

Advances in Pipe and Channel Flow Modeling

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

The fluid flow within confined conduits (pipes and channels) remains a cornerstone of hydraulic engineering, underpinning the design, analysis and safe operation of countless industrial, environmental and infrastructural systems. The ongoing progress in computational power, measurement techniques and multi-physics understanding continuously refines our ability to model these flows, from predictable steady-state operations to complex transient events. This Special Issue, “Modelling Flows in Pipes and Channels” seeks to capture and contribute to these advancements by presenting cutting-edge research that bridges fundamental insight with practical application.

To illustrate this, we confine our focus to pressurized flows, where the modeling of transient phenomena like water hammer exemplifies the domain’s vibrant challenges and innovations. The ongoing research is pushing boundaries on multiple fronts: refining unsteady friction models for more accurate damping prediction [1–3]; developing novel analytical and quasi-two-dimensional solutions for rapid benchmarking and deeper physical insight [4–6] and understanding complex interactions in series-connected pipelines of different materials [7–9]. Furthermore, advanced studies focus on the cavitation effects, employing sophisticated two-phase models and experimental visualization [10–13], while high-fidelity CFD simulations provide unprecedented spatial detail of wave propagation and fluid–structure interaction [14–17]. Critical operational scenarios like rapid filling and emptying, with their inherent air–water interactions, are also being elucidated through combined experimental and 3D computational approaches [18–21]. This has naturally led to applied research on innovative damping and mitigation strategies, from compliant polymeric liners and bypass systems to system vibration analysis [22–25].

This Special Issue serves as a forum for the latest developments across this broad spectrum of problems. We invite contributions that address mathematical model development, innovative numerical schemes, advanced experimental validation and novel applications related to both steady and transient flows in all types of conduits. By collating diverse perspectives, we aim to foster the necessary interdisciplinary dialog to tackle the enduring and emerging challenges in pipe and channel flow modeling, ultimately contributing to more resilient, efficient and sustainable engineering systems.



Received: 19 January 2026

Accepted: 26 January 2026

Published: 28 January 2026

Copyright: © 2026 by the author.

Licensee MDPI, Basel, Switzerland.

This article is an open access article distributed under the terms and conditions of the [Creative Commons Attribution \(CC BY\)](https://creativecommons.org/licenses/by/4.0/) license.

2. Review of Contribution to the Special Issue

The study by González et al. (Contribution 1) introduces a practical and efficient numerical methodology for predicting minor losses in pipes with abrupt section changes. Both sudden contraction and sudden expansion configurations are investigated through a combined theoretical and numerical approach. An initial one-dimensional theoretical model is developed to provide physical insight, while its limitations are clearly identified. The main contribution of the work is a two-dimensional axisymmetric CFD model that

is capable of accurately describing pressure losses in both geometries. The numerical results for sudden contractions show strong agreement with established experimental data for both air and water flows. The model demonstrates robust mesh independence, even producing reliable results with relatively coarse discretizations. Comparisons among the Spalart–Allmaras, $k-\epsilon$ and $k-\omega$ SST turbulence models reveal negligible differences in the predicted loss coefficients. Additional simulations using a Reynolds stress model improve the accuracy of loss predictions for the intermediate contraction ratios. The evolution of the loss coefficient as a function of the diameter ratio emerges as the most relevant quantitative finding of the study. Although the model tends to overpredict losses in pipe expansions, it remains a valuable and rapid tool for the preliminary design and analysis of geometrical changes in pipe flows.

The paper by Kumar et al. (Contribution 2) introduces a novel semi-inverse potential flow framework that models flow over round-crested weirs by constructing only streamlines, rather than a full flow net. A key innovation is the use of the Serre–Green–Naghdi equation, instead of traditional Bernoulli-type Boussinesq formulations, to generate a more accurate initial free-surface profile. The study advances earlier potential flow models by integrating this initial profile into Boadway’s potential flow formulation. The numerical accuracy is improved through the use of a five-point central finite difference scheme and fourth-order spatial discretization of the Laplace equation. The approach enables a precise computation of pressure and velocity fields in transcritical flow conditions. Unlike prior studies, the model explicitly quantifies the influence of streamline curvature on pressure variations along the weir bed. The methodology is demonstrated across multiple weir geometries, including Gaussian, parabolic and semicircular shapes. A dedicated semicircular weir experiment provides direct validation of the predicted pressure and velocity profiles. The close agreement between numerical results and experimental data highlights the robustness of the proposed 2-D ideal flow model. Overall, the study establishes a computationally efficient and physically insightful alternative to the existing potential flow approaches for analyzing complex weir flows.

Sofiadis et al.’s contribution (Contribution 3) presents one of the first applications of the micropolar fluid model to the study of the turbulent open-channel flows that are relevant to environmental applications. The main novelty lies in treating particulate and secondary-phase effects within a fully Eulerian framework, avoiding the complexity of traditional Lagrangian approaches. The micropolar model is coupled with direct numerical simulations (DNS) to rigorously assess its capability in describing turbulent wall-bounded flows. The study demonstrates that the micropolar formulation can accurately reproduce both unladen and particulate flow characteristics at moderate Reynolds numbers. Validation against experimental data and previous DNS results shows excellent agreement, often outperforming existing turbulence models. A key contribution is the systematic investigation of the effect of micropolar viscosity as a proxy for secondary-phase properties. The results clarify previously contradictory findings by revealing mixed velocity behavior between particulate and single-phase flows in the inner and outer regions. The proposed model provides improved physical interpretation of particle–turbulence interactions near the wall. Turbulence intensities and Reynolds shear stresses are captured consistently, highlighting the model’s robustness. Overall, the work establishes the micropolar model as a computationally efficient and physically reliable alternative for simulating turbulent particulate open-channel flows.

Fuertes-Miquel et al.’s paper (Contribution 4) introduces a generalized one-dimensional mathematical model to analyze transient events caused by entrapped air pockets during filling operations in pipelines with undulating profiles. The main novelty lies in explicitly incorporating the effects of blocking water columns, which are neglected in previously

developed filling models. The proposed formulation accounts for the interaction between multiple air pockets and air valves within a single unified framework. Unlike in earlier approaches, the model includes the dynamic behavior of expelled air through air valves during transient events. A polytropic formulation is adopted to realistically describe the air pocket compression and expansion processes. The methodology is applied to a pipeline configuration with two entrapped air pockets and an air valve located at the highest point. Experimental data from a laboratory-scale installation are used to assess the pressure response associated with blocking columns. The results demonstrate that air valves can significantly reduce pressure peaks when air expulsion is unrestricted, but this may lead to new transients upon valve closure. The study further examines the influence of air valve size and discharge characteristics on pressure surges. Overall, this work provides a practical modeling framework that enhances the understanding and prediction of hazardous pressure peaks during pipeline filling operations.

Vardy's paper (Contribution 5) introduces a novel numerical methodology for determining the possible asymptotic shapes of pressure wavefronts in long ducts without simulating their full propagation history. The key innovation lies in predicting final wavefront states directly, rather than computing their evolution from the prescribed initial disturbances. By avoiding long-domain unsteady simulations, the method overcomes the severe computational limitations associated with extremely fine spatial discretization. The approach is specifically tailored to study asymptotic behavior and is therefore complementary to conventional cause-and-effect wave propagation models. A major advantage of the methodology is its ability to achieve very high accuracy using minimal computational resources. The analysis requires simulations that only use short duct segments and short time intervals, even when using grid sizes that are orders of magnitude smaller than the duct diameter. The method is shown to be numerically stable, despite relying on extrapolation for two governing parameters. A detailed assessment demonstrates that the influence of these extrapolations on the results is minimal and acceptable. The framework is applied to ducts with enclosed air chambers: a configuration that was previously difficult to analyze accurately, due to CPU constraints. The study reveals unique one-to-one relationships between asymptotic wavefront pressure amplitude, propagation speed and maximum steepness, highlighting the physical characteristics that do not exist in non-asymptotic wavefronts.

The paper by Ferreira and Covas (Contribution 6) introduces a systematic and comprehensive assessment of radial mesh sensitivity in axisymmetric quasi-2D models for smooth-turbulent transient pipe flows. The main novelty lies in establishing clear guidelines to balance the numerical accuracy and computational efficiency when simulating hydraulic transients. An extensive parametric study is carried out, using 80 different radial meshes and varying the number of cylinders, the geometric common ratio and the axial discretization. The study demonstrates that geometric sequence meshes significantly improve the resolution near the pipe wall, where unsteady shear stress gradients are the most critical. A key contribution is the identification of optimal common ratio values that allow for accurate transient predictions with a substantially reduced computational cost. The comparison between the two best-performing meshes highlights that increasing the common ratio can halve the computation time while maintaining comparable accuracy. The work shows that discrepancies between meshes are confined to a very short time window after the first pressure surge. The transient response is characterized in detail through piezometric head, wall shear stress and velocity histories, confirming physical consistency. The analysis provides new insights into the localization of shear stress gradients within the viscous and buffer layers. Overall, the study offers practical and transferable mesh design recommendations for efficient axisymmetric modeling of turbulent hydraulic transients.

The study by Dmitriev et al. (Contribution 7) presents a combined experimental and numerical investigation of swirling single-phase coolant flow and heat transfer in channels of complex geometry under nuclear power plant operating conditions. The main novelty lies in the systematic comparison of swirl intensifiers with constant and variable twist pitches in a realistic heat exchanger configuration. Variable pitch swirlers are shown to enhance heat transfer efficiency while limiting hydraulic power penalties. The study introduces a detailed experimental validation of temperature fields and pressure drops for multiple swirling configurations. Advanced turbulence modeling, including an omega-based Reynolds stress model and an SST model with curvature correction, is applied and critically assessed. The work demonstrates that Reynolds stress modeling captures swirling flow structures and boundary layers more accurately than two-equation models. Numerical simulations reveal previously undocumented flow and thermal features arising from changes in swirl pitch along the channel. An effectiveness criterion is introduced to quantify the performance of each swirl intensifier, relative to a smooth pipe. The results highlight the trade-off between computational cost and accuracy among turbulence modeling approaches. Overall, the study provides new insights into the design and modeling of swirl-enhanced heat exchangers for high-performance thermal systems.

Mairal et al.'s study (Contribution 8) presents a one-dimensional methodology for modeling transitory and supercritical flow regimes in channel networks, using the Junction Riemann Problem (JRP). The novelty of the work lies in the inclusion of limiting coefficients in the junction coupling conditions, allowing the method to handle a wide range of flow regimes robustly. The method is validated against reference 2D numerical solutions in challenging geometries, demonstrating its ability to reproduce the main features of wave propagation. Unlike conventional 1D methods, this approach captures the long-term behavior of flow in networks with multiple junctions. The proposed formulation offers a computationally efficient alternative to full 2D simulations while maintaining high accuracy for practical applications. The study emphasizes the transitory flow conditions, showing that shock waves at junctions are properly managed by the JRP method. The methodology provides a framework to include energy conservation in 1D network models, though frictional losses and local reflections are not yet considered. The results highlight the potential of 1D methods for complex channel networks, even under supercritical conditions. Overall, this work demonstrates that 1D modeling, when combined with the JRP, can be a reliable and efficient tool for engineering and environmental studies.

Popov et al.'s paper (Contribution 9) focuses on improving engineering predictions of turbulence in narrow, curved channels, specifically in high-flux isotope reactor (HFIR) geometries. The study enhances a standard Reynolds-averaged Navier–Stokes (RANS) k - ω shear-stress transport (SST) turbulence model by modifying the dissipation equation's loss term and using high wall-distance (high y^+) computational grids. These changes enable fast, computationally efficient simulations while maintaining sufficient accuracy for engineering purposes. The methodology was validated against high-resolution numerical data, showing improved prediction of mean flow parameters and turbulence behavior in parallel channel flows. The approach allows for practical computation on coarse meshes with modest computational resources, making it suitable for reactor thermal hydraulics and other industrial channel flow applications. The study demonstrates that proper turbulence modeling is essential for predicting heat transfer and pressure losses in narrow channels, providing a foundation for safe and efficient reactor operation, particularly in the transition to low-enriched fuels.

Martins et al.'s study (Contribution 10) investigates the transient behavior of extended partial blockages (EPBs) in pressurized pipe systems, using 3D CFD simulations based on compressible Navier–Stokes equations in OpenFOAM. EPBs, which represent the lon-

gitudinal growth of discrete blockages, significantly alter flow dynamics and pressure responses. The simulations show that EPBs produce distinct pressure signatures compared to discrete blockages: the first reflected pressure wave is similar but inversely proportional to the blockage diameter, while the second peak can exceed conventional predictions due to multiple wave interactions. The length of the EPB strongly affects peak amplitudes and the complexity of the pressure response. These findings demonstrate that transient test-based techniques can accurately detect and characterize EPBs, allowing water utilities to implement targeted maintenance, optimize hydraulic efficiency, reduce energy consumption and prevent blockages from progressing to total obstruction. The study provides a foundation for improved non-invasive diagnostic strategies and proactive water network management.

Gandía-Barberá and Hoyas' study (Contribution 11) investigates coherent structures in turbulent channel flows across a transition from pure Poiseuille to pure Couette flows, at a friction Reynolds number of approximately 250. Using DNS data and a percolation-based method to identify structures, the authors analyze low-speed streaks and intense Reynolds stress structures, focusing on their geometry, energy content and spatial organization. The results show that increasing the mean shear in Couette-like flows enhances the energetic intensity and spanwise organization of the attached streaks and Q structures, while Poiseuille-like flows exhibit a more balanced distribution between the attached and detached structures. Despite changes in shear, the geometric characteristics of the streaks and Q structures, including dimensions, volume ratios and angular inclination, remain largely consistent. The wall normal distribution of Q structures reveals that near-wall regions are dominated by Q4 sweeps, whereas further from the wall, Q2 bursts prevail, with the overall intensity increasing with shear and reaching a maximum in pure Couette flows. The spacing between same-type Q pairs decreases in the streamwise direction with the increasing shear, while Q2–Q4 pairs tend to align spanwise and increase their distance from each other, reflecting the influence of the very large streamwise rolls that are present in Couette-like configurations. Overall, the study demonstrates that the mean shear primarily modifies the energetic intensity and spatial distribution of coherent structures, whereas their geometry and topology remain robust, providing new insight into the interaction between large-scale rolls and near-wall turbulence in mixed Couette–Poiseuille flows and laying the groundwork for future investigations at higher Reynolds numbers and with refined detection techniques.

Nachbin's contribution (Contribution 12) addresses the modeling of water waves on graphs, focusing on the reflection and transmission of waves at graph vertices. This is an area that has received limited attention in the literature. The author compares different mathematical approaches on a two-dimensional "fattened" graph, from hyperbolic conservation laws to weakly nonlinear and weakly dispersive systems, and explores the reduction in these models to one-dimensional graphs. A novel aspect of the work is the systematic inclusion of junction angles in the 1D graph model, which allows for a more general compatibility condition at the vertex when compared with the 2D fattened-graph solutions. The discussion highlights that the existing wave-on-graph models do not account for junction angles or provide detailed comparisons with mesoscale 2D models, which significantly affects the reflection–transmission dynamics. Nachbin proposes that future work steps back from weakly dispersive Boussinesq systems and studies nondispersive hyperbolic systems, using 2D shallow water equations, including more comprehensive comparisons between thick-Y-graph solutions and reduced 1D models with variable angles. Other modeling extensions that are under consideration include channels with non-rectangular or spatially varying cross sections, as well as different types of solutions beyond solitary waves, particularly to understand discrepancies in asymmetric junctions where forking angles affect the accuracy of 1D–2D model agreement. The study thus lays the groundwork

for the improved modeling of wave propagation on networks, highlighting both the current gaps and some promising directions for future research.

3. Conclusions

This Special Issue presents a diverse array of innovative numerical and experimental methodologies that address key challenges in fluid mechanics. Collectively, the contributions advance the prediction of complex flows, from internal pipe systems to open channels and turbulent structures (Table 1). Several studies introduce efficient and accurate computational frameworks to model transient phenomena and complex geometries, while others successfully apply advanced physical models, validating them against experimental data.

Table 1. Short overview of published papers.

#	Author/s	Flow Type	Flow Regime	System Type	Main Focus/Phenomena
1	González et al.	Turbulent	Steady	Pressurized	Minor loss coefficients for abrupt pipe section changes (CFD-RANS)
2	Kumar et al.	Potential/Ideal	Steady	Open Channel	Pressure/velocity profiles over a weir, using potential flow model
3	Sofiadis et al.	Turbulent (Micropolar)	Steady	Open Channel	Turbulent open-channel flow with suspended sediments (micropolar theory)
4	Fuertes-Miquel et al.	Transient Mixed-Phase	Transient	Pressurized	Effects of expelled air during pipeline filling in undulating profiles
5	Vardy	Wave Propagation	Transient	Pressurized/Ducts	Modeling asymptotic pressure wavefronts in long ducts with chambers
6	Ferreira and Covas	Transient Turbulent	Transient	Pressurized	Mesh sensitivity analysis for axisymmetric modeling of smooth–turbulent transients
7	Dmitriev et al.	Turbulent	Steady	Complex Channel	Swirling flows and heat transfer in channels of complex geometry (PWR-related)
8	Mairal et al.	Shallow Water Flow	Transient	Open Channel (Network)	Validation of Junction Riemann Problem solver for transitory flows in channel networks
9	Popov et al.	Turbulent	Steady (RANS)	Pressurized (Narrow)	High y^+ turbulence modeling for narrow channel flows in nuclear reactors (HFIR)
10	Martins et al.	Transient Turbulent	Transient	Pressurized	CFD-based detection and characterization of extended partial blockages in pipes
11	Gandía-Barberá and Hoyas	Turbulent	Steady (DNS)	Channel Flow	Spatial/energetic organization of coherent structures in turbulent channel flow
12	Nachbin	Wave Propagation	Steady/Quasi	Open Channel (Graph)	Mathematical modeling of water wave propagation on network graphs

A strong focus is placed on enhancing the safety and design of engineering systems. The research provides critical insights into transient pressure mechanisms, the characterization of flow obstructions and the performance of flow-control devices. Furthermore, the volume offers practical improvements to turbulence modeling for industrial applications and explores fundamental coherent structures in shear flows. By also examining the foundational methodologies for complex networked systems, this Special Issue suggests essential future research directions. This collection bridges fundamental research and engineering application, delivering both practical tools and a deeper physical understanding across a wide spectrum of fluid flow phenomena.

Funding: This research received no external funding.

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

List of Contributions

1. González, J.; Meana-Fernández, A.; Pérez, I.V.; Oro, J.M.F. Minor Loss Coefficient for Abrupt Section Changes in a Cylindrical Pipe Using a Numerical Approach. *Fluids* **2024**, *9*, 152. <https://doi.org/10.3390/fluids9070152>.
2. Kumar, M.R.A.; Hanmaiahgari, P.R.; Pu, J.H. Pressure and Velocity Profiles over a Weir Using Potential Flow Model. *Fluids* **2024**, *9*, 182. <https://doi.org/10.3390/fluids9080182>.
3. Sofiadis, G.; Liakopoulos, A.; Palasis, A.; Sofos, F. Turbulent Micropolar Open-Channel Flow. *Fluids* **2024**, *9*, 202. <https://doi.org/10.3390/fluids9090202>.
4. Fuertes-Miquel, V.S.; Coronado-Hernández, O.E.; Arrieta-Pastrana, A. Effects of Expelled Air during Filling Operations with Blocking Columns in Water Pipelines of Undulating Profiles. *Fluids* **2024**, *9*, 212. <https://doi.org/10.3390/fluids9090212>.
5. Vardy, A.E. On the Modelling of Asymptotic Wavefronts in Long Ducts with Chambers. *Fluids* **2024**, *9*, 240. <https://doi.org/10.3390/fluids9100240>.
6. Ferreira, P.L.; Covas, D.I.C. Mesh Sensitivity Analysis of Axisymmetric Models for Smooth-Turbulent Transient Flows. *Fluids* **2024**, *9*, 268. <https://doi.org/10.3390/fluids9110268>.
7. Dmitriev, S.; Sobornov, A.; Kurkin, A. Features of Motion and Heat Transfer of Swirling Flows in Channels of Complex Geometry. *Fluids* **2024**, *9*, 293. <https://doi.org/10.3390/fluids9120293>.
8. Mairal, J.; Murillo, J.; Garcia-Navarro, P. A Validation of the Junction Riemann Problem in Networks of Channels Under Transitory Flow Conditions. *Fluids* **2025**, *10*, 53. <https://doi.org/10.3390/fluids10030053>.
9. Popov, E.; Mecham, N.; Grubbs, T. High y^+ Shear-Stress Turbulence Implementation for High Flux Isotope Reactor Narrow Channel Flows. *Fluids* **2025**, *10*, 85. <https://doi.org/10.3390/fluids10040085>.
10. Martins, N.M.C.; Covas, D.I.C.; Brunone, B.; Meniconi, S.; Capponi, C. CFD-Based Transient Analysis for the Detection and Characterisation of Extended Partial Blockages in Pipes. *Fluids* **2025**, *10*, 291. <https://doi.org/10.3390/fluids10110291>.
11. Gandía-Barberá, S.; Hoyas, S. Spatial and Energetic Organization of Coherent Structures in Couette-Poiseuille Turbulent Channels. *Fluids* **2026**, *11*, 18. <https://doi.org/10.3390/fluids11010018>.
12. Nachbin, A. Modelling Water Waves on Graphs. *Fluids* **2025**, *10*, 140. <https://doi.org/10.3390/fluids10060140>.

References

1. Jiang, D.; Zeng, C.; Lu, Q.; Guo, Q. Water hammer in pipelines based on different friction models. *Sci. Rep.* **2024**, *14*, 953. [\[CrossRef\]](#)
2. Mohsin, Z.; Sulaiman, Z.; Khilqa, S. Comparative analysis of steady and unsteady friction models in numerical simulation of water hammer phenomenon. *Eng. Appl. Sci. Res.* **2025**, *52*, 207–220. Available online: <https://ph01.tci-thaijo.org/index.php/easr/article/view/259832> (accessed on 10 January 2026).
3. Urbanowicz, K.; Bergant, A.; Stosiak, M.; Deptuła, A.; Karpenko, M.; Kubrak, M.; Kodura, A. Water Hammer Simulation Using Simplified Convolution-Based Unsteady Friction Model. *Water* **2022**, *14*, 3151. [\[CrossRef\]](#)
4. Urbanowicz, K.; Di Nucci, C.; Sharma, B.K.; K., A.; Kawiak, M.; Krajewski, S. Revisiting Yamaguchi-Ichikawa water hammer model. *Chin. J. Phys.* **2025**, *94*, 487–503. [\[CrossRef\]](#)
5. Brühl, M. Analytical Solution for Laminar Water Hammer With Frequency-Dependent Friction. *J. Fluids Eng.* **2022**, *144*, 111302. [\[CrossRef\]](#)
6. Bordbar, M. Analytical Solution of Quasi-Two-Dimensional Method for Unsteady Transient Pressurized Pipe Flows in the Time Domain. *J. Hydraul. Eng.* **2025**, *151*, 04025035. [\[CrossRef\]](#)
7. Kubrak, M.; Kodura, A.; Malesinska, A.; Urbanowicz, K. Water Hammer in Steel-Plastic Pipes Connected in Series. *Water* **2022**, *14*, 3107. [\[CrossRef\]](#)
8. Lewandowski, M.; Adamkowski, A. Experimental Verification of 1D-Simulation Method of Water Hammer Induced in Two Series-Connected Pipes of Different Diameters: Determination of the Pressure Wave Speed. *Appl. Sci.* **2024**, *14*, 7173. [\[CrossRef\]](#)
9. Kraiem, D.; Triki, A. Tackling of hydraulic cavitation in pressurized pipe flow using high- or low-density polyethylene penstock and short-section. *AQUA Water Infrastruct. Ecosyst. Soc.* **2023**, *72*, 2126–2137. [\[CrossRef\]](#)

10. Sun, Q.; Wang, F.; Wu, Y.B.; Xu, Y.; Hao, Y. Energy analysis of transient flow with cavitation by considering the effect of water temperature in viscoelastic pipes. *J. Hydroinform.* **2023**, *25*, 2034–2052. [[CrossRef](#)]
11. Jansson, M.; Andersson, M.; Karlsson, M. High-Speed Imaging of Water Hammer Cavitation in Oil–Hydraulic Pipe Flow. *Fluids* **2022**, *7*, 102. [[CrossRef](#)]
12. Urbanowicz, K.; Bergant, A.; Kodura, A.; Kubrak, M.; Malesińska, A.; Bury, P.; Stosiak, M. Modeling Transient Pipe Flow in Plastic Pipes with Modified Discrete Bubble Cavitation Model. *Energies* **2021**, *14*, 6756. [[CrossRef](#)]
13. He, J.; Li, C.; Guo, Y. Modeling Transient Vaporous Cavitating Flow in Pipelines by a Two-Phase Homogeneous Flow Model. *Processes* **2025**, *13*, 3510. [[CrossRef](#)]
14. Neyestanaki, M.K.; Dunca, G.; Jonsson, P.; Cervantes, M.J. A Comparison of Different Methods for Modelling Water Hammer Valve Closure with CFD. *Water* **2023**, *15*, 1510. [[CrossRef](#)]
15. Cao, Y.; Zhou, L.; Ou, C.; Fang, H.; Liu, D. 3D CFD simulation and analysis of transient flow in a water pipeline. *AQUA Water Infrastruct. Ecosyst. Soc.* **2022**, *71*, 751–767. [[CrossRef](#)]
16. Zhang, Y.; Duan, H.F.; Keramat, A. CFD-aided study on transient wave-blockage interaction in a pressurized fluid pipeline. *Eng. Appl. Comput. Fluid Mech.* **2022**, *16*, 1957–1973. [[CrossRef](#)]
17. Kumar, A.M.R.; Pu, J.H.; Hanmaiahgari, P.R.; Lambert, M.F. Insights into CFD Modelling of Water Hammer. *Water* **2023**, *15*, 3988. [[CrossRef](#)]
18. Kumar, A.M.R.; Hanmaiahgari, P.R.; Lambert, M.F. CFD modelling of transient flow during rapid filling of horizontal pipeline with air. *Ocean Eng.* **2026**, *349*, 124174. [[CrossRef](#)]
19. 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](#)]
20. Ferreira, J.P.; Ferràs, D.; Covas, D.I.C.; Kapelan, Z. Air entrapment modelling in water supply networks during pipe filling events. *Urban Water J.* **2024**, *21*, 685–697. [[CrossRef](#)]
21. Paternina, 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*, 04023007. [[CrossRef](#)]
22. Kriaa, K.; Elgamal, M.; Farouk, M. Experimental study on reducing water hammer effects in uPVC pipes using rubber bypass tubes. *Ain Shams Eng. J.* **2024**, *15*, 102562. [[CrossRef](#)]
23. Vardy, A.E. On Sources of Damping in Water-Hammer. *Water* **2023**, *15*, 385. [[CrossRef](#)]
24. Kubrak, M. Water hammer mitigation by internal rubber hose. *Arch. Civ. Eng.* **2024**, *70*, 275–288. [[CrossRef](#)]
25. Henclik, S.; Janicki, W. Water hammer runs in elastically supported pipeline and the impact of system vibrations on pressure amplitudes reduction. *Int. J. Eng. Sci.* **2026**, *218*, 104400. [[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.