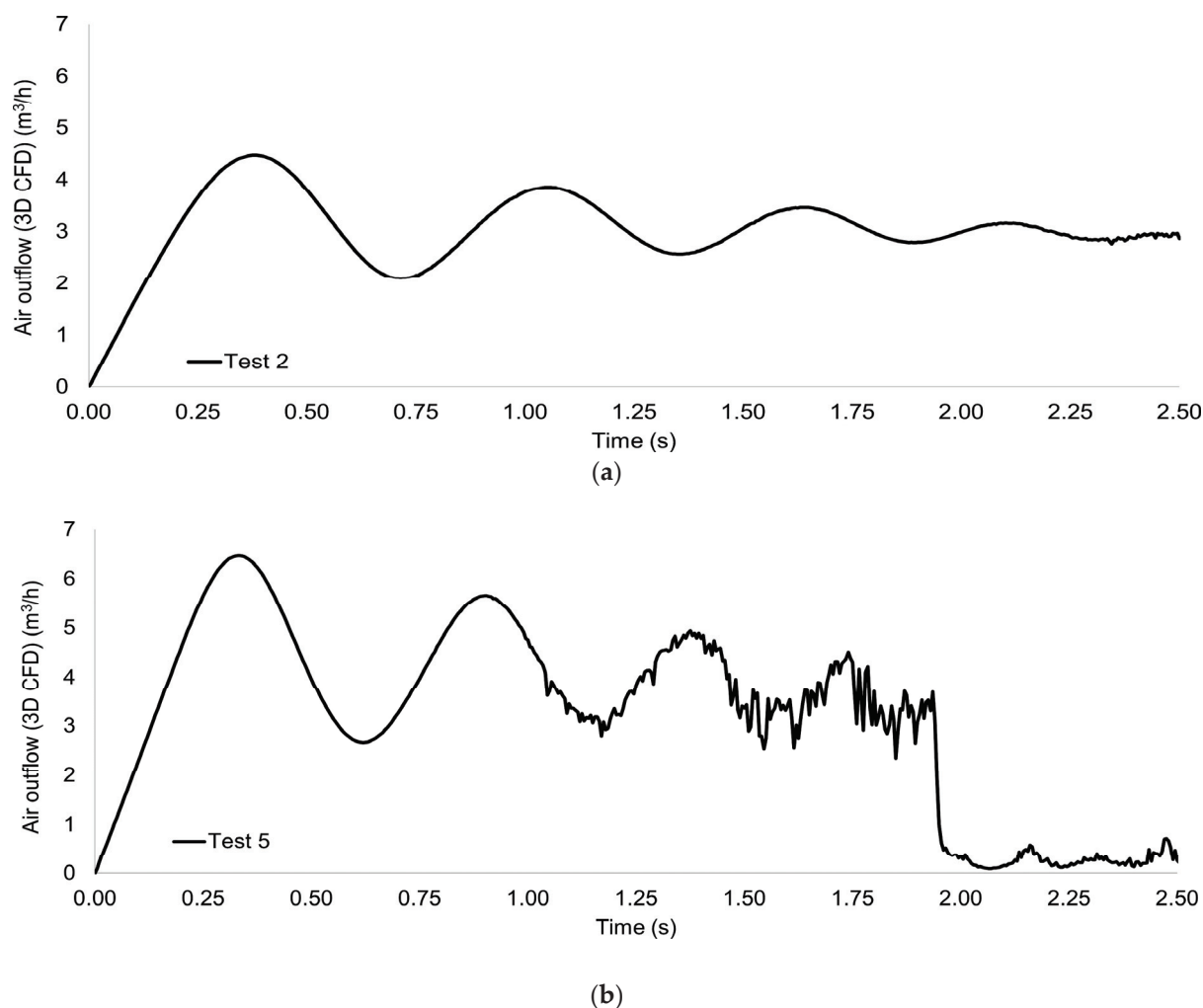


Figure 8. Numerical pattern of air outflow through air valve in 3D CFD model: (a) Test 2; and (b) Test 5.

Figure 8b shows the air outflow behavior. After the hydro-pneumatic tank is actuated to supply water to the pipeline, the airflow rate increases to $6.24 \text{ m}^3/\text{h}$ at $t = 0.29 \text{ s}$. After that, the air outflow rate decreases gradually to $2.66 \text{ m}^3/\text{h}$ at $t = 0.62 \text{ s}$. A second peak is generated with a value of $5.65 \text{ m}^3/\text{h}$ at $t = 0.9 \text{ s}$, and subsequently, a second valley value is generated at $t = 1.17 \text{ s}$ with a flow of $2.79 \text{ m}^3/\text{h}$. In these time instants mentioned, the flow oscillations are stable; this is because the air-water interface had low susceptibility to violent mixing processes between both fluids; this is evidenced by the time instants of $t = 0.25 \text{ s}$ and $t = 0.50 \text{ s}$ from Figure 5. On the other hand, after the time instant $t = 1.25 \text{ s}$, the airflow pattern presents secondary oscillations in the central air discharge flow oscillation pattern until an instant of time $t = 1.94 \text{ s}$. These secondary oscillations occur due to the mixing of air and water in the upper area of the hydraulic system (air valve area S050), which is visible at $t = 1.54 \text{ s}$ and $t = 2.00 \text{ s}$ from Figure 5. After the instant $t = 2.00 \text{ s}$ (when the filling process ends), a minimum discharge flow with an average value of $0.27 \text{ m}^3/\text{h}$ is present.

In post-processing tools, CFD models show streamlines resulting from the movement of fluids, in this case, air, which is a highly compressible fluid. Knowledge of the trajectory of the streamlines over the hydraulic system helps to identify whether the geometrical conditions of the air valve's exhaust orifice influence the efficiency of air exhaust processes.

Figure 9 shows the evolution of the trajectory of the streamlines of the air expelled in Test 5 at five (5) time instants, which are analyzed below:

- $t = 0.00$ s: There are no air streamlines at this instant due to the hydraulic system being at rest.
- $t = 0.25$ s: At this instant, air streamlines parallel the pipe axis with a velocity between 0 and 5 m/s. These are located between the initial air-water interface, up to the upper end of the pipe, where there is a vorticity at the corner. On the other hand, the air streamlines increase in velocity at the end of the air valve to values between 10 and 15 m/s. At the discharge orifice, the velocity increases to a value higher than 20 m/s, according to the observed contours.
- $t = 0.50$ s: The air streamlines in the air valve zone decrease in velocity magnitude due to the number of streamlines, while vortices occur in the air pocket as a result of the transient event, in which the air pocket releases energy after compressing in the first oscillation. Velocity contours are the same at time instant $t = 0.25$ s.
- $t = 1.50$ s: At this instant, the air streamlines are chaotically distributed in the upper zone of the pipe without being traced in the air valve structure. At this time, the airflow velocity is less than 5 m/s.
- $t = 2.00$ s: The filling process ends, so it is not easy to visualize the tracing of the air streamlines.

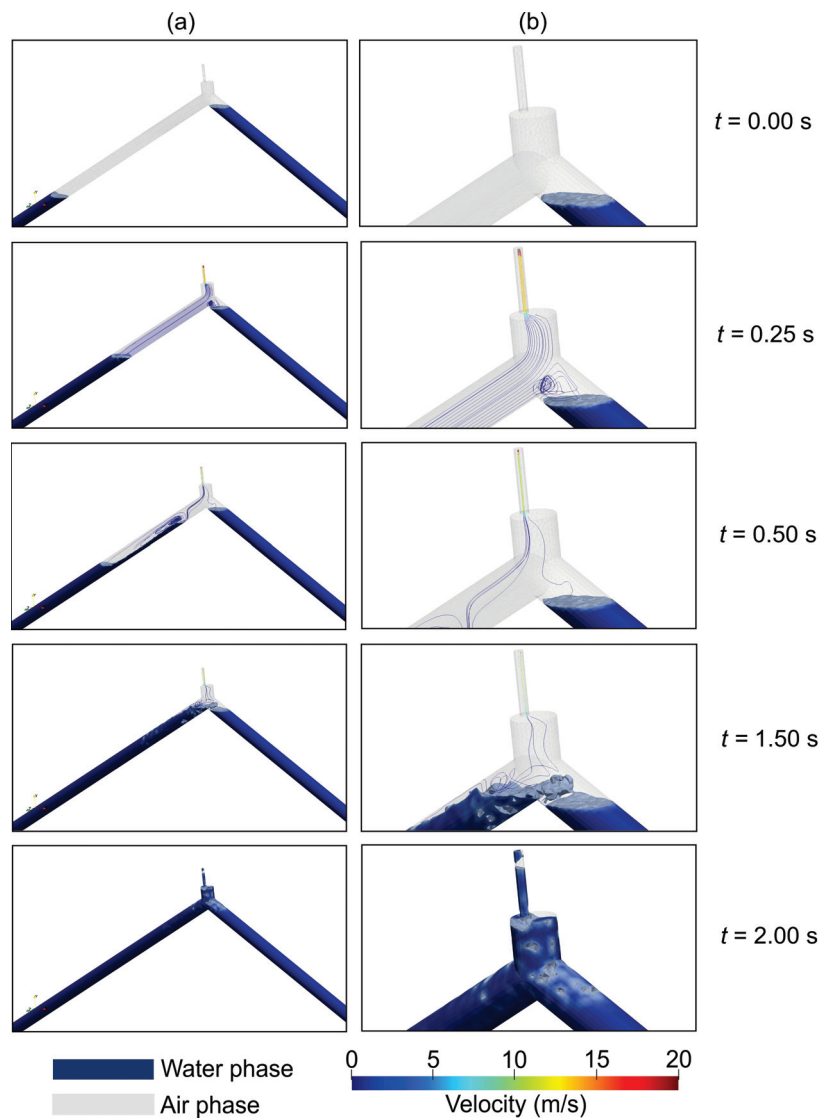


Figure 9. Air streamlines during the expulsion of trapped air for Test 2: (a) overview; (b) close-up at the highest point.

To bear in mind, the proposed Digital Twin can be applied to real-world pipeline systems by addressing several key aspects: (i) 3D CFD simulations, which can take months depending on available computational power, invariably require high-performance resources; (ii) real-time data acquisition using pressure transducers distributed along the pipelines is essential to validate the Digital Twin's results; and (iii) water utilities should utilize cloud computing to meet the demands associated with modeling large-scale hydraulic systems.

This research can be utilized to understand ruptures in pressurized hydraulic installations during filling operations in complex systems containing multiple air pockets. Water utilities often replace pipe materials; however, a deeper understanding could help reduce maintenance costs by enabling the appropriate operation of regulating valves and establishing an air valve maintenance plan. Additionally, it can aid in minimizing water supply interruptions for repairs, particularly in systems prone to frequent ruptures.

5. Conclusions

This study applied real-time data and CFD modeling to examine air-release dynamics during irregular pipe-filling processes. By integrating a Digital Twin platform, 3D CFD simulations, and Artificial Intelligence (AI) technologies, the research identified critical aspects of trapped air management, such as the air pocket extension, air outflow patterns through valves, the evolution of filling, and the aerodynamic behavior of expelled air. These insights significantly improve water conveyance systems' efficiency, reliability, and safety, particularly in complex pipeline networks. The results show that the 3D CFD model, calibrated with real-time experimental data, accurately represents air-release dynamics, offering a reliable tool for analyzing the performance of air valves under various conditions. ML can provide flexible techniques to facilitate the conceptual development of new robust predictive models for multiphase flows: (i) development of multiphase integrated models filling pipes, patterns of air flow, and evolution of air fraction and water to enhance the accuracy and efficiency of standard CFD simulations; (ii) image reconstruction of air bubbles, type of regime identification, and optimization of multiphase flow and contour of air-water fields; and (iii) modeling of two-phase flows, comparison of pressure variation in air pocket dynamics and pressure and kinetic parameters.

Key findings regarding the integration of Digital Twins platforms and CFD modeling are highlighted in this context.

- This study contributes to developing an integrated system for real-time monitoring of pipeline operations. It includes the three-dimensional representation from computational fluid dynamics tools that allows the identification of the air-water interaction from this study scenario and the detailed evolution of fluid parameters such as pressure, velocity–velocity, and temperature spatially and temporally.
- This study represents a breakthrough in the simulation of the filling process with entrapped air and air valves in three dimensions, going beyond 2D CFD simulation that relied on discharge orifices to approximate air ejection effects.
- The integration of CFD models and Digital Twins has several limitations: data sources, interpretability of results, robustness of the integrated model, phenomenon complexity, machine learning efficiency, and extrapolation of results. However, this research also faced several challenges and opportunities: analysis of results, discussion, and future directions.

Therefore, future research should focus on several areas, including studying trapped air management in different branches of water distribution networks and the interaction between hydraulic systems and the mechanical features of air valves, such as the valve body, float, and closing mechanisms. Integrating Geographic Information Systems (GIS), Digital Twin technology, and AI in detecting anomalies, like abnormal pressure patterns, is more used to enhance system safety. Additionally, improving real-time data acquisition through smart sensors and further refining mathematical models will help address current

limitations in field applications, providing more reliable and efficient management of transient events in water conveyance systems.

Author Contributions: Conceptualization, D.A.P.-V. and O.E.C.-H.; methodology, D.A.P.-V., O.E.C.-H., H.M.R. and M.P.-S.; formal analysis, D.A.P.-V., O.E.C.-H. and H.M.R.; writing—original draft preparation, D.A.P.-V. and O.E.C.-H.; supervision, M.P.-S. and H.M.R. All authors have read and agreed to the published version of the manuscript.

Funding: The authors thank MinCiencias (Colombia) for financing the Ph.D. studies of the principal author of this research at Universidad Tecnológica de Bolívar through the Science, Technology, and Innovation Fund (STel). In addition, this work was supported by project HY4RES (Hybrid Solutions for Renewable Energy Systems) EAPA0001/2022 from the ERDF Interreg Atlantic Area Pro-Programme 2021–2027. The research was carried out during Modesto Pérez-Sánchez’s stay at the CERIS-IST research center, called “Incorporation of New Water Resources in Irrigation Systems through the Use of Sustainable Technologies and Computational Tools to Mitigate Water Scarcity”.

Data Availability Statement: Data are contained within the article.

Conflicts of Interest: The authors declare no conflicts of interest.

Nomenclature

The following symbols are used in this paper:

a_{eff} :	thermal conductivity (W/m/°K);
C_s :	roughness constant (-);
$CD_{k\omega}$:	closure coefficient (-);
e :	specific internal energy (J/kg);
D :	pipe diameter (m);
D_k :	diffusion term for k (m ² /s);
D_ω :	diffusion term for ω (m ² /s);
F_1 :	blending function (-);
F_s :	surface tension force (N/m ²);
f_n :	roughness function parameter (-);
$\mathbf{g}$:	gravitational acceleration vector (m/s ²);
G :	turbulent kinetic energy generation (m ² /s ³);
k :	turbulence kinetic energy (m ² /s ²);
K_s :	pipe absolute roughness (m);
K_s^+ :	sand-grain roughness height (-);
L_l and L_r :	length of left and right pipe branches (m);
L_0 :	initial air pocket length (m);
S_T :	energy source term (Nm);
p_0 :	hydro-pneumatic gauge pressure (Pa);
p_{atm} :	atmospheric pressure (Pa);
q :	surface heat flux (W/m ²);
p :	air pocket pressure (N/m ²);
T :	temperature (°K);
R :	universal gas constant (J/°K/mol);
$\mathbf{u}$:	velocity vector (m/s);
u_r :	velocity field associated with the air phase (m/s);
u^* :	shear velocity (m/s);
ν_t :	turbulent kinematic viscosity (m ² /s);
ν_{wall} :	kinematic viscosity of fluid in a near-wall cell (m ² /s);
ω :	specific turbulence dissipation rate (1/s);
t :	time (s);
α :	volume fraction (-);
ρ :	mixing density (kg/m ³);
μ :	dynamic viscosity (Ns/m ²);
X_0 :	initial air pocket size (m);
$\gamma, \beta, \beta^*, S_k$ and S_ω :	constants of the turbulence model (-).

Subscripts

w :	refers to the water phase
a :	refers to the air phase

Appendix A. Governing Equations of the 3D CFD Model

The calculation of mixing density (ρ) and dynamic viscosity (μ) is based on values of water and air density (ρ_w and ρ_a , respectively) and dynamic viscosity of these fluids (μ_w and μ_a), and calculated using a water phase fraction (α_w), which ranges between 0 and 1, where $\alpha_w = 0$ means that cell of fluid is occupied by air, and $\alpha_w = 1$ means that cell of fluid is occupied by water. Equations (A1) and (A2) show the formulations used to calculate mixing density and dynamic viscosity:

$$\rho = \alpha_w \rho_w + (1 - \alpha_w) \rho_a \quad (\text{A1})$$

$$\mu = \alpha_w \mu_w + (1 - \alpha_w) \mu_a \quad (\text{A2})$$

A molecular transport equation is used to consider the dynamic mixing processes of water and air in cells of the CFD model, which contains conservation, advection, and diffusion terms. Equation (A3) shows the transport equation for α_w :

$$\frac{\partial \alpha_w}{\partial t} + \nabla \cdot (\alpha_w \mathbf{u}) + \nabla \cdot [(1 - \alpha_w) \alpha_w \mathbf{u}_r] \quad (\text{A3})$$

where $\mathbf{u}$ = velocity vector, and $\mathbf{u}_r$ = velocity field associated with the complementary phase (air phase). On the other hand, two equations represent the fluid dynamics concerning mass conservation and the acceleration of fluids in cells of the CFD model. Equations (A4) and (A5) represent fluids' continuity and momentum equations:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \quad (\text{A4})$$

$$\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla \cdot (\mu \nabla \mathbf{u}) + \rho \mathbf{g} - F_s \quad (\text{A5})$$

Two-phase transient flows generate heat transfer for entrapped air compression during rapid filling events. This simulation requires thermophysical equations to represent thermodynamic behaviors during transient phenomena during air-water interaction. The conservation of energy equation considers the interaction of heat, mechanics, and pressure energy, as shown in Equation (A6):

$$\frac{\partial (\rho e)}{\partial t} + \nabla \cdot (\rho e \mathbf{u}) = \nabla \cdot \mathbf{q} + p(\nabla \cdot \mathbf{u}) + S_T \quad (\text{A6})$$

where e = specific internal energy, $\mathbf{q}$ = heat flux vector, and S_T = energy source for energy balance. Heat flux is calculated through multiplication between a thermal diffusivity (a_{eff}) and a temperature gradient (∇T), as shown in Equation (A7):

$$\mathbf{q} = -a_{eff} \nabla T \quad (\text{A7})$$

The ideal gas law equation is applied to simulate the thermodynamic behavior of the air phase. This fluid mainly comprises gases such as oxygen (O_2) and nitrogen (N_2). These molecules are diatomic, non-polar, and relatively small in size. These factors contribute to the behavior of ideal gases under many common conditions. Based on the ideal gas equation, the density of the air phase is calculated from the pressure of the trapped

air (p_a), the air temperature (T_a), and the universal ideal gas constant (R), as shown in Equation (A8):

$$\rho_a = \frac{p_a}{RT_a} \quad (\text{A8})$$

The two-equations $k - \omega$ turbulence model with shear stress transport (SST) was used to predict the behavior of turbulence effects in different regions of pipe cross-section such as near-wall and far-wall regions. SST $k - \omega$ model unifies the physical conditions of turbulence models as ϵ , which several authors have used due to the optimization of grid resolution [26,27], and the standard $k - \omega$ physical conditions. The fundamental equations of the turbulence model are presented in Equations (A9) and (A10):

$$\frac{D(\rho k)}{Dt} = S_k - \beta^* \rho k \omega + \nabla \cdot (\rho D_k \nabla k) - \frac{2}{3} \rho k (\nabla \cdot \mathbf{u}) + \rho G \quad (\text{A9})$$

$$\frac{D(\rho \omega)}{Dt} = \nabla \cdot (\rho D_\omega \nabla \omega) + \frac{\rho \gamma G}{\nu_t} - \frac{2}{3} \rho \gamma \omega (\nabla \cdot \mathbf{u}) - \rho \beta \omega^2 + S_\omega + \rho (1 - F_1) C D_{k\omega} \quad (\text{A10})$$

where F_1 = blending function, k = turbulence kinetic energy, ω = specific turbulence dissipation rate, and ν_t = turbulent kinematic viscosity. G represents the generation of turbulence kinetic energy. D_k and D_ω are diffusion terms for k and ω , respectively, and the term $C D_{k\omega}$ is known as a closure coefficient that depends on k and ω . The terms γ , β , β^* , S_k and S_ω are constants of the turbulence model. This model was complemented with wall functions to describe the flow and the resulting turbulence processes, guaranteeing a suitable near-wall treatment in viscous sub-layer and logarithmic region, as a condition independent of y^+ value. The pipe roughness effects in the 3D CFD model were simulated using a wall function defined by the eddy viscosity and the roughness height of the pipe material as follows:

$$f_n = \left(\frac{K_s^+ - 2.25}{87.75} + C_s K_s^+ \right) \quad (\text{A11})$$

$$K_s^+ = \frac{u^* K_s}{\nu_{wall}} \quad (\text{A12})$$

where f_n = roughness function parameter, K_s^+ = sand-grain roughness height in walls, and C_s = roughness constant (0.50). The term K_s^+ is defined as: where u^* = shear velocity, K_s = roughness height, and ν_{wall} = kinematic viscosity of fluid in a near-wall cell.

References

1. Ramos, H.M.; Fuertes-Miquel, V.S.; Tasca, E.; Coronado-Hernández, O.E.; Besharat, M.; Zhou, L.; Karney, B. Concerning Dynamic Effects in Pipe Systems with Two-Phase Flows: Pressure Surges, Cavitation, and Ventilation. *Water* **2022**, *14*, 2376. [CrossRef]
2. John, H.B., Jr.; Dechant, D.A. Current Proposed Changes to AWWA M11 Steel Pipe: A Guide for Design and Installation. In *Proceedings of the Pipelines 2014: From Underground to the Forefront of Innovation and Sustainability*, Portland, OR, USA, 3–6 August 2014; pp. 2221–2228. [CrossRef]
3. Boulos, P.F.; Karney, B.W.; Wood, D.J.; Lingireddy, S. Hydraulic Transient Guidelines for Protecting Water Distribution Systems. *J. Am. Water Work. Assoc.* **2005**, *97*, 111–124. [CrossRef]
4. Crabtree, A.; Oliphant, K. Resistance of PE4710 Piping to Pressure Surge Events in Force Main Applications. 2012. Available online: <https://conduitalc.plasticpipe.org/pdf/resistance-of-pe4710-piping-to-pressure-surge-evnets-in-force-main-applications.pdf> (accessed on 10 September 2024).
5. Hussam, F.; Tarek, Z. Hierarchical Fuzzy Expert System for Risk of Failure of Water Mains. *J. Pipeline Syst. Eng. Pr.* **2010**, *1*, 53–62. [CrossRef]
6. Ramezani, L.; Karney, B.; Malekpour, A. Encouraging Effective Air Management in Water Pipelines: A Critical Review. *J. Water Resour. Plan. Manag.* **2016**, *142*, 4016055. [CrossRef]
7. Lauchlan, C.; Escarameia, M.; May, R.; Burrows, R.; Gahan, C. Air in Pipelines—A Literature Review. 2005. Available online: <https://eprints.hrwallingford.com/551/1/SR649.pdf> (accessed on 10 September 2024).
8. Zhou, L.; Cao, Y.; Karney, B.; Bergant, A.; Tijsseling, A.S.; Liu, D.; Wang, P. Expulsion of Entrapped Air in a Rapidly Filling Horizontal Pipe. *J. Hydraul. Eng.* **2020**, *146*, 4020047. [CrossRef]

9. Malekpour, A.; Shahroodi, A. What Is a Safe Filling Velocity in Water and Wastewater Pipelines? In Proceedings of the Pipelines 2023, San Antonio, TX, USA, 12–16 August 2023; pp. 301–312. [CrossRef]
10. Tasca, E.S.A.; Karney, B. Improved Air Valve Selection through Better Device Characterization and Modeling. *J. Hydraul. Eng.* **2023**, *149*, 4023019. [CrossRef]
11. Ramezani, L.; Karney, B.; Malekpour, A. The Challenge of Air Valves: A Selective Critical Literature Review. *J. Water Resour. Plan. Manag.* **2015**, *141*, 4015017. [CrossRef]
12. García-Todolí, S.; Iglesias-Rey, P.L.; Mora-Meliá, D.; Martínez-Solano, F.J.; Fuertes-Miquel, V.S. Computational Determination of Air Valves Capacity Using CFD Techniques. *Water* **2018**, *10*, 1433. [CrossRef]
13. Zhou, F.; Hicks, F.E.; Steffler, P.M. Transient Flow in a Rapidly Filling Horizontal Pipe Containing Trapped Air. *J. Hydraul. Eng.* **2002**, *128*, 625–634. [CrossRef]
14. De Martino, G.; Fontana, N.; Giugni, M. Transient Flow Caused by Air Expulsion through an Orifice. *J. Hydraul. Eng.* **2008**, *134*, 1395–1399. [CrossRef]
15. Carlos, M.; Arregui, F.J.; Cabrera, E.; Palau, C.V. Understanding Air Release through Air Valves. *J. Hydraul. Eng.* **2011**, *137*, 461–469. [CrossRef]
16. Fuertes-Miquel, V.S.; López-Jiménez, P.A.; Martínez-Solano, F.J.; López-Patiño, G. Numerical Modelling of Pipelines with Air Pockets and Air Valves. *Can. J. Civ. Eng.* **2016**, *43*, 1052–1061. [CrossRef]
17. Coronado-Hernández, Ó.E.; Besharat, M.; Fuertes-Miquel, V.S.; Ramos, H.M. Effect of a Commercial Air Valve on the Rapid Filling of a Single Pipeline: A Numerical and Experimental Analysis. *Water* **2019**, *11*, 1814. [CrossRef]
18. Zhou, L.; Pan, T.; Wang, H.; Liu, D.; Wang, P. Rapid Air Expulsion through an Orifice in a Vertical Water Pipe. *J. Hydraul. Res.* **2019**, *57*, 307–317. [CrossRef]
19. Romero, G.; Fuertes-Miquel, V.S.; Coronado-Hernández, Ó.E.; Ponz-Carcelén, R.; Biel-Sanchis, F. Analysis of Hydraulic Transients during Pipeline Filling Processes with Air Valves in Large-Scale Installations. *Urban. Water J.* **2020**, *17*, 568–575. [CrossRef]
20. Li, L.; Zhu, D.Z.; Huang, B. Analysis of Pressure Transient Following Rapid Filling of a Vented Horizontal Pipe. *Water* **2018**, *10*, 1698. [CrossRef]
21. Chan, S.N.; Cong, J.; Lee, J.H.W. 3D Numerical Modeling of Geyser Formation by Release of Entrapped Air from Horizontal Pipe into Vertical Shaft. *J. Hydraul. Eng.* **2018**, *144*, 04017071. [CrossRef]
22. Wang, J.; Vasconcelos, J.G. Investigation of Manhole Cover Displacement during Rapid Filling of Stormwater Systems. *J. Hydraul. Eng.* **2020**, *146*, 4020022. [CrossRef]
23. Fang, H.; Zhou, L.; Cao, Y.; Cai, F.; Liu, D. 3D CFD Simulations of Air-Water Interaction in T-Junction Pipes of Urban Stormwater Drainage System. *Urban. Water J.* **2022**, *19*, 74–86. [CrossRef]
24. Aguirre-Mendoza, A.M.; Paternina-Verona, D.A.; Oyuela, S.; Coronado-Hernández, O.E.; Besharat, M.; Fuertes-Miquel, V.S.; Iglesias-Rey, P.L.; Ramos, H.M. Effects of Orifice Sizes for Uncontrolled Filling Processes in Water Pipelines. *Water* **2022**, *14*, 888. [CrossRef]
25. American Water Works Association (AWWA). *Air-Release, Air/Vacuum, and Combination Air Valves: M51*; American Water Works Association (AWWA): Denver, CO, USA, 2001; Volume 51.
26. Paternina-Verona, D.A.; Coronado-Hernández, O.E.; Espinoza-Román, H.G.; Arrieta-Pastrana, A.; Tasca, E.; Fuertes-Miquel, V.S.; Ramos, H.M. Attenuation of Pipeline Filling Over-Pressures through Trapped Air. *Urban. Water J.* **2024**, *21*, 698–710. [CrossRef]
27. Martins, N.M.C.; Delgado, J.N.; Ramos, H.M.; Covas, D.I.C. Maximum Transient Pressures in a Rapidly Filling Pipeline with Entrapped Air Using a CFD Model. *J. Hydraul. Res.* **2017**, *55*, 506–519. [CrossRef]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

Article

Simulation of Flow and Pressure Loss in the Example of the Elbow

Emil Smyk ^{1,*}, Michał Stopel ¹ and Mikołaj Szyca ²

¹ Department of Mechanical Engineering, Bydgoszcz University of Science and Technology, Al. Prof. S. Kaliskiego 7, 85-796 Bydgoszcz, Poland; michal.stopel@pbs.edu.pl

² PESA Bydgoszcz S.A., Zygmunt Augusta 11, 85-082 Bydgoszcz, Poland; mikolaj.szyca@pesa.pl

* Correspondence: emil.smyk@pbs.edu.pl

Abstract: One of the most basic issues in fluid mechanics is the description of flow in closed flows; more precisely, the calculation of pressure drops and the description of the flow form. Therefore, in this paper, the numerical simulation of the flow through the elbow was presented. This case was used to comprehensively describe the most important phenomena that should be taken into account during closed flows. The elbow was chosen as one of the most frequently used fittings in practice. The simulation was made with ANSYS Fluent, with the use of the turbulent model $k-\omega$, SIMPLE simulation method, and at Reynolds number $Re = 500 - 100,000$. The minor and major pressure loss were presented and discussed in the paper. The minor loss coefficient at the high Reynolds number was equal to around 0.2, which is close to the value of 0.22 used in engineering calculations. The influence of the Reynolds number on the shift of the stream separation point in the elbow was described. The secondary flow in the elbow was observed and the vortex structure was discussed and shown with the use of the Q-criterion (Q iso surface for level 0.005). This analysis allowed us to better visualize and describe the complex flow structure observed in the investigated case.

Keywords: minor losses; major losses; head loss; pressure drop; CFD

1. Introduction

The elbows are one of the most frequently used fittings in duct engineering installations. Examples of their application include in gaseous fuel supplies, exhaust installations, ventilation systems, technical gas installations, and many others. The flow through the elbow is also very complex. The widespread use and complexity of the flow make it an excellent facility for operational research and flow analysis, as it is used much more often than other types of fittings. The elbow is very often used as an example fitting for major and minor loss calculation [1,2], mass transfer [3,4], and multi-phase analyses [5–7].

Ono et al. [8] investigated the influence of the elbow curvature on the flow structure for high Reynolds numbers (180,000 and 540,000). They showed that the use of a large bending radius of the elbow significantly affects the flow separation at the outlet—the separation region in the long elbow is formed intermittently. They also described secondary flow in the outflow. A more complex description of the secondary flow was made by Sudo et al. [9]. They observed that secondary flow develops into a pair of vortices at a 30-degree angle (calculated from the inlet to the elbow). The secondary flow was observed even for a distance from the elbow equal to $10D$, where D was the diameter of the channel. An analysis of the secondary flow was also undertaken by Röhrig et al. [10], Kim et al. [11], and others [12–15]. On the other hand, Patiño-Jaramillo et al. [16] analyzed flow through the tee joints for different configurations but only in 2D simulation without secondary flow analysis. Recently, research in the field of the numerical analysis of flow in elbows due to erosion occurring during pipeline operation has also been developing dynamically; see Sun et al. [17], Liu et al. [18], and Chen and Tan [19]. These investigations connect the flow structure with the shape of erosion occurring in pipelines.

The elbow is very often used in the analysis of the loss coefficient. dos Santos et al. [1] investigated the minor loss of elbow at different Reynolds numbers and proposed an equation describing this relationship. Smyk et al. [20] investigated minor and major loss coefficients for straight ducts and elbows for different duct diameters, elbow angles, and Reynolds numbers. They used different simulation models during the investigation. A similar investigation was also undertaken by Salehi et al. [21]. Lui et al. [22] proposed a procedure for predicting pressure loss coefficients based on the example of the tee. However, the simulation model was chosen based on the simulation of losses in the elbow. The interaction between two elbows and their impact on the flow field and losses was investigated by Sami and Ciu [23], Zmrhal and Schwarzer [24], and Weissenbrunner et al. [25]. All these studies were numerical. This is due to the search for a reliable method of determining local loss coefficients for fittings. Cheap methods that do not require time-consuming and expensive experimental research are also sought. All this research indicated the unreasonableness of treating the local loss coefficient as one value (in engineering practice).

The article presents the simulation of the flow in the elbow. The simulation was made for nine different Reynolds numbers in a range from 500 to 100,000 and the results were discussed due to the influence of the Reynolds number. The paper combines issues in the field of local losses and flow characteristics in the elbow, providing a broader context for understanding this basic but complex phenomenon. The paper paid particular attention to the hydraulic entry length that had to be maintained during the simulation, and the secondary flow was described in detail, through the use of a Q iso surface. Additionally, the impact of the Reynolds number on the angle of separation point in the elbow was discussed. For this reason, in the authors' opinion, the work is a particularly extensive study of the described topic, which may constitute a springboard for a broader discussion and further description and the investigation of flows in closed channels.

2. Materials and Methods

The simulation of the flow through the elbow was made in ANSYS Fluent 2019 and the investigated model was presented in Figure 1. The diameter of the duct was equal to $D = 200$ mm and the bending radius of the elbow was equal to $R = 200$ mm. The length of the inlet duct was equal to $200D$ and the outlet duct was $20D$. Steady simulation was used, and gravity was not taken into account. The turbulent model $k-\omega$ and SIMPLE simulation method were used. During all simulations, y^+ values were below 1. The velocity inlet boundary condition was used on the inlet and the pressure outlet one on the outlet. The roughness of the wall was defined as 0 mm. Table 1 presents the parameters of the simulation.

The Reynolds number was calculated as

$$Re = \frac{V_{MEAN}D}{\nu} \quad (1)$$

where V_{MEAN} is the mean velocity, defined in Table 1 [m/s]. The simulation was made for water with density $\rho = 998.2$ kg/m³ and kinematic viscosity $\nu = 1.79 \cdot 10^{-5}$ m²/s. The Dean number was calculated as

$$De = Re \sqrt{\frac{D}{2R}} \quad (2)$$

Table 1. Parameters of simulation.

Number	Re	V_{MEAN} , m/s	De
1	100,000	8.95	70,711
2	80,000	7.16	56,569
3	50,000	4.47	35,355
4	20,000	1.79	14,142
5	10,000	0.89	7071
6	5000	0.45	3536
7	2300	0.21	1626
8	1000	0.09	707
9	500	0.04	354

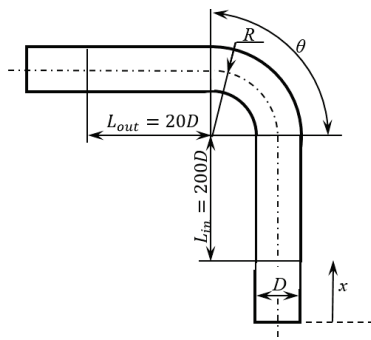
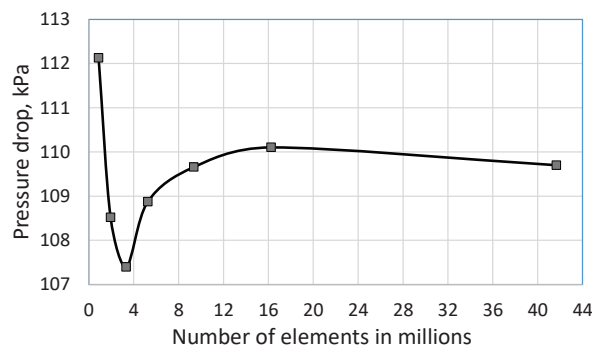


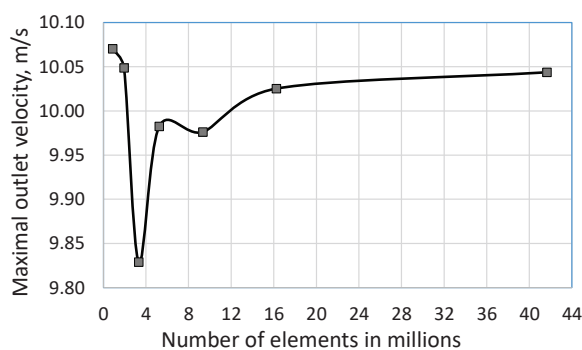
Figure 1. Model of measuring system.

Grid Independence

The first step of the simulation was an analysis of the grid independence. Figure 2 presents the dependence between the pressure drop and the number of elements (Figure 2a) and between the maximal velocity on the outlet and the number of elements (Figure 2b). Based on this dependence, selected mesh parameters corresponding to 5.26 million elements were chosen. It must be noted that the difference between the investigated parameters obtained for this and the best mesh was smaller than 1%.



(a)



(b)

Figure 2. The pressure drop (a) and maximal velocity on the outlet (b) depending on the number of elements at $Re = 100,000$.

3. Results and Discussion

3.1. Hydrodynamic Entry Length and Head Loss

Figure 3 presents the profile of the ratio of centerline velocity (U_C) to mean velocity for different Reynolds numbers. This profile can be used to determine the hydrodynamic entry length [26,27]. Based on the ratio velocity profile presented in Figure 3, the entry length can

be determined as $50D$ in case of $Re = 500$ and as $70D$ in case of $Re = 100,000$, and generally the higher Reynolds number, the longer entry length (the length at which the velocity values in the axis stabilize). Additionally, at $Re = 500$ for $x/D > 140$, fluctuation can be observed in the ratio velocity, which was not observed for other Reynolds numbers. It is caused by the elbow. Therefore, the head pressure loss was calculated from $80 \leq x/D \leq 150$ for all investigated cases.

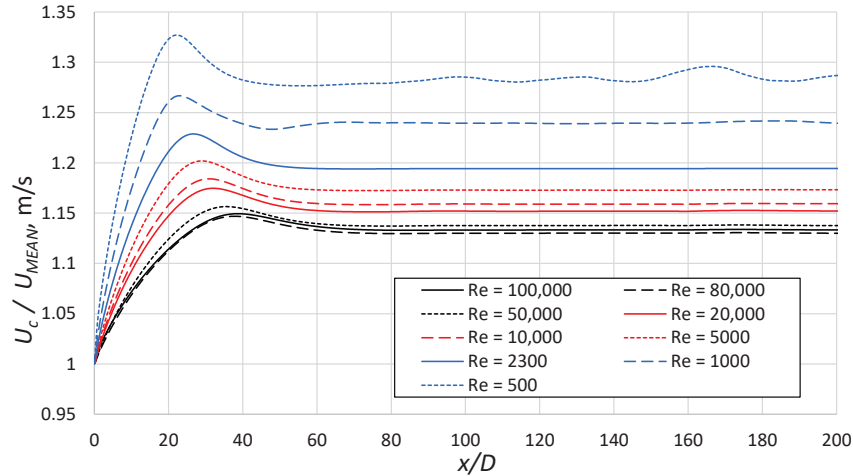


Figure 3. The profile of the ratio of centerline velocity (U_C) to mean velocity (U_{MEAN}) for different Reynolds numbers in the inlet duct.

Figure 3 shows that the ratio of centerline velocity to mean velocity decreased with the Reynolds number. This is a phenomenon that should be expected due to the flattening of the velocity profile with an increase in the Reynolds number [28–30]. The velocity ratio stabilized around the value of $\frac{U_C}{U_{MEAN}} = 1.13$ at $Re > 50,000$.

The numerical major/head loss coefficient (called Darcy friction factor or Darcy–Weisbach friction factor) was calculated from the Darcy–Weisbach equation, as follows:

$$f_D = \frac{2\Delta p D}{\rho V^2 L} \quad (3)$$

where Δp is a static pressure drop in the inlet duct at a distance $80 \leq x/D \leq 150$ [Pa], and L is the measuring length ($L = 14$ m). The theoretical value of the Darcy factor was calculated from Poiseuille’s equation (laminar flow, $Re \leq 2300$) and Blasius’s equation (turbulent flow, $Re > 2300$) [31] successively, as follows:

$$f_D = 64/Re \quad (4)$$

$$f_D = 0.3164 \cdot Re^{-0.25} \quad (5)$$

Figure 4 presents a comparison of the Darcy friction factor calculated based on the simulation for different Reynolds numbers, where the Reynolds number is presented in a logarithmic scale. The numerical and theoretical simulations were similar at a high Reynolds number $Re \geq 2300$. Additionally, Figure 4 presents the numerical Darcy friction factor calculated for the k- ω simulation by Smyk et al. [20]. The course of the curve obtained in [20] is more similar to the theoretical course, but the errors in the laminar range are similar to those in the presented simulation. It is worth noting that data from [20] were meant for eight different diameters, and the presented data are calculated for only one diameter of the duct.

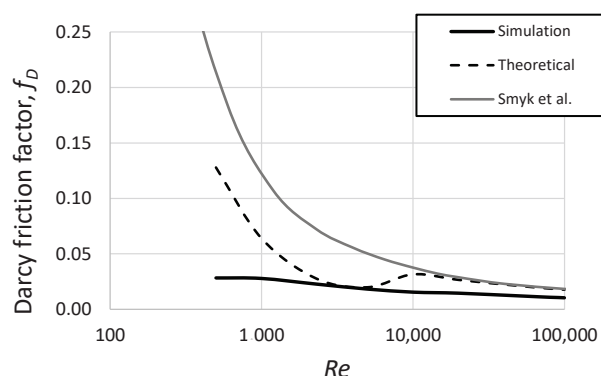


Figure 4. The Darcy friction factor from the simulation, the theoretical one based on Equations (4) and (5), and the one from Smyk et al. [20].

The differences between the obtained and theoretical Darcy friction factors do not seem to be big. However, the smallest percentage difference between them was obtained at $Re = 50,000$ and was equal to 7.5%. The theoretical Darcy friction factor at $Re = 500$ was 12 times bigger than the numerical one. The analysis of the major pressure coefficient is not a goal of this paper and is useful only for calculating the minor/local pressure coefficient. For this reason, the calculated and theoretical coefficients do not need to be equal in value. Smyk et al. [20] showed that the best turbulence model for the calculation of the Darcy friction factor is the Spalart–Allamars turbulence model. It must be noted that Poiseuille’s and Blasius’s equations were experimentally designed. However, their theoretical justification was given by George Stokes. The problem of calculating the Darcy friction factor was discussed in more detail in [32–34].

3.2. Minor Loss Coefficient

To calculate the local loss factor, two steps were performed. These were (1) the calculation of the static pressure drop between three points: the inlet duct at a distance $x/D = 80$ and the outlet of the outlet duct; (2) a reduction in the obtained pressure by major pressure losses calculated with the use of numerical Darcy friction factor (presented in Figure 4), and calculation of the minor loss coefficient based on the following equation:

$$\zeta = \frac{2\Delta p}{\rho v^2} \quad (6)$$

where Δp is a pressure drop calculated in the second step [Pa]. The calculated minor loss coefficient is presented in Figure 5. Additionally, the value $\zeta_{practice} = 0.22$ was marked on the chart. The value was taken from [35]. This book is one of the most commonly used (in Poland) for designing and calculating ventilation systems.

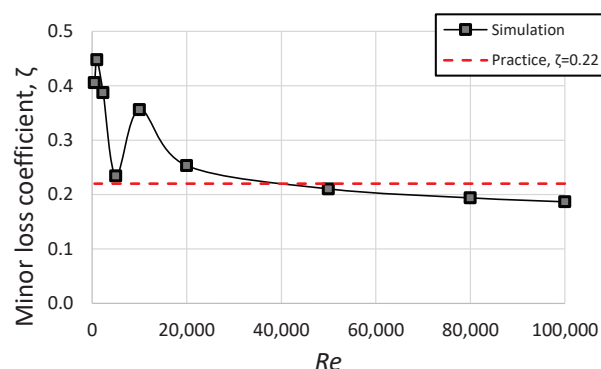


Figure 5. The simulated minor loss coefficient vs. the Reynolds number. The value $\zeta_{practice} = 0.22$ was marked with the red dashed line.

The highest value of the minor loss coefficient was obtained at $Re = 1000$. The local maximum was also observed at $Re = 10,000$. The decrease in the minor loss coefficient for $Re > 10,000$ was also observed in other papers [1,21,36]. The increase in the pressure loss in the area of transition from laminar to turbulent flow was observed in [12].

3.3. Flow Analysis

The elbow and other types of fittings can cause velocity distribution, not only in a plane of the flow (YZ plane in investigated case, see Figure 6) but also in the perpendicular plane (XY plane in the investigated case, see Figure 6), observed as secondary flow [9,11,14,37]. The secondary flow is most often observed with the use of proper orthogonal decomposition (POD) of the velocity field [37,38]. The secondary flow is related to the appearance of a velocity component perpendicular to the flow plane (the velocity component u parallel to the X-axis; see Figures 6 and 7). The line of the u velocity was measured was shown in Figure 7. All lines were parallel to the X-axis.

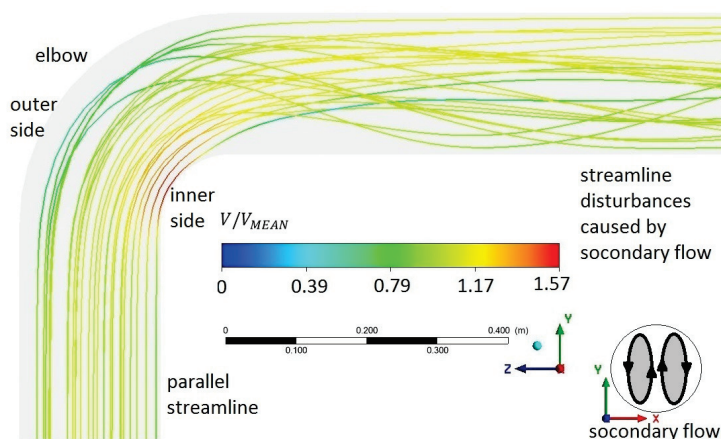


Figure 6. The streamline in the analyzed system at $Re = 100,000$.

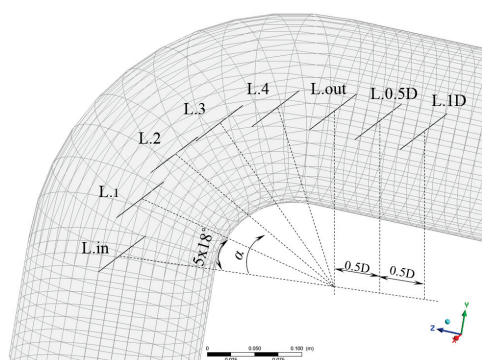


Figure 7. The location of measurement line—the L.in line is located at the beginning of the elbow and the L.out line at the end of the elbow.

The u velocity was also analyzed at distances $0.5D$ and $1D$ before the elbow, but even at $Re = 100,000$, the u velocity was equal to zero at a distance $1D$ before the elbow. At a distance $0.5D$ before the elbow, the u velocity was at least 100 times lower than for the other lines tested. Such low values are negligible; therefore, the analysis of the u component was started from the L.in line located at the beginning of the elbow (Figure 7). The above observations are consistent with experimental studies by Sudo et al. [9].

Figure 8 presents the radial profile of the u velocity on the measurement line presented in Figure 7 at different Reynolds numbers. The profiles were presented as r/D function, where r is a distance from the duct axis. The u velocity was equal to zero in $r/D = 0$ at all investigated Reynolds numbers and for all measurement lines. The higher the Reynolds

number, the higher the value of u velocity. The velocity profile was generally sinusoidal; however, at lines L.2 and L.3 (Figure 8c,d) a distribution of the profile was observed. Particular attention should be paid to the change in the direction of the u velocity vector near the elbow wall at line L.2 (Figure 8c). This phenomenon was observed at a distance $\frac{r}{D} = 0.36 - 0.50$ at $Re = 1000$, and the distance decreased as the Reynolds number increased. It was not observed at $Re = 500$. The different direction of the u velocity indicates the presence of secondary flow in the elbow. It can be seen better in the presentation of the tangential velocity vector. Figure 9 shows the tangential velocity vector in the plane passing through line L.2. The change in the direction of the u velocity vector was not visible in this case. But the shape of the vector clearly indicates the formation of vortex structures in this elbow. These structures, called secondary flow, were also observed at $Re = 500$. According to the classification of the secondary flow patterns by Sudo et al. [9], this type of the vortex structure should be classified as the first type of secondary flow. This structure was observed for all investigated Dean and Reynolds numbers (Table 1).

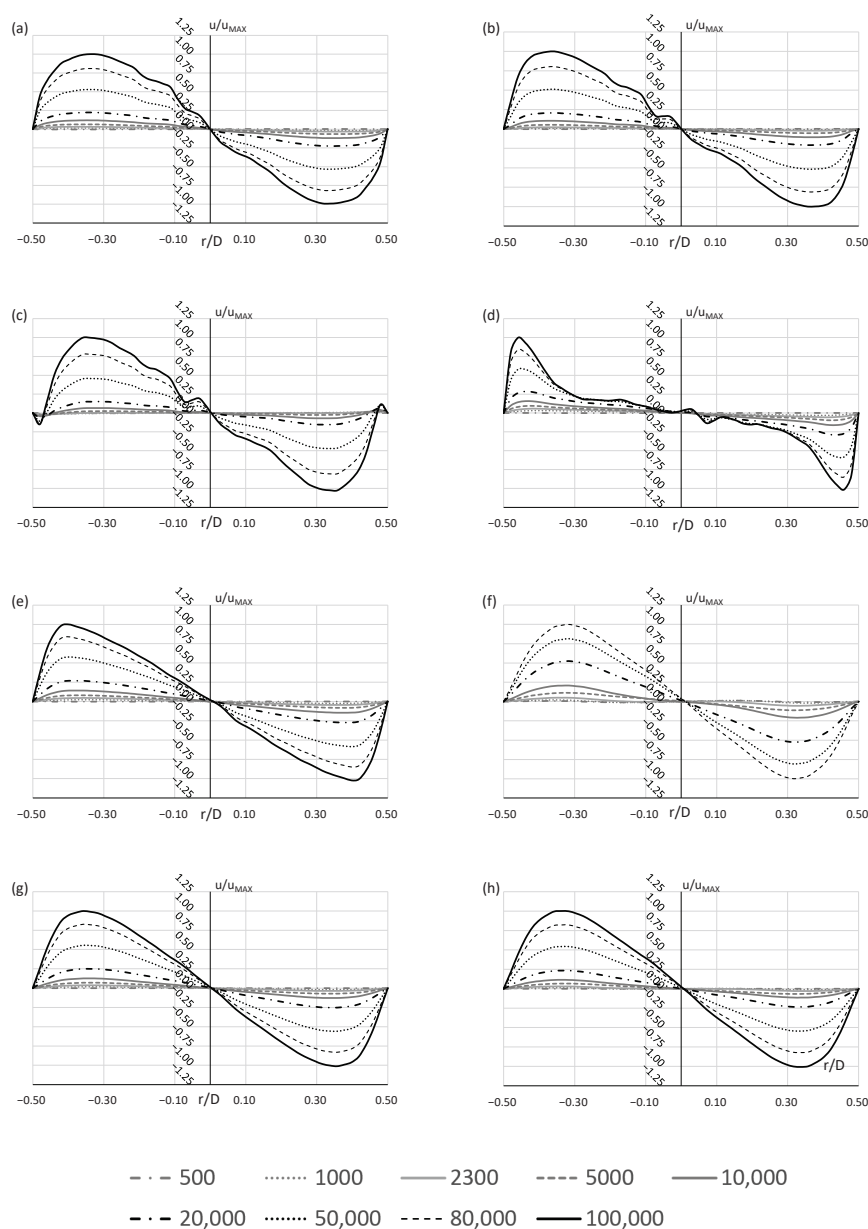


Figure 8. The radial profile of the u velocity on the L.in (a), L.1 (b), L.2 (c), L.3 (d), L.4 (e), L.out (f), L.0.5D (g), and L.1D (h).

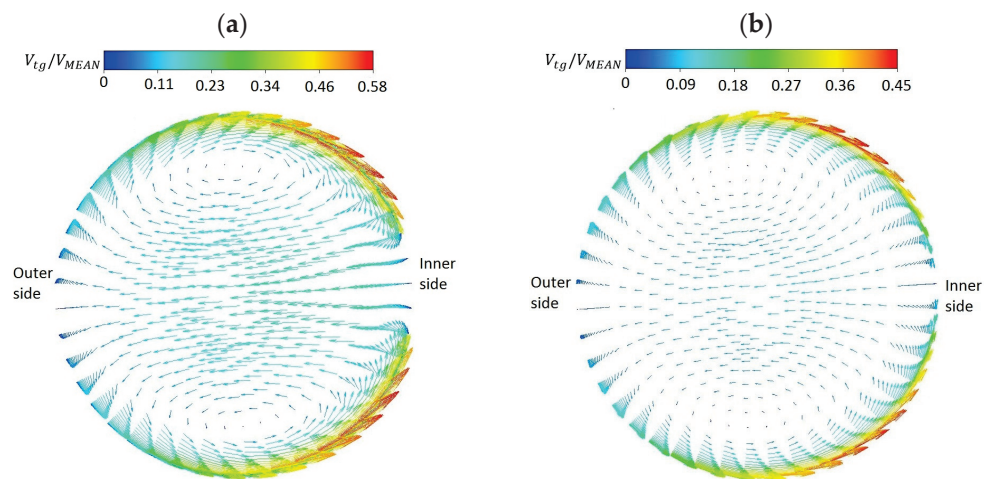


Figure 9. The tangential velocity (V_{tg}) vector in a plane perpendicular to the elbow and passing through line L.2 at $Re = 1000$ (a) and $Re = 100,000$ (b).

Kalpakli et al. [38] observed that the change in the secondary flow (tangential velocity vector) depends on the location of measurements in the elbow and outer pipe. Röhrig et al. [10] carried out the same experiment numerically and expanded Kalpakli et al. [38]’s investigation. The change in the tangential velocity vector is caused mainly by the separation of the flow and the formation of a vortex structure behind the elbow. Figure 10 presents the dependence between the separation angle α (the angle determines the location of the separation point of the stream on the inner wall of the elbow) and the Reynolds number. The separation angle is presented in Figure 7. Generally, the higher the Reynolds number, the further the separation point is moved towards the outlet. This behavior is an effect of the pressure distribution in the duct and the Coanda effect [39,40].

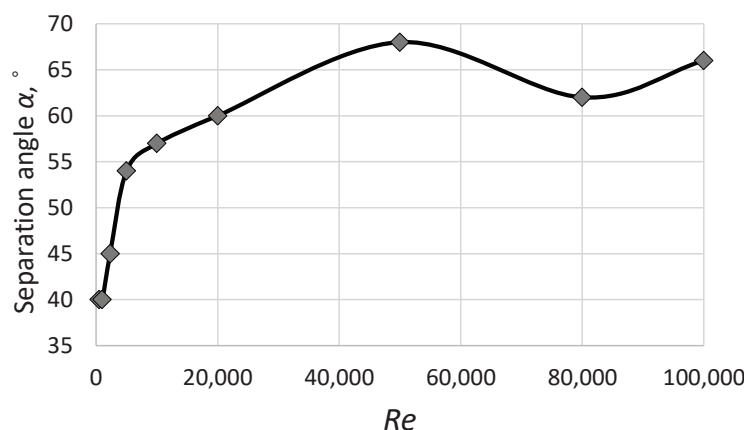
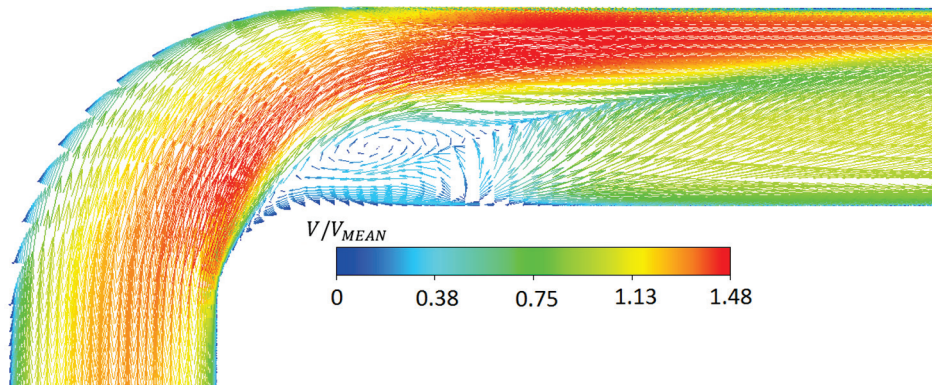


Figure 10. The separation point location vs. the Reynolds number.

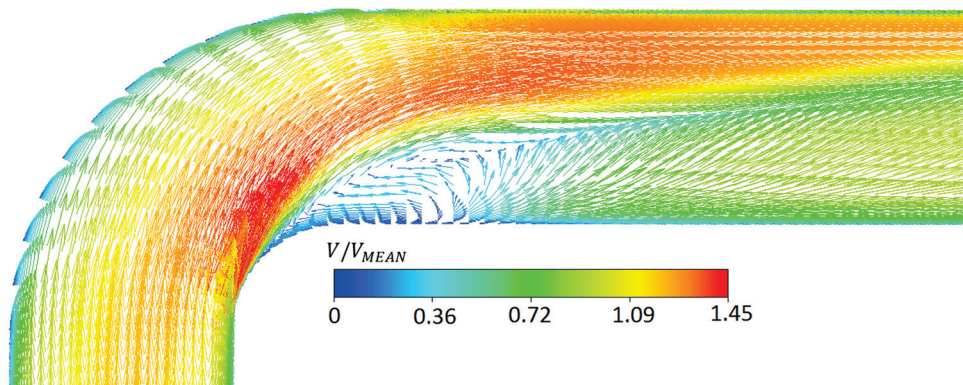
The separation angle increased with the increase in the Reynolds number (Figure 10). Firstly, it was linear in a laminar and transitional flow region ($Re < 7000$). At a high Reynolds number ($Re < 50,000$), the value of the separation angle oscillated around 65° . Figure 11 presents the velocity vectors in a ZY plane (see Figures 6 and 7) at three different Reynolds number values. The separation point is visible as a sudden decrease in the velocity value near the inner wall of the elbow. Behind the separation point, the pressure decreases and the vortex structure is generated. In general, the higher the Reynolds number, the larger the vortex structure. It is very visible when comparing the velocity vector in the elbow at $Re = 500$ (Figure 11a) and $Re = 100,000$ (Figure 11c). The vortex structure visible in Figure 11 creates a flow dead zone. The size of the flow dead zone generally depends on the location of the separation point. It is important to pay attention to the stabilization of

the value of the separation angle (Figure 10) and the minor loss coefficient (Figure 5) at a high Reynolds number ($Re \geq 50,000$). Of course, the minor loss coefficient also depends on other vortex structures and losses generated in an elbow as a secondary flow (Figure 9).

(a)



(b)



(c)

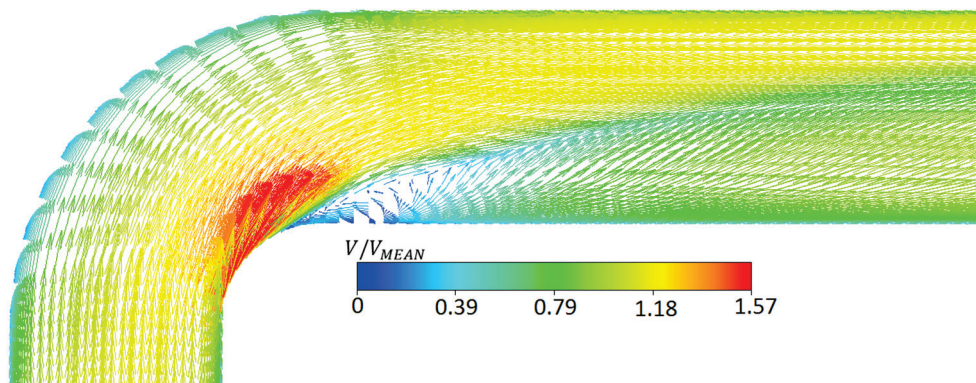


Figure 11. The velocity vector in elbow and outlet duct at $Re = 500$ (a), $Re = 2300$ (b), and $Re = 100,000$ (c).

The secondary flow and the vortex structure visible in Figure 11 overlap and create a relatively complex vortex structure inside the elbow. One of the methods for vortex identification is a Q-criterion [41]. Figure 12 shows the Q iso surface for the level equal to 0.005. The vortex shape indicates how much secondary flow contributes to the flow in the elbow. Additionally, the size of the structure shows that the impact of the secondary

flow decreases with an increase in the Reynolds number. The characteristic structure of the vortex shown in Figure 11 could be observed along the entire length of the outlet channel in all investigated cases.

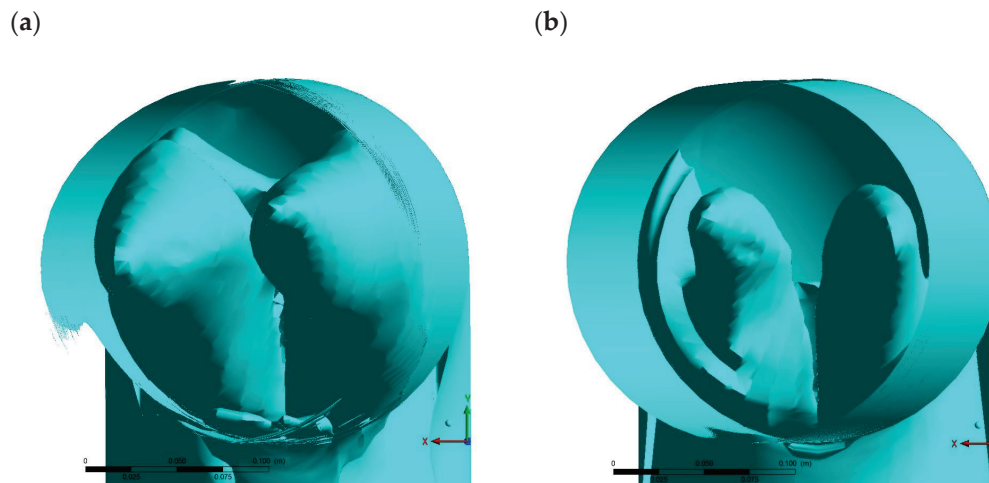


Figure 12. Q iso surface for level 0.005 on the outlet of the elbow at $Re = 1000$ (a) and $Re = 100,000$ (b).

The occurrence of vortices presented by the Q iso surface in Figure 12 is one of the reasons for the pressure loss in the elbow–vortex structures, which dissipates the energy of the flow. One of the methods for extinguishing them is the use of straighteners, before, after, or during the fittings [42–44]. However, the operation of straighteners is based on the use of appropriate elements to extinguish turbulence due to increased local pressure losses or physically preventing the formation of vortex structures [45]. Therefore, the quenching of vortices may be energetically unjustified, even though, with an increase in the Reynolds number, they appear faster and have greater intensity (understood in this case as Q being a function of strain rate tensor and vorticity tensor). However, the vortex structure can affect measurements that take place immediately behind chokes such as elbows [25].

4. Conclusions

The paper deals with the simulation of the flow in the elbow. The turbulent k- ω model and SIMPLE simulation method were used and the Reynolds number was analyzed in a range from 500 to 1,000,000. Based on the presented discussion and analysis, the following conclusions can be drawn:

1. The k- ω model can be used to calculate head pressure loss. However, the Darcy friction factor was similar to the theoretical value only in the case of the high Reynolds number ($Re \geq 2300$).
2. The calculated minor loss coefficient depends on the Reynolds number. The calculated value of the minor loss coefficient was similar to the engineering/literature value at $Re \geq 20,000$. The minor loss coefficient should be better investigated for Reynolds numbers $Re < 10,000$.
3. The secondary flow was observed in all investigated cases. It was observed also by the analysis of the u velocity profile.
4. The separation point location depends on the Reynolds number and is stabilized for $Re > 20,000$, similar to the minor loss coefficient. The analysis of the velocity vectors showed the flow dead zone behind the elbow. The flow dead zones were bigger the lower the Reynolds number.
5. The Q-criterion vortex identification method showed two vortex structures. They were the result of overlapping of the main flow and the secondary flow. The impact of the secondary flow decreased with the Reynolds number increasing.

Author Contributions: Conceptualization, software, investigation, resources, data curation, writing—original draft preparation, supervision, funding acquisition, visualization, project administration, E.S.; methodology, validation, formal analysis, E.S. and M.S. (Michał Stopel); writing—review and editing, M.S. (Michał Stopel) and M.S. (Mikołaj Szyca). All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Data Availability Statement: All data generated or used during the study are available from the corresponding author upon request.

Conflicts of Interest: The authors declare no conflicts of interest.

References

- dos Santos, A.P.P.; Andrade, C.R.; Zaparoli, E.L. CFD Prediction of the Round Elbow Fitting Loss Coefficient. *Int. J. Mech. Mechatron. Eng.* **2014**, *8*, 743–747.
- Smyk, E.; Mrozik, D.; Wawrzyniak, S.; Peszyński, K. Tabular air deflector in ventilation ducts. In Proceedings of the 23rd International Conference Engineering Mechanics, Svratka, Czech Republic, 15–18 May 2017; pp. 882–885.
- Gallegos-Muñoz, A.; Uzárraga Rodríguez, N.C.; Belman Flores, J.M.; Rangel Hernández, V.H. Analysis of effect caused by fitting in the measurements of flow in air conditioning system. *Appl. Therm. Eng.* **2012**, *33–34*, 227–236. [CrossRef]
- Wang, J.; Shirazi, S.A. CFD based correlation for mass transfer coefficient in elbows. *Int. J. Heat Mass Transf.* **2001**, *44*, 1817–1822. [CrossRef]
- Spedding, P.L.; Benard, E.; Crawford, N.M. Fluid flow through a vertical to horizontal 90° elbow bend III three phase flow. *Exp. Therm. Fluid Sci.* **2008**, *32*, 827–843. [CrossRef]
- Zahedi, R.; Babaee Rad, A. Numerical and experimental simulation of gas-liquid two-phase flow in 90-degree elbow. *Alex. Eng. J.* **2022**, *61*, 2536–2550. [CrossRef]
- Cao, X.; Zhang, P.; Li, X.; Li, Z.; Zhang, Q.; Bian, J. Experimental and numerical study on the flow characteristics of slug flow in a horizontal elbow. *J. Pipeline Sci. Eng.* **2022**, *2*, 100076. [CrossRef]
- Ono, A.; Kimura, N.; Kamide, H.; Tobita, A. Influence of elbow curvature on flow structure at elbow outlet under high Reynolds number condition. *Nucl. Eng. Des.* **2011**, *241*, 4409–4419. [CrossRef]
- Sudo, K.; Sumida, M.; Hibara, H. Experimental investigation on turbulent flow through a circular-sectioned 180° bend. *Exp. Fluids* **2000**, *28*, 51–57. [CrossRef]
- Röhrig, R.; Jakirlić, S.; Tropea, C. Comparative computational study of turbulent flow in a 90° pipe elbow. *Int. J. Heat Fluid Flow* **2015**, *55*, 120–131. [CrossRef]
- Kim, J.; Yadav, M.; Kim, S. Characteristics of secondary flow induced by 90-degree elbow in turbulent pipe flow. *Eng. Appl. Comput. Fluid Mech.* **2014**, *8*, 229–239. [CrossRef]
- Spedding, P.L.; Benard, E.; McNally, G.M. Fluid Flow through 90 Degree Bends. *Dev. Chem. Eng. Miner. Process.* **2010**, *12*, 107–128. [CrossRef]
- Dutta, P.; Nandi, N. Effect of Reynolds number and curvature ratio on single phase turbulent flow in pipe bends. *Mech. Mech. Eng.* **2015**, *19*, 5–16. [CrossRef]
- Jarrahi, M.; Castelain, C.; Peerhossaini, H. Secondary flow patterns and mixing in laminar pulsating flow through a curved pipe. *Exp. Fluids* **2011**, *50*, 1539–1558. [CrossRef]
- Rup, K.; Sarna, P. Analysis of turbulent flow through a square-sectioned duct with installed 90-degree elbow. *Flow Meas. Instrum.* **2011**, *22*, 383–391. [CrossRef]
- Patiño-Jaramillo, G.A.; Iglesias, I.; Vera, M. Laminar flow and pressure loss in planar Tee joints: Numerical simulations and flow analysis. *Eur. J. Mech. B/Fluids* **2022**, *92*, 75–89. [CrossRef]
- Sun, Q.L.; Xia, L.; Deng, L.; Wang, J.G.; Wang, G.L.; Feng, D. Experimental and Numerical Simulation Analyses of Elbow Erosion in Surface Process of Deepwater Gas Well Testing. *J. Fail. Anal. Prev.* **2024**, *24*, 202–215. [CrossRef]
- Liu, E.B.; Huang, S.; Tian, D.C.; Shi, L.M.; Peng, S.B.; Zheng, H. Experimental and numerical simulation study on the erosion behavior of the elbow of gathering pipeline in shale gas field. *Pet. Sci.* **2024**, *21*, 1257–1274. [CrossRef]
- Chen, M.; Tan, L. An improved method combined modified drag model and modified erosion model based on LES for solid-liquid multiphase flow and erosion with application in a 90° elbow. *Wear* **2024**, *538–539*, 205214. [CrossRef]
- Smyk, E.; Mrozik, D.; Łukasz, O.; Peszyński, K. Numerical simulation of minor losses coefficient on the example of elbows. *EPJ Web Conf.* **2018**, *180*, 02093. [CrossRef]
- Salehi, M.; Sleiti, A.K.; Idem, S. Study to identify computational fluid dynamics models for use in determining HVAC duct fitting loss coefficients. *Sci. Technol. Built Environ.* **2017**, *23*, 181–191. [CrossRef]
- Liu, W.; Long, Z.; Chen, Q. A procedure for predicting pressure loss coefficients of duct fittings using computational fluid dynamics (RP-1493). *HVAC R Res.* **2012**, *18*, 1168–1181. [CrossRef]
- Sami, S.; Cui, J. Numerical study of pressure losses in close-coupled fittings. *HVAC&R Res.* **2004**, *10*, 539–552. [CrossRef]

24. Zmrhal, V.; Schwarzer, J. Numerical simulation of local loss coefficients of ventilation duct fittings. In Proceedings of the Eleventh International IBPSA Conference, Glasgow, Scotland, 27–30 July 2009; pp. 1761–1766.
25. Weissenbrunner, A.; Ekat, A.; Straka, M.; Schmelter, S. A virtual flow meter downstream of various elbow configurations. *Metrologia* **2023**, *60*, 5. [CrossRef]
26. Durst, F.; Ray, S.; Ünsal, B.; Bayoumi, O.A. The development lengths of laminar pipe and channel flows. *J. Fluids Eng. Trans. ASME* **2005**, *127*, 1154–1160. [CrossRef]
27. Kahramanoglu, E.; Sezen, S.; Bayraktar, S. Computational fluid dynamics analyses on the hydrodynamic entry length in internal flows. *Pamukkale Univ. J. Eng. Sci.* **2017**, *23*, 372–377. [CrossRef]
28. Grossmann, S.; Lohse, D. Curvature effects on the velocity profile in turbulent pipe flow. *Eur. Phys. J. E* **2017**, *40*, 7–12. [CrossRef] [PubMed]
29. Bariya, H.G.; Patel, M.P. On the Solution of Blasius Boundary Layer Equations of Prandtl-Eyring Fluid Flow Past a Stretching Sheet. *Indian J. Sci. Technol.* **2024**, *17*, 1237–1244. [CrossRef]
30. Wang, Y.; Zhang, Z. Asymptotic behavior of the steady Prandtl equation. *Math. Ann.* **2023**, *387*, 1289–1331. [CrossRef]
31. Cengel, Y.A.; Cimbala, J.M. *Fluid Mechanics: Fundamentals and Applications*; Mc Graw-Hill, Higher Education: New York, NY, USA, 2006.
32. Turgut, O.E.; Asker, M.; Çoban, M.T. A review of non iterative friction factor correlations for the calculation of pressure drop in pipes. *Bitlis Eren Univ. J. Sci. Technol.* **2014**, *4*, 1–8. [CrossRef]
33. Vatankhah, A.R. Approximate Analytical Solutions for the Colebrook Equation. *J. Hydraul. Eng.* **2018**, *144*, 1–8. [CrossRef]
34. Kijarvi, J. Darcy Friction Factor Formulae in Turbulent Pipe Flow. *Lunowa Fluid Mech. Pap.* **2011**, 110727, 1–11.
35. Hendiger, J.; Zietek, P.; Chlidzinska, M. *Wentylacja i klimatyzacja. Materiały pomocnicze do projektowania (From Polish: Ventilation and Air Conditioning. Support Materials for Design)*; Venture Industries Sp. z o.o.: Warszawa, Poland, 2013.
36. Dutta, P.; Nandi, N. Study on pressure drop characteristics of single phase turbulent flow in pipe bend for high reynolds number. *ARPJ. Eng. Appl. Sci.* **2015**, *10*, 2221–2226.
37. Glenn, A.L.; Bulusu, K.V.; Shu, F.; Plesniak, M.W. Secondary flow structures under stent-induced perturbations for cardiovascular flow in a curved artery model. *Int. J. Heat Fluid Flow* **2012**, *35*, 76–83. [CrossRef]
38. Kalpakli, A.; Örlü, R.; Alfredsson, P.H. Vortical patterns in turbulent flow downstream a 90° curved pipe at high Womersley numbers. *Int. J. Heat Fluid Flow* **2013**, *44*, 692–699. [CrossRef]
39. Valeriu, D. A New Mathematical Model for Coandă Effect Velocity Approximation. *Incas Bull.* **2012**, *4*, 85–92. [CrossRef]
40. Dumitrache, A.; Frunzulica, F.; Ionescu, T.C. Mathematical Modelling and Numerical Investigations on the Coanda Effect. In *Nonlinearity, Bifurcation and Chaos-Theory and Applications*; Hagedorn, P., Awrejcewicz, J., Eds.; InTech: London, UK, 2012; pp. 101–132.
41. Jeong, J.; Hussain, F. On the identification of a vortex. *J. Fluid Mech.* **1995**, *285*, 69–94. [CrossRef]
42. Peszynski, K.; Tesař, V.; Kuliš, E. Evaluation of linear losses in ventilation ducts with a rounded rectangle cross-section. *MATEC Web Conf.* **2019**, *302*, 01021. [CrossRef]
43. Jurga, A.P.; Janocha, M.J.; Ong, M.C.; Yin, G. Numerical Investigations of Turbulent Flow Through a 90-Degree Pipe Bend and Honeycomb Straightener. *J. Fluids Eng.* **2024**, *146*, 021307. [CrossRef]
44. Puspitasari, D.; Pramadhony, P.; Ellyanie, E.; Marwani, M. The effects of flow straightener inclination on distribution of flue gas flow. *AIP Conf. Proc.* **2018**, *2001*, 060002.
45. Scheiman, J. *Comparison of Experimental and Theoretical Turbulence Reduction Characteristics for Screens, Honeycomb, and Honeycomb-Screen Combinations*; NASA Technical Paper; NASA: Washington, DC, USA, 1981.

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.

MDPI AG
Grosspeteranlage 5
4052 Basel
Switzerland
Tel.: +41 61 683 77 34

Water Editorial Office
E-mail: water@mdpi.com
www.mdpi.com/journal/water



Disclaimer/Publisher's Note: The title and front matter of this reprint are at the discretion of the Guest Editor. The publisher is not responsible for their content or any associated concerns. The statements, opinions and data contained in all individual articles are solely those of the individual Editor and contributors and not of MDPI. MDPI disclaims responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.



Academic Open
Access Publishing

mdpi.com

ISBN 978-3-7258-8272-4