

Article

Improving Flow Efficiency via Internal Flow Channel Optimization Design in a Novel Non-Pressurized Diaphragm Deluge Alarm Valve

Yan Zheng ^{1,2}, Jun Wang ², Zijie Yin ^{1,2}, Jinhao Zhang ², Wenfeng Shen ^{1,2}, Tianyi Zheng ^{1,*}, Zongheng Chen ³ and Jieqing Zheng ^{1,2}

¹ College of Marine Equipment and Mechanical Engineering, Jimei University, Xiamen 361021, China; 18760288175@163.com (Y.Z.); 13533224979@163.com (Z.Y.); shenwf20@jmu.edu.cn (W.S.); zhengjieqing@jmu.edu.cn (J.Z.)

² Key Laboratory of Energy Cleaning Utilization and Development of Fujian Province, Jimei University, Xiamen 361021, China; 202512855064@jmu.edu.cn (J.W.); 15980586953@163.com (J.Z.)

³ Xiamen Aids to Navigation Department of the Eastern Navigation Service Center, Ministry of Transport, Xiamen 361005, China; xmhbc@126.com

* Correspondence: zhengtianney@jmu.edu.cn

Abstract

Automatic sprinkler systems are widely used for fire protection in various buildings, with deluge valves serving as the core component of these systems. Traditional deluge valves employ a diaphragm-type design (Zoning Sprinkler Fire Monitor, ZSFM), which is prone to significant safety hazards such as corrosion and damage due to uneven pressure distribution on the diaphragm. This study modified a 150 mm diameter ZSFM to a non-pressure diaphragm type, establishing and validating a CFD model of the internal flow field. Based on the original structure, six drag reduction optimization cases are designed. Among these, case 5 exhibits the minimum inlet-to-outlet pressure drop of 0.050 MPa under rated operating conditions, meeting and significantly exceeding the fire protection industry standard (≤ 0.08 MPa).

Keywords: fire protection; deluge valve; non-pressurized diaphragm; CFD simulation; flow channel optimization

1. Introduction

In the process of modern urbanization, fire is a common and highly destructive disaster, especially in complex environments such as high-rise buildings, underground structures, and large complexes. The rapid spread of fire and the generation of high-temperature smoke make early fire control particularly crucial. High-rise building fires pose unique evacuation challenges due to stack effects [1]; underground structure fires generate toxic smoke that compromises escape routes [2]; and large complex fires exhibit complex flame spread patterns across interconnected spaces [3]. These scenarios collectively demonstrate the critical need for rapid suppression systems [4–6]. Automatic sprinkler systems are widely used in various buildings due to their advantages of simple operation, rapid response, and extensive coverage. Among these, deluge systems have a higher usage rate due to their extensive coverage and high efficiency [7–13]. The deluge valve is the core component of the entire deluge system, controlling the flow of water from the water

Academic Editor: Andrea Carpinteri

Received: 30 December 2025

Revised: 16 February 2026

Accepted: 17 February 2026

Published: 21 February 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 \(CC BY\) license](https://creativecommons.org/licenses/by/4.0/).

supply pipe network into the distribution network. Typically, the valve remains closed and only opens upon detection of a fire signal. With the increasing complexity of building structures and the growing demands for fire safety, traditional deluge valve systems face numerous challenges in terms of response speed, reliability, and intelligence. For example, regarding response speed, delays may occur due to mechanical component wear or corrosion, impacting immediate response [3,6].

Since the 1990s, diaphragm-type deluge valves (Zoning Sprinkler Fire Monitor, ZSFM) have been the most widely used in deluge fire suppression systems [6–8]. The basic structure of these valves, as illustrated in Figure 1, utilizes a rubber diaphragm to separate the pressure chamber from the working chamber, achieving sealing through differential hydraulic forces. Specifically, Deurloo [6] patented a diaphragm-controlled deluge valve mechanism where the diaphragm deformation under water pressure creates the seal. However, this design inherently subjects the diaphragm to uneven pressure distribution: the inner surface and a portion of the outer surface are exposed to water pressure, while the remaining outer surface is not [7,8]. Weingarten [7] further developed this concept by introducing a valve seat drain to mitigate pressure imbalances, yet the fundamental issue of prolonged uneven loading persists. Consequently, as the valve remains normally closed, the diaphragm experiences sustained non-uniform stress, increasing the risk of material fatigue and potential system failure [8,9]. This reliability concern has motivated the development of non-pressurized diaphragm designs, as discussed in the following section.

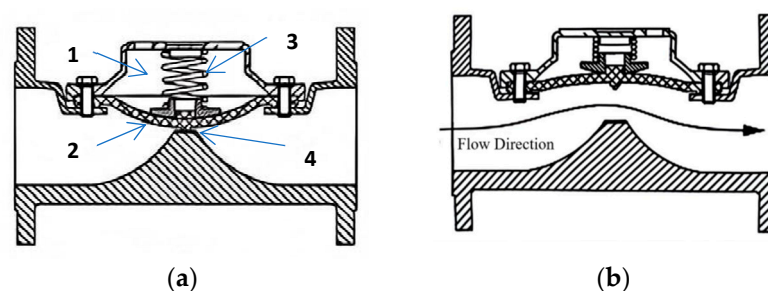


Figure 1. Schematic diagram of the closed and activated states of a ZSFM deluge valve (1—Pressure Chamber; 2—Rubber Diaphragm; 3—Spring; 4—Valve Disc) [6]. (a) Closed state; (b) Activated state.

The hydraulic performance requirements for deluge valves are strictly regulated by international and national standards. The National Fire Protection Association (NFPA) Standard 15 [9] specifies that deluge valves should maintain a pressure drop below 0.08 MPa under rated flow conditions to ensure adequate water supply for fire suppression. Similarly, the Chinese national standard GB 5135.5-2018 [10] establishes comparable performance criteria for automatic sprinkler systems. These standards provide the critical design constraint for the present study: any modification to the traditional diaphragm structure must not compromise the hydraulic efficiency below this threshold. Previous optimization studies on deluge systems, such as the genetic algorithm approach by Andrews and Bartlett [11], have demonstrated the potential of systematic design improvements to enhance system reliability while maintaining regulatory compliance.

The diaphragm functions as a valve disc in deluge valves, its material must possess a variety of properties, including high-pressure resistance, corrosion resistance, elasticity and flexibility, wear resistance, non-toxicity, and safety [9,10,14,15]. Therefore, it is extremely difficult to simultaneously enhance the diaphragm's strength and ensure its resistance to damage under long-term stress. Against this background, non-pressure diaphragm deluge valves have emerged, with their basic structure shown in Figures 1–3.

Traditional disc-shaped rubber diaphragms need to deform into a hemispherical shape under water pressure to separate the inlet and working chambers [16–18]. In contrast, the new design employs a cylindrical valve disc, which utilizes changes in the top rubber structure to achieve valve opening and closing through the vertical movement of the cylindrical valve disc. Consequently, the cylindrical valve disc does not require deformation and can be strengthened by embedding metal sheets within it. This design retains the advantages of ZSFM deluge valves while eliminating the risk of diaphragm failure, thereby enhancing the overall safety of automatic sprinkler systems.

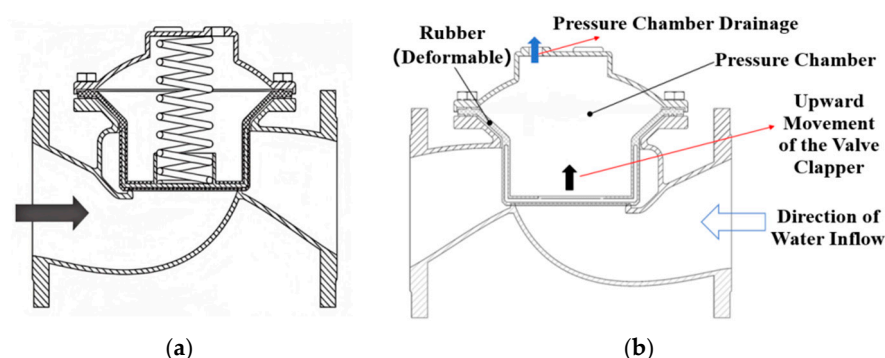


Figure 2. The closed and activated states of a Non-Pressurized ZSFM deluge valve. (a) Closed state; (b) Activated state.

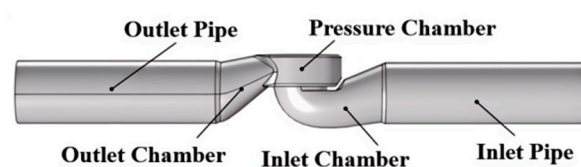


Figure 3. CFD computational domain comprising the valve flow channel and extended straight pipe sections (0.5 m upstream and downstream) to eliminate boundary effects on the flow field.

However, the non-pressurized diaphragm design replaces the original, variable streamlined diaphragm with a fixed planar shape. Consequently, if the remaining flow channel geometry remains unchanged, the flow pressure loss will exceed the Chinese fire protection industry standard of less than 0.08 MPa. Therefore, redesigning the flow channel to effectively optimize the flow path and eliminate unnecessary turbulence and local pressure losses is crucial for improving the overall performance and reliability of deluge fire suppression systems.

This paper focuses on the internal flow channel of a non-pressurized diaphragm deluge valve as the study object. By combining experimental and numerical simulation methods to analyze the flow field within the channel, the geometric structure of the flow channel of the non-pressurized diaphragm deluge valve is optimized to ensure that it can provide the best fluid transport performance in automatic sprinkler systems while meeting fire protection industry standards (≤ 0.08 MPa).

2. CFD Method

The non-pressure diaphragm deluge valve is shown in Figure 2. The main parameters of the original geometric structure are detailed in Table 1.

Table 1. Original geometric structural parameters of the deluge valve.

Inlet Diameter (mm)	Outlet Diameter (mm)	Valve Disc Base Area (mm ²)	Valve Disc Travel (mm)
150	150	25,300	53

When constructing the internal geometric model, the inlet and outlet pipes were extended by 0.5 m to ensure that the inlet and outlet water flows were in a fully developed region, as shown in Figure 3. Water at 25 °C was used as the fluid medium. The inlet boundary condition was set as pressure inlet, and the outlet boundary condition was set as mass flow outlet. The wall was set as a no-slip wall. According to the Chinese fire protection standard manual, the rated inlet pressure for a 150 mm diameter deluge valve is 1.5 MPa, and the rated flow rate is 5300 L/min. The mass flow outlet boundary condition (5300 L/min) is physically appropriate for fire protection systems, which operate at constant pump flow rates. The solver computes the outlet pressure as part of the solution, yielding the pressure drop directly. Pressure outlet was not used because it would specify back pressure rather than flow rate, resulting in a different operating point inconsistent with rated conditions [19]. To ensure accuracy, multiple sets of pressures and flow rates will be used for comparative validation in both the simulation calculations and experimental tests.

The control equations for CFD include the mass conservation equation and the N-S equations [20,21]. The RANS k - ϵ turbulence model was applied. The computational domain was discretized using a free tetrahedral mesh, with local refinement applied to the boundary layer, as shown in Figure 4. COMSOL Multiphysics software was used for the simulation calculations. To verify mesh independence, six numerical models with different mesh numbers were selected and simulated under the same operating conditions, with the results presented in Table 2. Considering computational efficiency and ensuring result accuracy, a mesh quantity of approximately 1,200,000 was chosen.

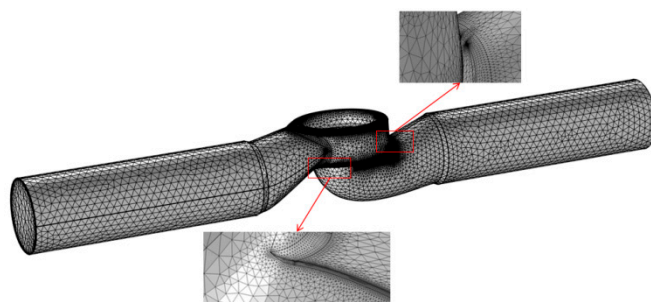


Figure 4. Unstructured tetrahedral mesh (approximately 1.2 million elements) with local refinement in the boundary layer and critical regions (valve seat, inlet/outlet chambers). Insets show detailed mesh distribution near the valve seat and curved inlet channel.

Table 2. Simulated pressure drop at rated conditions with different mesh numbers.

Mesh Number	Pressure Drop (MPa)	Inlet Velocity (m/s)	Outlet Velocity (m/s)
32,313	0.104	4.81576	4.60350
48,259	0.100	4.75578	4.54804
103,205	0.092	4.71646	4.50862
331,696	0.096	4.65973	4.45345
1,196,332	0.095	4.62614	4.42134
2,696,354	0.095	4.62619	4.42157

3. Experimental Method

Figures 5 and 6 present the schematic of the experimental system and photographs of the experimental setup, respectively. The main components of the experimental system include a water reservoir, a centrifugal pump, an accumulator tank, a gate valve, the test deluge valve, and various devices for collecting experimental data, such as pressure gauges and data acquisition instruments. During the experiment, water is supplied by a

centrifugal pump, flows to the deluge valve after passing through an accumulator tank, and then returns to the reservoir through piping. To test the pressure loss of the deluge valve under multiple different operating conditions, two gate valves are used to control the test valve's different inlet flow rates and pressures by adjusting their opening degrees, thereby establishing various inlet conditions. The data acquisition system comprises a differential pressure gauge to obtain the pressure loss of the test valve under different inlet conditions, along with a flow meter and a pressure gauge to monitor the inlet pressure and flow rate of the test valve. We utilized three centrifugal pumps operating simultaneously. The parameters for a single centrifugal pump are a flow rate of 40 L/s, a head of 160 m, and a motor power of 90 KW. Each operating condition was tested three times, and the reported pressure drops represent the arithmetic mean of these repeated measurements.

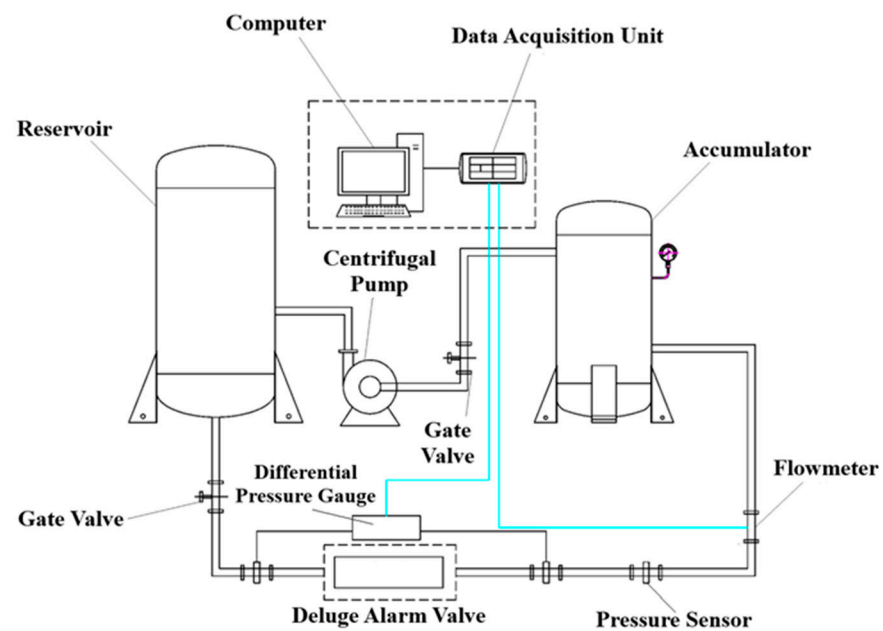


Figure 5. Schematic of the experimental system.

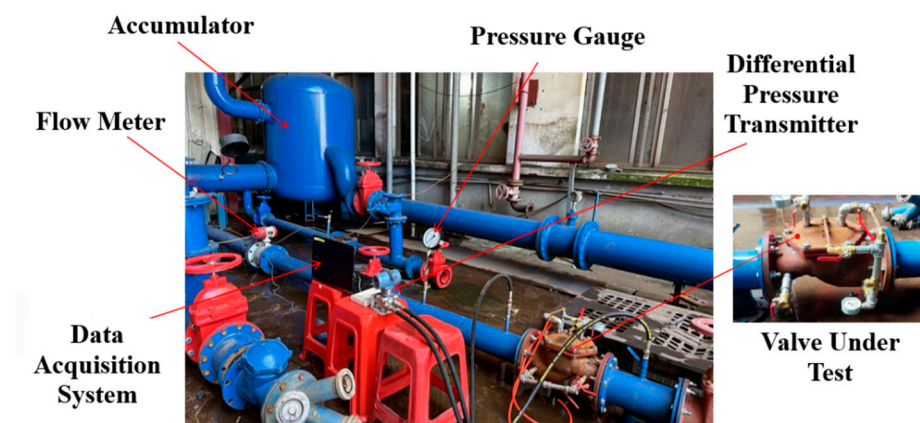


Figure 6. Photographs of the experimental setup.

The uncertainty of the Reynolds number is calculated using the error propagation formula [22]. For a function $Re = f(v, \rho, d_h, \mu)$, the combined standard uncertainty is given by:

$$\delta Re = \sqrt{\sum_{i=1}^n \left(\frac{\partial Re}{\partial X_i} \delta X_i \right)^2} \quad (1)$$

where X_i represents the independent variables (v , ρ , d_h , μ), and δX_i is the standard uncertainty of each variable. This formula is derived from the first-order Taylor series expansion and is widely adopted in experimental fluid mechanics.

Substituting $Re = \rho v d_h / \mu$ into the above equation yields:

$$\frac{\delta Re}{Re} = \sqrt{\left(\frac{\delta v}{v}\right)^2 + \left(\frac{\delta \rho}{\rho}\right)^2 + \left(\frac{\delta d_h}{d_h}\right)^2 + \left(\frac{\delta \mu}{\mu}\right)^2} \quad (2)$$

In this study, the uncertainties of density (ρ) and hydraulic diameter (d_h) are negligible compared to velocity and viscosity measurements. Therefore, Equation (1) is simplified to consider only the dominant contributions from velocity and dynamic viscosity.

The pressure drop and flow velocity are measured multiple times during the experiment. The standard deviation of these measured values is calculated to estimate the uncertainty, using the following equation:

$$S = \sqrt{\frac{1}{n-1} \sum_{i=1}^n (X_i - \bar{X})^2} \quad (3)$$

where $\bar{X}$ is the average of the measured values:

$$\bar{X} = \frac{1}{n} \sum_{i=1}^n X_i \quad (4)$$

The calculated results of the uncertainties based on the above equations are shown in Table 3.

Table 3. Uncertainty.

Variable	Volumetric Flow Rate G	Pressure Drop Δp	Reynolds Number Re
Uncertainty	$\pm 1.9\%$	$\pm 1.1\%$	$\pm 2.5\%$

4. Model Verification

To determine which turbulence model is suitable for this study, simulations were performed using the RANS k- ϵ , RANS k- ω , and Realizable k- ϵ turbulence models under the same experimental conditions. The inlet pressure was kept constant at the rated pressure of 1.5 MPa, and different simulation results were obtained by varying the inlet flow rate. These results were then compared with the experimental data and are given in Table 4. The results indicate that the RANS k- ϵ model's results are closer to the experimental results compared to the RANS k- ω and Realizable k- ϵ models. Turbulence model selection was based on quantitative comparison of mean pressure drop prediction (Mean Absolute Percentage Error-MAPE, R^2). While k- ω SST is theoretically preferred for strong curvature [23,24], it overpredicted separation severity in the valve seat region, resulting in inferior overall ΔP accuracy (MAPE 14.3% vs. 7.2%). The standard k- ϵ model better captured the jet-dominated loss mechanisms despite its known curvature limitations [25]. Therefore, the RANS k- ϵ model will continue to be used for subsequent numerical calculations.

Table 4. Experimental results of varying inlet flow rate for the deluge valve.

Inlet Flow Rate (L/s)	Inlet Pressure (MPa)	Outlet Pressure (MPa)	Pressure Drop (MPa)
88	1.5	1.395	0.105

78	1.5	1.416	0.084
68	1.5	1.4	0.070
58	1.5	1.445	0.055
48	1.5	1.465	0.035

By varying the inlet flow rate and inlet pressure of the test deluge valve, Figure 7 presents the simulation and experimental error results for the pressure drop as it changes under different inlet conditions. As observed in Figure 7, the trend of pressure drop with respect to variations in inlet pressure and flow rate is relatively consistent between the numerical simulations and the experiments, with errors within 10%. Therefore, the numerical simulation model established in this paper meets the engineering-acceptable requirements for subsequent study.

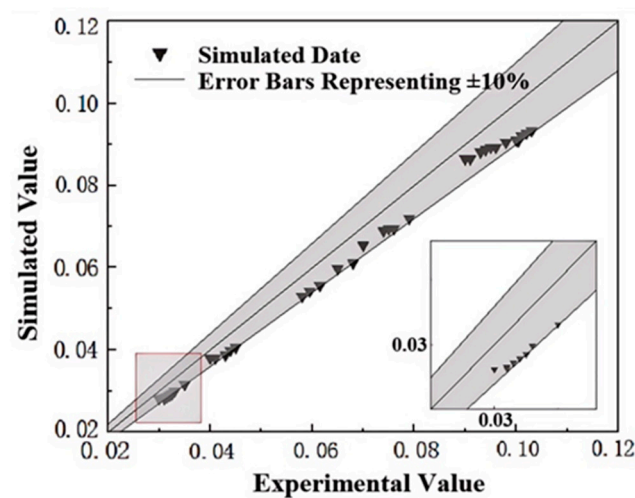


Figure 7. Comparison of simulated and experimental pressure drops across the deluge valve at an inlet pressure of 1.5 MPa and varying flow rates (88–48 L/s). The diagonal line represents engineering-acceptable agreement, and the shaded region indicates $\pm 10\%$ error bounds ($R^2 = 0.987$, slope = 0.94).

Notably, pressure drops are calculated as the difference between area-weighted average static pressures at the inlet and outlet boundaries, obtained using COMSOL's built-in integration operators. While the software reports values with high numerical precision (6–8 decimal places) due to double-precision arithmetic, we emphasize that (i) experimental measurements cannot achieve such precision, and (ii) the physically meaningful accuracy of CFD results is limited to approximately three significant figures based on model validation uncertainty ($\pm 10\%$). All reported values are accordingly rounded to reflect this physical reality.

5. Results and Discussion

Minor changes in the geometry of the flow channels within the deluge valve's pressure chamber can result in significant alterations in the entire flow field. This influence is ultimately amplified and reflected in the level of pressure drop. This is the reason why the original structure of the deluge valve is no longer suitable for the non-pressurized-bearing deluge valve, which has undergone a complete change in the valve disc diaphragm structure, leading to excessive flow pressure drop that does not meet industry practice standards. Among these, the curvature R of the bend in the valve inlet flow channel, the height H of the valve seat, the length L of the outlet chamber, and the inclination angle Θ of its bottom surface have the most significant impact on pressure drop, as shown in Figure 8. Therefore, based on the original structure and considering the aforementioned factors, we

have designed six structural optimization cases, with the modified structural dimensions listed in Table 5. The six optimization cases were designed using a sequential adaptive strategy rather than full factorial design. Stage I (Cases 1–3) identified curvature radius as the dominant parameter. Stage II (Case 4) tested thickness reduction at near-optimal radius. Stage III (Cases 5–6) fine-tuned outlet geometry with established radius and thickness. This approach efficiently allocated computational resources to the most influential parameters, as informed by preliminary sensitivity analysis. The simulation results for the inlet and outlet pressure drop are given in Table 6.

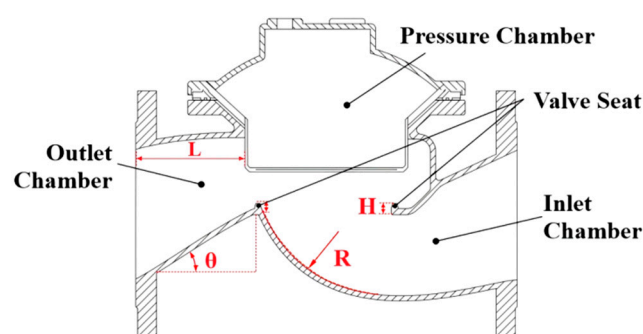


Figure 8. Schematic diagram of four geometric factors significantly affecting pressure drop.

Table 5. Configuration of six geometric optimization cases.

Case	1	2	3	4	5	6
Curvature Radius (m)	0.1	0.115	0.15	0.165	0.165	0.17
Seat Thickness (m)	0.01	0.01	0.01	0.0065	0.0065	0.0065
Outlet Chamber Length (m)	0.135	0.135	0.13	0.135	0.15	0.15
Outlet Chamber Inclination Angle (°)	34	34	35	34	30	30

Table 6. Simulation pressure drop of non-pressurized ZSFM under identical inlet conditions.

Case	Original Structure	1	2	3	4	5	6
Inlet Flow Rate (L/min)	5300	5300	5300	5300	5300	5300	5300
Inlet Pressure (MPa)	1.5	1.5	1.5	1.5	1.5	1.5	1.5
Pressure Drop (MPa)	0.095	0.078	0.086	0.052	0.050 ⁺	0.050 ⁺	0.052

Figure 9 shows the streamline diagrams of the simulation results for six different optimization schemes. In case 1, reducing the radius of curvature of the bent inlet flow channel led to a decrease in pressure drop. However, this design induced multiple small-scale vortices on the bottom surface of the bent flow channel, reflecting phenomena of local flow separation and instability, thus reducing flow stability. Additionally, recirculation within the outlet chamber persisted, further diminishing the orderliness of the flow field and resulting in poorer overall flow stability. Case 2 did not alter the curvature of the bent inlet flow channel; instead, it adjusted the length and inclination angle of the outlet chamber. Although the modification of the outlet chamber shape aimed to optimize the flow, the actual results showed increased pressure drop and intensified recirculation, indicating that this scheme has poor adaptability in flow control and a further exacerbation of flow instability within the outlet chamber. Case 3 increased the radius of curvature of the bent outlet flow channel and optimized the shape and inclination angle of the outlet flow channel, as well as the valve seat structure, while also reducing the overall height of the valve. These improvements effectively reduced the pressure drop and mitigated the recirculation phenomenon in the outlet chamber, leading to a significant enhancement in flow stability. However, some vortices still existed within the valve chamber, indicating that local

flow separation has not been completely eliminated. Case 4 further increased the radius of curvature of the outlet chamber based on Case 3, while also reducing the height of the pressure chamber to narrow the valve chamber space and adjusting the valve seat structure. Although these adjustments slightly reduced the pressure drop and decreased localized high-velocity regions, they induced new vortices within the valve chamber, suggesting that further compression of space might have introduced new sources of instability. In case 5, building upon case 4, extending the outlet chamber and adjusting the inclination angles of the upper and lower surfaces further reduced the pressure drop, albeit with a limited reduction, indicating a diminishing marginal benefit of this improvement in reducing energy losses. Case 6, on the other hand, involved reducing the gap between the right side of the valve chamber and the pressure chamber, along with a slight adjustment of the outlet chamber's inclination angle. This minor adjustment led to a slight increase in pressure drop and an increase in vortices within the valve chamber, consequently reducing flow stability. This indicates that this scheme performs poorly in balancing pressure drop and flow stability.

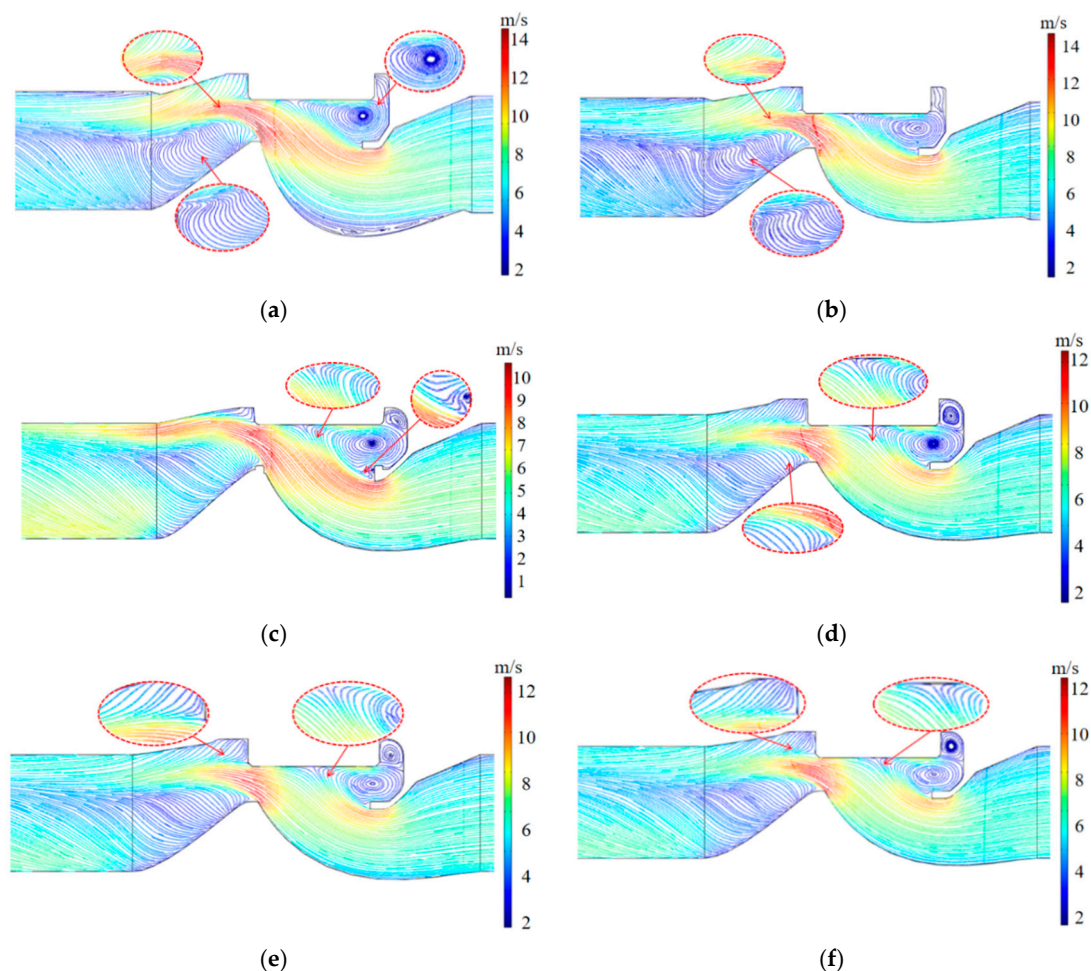


Figure 9. Velocity streamlines in the valve flow channel. (a) Case 1; (b) Case 2; (c) Case 3; (d) Case 4; (e) Case 5; (f) Case 6.

To further clarify the mechanisms of drag reduction in depth, Figure 10 compares the pressure fields across six redesigned valve cavities. Case 1 exhibits steep pressure gradients in the inlet bend and outlet regions, indicating high local resistance and imminent flow separation. Case 2 further elevates pressure loss and degrades uniformity, as an expanded high-pressure zone near the outlet cavity intensifies backflow and vorticity. Case 3 reshapes the flow path to slash pressure loss and homogenize the distribution, yet a low-

pressure pocket on the right side of the valve cavity approaches atmospheric pressure, suggesting potential for cavitation inception if local conditions degrade. Cavitation assessment in this study is based on static pressure fields only. Full cavitation modeling (two-phase flow, bubble dynamics) was not performed, as the primary design criterion (pressure drop < 0.08 MPa) is satisfied with adequate margin in all cases. Building on this, case 4 again reduces pressure loss and suppresses high-pressure regions, but the enlarged low-pressure domain keeps the cavitation threat alive. Case 5 achieves the smoothest pressure profile to date, with the lowest loss; only an extreme point in the outlet cavity retains a slight cavitation potential, making it the optimal choice for minimizing energy dissipation. Case 6, by contrast, shows the low-pressure zone expanding and gradients steepening, compromising stability and cavitation suppression; although its pressure loss remains acceptable, it fails to continue the advantages of case 5. In summary, case 5 delivers the best design, combining minimal pressure loss with the most uniform pressure distribution while maintaining flow stability.

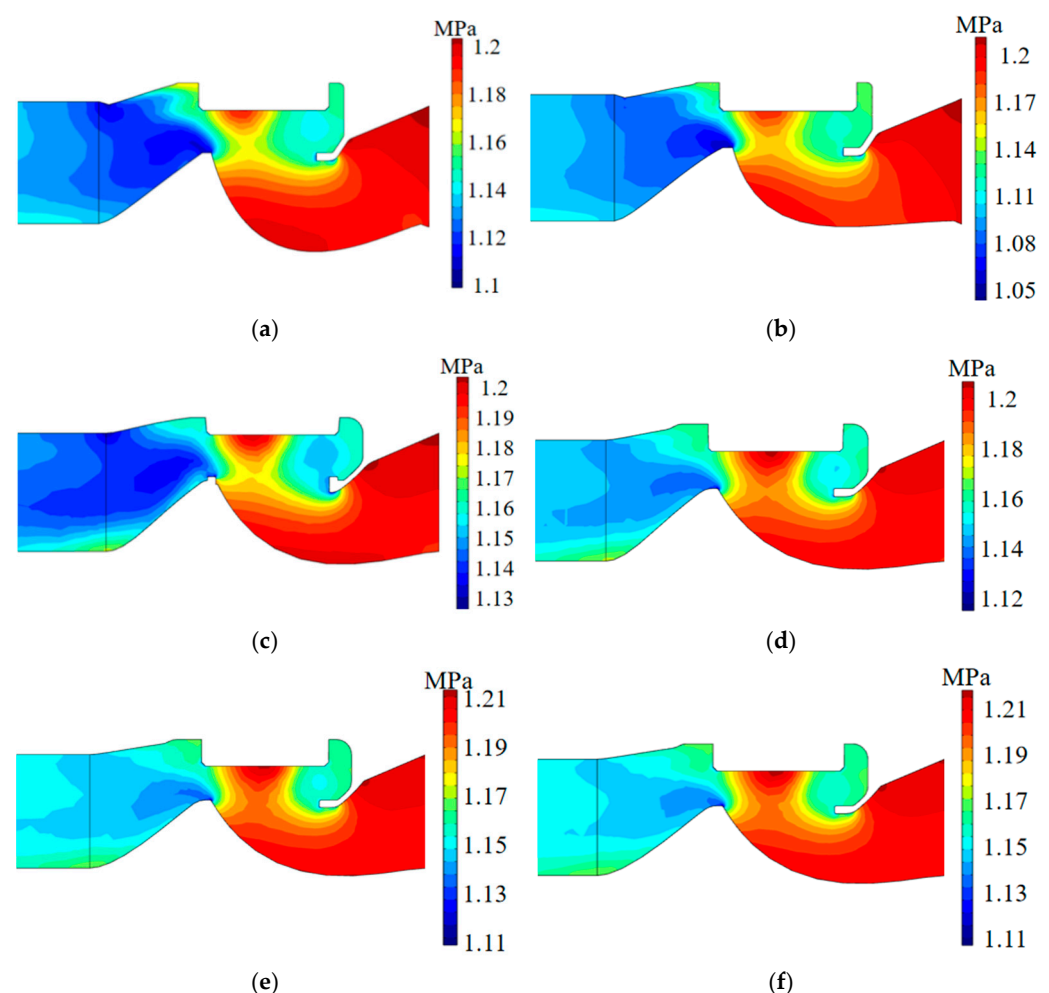
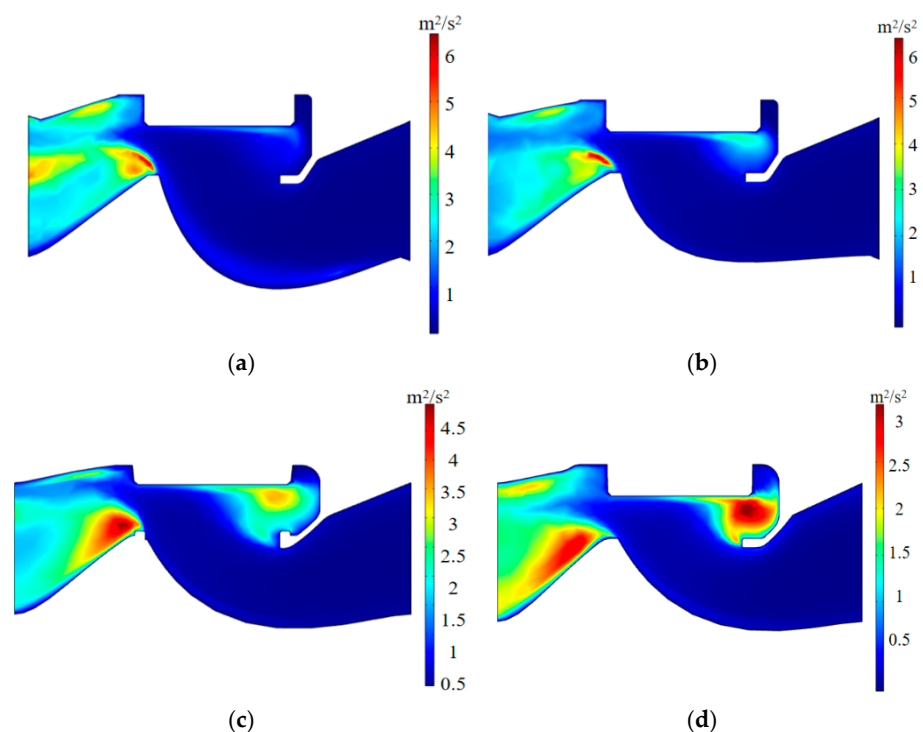


Figure 10. Pressure contour in the valve flow channel. (a) Case 1; (b) Case 2; (c) Case 3; (d) Case 4; (e) Case 5; (f) Case 6.

Figure 11 shows the turbulent kinetic energy distribution in the valve channel for six different modification schemes under the same boundary conditions, revealing the turbulence intensity of each case and its impact on flow stability. The turbulent kinetic energy distribution in case 1 shows higher turbulence intensity in the inlet curved region and near the outlet cavity, indicating significant energy losses in these areas. Furthermore, turbulence disturbances are intensified, easily leading to local flow separation and vortices,

which decrease flow stability and exacerbate the complexity of the flow field. Case 2 is similar to case 1, but the turbulence intensity in the outlet cavity further increases, especially in the recirculation zone. The elevated turbulent kinetic energy in this region signifies increased energy losses and more chaotic local flow, further weakening flow stability. This turbulence diffusion can lead to more complex flow structures and higher pressure drops. In case 3, the turbulent kinetic energy on the right side of the valve cavity is lower, indicating that the flow in this region tends to be stable with less energy loss. However, the turbulent kinetic energy in the outlet recirculation zone remains high, showing that strong local turbulence still exists in this area. Although overall flow stability has improved, local energy losses and turbulence disturbances are still evident. Case 4 is relatively uniform, with fewer high turbulent kinetic energy regions compared to case 3. This suggests that the optimized design has effectively reduced turbulence intensity and improved the overall flow field stability. Although turbulence within the valve cavity is relatively dispersed, the turbulent kinetic energy near the outlet cavity remains concentrated, and the issue of local energy loss still requires attention. In case 5, the turbulent kinetic energy distribution is further improved, with reduced concentration in high turbulent kinetic energy regions, leading to a significant enhancement in overall flow stability. While the turbulent kinetic energy in local turbulent regions remains high, the overall energy loss is reduced compared to the preceding schemes, making the flow channel structure more efficient. Case 6 is similar to case 5, but the turbulent regions are slightly expanded, particularly the turbulence intensity in the outlet cavity has increased. This increase in turbulent kinetic energy indicates a slight decrease in flow stability and potentially increased energy losses. Therefore, the overall flow characteristics of Scheme Case 6 are inferior to those of Case 5.



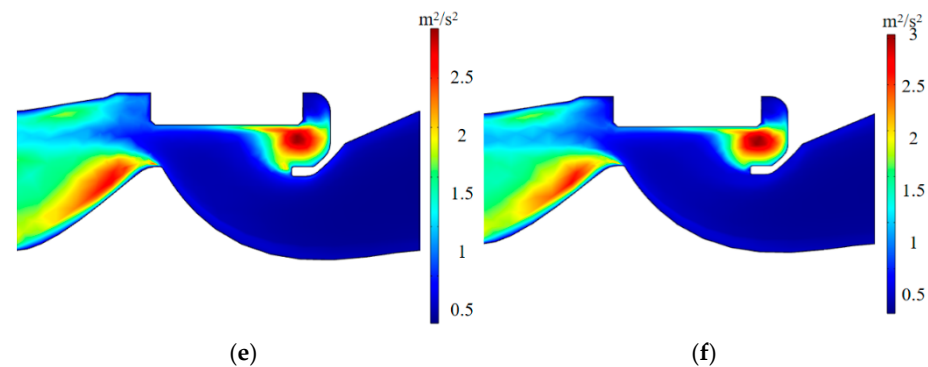


Figure 11. Turbulent kinetic energy contours in the valve flow channel. (a) Case 1; (b) Case 2; (c) Case 3; (d) Case 4; (e) Case 5; (f) Case 6.

Although flow stabilization in Case 5 is indicated by reduced vorticity concentration and more uniform streamline patterns (Figures 9–11), quantitative stability metrics, such as sectional loss coefficients and coherent structure identification, are recommended for future work to provide formal engineering proof of stability improvement [22,23].

The reported optimal parameters are nominal design targets. Sensitivity analysis indicates that $\pm 5\%$ geometric variations result in $< 5\%$ pressure drop change, providing adequate margin against manufacturing tolerances. Surface roughness typical of cast steel construction increases ΔP by $< 3\%$, remaining within the 0.08 MPa criterion.

6. Conclusions

Fabricating numerous non-pressurized deluge alarm valve cavities under a multi-factor structural variation and relying on extensive experimental comparisons would unduly prolong the R&D cycle and generate prohibitive tooling costs. Instead, this study combines targeted experiments with CFD to dissect the internal flow of non-pressurized deluge valves and to optimize the flow channel geometry for drag reduction. The developed CFD model agrees with measured data within $\pm 10\%$, confirming its reliability. Setting an inlet-to-outlet pressure drop below 0.08 MPa as the design criterion, the investigation identifies the optimal configuration as a curvature radius of 0.165 m, a seat thickness of 0.0065 m, an outlet-chamber length of 0.15 m and an outlet-chamber inclination of 30° , at which the pressure drop is reduced from 0.095 MPa to 0.050 MPa. The optimum is validated at the rated condition (1.5 MPa, 5300 L/min, 25°C). While preliminary CFD suggests robustness to pressure variations (1.0–2.0 MPa), full experimental validation across temperature and extended pressure ranges is ongoing future work.

Beyond the practical engineering outcomes, we demonstrate a systematic approach combining validated CFD modeling with targeted experimental verification to solve inverse design problems in valve flow channels. Unlike previous studies that treated valve optimization as single-parameter sensitivity analysis, our work establishes a multi-factor coupling methodology that identifies parameter interactions (e.g., the synergistic effect between curvature radius and outlet chamber length) that would be inaccessible through conventional one-factor-at-a-time approaches. This framework is transferable to other valve types where geometric constraints conflict with hydraulic performance requirements.

Author Contributions: Conceptualization, T.Z. and J.Z. (Jieqing Zheng); methodology, Y.Z. and W.S.; software, Y.Z. and J.W.; validation, Y.Z., J.W. and Z.Y.; formal analysis, Y.Z. and J.Z.; investigation, Y.Z., J.W. and J.Z. (Jinhao Zhang); resources, T.Z.; data curation, Y.Z., J.W. and Z.Y.; writing—original draft preparation, Y.Z.; writing—review and editing, T.Z. and J.Z. (Jieqing Zheng);

visualization, Y.Z. and Z.C.; supervision, T.Z. and J.Z. (Jieqing Zheng); project administration, J.Z. (Jinhao Zhang); funding acquisition, W.S. All authors have read and agreed to the published version of the manuscript.

Funding: This research was funded by the Natural Science Foundation of Xiamen (3502ZZ202571044), the Natural Science Foundation of Fujian Province (2025J01873) and the Scientific Research Foundation of JiMei University (ZQ2020013).

Data Availability Statement: The data presented in this study are available upon request from the corresponding author.

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

References

1. Liu, Z.Q.; He, H.Z.; Zheng, J.Q.; Huang, Y.J.; Zhuang, H.H.; Chen, Y.G.; Lai, D.M. Effect of ventilation ducts on smoke spread between two adjacent cabins arranged along a corridor: An experimental and numerical investigation. *Int. Commun. Heat Mass Transfer* **2024**, *159*, 108245.
2. Hystad, P.W.; Keller, P.C. Towards a destination tourism disaster management framework: Long-term lessons from a forest fire disaster. *Tourism Manag.* **2008**, *29*, 151–162.
3. Zhang, Y.X.; Zhang, X.N.; Huang, X.Y. Design a safe firefighting time (SFT) for major fire disaster emergency response. *Int. J. Disaster Risk Reduct.* **2023**, *88*, 103606.
4. Kim, D.; Cha, H.; Jiang, S. The prediction of fire disaster using BIM-based visualization for expediting the management process. *Sustainability* **2023**, *15*, 3719.
5. Robinet, A.; Chetehouna, K. A review of additives for water mist fire suppression systems. *Fire Technol.* **2024**, *60*, 2923–2961.
6. Deurloo, D. Dry pipe/deluge valve for automatic sprinkler systems. U.S. Patent 7,673,695, 10 March 2010.
7. Weingarten, Z. Deluge valve with valve seat drain. U.S. Patent 9,803,755, 31 October 2017.
8. Andrews, J.D.; Bartlett, L.M. Genetic algorithm optimization of a firewater deluge system. *Qual. Reliab. Eng. Int.* **2003**, *19*, 39–52.
9. NFPA 15; Standard for Water Spray Fixed Systems for Fire Protection, 2019th ed. National Fire Protection Association: Quincy, MA, USA, 2019.
10. GB 5135.5-2018; Automatic Sprinkler System—Part 5: Deluge Alarm Valves. Standardization Administration of China: Beijing, China, 2018.
11. Ingason, H.; Li, Y.Z.; Arvidson, M.; Jiang, L. Fire tests with automatic sprinklers in an intermediate scale tunnel. *Fire Saf. J.* **2022**, *129*, 103567.
12. He, Q.Q.; Liu, N.A.; Xie, X.D.; Zhang, L.H.; Lei, J.; Zhang, Y.; Wu, D. Experimental study on fire spread over discrete fuel bed—Part II: Combined effects of wind and packing ratio. *Fire Saf. J.* **2022**, *128*, 103520.
13. Arvidson, M.; Westlund, Ö. The Development of Automatic Sprinkler System Concepts for Maritime Vehicle Carriers. *Fire Technol.* **2024**, *60*, 2125–2153.
14. Li, X.; Wan, W.; Zhao, Y.; Bai, G.; Shi, X.; Chen, B.; Zhang, Y. Feasibility assessment of a clean and efficient fire extinguishing system for pottery jar liquor warehouses. *Sci. Rep.* **2024**, *14*, 13449.
15. Berardi, D.; Galuppi, M.; Lombardi, M. Experimental Study on the Effectiveness of Water Mist Extinguishing Systems for Protecting Paintings in Art Gallery Museums: Developing an Initial Methodological Model. *Buildings* **2023**, *13*, 1806.
16. Till, R.C.; Coon, J.W. Sprinkler Systems and Their Types. In *Fire Protection: Detection, Notification, and Suppression*; Springer International Publishing: Cham, Switzerland, 2018; pp. 89–124.
17. Lai, C.M.; Ho, M.C.; Chen, C.J.; Tsai, M.J.; Lin, T.H. Experimental investigation of an office fire with a partially impaired sprinkler system. *Fire Technol.* **2010**, *46*, 611–627.
18. Cablé, A.; Chetehouna, K.; Gascoin, N.; Settari, A.; Van Herpe, K.; Kadoche, M. Experimental and numerical study of sprinkler fire protection systems: Effect of initial pressure on system performance. *J. Loss Prev. Process Ind.* **2019**, *60*, 76–86.
19. Liatsikouras, A.G.; Pierrot, G.; Megahed, M. A coupled CAD-free parameterization-morphing method for adjoint-based shape optimization. *Int. J. Numer. Methods Fluids* **2022**, *94*, 1745–1763.
20. Lisowski, E.; Filo, G. Analysis of a proportional control valve flow coefficient with the usage of a CFD method. *Flow Meas. Instrum.* **2017**, *53*, 269–278.

21. Chu, S.; Chen, T.; Gan, Y.; Liu, Y.; Zheng, W.; Tang, Y.; Zhou, W. Numerical Simulation and Experimental Study of Gas–Solid Two-Phase Spraying of Dry Powder Fire-Extinguishing System Based on Fire-Extinguishing Inspection Robot. *Processes* **2024**, *12*, 1239.
22. Celik, I.B.; Ghia, U.; Roache, P.J.; Freitas, C.J. Procedure for estimation and reporting of uncertainty due to discretization in CFD applications. *J. Fluids Eng.* **2008**, *130*, 078001.
23. Menter, F.R. Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications. *AIAA J.* **1994**, *32*, 1598–1605.
24. Asnaghi, A.; Svennberg, U.; Bensow, R.E. Evaluation of curvature correction methods for tip vortex prediction in SST $k-\omega$ turbulence model framework. *Int. J. Heat Fluid Flow* **2019**, *75*, 135–152.
25. Huang, X.; Yang, W.; Li, Y.; Qiu, B.; Guo, Q.; Liu, Z. Review on the sensitization of turbulence models to rotation/curvature and the application to rotating machinery. *Appl. Math. Comput.* **2019**, *341*, 46–69.

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.