water

MDPI

Editorial

New Advances in Water Hammer Problems

Kamil Urbanowicz [1,]* and Helena Margarida Ramos [2]

[1] Faculty of Mechanical Engineering and Mechatronics, West Pomeranian University of Technology in Szczecin, 70-310 Szczecin, Poland
[2] Department of Civil Engineering, Architecture and Georesources, CERIS, Instituto Superior Técnico, University of Lisbon, 1049-001 Lisbon, Portugal; helena.ramos@tecnico.ulisboa.pt
* Correspondence: Kamil.Urbanowicz@zut.edu.pl

1. Introduction

When the flow within pressurized pipes experiences abrupt stoppages, initiation, or directional alteration, it gives rise to the phenomenon of water hammer, characterized by the propagation of waves. This phenomenon, which was relatively obscure in the past, has gained significant prominence in contemporary times. Given its profound relevance in practical engineering, the body of literature concerning this intricate subject matter has exhibited a consistent and substantial growth over the years.

From a historical vantage point, engineers have grappled with the challenges posed by water hammer since the inception of pipe systems for the conveyance of liquids. More than two millennia ago, Marcus Vitruvius Pollio [1] had already delineated the repercussions of water hammer and cavitation in the context of clay and lead conduits supplying water to the water distribution systems of ancient Rome. On a mathematical front, the formal description of this phenomenon commenced with the early works of von Kries [2,3], Joukowsky [4], and Allevi [5]. Joukowsky [4] delved into the intricacies of wave reflections within pipes and the deployment of air chambers, such as equalizing tanks and spring-loaded safety valves, ultimately furnishing a fundamental formula for the calculation of pressure surges resulting from rapid valve closures, which remains applicable even in modern times.

Contemporarily, the discourse surrounding this subject has expanded considerably. The central issues associated with water hammer problems encompass various dichotomies, including single versus multiphase flow, laminar versus turbulent flow, the elastic versus viscoelastic behavior of pipe materials under strain, the presence of gaseous or vaporous cavitation, Newtonian versus non-Newtonian flow, the distinction between rigid and flexible pipe walls, and the differentiation between fast (impulsive) and slow–transient flow. The deliberation of these aforementioned issues often necessitates the consideration of related phenomena, such as mechanical energy dissipation due to fluid friction, the manifestation of viscoelastic delayed deformations in pipe walls resulting from cavitation-induced liquid column separation, and fluid–structure interaction.

This Special Issue is dedicated to all aspects of modeling water hammer phenomena and the experimental verification of this particular form of unsteady flow. We have examined works that encompass (a) the exploration of accompanying phenomena (such as unsteady friction, delayed strain, cavitation in both vapor and gas phases, and fluid–structure interaction); (b) methods and devices aiming to safeguard pipe systems (such as cushion surge chambers and air valves) against the adverse consequences of water hammer (e.g., noise, vibrations, leakages, etc.); (c) advancements in numerical methodologies; and (d) the overall expansion of our understanding of water hammer.

In this Special Issue, entitled "About an Important Phenomenon—Water Hammer", we have been privileged to receive contributions from esteemed authors hailing from diverse corners of the globe, each of whom has devoted many years of professional expertise to the exploration of transient pipe flow challenges.

Citation: Urbanowicz, K.; Ramos, H.M. New Advances in Water Hammer Problems. *Water* **2023**, *15*, 4004. https://doi.org/10.3390/w15224004

Received: 5 November 2023
Accepted: 13 November 2023
Published: 17 November 2023

2. Review of New Advances

Tasca et al. (contribution 1) analyzed the identification of the typical water hammer stages associated with the pump trip scenario. The two key power-loss behaviors were found: (a) attenuated and (b) water-hammer-dominated. Deep research of the sensitivity of the transient response to the air valve outflow capacity and location was carried out. The impact of different sources was taken into account in the analyses of (a) variations in pipeline length; (b) the initial steady water velocity; (c) the elevation difference between downstream and upstream reservoirs; (d) changes in both wave speed and friction on the transient response; and (e) negative pressures. The realized study explored (a) maximum hydraulic grade (HG) values; (b) maximum head values; (c) air pocket collapse times; (d) the timing of peak transient HGs; and (e) critical orifice sizes. Finally, the paper investigated how non-slam air valve design parameters impact the transient response.

This work is of great practical importance because the selection of proper air valves is very problematic to many system designers and pipeline operators. This is mainly "because their behavior is highly non-linear, sometimes showing high sensitivity to assumed conditions and sometimes displaying robust and stable results". The authors defined four stages occurring just after a pump trip-off: depressurization, air admission, air expulsion, and the creation of a secondary wave. The minimum hydraulic grade line (HGL) is formed during the initial two stages while the maximum HGL is often only completed after the first instance of air pocket collapse. Three main transient behaviors have been identified: attenuated (type 1 behavior) for small d/D values, intermediate (type 2 behavior) for moderate d/D values, and water-hammer-dominated (type 3 behavior), which typically occurs for large d/D values (d—diameter of the outflow orifice of the air valve; D—pipe diameter). An analysis of variation of certain key initial parameters (initial velocity, elevation difference, pressure wave speed, and friction coefficients, which were initially locked in for the main set of numerical simulations) revealed that the influence of these parameters on the transient response is more evident in relation to type 3 behavior than in type 1 behavior. It was found that a well-sized non-slam air valve can mimic the behavior of a large inflow orifice and small outflow orifice air valve during the pump trip scenario. For a non-slam air valve to be effective in mitigating the transient event, the selected values should be sufficiently small, both those of the transition head (to enable the utilization of the small outflow orifice for expulsion) and those of the large outflow orifice.

Kim (contribution 2) wrote governing transient flow equations in dimensionless form and used them for two widely adapted water supply systems: a reservoir pipeline surge tank valve reservoir system and a reservoir pump check valve pipeline surge tank valve reservoir system. Inertia (lumped) and expression (integrated solution) for surge tank (located in specific section of the pipeline) were taken into account in a simplified, dimensionless way.

Both frequency and time domain solutions of the pressure response for the analyzed systems are presented and compared (partially with experimental results). The proposed solution is interesting as eliminates the main restriction: the discretization problem of numerical methods based on the method of characteristics (MOC). In MOC, the requirement of equality of the Courant number to one is hard to fulfill for existing pipeline systems.

Kim's (contribution 2) analysis of the surge tank in pipeline systems with and without pumping stations and check valves indicated that "the resonance of the system can be explained by the pipeline length and the locations of the pipeline structures, such as the surge tanks, pumps, and check valves". The difference in the time domain response between the two analyzed systems indicates that the amplification and mitigation originated from the boundary conditions of the interaction with the surge tank. The proposed normalization of the main equations (written in a dimensionless form) provides an intuitive understanding of the system response.

The air cushion surge chamber is a useful device that protects large-diameter pipeline systems (e.g., long-distance water transfer and hydropower systems). It is a closed chamber that is partially filled with water and compressed air [6,7]. "As compared to an open-type pressure regulating chamber, it is rarely restricted by geological or topographical features, and offers numerous advantages, such as shorter construction supply, lower excavation volume, cost-effectiveness, and minimal ecological impact" (contribution 3). Liu et al. (contribution 3) improved a second-order finite volume method (FVM) taking into account unsteady friction models. With the use of FVM, a novel methodology to simulate the dynamic behavior of the air cushion surge chamber in a water pipeline system was developed, and its usefulness was validated through comparisons with experimental pipe system results. The pipeline was a 582-meter-long system based on a coiled brass pipeline with a small diameter (21 mm). Four tests were conducted (two in laminar regime, Re = 1284, and two in a low-Reynolds-number turbulent one, Re = 4334).

The performed comparisons reveal that the steady friction model only accurately predicts the first pressure peak and it seriously underestimates pressure attenuation in later stages. The incorporation of unsteady friction models resulted in much better predictions of the complete pressure attenuation process. Among the two tested friction models, in particular IAB (instantaneous acceleration-based) and CBM (convolution-based models), the CBM unsteady friction model with acceptable accuracy reproduced pressure peaks and whole-pressure oscillation periods. In these systems, air cushion surge chamber energy attenuation is primarily due to pipe friction (occurring in elastic pipes) and air cushion. Some differences in peak values of subsequent amplitudes were noticed that the authors (contribution 3) related to the absence of wall heat exchange in the mathematical model.

Pezzinga (contribution 4) analyzed the results obtained using numerical 1D and 2D models with experimental results (long-duration experimental tests—about 70 periods) in which no vapor areas appeared. A numerical solution named MOC-Z by the author was used, which operates without interpolation for flows with liquid and gas. Pezzinga's experimental test stand contained several air-release valves. According to the author, it was very difficult to completely eliminate the air from the circuit. The gas release effect is considered a possible reason for the further oscillation damping noticed in the experimental results. To overcome the above problem, a proper mass balance equation involving gas release was taken into account. The calibration of the proposed model parameters was carried out with a micro-genetic algorithm. A different turbulence model [8] was also considered for comparison.

Both constant and variable gaseous mass for water hammer flow were taken into account. "Taking into account the mass of gas, considered as constant, reduces the MAE because it allows to phase the computed oscillations, approaching it to the observed one. If the mass of free gas is considered as a variable, taking into account a gas release and solution process, the oscillation damping is caught altogether, provided that a proper calibration of the parameters of the model is made" (contribution 4). Pezzinga noticed a significant improvement in the modeling of the head oscillation damping results from the 2D flow schematization with respect to the 1D one with quasi-steady friction and concluded that the oscillation damping observed in water hammer flow is mainly due to unsteady friction, but other mechanisms of dissipation exist; for example, the thermic exchange between bubbles and the surrounding liquid. These mechanisms can influence the values of the calibrated parameters, but they do not seem capable of fully reproducing the observed pressure traces. Finally, Pezzinga mentioned that more complex models of gas release could be considered in future studies.

Neyestanaki et al. (contribution 5) analyzed the literature in the direction of modeling different types of valve closure with a transient water hammer with the help of three-dimensional computational fluid dynamics (CFD). In their opinion, 3D models are more accurate than numerical simulation with use of 1D solutions. The performed literature survey showed that there is no study comparing the different methods for modeling valve closure in terms of modeling accuracy and computational cost. Moreover, a sliding mesh has not been used before for modeling axial valve closure. In their paper, water hammer in a straight 3D pipe (36 m long with relatively large inner diameter D = 0.3 m; Reynolds number Re = 7×10^5) during an axial gate valve closure was modeled using CFD. Three methods were used to model the valve closure: dynamic mesh, sliding (motion) mesh (Ansys Fluent), and immersed solid (Ansys CFX) methods were used for modeling the valve closure. The calculation results were compared with experimental results provided by Sundstrom and Cervantes [9] that include the variation in the differential pressures between two cross-sections and the wall shear stress. The performed result analysis showed that the immersed solid method has a delay in flow rate reduction and therefore underestimates differential pressure rise and overestimates wall shear stress close to the end of the valve closure. The dynamic mesh method models are more time-consuming and three times more expensive in terms of computational cost than other methods. Additionally, the dynamic mesh was unstable, with the possibility of divergence. The best results were received with help of the sliding mesh method, which is an inexpensive and stable technique that provides results closest to those of the experimental method. The authors proved that, for a thin gate valve, the axial movement of the valve can be modeled using mesh movement without any mesh deformation (predicting more physical results). The three-dimensionality of the flow after the valve closure was addressed as a non-symmetrical recirculation region that appears near the gate valve with its movement. The computational cost of the mentioned methods was discussed. "Immersed solid method and sliding mesh have quite the same computational cost. For dynamic mesh, the process of re-meshing is added to the calculation. Moreover, the lower quality of regenerated mesh makes it more possible to diverge. Therefore, a lower under-relaxation factor and, consequently, a higher number of iterations per time step is applied in the simulation. The mentioned drawbacks made the computational cost of the dynamic mesh method around three times more than the sliding mesh and immersed solid method".

Ferreira and Covas (contribution 6) rightly pointed out that, in any numerical model (1D, 2D, or 3D), the mesh size and configuration strongly affect the computational effort, as well as the accuracy of the results. The 2D mesh adaptation is very important for the geometrical boundaries. Flow physics needs ensure that the velocity variation is of the same order along the numerical mesh—a non-uniform grid should be generated (higher resolution near the pipe wall) due to the high gradients observed. The calibration of the grid (according to the velocity gradient history) can reduce the number of mesh points, maintaining the same accuracy level. An in-depth assessment of the 2D radial mesh's influence on the computation of unsteady energy dissipation in pressurized pipes was conducted. An extensive numerical analysis of the effect of the numerical schemes and of the radial mesh (several radial meshes were defined) on the computation of unsteady energy was carried out. The simulation results with use of 1D and 2D models were compared to the experimental method for laminar flows (Zielke solution) and for two valve closure maneuvers (i.e., an instantaneous and an S-shaped closure).

A new optimized equal area cylinder (OEAC) radial mesh (with 40 cylinders) was proposed using 2D simulations. It is defined by a high-resolution grid near the pipe wall and a lower-resolution grid in the pipe core. Its main advantage is achieving good model accuracy without increasing the computation effort. For an instantaneous valve closure scenario, this mesh reduced the calculation time by four times with a similar simulation error to that of the standard equal area cylinder (EAC) mesh. The comparisons of the results of the new mesh compared to those of the traditional mesh geometries (GS, ETC, and EAC) for a calibrated S-shaped valve closure revealed that (a) compared to the geometric sequence (GS) mesh, the new geometry achieved a significant improvement in accuracy for meshes with fewer cylinders (NC $\leq$ 60); (b) the other two radial meshes (ETC and EAC) do not provide the same accuracy; and (c) new mesh correctly describes the experimental data with only 20 cylinders. One of the notable advantages of the Ferreira and Covas approach is its adaptability to turbulent flows, achieved by incorporating a suitable turbulent model into the fundamental equations.

Urbanowicz et al. (contribution 7) presented an extensive review of two modeling techniques of unsteady friction related to IAB and CBM. The filtering method of weighting function used in CBM proposed in Urbanowicz's earlier paper [10] were further analyzed.

Its usefulness was checked for cases without vapor (cavitation) zones forming in the pipe. The second objective of this paper was the verification of the effectiveness of Johnston's lumped friction model [11], according to which the unsteady friction can be concentrated only at the boundary nodes of the numerical grid (in inner nodes the simplified quasi-steady assumption was used). The weighting function in this proposed computationally effective and accurate model was simply composed of two exponential terms, and its values are chosen to be dimensionless and time-step-dependent. The simplification of the weighting function in conjunction with the corrected effective method for solving the convolution integral enabled the proper determination of resistances. The proposed method's mathematical complexity is similar to that of the IAB model. Contrary to the IAB model, in the proposed method, there is no need to calibrate any parameters describing wall shear stress. The simulations carried out with the use of Johnston's model (lumping unsteady friction in boundary nodes of the MOC grid) showed that the analyzed transient histories of pressure can be simulated with sufficient compliance, maintaining acceptable simulation accuracy even more quickly.

Kubrak et al. (contribution 8) numerically (using a fixed-grid method of characteristics and IAB unsteady friction model) and experimentally studied the water hammer phenomenon in a serially connected steel–plastic pipeline system with significantly different inner diameters and for several different lengths of each section (maintaining a constant total length of the pipeline system). The experimental data revealed that the maximum pressure increase linearly depended on the share of the steel section in the total pipeline length. The combination of the IAB model applied to the steel section and the one-element Kelvin–Voigt model accounting for the viscoelastic properties of the HDPE pipe made it possible to obtain a satisfactory agreement between the calculated and the measured pressure signals.

Parameters defining the IAB model and the creep compliance of the HDPE plastic pipe were calibrated for single-pipeline systems. They were then introduced into a proposed numerical model to simulate the water hammer in different configurations of lengths of steel and HDPE sections. It was demonstrated that this approach failed to reproduce pressure head histories for a pipeline system with significantly different lengths of each section, as any change in the configuration of the pipeline system caused the creep parameters to be recalibrated.

Vardy (contribution 9), in his review paper, pointed out that during unsteady pipe flows, many different phenomena can occur simultaneously and interact with one another. Because of this, it is beneficial to have a clear understanding of the potential sources of damping (of pressure waves propagating in pipes) and their likely importance in any particular situation. It is common that two or more causes of damping exist simultaneously, but usually one of these is dominant and the others are of secondary importance.

In events forced by valve closure or pump trip-off, low-frequency components of waves are dominant. In studies of acoustics or leak detection in pipelines (where acoustic sensors are used) where waves are of a sufficiently high frequency (their wavelengths are shorter than a few pipe diameters), cross-sectional variations in pressure become significant (radial disturbances dominate axial ones). The most well-known cause of damping is skin friction (SF). Vardy explains popular rough assumptions in detail, according to which SF is approximated in a quasi-steady way. Notably, shear stresses are almost never measured directly except in highly specialized laboratory experiments designed expressly for this purpose. In practical applications, the SF influence is less strong than seen in the Holmboe experiment. Vardy suggests that there are three reasons for this: (a) the measurements were made in a pipe with a very small L/D ratio (time intervals between successive steps are short; L—pipe length); (b) the valve closure was as rapid as the experimenters could achieve; and (c) the Reynolds numbers of the initial steady flows were smaller than usual in large-scale engineering.

After analyzing the skin friction effect, the pipe wall properties are discussed. Here, typical steel pipes (and other metal pipes) are conventionally treated as linearly elastic, while in water supply systems, plastic pipes play the main role. The wall material of such pipes behaves in a viscoelastic manner, which physically means that the pipe diameter responds relatively slowly to changes in the fluid pressure (a delayed strain occurs). The influence of the assumed viscoelastic (VE) properties of the pipe wall are sometimes so dominant that little useful purpose would be served by taking account of unsteady components of skin friction. Vardy's comments about wave speed (WS) state that it would not be safe to infer the WS from the time intervals between the pressure histories maxima.

In the following sections, Vardy discusses the role of:

(a) Axial and lateral structural movement; he discusses its widely fluid structure interaction effect (FSI). In FSI, structural movement can cause a behavior that complicates attempts to make reliable inferences from the measurements of pressure alone. FSI's importance depends significantly on the interactions between internal pressure forces and forces due to structural movements. This effect can decrease the pressure amplitudes in some locations, but increases amplitudes in others.
(b) Variable wave speed. A small amount of gas—free (bubbles of undissolved gas) or vapor (as an effect of cavitation)—can cause changes in the WS, which also causes dispersive behavior and, hence, influences damping (either increasing or countering it).
(c) Porous surfaces. "Strong damping can exist when fluid can discharge laterally through pipe walls. It will be rare for this to be desirable in the case of liquid flows, but it can be beneficial in some gas flows" (sonic boom-like disturbances occurring in tunnels exit portals).
(d) Delayed reflections. "Disturbances propagating from the pipe into the reservoir radiate in a spherical-like manner whereas the reflections along the pipe approximate closely to planar. The time required for the changes in pressure at the outlet to die away is short—typically in the order of the time required for a wave to travel one pipe diameter—so the phenomenon is justifiably neglected in many practical applications".
(e) Experimental measurements. Wave superpositions can have a strong influence on pressure histories at any particular location and can complicate the interpretation of the physical measurements exhibiting damping. The author also discussed problems with possible unidentified oscillations.
(f) Numerical damping, which can be caused by interpolation algorithms or by incorrect estimations of flux across interfaces between adjacent cells. It can also be introduced intentionally for special reasons; for instance, through the use of "artificial viscosity" to suppress unrealistic oscillations close to locations of especially rapid change.

Lu et al. (contribution 10)'s paper concerned problems related to modeling the load-rejection process of a hydropower plant with an air cushion surge chamber. The main aim of this paper was to develop an accurate and efficient water hammer numerical model, which is significant for the proper design and safe operation of hydropower plants. The authors worked on a second-order finite volume method (FVM) based on a Godunov-type scheme (GTS) to solve the main set of equations, motivated by the fact that with its help, results would be more accurate and more stable with less numerical dissipation than in MOC. This FVM-GTS model was validated earlier in simple scenarios. In their paper, hydraulic transients of the load-rejection process occurring in the hydropower plant with an air cushion surge chamber (rarely analyzed previously) were studied. "The results calculated by the proposed second-order FVM GTS models were compared with the exact solution and the measured values as well as predictions by the MOC scheme. The accuracy and efficiency of the proposed approach were discussed. Another important purpose is that the proposed accurate model was used to explore the possible computation error caused by the MOC scheme in a hydropower plant with a complex pipe system". As sometimes complicated boundaries need to be taken into account, virtual boundaries were introduced (upstream, downstream, and hydraulic component connection sections) to provide a connection between the air chamber and the unit to achieve uniformity in the calculation of the control cells inside the pipeline and at the boundaries.

The performed numerical comparisons revealed that for a Courant number Cr = 1, both models, FVM and MOC, were consistent with the exact solution. When Cr < 1, both computational results had numerical dissipation. For a Cr number with a gradual decrease, the second-order FVM simulation results were more stable as the numerical dissipation of MOC was more serious. For complex pipe systems, the proposed FVM model better reproduced the experimental data, which were more accurate than those of MOC. For the load-rejection process of hydropower units containing an air chamber, the results calculated using the proposed FVM model were basically consistent with the measured rotational speed variation. The second-order FVM does not need to adjust the wave speed for the pipes; it only needs to reduce the Cr condition appropriately. The error in the MOC calculation was associated with the air chamber parameters. The second-order FVM is robust in simulating the water hammer problems in a simple or complex pipe system. "Considering the higher accuracy, stability, and efficiency, the high-order FVM is feasible and suggested for water hammer simulation in real hydraulic systems with more complicated pipe components and devices".

Paternina-Verona et al. (contribution 11) developed a study in which they explored various issues related to transient flows during different filling events through the use of 3D computational fluid dynamics (CFD) models. The main findings and issues addressed in the study are as follows:

Gauge Pressure Oscillations: The 3D CFD models accurately predicted gauge pressure oscillations in the air pocket during filling events. The entrapped air pocket experienced cyclic changes in volume, and this pressure variation had a significant impact on the hydraulic behavior.

Water flow velocities: The study examined water flow velocities during transient events. The authors observed that the pressure damping of the entrapped air pocket led to water flow velocity transitions, with higher inlet gauge pressures resulting in more intense mixing of air and water.

Thermodynamic phenomena: Temperature changes occurred during the filling processes due to the compression of the trapped air pocket. The study found non-uniform temperature distributions, with the highest temperatures away from the air–water interface. This adiabatic behavior in the air pocket can affect system efficiency and safety.

Backflows and hydraulic efficiency: Transient flows can lead to backflows towards the pumping source, potentially causing a loss of hydraulic efficiency during filling events. The study showed that these backflows were more critical with higher inlet gauge pressures and gradually dissipated over time due to the damping pressure of the entrapped air pocket.

Detailed visualization: The use of 3D CFD models with an unstructured mesh allowed the detailed visualization of various variables, including streamlines, velocity contours, and temperature distributions, providing a comprehensive understanding of hydraulic–thermodynamic phenomena.

Selection of pipe class: The study highlighted the importance of evaluating the selection of pipe class in large-scale water installations, as the compression of entrapped air pockets can result in absolute pressure values higher than those observed with monophasic fluids. Current regulations often do not consider such two-phase flow models when designing systems, and the use of 3D CFD models can improve system reliability during filling operations.

The study suggests directions for future research, including the investigation of the three-dimensional behavior of hydraulic events with air expulsion orifices, which has not been extensively studied using 3D CFD models. Additionally, the simulation of large-scale hydraulic scenarios, with and without air expulsion orifices, is recommended for further exploration. In summary, this study delves into the complex interplay of hydraulic and thermodynamic phenomena during filling events in pipes with entrapped air pockets. It emphasizes the utility of 3D CFD models for understanding and predicting these phenomena and suggests potential improvements and areas for future research.

The investigation presented by Wu et al. (contribution 12) employed the integer total energy method to formulate the energy equation within a quasi-2D model representing viscoelastic pipes. Subsequently, a comparative analysis of work variations pertaining to the friction term and viscoelastic term in both 1D and 2D models was conducted across different initial Reynolds numbers (Re). The key findings are as follows: (i) When the initial Reynolds number was less than 3.0×10^5, the 1D model exhibited a tendency to underestimate the work associated with the friction term, a trend not observed in the 2D model. This discrepancy lessened as the initial Re exceeded 3.0×10^5. (ii) The work performed with the viscoelastic term of the pipe wall demonstrated a relative constancy across varying initial Reynolds numbers in both the 1D and 2D models. (iii) As the Reynolds numbers increased, both the viscoelastic and the frictional work within the 1D and 2D models showed a progressive increase over time. However, for cases with a high initial Re, the work due to friction surpassed that of the viscoelastic term. (iv) The dissipation of energy related to the friction term displayed an initial significant increase, followed by a gradual deceleration, eventually reaching a constant value. (v) The energy transformation associated with the viscoelastic term exhibited sinusoidal fluctuations during the initial phase of transient flow. In cases with lower initial Reynolds numbers, these oscillations persisted for an extended duration, portraying an overall upward trend before converging to a steady state. (vi) With an escalation in initial Reynolds numbers, the proportion of energy dissipation attributed to the friction term within the total energy transformation exhibited continuous growth, while the proportion originating from the viscoelastic term decreased proportionally.

Zeng et al. (contribution 13) presented hydraulic systems encompassing both pipelines and open channels, posing a challenge for hydraulic transient analysis. This study introduced a novel coupling method that combines the method of characteristics (MOC) for pipeline modeling and the finite volume method (FVM) for open-channel modeling. The interface between these two simulation domains was established using Riemann invariants. Parameters were exchanged between the MOC and FVM regions through the coupling boundaries in both directions. To validate the method, the authors developed tests on a simple tank-pipe system, and the results were compared with those obtained through 3D computational fluid dynamics (CFD) analysis. Subsequently, the method was applied to a practical hydropower station featuring a sand basin situated between the upstream reservoir and the turbines. The sand basin was treated as an open channel, integrated with the pipes within the system, and transient processes were simulated by modeling the sand basin as a surge tank. Comparing the results obtained through the new coupling method with those from the MOC-FVM coupling method, it became evident that the new approach offers increased reliability and accuracy. This improved performance stems from considering horizontal flow velocity in the sand basin, a factor that was previously neglected when modeling the sand basin as a surge tank within the MOC framework.

Cao et al. (contribution 14) enhanced hydraulic transients with significant operational risk due to the potentially destructive surge waves they generate. This study delved into the analysis of hydraulic surge phenomena and the control of surge damping in the context of pipe flow modeling and valve optimization. The study involved the development of a one-dimensional transient model using the modified instantaneous-accelerations-based (IAB) model, which incorporates considerations of energy dissipation, specifically the compression–expansion effect. This model was subsequently solved through the method of characteristics (MOC).

In a manner analogous to addressing valve operations via the traveling salesman problem (TSP), an innovative surge damping strategy was proposed, leveraging an improved artificial fish swarm algorithm (AFSA). To validate the unsteady model and the optimization algorithm, the effectiveness of surge wave damping was evaluated through case studies encompassing various pump operation scenarios.

The results revealed that the nonlinear optimized control method, as presented, is capable of reducing surge amplitudes by 9.3% and 11.4% in pipe systems, with and without a centrifugal pump in operation, respectively. Moreover, the method can yield a substantial 34% increase in time margin or a maximum surge reduction of 75.2% when employing a positive displacement pump. The optimized nonlinear valve closure exhibits distinct profiles in scenarios involving both rapid and gradual closure.

The surge damping strategy presented in this study not only holds practical implications for guiding real-time valve control but also provides valuable insights for valve design geared toward safeguarding against wave surge occurrences.

Liu et al. (contribution 15) presented a study to assess the sensitivity of input parameters concerning output results in hydraulic transient simulations employing the method of characteristics (MOC). In the context of a gravity flow water delivery project, six primary parameters influencing hydraulic transient simulations were identified. The study focused on the maximum pressure as the output parameter to conduct a sensitivity analysis. Two distinct approaches, namely Morris sensitivity analysis (Morris) and the partial rank correlation coefficient method based on Latin Hypercube Sampling (LHS-PRCC), were employed for this analysis. The findings indicated that the sensitivity of each parameter was generally consistent, with the exception of the friction factor. Specifically, the flow rate and Young's modulus exhibited a positive correlation with the maximum pressure, whereas the pipe diameter, valve closing time, and wall thickness demonstrated a negative correlation. It was noted that the variability in the friction factor was primarily attributed to the functioning of the flow and pressure regulating valve. In situations where other conditions within the gravity flow project remained constant, an increase in the friction factor corresponded to an elevation in maximum pressure. Significantly, the flow rate, pipe diameter, and valve closing time emerged as the critical parameters influencing the model. Moreover, the study underscored that both the Morris and LHS-PRCC methods proved effective in assessing parameter sensitivity in hydraulic transient simulations.

In conclusion, this study focused on analyzing parameter sensitivity in hydraulic transient simulations employing the method of characteristics (MOC) within the context of gravity flow. The examination was conducted using a single engineering case, demonstrating the effectiveness of the two sensitivity analysis methods. To broaden the scope and achieve a more comprehensive understanding, further analyses involving additional examples are warranted.

3. Conclusions

The papers included in the *Water* Journal's Special Issue titled "About an Important Phenomenon—Water Hammer" provide a contemporary snapshot of research trends in this field. All fourteen research papers in this collection involve a comparative analysis between simulation results and experimental findings. Among these papers, the method of characteristics (MOC) was employed in eight instances to model unsteady flow, while two papers utilized the second-order finite volume method (FVM). Additionally, one paper introduced a coupled approach combining MOC and FVM, which was also found applicable for open-channel flows. Notably, one paper presented a frequency-domain solution, while two other works relied on computational fluid dynamics software, specifically Ansys and OpenFoam. This reaffirms the ongoing significance of the MOC in the numerical analysis of water hammer problems.

In all the research papers, the transient water hammer flow was induced through valve closures (both hypothetical–instantaneous and realistic valve closing performance) and/or by deactivating the pump power, mirroring the conditions often encountered in real-world operational systems. In the majority of these papers (eight in total), the instantaneous-accelerations-based (IAB) unsteady friction model was employed by the authors to simulate friction losses using a 1D modeling approach. This preference suggests a desire for friction models that are relatively straightforward in their application. An interesting model meeting these expectations (contribution 7) focuses on unsteady friction mainly at the boundary nodes of the numerical mesh, based on the assumption proposed by Johnstone [11], and utilizes a convolution-based model (CBM) with a weighting function relying on only two exponential terms. The future, however, is likely to belong to meshless methods [12].

The majority of the comparative analyses (twelve in total) centered on elastic (metal) pipelines, while only three papers extended their simulations to scenarios found in contemporary water supply systems employing plastic pipes with viscoelastic wall behavior characterized by delayed strain.

Returning to notable observations made by Tasca et al. and Vardy, Tasca et al. (contribution 1) aptly emphasized the inevitability of real-world systems deviating from numerical idealizations due to unforeseeable future conditions. As such, they stressed the importance of design and operational engineers assessing the potential impact of real-world variability in system parameters on system performance. Vardy (contribution 9) highlighted the positive influence of damping in the context of water hammer in pipelines. Damping can facilitate the dissipation of strong disturbances before subsequent disturbances arrive, ultimately mitigating the potential consequences of superpositions.

The ongoing relevance of water hammer, particularly with the consideration of accompanying phenomena such as cavitation, unsteady friction, delayed strain, and fluid–structure interaction, underscores the continuous need for advancements in modeling methods. However, it is essential to exercise caution during analyses, as dissipative phenomena typically induce damping, and dispersive phenomena often do the same, though exceptions can arise where energy redistributions lead to superpositions that would not occur otherwise (contribution 9).

Author Contributions: Conceptualization, K.U. and H.M.R.; investigation, K.U. and H.M.R.; writing—original draft preparation, K.U. and H.M.R.; writing—review and editing, K.U. and H.M.R.; All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

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

List of Contributions:

1. Tasca, E.; Besharat, M.; Ramos, H.M.; Luvizotto, E., Jr.; Karney, B. Exploring the Sensitivity of the Transient Response following Power Failure to Air Valve and Pipeline Characteristics. *Water* **2023**, *15*, 3476. https://doi.org/10.3390/w15193476
2. Kim, S. Dimensionless Pressure Response Analysis for Water Supply Pipeline Systems with or without Pumping Station. *Water* **2023**, *15*, 2934. https://doi.org/10.3390/w15162934
3. Liu, Y.; Lu, J.; Chen, J.; Xia, Y.; Liu, D.; Hu, Y.; Feng, R.; Liu, D.; Zhou, L. Finite Volume Method for Transient Pipe Flow with an Air Cushion Surge Chamber Considering Unsteady Friction and Experimental Validation. *Water* **2023**, *15*, 2742. https://doi.org/10.3390/w15152742
4. Pezzinga, G. Gas Release and Solution as Possible Mechanism of Oscillation Damping in Water Hammer Flow. *Water* **2023**, *15*, 1942. https://doi.org/10.3390/w15101942
5. 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. https://doi.org/10.3390/w15081510
6. Ferreira, P.L.; Covas, D.I.C. New Optimized Equal-Area Mesh Used in Axisymmetric Models for Laminar Transient Flows. *Water* **2023**, *15*, 1402. https://doi.org/10.3390/w15071402
7. 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. https://doi.org/10.3390/w14193151
8. Kubrak, M.; Kodura, A.; Malesińska, A.; Urbanowicz, K. Water Hammer in Steel–Plastic Pipes Connected in Series. *Water* **2022**, *14*, 3107. https://doi.org/10.3390/w14193107
9. Vardy, A.E. On Sources of Damping in Water-Hammer. *Water* **2023**, *15*, 385. https://doi.org/10.3390/w15030385
10. Lu, J.; Wu, G.; Zhou, L.; Wu, J. Finite Volume Method for Modeling the Load-Rejection Process of a Hydropower Plant with an Air Cushion Surge Chamber. *Water* **2023**, *15*, 682. https://doi.org/10.3390/w15040682

11. Paternina-Verona, D.A.; Coronado-Hernández, O.E.; Espinoza-Román, H.G.; Fuertes-Miquel, V.S.; Ramos, H.M. Rapid Filling Analysis with an Entrapped Air Pocket in Water Pipelines Using a 3D CFD Model. *Water* **2023**, *15*, 834. https://doi.org/10.3390/w15050834
12. Wu, K.; Feng, Y.; Xu, Y.; Liang, H.; Liu, G. Energy Analysis of a Quasi-Two-Dimensional Friction Model for Simulation of Transient Flows in Viscoelastic Pipes. *Water* **2022**, *14*, 3258. https://doi.org/10.3390/w14203258
13. Zeng, W.; Wang, C.; Yang, J. Hydraulic Transient Simulation of Pipeline-Open Channel Coupling Systems and Its Applications in Hydropower Stations. *Water* **2022**, *14*, 2897. https://doi.org/10.3390/w14182897
14. Cao, Z.; Xia, Q.; Guo, X.; Lu, L.; Deng, J. A Novel Surge Damping Method for Hydraulic Transients with Operating Pump Using an Optimized Valve Control Strategy. *Water* **2022**, *14*, 1576. https://doi.org/10.3390/w14101576
15. Liu, J.; Wu, J.; Zhang, Y.; Wu, X. Sensitivity Analysis of Hydraulic Transient Simulations Based on the MOC in the Gravity Flow. *Water* **2021**, *13*, 3464. https://doi.org/10.3390/w13233464

References

1. Pollio, V. *The Ten Books on Architecture*; Morgan, M.H., Translator; Harvard University Press: Cambridge, MA, USA, 1914; Book 8, Chapter 6, Sections 5–8, pp. 245–246.
2. Tijsseling, A.S.; Anderson, A. A precursor in water hammer analysis—Rediscovering Johannes von Kries. In Proceedings of the 9th International Conference on Pressure Surges, Chester, UK, 24–26 March 2004; pp. 739–751.
3. Tijsseling, A.S.; Anderson, A. Johannes von Kries and the history of water hammer. *J. Hydraul. Eng.* **2007**, *133*, 1–8. [CrossRef]
4. Tijsseling, A.S.; Anderson, A. *The Joukowsky Equation for Fluids and Solids*; CASA Reports; Department of Mathematics and Computer Science, Eindhoven University of Technology: Eindhoven, The Netherlands, 2006; pp. 1–11.
5. Allievi, L. General theory of the perturbed motion of water in pipes under pressure (water hammer). *Ann. Della Soc. Degli Ing. Ed Archit. Ital. (Ann. Soc. Ital. Eng. Archit.)* **1902**, *17*, 285–325.
6. Pandey, M.; Winkler, D.; Vereide, K.; Sharma, R.; Lie, B. Mechanistic Model of an Air Cushion Surge Tank for Hydro Power Plants. *Energies* **2022**, *15*, 2824. [CrossRef]
7. Xu, T.; Chen, S.; Zhang, J.; Yu, X.; Lyu, J.; Yan, H. Comparison on Hydraulic Characteristics of Vertical and Horizontal Air-Cushion Surge Chambers in the Hydropower Station under Load Disturbances. *Energies* **2023**, *16*, 1501. [CrossRef]
8. Lam, C.K.G.; Bremhorst, K.A. Modified Form of the k-ε Model for Predicting Wall Turbulence. *J. Fluids Eng.* **1981**, *103*, 456–460. [CrossRef]
9. Sundstrom, L.R.J.; Cervantes, M.J. Transient Wall Shear Stress Measurements and Estimates at High Reynolds Numbers. *Flow Meas. Instrum.* **2017**, *58*, 112–119. [CrossRef]
10. Urbanowicz, K. Fast and accurate modelling of frictional transient pipe flow. *Z. Angew. Math. Mech.* **2018**, *98*, 802–823. [CrossRef]
11. Johnston, D.N. Efficient methods for numerical modelling of laminar friction in fluid lines. *J. Dyn. Syst. Meas. Control* **2006**, *128*, 829–834. [CrossRef]
12. Xu, Y.; Deng, Y.; Jiao, Z. Fast Meshless Solution with Lumped Friction for Laminar Fluid Transients. *J. Press. Vessel Technol.* **2023**, *145*, 061401. [CrossRef]