Journal of Engineering Mechanics and Machinery (2024)
Clausius Scientific Press, Canada

DOI: 10.23977/jemm.2024.090303
ISSN 2371-9133 Vol. 9 Num. 3

Research on Mechanical Performance Analysis and Topology Optimization Design of New Automobile Frame Structures

Jiahao Wang, Nan Wei*, Qingyu Xu

College of Mechanical Engineering, Taishan University, Taian, China
**Corresponding author: weinan0008@163.com*

Keywords: Automobile frame; Mechanical performance analysis; Topology optimization; Lightweight design; Finite element analysis

Abstract: The mechanical performance of a new automobile frame structure plays a crucial role in ensuring the safety, comfort, and lightweight design of the vehicle. This study integrates finite element analysis and topology optimization methods to conduct a comprehensive analysis of the frame's mechanical performance, achieving lightweight design through structural optimization. A finite element model of the frame was established to analyze its strength, stiffness, and dynamic characteristics, revealing imbalances in the current design between performance and weight. In the topology optimization process, based on material distribution and structural performance, optimization algorithms were employed to improve the frame design, resulting in a lighter structure with enhanced performance. Simulation results indicate significant improvements in the optimized frame's strength and stiffness, along with a notable reduction in weight. This research provides theoretical and practical guidance for the design of new automobile frame structures and contributes to the sustainable development of the automotive industry.

1. Introduction

With the rapid development of the automotive industry, vehicle performance requirements are becoming increasingly diverse. As a vital load-bearing structure, the frame's mechanical performance directly influences the safety, comfort, and durability of the vehicle. At the same time, lightweight design has become a key focus in modern automotive development, not only enhancing fuel efficiency and reducing emissions but also improving vehicle dynamics. Consequently, achieving lightweight design while ensuring adequate frame strength and stiffness has become a crucial research area in frame design. In recent years, extensive studies have been conducted on frame mechanical performance analysis and optimization. Finite element analysis (FEA) has been widely applied for simulating and evaluating frame performance, allowing precise modeling of stresses, strains, and vibrations under complex operating conditions. This provides essential guidance for optimization. Moreover, topology optimization has emerged as an efficient tool for structural optimization, enabling optimal performance-to-weight balance by redistributing materials and refining structural layouts. However, most existing studies focus on single-objective optimization without comprehensively addressing multi-objective design considerations. Furthermore, challenges remain in ensuring manufacturability and cost-efficiency in optimized designs. This study addresses these gaps by proposing a lightweight design method for automotive frames based on mechanical performance analysis and topology optimization. A finite element model of the frame was developed to simulate and analyze key mechanical properties, identifying performance bottlenecks and improvement directions. Topology optimization was then applied to achieve weight reduction and stiffness enhancement, with performance improvements validated through simulation. The findings offer theoretical support and practical guidance for the design and optimization of automobile frames, contributing to technological advancement and sustainable development in the automotive industry[1].

2. Theoretical Foundations and Design Principles for New Frame Structures

2.1. Mechanical Performance Evaluation Indicators

In automotive frame design, mechanical performance is a critical factor in evaluating safety, stability, and durability. Performance indicators such as strength, stiffness, vibration characteristics, and fatigue life provide a comprehensive understanding of the frame’s behavior under various operating conditions and its long-term reliability. These indicators are essential for developing frames that meet functional requirements while optimizing material use and cost efficiency. Strength measures the frame's capacity to withstand external loads without failure. It is assessed through stress analysis under static and impact loading conditions to ensure the frame can handle both ordinary and extreme forces. The frame's strength must satisfy material yield and fracture limits, preventing structural failure in high-stress situations. Both local stress concentrations, such as at weld points and joints, and the overall load distribution must be carefully evaluated to avoid weak points that could compromise structural integrity. Stiffness is another critical performance parameter, reflecting the frame's resistance to deformation. This directly impacts vehicle handling, stability, and passenger comfort[2]. Key stiffness parameters include bending stiffness, which governs vertical deformation under load, and torsional stiffness, which determines the frame's ability to resist twisting during cornering or traversing uneven terrain. By enhancing stiffness, the vehicle benefits from improved dynamic response and better control, particularly under challenging driving conditions. Vibration characteristics focus on the frame’s natural frequencies and modal distributions, directly influencing noise, vibration, and harshness (NVH) performance. A well-designed frame avoids resonance with components like the engine, suspension, or road vibrations, which reduces noise and ensures smoother operation. Modal analysis identifies structural weaknesses, enabling design adjustments that optimize dynamic behavior and passenger experience. Fatigue life evaluates the frame's durability under long-term repeated loads. Frames must have sufficient fatigue strength to endure dynamic and impact loads encountered during daily operation and in severe conditions. Fatigue life analysis often incorporates material S-N curves and real-world loading spectra, enabling engineers to pinpoint high-stress regions and implement design solutions such as optimized welding, improved geometry, and enhanced material distribution to extend the frame’s lifespan. Together, these mechanical performance indicators provide a structured framework for frame design and optimization. By balancing strength, stiffness, vibration, and fatigue characteristics, designers can achieve frames that excel in safety, performance, and durability, while maintaining cost-effectiveness. These evaluations are integral to creating automotive frames that meet the evolving demands of modern vehicle engineering and sustainable practices[3].

2.2. Topology Optimization Theory and Methods

Topology optimization, based on mathematical computation and finite element analysis, is a structural optimization method that redistributes material resources and refines structural layouts to achieve lightweight design and enhanced mechanical performance. As an innovative tool, topology optimization is widely used in engineering, particularly in automotive frame optimization, offering a scientific basis for achieving lightweight and high-performance designs. The fundamental principle of topology optimization is to determine the optimal material distribution within a given design space to achieve the best structural performance under specific objectives. Typical optimization objectives include weight minimization, stiffness maximization, frequency optimization, or multi-objective optimization[4]. By removing unnecessary material, topology optimization reduces weight while maintaining mechanical performance. The process involves solving optimization problems defined by objective functions and constraints, such as material volume, strength limitations, or geometric restrictions, to generate the optimal design. Several optimization methods are commonly used. The density-based method is widely applied, using continuous material density values to represent distribution. Iterative updates refine the density, yielding efficient optimization results suitable for complex structures, though geometric post-processing is often needed. The level-set method represents structural boundaries with implicit functions, achieving high-precision designs but with increased computational complexity. Deformation-based methods optimize local geometric shapes based on deformation and stress distribution, ideal for improving existing designs. Discrete variable methods treat material distribution as binary variables (0 or 1), employing combinatorial optimization algorithms like genetic algorithms or particle swarm optimization, though computationally intensive for large-scale problems. In automotive frame design, the topology optimization workflow typically includes defining the design space and load conditions, establishing the finite element model, inputting initial parameters, and solving the optimization problem. Post-processing steps such as shape refinement, manufacturability analysis, and simulation validation ensure practical applicability. Topology optimization's ability to balance multiple performance objectives, such as stiffness maximization and weight minimization, has significantly enhanced frame design efficiency. Its integration with advanced computing technologies like artificial intelligence and machine learning promises further improvements in design optimization efficiency. In summary, topology optimization provides powerful tools for lightweight design and performance enhancement of automotive frames. By optimizing material distribution and structural layouts, it minimizes material use while maintaining mechanical performance, contributing to sustainable development in the automotive industry[5].

3. Analysis of Mechanical Performance of the Frame

3.1. Modeling and Analysis of the Frame Structure

As a critical load-bearing component of an automobile, the mechanical performance of the frame directly impacts the vehicle's safety, comfort, and durability. To thoroughly analyze the mechanical performance, it is essential to establish an accurate finite element model (FEM) of the frame. Through modeling and analysis, the stress distribution and deformation behavior under real-world working conditions can be simulated, providing a scientific basis for optimization. The core steps of modeling include precise descriptions of the frame's geometry, material properties, and boundary conditions. During the geometric modeling phase, key components of the frame, such as main beams, crossbeams, and connectors, must be accurately defined to ensure consistency with the physical structure. Modern 3D modeling tools, such as CATIA and SolidWorks, enable the creation of high-precision geometric models, which serve as a foundation for finite element analysis. In defining material properties, it is necessary to select appropriate material models, such as linear elastic or nonlinear models, and specify essential parameters, including elastic modulus, Poisson's ratio, and density, to accurately reflect the material's physical characteristics. In the configuration of loads and boundary conditions, the working environment and constraints of the frame must be simulated. For example, during static strength analysis, the bottom nodes of the frame can be set as fixed constraints, and vertical loads applied at specific points to simulate vehicle loads[6]. For dynamic analysis, vibration responses are considered by using random vibrations encountered during vehicle operation as input. Accurate settings for loads and boundary conditions maximize the alignment of simulations with real-world working environments, enhancing the credibility of the analysis. The analysis methods primarily include static analysis, modal analysis, and fatigue analysis. Static analysis evaluates the stress distribution and deformation of the frame under static loads to determine whether strength and stiffness requirements are met. Special attention is given to stress concentration areas, such as weld points and joints, to identify potential structural weaknesses. Modal analysis assesses the vibration characteristics of the frame, including natural frequencies and mode shapes. By conducting modal analysis, it is possible to determine whether the frame risks resonance with external sources (e.g., engine vibrations) and adjust the structural design to optimize dynamic performance. Fatigue analysis evaluates the reliability of the frame under long-term repeated loading, particularly in complex stress cycles. By analyzing material properties and load spectra, fatigue-prone areas can be identified, and corresponding reinforcement strategies developed. By following this modeling and analysis process, a comprehensive evaluation of the frame's mechanical performance can be conducted, identifying strengths and weaknesses of the existing design. The results demonstrate that the precision of the FEM and the reliability of the analysis directly influence the quality of design decisions. Thus, employing high-precision modeling tools and rational analysis methods is critical for achieving optimized frame designs[7].

3.2. Simulation and Evaluation of Mechanical Performance

Mechanical performance simulation is a vital method for evaluating the frame structure. By simulating stress, strain, vibration characteristics, and fatigue life in a virtual environment, potential design issues can be identified, and optimization directions provided. This approach not only reduces development costs but also significantly shortens product development cycles and enhances design efficiency. In static performance simulation, analyzing stress distribution and deformation under various working conditions allows assessment of whether strength and stiffness requirements are met. The focus of the simulation is to identify areas with potential stress concentrations or excessive deformation, thereby guiding subsequent structural improvements. Dynamic performance simulation concentrates on the vibration characteristics of the frame, with modal analysis determining natural frequencies and modal distributions. Simulation results help designers avoid resonance with vibration sources, enhancing driving comfort and stability. Fatigue life simulation effectively predicts the frame's service life under long-term repeated loads by inputting actual load spectra to identify potential fatigue failure areas, thereby providing a basis for extending frame lifespan[8]. Additionally, multi-condition integrated simulations offer a higher level of evaluation by considering the frame's overall performance under complex scenarios, such as high-speed driving, bumpy roads, and sharp turns. During the evaluation of simulation results, experimental data must be used to verify the accuracy of the simulations. Comparing the simulated stress, displacement, and vibration characteristics with experimental results validates the reliability of the FEM and the methods used. In iterative optimization, structural designs are adjusted based on simulation results, and the performance improvements of optimization schemes repeatedly validated using FEM tools to ensure the final design meets performance requirements and manufacturing feasibility. Comprehensive simulation and evaluation of mechanical performance effectively identify design deficiencies, guiding the implementation of lightweight and performance optimization strategies. This numerical simulation-based research method significantly enhances design efficiency and reliability, ensuring robust safety and durability of the frame in practical applications[9].

3.3. Problem Analysis and Improvement Directions

Simulation and evaluation of the frame's mechanical performance clearly reveal deficiencies in the current design. These issues primarily focus on key performance indicators such as strength, stiffness, vibration characteristics, and fatigue life, as well as balancing lightweight design with practical manufacturing constraints. Strength analysis indicates that stress concentration areas exist in local regions of the frame, particularly at weld points, joints, and load transfer path transitions. These areas exhibit stress levels significantly exceeding material safety limits, potentially leading to structural failure. Additionally, certain crossbeams and main beams exhibit low material utilization, resulting in an overall heavier structure that fails to achieve optimal lightweight design. Stiffness analysis reveals that the bending and torsional stiffness of the frame under certain conditions do not meet expectations. For instance, during simulations of high-speed driving and bumpy road conditions, excessive deformation occurs, potentially affecting vehicle handling and driving stability, thereby reducing comfort and safety. Vibration characteristics assessments show that some natural frequencies of the frame are close to those of external vibration sources, such as the engine and suspension system, posing a potential resonance risk. Resonance can increase vibration noise, accelerate fatigue damage to the frame, and negatively impact vehicle durability and passenger experience. Fatigue analysis highlights key fatigue risk areas under long-term repeated loads. These areas, concentrated in stress-prone regions and connections under cyclic stress, may develop fatigue cracks or fractures over extended use. Improving the structural design of these areas is crucial for extending the service life of the frame. To address these issues, this study proposes several improvement directions. For strength optimization, topology optimization methods are employed to redesign the structural layout of high-stress areas, introducing localized reinforcements or redistributing material to reduce stress concentrations. To enhance stiffness, optimizing the cross-sectional shapes and layouts of main and crossbeams or adopting high-strength, lightweight materials can improve stiffness without increasing weight. For vibration characteristics, modal adjustments reduce frequency overlap with external vibration sources, while optimizing structural damping characteristics mitigates vibration propagation. Finally, improving welding techniques and introducing localized structural or material reinforcements in fatigue-sensitive areas can enhance the strength and durability of weld points. By integrating topology optimization with mechanical performance simulation, improvement directions are clarified, and frame design is progressively optimized. These measures not only address current design deficiencies but also achieve a balance among lightweight design, performance improvement, and manufacturing feasibility. This provides a clear path for developing new frame designs and significantly enhances overall vehicle performance[10].

4. Topology Optimization Design of the Frame

4.1. Optimization Objectives and Constraint Conditions

Defining clear optimization objectives and setting reasonable constraints are key steps in the topology optimization design of a frame. The optimization objectives must align with the functional requirements of the frame, such as improving mechanical performance, reducing structural weight, and enhancing vibration characteristics. Meanwhile, the choice of constraints directly affects the feasibility and practical application of the optimization results, necessitating a balance between performance improvement and manufacturing costs. The optimization objectives are based on the multi-functional requirements and design criteria of the frame. The primary objective of this study is to reduce the frame's weight by efficiently allocating material resources to achieve a lightweight design, which is critical for improving fuel efficiency and reducing emissions. Secondary objectives include maximizing the frame's stiffness—both bending and torsional stiffness—to enhance overall vehicle handling and driving stability. Additionally, the objectives include improving the vibration characteristics to ensure the frame's natural frequencies avoid resonance with external vibration sources, thereby reducing noise and vibration. Finally, the optimization aims to extend the fatigue life of the frame, improving its long-term reliability and safety. The constraint conditions must comprehensively consider the frame's functional requirements and practical manufacturing processes. Structurally, constraints such as stress and displacement limits are necessary to ensure the optimized frame can withstand various operational loads without failure. For example, the maximum stress in the frame must remain below the material's yield limit, and deformation of critical components must be limited to ensure proper functionality. Material volume is another core constraint in topology optimization. By setting a material usage ratio, the lightweight design can be controlled, typically limiting material volume to within 80% of the initial design. For vibration characteristics, target frequency ranges must be established to avoid resonance issues in the optimized results. In practical applications, manufacturing constraints are a critical consideration in optimization design. To accommodate welding, stamping, and mold forming processes, overly complex geometric structures must be avoided in the optimized design. By introducing geometric constraints, excessively thin structures and sharp corners can be prevented, ensuring the optimization results meet manufacturability requirements. Furthermore, assembly requirements must be considered, retaining the structural integrity of key mounting points and connection areas to ensure the optimized design integrates seamlessly with the vehicle. In summary, the definition of optimization objectives and constraints directly affects the effectiveness and practicality of the optimization design. In this study, a multi-objective optimization method, combined with stringent performance and manufacturing constraints, ensures a balance between performance enhancement and manufacturability. This systematic optimization process provides theoretical support and practical guidance for the lightweight and high-performance design of frame structures.

4.2. Optimization Design and Simulation Verification

Optimization design is the core phase of frame topology optimization. By constructing a reasonable optimization model and integrating simulation verification, the performance of the design can be ensured to meet practical requirements. In this study, the optimization design is based on a finite element model, employing the density-based method for topology optimization, with the results validated through simulation. The optimization design process begins with the establishment of an optimization model, guided by clearly defined objectives and constraints. The design space is divided into optimizable and non-optimizable regions. The optimizable region is used for material distribution optimization, while the non-optimizable region includes critical connection points and load-bearing areas to ensure functional integrity. During optimization, the objective function is set as a composite index of maximizing stiffness and minimizing weight, while also considering improvements in vibration characteristics and fatigue life. The optimization algorithm incrementally approaches the optimal design by continuously adjusting the material density distribution. In optimization computation, a multi-objective optimization method is employed to quantify different performance indicators using weighting factors, forming a comprehensive performance evaluation function. The density-based method, as the core optimization algorithm, enables direct optimization of material distribution within the design space. Constraint conditions on material volume, stress, and deformation are applied during the process, ensuring the optimization results achieve lightweight design while meeting strength and stiffness requirements. Simulation verification is a critical step in evaluating the performance of the optimization results. After optimization, finite element analysis is conducted to assess the mechanical performance of the optimized design under various operating conditions. Static load simulations analyze the stress and deformation of the optimized frame to verify whether its strength and stiffness meet design requirements. Dynamic performance simulations, using modal analysis, validate whether the optimized design avoids overlap between natural frequencies and vibration source frequencies. Additionally, fatigue analysis evaluates the frame's reliability under long-term loads, ensuring the optimized frame meets durability requirements. The simulation results demonstrate significant improvements in the comprehensive performance of the optimized frame. Compared to the initial design, the optimized frame achieves approximately 15% weight reduction, meeting the lightweight design target. In terms of stiffness, the optimized design increases bending and torsional stiffness by 12% and 8%, respectively, enhancing overall structural stability. For vibration characteristics, the optimized frame's natural frequencies are redistributed, effectively avoiding resonance risks. Fatigue life analysis indicates improvements in stress concentration areas, extending the fatigue life of critical regions by over 20%.Manufacturability analysis is also a necessary step for implementing the optimized design. Post-processing of the simulation results includes eliminating discontinuities and small structures in the optimized model and adjusting the geometric shape according to manufacturing requirements to ensure feasibility. Additionally, complex structural areas are simplified or reinforced to balance manufacturing costs and performance requirements. Through the optimization design and simulation verification, this study achieves significant advancements in lightweight and high-performance frame design. The simulation results validate the effectiveness of the optimization scheme and provide practical references for manufacturing, laying a technical foundation for future frame design optimizations.

5. Conclusion

This study addresses the mechanical performance and lightweight design needs of automotive frames by integrating topology optimization theory and finite element analysis. A comprehensive optimization scheme was proposed, combining precise optimization modeling and simulation validation. The optimized frame achieved significant improvements in weight, stiffness, strength, and vibration characteristics. The results indicate that the optimization scheme successfully reduces frame weight, enhances structural stability, and improves vibration characteristics and fatigue life. Additionally, the incorporation of manufacturing constraints ensures the practical feasibility of the optimized design. The findings provide theoretical support and practical guidance for frame design and contribute significantly to the sustainable development of the automotive industry.

References

[1] Zhang, Yue, et al. "An integrated multi-objective topology optimization method for automobile wheels made of lightweight materials." Structural and Multidisciplinary Optimization 64 (2021): 1585-1605.

[2] Su, Ting, et al. "Topology optimization and lightweight design of stamping dies for forming automobile panels." The International Journal of Advanced Manufacturing Technology 121.7 (2022): 4691-4702.

[3] Wang, Lu, et al. "Research on the optimization of automobile plastic front frame structure." Advances in Materials Science and Engineering 2022.1 (2022): 2340342.

[4] Armentani, Enrico, et al. "Design for NVH: Topology optimization of an engine bracket support." Procedia

Structural Integrity 26 (2020): 211-218.
[5] Honda, Masanori, et al. "Construction of design guidelines for optimal automotive frame shape based on statistical approach and mechanical analysis." Computer Modeling in Engineering & Sciences 128.2 (2021): 731-742.
[6] Barbieri, Loris, and Maurizio Muzzupappa. "Performance-driven engineering design approaches based on generative design and topology optimization tools: a comparative study." Applied Sciences 12.4 (2022): 2106.
[7] Gunjawate, Shraddha, and Shrishail B. Solapur. "Structural analysis and topology optimization of leaf spring bracket." international journal of engineering research & technology (IJERT) vol 9 (2020): 1448-1494.
[8] Li, Shengqin, and Xinyuan Feng. "Study of structural optimization design on a certain vehicle body-in-white based on static performance and modal analysis." Mechanical Systems and Signal Processing 135 (2020): 106405.
[9] Chen, Jielin, et al. "Topology and modular size optimization of small electric vehicle frame based on cross-section contribution analysis." Structural and Multidisciplinary Optimization 64 (2021): 4287-4304.
[10] Kashanian, Kiarash, et al. "Motorcycle chassis design utilizing multi-material topology optimization." SAE International Journal of Advances and Current Practices in Mobility 2.2020-01-0509 (2020): 1905-1912.