DOI: 10.21062/mft.2024.053 http://www.journalmt.com

Analysis of Static and Dynamic Characteristics and Lightweight Design of Titanium Alloy Frame

Bin Zheng (0000-0002-8614-6355)
School of Intelligent Manufacturing, Panzhihua University, Panzhihua, 617000, China. Email: 22198334@qq.com

In the field of modern automobile manufacturing, the frame, as the core load-bearing component of automobiles, has always been a hot research topic in terms of strength, stiffness, and lightweight design. In response to the shortcomings of traditional car frames in strength and stiffness, as well as the problem of weight redundancy, this research focuses on titanium alloy frames and conducts in-depth analysis of static and dynamic characteristics and lightweight design. Firstly, this article used the finite element analysis method to simulate four representative working conditions and performed a static analysis on the titanium alloy frame. Through analysis, the stress distribution and deformation of the frame under different load conditions were obtained, providing an important basis for the subsequent optimization design. Second, to gain a more comprehensive understanding of the dynamic performance of the frame, this research conducted a modal analysis and explored the natural frequencies and vibration modes of the frame. These analysis results not only reveal potential problems in the vibration process of the frame but also provide strong guidance for subsequent optimization design. Third, through harmonic response analysis, the changes in amplitude and frequency of the frame during use were thoroughly studied, further verifying the rationality of the frame structure and the stability of dynamic performance. On the basis of the analysis of the static and dynamic characteristics of the frame, topology optimization and lightweight design were carried out on the frame. Through refined design adjustments, the weight of the frame has been successfully reduced while improving its strength and stiffness. Finally, this research validated and compared the optimized titanium alloy frame. The results showed that the optimized frame weight was reduced by 13.76%, the maximal stress was reduced by 5.19%, and the maximal deformation was reduced by 0.37%. The improved performance of the optimized frame fully demonstrates the feasibility and superiority of the optimization plan. This research not only provides a method to analyze the static and dynamic characteristics and the design of the topology optimization of the frame but also provides strong technical support for the lightweight development of the automotive manufacturing industry.

Keywords: Titanium alloy frame, Topology optimization, Finite element analysis, Lightweight design

1 Introduction

The frame is the skeleton of a car, which combines components such as the chassis, engine, and body as a whole. A metal skeleton structure is used to connect major components such as wheels, suspension system, engine, and transmission, which play an important role in supporting vehicle weight, ensuring driving stability, connecting various components, and providing safety assurance [1, 2]. The frame bears loads and moments during driving, therefore, it must withstand various static and dynamic loads and maintain strength and stiffness under complex usage conditions. Many researchers have carried out various studies on the frame, including static and dynamic characteristics and stiffness analysis [3, 4].

Li et al. [5] analyzed and studied the strength of a certain frame using the finite element method, combined with the finite element model of the frame and the stress testing of the sample car frame, indicating that the frame strength is insufficient under four-point support and soil pushing conditions. He optimized the design of the frame and provided a frame improvement plan, providing a technical method for the analysis and research of related vehicle frames. To evaluate the safety performance of a new pure electric bus frame, Zhang et al. [6] conducted analysis on the bus frame and obtained the maximal stress and deformation of the frame. On the basis of the static analysis results, the size optimization was used to obtain the main size parameters. Gong et al. [7] focused on the issue that modifying car parameters affects overall performance of the frame. Taking the trapezoidal frame of a truck as the research object, he designed a parameterized platform for the car frame as a design tool and used ANSYS software to verify the rationality of the frame designed by the platform.

Tian [8] established a discretized model of the front sub-frame by using the finite element method to analyze vibration and noise faults generated by the frame. The results of the analysis showed that its first natural frequency was lower than the engine excitation frequency, resulting in resonance. On the basis of the optimization platform, optimization analysis was performed on the structural parameters of the frame and actual testing was conducted. The test results showed that after optimization, the vibration acceleration was significantly reduced and the noise was significantly reduced. He et al. [9] took a certain car frame as the research object, researched vibration shapes of the first to twelve orders of the frame, and conducted modal tests on the frame using the LMS vibration testing system. A static strength analysis was conducted under two working conditions, full load bending and full load torsion, to verify whether the strength can meet the normal driving strength requirements. Jiang et al. [10] researched the vibration shapes of a racing car frame in a free state using a combination of computational modes and experimental verification. The modal confidence criteria for obtaining experimental modes and verifying the experimental parameters indicate that the natural frequency error between the calculated and experimental modes is around 5%, verifying the accuracy of the calculated modes.

To analyze the static and dynamic characteristics of a certain frame, Mi et al. [11] conducted a static strength analysis under full load and then verified the correctness of the model through road stress tests. Subsequently, dynamic strength analysis, stiffness analysis, and modal analysis were carried out under fully loaded horizontal bending conditions, ultimate torsion conditions, and emergency braking conditions. Then, topology optimization technology was used to perform multi-objective optimization under multiple working conditions. Liu et al. [12] take a low-speed electric vehicle frame as the research object, then establish its finite element model, and conducted modal, strength, and bending torsional stiffness analysis. Then, the multi-objective optimization was carried out on the low-speed electric vehicle frame. The weight has been reduced by 27.6 kg, and its first-order frequency, strength, and bending torsional stiffness meet the design requirements. Zou et al. [13] solved the failure problem of the second crossbeam and verified the correctness of the frame and its finite element model of strength calculation through modal and stress tests. Nonlinear strength calculations were conducted on the frame based on the actual usage conditions of the user, identifying the main reasons for the failure of the second crossbeam and optimizing it to solve the problem of frame failure. Gu et al. [14] established a three-dimensional model and finite element model of a semi-trailer frame and conducted the strength, stiffness, and modal analysis. The research results showed that the optimized frame can meet the requirements, the modal frequency also avoided the vibration frequency caused by the excitation of the engine, and achieved the optimization goal.

To achieve the lightweight design of the frame, many scholars have conducted optimization designs based on different methods [15, 16]. He Rui [17] established a frame model for performance analysis and lightweight problem of a certain truck frame, conducted a bending stiffness analysis and bending stiffness tests on it, and conducted a multidisciplinary and multi-objective optimization design on the thickness values of the main components based on the integrated platform. The results of the analysis showed that the static and dynamic performance can meet the design requirements, with a total weight reduction of 9.3 %. Liao et al. [18] studied the lightweight design method for the aluminum alloy rear frame, selecting 12 parameterized frame shape as lightweight multidisciplinary optimization design variables to obtain the optimal shape that meets the target requirements. After lightweight, the weight decreased by 1.84 kg, approximately 7.4 %. Jiang and Wu [19] established a certain frame grid model and conducted the modal performance analysis and strength analysis on it. Multi-objective genetic algorithm is used to obtain the best size parameters. After optimization, the weight is successfully reduced by 9.9 %.

Li et al. [20] used finite element analysis to analyze the lightweight problem of a heavy-duty truck frame, taking the thickness of each main beam as a design variable. Through size optimization methods, he conducted lightweight research to reduce frame weight while ensuring its reliability. After optimization, the weight has been reduced by 7.3 %, and its strength, natural frequency, and fatigue life can all meet the usage requirements, achieving the goal of a lightweight frame. Wu et al. [21] took a certain frame as the research object and established a finite element model. The stress and deformation were obtained under two working conditions of bending and torsion. Based on the analysis results, the frame is optimized to be lightweight by changing the cross-sectional area of the beam and changing the volume of the frame, ultimately achieving a lightweight frame. The results indicate that the frame mass has decreased by 10.6 % compared to the initial design. To improve the utilization rate of a certain car frame material, Yu et al. [22] conducted static and modal analysis on the frame model. On the basis of the analysis results, structural improvements were made. By establishing a multi-objective size optimization model for the frame, a more reasonable optimization plan was obtained. The weight of the lightweight frame was reduced by 74.58 kg compared to the original frame.

Xu et al. [23] used an experimental design method to analyze the sensitivity of design variables for a certain frame as an optimization object and established a structural optimization model based on the necessary selection of design variables. The genetic algorithm was used to calculate maximal stress, displacement and natural frequency to achieve the goal of satisfying natural frequency optimization of the discretized dimensions of each plate of the frame under constraints such as geometric dimensions and strength. Wang et al. [24] proposed a lightweight optimization design evaluation method for the frame. The results of both finite element analysis and experimental verification were within the error range, demonstrating the correctness of the evaluation method and providing strong support for the lightweight. Zhou et al. [25] researched a certain car frame and carried out static and modal analysis, and optimized it. The frame crossbeam material was changed from A610L to Q345, reducing the weight of the entire vehicle by 142.6 kg, and improving the overall strength and stiffness of the entire vehicle and its components.

To investigate the static and dynamic performance of titanium alloy frame in depth, and to make up for the shortcomings of frame optimization and lightweight design in the current research field of titanium alloy frames, this research adopted a series of numerical simulation methods. Firstly, a solid model of the titanium alloy frame is constructed using 3D modeling software, followed by static and dynamic analysis in ANSYS software. Finite element analysis was conducted on the titanium alloy frame through meticulous mesh division, precise application of constraint, and reasonable load loading. This analysis process not only reveals the deformation and equivalent stress distribution of the titanium alloy frame under different working conditions but also obtains its natural frequency, modal vibration mode, and displacement frequency response curve, providing rich data support for the dynamic performance evaluation of the frame. Based on the above analysis results, maximum utilization of titanium alloy frame materials was explored through topology optimization design. This optimization process aims to achieve lightweight of the frame while ensuring that its stress and strength meet the design requirements.

2 3D Modeling of Titanium Alloy Frame and Definition of Material Properties

The research object of this article is the titanium alloy frame, which adopts a side beam structure and mainly consists of two longitudinal beams and eight crossbeams. In the frame design, the longitudinal beams and crossbeams are connected by riveting and welding. The longitudinal beam is made of the 7 mm thick steel plate by stamping and forming, with a total length of 5500 mm. The flange width of the longitudinal beam is 70 mm, and the height of the web plate is 200 mm. To connect the crossbeam and the steel plate spring, there are connection holes in the flange and web plate. This connection method helps to ensure overall stiffness and strength. The crossbeam is divided into front, middle, rear crossbeams, and rear suspension crossbeams, with varying lengths and heights, but uniformly having a width of 75 mm. The connection between the longitudinal beam and the crossbeam is completed through riveting and welding [26]. The connecting piece serves as an intermediate part and connects the longitudinal beam and crossbeam through the rivets. In addition, fillet treatment is carried out on the corresponding parts of the longitudinal beam to reduce stress concentration, improve the strength and durability of the structure, and ensure the service life of the frame assembly. Based on the above data, a solid model of the titanium alloy frame was established, as shown in Fig. 1.

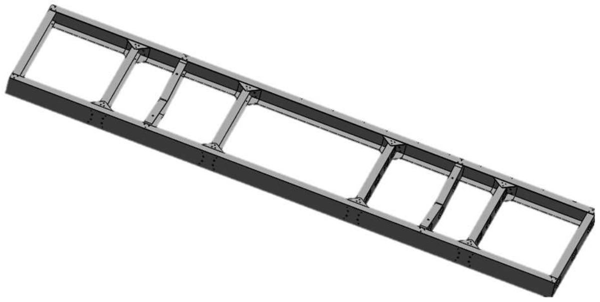


Fig. 1 *Solid model of the titanium alloy frame*

The main parameters of the titanium alloy used for the frame material, such as elastic modulus, Poisson's ratio, density, etc., are shown in Tab. 1.

Tab. 1 *Material properties of titanium alloy*

Density [kg.m^{-2}]	Poisson's ratio	Elastic modulus [Pa]	Yield strength [MPa]	Tensile strength [MPa]
4620	0.36	9.6E+10	930	930

3 Static Analysis of Titanium Alloy Frame

3.1 Mesh division of the titanium alloy frame model

Mesh partitioning is a crucial step in finite element analysis, which discretizes complex continuous problems into a finite number of connected elements. The unknown quantity of each node can be expressed through approximate functions, which in turn approximate the distribution of the internal field problem of each element, thus approximating the field function of the entire continuum [27, 28].

When the titanium alloy frame is meshed, an automatic meshing method is used. Through this grid division method, the precise grid division of the titanium alloy frame was achieved, and the final established grid model is shown in Fig. 2. By fine mesh division, the detailed features of the frame structure can be better captured, providing a reliable foundation for subsequent simulation analysis.

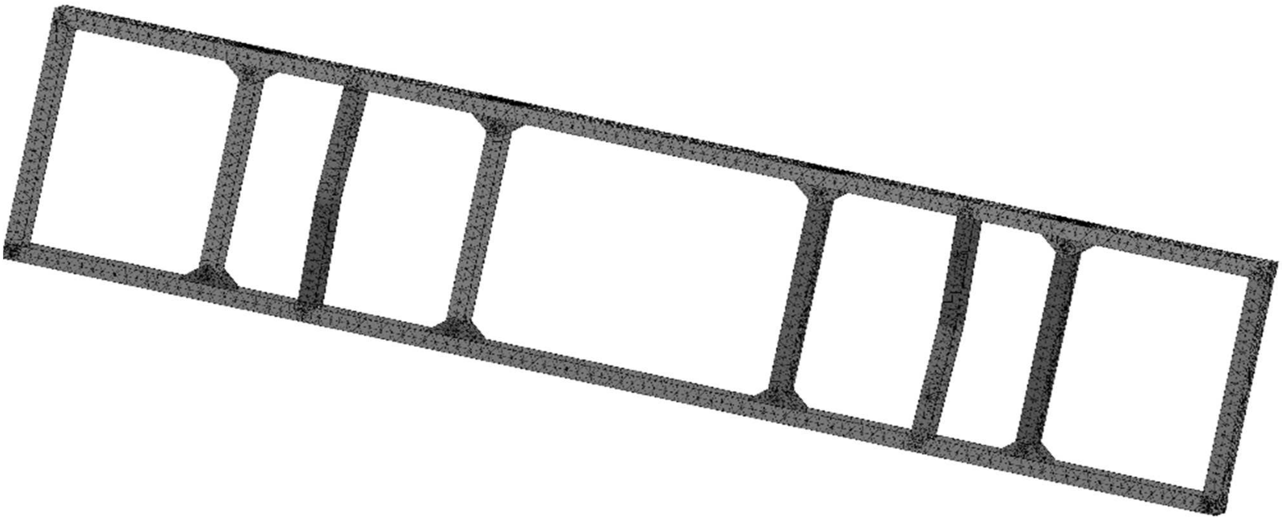


Fig. 2 *Titanium alloy frame grid model*

3.2 Quality inspection of the titanium alloy frame grid

Due to the direct impact of grid quality on subsequent finite element analysis, optimizing grid quality can be achieved through grid topology, appropriate grid density, and excellent node connection methods. Through the above operation, the grid quality of the titanium alloy frame was checked, and the element quality distribution histogram is shown in Fig. 3. The element quality is mainly concentrated between 0.50 and 1.00. Within this range, the closer the element quality is to 1, the better the element quality is, while closer to 0, the worse the grid quality is.

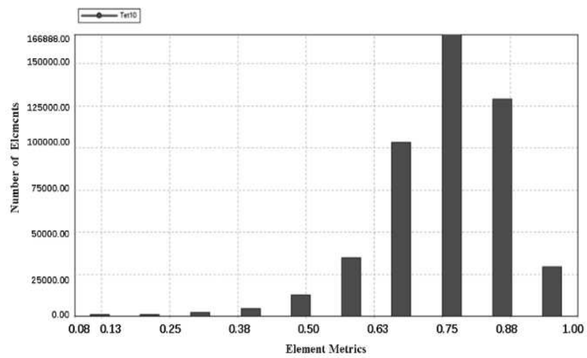


Fig. 3 *Element quality distribution of titanium alloy frame*

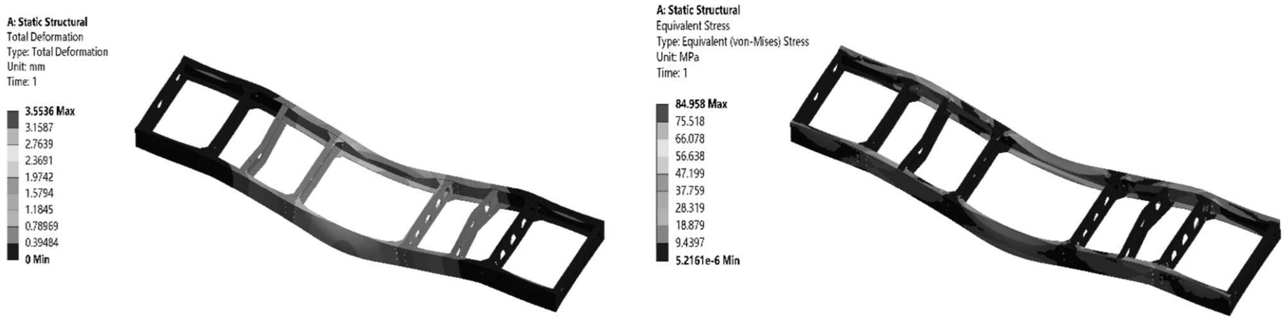
3.3 Setting boundary condition and analysis of different working conditions

After the titanium alloy frame is meshed, to accurately calculate the stress and deformation, it is necessary to set appropriate boundary conditions in the model. By correctly setting constraints and adding boundary conditions such as external loads, the accuracy of the finite element model can be ensured.

3.3.1 Bending conditions

Under this working condition, set the load as follows. The rated load of the truck (six tons) is applied as a uniformly distributed load on the surface of two longitudinal beams, and the calculated pressure is $6\times1000\times10\div0.47=127659$ Pa. There are a total of eight lifting lugs and fixed supports connecting the steel plate springs to the frame assembly model, and four key points on each longitudinal beam are set as fixed constraints. The boundary conditions for these 8 key points are set as follows. The degrees of freedom X, Y, and Z at the front end of the left longitudinal beam are constrained, the degrees of freedom Y and Z at the end of the left longitudinal beam are constrained, the degrees of freedom X and Z at the front end of the right longitudinal beam are constrained, and the degrees of freedom Z at other positions are constrained. Finally, add an inertial load to the frame, which is standard gravitational acceleration.

After analysis and calculation, the maximal deformation occurs at the outer edge of the upper surface of the middle section of the right longitudinal beam, with a maximum deformation of 3.5536 mm, as shown in Fig. 4. In the bending condition, where the vehicle is stationary or traveling at a constant speed with all four wheels on the ground, under a load of 6 tons, there will be a deformation of 3.5536 mm along the -Y direction at the outer edge of the upper surface of the left and right longitudinal beams.



(a) Total deformation cloud map (b) Equivalent stress cloud map

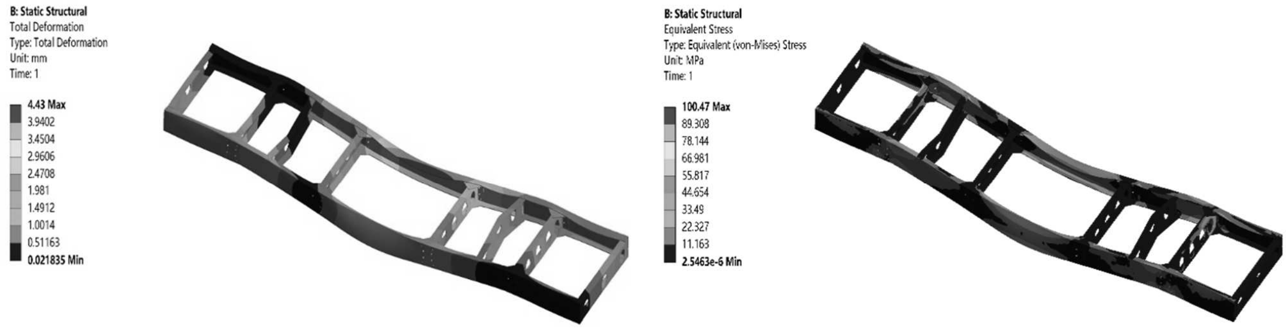
Fig. 4 *Static analysis results under bending condition*

The maximal stress is 84.958 MPa, and the maximal stress point appears at the corner position of the lower surface in the middle of the left longitudinal beam. There is a phenomenon of stress concentration, but the maximum stress is within a reasonable range. In the bending condition, where the vehicle is stationary or traveling at a constant speed with all four wheels on the ground, with a load of six tons, the strength and stiffness of the frame are sufficient, far lower than the tensile strength of titanium alloy materials, and there is no load-bearing problem. However, further analysis and improvement are needed in this area.

3.3.2 Torsional working conditions

Under the torsion condition, the load is the same as the bending condition load setting. According to the limited degrees of freedom under frame torsion conditions, the front end of the left longitudinal beam is constrained in the X, Y, and Z directions, the end of the left longitudinal beam is constrained in the Y and Z directions, the front end of the right longitudinal beam is constrained in the X and Z directions, the rear two points of the right longitudinal beam are constrained in the X and Y directions, and the degrees of freedom in the Z direction of other positions are constrained.

After analysis, the maximal deformation is located at the outer edge of the upper surface of the right longitudinal beam, with a maximal deformation of 4.43 mm, as shown in Fig. 5. Under torsional conditions, where the vehicle is loaded with three wheels and the right front wheel is suspended, a 4.43 mm deformation along the - Y direction will appear at the right middle section of the frame. From the above analysis, the maximal deformation of the frame is relatively large and further improvement and optimization are needed in this area. By adjusting the structural design, deformation can be reduced and the stiffness and strength of the body structure can be improved, thus improving the safety and stability of the vehicle. The maximal stress point appears on the lower surface of the rear section of the left longitudinal beam, with a maximal stress of 100.47 MPa, which is less than the tensile strength of the longitudinal beam of 930 MPa.

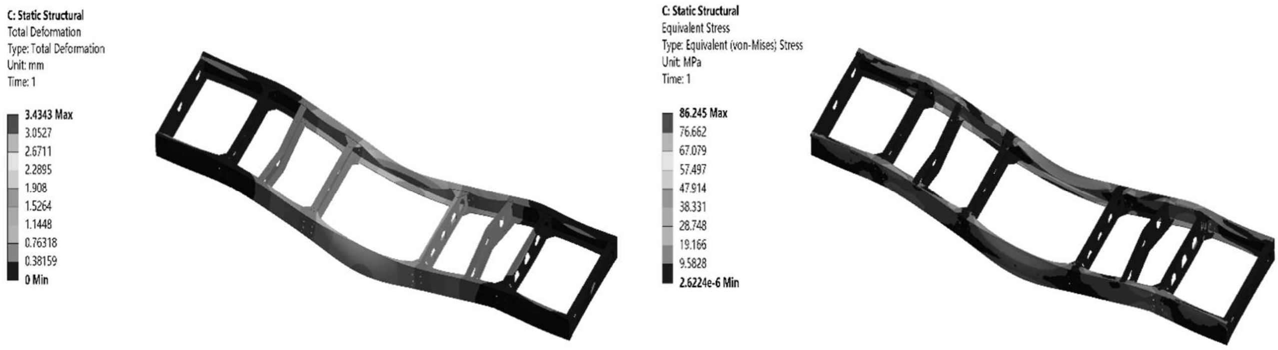


(a) Total deformation cloud map (b) Equivalent stress cloud map

Fig. 5 *Static analysis results under torsional working conditions*

3.3.3 Emergency braking conditions

Under emergency braking conditions, the load and boundary conditions are the same as under bending conditions. However, in addition to applying the inertia load to the frame, the standard gravity acceleration, it is also necessary to set an emergency braking acceleration of -0.8×9.8 m.s^{-2}. After analysis and calculation, the maximal deformation position occurs at the outer edge of the upper surface of the left longitudinal beam, with a maximal deformation of 3.4343 mm, as shown in Fig. 6.



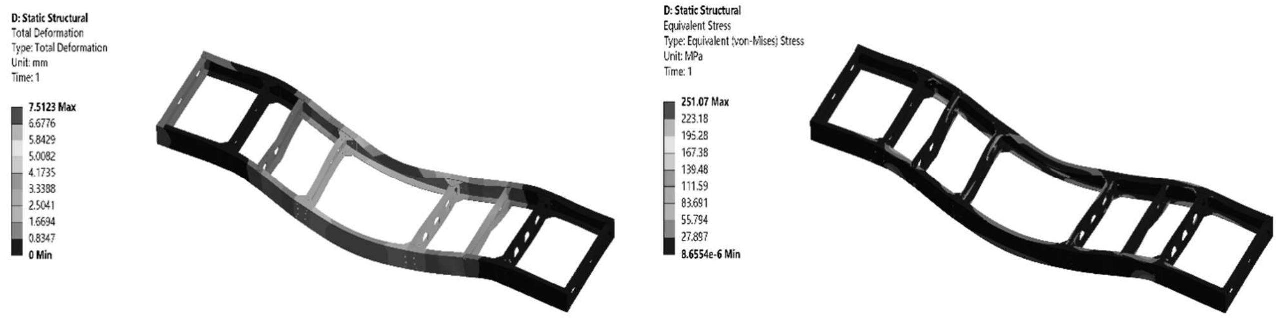
(a) Total deformation cloud map (b) Equivalent stress cloud map

Fig. 6 *Static analysis results under emergency braking conditions*

The maximal stress occurs in the rear corner of the left longitudinal beam, with a maximal stress of 86.245 MPa, which is less than the tensile strength of the longitudinal beam of 930 MPa. Further analysis and improvement are needed in this area. It may be necessary to improve the stiffness and strength of the body structure to enhance the safety and stability of the vehicle.

3.3.4 Emergency turning conditions

Under emergency turning conditions, the load and boundary conditions are the same as those under bending conditions. Apply the inertia load, the standard gravity acceleration, to the frame assembly, while also adding an emergency turning acceleration of 0.5×9.8 m.s^{-2}. After analysis and calculation, the maximal deformation occurs at the outer edge of the upper surface of the middle section of the right longitudinal beam, with a maximal deformation of 7.5123 mm, as shown in Fig. 7. The analysis results indicate that the occurrence of the maximal deformation variable at this location bears significant stress and deformation, and further analysis and improvement are needed in this area. It may be necessary to adjust the frame structure or take other design measures to reduce deformation.



(a) Total deformation cloud map (b) Equivalent stress cloud map

Fig. 7 *Static analysis results under emergency turning working conditions*

The stress in the rear corner of the left longitudinal beam is the maximum value, with a maximal stress of 251.07MPa, which is less than the tensile strength of the longitudinal beam of 930MPa.

4 Dynamic Analysis of the Titanium Alloy Frame

4.1 Modal analysis of the titanium alloy frame

Modal analysis is an engineering technique that combines structural dynamics principles and finite element analysis techniques to analyze and describe the inherent vibration characteristics of structural systems. Modal analysis is not affected by external excitation but is based on the internal characteristics of the structure, such as mass distribution, damping, and stiffness. Modal analysis can provide important information for optimizing structural design and can predict the vibration response with different external excitations.

The expression of its dynamic equation is shown in Equation (1).

$$[M]\{\ddot{x}\}+[C]\{\dot{x}\}+[K]\{x\}=[F] \tag{1}$$

Where:

[*M*]…The total mass matrix.

[*C*]…The total damping matrix.

[*K*]…The equivalent load array for the nodes.

[*F*]…The node displacement array.

When the external excitation is zero, the undamped free vibration equation is obtained, as shown in Equation (2).

$$[M]\{\ddot{x}\}+[K]\{x\}=0 \tag{2}$$

This equation is a system of homogeneous differential equations with constant coefficients, and its solution is shown in Equation (3).

$$\{x\}=\{x_0\}\sin(\omega t+\varphi) \tag{3}$$

Then, Equation (3) is added to the Equation (2) to obtain the Equation (4).

$$\left(|K|-\omega^2|M|\right)\{x_0\}=0 \tag{4}$$

If Equation (4) has a solution, there is a characteristic equation, as shown in Equation (5).

$$\left(|K|-\omega^2|M|\right)=0 \tag{5}$$

Based on static analysis, the corresponding boundary conditions are added, the damping is turned off, and the type of solution is controlled spontaneously by the system. Meanwhile, considering that high-order natural frequencies do not have much significance for the actual analysis and also consume too much calculation time, the first-order to sixth-order modes of the titanium alloy frame are solved and the analysis results are as in Fig. 8.

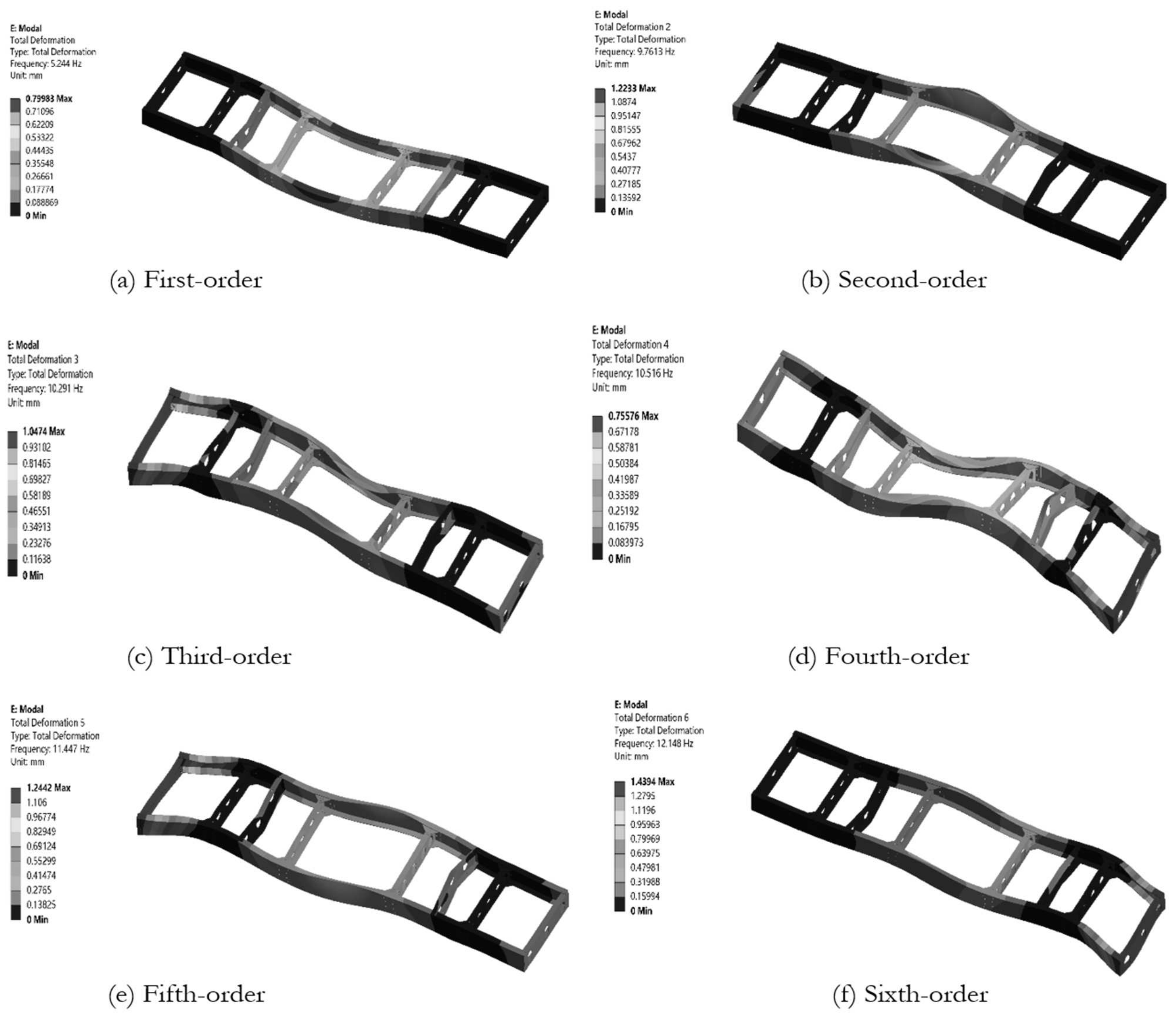


Fig. 8 *Results of modal analysis*

According to results and analysis of the ANSYS calculation, the first-order natural frequency of the titanium alloy frame is 5.244 Hz, and the maximum amplitude point is determined to occur at the outer edge of the upper surface of the middle section of the right longitudinal beam, with a maximum amplitude of 0.79983 mm. The second natural frequency is 9.7613 Hz, and the maximum amplitude point is located at the outer edge of the upper surface of the left longitudinal beam. At this position, the vibration of the frame reaches its maximum value, with a maximum amplitude of 1.2233 mm. The third natural frequency is 10.291Hz, and the maximum amplitude point occurs at the corner of the junction between the front end of the right longitudinal beam and the first crossbeam, with the maximum amplitude of 1.0474 mm. The fourth natural frequency is 10.516 Hz, and its maximum amplitude occurs at the corner between the rear end of the right longitudinal beam and the eighth crossbeam, with the maximum amplitude of 0.75576 mm. The fifth natural frequency is 11.447 Hz, and the maximum amplitude point occurs in the middle of the first crossbeam. This position may be determined by factors such as beam length, fixing conditions, or load distribution. The amplitude of the frame structure in the fifth mode reached 1.2442 mm, indicating that the vibration amplitude in this area is relatively large. The sixth natural frequency is 12.148Hz, and the maximum amplitude occurs at the corner where the rear end of the right longitudinal beam intersects with the eighth crossbeam, with the maximum amplitude of 1.4394 mm. The maximum amplitude point at the corner where the rear end of the right longitudinal beam intersects the eighth crossbeam is a location that needs attention. This may indicate significant stress concentration or low structural stiffness in the area, which can easily lead to vibration problems or fatigue damage.

4.2 Harmonic response analysis of titanium alloy frame

Harmonic response analysis is a method used to study the steady-state response characteristics of structures under sinusoidal loads. It can be used to predict the response behavior of structures under continuous dynamic excitation and identify potential structural or design defects for optimal design and improvement [29]. By conducting a modal analysis on the frame, multiple natural frequencies can be calculated, each corresponding to a vibration mode of the frame. Then, for sinusoidal loads, the response analysis of the frame is carried out to calculate the response amplitude and phase angle at different excitation frequencies. The response amplitude represents the vibration amplitude of the frame at a specific excitation frequency, while the phase angle describes the relative phase difference of the vibration. By comparing the response amplitude and phase angle with the excitation frequency, one can determine whether there is a resonance phenomenon. If the resonance phenomenon is found, appropriate measures need to be taken to reduce the vibration amplitude to ensure stability and safety.

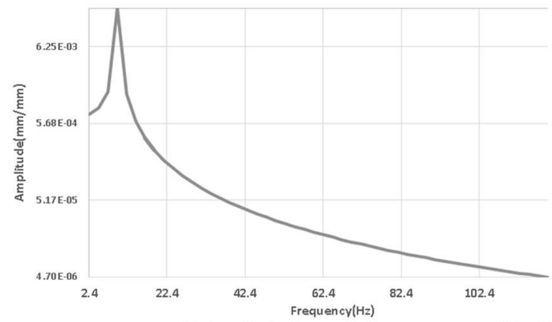
The dynamic equation under harmonic excitation is shown in Equation (6).

$$[M]\ddot{x}+[C]\dot{x}+[K]x=P(t) \tag{6}$$

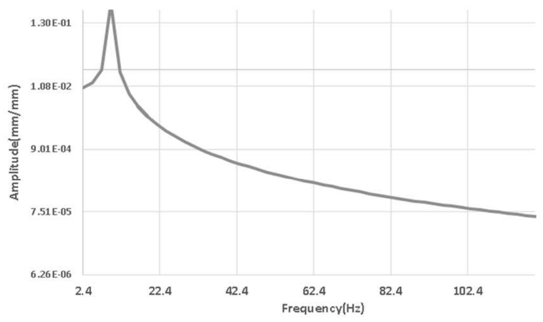
In the harmonic response analysis, the excitation force is a simple harmonic force, so $P(t)$ is shown in Equation (7).

$$P(t) = P\sin\omega t = Pe^{j\omega t} \tag{7}$$

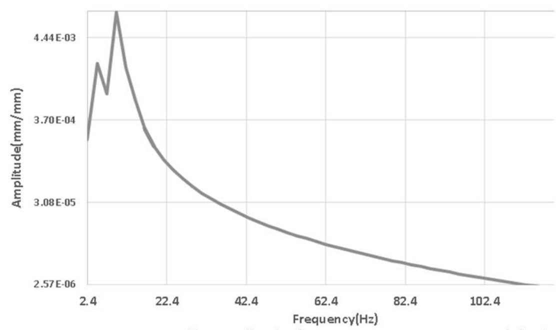
The harmonic response analysis is used to determine the vibration amplitude changes of titanium alloy frames at different frequencies and evaluate their interaction with excitation loads. The vibration response of the frame under actual working conditions can be predicted. Through harmonic response analysis, corresponding optimization decisions can be made to improve vibration performance and reliability. The results of the harmonic response analysis are usually presented in a graphical form, including vibration amplitude, vibration modes, and their corresponding frequencies. When these results are studied, the potential resonance frequencies and corresponding vibration modes can be determined, and targeted design measures can be taken, such as adjusting structural stiffness and increasing damping, to avoid resonance or reduce its impact. The results of the harmonic response analysis on the titanium alloy frame are shown in Fig. 9.



(a) Amplitude-frequency response curve of the X-axis



(b) Amplitude-frequency response curve of the Y-axis



(c) Amplitude-frequency response curve of the Z-axis

Fig. 9 *Harmonic Response Analysis Results*

As shown in Fig. 9, when the frequency is between 5 Hz and 15 Hz, the displacement value in the X direction reaches its maximum, with a maximum value of 6.25×10^{-3} mm, the displacement changes tend to flatten with increasing frequency. The displacement value in the Y direction reaches its maximum, with a maximum value of 0.13mm, and the displacement changes tend to flatten out with increasing frequency. The displacement value in the Z direction reaches its maximum, with a maximum value of 4.44×10^{-3} mm, the displacement changes tend to flatten with increasing frequency. Through the analysis of harmonic response in three directions, it can be seen that the frequency range from 5 Hz to 15 Hz has the greatest impact on frame vibration, especially in the Y direction.

5 Optimization Design of the Titanium Alloy Frame

5.1 Overview of the optimized design

Modern automotive structural optimization design uses mathematical theory, mechanical analysis methods, and computer-aided design software to achieve optimal design of parts. Structural optimization design can target single or multiple optimization objectives, such as improving performance, reducing weight, increasing stiffness, reducing vibration and noise, etc. Size optimization mainly achieves the optimization of design requirements and performance indicators by changing their size parameters [30]. Shape optimization refers to achieving optimal design by adjusting the shape of a part. Topology optimization is a more advanced optimization method that achieves optimal use of materials by reallocating their positions in the structure.

Topology optimization determines the main load path and the area of stress concentration by analyzing the structure of the frame, providing a basis for the strength requirements and material distribution of the frame. The purpose of topology optimization is to minimize the weight of the frame while meeting requirements such as strength and stiffness. When optimizing the frame, the load and boundary conditions are set according to static analysis. The adopted mathematical model is shown in Equation (8).

$$\begin{aligned} &Find \quad x = (x_1, x_2, \cdots, x_n)^T \\ &Min \quad C(x) = F^T U \\ &s.t. \quad \begin{cases} V \le V^* \\ F = KU \\ 0 < x_{\min} \le x_i \le 1 (i = 1, 2, \cdots, n) \end{cases} \end{aligned} \tag{8}$$

Where:
x_i…The design variable,
0…Means to delete the element,
1…Means to keep the element,
n…The number of optimization design variables,
K…The total stiffness matrix,
U…The displacement vector,
F…The force vector,
V…The original volume before optimization,
V^*…The volume after optimization.

5.2 Optimization Analysis of Topologies

By topology optimization, the red areas are marked as removed regions during the optimization process, which means that the materials in these areas can be deleted or reduced in the optimal design. Brown areas are identified as edge areas, which may indicate boundaries or special requirements in the design. The gray part represents the reserved areas, which are determined as the material areas that need to be retained in the optimal design. After optimization, an optimized model was generated. The model has undergone structural optimization, minimizing the weight of the model and improving performance indicators such as stiffness and strength without changing the overall size of the model. The optimized finite element analysis model can be used for further analysis and validation, and the necessary adjustments and improvements can be made. The convergence curve for model optimization is shown in Fig. 10, and the optimization and exclusion region of the frame model is shown in Fig. 11.

Fig. 12 shows the results of topology optimization of titanium alloy frame. In order to optimize the frame structure, the deleted areas and reserved areas are clearly marked in the model.

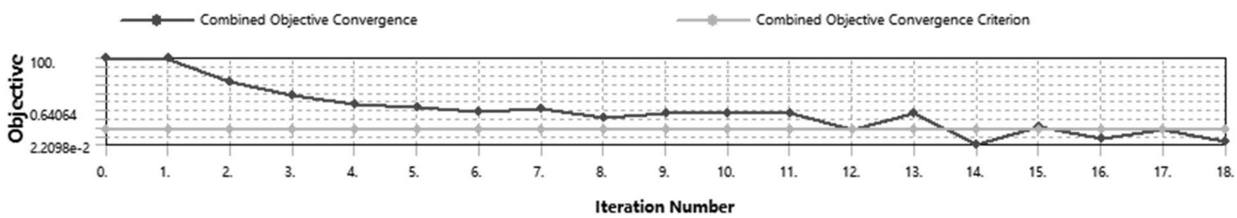


***Fig. 10** Convergence curve of the titanium alloy frame optimization*

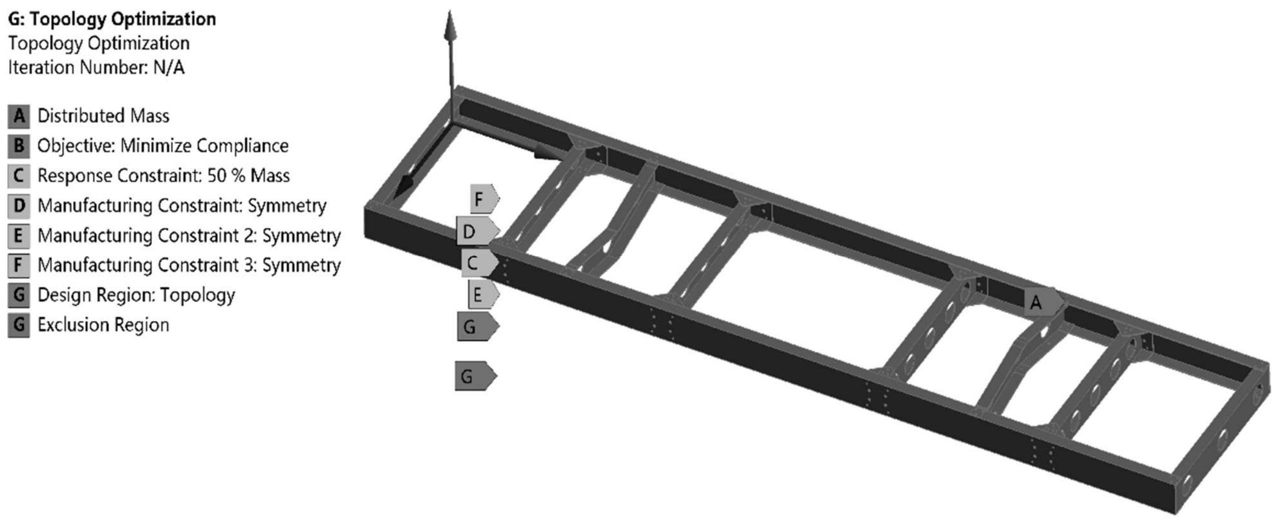


Fig. 11 *Optimization and exclusion region of the titanium alloy frame model*

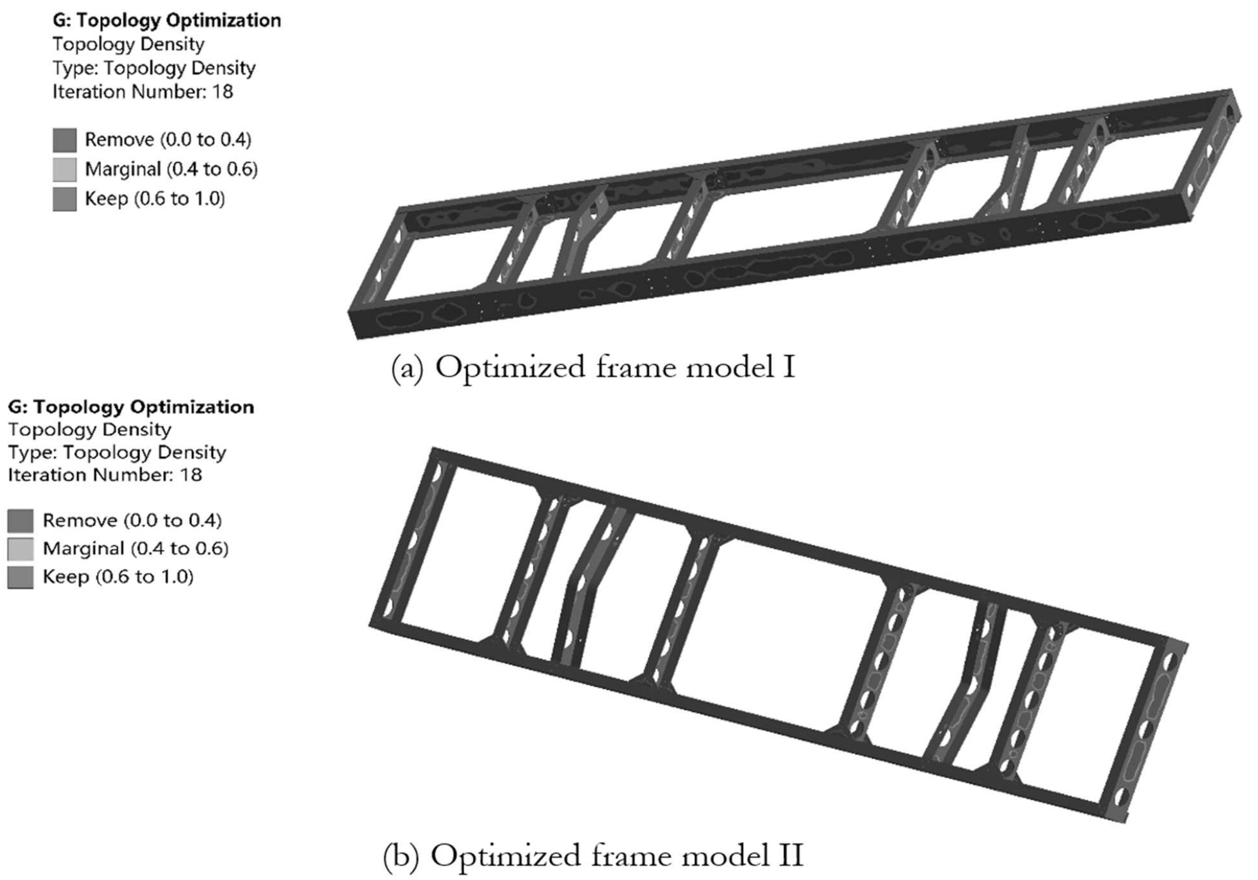
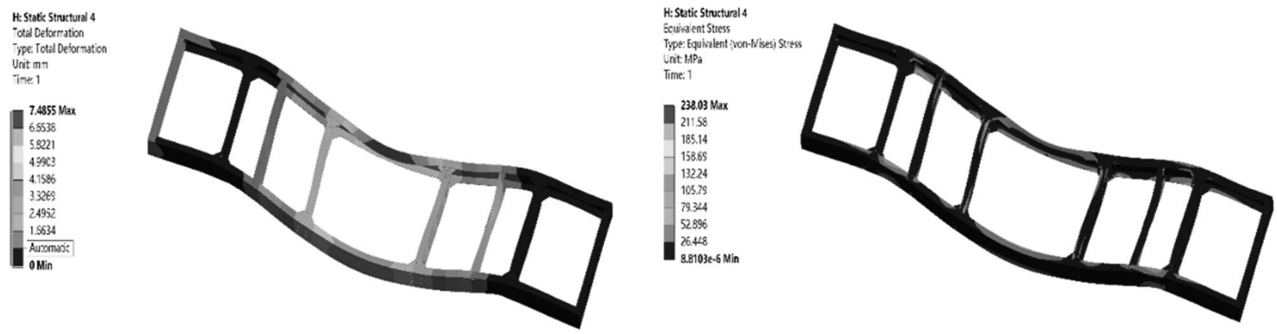


Fig. 12 *Optimized titanium alloy frame model diagram*

5.3 Comparison of analysis data before and after optimization

To compare and analyze the static characteristics of the titanium alloy frame before and after optimization, a static analysis was conducted again. According to static analysis, when the titanium alloy frame is in emergency turning condition, its deformation and stress are greater than the other three working conditions. Therefore, when conducting a static analysis on the optimized titanium alloy frame model under emergency turning conditions, the added boundary conditions and loads are the same as the above analysis. Maximal deformation occurs at the outer edge of the upper surface in the middle of the right longitudinal beam, with a maximum deformation of 7.4855mm, as shown in Fig. 13.



(a) Total deformation cloud map (b) Equivalent stress cloud map

***Fig. 13** Static analysis results under emergency turning conditions after titanium alloy frame optimization*

A large stress point was found to appear on the outer edge of the lower surface of the rear end of the left longitudinal beam. After calculation, the maximal stress at this point is 238.03 MPa, which is much lower than the tensile strength of the longitudinal beam of 930 MPa. This result indicates that during emergency turns, the designed tensile strength of the longitudinal beam can effectively withstand the stress generated by the turn. The stress point appears at the outer edge of the lower surface of the longitudinal beam, which is subjected to significant lateral force during turning. The results of the analysis are crucial for the safety assessment of the structure. By confirming that the maximal stress is less than the tensile strength of the longitudinal beam, it can be ensured that, under emergency turning conditions, the longitudinal beam can safely withstand stress without damage or failure. The comparison of the frame data before and after optimization is shown in Tab.2. The results showed that the optimized frame weight was reduced by 13.76%, the maximal stress was reduced by 5.19%, and the maximal deformation was reduced by 0.37%.

***Tab. 2** Comparison of data from the titanium alloy frame before and after optimization*

No.	Item	Before optimization	After optimization	Optimization amount	Optimization percentage
1	Maximal deformation	7.5123 mm	7.4855 mm	0.0277 mm	0.37%
2	Maximal equivalent stress	251.07 MPa	238.03 MPa	13.04 MPa	5.19%
3	Mass	197.3 Kg	170.15 Kg	27.15 Kg	13.76%

6 Conclusions

This article used a finite element method to conduct static and modal analysis on the titanium alloy frame, with the aim of comprehensively evaluating its static and dynamic characteristics. Through this series of analysis, the stress distribution and deformation of the frame under various working conditions are deeply understood, which provided solid data support for its subsequent optimization design. On the basis of the results of static and modal analysis, a lightweight design was carried out on the titanium alloy frame using the topology optimization method. Topology optimization, as an efficient method of optimizing material layout, redistributes materials within the frame to minimize material usage while maintaining sufficient strength. After the lightweight design, the performance of the titanium alloy frame has been significantly improved. Firstly, the maximal deformation was reduced to 7.4855mm, which is 0.0277mm less than the original design, and the optimization percentage reached 0.37%. Reducing maximal deformation is of great significance for improving the driving stability and riding comfort of vehicles. Second, the maximal stress was reduced to 238.03 MPa, which is 13.04 MPa less than the original design, and the optimization percentage was as high as 5.19 %. This significant stress reduction not only improved the durability and safety of the frame, but also helped to extend its service life. In terms of lightweight design, before optimization, the total weight of the titanium alloy frame was 197.3kg, after topology optimization, its weight was reduced to 170.15 kg, and the optimization percentage was as high as 13.76 %. This significant weight reduction is of great importance to improve fuel economy, reduce emissions, and improve the handling performance of the entire vehicle. Through the lightweight design, the maximal stress and deformation of the titanium alloy frame have been effectively reduced, while significantly reducing the weight of the frame. These improvements not only enhanced the performance of the frame, but also contributed to enhancing the overall performance of the vehicle.

The finite element analysis method and topology optimization theory used in this article provide effective technical means for the lightweight design of titanium alloy frames. This research not only provided a method for optimizing the design of titanium alloy frames, but also provided useful references for lightweight design of other similar structures.

Acknowledgement

This work was supported by the Vanadium and Titanium Resources Comprehensive Utilization Key Laboratory of Sichuan Province (2022FTSZ01) and Panzhihua Municipal Guiding Science and Technology Plan Project (2023ZD-G-1).

References

[1] YANG, R., ZHANG, W., LI, S., XU, M., HUANG, W., & QIN, Z., (2023). Finite element analysis and optimization of hydrogen fuel cell city bus body frame structure. *Applied Sciences*, 13(19): 10964. https://doi.org/10.3390/app131910964

[2] MA, W., LU, Y., WANG, P., WANG, Y., & WANG, J. (2023). Double optimization design of the formula racing car frame based on the variable density method and the joint variable method. *Applied Sciences*, 13(18): 10155. https://doi.org/10.3390/app131810155

[3] WANG, L., TANG, L., WU, P., & CHEN, J. (2022). Research on the optimization of automobile plastic front frame structure. *Advances in Materials Science and Engineering*, 2022: 2340342. https://doi.org/10.1155/2022/2340342

[4] CHEN, J., KWAK, Y., XU, M., KURNIAWAN, R., LI, C., & KO, T. J. (2021). Topology and modular size optimization of small electric vehicle frame based on cross-section contribution analysis. *Structural and Multidisciplinary Optimization*, 64: 4287-4304. https://doi.org/10.1007/s00158-021-03075-y

[5] LI, W., YANG, C., WANG, G.L. (2023). Finite element analysis and experimental study on the frame of automotive multifunctional obstacle breaking vehicle. *Machinery Design & Manufacture*, (9): 137-140. DOI: 10.19356/j.cnki.1001-3997.2023.09.001

[6] ZHANG, Z.B., SUN, Y.B., LIU, Q. (2022). Static analysis and optimization of a pure electric bus frame. *Machinery Design & Manufacture*, (11): 154-157+163. DOI: 10.19356/j.cnki.1001-3997.2022.11.010

[7] GONG, J.F., SHAN, D.S., LIU, H., LI, X.Y. (2022). Design and finite element analysis of parametric platform for trapezoidal frame of truck. *Machinery Design & Manufacture*, (7): 6-9. DOI: 10.19356/j.cnki.1001-3997.20211105.015.

[8] TIAN, K. (2020). Vibration performance analysis and optimization of a passenger vehicle front subframe. *Machine Design and Research*, 36(5): 180-183+188. DOI: 10.13952/j.cnki.jofmdr.2020.0215

[9] HE, W.B., YAO, H.Y., MA, J., LI, Y.H. (2020). Strength analysis and modal analysis of frame structure of pure electric passenger car. *Machinery Design & Manufacture*, (9): 235-238. DOI: 10.19356/j.cnki.1001-3997.2020.09.055

[10] JIANG, Z.H., GUO, H.Z., GUO, R.N., LIU, B.J. (2020). Optimization design of racing frame based on modal test verification. *Manufacturing Automation*, 42(11): 111-114+118.

[11] MI, C.J., LI, W.T., NI, Z.S., XIAO, X.W., YANG, Q.Q., XIAO, Y.P., WANG, M.L. (2020). Static and dynamic characteristics analysis and optimization of electric wheel dump truck frame. *Journal of Mechanical Strength*, 42(3): 639-647. DOI: 10.16579/j.issn.1001.9669.2020.03.019

[12] LIU, Y., JIANG, R.C., LI, X.F., SONG, F.J., LIU, D.W. (2020). Multi-objective optimization of the low-speed electric vehicle frame based on the surrogate model. *Journal of Machine Design*, 37(1): 105-109. DOI: 10.13841/j.cnki.jxsj.2020.01.019

[13] ZOU, Y., SHEN, B.S., ZHONG, B., YOU, Z., SUN, Q. (2023). Structural optimization of tractor frame based on user operating conditions. *Journal of Machine Design*, 40(9): 108-114. DOI: 10.13841/j.cnki.jxsj.2023.09.015

[14] GU, F.Q., SU, X.P., MIAO, X.D. (2022). Performance analysis and structural optimization design of a semi-trailer frame. *Journal of Chongqing University of Technology* (Natural Science), 36(7): 94-101.

[15] MENG, X., SUN, Y., HE, J., LI, W., & ZHOU, Z. (2023). Multi-objective lightweight optimization design of the aluminium alloy front subframe of a vehicle. *Metals*, 13(4): 705. https://doi.org/10.3390/met13040705

[16] TANG, P., YANG, R., NI, X., ZHAO, H., & XU, F. (2022). Research on a lightweight unmanned sightseeing vehicle frame based on multi-condition and multi-objective optimization. *Advances in Mechanical*

Engineering, 14(10): 16878132221131748. https://doi.org/10.1177/16878132221131748

[17] HE, R. (2022). Lightweight design of frame structures based on multidiscipline and multi-objective. *Noise and Vibration Control*, 42(2): 173-178.

[18] LIAO, Y., LI, F., LI, Z. (2020). Lightweight design of aluminum rear subframe in conceptual design stage. *Automotive Engineering*, 42(12): 1737-1743. DOI: 10.19562/j.chinasae.qcgc.2020.12.017

[19] JIANG, L.H., WU, Q.J. (2020). Performance analysis and lightweight design of a front subframe. *Journal of Mechanical Strength*, 42(6): 1503-1508.
DOI: 10.16579/j.issn.1001.9669.2020.06.033

[20] LI, S.H., ZHANG, B., FENG, G. Z. (2020). Lightweight research on flexible frame of heavy truck. *Machinery Design & Manufacture*, (10): 110-114. DOI: 10.19356/j.cnki.1001-3997.2020.10.025

[21] WU, Z.M., XU, L.H., GUO, Y. (2020). Design and lightweight improvement of small electric vehicle frame. *Modern Manufacturing Engineering*, (4): 77-82. DOI: 10.16731/j.cnki.1671-3133.2020.04.013

[22] YU, Y.Z., LI, W.L., LI, L. (2019). Research on lightweight design of the frame structure for micro electric vehicles. *Modern Manufacturing Engineering*, (1): 75-81. DOI: 10.16731/j.cnki.1671-3133.2019.01.014

[23] XU, Z.G., ZHU, Z.M., YU, C. G. (2018). Base on discrete variable three-stage of frame light weight design. *Machinery Design & Manufacture*, (5): 25-27+31. DOI: 10.19356/j.cnki.1001-3997.2018.05.008

[24] WANG, Q.C., ZHAO, J.N., YANG, F., MU, Z. (2017). Lightweight evaluation method and optimal design on heavy truck frame. *Forging & Stamping Technology*, 42(9): 174-181. DOI: 10.13330/j.issn.1000-3940.2017.09.032

[25] ZHOU, Q. H., LIU, L. Y., SHEN, M. B. (2017). Lightweight optimization design for sub-frame of front lifting dump truck. *Chinese Journal of Construction Machinery*, 15(5): 409-414. DOI: 10.15999/j.cnki.311926.2017.05.007

[26] YANG, S, LIAO, X, ZHI, G, LIU, G, ZOU, J. (2023). Finite Element Optimization Analysis of the Frame Structure of the New Steel Mold Trolley. *Manufacturing Technology*, 23(6): 1013-1019. DOI: 10.21062/mft.2023.112.

[27] WANG, T., WANG, L., WANG, C., & ZOU, X. (2018). Crashworthiness analysis and multi-objective optimization of a commercial vehicle frame: A mixed meta-modeling-based method. *Advances in Mechanical Engineering*, 10(5): 1687814018778480.
https://doi.org/10.1177/1687814018778480

[28] WANG, S., & WANG, D. (2021). Crashworthiness based multi objective integrated optimization of electric vehicle chassis frame. *Archives of Civil and Mechanical Engineering*, 21(3): 103. https://doi.org/10.1007/s43452-021-00242-2

[29] UHRÍČIK, M, PALČEK, P, CHALUPOVÁ, M, KUCHARIKOVÁ, L, PASTIEROVIČOVÁ, L, MEDVECKÁ, D, et al. (2023). Structural and Fractographic Analysis of Aluminum Alloy before and after Fatigue Loading. *Manufacturing Technology*, 23(5): 725-731. DOI: 10.21062/mft.2023.067

[30] TEICHMANOVA, A, MICHALCOVA, A, NECAS, D. (2022). Microstructure and Phase Composition of Thin Protective Layers of Titanium Aluminides Prepared by Self-propagating High-temperature Synthesis (SHS) for Ti-6Al-4V Alloy. *Manufacturing Technology*, 22(5): 605-609. DOI: 10.21062/mft.2022.069