

Article

Topology Optimization of the Internal Flow Domain of a Transformer Pressure Relief Valve

Pengfei Bao ¹, Ke Wang ² , Jiayi Li ^{2,*}, Yikun Zhao ² and Mingyao Yao ¹ ¹ School of Automobile and Transportation Engineering, Hefei University of Technology, Hefei 230009, China; q1135275698@163.com (P.B.); yaomingyao@126.com (M.Y.)² China Electric Power Research Institute, Beijing 100192, China; wangke1@epri.sgcc.com.cn (K.W.); zhaoyikun@epri.sgcc.com.cn (Y.Z.)

* Correspondence: lijiaxi2022@yeah.net

Abstract

Pressure relief valves (PRVs) are key safety components in power transformers, and their discharge performance directly influences pressure mitigation during internal fault events. This study presents a density-based topology optimization approach to improve the internal flow passage of a transformer PRV. Optimization is carried out within a predefined design domain while preserving the original external geometry and functional constraints of the valve. Pressure drop is defined as the objective function, and the Brinkman penalization method is used to model the fluid–solid transition during the optimization process. Based on the optimized topology, a reconstructed valve configuration is developed and evaluated using transient computational fluid dynamics (CFD) simulations. The transient inlet pressure boundary condition is obtained from a 6 MJ transformer arcing experiment to ensure realistic operating conditions. The hydraulic performance of the baseline and optimized designs is compared in terms of cumulative discharged volume, pressure field evolution, velocity distribution, and turbulent kinetic energy. The results indicate that the optimized flow passage reduces internal flow resistance and improves flow organization within the valve chamber under transient conditions. Over a 19 ms discharge period, the cumulative discharged volume increases from 13.219 L to 13.601 L, corresponding to an improvement of approximately 3%. These findings demonstrate that topology optimization can effectively enhance the transient discharge performance of transformer PRVs and provide a basis for improving internal flow design in similar hydraulic safety devices.

Keywords: transformer pressure relief valve; topology optimization; transient discharge; computational fluid dynamics; hydraulic loss reduction; flow-field optimization



Academic Editor: Andrea
Gaetano Chiariello

Received: 24 June 2026

Revised: 15 July 2026

Accepted: 21 July 2026

Published: 23 July 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

During the service life of oil-immersed electrical equipment such as power transformers and on-load tap changers, low-impedance faults can occur due to aging insulation and changes in the physical and chemical properties of the insulating oil [1]. These faults often give rise to high-energy arcs, which rapidly decompose the surrounding transformer oil into a mixture of flammable gases, including hydrogen, acetylene, and ethylene [2]. The intense heat generated by the arc also causes the oil to vaporize almost instantaneously, producing a sudden volumetric expansion. Given the high density and inertia of the liquid phase, the oil cannot adjust immediately to this rapid expansion, leading to localized pressure spikes within gas bubbles [3]. The resulting pressure wave propagates through

the oil, interacting with solid components inside the transformer, reflecting and refracting multiple times before contributing to an overall rise in tank pressure [4]. If this excess pressure is not quickly relieved, it can impose severe mechanical stress on the tank walls, potentially causing deformation, oil leakage, or even catastrophic rupture, posing serious risks to both equipment and operational safety.

Installed as a final critical safety barrier on transformer tanks, PRVs are widely used to mitigate overpressure risks from internal arc faults [5]. Their typical operation relies on a spring-loaded disc mechanism: the disc lifts to open a relief passage when internal pressure exceeds a preset threshold and resets once pressure drops below the reseating value, enabling rapid pressure regulation and isolation [6,7]. Extensive practical evidence confirms that high-performance PRVs substantially reduce explosion risks associated with arc faults [8,9].

Previous investigations on transformer PRVs have primarily focused on opening pressure characteristics, spring stiffness design, dynamic response behavior, and structural reliability. Experimental and numerical studies have demonstrated that valve opening dynamics considerably influence pressure-relief effectiveness and discharge performance [10–12]. Similar research has also been conducted on safety valves and relief devices used in hydraulic, petrochemical, and process engineering systems, where valve motion, flow-induced forces, structural vibration, and dynamic stability have been identified as critical factors affecting operational performance [13,14]. Recent studies have further expanded the investigation of hydraulic valve performance from conventional structural design to dynamic operating characteristics under complex working conditions. For example, Stosiak et al. investigated the influence of mechanical vibration on a micro-hydraulic relief valve and analyzed the resulting pressure pulsation characteristics through theoretical and experimental approaches [15]. Another study combined experimental measurements with numerical analysis to investigate vibration-related behaviors of hydraulic valves, further improving the understanding of valves' dynamic characteristics under practical operating conditions [16]. These studies have considerably improved the understanding of valve mechanics and dynamic response characteristics.

However, compared with the extensive research on valve opening mechanisms, considerably less attention has been devoted to the internal flow organization during transient pressure-relief processes. In practical transformer PRVs, the discharge flow commonly experiences abrupt contractions, sharp turning passages, and sudden expansions before reaching the outlet. Such complex geometric features can induce flow separation, recirculation zones, vortex structures, and localized pressure losses, thereby reducing the effective conversion of pressure energy into discharge flow [17]. Previous numerical investigations of safety valves and pressure-relief devices have shown that vortex formation and flow stagnation regions may substantially increase hydraulic losses and weaken discharge performance [18,19]. Therefore, improving the internal flow-path configuration represents a promising strategy for enhancing transient discharge efficiency without altering the fundamental operating mechanism of the valve.

Conventional geometric modifications generally rely on engineering experience and iterative parameter adjustments, which may fail to identify globally optimal flow configurations. In recent years, topology optimization has emerged as a powerful computational design methodology for fluid systems. Unlike traditional size or shape optimization techniques, topology optimization enables material distribution to evolve freely within a prescribed design domain, allowing the automatic generation of non-intuitive flow-guiding structures that are difficult to obtain through conventional engineering intuition [20].

Since the seminal work of Borrvall and Petersson [21], density-based topology optimization has become an effective computational design methodology for fluid systems,

enabling the automatic generation of optimized flow paths through material redistribution within a prescribed design domain. Olesen et al. [22] extended the approach to Navier–Stokes flow problems and demonstrated its capability to improve hydraulic performance in fluidic devices. Subsequent developments in topology optimization theory and numerical methods have considerably expanded their applicability to engineering design problems [23]. More recently, topology optimization has been successfully applied to channel-flow systems [19], unsteady flow configurations [24], turbulent flow problems [25], thermal management devices and heat exchangers [26], as well as proton-exchange membrane fuel cell flow fields [27]. These studies have consistently shown that topology optimization can identify non-intuitive flow-guiding structures and achieve substantial reductions in hydraulic losses, offering design solutions that are difficult to obtain through conventional geometric modification and engineering experience alone.

Despite the growing application of topology optimization in fluid engineering, its use in transformer pressure relief valves remains limited. Existing studies have primarily focused on valve actuation and structural design, while the role of internal flow-path configuration in determining transient discharge performance has received comparatively little attention. Therefore, the objective of this study is to investigate the feasibility and effectiveness of applying density-based topology optimization to the internal flow domain of a transformer pressure relief valve. The optimization problem is formulated to minimize hydraulic resistance while preserving the external geometry and functional constraints of the original valve. The topology optimization result is reconstructed into a manufacturable configuration and subsequently evaluated using transient computational fluid dynamics (CFD) simulations. Particular attention is devoted to transient discharge behavior, cumulative venting capacity, pressure evolution, velocity distribution, and flow-field characteristics.

2. Materials and Methods

2.1. Valve Geometry

Pressure relief valves usually contain elastic components, sealing elements, and a housing. The elastic component is preloaded, applying enough force to the valve plate to press it through a sealing element to close off the oil tank. When the pressure from the tank rises, the sealing element rises against the elastic component to relieve the pressure. After that, the elastic component provides force to the sealing element, allowing it to move back and close off the oil tank again. In our previous work, a symmetrical dual-outlet housing PRV was designed [28]. The present work builds upon this novel valve configuration, as shown in Figure 1.

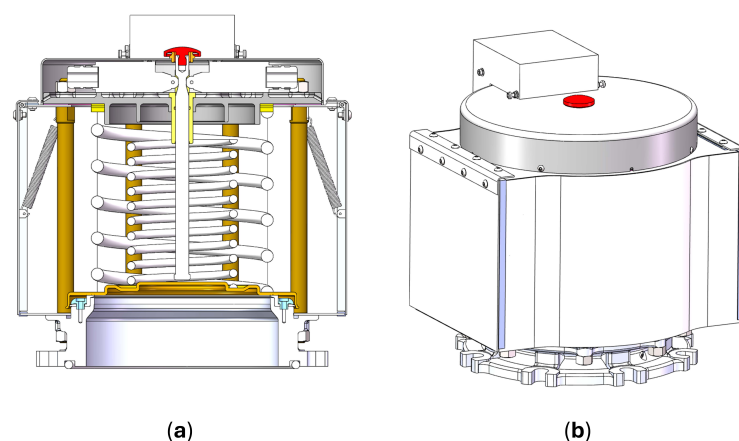


Figure 1. (a) Section view of the novel valve; (b) axonometric view of the novel valve.

2.2. Governing Equations

In topology optimization of the pressure relief valve, the fluid motion is described by the incompressible Navier–Stokes equations [29]:

$$\nabla \cdot \mathbf{u} = 0 \quad (1)$$

$$\rho (\mathbf{u} \cdot \nabla) \mathbf{u} = -\nabla p + \mu \nabla^2 \mathbf{u} - \alpha \mathbf{u} \quad (2)$$

Here $\mathbf{u}$ denotes the velocity vector, p is the pressure, ρ is the fluid density, and μ is the dynamic viscosity.

The design domain is treated as a combination of fluid and solid regions, with a density-based representation applied throughout the optimization. To account for the effect of solid regions on the fluid, volumetric resistance proportional to the velocity is included. This is implemented through the Brinkman penalization method [30], which introduces the term $-\alpha \mathbf{u}$ into the momentum equation. The coefficient α is linked to the design variable γ using a Darcy-type interpolation function, providing a smooth transition between fluid and solid regions [31]:

$$\alpha(\gamma) = (\alpha_{\max} - \alpha_{\min}) \frac{q(1-\gamma)}{q+\gamma} \quad (3)$$

$$\alpha_{\max} = \frac{\mu}{Da \cdot L^2} \quad (4)$$

where q is the penalty factor of the interpolation model (set to 0.01); $\alpha_{\min}$ is set to 0; $\alpha_{\max}$ is a large constant; L represents the characteristic length of the system; and $\alpha(\gamma)$ serves as the volumetric resistance coefficient applied to the fluid.

In the density-based topology optimization framework, the maximum flow resistance coefficient $\alpha_{\max}$ is determined from the dimensionless Darcy number (Da), which characterizes the relative influence of viscous forces and flow resistance in the porous medium. To approximate an impermeable solid, Da should be sufficiently small, typically less than 10^{-5} in practical applications [30]. During the optimization, the design variable γ varies continuously between 0 and 1, controlling the local material properties. The transition of material behavior from solid-like to fluid-like is achieved by interpolating the relevant physical properties. Specifically, the dynamic viscosity μ and density ρ are expressed as functions of γ using a Darcy-type interpolation:

$$\mu(\gamma) = \mu_l - (\mu_s - \mu_l) \frac{q(1-\gamma)}{q+\gamma} \quad (5)$$

$$\rho(\gamma) = \rho_l - (\rho_s - \rho_l) \frac{q(1-\gamma)}{q+\gamma} \quad (6)$$

where μ_l and μ_s denote the viscosities of the fluid and solid phases, respectively, while ρ_l and ρ_s are the corresponding densities. The interpolation ensures a gradual variation in material properties across the fluid–solid interface during the topology optimization process.

2.3. Objective Function and Constraints

The objective of the topology optimization is to enhance the hydraulic performance of the valve housing by reducing the flow resistance within the discharge passage. Geometric discontinuities such as local contractions, sharp bends, and sudden expansions lead to additional energy dissipation and consequently increase the pressure loss during fluid transport. Therefore, the pressure difference between the inlet and outlet is adopted as the objective function. Minimization of this pressure drop enables the optimization algorithm

to redistribute the internal flow domain toward a lower-resistance flow configuration, which is expected to improve the subsequent discharge performance.

Although cumulative discharged volume provides a more direct evaluation of the transient discharge capability, it requires repeated time-dependent CFD simulations and is therefore not directly adopted as the optimization objective in the present topology optimization framework due to the significantly increased computational cost. Instead, cumulative discharged volume is used as a key performance indicator in the subsequent transient CFD simulations to evaluate the effectiveness of the optimized configuration. The objective function is defined as follows:

$$\varphi = \Delta p = p_{in} - p_{out} \quad (7)$$

where p_{in} and p_{out} denote the inlet and outlet pressures, respectively.

The topology optimization problem is then formulated as follows:

$$\begin{aligned} & \text{find } \gamma \\ & \min \varphi = \Delta p = p_{in} - p_{out} \\ & \text{subject to } \nabla \cdot u = 0 \\ & \rho (u \cdot \nabla) u = -\nabla p + \mu \nabla^2 u - \alpha(\gamma)u \\ & \alpha(\gamma) = (\alpha_{max} - \alpha_{min}) \frac{q(1-\gamma)}{q+\gamma} \\ & \frac{\int_{\Omega} \gamma d\Omega}{\int_{\Omega} 1 d\Omega} \leq V_f \\ & 0 \leq \gamma \leq 1 \end{aligned} \quad (8)$$

where Ω represents the entire valve topology domain, V_f represents the volume fraction of the fluid domain, and in this topology optimization, the upper limit of V_f is 0.9.

2.4. Filtering and Projection

In topology optimization, the continuous variation in the design variable $\gamma \in [0, 1]$ may lead to numerical instabilities, which are commonly manifested as checkerboard patterns and gray-scale regions. The checkerboard pattern refers to non-physical alternating distributions of solid and void phases that arise from mesh-dependent numerical oscillations, particularly at the element scale. Such patterns are primarily associated with discretization sensitivity and the lack of numerical regularization, resulting in artificial spatial oscillations in the design field. These artifacts are unfavorable in practice, as they violate structural continuity and pose difficulties for manufacturing.

Another typical issue is the emergence of gray-scale regions, where the design variable takes intermediate values between 0 and 1. In these regions, the material state becomes ambiguous due to the interpolation of physical properties, leading to discrepancies between numerical representation and real material behavior.

To alleviate these issues, a Helmholtz-type density filter is applied to regularize the design field [32]. The filtered variable γ_f is governed by the following:

$$-R_{min}^2 \nabla^2 \gamma_f + \gamma_f = \gamma \quad (9)$$

where γ is the original design variable, γ_f is the filtered field, and R_{min} denotes the filter radius, typically chosen in relation to the mesh size.

However, density filtering alone may increase the extent of gray-scale regions. To further enforce a clear separation between solid and fluid phases, a hyperbolic tangent projection is introduced:

$$\gamma_h = \frac{\left(\tanh\left(\beta(\gamma_f - \gamma_\beta)\right) + \tanh(\beta\gamma_\beta) \right)}{\left(\tanh(\beta(1 - \gamma_\beta)) + \tanh(\beta\gamma_\beta) \right)} \quad (10)$$

where γ_h is the projected design variable; β is the projection steepness parameter; and γ_β defines the projection threshold. In this study, $\beta = 8$ and $\gamma_\beta = 0.5$. The projection enhances the contrast between solid and fluid regions while maintaining numerical stability.

2.5. Topology Optimization Region

The optimization problem in this study focuses on the design of the inner housing of the pressure relief valve. Before optimization, the fluid domain of the valve is extracted. To prevent interference between the topology optimization region and the existing internal components of the valve body, the design domain is separated accordingly. In Figure 2, the blue region represents the topology optimization domain, and the blue region represents the non-design region.

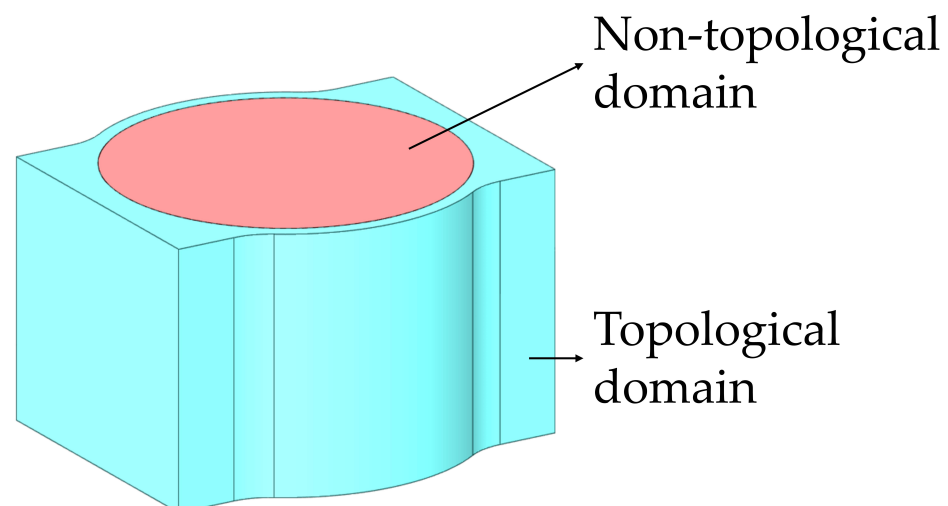


Figure 2. Topological region division.

The topology optimization is carried out using COMSOL Multiphysics 6.3. The computational model is implemented on a workstation equipped with a 16-core AMD64 processor and 64 GB of RAM. The entire design domain is treated as a fluid region in the initial configuration. The initial relative density is assigned as 0.5 in the design (optimization) domain, while the non-design region is fixed with a density value of 1. Sensitivity information is obtained using the discrete adjoint method, which is employed to evaluate the derivatives of the objective and constraint functions with respect to the design variables. The optimization problem is solved using the method of moving asymptotes (MMAs). In the optimization procedure, the maximum number of iterations is set to 200, the convergence tolerance is 0.001, and the move limit is set to 0.2. The flowchart of the topology optimization process is shown in Figure 3.

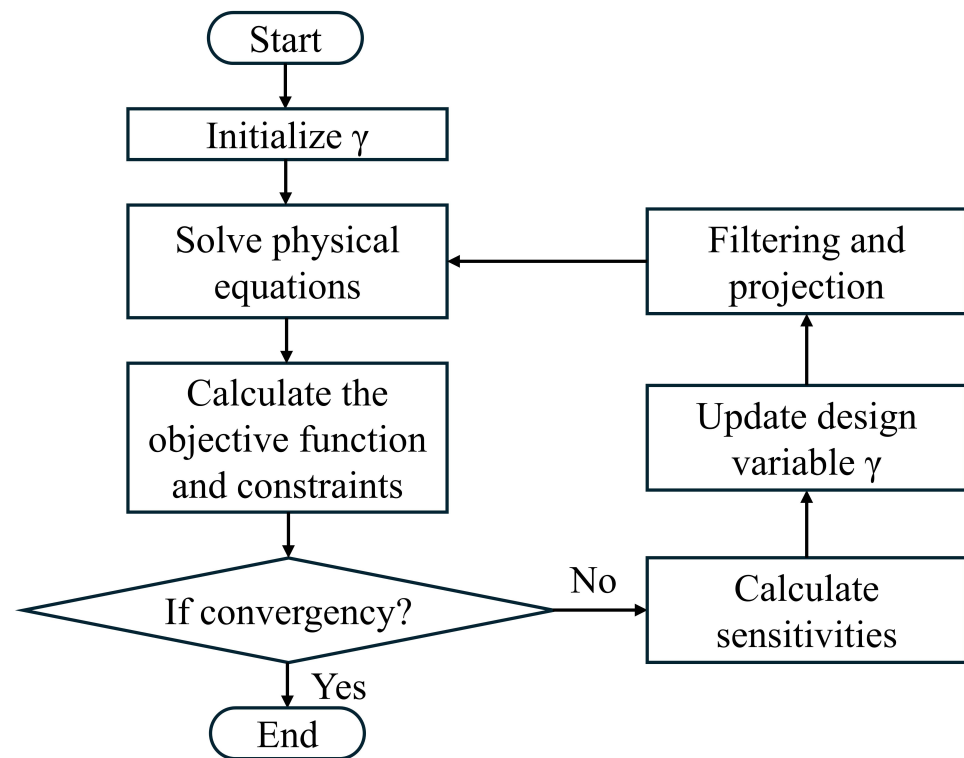


Figure 3. Flowchart of the topology optimization process.

3. Results

3.1. Topology Optimization Result

The comparison between the optimized valve flow channel and the original prototype is presented in Figure 4. To eliminate the influence of intermediate density values, a thresholding operation is applied to the optimized design field based on a commonly used criterion in topology optimization. A threshold value of $\theta = 0.5$ is adopted, where regions with $\theta \leq 0.5$ are classified as solid, and regions with $\theta > 0.5$ are identified as fluid.

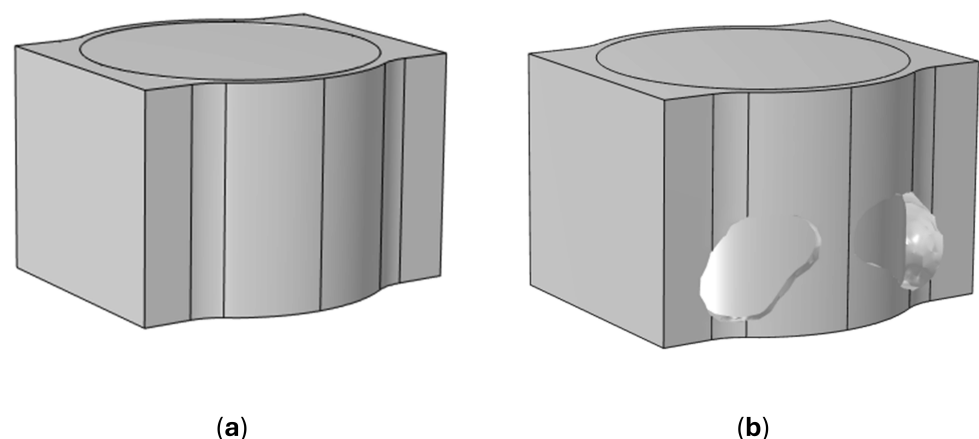


Figure 4. (a) Fluid domain before optimization; (b) fluid domain after optimization.

It should be noted that the additional solid regions generated in the optimized flow domain do not represent defects or unnecessary blockages. Instead, they result from the material redistribution process during topology optimization. Although the fluid domain is locally reduced, these regions generally correspond to areas with lower contribution to the optimization objective. The resulting redistribution of the flow domain modifies the internal flow path and provides additional flow guidance toward the outlet.

In practical engineering applications, the topology optimization result is considered a conceptual flow-guiding design rather than a directly manufactured geometry. In the present study, the optimized flow configuration can be implemented by modifying the internal wall surface of the valve housing while maintaining the original external dimensions and functional components. The corresponding solid features may be introduced through local material addition, machining, or integrated manufacturing approaches. Therefore, the optimized topology provides a feasible reference for improving the internal flow configuration of transformer pressure relief valves without requiring fundamental modifications to the existing valve structure.

3.2. CFD Numerical Methodology

Both models were simulated under identical operating conditions to ensure a reliable comparison. The internal flow was treated as an incompressible turbulent flow, and the Reynolds-averaged Navier–Stokes (RANS) approach was adopted. The standard $k - \epsilon$ turbulence model was employed to describe the turbulent characteristics of the discharge process. Although more advanced turbulence models, such as SST $k - \omega$, Reynolds stress models (RSMs), and large eddy simulation (LES), can provide improved predictions for near-wall effects, turbulence anisotropy, or large-scale unsteady vortex structures, they require higher computational costs and stricter mesh requirements. Since the objective of this study is to compare the relative performance difference between the original and topology-optimized valve designs under identical transient conditions, the standard $k - \epsilon$ model was selected as a compromise between computational efficiency and prediction capability for engineering applications.

The bottom face of the valve base was defined as the inlet with a prescribed pressure load, as illustrated in Figure 5. The inlet pressure profile was obtained from the 6 MJ transformer arcing experiment of the prototype valve, thereby representing the actual pressure variation during an internal fault event. The outlet boundaries were defined as pressure outlets, and all solid surfaces were treated as stationary no-slip walls. The valve plate was fixed at its maximum opening position because the opening duration of the valve plate is much shorter than the discharge period investigated in this study. In addition, components including the signal rod, connecting rods, and springs were neglected because their influence on the internal flow field was considered negligible.

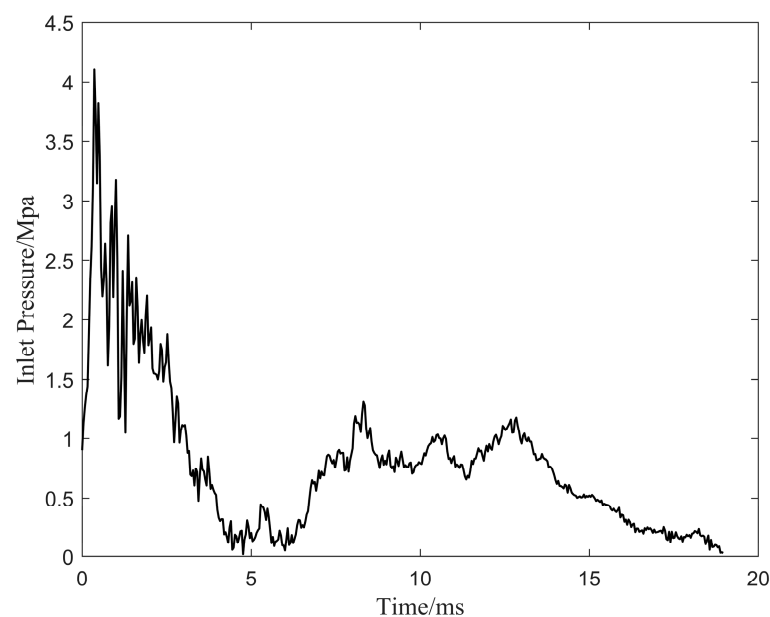


Figure 5. Pressure at the inlet of the developed valve in the arcing experiment.

The simulations were performed using a pressure-based solver with a transient formulation. The SIMPLE algorithm was applied for pressure–velocity coupling. Second-order upwind discretization schemes were adopted for the momentum equation, turbulent kinetic energy equation, and turbulent dissipation rate equation. The convergence criterion was defined as residual values below 10^{-3} for the continuity, momentum, turbulent kinetic energy, and turbulent dissipation rate equations. The working fluid was transformer insulating oil with a density of 885.6 kg/m^3 and a kinematic viscosity of $9.64 \text{ mm}^2/\text{s}$. A time step of 0.04 ms was employed to capture the rapid pressure variation during the discharge process.

A grid independence study was performed to evaluate the influence of mesh resolution on the numerical results. Three mesh resolutions were generated for both the original and optimized valve models. As summarized in Table 1, the differences between the medium and fine meshes were 0.21% for the original valve and 0.12% for the optimized valve compared with the medium mesh results, respectively. These small variations indicate that the predicted discharge performance is nearly independent of mesh resolution. Therefore, fine meshes containing approximately 0.8 million cells were selected for the subsequent CFD analysis to achieve a balance between computational accuracy and efficiency.

Table 1. Grid independence study of the original and optimized valve models.

Model	Mesh Cells	Accumulated Discharge Volume (L)	Relative Difference (%)
Original—Coarse	422,963	13.276459	0.43
Original—Medium	524,016	13.192297	0.21
Original—Fine (Selected)	780,581	13.219593	—
Optimized—Coarse	512,355	13.648408	0.35
Optimized—Medium	602,327	13.584578	0.12
Optimized—Fine (Selected)	847,487	13.601257	—

For the selected fine meshes, the original and optimized valve models contained approximately 0.78 million and 0.85 million cells, respectively. Both fluid domains were discretized using polyhedral elements with an average size of 3.5 mm , and five layers of inflation mesh were applied around the valve plate to better capture the near-wall flow characteristics, as shown in Figure 6. The cell orthogonal quality values of the two meshes were 0.950 and 0.942, respectively, indicating that the generated meshes possess good quality and are suitable for the subsequent transient CFD simulations.

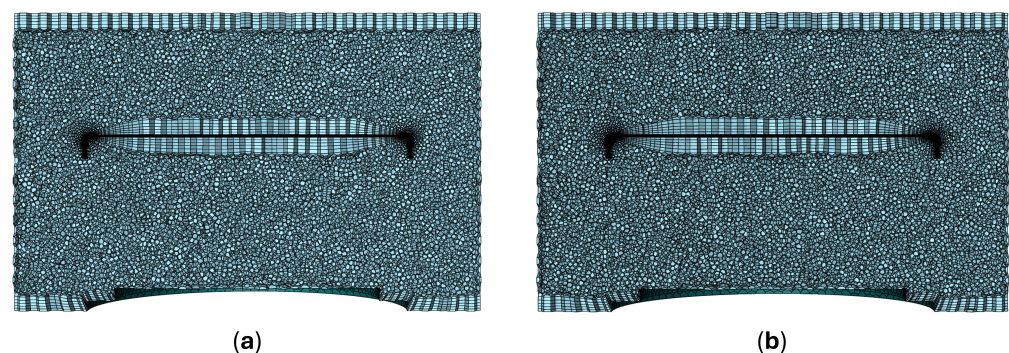


Figure 6. (a) CFD mesh of the original valve; (b) CFD mesh of the optimized valve.

3.3. Transient Discharge Performance

Figure 7 shows the outlet flow rate difference between the two valves. At the initial stage of discharge, the original valve exhibits a slightly higher outlet flow rate than the

optimized valve. This difference may be attributed to the early-stage development of the transient flow field, where the effect of the optimized internal flow configuration on the discharge process is not yet fully reflected. After approximately 3.5 ms, the outlet flow rate of the topology-optimized valve becomes higher than that of the original configuration and remains higher throughout the subsequent discharge process.

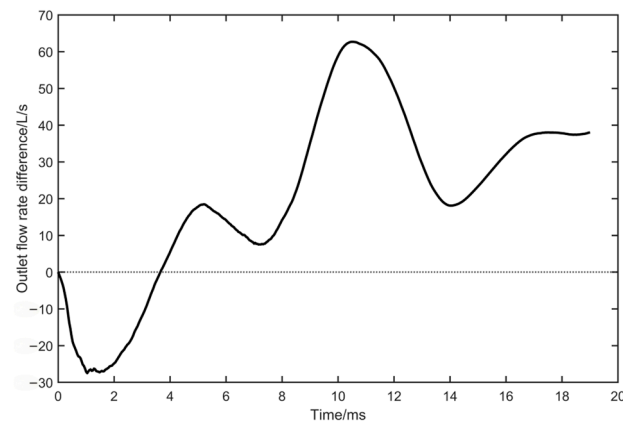


Figure 7. Outlet flow rate difference (optimized valve outlet flow rate minus original valve outlet flow rate).

Figure 8 presents the cumulative discharged volume of the two valves under identical transient pressure boundary conditions. The original valve achieves a total discharged volume of 13.219 L, whereas the topology-optimized valve reaches 13.601 L at the end of the discharge period. This corresponds to an improvement of approximately 3% in cumulative discharge capacity. The increased cumulative discharge volume indicates that the optimized internal flow configuration enhances the transient discharge performance of the valve.

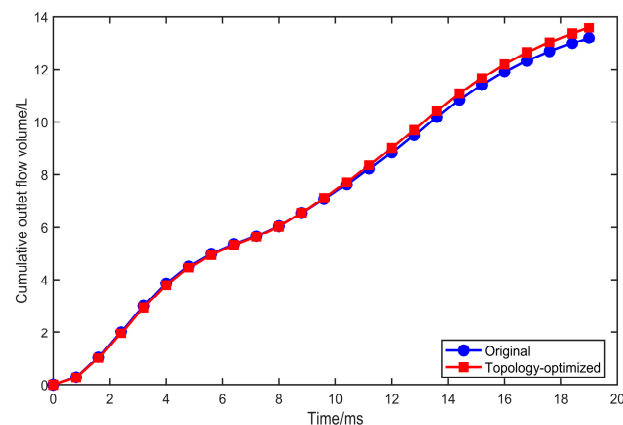


Figure 8. Comparison of cumulative outlet flow volume between the optimized valve and the original valve.

3.4. Pressure Field Characteristics

Pressure distribution is one of the primary indicators for evaluating the hydraulic performance of a pressure relief valve. During the discharge process, the pressure difference between the inlet and outlet provides the driving force for fluid transport. Excessive local pressure accumulation usually indicates increased flow resistance and inefficient pressure-energy conversion, while severe low-pressure regions may be associated with flow separation and hydraulic losses. Therefore, analysis of the pressure field provides insight into the effectiveness of the internal flow path in utilizing the available pressure potential.

Figure 9 presents the static-pressure distributions of the original and optimized valves at three representative stages of the discharge process. At 1.6 ms, both configurations exhibit a high-pressure region beneath the valve plate due to the rapid pressure rise at the inlet. The overall pressure distributions are similar because the fluid has only begun to accelerate and the influence of the optimized internal geometry remains limited at this stage.

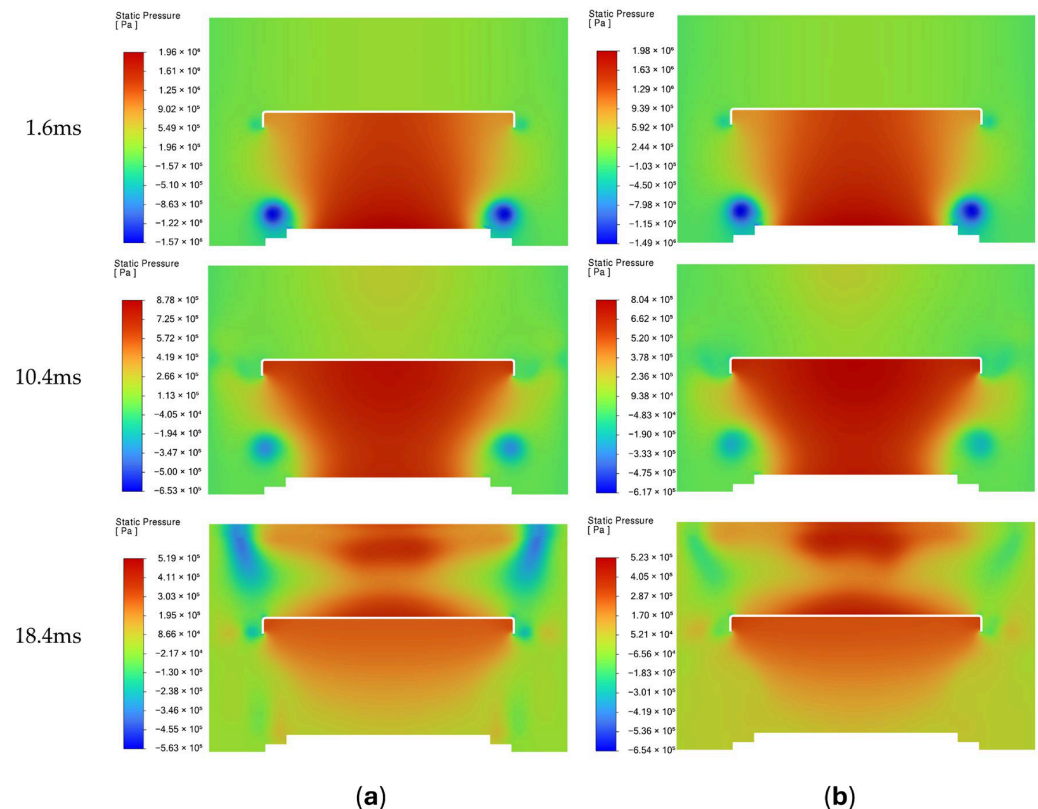


Figure 9. (a) Static-pressure distributions of the original valve; (b) static-pressure distributions of the optimized valve.

At 10.4 ms, the original valve still exhibits relatively obvious low-pressure zones in the bottom-inlet corner of the valve. The optimized configuration exhibits a smoother pressure transition, and the low-pressure zones become less pronounced. As the flow rate increases, the influence of the optimized flow passage becomes more evident, leading to a more continuous pressure distribution within the valve chamber.

At 18.4 ms, when the discharge process approaches its final stage, the influence of topology optimization becomes most apparent. In the original valve, extensive low-pressure regions are observed near the top of the valve chamber and close to the outlet. After topology optimization, the pressure field becomes more evenly distributed throughout the chamber, and the extent of the low-pressure regions is reduced. The more uniform pressure distribution indicates that the optimized geometry promotes a more effective redistribution of the driving pressure during discharge, thereby reducing localized adverse pressure gradients.

These results indicate that topology optimization improves the pressure distribution characteristics inside the valve chamber and contributes to a more favorable transient discharge process.

3.5. Velocity Field Characteristics

Velocity distribution reflects the utilization efficiency of the available flow domain. In an efficient discharge process, a more uniform velocity distribution generally indicates better utilization of the available flow passage. Conversely, extensive low-velocity regions indicate stagnant flow and ineffective use of the valve chamber volume. Therefore, velocity contours can be used to evaluate the effectiveness of flow development within the valve.

Figure 10 compares the velocity-magnitude distributions of the original and optimized valves. At 1.6 ms, the flow is mainly concentrated near the inlet region, and the velocity fields of the two valves remain similar. The highest velocities are observed around the lower corners adjacent to the inlet, where the fluid first enters the chamber and undergoes rapid acceleration. Since the discharge process has only just begun, the influence of the optimized internal geometry on the overall velocity distribution is still limited.

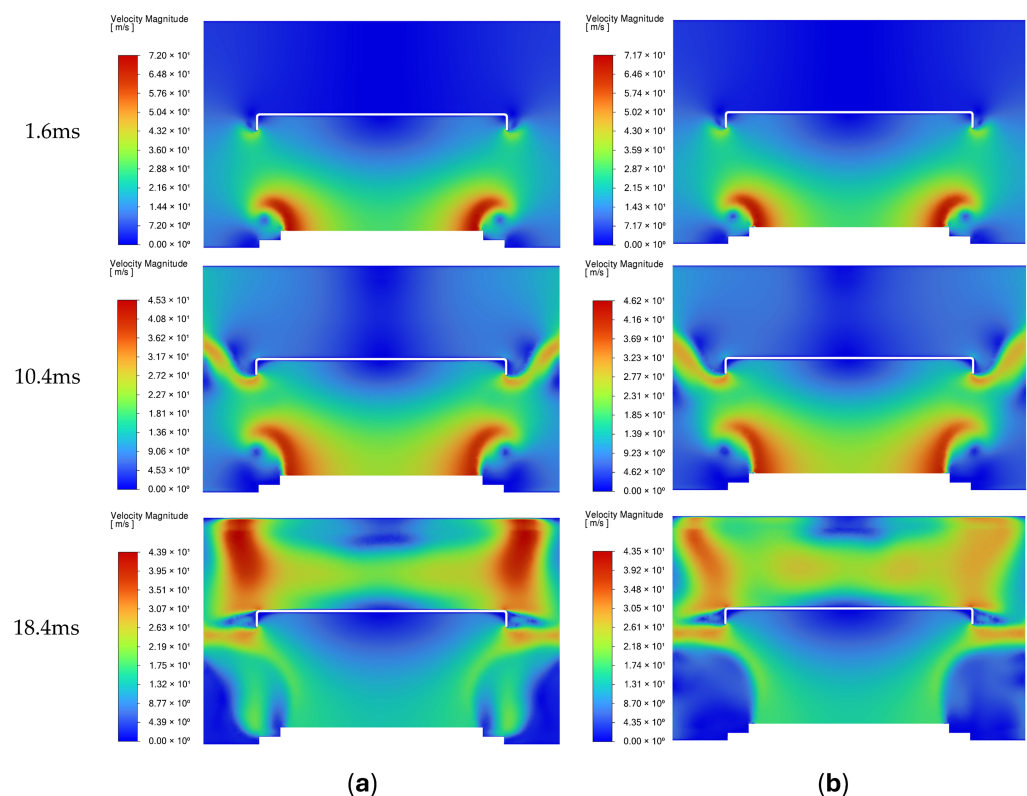


Figure 10. (a) Velocity distributions of the original valve; (b) velocity distributions of the optimized valve.

At 10.4 ms, the velocity fields of the two valves are still similar. However, the optimized valve exhibits slightly higher flow velocity in several regions of the chamber. Although the overall flow pattern changes little, the increase in flow velocity indicates a more effective utilization of the available flow passage during the discharge process.

At 18.4 ms, localized high-velocity regions are observed in the upper chamber of the original valve together with a localized low-velocity zone near the inlet. The velocity distribution suggests the existence of complex local flow structures within the chamber. After topology optimization, the high-velocity zones become less concentrated and the velocity contours in the upper chamber appear more continuous. Meanwhile, the localized low-velocity region near the inlet was substantially reduced. The more uniform velocity distribution suggests that more fluid contributes effectively to the discharge process.

After topology optimization, the overall velocity field remains similar to that of the original valve, while moderately higher flow velocities are observed during the middle

and late stages of discharge. The reduction in localized low-velocity regions indicates that the available flow passage is utilized more effectively, which is consistent with the improvement in cumulative discharged volume presented in Section 3.3.

3.6. Turbulent Kinetic Energy Characteristics

Turbulent kinetic energy (TKE) represents the kinetic energy associated with turbulent velocity fluctuations and is widely used to characterize turbulence intensity and energy dissipation in internal flows. High-TKE regions are generally associated with strong shear layers, recirculation zones, and flow instability. Since part of the mechanical energy is converted into turbulent fluctuations, excessive TKE generally indicates stronger turbulent activity and additional energy dissipation. Therefore, the TKE distribution can be used to evaluate the degree of energy dissipation inside the valve.

Figure 11 shows the turbulent kinetic energy distributions of the original and optimized valves. At 1.6 ms, the TKE distributions of the two valves are similar, and the levels are relatively low. Localized turbulence is mainly observed near the sharp corners adjacent to the valve plate and inlet region, where the flow undergoes rapid acceleration.

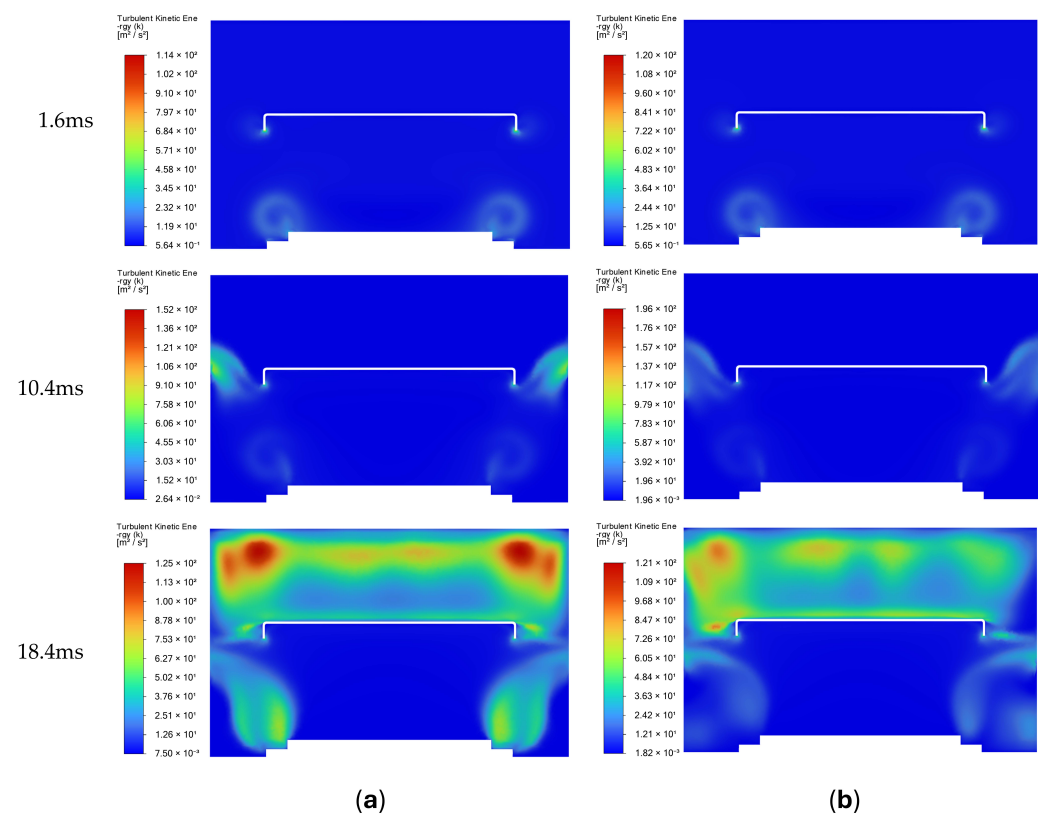


Figure 11. (a) TKE distributions of the original valve; (b) TKE distributions of the optimized valve.

At 10.4 ms, turbulence production becomes more pronounced as the discharge process develops. For the original valve, elevated TKE regions appear near the outlet entrances. The optimized valve exhibits a similar overall distribution pattern, but the intensity of the high-TKE regions is reduced, indicating weaker turbulent fluctuations under comparable flow conditions.

At 18.4 ms, significant differences emerge between the two configurations. The original valve develops extensive high-TKE regions in the upper chamber and near the outlet, indicating strong turbulent activity and enhanced energy dissipation. In contrast, the optimized configuration exhibits a noticeable reduction in both the intensity and spatial extent of these regions. The high-TKE zones become less concentrated, and the overall

turbulence level inside the chamber decreases. Compared with the pressure and velocity fields, the difference in TKE between the two configurations becomes more apparent during the late stage of discharge.

The reduction in TKE indicates that the optimized valve experiences weaker turbulent fluctuations during the discharge process. This observation is consistent with the quantitative comparison presented in Section 3.7, where the optimized valve exhibits a lower volume-averaged TKE while maintaining a higher discharge capacity.

3.7. Quantitative Performance Evaluation

To provide a quantitative comparison between the original and topology-optimized valves, several representative performance indicators are summarized in Table 2. The flow-field-related parameters, including the outlet mass-weighted average velocity and volume-averaged TKE, are evaluated at 18.4 ms. This instant corresponds to the late stage of the transient discharge process, where the differences between the two valve configurations become more evident in the pressure, velocity, and turbulent kinetic energy fields, making it suitable for quantitative comparison. The cumulative discharged volume is selected as the primary indicator of discharge performance and is calculated over the entire transient discharge period. The outlet mass-weighted average velocity is used to characterize the overall flow transport capability at the outlet because it reflects the characteristic velocity of the discharged fluid while accounting for the non-uniform velocity distribution across the outlet. The volume-averaged TKE is used to evaluate the overall turbulence characteristics inside the valve chamber, providing a quantitative measure of the turbulence intensity within the flow domain.

Table 2. Quantitative comparison of representative performance indicators.

Parameter	Original Valve	Topology-Optimized Valve	Difference
Cumulative discharged volume (L)	13.219	13.601	+2.9%
Outlet mass-weighted average velocity (m/s)	16.724	17.230	+3.0%
Volume-averaged TKE (m^2/s^2)	26.441	24.545	−7.2%

As summarized in Table 2, the topology-optimized valve exhibits an increase in cumulative discharge capacity compared with the original configuration, which is consistent with the transient discharge behavior discussed in Section 3.3. In addition, the outlet mass-weighted average velocity increases from 16.724 m/s to 17.230 m/s, indicating enhanced flow transport through the optimized flow passage.

Meanwhile, the volume-averaged turbulent kinetic energy decreases from 26.441 m^2/s^2 to 24.545 m^2/s^2 , representing a reduction of approximately 7.2%. This reduction is consistent with the TKE distributions discussed in Section 3.6, where the optimized valve exhibits weaker and less concentrated high-turbulence regions. The lower overall turbulence intensity suggests that the optimized geometry suppresses excessive turbulent fluctuations associated with flow separation and recirculation.

Overall, the quantitative comparison confirms that the proposed topology optimization improves the transient discharge performance while modifying the internal turbulence characteristics of the valve. The agreement between the quantitative indicators and the flow-field analyses presented in Sections 3.3–3.6 provides further support for the effectiveness of the proposed optimization strategy.

4. Discussion

The topology optimization results indicate that the optimized configuration differs from the baseline design primarily through the formation of several localized geometric protrusions along the sidewall regions of the valve chamber. These features are mainly located in the middle and lower portions of the side walls near the outlet region. Because the volume-fraction constraint limits the proportion of solid material within the topology domain, the overall geometric characteristics of the flow passage remain similar before and after optimization. Therefore, the observed improvement in discharge performance is primarily associated with changes in the internal flow development process caused by the optimized geometry.

The influence of these geometric modifications can first be observed from the pressure distributions. During the early stage of discharge, both configurations exhibit similar pressure-field characteristics because the pressure difference across the valve is still relatively large. As the discharge process develops, the differences between the two configurations become more apparent. At 18.4 ms, the baseline configuration exhibits a higher pressure-concentration in the upper chamber region, whereas the optimized configuration presents a more distributed pressure field. In addition, the low-pressure regions located near the outlet-side corners become less pronounced after optimization. These observations indicate that the optimized geometry modifies the pressure distribution within the valve chamber and reduces the degree of local pressure concentration. As a result, the pressure field becomes more evenly distributed during the discharge process.

The velocity contours further demonstrate that topology optimization alters the flow-development pattern within the valve chamber. In the baseline configuration, two relatively high-velocity regions are observed above the valve plate, while a lower-velocity zone remains between them. This feature becomes particularly evident during the later stage of discharge. The coexistence of adjacent high- and low-velocity regions indicates that the flow is unevenly distributed across the upper chamber. After topology optimization, the shape and spatial distribution of these velocity regions change noticeably. The low-velocity region located between the two high-velocity zones becomes less pronounced, and the surrounding velocity field is altered accordingly. Meanwhile, the velocity distribution near the outlet region also changes. These observations suggest that the optimized geometry influences the internal flow-development process and modifies the manner in which fluid moves through the valve chamber during discharge.

The effect of topology optimization is also reflected in the turbulent kinetic energy distributions. At the beginning of the discharge process, both configurations exhibit relatively low turbulence intensity because the flow field is still developing. As the discharge proceeds, high-TKE regions gradually appear in the upper chamber and near the outlet region. Compared with the baseline configuration, the optimized design exhibits a different spatial distribution of turbulent kinetic energy during the later stage of discharge. In particular, the locations and extent of the high-TKE regions are modified after topology optimization. This indicates that the optimized geometry affects the turbulence development process within the valve chamber and changes the interaction between the main flow and the surrounding fluid.

Combining the pressure, velocity, and turbulent kinetic energy results, it can be inferred that the localized geometric features generated by topology optimization influence the transient flow development throughout the valve chamber. Rather than substantially changing the overall flow passage, these structures modify local flow characteristics, leading to changes in pressure distribution, velocity distribution, and turbulence intensity. The resulting flow field exhibits a more uniform pressure distribution and a modified flow-

development pattern during discharge. These changes collectively contribute to the increase in cumulative discharge volume observed in the optimized design.

Under the same transient pressure-loading condition, the cumulative discharged volume increases from 13.219 L to 13.601 L during the 19 ms discharge process. Considering that the optimization was performed under strict geometric constraints and a prescribed solid-volume constraint, the observed improvement demonstrates that localized modifications of the internal flow domain can influence transient flow behavior and contribute to enhanced discharge performance in transformer pressure relief valves.

5. Conclusions

In this study, a density-based topology optimization approach was proposed to improve the transient discharge performance of a transformer pressure relief valve by modifying its internal flow-path configuration. Unlike conventional geometric optimization methods that rely on predefined parameters, the proposed approach enables automatic redistribution of the internal flow domain while maintaining the original external dimensions and functional components. The optimized configuration was evaluated through transient CFD simulations under experimentally derived pressure loading conditions, providing a numerical framework for assessing the influence of internal flow-path design on rapid pressure-relief processes. The proposed methodology provides a potential strategy for improving internal flow configurations of hydraulic safety devices with complex flow passages. The main conclusions are as follows:

- (1) The topology optimization successfully generated a new internal flow-guiding structure without increasing the overall valve size. The optimized design redistributed the internal flow passage by reducing ineffective flow regions associated with low-velocity zones and complex flow structures.
- (2) Under identical transient pressure conditions, the topology-optimized valve achieved a cumulative discharge volume of 13.601 L, compared with 13.219 L for the original valve, corresponding to an improvement of approximately 3%. The improvement was mainly attributed to enhanced flow organization rather than an increase in the available flow area.
- (3) The CFD results demonstrated that topology optimization modified the internal pressure and velocity distributions and reduced intense turbulent regions. The optimized valve exhibited a more uniform flow field and weaker turbulence intensity, indicating more favorable internal flow characteristics during transient discharge.

Although the improvement in cumulative discharge capacity is moderate, this enhancement is obtained without changing the external dimensions, valve opening mechanism, or major structural components of the existing pressure relief valve. For safety-critical devices such as transformer PRVs, even a moderate increase in discharge capability can provide an additional pressure-relief margin during extremely rapid internal fault events. Therefore, the proposed method provides an effective approach for improving valve performance through internal flow optimization rather than complete structural redesign.

The proposed methodology still has several limitations. The present CFD evaluation was performed with the valve plate fixed at the fully opened position, which represents the flow characteristics after valve opening but does not capture the transient interaction between valve motion and fluid flow during the opening process. In addition, the topology-optimized structure obtained in this study represents a conceptual flow-guiding design and requires further engineering refinement before practical implementation. Future investigations may focus on more comprehensive transient modeling considering valve motion effects, experimental validation of the optimized configuration, and the develop-

ment of multi-objective optimization strategies as well as both discharge performance and hydraulic losses.

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

Funding: This research was funded by the National Key Research and Development Program of China (Grant No. 2023YFB2407004) and the Fundamental Research Funds for the Central Universities (Grant No. JZ2024HGTB0204).

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.

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

References

1. Brodeur, S.; Lê, V.N.; Champiaud, H. A Nonlinear Finite-Element Analysis Tool to Prevent Rupture of Power Transformer Tank. *Sustainability* **2021**, *13*, 1048. [\[CrossRef\]](#)
2. Guo, Z.; Guo, X.; Li, X.; Tan, M.; Li, D. Study on Decomposition Gas Explosion of Transformer Oil Under Arc. In *Proceedings of the 2018 2nd IEEE Conference on Energy Internet and Energy System Integration (EI²), Beijing, China, 20–22 October 2018*; IEEE: Piscataway, NJ, USA, 2018; pp. 1–5. [\[CrossRef\]](#)
3. Valenta, L.; Černý, I.; Maternik-Demontoux, A. Power Transformers and Environmental Protection. *IFAC-PapersOnLine* **2016**, *49*, 18–22. [\[CrossRef\]](#)
4. Meitei, S.N.; Borah, K.; Chatterjee, S. Modelling of Acoustic Wave Propagation Due to Partial Discharge and Its Detection and Localization in an Oil-Filled Distribution Transformer. *Frequenz* **2020**, *74*, 73–81. [\[CrossRef\]](#)
5. Rui, J.; Lidong, Z. A High-Pressure Relief Valve Life Measuring System Based on LabVIEW. *J. Phys. Conf. Ser.* **2024**, *2785*, 012037. [\[CrossRef\]](#)
6. Ye, Z.; Li, W.; Xu, Y.; Jin, Z.; Qian, J. Effects of Adjusting Ring and Spring Stiffness on Fluid Dynamics of Steam Spring-Loaded Safety Valve. *Nucl. Eng. Technol.* **2025**, *57*, 103338. [\[CrossRef\]](#)
7. Hao, Q.; Wu, W.; Liang, X.; Liu, Z. Effects of Structure Parameters on Abnormal Opening of Pilot-Operated Relief Valve Under Alternating Pressure. *IEEE Access* **2019**, *7*, 33932–33942. [\[CrossRef\]](#)
8. Seo, Y.M.; Noh, H.W.; Koo, T.H.; Ha, D.W.; Ko, R.K. Experimental Study on the Operation of Pressure Safety Valve in the Liquid Hydrogen Environment. *Results Eng.* **2025**, *28*, 107328. [\[CrossRef\]](#)
9. Liu, C.; Liu, Y.; Cheng, Z.; Li, Y.; Liu, P.; Liu, L.; Min, Y.; Duan, Q.; Mei, W.; Wang, Q. Experimental Study on the Impact of Safety Valve Venting Pressure on Thermal Runaway in Large-Format Lithium Iron Phosphate Battery. *Process Saf. Environ. Prot.* **2025**, *201*, 107563. [\[CrossRef\]](#)
10. Ma, W.; Ma, F.; Guo, R. Experimental Research on the Dynamic Instability Characteristic of a Pressure Relief Valve. *Adv. Mech. Eng.* **2019**, *11*, 1687814019833531. [\[CrossRef\]](#)
11. Bázsó, C.; Hős, C.J. An Experimental Study on the Stability of a Direct Spring Loaded Poppet Relief Valve. *J. Fluids Struct.* **2013**, *42*, 456–465. [\[CrossRef\]](#)
12. Lei, J.; Tao, J.; Liu, C.; Wu, Y. Flow Model and Dynamic Characteristics of a Direct Spring Loaded Poppet Relief Valve. *Proc. Inst. Mech. Eng. Part C J. Mech. Eng. Sci.* **2018**, *232*, 1657–1664. [\[CrossRef\]](#)
13. Beune, A.; Kuerten, J.G.M.; Van Heumen, M.P.C. CFD Analysis with Fluid–Structure Interaction of Opening High-Pressure Safety Valves. *Comput. Fluids* **2012**, *64*, 108–116. [\[CrossRef\]](#)
14. Song, X.; Cui, L.; Cao, M.; Cao, W.; Park, Y.; Dempster, W.M. A CFD Analysis of the Dynamics of a Direct-Operated Safety Relief Valve Mounted on a Pressure Vessel. *Energy Convers. Manag.* **2014**, *81*, 407–419. [\[CrossRef\]](#)
15. Stosiak, M.; Karpenko, M.; Skačkauskas, P. Variations in the pressure pulsation spectrum of a hydraulic system with an oscillating relief valve. *Adv. Mech. Eng.* **2025**, *17*, 16878132251343945. [\[CrossRef\]](#)

16. Stosiak, M.; Skačkauskas, P.; Towarnicki, K.; Deptuła, A.; Deptuła, A.M.; Prażnowski, K.; Grzywacz, Ż.; Karpenko, M.; Urbanowicz, K.; Łapka, M. Analysis of the Impact of Vibrations on a Micro-Hydraulic Valve Using a Modified Induction Algorithm. *Machines* **2023**, *11*, 184. [[CrossRef](#)]
17. Elmayyah, W.; Dempster, W. Prediction of two-phase flow through a safety relief valve. *Proc. Inst. Mech. Eng. Part E J. Process Mech. Eng.* **2013**, *227*, 42–55. [[CrossRef](#)]
18. Bassi, F.; Crivellini, A.; Dossena, V.; Franchina, N.; Savini, M. Investigation of Flow Phenomena in Air–Water Safety Relief Valves by Means of a Discontinuous Galerkin Solver. *Comput. Fluids* **2014**, *90*, 57–64. [[CrossRef](#)]
19. Gersborg-Hansen, A.; Sigmund, O.; Haber, R.B. Topology Optimization of Channel Flow Problems. *Struct. Multidiscip. Optim.* **2005**, *30*, 181–192. [[CrossRef](#)]
20. Alexandersen, J.; Andreasen, C.S. A Review of Topology Optimisation for Fluid-Based Problems. *Fluids* **2020**, *5*, 29. [[CrossRef](#)]
21. Borrvall, T.; Petersson, J. Topology Optimization of Fluids in Stokes Flow. *Numer. Methods Fluids* **2003**, *41*, 77–107. [[CrossRef](#)]
22. Olesen, L.H.; Okkels, F.; Bruus, H. A high-level programming-language implementation of topology optimization applied to steady-state Navier–Stokes flow. *Int. J. Numer. Meth. Eng.* **2006**, *65*, 975–1001. [[CrossRef](#)]
23. Sigmund, O.; Maute, K. Topology Optimization Approaches: A Comparative Review. *Struct. Multidiscip. Optim.* **2013**, *48*, 1031–1055. [[CrossRef](#)]
24. Kreissl, S.; Pingen, G.; Maute, K. Topology optimization for unsteady flow. *Int. J. Numer. Meth. Eng.* **2011**, *87*, 1229–1253. [[CrossRef](#)]
25. Dilgen, S.B.; Dilgen, C.B.; Fuhrman, D.R.; Sigmund, O.; Lazarov, B.S. Topology Optimization of Turbulent Flows. *Comput. Methods Appl. Mech. Eng.* **2018**, *331*, 363–393. [[CrossRef](#)]
26. Høghøj, L.C.; Nørhave, D.R.; Alexandersen, J.; Sigmund, O.; Andreasen, C.S. Topology Optimization of Two Fluid Heat Exchangers. *Int. J. Heat Mass Transf.* **2020**, *163*, 120543. [[CrossRef](#)]
27. Behrou, R.; Pizzolato, A.; Forner-Cuenca, A. Topology Optimization as a Powerful Tool to Design Advanced PEMFCs Flow Fields. *Int. J. Heat Mass Transf.* **2019**, *135*, 72–92. [[CrossRef](#)]
28. Luo, L.; Li, J.; Zhang, N.; Wang, K.; Zhang, S.; Liang, N. Design and Analysis of a New Pressure Relief Valve for Oil-Immersed Electrical Equipment. *J. Press. Vessel Technol.* **2023**, *145*, 021701. [[CrossRef](#)]
29. Yu, M.; Ruan, S.; Wang, X.; Li, Z.; Shen, C. Topology Optimization of Thermal–Fluid Problem Using the MMC-Based Approach. *Struct. Multidiscip. Optim.* **2019**, *60*, 151–165. [[CrossRef](#)]
30. Zhang, K.; Li, Y.; Chang, S.-M.; Hu, L.; Wang, X.; Yu, M. Hydraulic and Thermal Performance Enhancement for the Cold Plate Using Topology Optimization. *Appl. Therm. Eng.* **2024**, *236*, 121829. [[CrossRef](#)]
31. Wang, J.; Chen, F.; Shao, Z.; He, L. Study of the Influence of Objective Functions on the Topology Optimization Design of Battery Cold Plate. *Appl. Therm. Eng.* **2023**, *226*, 120326. [[CrossRef](#)]
32. Wang, Y.; Yu, M.; Fan, H.; Chung, J.D. Multi-Objective Topology Optimization of Cold Plates for Enhanced Battery Thermal Management in Electric Vehicles. *Case Stud. Therm. Eng.* **2025**, *66*, 105794. [[CrossRef](#)]

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