

Article

The Influence of Internal Geometry on Pressure Losses, Hydraulic Stability, and Cavitation Risk in a Firefighting Monitor

Michał Zielina ^{1,*} , Bartosz Kopiczak ² , Artur Cebula ¹  and Mehmet Yildirim ¹ 

¹ Faculty of Environmental Engineering and Energy, Cracow University of Technology, 24 Warszawska Street, 31-155 Krakow, Poland; artur.cebula@pk.edu.pl (A.C.); mehmetaliyldirmm@gmail.com (M.Y.)

² Faculty of Mechanical Engineering, Cracow University of Technology, Jana Pawła II 37 Street, 31-864 Krakow, Poland; bartosz.kopiczak@pk.edu.pl

* Correspondence: michal.zielina@pk.edu.pl; Tel.: +48-12-628-21-86

Abstract

This paper presents a combined CFD and experimental investigation of water flow through a Shootfire-1000 water-foam fire monitor. The study aimed to assess pressure losses, flow characteristics, hydraulic stability, and cavitation risk, and to evaluate the influence of selected geometric and operating parameters on hydraulic performance. A numerical model based on the actual geometry of the device was developed and experimentally validated using pressure loss measurements obtained under representative operating conditions. The influence of flow rate, monitor elevation angle, water temperature, nozzle cone position, and cone plate fillet radius was analyzed. The results showed that the dominant pressure losses occur within the monitor head, particularly in the annular constriction formed by the nozzle cone and the housing. Water temperature and monitor elevation angle had only a minor effect on the overall hydraulic performance, whereas relatively small modifications of the nozzle cone geometry significantly affected pressure losses. The analyses further demonstrated that local geometric features play a key role in determining velocity distribution and hydraulic stability. No pressure values below the water vapour pressure were observed within the investigated operating range, indicating a negligible risk of cavitation. The results confirm that optimization of the monitor head geometry can effectively reduce hydraulic losses and improve the hydraulic performance of water-foam fire monitors.

Keywords: firefighting monitor; internal flow; pressure losses; hydraulic stability; cavitation risk; computational fluid dynamics (CFD); pressure distribution; hydraulic performance



Academic Editor: Gionata Fragomeni

Received: 17 July 2026

Revised: 20 August 2026

Accepted: 31 August 2026

Published: 2 September 2026

Copyright: © 2026 by the authors.

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.

1. Introduction

Fire monitors are key components of fire protection systems, enabling large water streams to be delivered over considerable distances and heights. Their operational effectiveness depends directly on hydraulic parameters such as pressure losses, flow capacity and flow stability, as well as on undesirable phenomena such as cavitation. Cavitation, which results from local pressure drops below the vapour pressure of the liquid, leads to the formation of vapour bubbles which, upon collapse, may cause erosion of structural components and degradation of the flow characteristics. The development and collapse of vapour bubbles are governed by complex interactions among pressure fluctuations, turbulence, and local flow structures. Bubble collapse may generate high-pressure microjets and shock waves, leading to material erosion and the deterioration of hydraulic performance. These

mechanisms have been extensively investigated in fundamental cavitation studies and hydraulic machinery applications [1–3]. Recent investigations of cavitating flows in nozzles and injectors have further demonstrated that cavitation behaviour strongly depends on local flow acceleration, pressure gradients, and nozzle geometry. Hu et al. [4] showed that changes in nozzle orifice geometry significantly affect internal flow structures and cavitation intensity in gasoline direct injection injectors. Similarly, Zhu et al. [5] reported that geometric modifications of bionic nozzles influence both cavitation development and flow performance, highlighting the key role of internal geometry in cavitation control. Moreover, Li et al. [6] demonstrated that cavitation bubble collapse may generate severe erosion effects in submerged water jets, emphasizing the importance of identifying cavitation-prone regions in hydraulic devices. This phenomenon is a well-documented problem in fire pumps and hydraulic systems; however, the literature concerning its occurrence in fire monitors remains limited. Although numerous studies have investigated cavitation mechanisms in nozzles, injectors, pumps, and water jets, comparatively few publications have addressed cavitation risk in firefighting monitors and their internal flow passages.

Recent studies have shown that both fire pumps and nozzle systems may be exposed to significant pressure fluctuations and cavitation phenomena under off-design operating conditions. For example, Hao et al. [7] demonstrated a significant increase in cavitation risk at increased flow rates, as well as fluctuations in radial forces in fire pumps, indicating the importance of stable hydraulic operating conditions. Similar observations were reported by Liu et al. [8], who determined critical values of net positive suction head (NPSH) and described the nature of pressure pulsations in fire pumps, emphasizing the direct relationship between hydraulic losses and cavitation inception.

In fire monitors and their nozzles, hydraulic performance is strongly governed by the internal geometry, including local flow constrictions, contraction and expansion angles, and changes in flow direction, all of which affect pressure losses and cavitation risk.

Studies on fire monitors clearly indicate that geometry modifications can lead to significant improvements in the hydraulic performance of the device. Kalender and Soyhan [9] showed that optimization of the internal geometry of a fire monitor using Computational Fluid Dynamics (CFD) methods can improve the velocity distribution and reduce hydraulic losses. In turn, Zhang et al. [10], analyzing fire monitors used in ultra-high-voltage (UHV) fire safety systems, demonstrated that properly designed flow straightening elements can reduce turbulent kinetic energy, although excessive use of such elements may increase pressure losses.

Although previous studies have demonstrated the importance of internal geometry for the hydraulic performance of fire monitors and nozzle systems, they have primarily focused on flow optimization, velocity distribution, and turbulence reduction. Experimental validation and cavitation assessment have received considerably less attention. Furthermore, the influence of specific geometric parameters, including nozzle cone position, cone plate geometry, monitor elevation angle, and water temperature, on both hydraulic losses and cavitation risk in commercially available water-foam fire monitors remains insufficiently documented. Consequently, there is still a need for experimentally validated studies addressing the combined effects of geometric and operating parameters on pressure losses, hydraulic stability, and cavitation susceptibility. To the authors' knowledge, such a combined assessment has not been reported for a commercially available Shootfire-1000 water-foam fire monitor. The novelty of the present study lies in the combined experimental validation and CFD-based assessment of pressure losses, hydraulic stability, and cavitation risk in a commercially available Shootfire-1000 water-foam fire monitor, together with a parametric evaluation of selected geometric and operating factors affecting its hydraulic performance.

The influence of geometric configuration on hydraulic performance has been extensively documented in the literature. Both experimental studies and CFD simulations have demonstrated that relatively small geometric modifications may considerably affect pressure losses, velocity fields, turbulence intensity, flow separation, and hydraulic efficiency. Similar observations have been reported for a variety of hydraulic devices, including intake structures, pumps, pipe fittings, and control valves, indicating that geometry plays a dominant role in determining flow behaviour and energy losses within fluid-flow systems [11–14].

Valuable insights are also provided by studies on the geometric parameters of nozzles. Zhang et al. [15] showed that increasing the contraction angle above 25° or the expansion angle above 30° leads to deterioration of velocity uniformity and an increase in pressure losses and turbulent kinetic energy, whereas an appropriate length of the straight section enables jet stabilization and loss reduction.

Recent CFD studies have demonstrated that cavitation inception and development are strongly influenced by turbulence structures, pressure gradients, and flow geometry. Investigations of injectors, nozzles, hydraulic pumps, and water jets have shown that relatively small geometric modifications may substantially affect cavitation intensity, vapour distribution, and cavitation-induced erosion [16–18]. These findings highlight the importance of high-fidelity numerical modelling for predicting cavitation-prone regions and assessing cavitation risk in hydraulic devices.

The literature also widely describes numerical methods used to model cavitation phenomena in nozzles and orifices, which are hydraulically similar to the conditions occurring in fire monitors. Mixture flow-based approaches and Schnerr–Sauer cavitation models are commonly used in CFD simulations [2,5,19,20].

The objective of this study was to perform an experimentally validated CFD investigation of a commercially available Shootfire-1000 water-foam fire monitor in order to assess pressure losses, hydraulic stability, and cavitation risk under representative operating conditions. Particular attention was devoted to identifying the dominant sources of hydraulic losses and evaluating the influence of selected geometric parameters, including nozzle cone position and cone plate edge geometry, on the hydraulic performance of the monitor. Unlike previous studies focused primarily on flow optimization and velocity-field improvement, the present work combines experimental validation with a parametric assessment of pressure losses, cavitation susceptibility, and design modifications in a real industrial device.

2. Materials and Methods

2.1. Fire Monitor and Geometric Reconstruction

The subject of the study was the Shootfire-1000 USGPM UL-Listed Foam Monitor manufactured by Vimal Fire Controls PVT LTD (Mumbai, India). The device is intended for use in fixed fire protection systems and is capable of discharging both water and low-expansion fire-fighting foam. The monitor is made of an aluminum alloy. Its operating pressure range is 7–10.5 bar, and the maximum flow rate declared by the manufacturer is 3785 L/min (1000 USGPM). The monitor provides continuous 360° horizontal rotation and allows for vertical elevation adjustment within a range of -52° to $+90^\circ$.

The general structure of the device is shown in Figure 1. The main components include the connection flange (1), rotation mechanisms (2, 3), stream pattern regulator (4), foam agent inlet (5), outlet nozzle (6), and flow rate control cone plate (7). The hydraulic operation of the device consists of pressurized water flowing through the monitor body and then through the guide system and nozzle, where either a solid or dispersed stream is formed. The head contains a foam pipe and a system of guide vanes, which enable the foam agent

to be entrained and mixed with water according to the Venturi effect. Stream regulation is achieved by changing the position of the cone plate (7) and rotating the regulating cover (4), as shown in Figure 1. From the perspective of flow analysis, the geometry of the head is of key importance, since this is the region where the stream is formed and where local pressure drops promoting cavitation may occur.

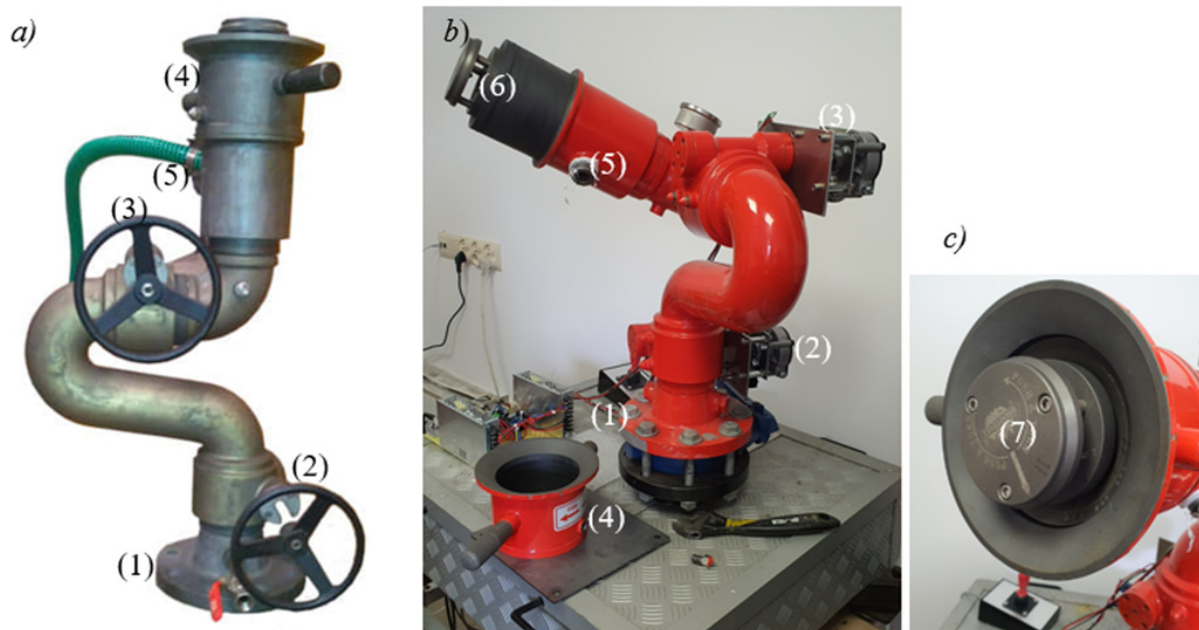


Figure 1. Shootfire-1000 USGPM UL-Listed Foam Monitor manufactured by Vimal Fire Controls PVT LTD: (a) assembled fire monitor, (b) monitor during disassembly, and (c) nozzle with the regulating cover.

In order to obtain the data required to develop the digital model, a detailed geometric inventory of the actual device was carried out. This was necessary due to the lack of available technical documentation that would allow the internal geometry to be directly reconstructed. All measurements were performed on a physical unit of the fire monitor under laboratory conditions.

The inventory included an analysis of the external structure of the device, dimensional measurements of components accessible after disassembly of the head, and measurements of internal components using video endoscopic inspection. Particular attention was paid to the elements having a significant influence on the flow conditions, including the foam pipe and guide vanes, shown in Figure 2a. Based on the measurements, a dimensioned cross-section of the head was prepared, as shown in Figure 2b.

From the perspective of the subsequent hydraulic analyses, precise reconstruction of the nozzle cone geometry was particularly important, as this element has a significant influence on the minimum flow cross-section, local jet acceleration, pressure drop, and the possible occurrence of low-pressure regions promoting cavitation. Based on geometric inventory, a detailed 3D CAD model of the fire monitor was developed. The model reproduced the geometry of the pipe section and the monitor head, including the foam pipe, guide vanes, cone plate, nozzle cone, and regulating cover (Figure 3).

2.2. Experimental Validation Setup

Experimental measurements used for CFD model validation were performed under field conditions at a dedicated outdoor test range due to the considerable throw distance of the water stream generated by the fire monitor. The tests were conducted using a

Rosenbauer SPA 100 portable fire pump equipped with the Logic Control System (LCS) (Rosenbauer International AG, Leonding, Austria).

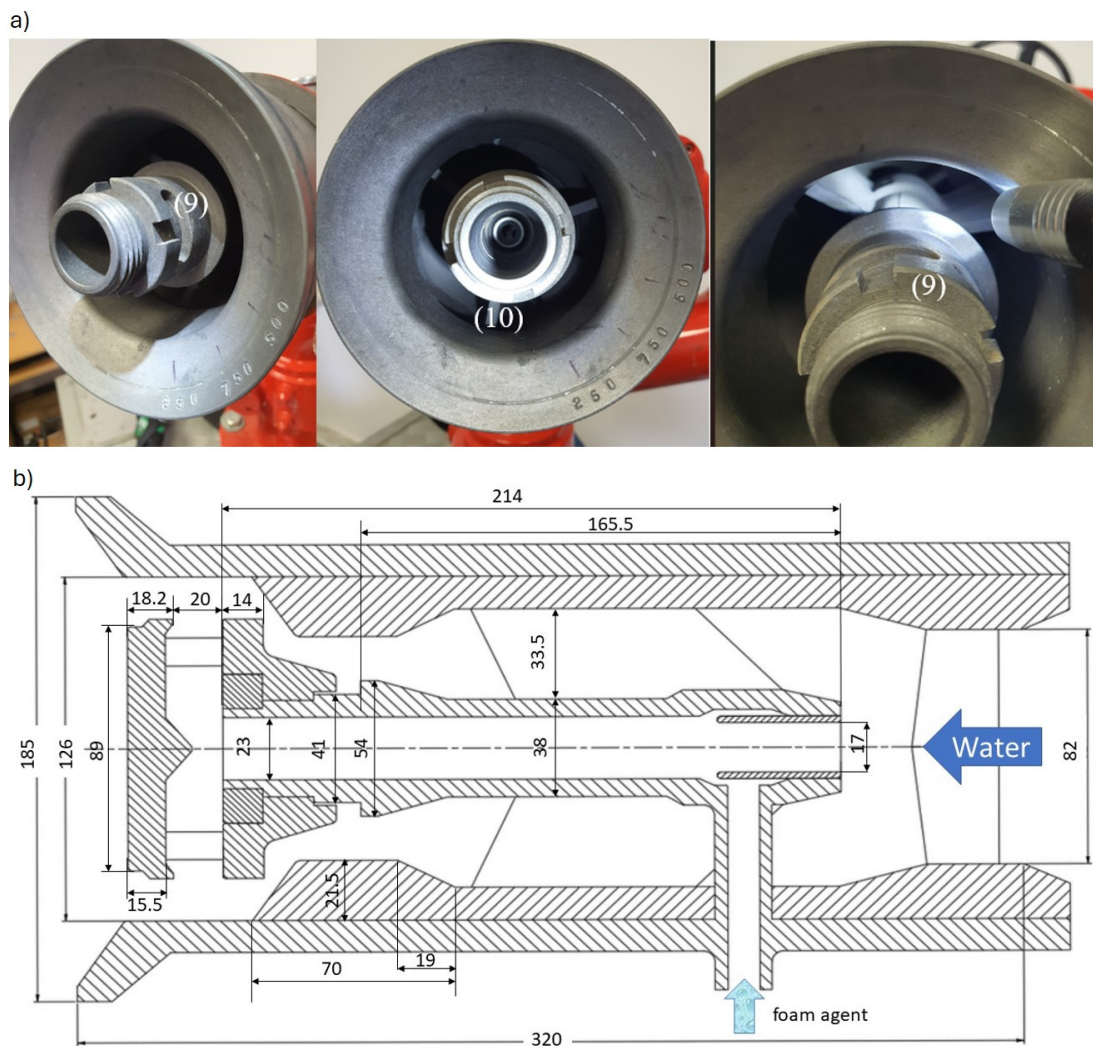


Figure 2. Monitor head: (a) head components, and (b) cross-section of head showing main elements: flow rate control cone plate (7), nozzle cone (8), foam pipe (9), and guide vanes (10).

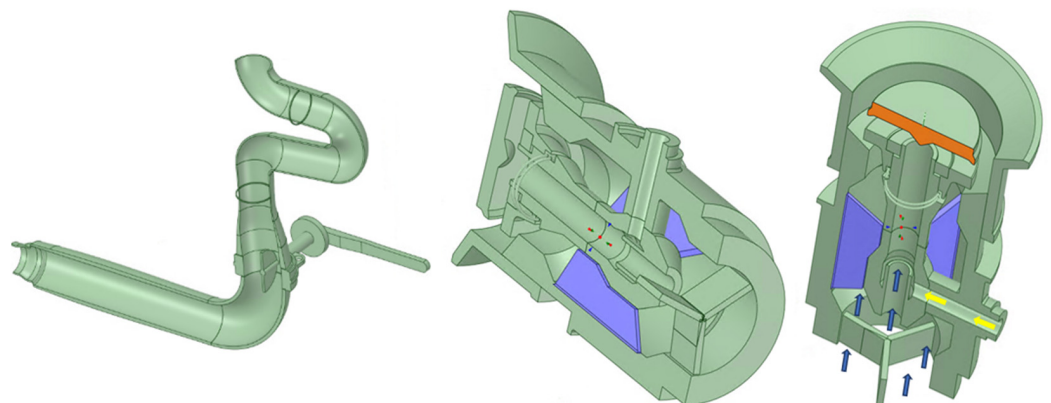


Figure 3. Three-dimensional CAD model of fire monitor incorporating components identified in Figures 1 and 2. Blue arrows indicate the water-flow direction, while yellow arrows indicate the foam-agent inlet direction. The guide vanes are highlighted in blue for clarity.

The flow rate was measured using an integrated electromagnetic flowmeter. An uncertainty of $\pm 1\%$ of the measured value was assumed for the flow rate measurements.

Pressure measurements were performed using a glycerine-filled pressure gauge with a measuring range of 0–15 bar and an accuracy class of 2.5, corresponding to an uncertainty of ± 37.5 kPa.

The experimental measurements were carried out under stabilized operating conditions corresponding to constant flow rates and fixed geometric configurations. The obtained pressure drop data were subsequently used for validation of the numerical model.

2.3. Computational Domain and Mesh Generation

After the solid CAD model had been developed, the flow domain representing the space occupied by water was generated. The model included the actual inlet surface with a diameter of 68 mm and an extended outlet surface with a diameter of 174 mm, located 350 mm downstream of the actual outlet cross-section. The outlet surface was shifted downstream in order to reduce the influence of the outlet boundary condition on the velocity and pressure distributions in the monitor head region (Figure 4).

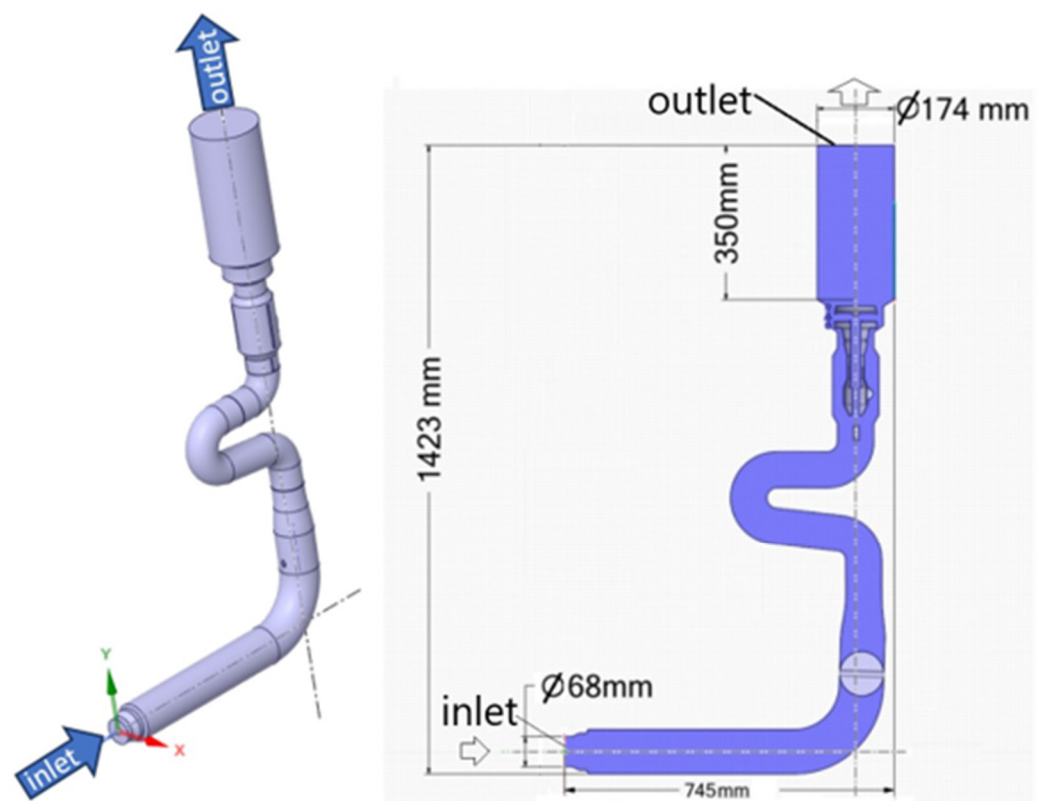


Figure 4. Geometry of computational domain.

A computational mesh was generated for the prepared flow domain. An unstructured tetrahedral mesh was used in the domain volume, supplemented with near-wall prism layers along the solid boundaries. The calculations were initiated using a coarse mesh consisting of approximately 16.1 million elements. After a stable and converged computational model had been obtained, the mesh was successively refined using a refinement factor of 1.2. Due to the three-dimensional and geometrically complex nature of the computational domain, this resulted in a substantially larger increase in the total number of elements. Consequently, the medium and fine meshes contained approximately 24.0 and 44.8 million elements, respectively. The total pressure drop was used as the main comparison criterion. The results of the grid independence analysis are summarized in Table 1.

As shown in Table 1, the difference between the medium and fine meshes was only approximately 1.79%, while the number of elements increased by about 87%. Therefore, the

medium mesh consisting of approximately 24 million elements was selected for the final simulations as a reasonable compromise between numerical accuracy and computational cost. After the initial mesh generation, additional mesh-quality improvement procedures were applied in order to obtain element-quality parameters within acceptable ranges. In particular, iterative mesh smoothing was performed to reduce excessive element skewness and improve orthogonality, with special attention paid to the geometrically complex regions of the monitor head including the cone plate edge and the narrow gap between the nozzle cone and the head housing, where strong velocity and pressure gradients were expected. The mesh quality was subsequently evaluated using the skewness and orthogonal quality metrics. For the final computational mesh, the average skewness was approximately 0.23, while the average orthogonal quality was approximately 0.77. The majority of the mesh elements exhibited skewness values below approximately 0.4 and orthogonal quality values above approximately 0.65, indicating a good overall quality of the computational mesh. The obtained distributions confirmed that the mesh smoothing procedure effectively limited the occurrence of highly distorted elements and provided a mesh suitable for the subsequent CFD calculations.

Table 1. Results of grid independence analysis.

Number of Elements $\times 10^6$	Δp [kPa]	Difference Relative to the Next Finer Mesh [%]
16.1	690.40	2.70
24.0	709.54	1.79
44.8	722.47	-

Therefore, a mesh consisting of approximately 24 million elements was adopted for the final calculations (Figure 5). The near-wall region was resolved by applying ten prism layers along the solid walls, with a growth factor of 1.15. The height of the first prism layer was selected to achieve y^+ values below 1, allowing for full resolution of the viscous sublayer and ensuring accurate near-wall predictions with the $k-\omega$ SST turbulence model [21,22].

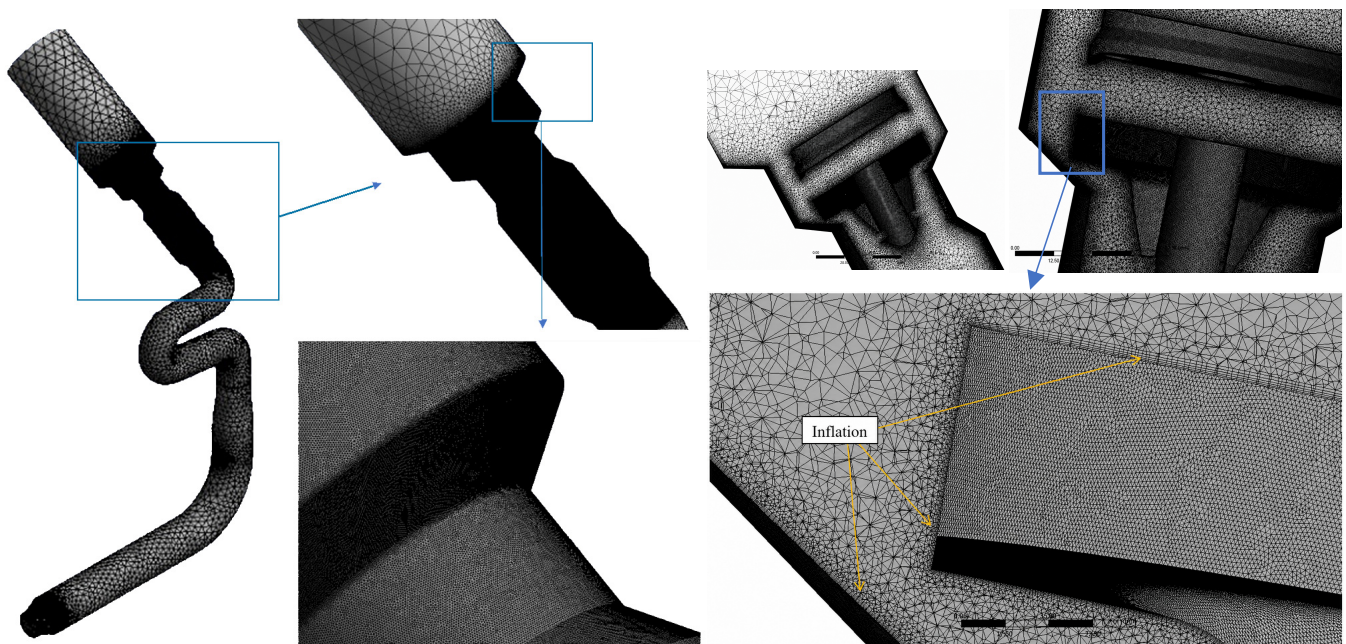


Figure 5. View of computational mesh with local refinements and near-wall layers.

2.4. Numerical Model and Boundary Conditions

The lowest Reynolds number, determined for the most unfavourable analyzed operating conditions of the fire monitor, i.e., at the minimum flow rate and a water temperature of 10 °C (corresponding to the highest viscosity considered in the study), was 225,760.

This value clearly indicates a fully turbulent flow regime across the entire analyzed operating range of the device. Consequently, transition effects were neglected. Due to the complex flow geometry, including changes in flow direction, local constrictions, and regions subjected to adverse pressure gradients, the $k-\omega$ SST turbulence model was selected. Compared with the standard $k-\varepsilon$ model, the SST formulation provides improved near-wall predictions in flows involving adverse pressure gradients and possible separation, while reducing the free-stream sensitivity associated with the standard $k-\omega$ model. These characteristics make the $k-\omega$ SST model particularly suitable for the present internal flow, which involves strong local acceleration, expansion, and recirculation [22].

The computational model included one inlet and one outlet, while all other surfaces were defined as no-slip walls. A mass flow rate ranging from 380 to 3610 kg/min, corresponding to the actual operating range of the fire monitor, was prescribed at the inlet. At the outlet, a static pressure condition of 0 Pa relative to the reference pressure of 101,325 Pa was applied, corresponding to atmospheric pressure at the outlet.

Only water flow was considered in the present simulations, and no foam concentrate was supplied through the foam-agent inlet. However, the complete geometry of the foam pipe was retained in the computational domain to account for its influence on the main water flow field and local hydraulic losses.

All simulations were performed under steady-state conditions. This approach was selected because the primary objective of the study was to assess average pressure losses, hydraulic stability, and cavitation susceptibility under representative operating conditions of the fire monitor. The analyzed cases corresponded to stabilized operating states with constant flow rates and fixed geometric configurations. Moreover, the experimental measurements used for model validation were conducted under the same stabilized operating conditions. Although transient pressure fluctuations and unsteady flow structures may occur during rapid operational changes, such phenomena were outside the scope of the present study. Therefore, a steady-state formulation was considered sufficient for reproducing the mean flow characteristics governing the hydraulic performance of the monitor.

The properties of water were considered to be temperature-dependent. Temperatures of 10 °C and 25 °C were taken into account when assessing the hydraulic parameters, whilst the additional temperature of 33 °C was analyzed solely for the purposes of assessing the risk of cavitation as an extreme scenario reflecting an exceptionally high water temperature that may only occur in storage tanks during hot weather.

Gravity was included in the model, with an acceleration of $g = 9.81 \text{ m/s}^2$. A wall roughness of 0.1 mm was assumed to represent the technical roughness of the internal surfaces of the analyzed flow system.

The convergence criteria for the numerical solution were set to normalized residuals not exceeding 10^{-4} for the momentum equations and turbulence model equations, and 10^{-3} for the continuity equation. In all analyzed cases, the residuals decreased by more than four orders of magnitude, while the monitored global quantities, in particular the mass flow rate and pressure drop, reached stable values. The mass balance between the inlet and outlet was also satisfied, confirming that a numerically converged solution had been obtained.

3. Results

The numerical analyses enabled a detailed assessment of water flow through the Shootfire water-foam fire monitor over a wide range of geometric configurations and operating conditions. The obtained pressure, velocity and flow-vector distributions made it possible to identify the regions with the highest flow dynamics, determine areas of potential cavitation risk and quantify the pressure drop ranges for all analyzed computational variants. The analyses were carried out using a reliably reconstructed geometry of the actual device, including the pipe sections, foam pipe, nozzle cone and guide vane system, which allowed the flow characteristics observed during device operation to be reproduced.

3.1. Experimental Validation of the Numerical Model

In order to verify the correctness of the developed CFD model, validation was performed based on experimental measurements of the total pressure drop across the water-foam fire monitor. The validation consisted of comparing the numerical results with the pressure-flow characteristic obtained during test-stand measurements.

Experimental measurements obtained using the setup described in Section 2 were used to validate the numerical model. The CFD calculations were performed for operating points corresponding to the experimental conditions. The total pressure drop was determined between cross-sections corresponding to the measurement locations used in the experiment, which enabled a direct comparison between the numerical and experimental results.

The measurement uncertainties associated with the flow rate and pressure measurements were used to define the error bars presented in the validation plot.

Figure 6 presents a comparison of the pressure drop characteristics obtained experimentally and numerically. The CFD results show very good agreement with the measurements over the entire analyzed flow rate range, accurately reproducing both the pressure drop values and the shape of the hydraulic characteristic of the tested fire monitor.

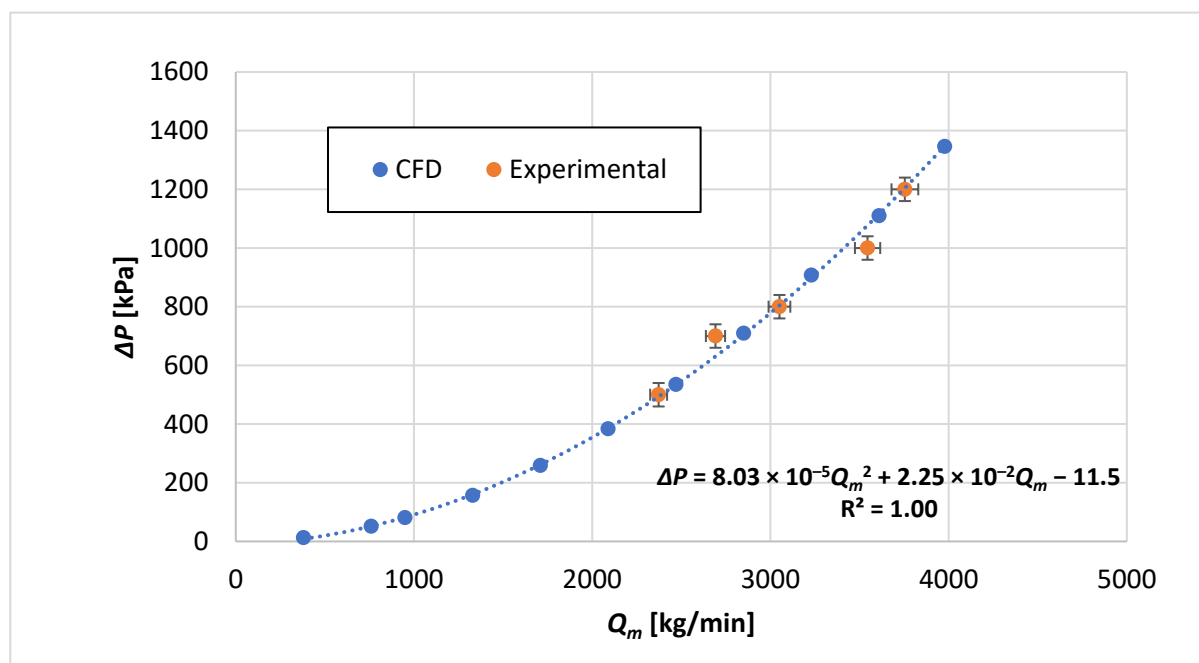


Figure 6. A comparison of the total pressure drop across the fire monitor obtained experimentally and numerically, including measurement uncertainty.

Most of the measurement points lie directly on the numerical curve or within the estimated measurement uncertainty, equal to $\pm 1\%$ for the flow rate and ± 37.5 kPa for the pressure drop. No systematic tendency of the model to overestimate or underestimate the

hydraulic losses was observed. The minor discrepancies between the experimental and numerical results can be attributed to measurement uncertainty, manufacturing tolerances of the actual device, and simplifications introduced into the numerical model. However, their influence remains negligible compared to the total pressure losses. The agreement between the numerical predictions and the experimental data was within the measurement uncertainty of the test setup, confirming the validity and reliability of the developed CFD model. Therefore, the model was considered experimentally validated and suitable for further analyses of internal flow characteristics, pressure distribution, and cavitation risk.

3.2. Influence of Monitor Body Geometry

Three fire monitor inclination angles, 30° , 45° , and 60° , were considered, as illustrated in Figure 7. In the first stage of the analysis, the total pressure distributions in the fire monitor cross-section were compared for these three configurations.

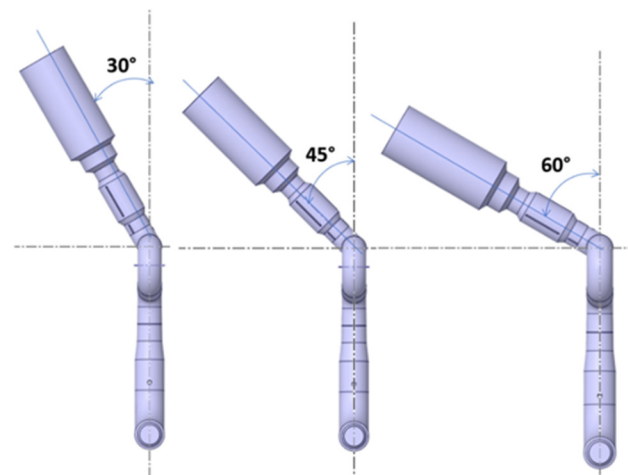


Figure 7. A comparative view of the fire monitor configurations for the three analyzed inclination angles: 30° , 45° , and 60° .

The results obtained for a flow rate of 2850 kg/min, presented in Figure 8, clearly indicate that the location of the lowest pressure regions is independent of the monitor inclination angle. In each case, the minimum pressure occurs in the vicinity of the constriction between the nozzle cone and the head housing, as well as near the foam pipe.

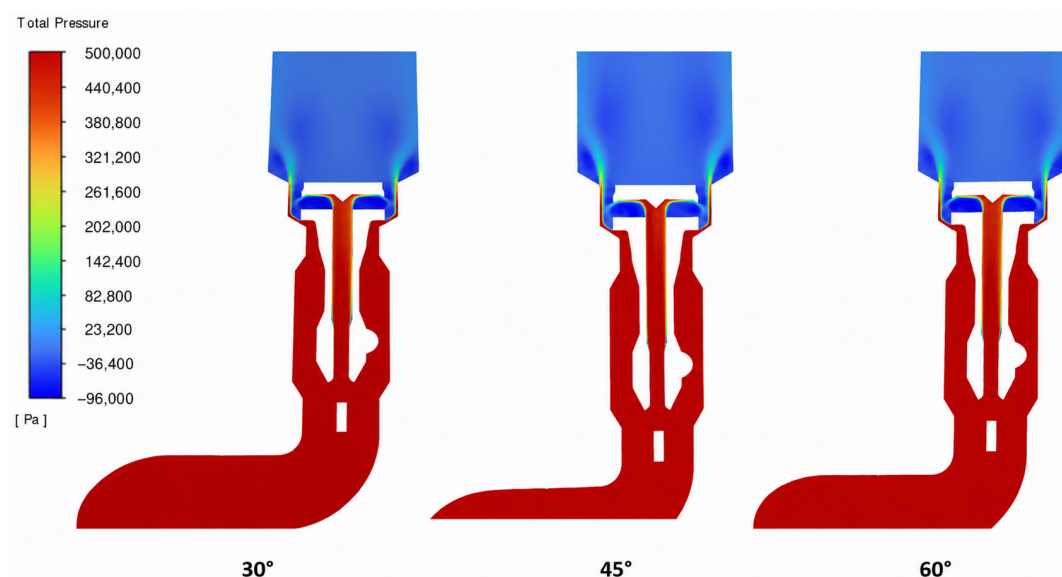


Figure 8. Pressure distributions in the fire monitor cross-section for three head inclination angles.

The velocity vectors shown in Figure 9 further confirm the similar flow pattern for all three inclination angles, revealing the same regions of local flow acceleration and similar flow directions within the monitor head, regardless of its position.

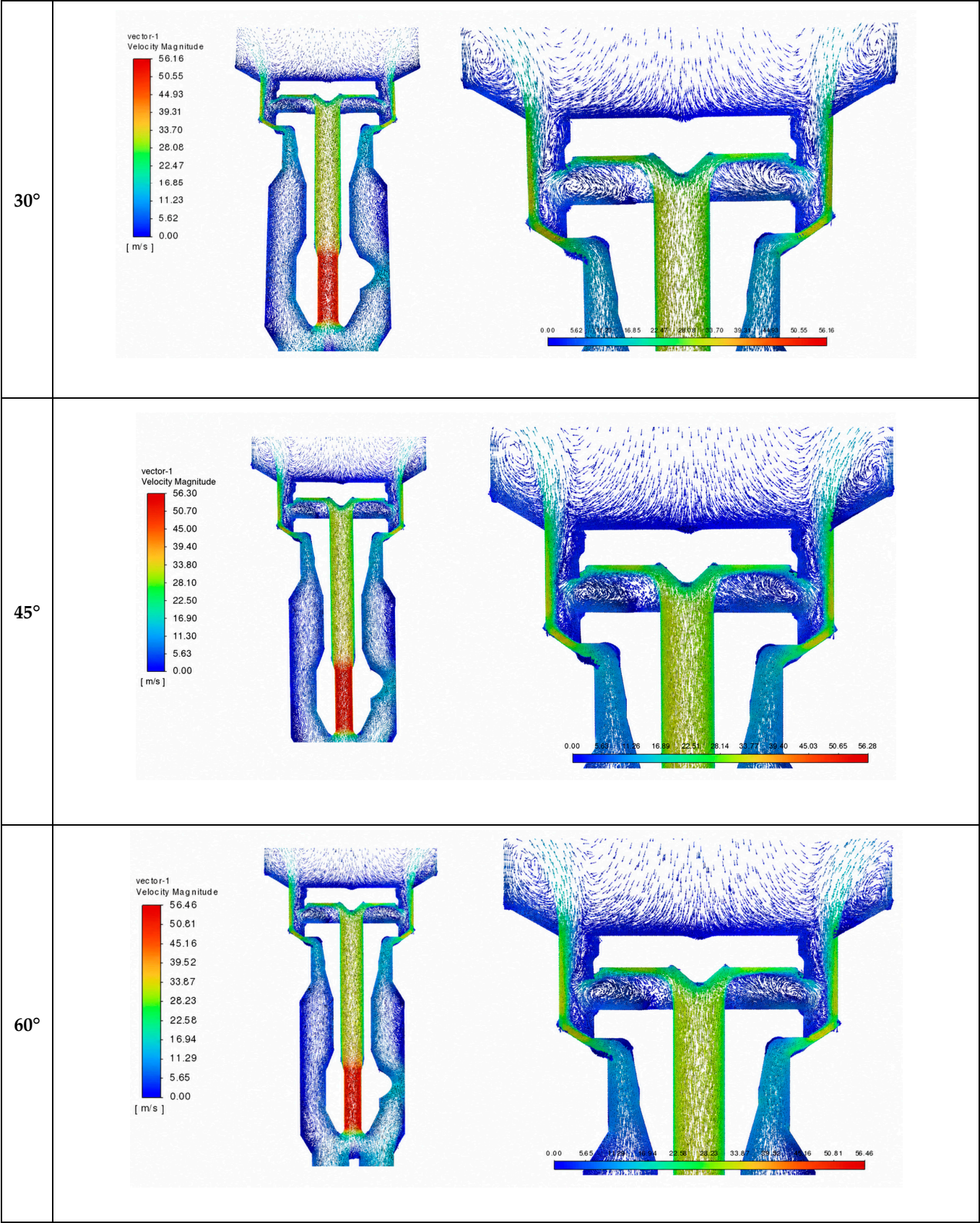


Figure 9. Velocity vector fields in the fire monitor cross-section for three head inclination angles.

The velocity distributions presented in Figure 10 likewise confirm that the flow through the head constriction is the dominant region of jet formation and the main source of pressure losses, and that the body inclination angle does not significantly alter the character of these phenomena.

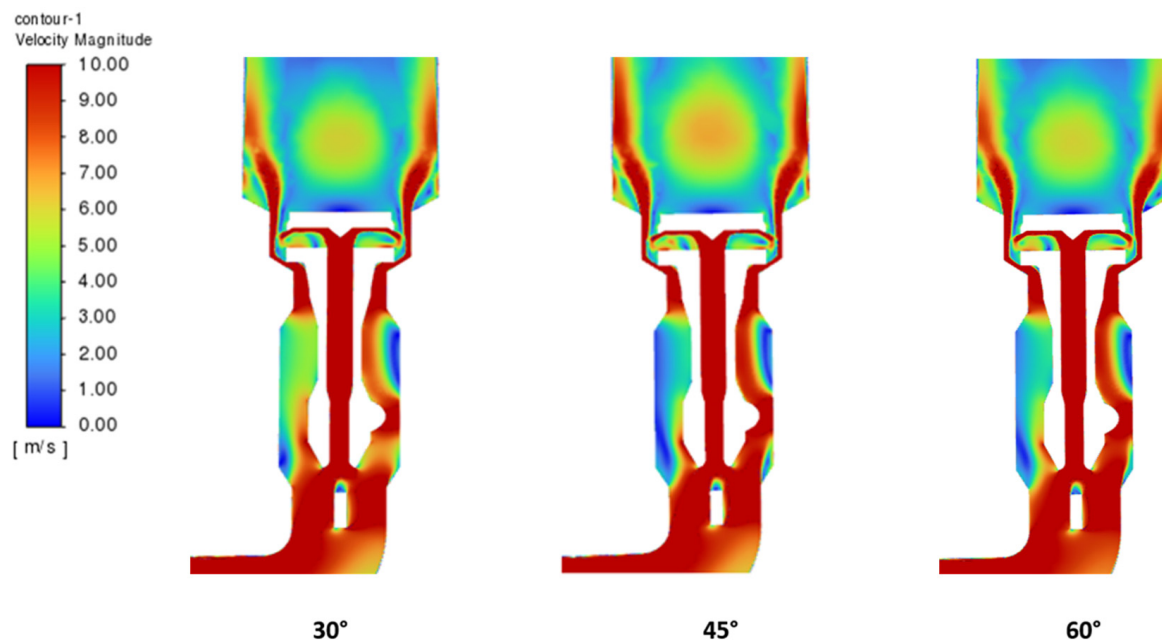


Figure 10. Velocity distributions in the fire monitor cross-section for three head inclination angles.

A quantitative comparison further confirms the limited influence of the monitor inclination angle. At a mass flow rate of 2850 kg/min, the pressure drop across the monitor head was 600.0, 623.4, and 628.9 kPa for inclination angles of 30°, 45°, and 60°, respectively, corresponding to a maximum difference of approximately 4.8%. For the complete fire monitor, the corresponding pressure drops were 688.9, 709.5, and 686.6 kPa, with a maximum difference of approximately 3.3%. The pressure drop characteristics remain similarly close over the entire investigated flow rate range, confirming that the monitor inclination angle has only a minor influence on its overall hydraulic performance.

3.3. Cavitation Risk

Cavitation may occur when the local pressure drops below the vapour pressure of water. The vapour pressure increases with temperature. Under typical operating conditions, the temperature of water supplied from a hydrant is approximately 10 °C, for which the vapour pressure is slightly above 1 kPa. However, under unfavourable conditions, water previously stored in a tank may heat up to approximately 33 °C, at which the vapour pressure increases to about 5 kPa. This conservative case was therefore adopted for the cavitation risk assessment.

The pressure values presented in the CFD results represent gauge pressure, referenced to the atmospheric pressure at sea level (101,325 Pa). The lower bound of the pressure scale was defined as −96 kPa gauge pressure, corresponding to an absolute pressure of 5.325 kPa. Therefore, regions exhibiting gauge pressure values below −96 kPa would indicate conditions potentially conducive to cavitation, where the local absolute pressure approaches or falls below the water vapour pressure. Such regions would appear as uncoloured areas in the pressure contour plots.

A comparison of the minimum pressure values obtained from the CFD simulations (Figure 8) with the water vapour pressure indicates that, for a flow rate of 2850 kg/min and the maximum expected water temperature of 33 °C, no regions were identified where

the local absolute pressure fell below the water vapour pressure. Similar observations were made for all other simulated flow rates within the range of 380–3610 kg/min.

As shown in Figure 8, the water pressure within the main pipe section remains relatively high. Lower pressure values, accompanied by elevated flow velocities and local flow recirculation, can be observed in the vicinity of the monitor outlet nozzle, particularly within the foam tube, nozzle cone, and cone plate regions (Figures 8–10). However, the local pressure remains above the vapour pressure of water throughout the entire flow domain.

To further verify the pressure-based cavitation assessment, a supplementary multi-phase calculation was performed using the Mixture model coupled with the Schnerr–Sauer cavitation model. Under the most critical conditions, the minimum absolute pressure was approximately 5.12 kPa, while the maximum predicted vapour volume fraction was only 2.6×10^{-5} . These results indicate that the flow approaches the cavitation inception threshold; however, only trace vapour formation was predicted and no developed cavitating region was observed.

3.4. Pressure Loss

Based on the CFD results, pressure loss characteristics were developed over the entire analyzed flow rate range and for three different inclination angles of the monitor head: 30°, 45°, and 60°. The characteristics were determined both for the nozzle section (Figure 11) and for the entire fire monitor (Figure 12).

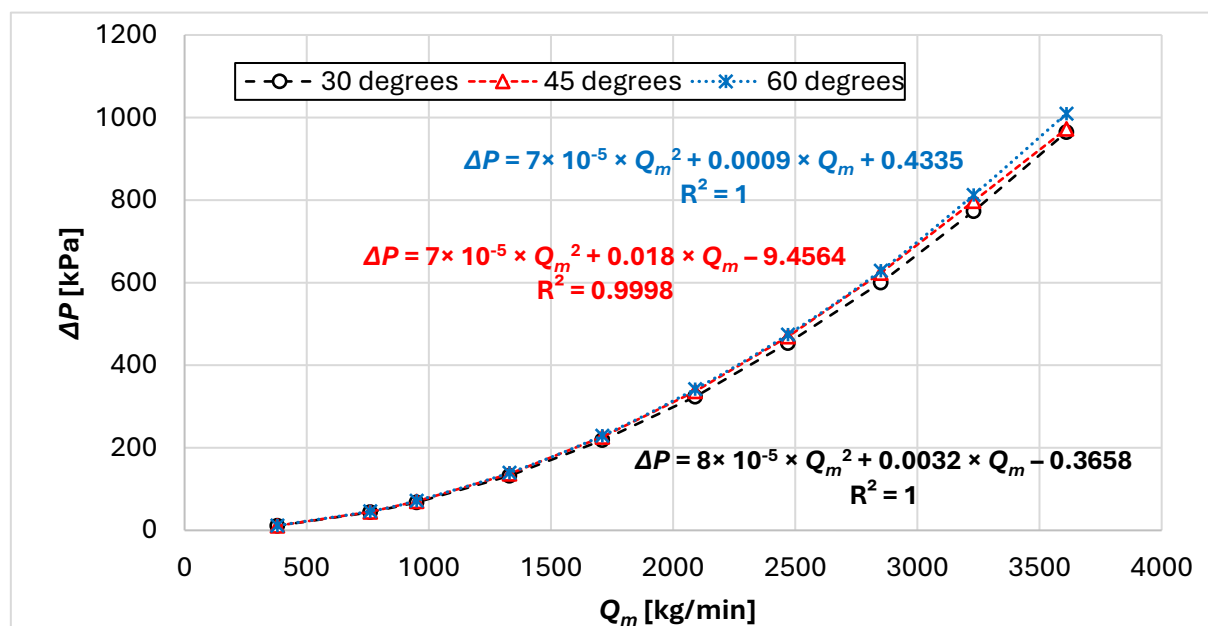


Figure 11. Pressure drops across the nozzle section of the fire monitor as a function of flow rate for different head inclination angles.

The locations of the surfaces defining the nozzle section (P2–P3) and the total length of the analyzed fire monitor (P1–P3) are shown in Figure 13.

The relationship between flow rate and pressure drop across the monitor head exhibits a quadratic trend, which is characteristic of turbulent flow through flow-restricting components. This relationship was further generalized over the entire range of monitor elevation angles, demonstrating that the external geometry has only a negligible effect on the hydraulic performance. Consequently, the flow behaviour within the monitor is governed almost exclusively by the geometry of the monitor head and the internal nozzle configuration.

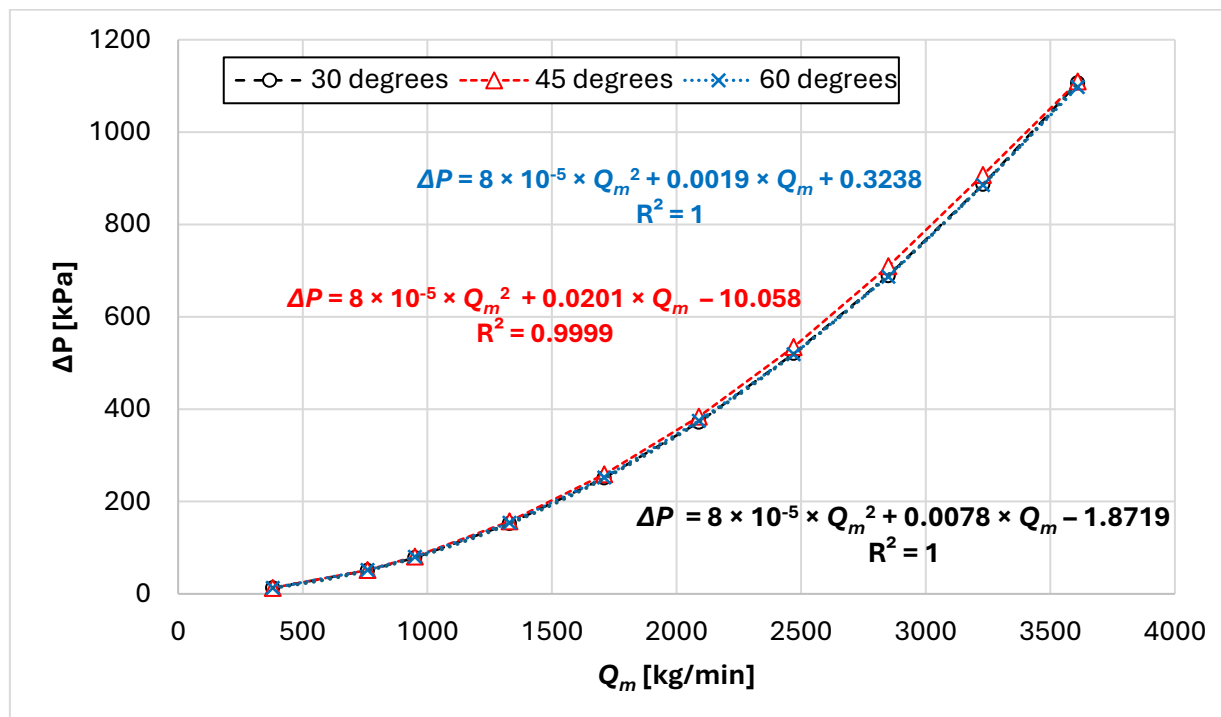


Figure 12. Total pressure drops across the fire monitor as a function of flow rate for different head inclination angles.

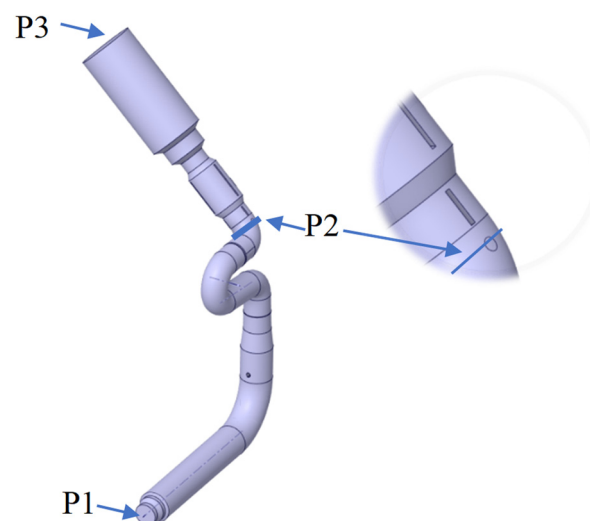


Figure 13. Cross-section locations for pressure drop calculations across the nozzle section and the entire fire monitor.

Based on the results presented in Figure 11, Equation (1) was derived to describe the pressure drop across the nozzle section of the fire monitor. The coefficients of Equation (1) were obtained by fitting a second-order polynomial function to the CFD results presented in Figure 11 using the least-squares method. Since only negligible differences were observed between the pressure loss characteristics corresponding to different monitor inclination angles, a single approximation equation was adopted for the entire analyzed range of head inclinations. The equation remains valid for monitor head inclination angles ranging from 0° to 90° , indicating that it can be regarded as a general hydraulic characteristic of the fire monitor.

$$\Delta P_{2-3} = 0.000071 \cdot Q_m^2 + 0.018021 \cdot Q_m - 9.456397 \quad (1)$$

where Q_m is the mass flow rate, kg/min; ΔP is the pressure loss, kPa.

Equation (1) may be used as a practical engineering tool for estimating hydraulic losses generated by the fire monitor within its operating range. The predicted pressure loss can be incorporated into the hydraulic balance of a firefighting system and combined with pipeline losses and elevation head requirements during pump selection. Consequently, the equation may support the determination of the minimum pump head required to maintain the desired flow rate at the monitor outlet under design operating conditions.

3.5. Effect of Water Temperature

To investigate the effect of water temperature on the flow parameters, simulations were performed for two temperatures: 10 °C and 25 °C. The comparison of velocity and pressure distributions for these two limiting temperatures of the supplied water, presented in Figure 14, indicates only minor differences, mainly resulting from the reduced viscosity of warmer water. As shown in Figure 15, increasing the water temperature from 10 °C to 25 °C reduces the pressure drop by approximately 2%, which is of marginal practical significance for the use of the fire monitor in fire protection systems.

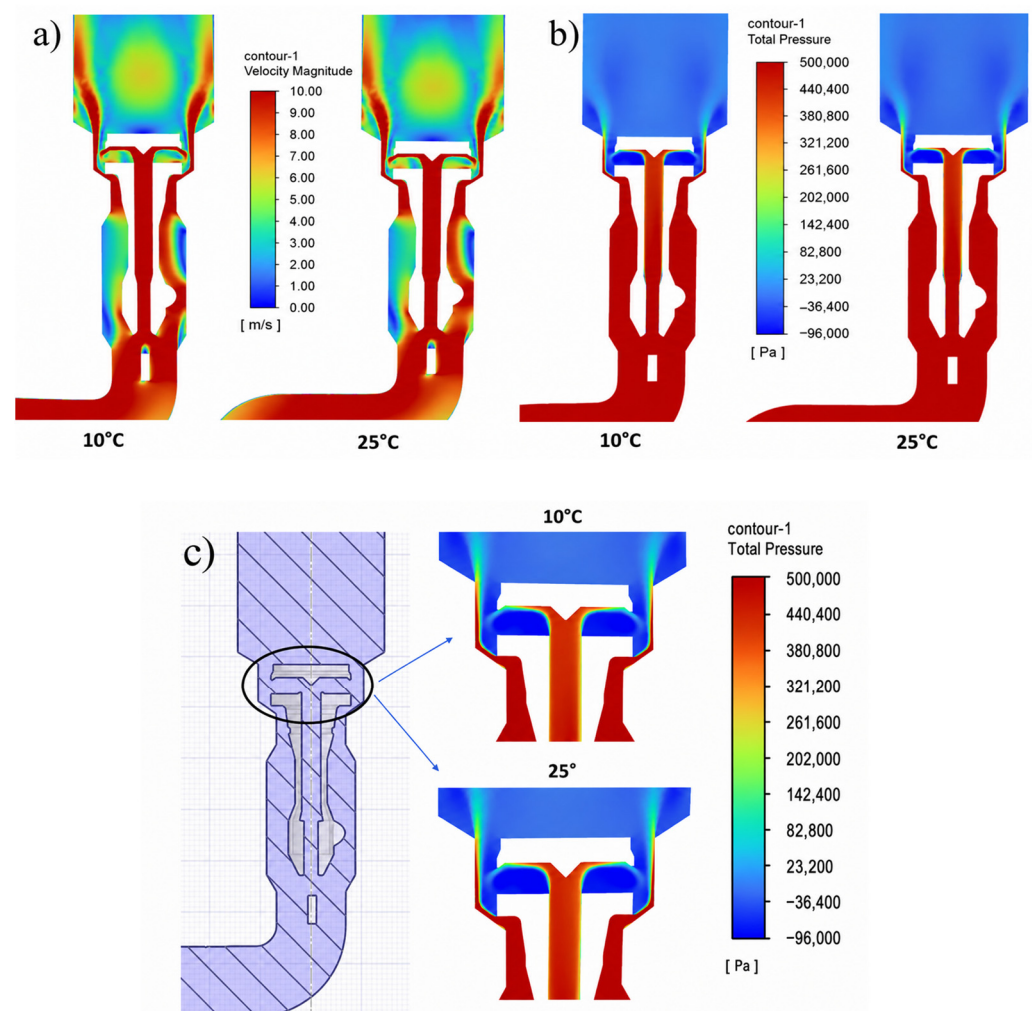


Figure 14. Velocity distributions (a) and pressure distributions (b) in the fire monitor cross-section, and pressure distributions (c) in the nozzle section, for different water temperatures at a mass flow rate of 2850 kg/min.

The single-phase pressure-based assessment did not identify any regions where the local pressure decreased below the water vapour pressure at the higher temperature. This indicates that the fire monitor maintains stable operating parameters both at temperatures

typical of hydrant water, approximately 10 °C, and for water drawn from heated tanks, approximately 25 °C.

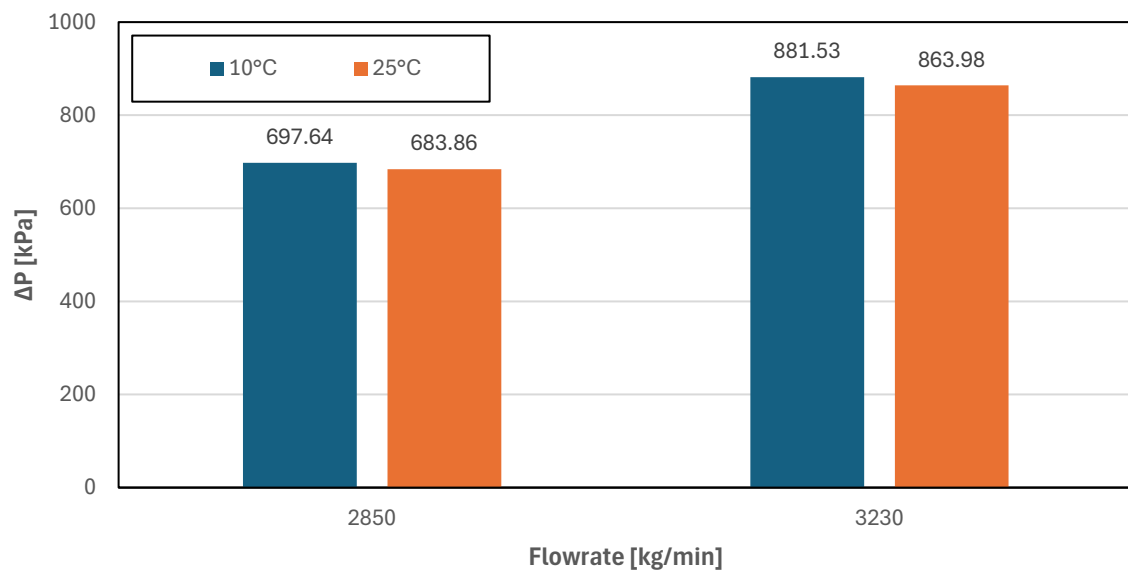


Figure 15. Pressure drops across the monitor head for two different water temperatures.

As part of the conservative cavitation risk assessment described in Section 3.3, an additional simulation was carried out for a water temperature of 33 °C. This case was not included in the temperature comparison graphs, as the differences compared with the 25 °C scenario were negligible and had no impact on the monitor's overall hydraulic characteristics. As the flow under analysis remained fully turbulent across the entire operating range, the effect of viscosity changes between 25 °C and 33 °C on pressure losses was minimal.

3.6. Effect of Cone Plate Geometry Modification

An important part of the numerical analyses was to investigate the effect of nozzle cone plate geometry modifications, which may result from wear, erosion or intentional design changes. The highest water velocity occurred in the restriction between the nozzle cone and the monitor head housing, i.e., in the region forming the outlet nozzle geometry. The analyzed cone plate modifications, consisting of different edge fillet radii, are shown in Figure 16.

Four fillet radius variants were analyzed: 0 mm, corresponding to the baseline geometry, and 2 mm, 4 mm and 7 mm. The results presented in Figure 16 show a clear dependence of pressure drop on the fillet radius for flow rate of 2850 kg/min. It was observed that rounding the cone plate edge increases the flow area in the narrowest section, thereby reducing pressure losses. The greatest improvement was obtained for a fillet radius of 4 mm. A further increase in the fillet radius did not lead to significant additional changes, suggesting the existence of a geometric optimum in the region of 4 mm.

The pronounced reduction in pressure loss observed with increasing cone plate fillet radius can be attributed to a smoother change in flow direction in the vicinity of the edge (Figure 17). In the sharp-edge configuration, the abrupt change in geometry promotes strong local flow acceleration and increases the tendency for flow separation and recirculation, resulting in higher local hydraulic losses. Introducing a fillet provides a more gradual transition of the flow path, thereby reducing local velocity gradients and the associated energy dissipation. The largest improvement is observed when the fillet radius is increased up to approximately 4 mm. A further increase from 4 mm to 7 mm reduces the pressure drop only from approximately 358 kPa to 348 kPa, indicating that most of the hydraulic ben-

enefit associated with smoothing the edge has already been achieved. Therefore, $R = 4$ mm may be regarded as a practical optimum or a point of diminishing hydraulic benefit rather than the absolute minimum pressure loss configuration.

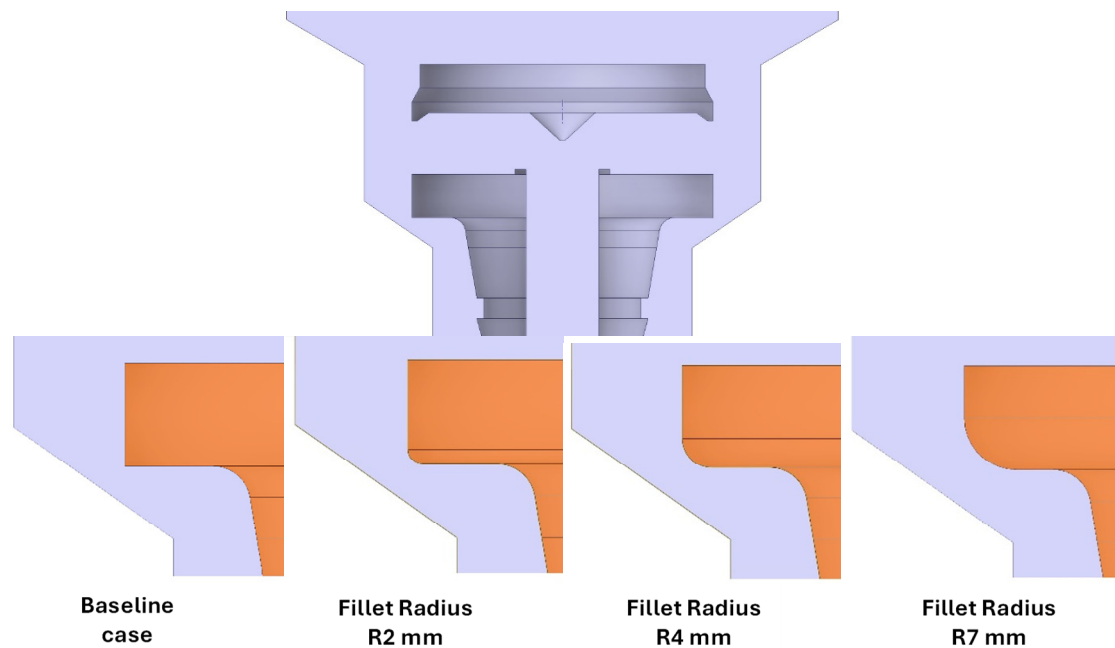


Figure 16. Modification of the cone plate geometry at the base of the nozzle cone.

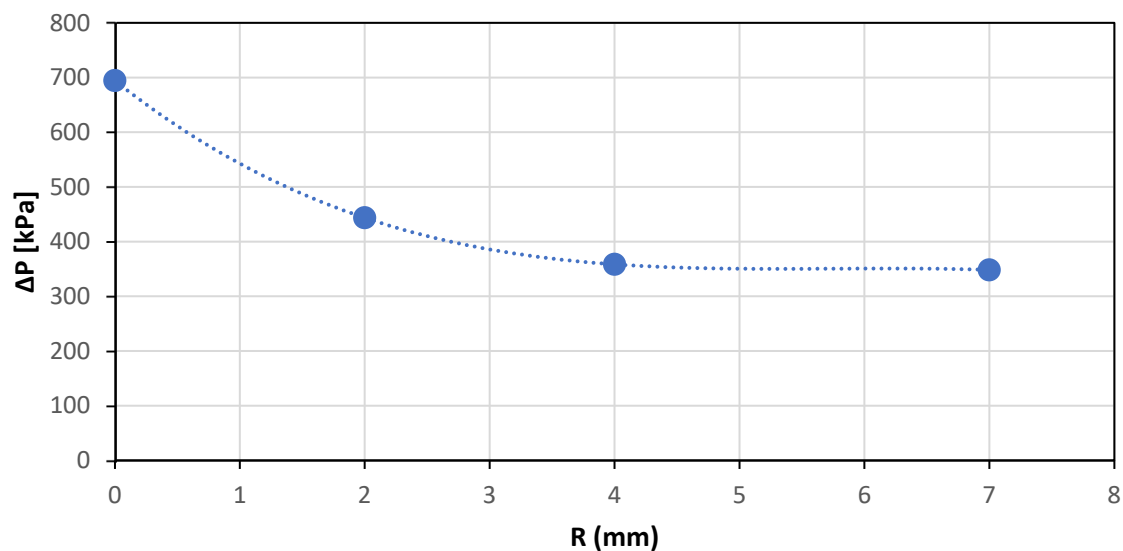


Figure 17. The effect of cone plate fillet radius on the pressure drop in the fire monitor head at a flow rate of 2850 kg/min.

The pressure distributions for different fillet radii (Figure 18) confirm that sharper edges generate stronger local flow acceleration and lower pressure values, whereas smoothing this edge results in a more uniform flow. According to the single-phase pressure-based assessment, the local pressure remained above the water vapour pressure in all analyzed cases, including the sharpest geometry with a fillet radius of 0 mm. The results clearly indicate that the cone plate geometry is one of the key factors determining the hydraulic characteristics of the fire monitor.

From a practical perspective, the results indicate that the cone plate edge geometry should be carefully controlled during manufacturing and maintenance. Introducing edge

rounding reduced pressure losses from approximately 694 kPa for the sharp-edge configuration to 445 kPa, 358 kPa, and 348 kPa for fillet radii of 2 mm, 4 mm, and 7 mm, respectively. This corresponds to a reduction of approximately 36%, 48%, and 50% relative to the baseline geometry. The results demonstrate that edge rounding substantially improves hydraulic performance, while the additional benefit obtained by increasing the fillet radius beyond 4 mm is relatively small. Consequently, manufacturing and maintenance procedures should ensure that excessive edge wear, deformation, or unintended geometric modifications do not occur.

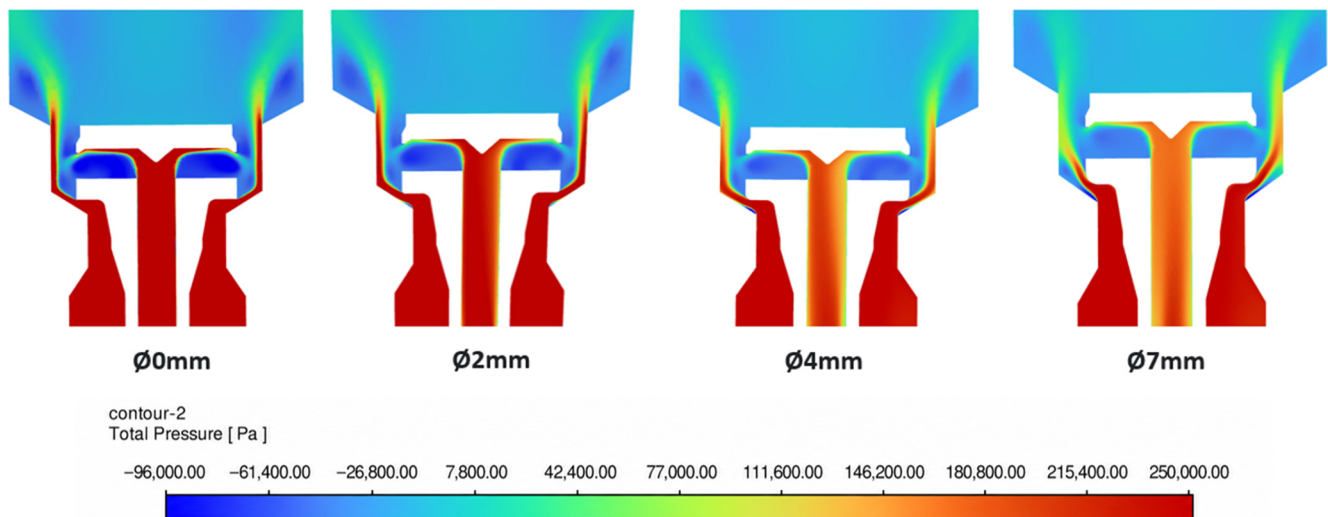


Figure 18. Pressure distributions in the fire monitor cross-section near the outlet nozzle for different fillet radii of the nozzle cone plate.

3.7. Effect of Nozzle Cone Position

An even more significant effect on pressure drop was observed when the position of the nozzle cone was changed relative to its nominal setting. A 1 mm downward displacement of the cone, corresponding to a reduction in the minimum flow cross-section, caused a substantial increase in pressure drop of nearly 250 kPa at a flow rate of 2850 kg/min, as shown in Figure 19. Conversely, a 1 mm upward displacement of the cone resulted in a reduction in pressure losses of approximately 120 kPa at 2850 kg/min.

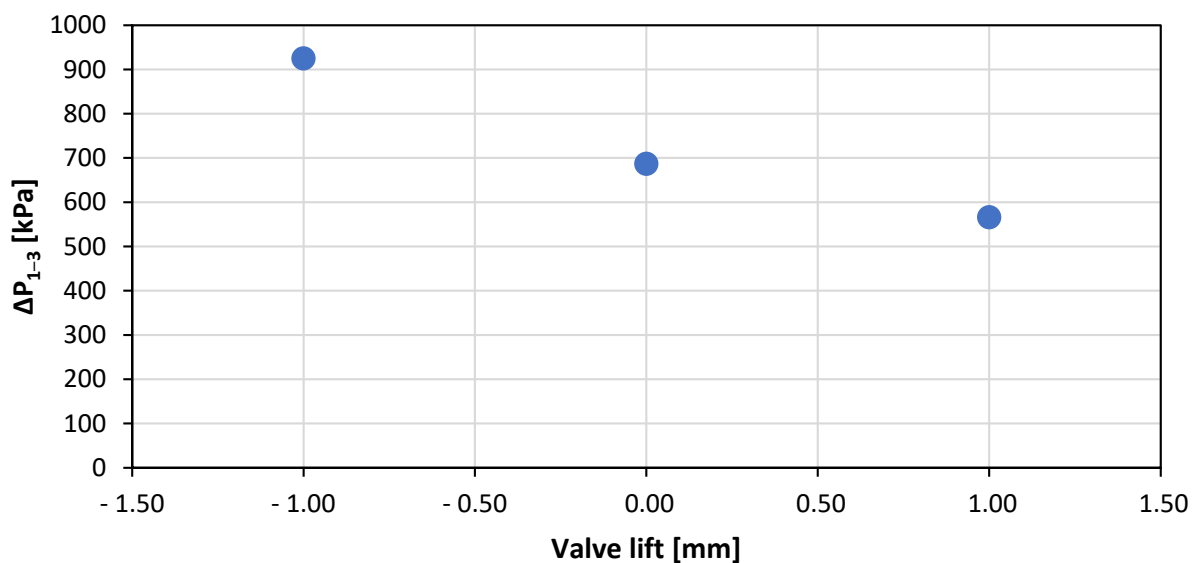


Figure 19. Relationship between pressure drop across fire monitor and nozzle cone position for flow rate of 2850 kg/min.

These changes are significantly greater than those caused by water temperature or the monitor inclination angle, confirming that the nozzle cone position is the most important parameter affecting the hydraulic performance of the device. The pressure and velocity distributions in the head region, shown in Figures 20 and 21, respectively, indicate that changes in cone position lead to significant variations in the local pressure and velocity gradients.

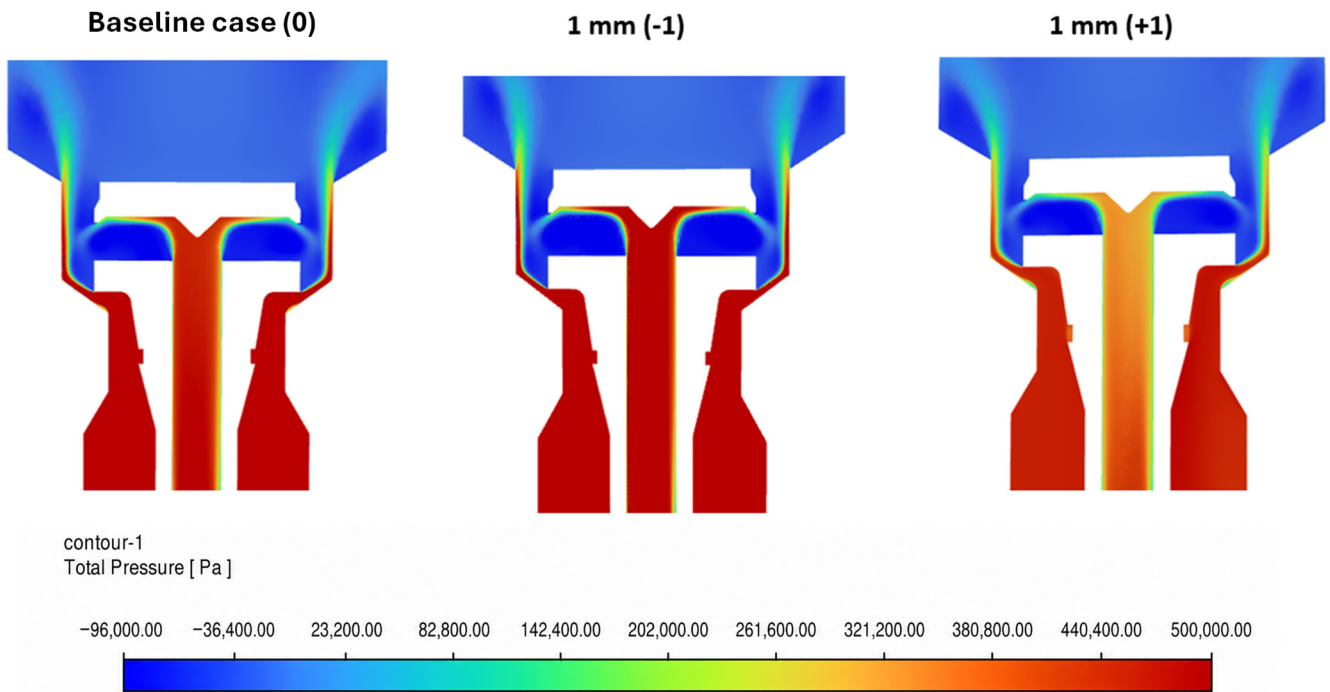


Figure 20. Pressure distribution in the fire monitor cross-section near the outlet nozzle for different cone plate displacements.

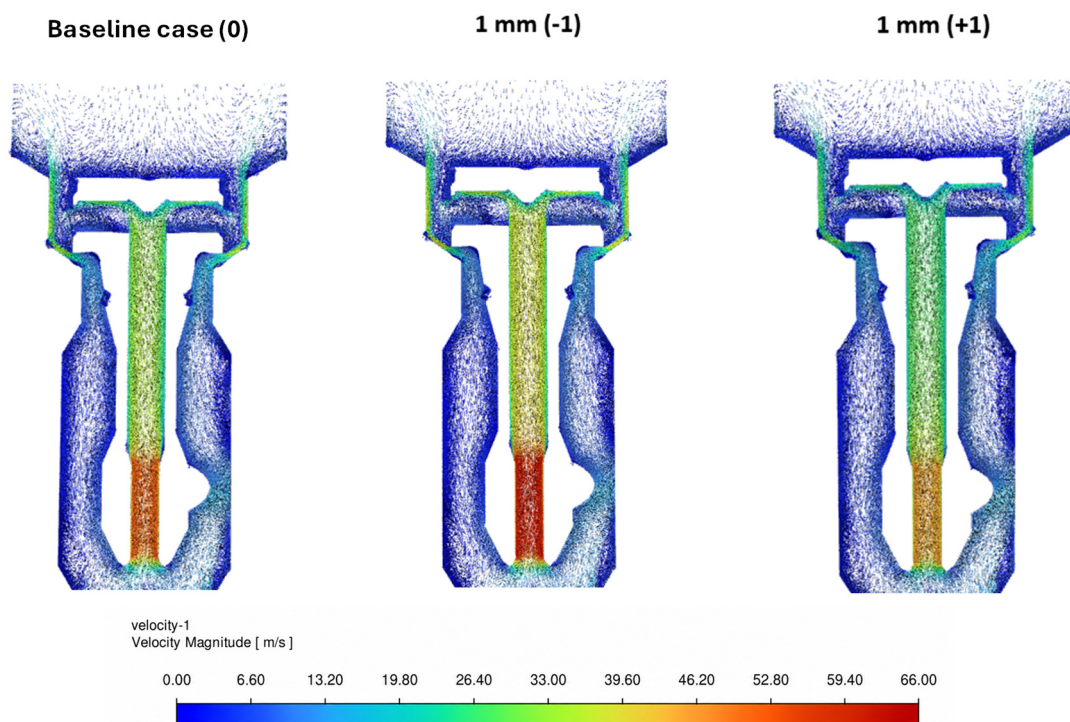


Figure 21. Velocity vector fields in the fire monitor cross-section near the outlet nozzle for different cone plate displacements.

The observed response is distinctly nonlinear. A downward displacement of the nozzle cone reduces the minimum flow cross-section, resulting in a considerable increase in local flow velocity and hydraulic resistance within the annular throttling region formed between the cone and the housing. Consequently, pressure losses increase disproportionately with decreasing throttling area. In contrast, an upward displacement enlarges the flow cross-section and reduces local resistance; however, the resulting decrease in pressure losses is less pronounced because other geometric features of the monitor increasingly contribute to the overall hydraulic resistance. This asymmetry indicates that the monitor performance is governed primarily by the local throttling effect generated by the nozzle cone, whose influence becomes particularly significant when the effective flow area is reduced.

Despite the large changes in pressure drop, no pressure decrease below the vapour pressure of water was observed, even for the most critical variant, corresponding to a -1 mm displacement. This means that the change in cone position is hydraulically significant but does not create a risk of cavitation.

From a practical perspective, the results indicate that accurate positioning of the nozzle cone is essential for maintaining the expected hydraulic performance of the monitor. A displacement of only 1 mm from the nominal position changed the pressure loss from 686.57 kPa to 924.58 kPa when the cone was moved downward and to 565.98 kPa when the cone was moved upward. These changes correspond to an increase of approximately 35% and a decrease of approximately 18%, respectively. The results demonstrate that even small assembly or adjustment deviations may significantly affect monitor performance and therefore should be minimized during manufacturing, installation, and maintenance procedures.

4. Conclusions

An experimentally validated CFD investigation of a commercially available Shootfire-1000 water-foam fire monitor was performed to assess pressure losses, hydraulic stability, and cavitation risk under representative operating conditions.

The results demonstrated that the hydraulic performance of the monitor is governed primarily by the geometry of the monitor head, which accounted for approximately 90% of the total pressure loss. The annular constriction between the nozzle cone and the housing, the cone plate edge, and the foam pipe region were identified as the dominant sources of hydraulic losses and should therefore be considered the primary targets for hydraulic optimization.

The monitor elevation angle and water temperature had only a minor influence on hydraulic performance within the investigated operating range. Increasing water temperature from 10 °C to 25 °C reduced pressure losses by approximately 2%, while variations in monitor elevation angle produced only negligible changes in pressure drop characteristics. These results indicate stable monitoring operation under typical firefighting conditions.

Local geometric modifications had a pronounced influence on hydraulic losses. The nozzle cone position was identified as the most influential design parameter, with a displacement of only ± 1 mm resulting in pressure drop changes of approximately +250 kPa and -120 kPa relative to the nominal configuration. Furthermore, introducing a 4 mm fillet radius at the cone plate edge reduced pressure losses by nearly twofold, whereas further radius increases provided only limited additional benefit. These findings indicate that manufacturing tolerances, adjustment procedures, and local geometric details play a critical role in monitor performance.

For the reference operating conditions, the minimum pressure remained above the water vapour pressure, corresponding to a cavitation margin of approximately 31–44 kPa. Under the more conservative 33 °C condition, supplementary calculations using the Schnerr–Sauer cavitation model showed that the flow approached the cavitation inception threshold;

however, only a trace vapour volume fraction was predicted and no developed cavitating region was observed. The monitor therefore demonstrated stable hydraulic behaviour over a wide range of flow rates, operating conditions, and geometric configurations.

The main novelty of this work lies in the combined experimental validation and CFD-based investigation of a commercially available Shootfire-1000 water-foam fire monitor. Unlike previous studies focused primarily on flow optimization and velocity field improvement, the present study provides a parametric assessment of the influence of nozzle cone position, cone plate geometry, monitor elevation angle, and water temperature on pressure losses, hydraulic stability, and cavitation risk, while also identifying the dominant sources of hydraulic losses within the monitor.

This study has several limitations. The analyses were performed for a single commercially available Shootfire-1000 fire monitor and were based on steady-state operating conditions. Therefore, the conclusions should be interpreted within the investigated geometry and operating range.

Future research should include investigations of additional fire monitor geometries, transient operating conditions, and advanced cavitation modelling approaches. Such studies could contribute to the development of generalized design guidelines for reducing hydraulic losses and improving hydraulic performance in firefighting monitors.

Author Contributions: Conceptualization, A.C., B.K. and M.Z.; methodology, A.C., B.K. and M.Z.; software, A.C., B.K. and M.Y.; validation, A.C., B.K. and M.Z.; formal analysis, A.C. and B.K.; investigation, A.C., B.K., M.Y. and M.Z.; resources, A.C., B.K. and M.Z.; data curation, A.C., B.K. and M.Y.; writing—original draft preparation, A.C., B.K. and M.Z.; writing—review and editing, A.C., B.K. and M.Z.; visualization, A.C. and B.K.; project administration, M.Z.; funding acquisition, M.Z. All authors have read and agreed to the published version of the manuscript.

Funding: The research and development work was co-financed by the European Regional Development Fund under the Smart Growth Operational Programme 2014–2020, Grant No. POIR.01.01.01-00-1524/20 entitled “Development of an innovative automatic control system integrated with a fire foam-water cannon with thermovision and/or fiber optic detection”.

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

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 thank the National Centre for Research and Development (NCBR) for its substantive and administrative support throughout the project. The authors would like to thank Anita Siwek ASPROJEKT and all contractors and subcontractors involved in the implementation of the project tasks.

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

Abbreviations

The following abbreviations are used in this manuscript:

CFD Computational Fluid Dynamics
UHV ultra-high voltage

References

1. Brennen, C.E. *Cavitation and Bubble Dynamics*; Oxford University Press: New York, NY, USA, 2014.
2. Folden, T.S.; Aschmoneit, F.J. A Classification and Review of Cavitation Models with an Emphasis on Physical Aspects of Cavitation. *Phys. Fluids* **2023**, *35*, 081301. [[CrossRef](#)]
3. Franc, J.P.; Michel, J.M. *Fundamentals of Cavitation*; Kluwer Academic Publishers: Dordrecht, The Netherlands, 2004.

4. Hu, C.; Wu, Z.; Ferrari, A.; Ji, M.; Deng, J.; Vento, O. Numerical Study on Internal Flow and Cavitation Characteristics of GDI Injectors for Different Nozzle Orifice Geometries. *Energies* **2024**, *17*, 4114. [[CrossRef](#)]
5. Zhu, Y.; Wang, Y.; Wang, Y. Research on the Internal Flow and Cavitation Characteristics of Petal Bionic Nozzles Based on Methanol Low-Pressure Injection. *Energies* **2024**, *17*, 5612. [[CrossRef](#)]
6. Li, H.; Chen, J.; Guo, J.; Zhu, H.; Lin, Y.; Ge, H. A Study of Cavitation Erosion in Artificial Submerged Water Jets. *Appl. Sci.* **2024**, *14*, 4804. [[CrossRef](#)]
7. Hao, J.; Yu, C.; Fu, Y.; Zhang, P.; Hou, D.; Shen, Y.; Yuan, J. Study on the Cavitation Performances and Unsteady Characteristics of a Centrifugal Fire Pump. *J. Appl. Fluid Mech.* **2025**, *18*, 3036–3052. [[CrossRef](#)]
8. Liu, H.; Hu, J.; He, Z.; Wang, K.; Lu, X. Experimental Research on Pressure Pulsation and Cavitation Characteristics of Centrifugal Fire Pump. *J. Low Freq. Noise Vib. Act. Control* **2023**, *42*, 1339–1349. [[CrossRef](#)]
9. Kalender, O.; Soyhan, H.S. Computational Fluid Dynamics (CFD) Optimization of a Fire Monitor for Enhanced Flow Performance. *Mach. Technol. Mater.* **2025**, *19*, 404–407.
10. Zhang, J.; Luo, S.; Huang, Y.; Guo, Y.; Zhang, J.; Li, D.; Zhao, C. Optimization of Rectifiers in Firefighting Monitors Used in UHV Fire Safety Applications. *Energies* **2023**, *16*, 3898. [[CrossRef](#)]
11. Pawłowska-Salach, A.; Zielina, M.; Kaczmarski, K. Passive Water Intake Screen to Reduce Entrainment of Debris and Aquatic Organisms under Various Hydraulic Flow Conditions. *Water* **2025**, *17*, 3424. [[CrossRef](#)]
12. Aliuly, A.; Amanzholov, T.; Seitov, A.; Momyshev, N.; Jaichibekov, N.; Kaltayev, A. Hydraulic Design and CFD-Based Parametric Study for Optimizing Centrifugal Pump Impeller Performance. *Appl. Sci.* **2024**, *14*, 10161. [[CrossRef](#)]
13. Warzocha, K.; Rzucidło, P. CFD-Based Steady-State Flow Force Compensation in Direct Drive Servo Valves. *Appl. Sci.* **2026**, *16*, 3262. [[CrossRef](#)]
14. Zhang, B.; Zhu, C.; Li, J.; Wang, H.; Liu, X.; Wang, K. Optimization Study on Nozzle Selection Based on the Influence of Nozzle Parameters on Jet Flow Field Structure. *Appl. Sci.* **2024**, *14*, 9098. [[CrossRef](#)]
15. Zhang, X.; Guo, C.; Ge, H.; Jiao, L. Analysis of Internal Flow Field Characteristics in the Nozzle of the Three-Phase Jet Fire Monitor. *Sci. Rep.* **2025**, *15*, 21968. [[CrossRef](#)] [[PubMed](#)]
16. Ji, B.; Luo, X.; Arndt, R.E.A.; Peng, X.; Wu, Y. Numerical Simulation of Three-Dimensional Cavitating Flow in a Centrifugal Pump Using a Cavitation Model Coupled with a Filter-Based Turbulence Model. *J. Fluids Eng.* **2013**, *135*, 091302.
17. Koukouvini, P.; Gavaises, M.; Supponen, O.; Farhat, M. Numerical Simulation of a Collapsing Bubble Subjected to Gravity. *Phys. Fluids* **2016**, *28*, 052103. [[CrossRef](#)]
18. Zhang, D.; Shi, W.; Pan, D.; Dubuisson, M. Numerical and Experimental Investigation of Cavitation Dynamics and Erosion in Hydraulic Machinery. *Renew. Energy* **2021**, *170*, 386–399.
19. Singhal, A.K.; Athavale, M.M.; Li, H.; Jiang, Y. Mathematical Basis and Validation of the Full Cavitation Model. *J. Fluids Eng.* **2002**, *124*, 617–624. [[CrossRef](#)]
20. Schnerr, G.H.; Sauer, J. Physical and Numerical Modeling of Unsteady Cavitation Dynamics. In Proceedings of the Fourth International Conference on Multiphase Flow (ICMF 2001), New Orleans, LA, USA, 27 May–1 June 2001.
21. ANSYS Inc. *ANSYS Fluent Theory Guide, Release 2025 R2*; ANSYS, Inc.: Canonsburg, PA, USA, 2025; Available online: <https://ansyshelp.ansys.com> (accessed on 9 March 2026).
22. Menter, F.R. Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. *AIAA J.* **1994**, *32*, 1598–1605. [[CrossRef](#)] [[PubMed](#)]

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.