International Journal of Mechatronics and Applied Mechanics
2025, Issue 20, Vol. 1

FINITE ELEMENT ANALYSIS AND OPTIMIZATION DESIGN OF THE FRAME OF HEAVY-DUTY TRUCKS

Shouxin Li[1,2], Xuexin Yang[1,2], Junjie Wang[1,2], Yu Wang[1,2], Chao Guo[1,2].*
[1]Hebei Institute of Mechanical and Electrical Technology， Xingtai 054000, Hebei, China
[2]New Energy Vehicle Lightweight Technology Research and Development Center of Xingtai，Xingtai 054000, Hebei, China

Abstract - The vehicle frame is an important assembly related to the normal driving and transportation of the vehicle. Performing a finite element analysis on the frame to ensure its mechanical properties such as static strength and dynamic performance is crucial for the frame design. Through the finite element analysis of the heavy-duty truck frame, the stress and deformation under four working conditions were analyzed respectively, ensuring that the force on the frame is within a reasonable range. Through the modal analysis of the frame, the structural damage caused by long-period resonance during the use of the frame was avoided. Finally, in order to further improve the quality of the frame, high-strength materials were replaced, and the finite element analysis was carried out again. Through the analysis, it was found that the designed frame basically meets the usage requirements and has a relatively high safety factor.

Keywords: Vehicle frame, Static analysis, Modal analysis, Structural optimization, Finite element.

1. Introduction

The performance of the vehicle frame plays a crucial role as an important component in ensuring the driving safety of automobiles. Finite element analysis of the vehicle frame is an efficient and cost-effective optimization design approach [1,2]. Numerous studies have indicated that during the finite element simulation process of the frame, various factors, including static strength, vibration, and optimization aspects of the frame, need to be taken into comprehensive consideration [3-5]. Conducting a static strength analysis of the frame under regular operating conditions can prevent the vehicle from experiencing structural damage resulting from impact loads caused by goods and different working conditions during driving, thereby guaranteeing the vehicle's driving safety [6, 7]. Modal analysis of the structure can help avoid structural damage to the frame due to resonance issues [8]. Moreover, optimizing the design of the frame can not only enhance its mechanical performance but also further improve its economic efficiency [9]. Through the finite element analysis of the designed frame, the structural safety of the vehicle can be further ensured.

2. Finite Element Modeling of the Vehicle Frame

2.1 Frame Model

The vehicle frame's dimensional structure is shown in Figure 1. It has a side - beam - type overall structure, consisting of a main frame and a sub - frame connected by high - strength bolts. In total, there are two longitudinal beams and thirteen cross beams. To be specific, the main frame has a material thickness of 8mm, the sub-frame has a thickness of 7mm, and the stiffener has a thickness of 8mm. The material thickness of the 1st to 3rd and 10th to 13th cross beams is 8mm, while that of the 4th to 9th cross beams is 6mm. The frame measures 11400mm in total length, 870mm in total width, and the main frame has a height of 320mm. The longitudinal beams are made of the commonly used low-carbon steel B550L, and the cross beams are made of low-carbon steel B510L. The mechanical performance parameters of the materials are shown in Table 1.

https://doi.org/10.17683/ijomam/issue20.4

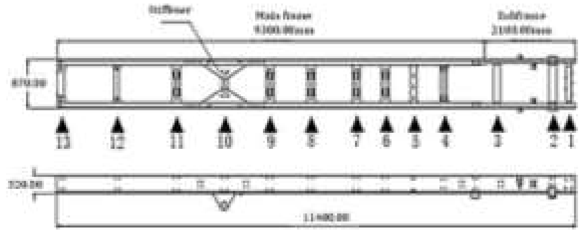

Figure 1: Dimensional Diagram of the Vehicle Frame Structure

Table 1: Mechanical Properties of Materials

Material Name	Density (kg/m^3)	Elastic Modulus (n/mm^2)	Poisson's Ratio	Yield Strength (MPa)
B550L	7.83e3	2.1e5	0.3	400
B510L	7.53e3	2.1e5	0.3	355

2.2 Three-Dimensional Modeling

Based on the provided dimensional model, a three-dimensional model of the vehicle frame is constructed by software SOLIDWORKS. In the modelling process, to carry out the finite element analysis of the vehicle frame effectively, the following simplifications are applied to the frame:

(1) Irrelevant chamfer and fillet structures are removed. To simplify the meshing procedure and enhance the mesh quality, certain chamfer and fillet structures that have minimal impact on the analysis are eliminated.

(2) There are numerous small process holes on the cross beams and longitudinal beams. These holes interfere with the meshing and are not beneficial for the vehicle frame's simulation analysis. Thus, they are removed to improve the mesh quality.

(3) Simplification of connection modes: Frame connection elements like bolts and rivets can increase the model's complexity. To simplify the model, these connection modes can be simplified into equivalent rigid connections to reduce the computational complexity.

The simplified three-dimensional model is depicted in Figure 2 below.

Figure 2: Three-Dimensional Model of the Vehicle Frame

2.3 Mesh Generation

The three-dimensional model is imported into the simulation analysis software ANSYS Workbench, and the tetrahedral mesh is generated for the vehicle frame through the automatic meshing approach. The mesh size is set to 20mm. Once the meshing process is completed, there are a total of 462134 nodes and 220243 elements. A partial mesh model of the meshed structure of the vehicle frame is presented in Figure 3 below.

Figure 3: Partial Mesh Structure Diagram of the Vehicle Frame

By constructing the finite element model of the vehicle frame, it facilitates subsequent finite element analysis.

3. Finite Element Strength Analysis of the Vehicle Frame

During the long-distance transportation of automobiles, due to complex driving environments, bumpy roads, and other factors, it is of crucial importance to ensure that the vehicle frame has a safe and reliable structure. To ensure the reliability of the vehicle frame under various working conditions and avoid local stress concentration that might lead to frame damage, fracture, or plastic deformation, static analysis of the frame structure is necessary during the frame design stage. The static analysis of the vehicle frame is typically conducted under four conditions, namely the bending condition, the torsional condition, the turning condition, and the braking condition.

The bending condition primarily simulates the scenario where the vehicle is traveling at a constant speed on a relatively flat road. In this scenario, all wheels are in contact with the ground, and the boundary conditions are relatively straightforward. Secondly, the torsional condition is more intricate compared to the bending condition. It simulates the vehicle traveling on a rough road with the left front wheel in a suspended state, and appropriate adjustments to the boundary conditions are required. The turning condition simulates the vehicle making a right turn, and in this case, a horizontal acceleration in the negative direction of the X-axis needs to be applied to the vehicle frame. Finally, the braking condition is simulated, which represents the situation where the vehicle performs an emergency stop on a straight road, and at this time, a horizontal acceleration in the negative direction of the Z-axis needs to be applied to the vehicle frame.

The main loads borne by the vehicle during transportation are presented in Table 2 below. And all the loads and conditions on the frame is shown in Figure 4.

Table 2: Mass Table of the Main Components of the Truck

NO.	Name	MASS/kg
1	The cab and the driver	700
2	The Engine, clutch and gearbox	900
3	The Storage battery, oil tank	350
4	The carriage	2000
5	The cargo	10000

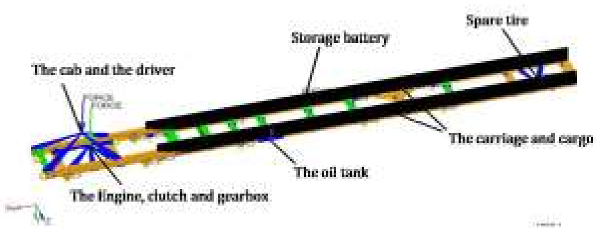

Figure 4: Loads and Conditions on the Frame

3.1 Bending Condition

Under the bending condition, the loads acting on the vehicle are applied to the vehicle frame in the form of uniformly distributed loads, while the translational degrees of freedom of the vehicle frame are constrained. Through finite element computation, the deformation distribution cloud diagram of the vehicle frame is obtained, as shown in Figure 5 below.

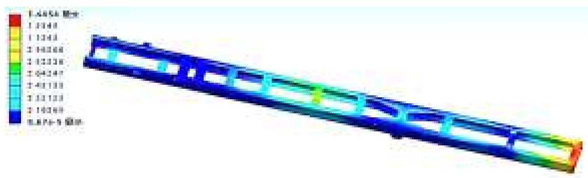

Figure 5: Stress Cloud Diagram under the Bending Condition

In Figure 5, the position where the maximum displacement occurs is at the rear of the vehicle frame, with the maximum displacement being 1.4454mm. Based on the analysis of the actual situation, the possible reason for this deformation is that the pressure exerted by the carriage and the cargo on the rear half of the vehicle frame is considerably greater than that from the cab, the driver, and the powertrain on the front half of the vehicle frame, thus resulting in a significant bend at the rear of the vehicle frame. According to the reference data, the displacement of the heavy-duty truck frame should not exceed 10mm[10]. The maximum displacement of this vehicle frame complies with the design requirements.

The stress distribution cloud diagram of the vehicle frame is obtained, as illustrated in Figure 6.

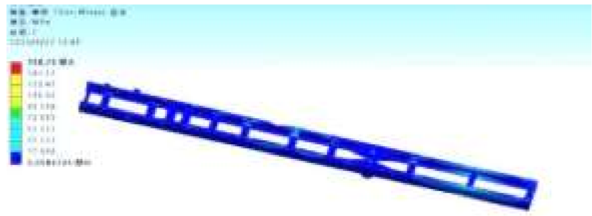

Figure 6: Stress Cloud Diagram of the Bending Condition

In Figure 6, the connection between the eighth crossbeam of the vehicle frame and its longitudinal beam is the location where the maximum stress occurs, with the maximum stress being 158.75MPa. Through analysis, it is considered that the weight of the goods at this location is the largest, thereby resulting in relatively large stress at the connection between the vehicle frame's longitudinal beam and the eighth crossbeam. However, the stress value is lower than the allowable stress of the material (400MPa), and thus it is within the safety range.

3.2 Torsion Condition

During the driving process of heavy-duty trucks, they frequently encounter relatively harsh road conditions, such as uneven road surfaces. In such circumstances, one of the vehicle's wheels may be suspended, which will alter the way in which the wheels support the vehicle frame, thereby resulting in a torsion phenomenon.

Under this working condition, the vehicle's load is identical to that in the bending condition, yet the constraint conditions vary. This working condition simulates the scenario where the left front wheel is suspended. The frame's constraint conditions are as follows: all degrees of freedom of shackles 1 and 3 of the frame are released, all translational degrees of freedom of shackles 2, 4, 5, 6, 7, and 8 are constrained, and simultaneously, the vertical degrees of freedom of shackles 9 to 16 are constrained, while other degrees of freedom are released. The displacement cloud diagram of the vehicle frame obtained through calculation is presented in Figure 7.

Figure 7: Displacement Cloud Diagram under the Torsion Condition

In Figure 7, the position where the maximum displacement occurs is on the left half of the sub-longitudinal beam of the vehicle frame, and the maximum displacement amounts to 3.6709mm. The reason for this is analyzed as below: the left front wheel is in a suspended state, and all constraints on its shackles are released, resulting in severe deformation of the front half of the sub-longitudinal beam due to the gravitational forces of the cab, powertrain, etc. However, it still complies with the design requirements.

The stress distribution cloud diagram of the vehicle frame is obtained, as shown in Figure 8.

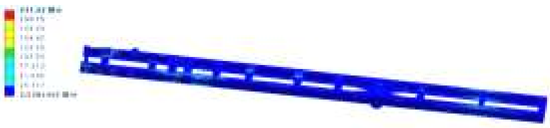

Figure 8: Stress Cloud Diagram under the Torsion Condition

The position of the maximum stress is at the connection between the second crossbeam of the vehicle frame and the right longitudinal beam, with the maximum stress being 231.92MPa. The possible reason, through analysis, might be the uneven force distribution on the sub-longitudinal beam caused by the left front wheel being suspended. It can be calculated that the safety factor under this working condition is 1.72, which conforms to the design requirements.

3.3 Turning Condition

This working condition involves a heavy-duty truck making a right turn. Under this condition, apart from the gravitational load, a centrifugal force load should also be applied to the vehicle frame. Generally, the speed of heavy-duty trucks when passing through curves is lower than 20km/h, and the turning radius is set at 12m, with the applied acceleration being 0.26g. The remaining loads are identical to those in the bending condition. At this time, the planar degrees of freedom in the X, Y, and Z directions of all shackles are constrained, while the rotational degrees of freedom in the X, Y, and Z directions are released. At this moment, the deformation distribution cloud diagram under the turning condition is obtained, as shown in Figure 9.

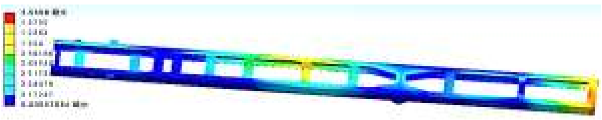

Figure 9: Displacement Cloud Diagram under the Turning Condition

In Figure 9, the position where the maximum displacement occurs is at the rear end of the vehicle frame, and the maximum displacement amounts to 1.5508mm. The most probable reason for this phenomenon is that the weight of the carriage and the cargo is considerably greater than the weight borne by the front end of the vehicle frame, resulting in significant deformation of the rear of the vehicle frame. Since the vehicle is turning, the right longitudinal beam of the vehicle frame bears a relatively large gravitational force and also undergoes a substantial displacement, yet it remains within the design requirements.

The stress distribution cloud diagram of the vehicle frame is depicted in Figure 10.

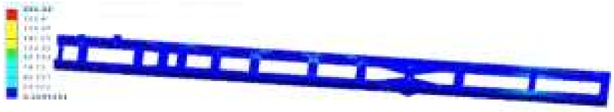

Figure 10: Stress Distribution Cloud Diagram under the Turning Condition

In Figure 10, the position where the maximum stress occurs does not change. However, due to the effect of centrifugal force, the maximum stress under this condition increases to 224.32MPa. After calculation, the safety factor under this working condition is 1.78, which still satisfies the design requirements.

3.4 Braking Condition

When a heavy-duty truck encounters an emergency situation, it may need to perform an emergency brake. In this scenario, the vehicle frame not only bears the gravitational loads generated by various components but also endures longitudinal inertial force loads due to the emergency braking. Here, the simulated situation is that the vehicle is on an asphalt road, with the braking acceleration set at $5.88m/s^2$, and the longitudinal inertial force for braking is applied simultaneously. The remaining loads are identical to those in the bending condition. All the planar degrees of freedom in the X, Y, and Z directions of the front wheel shackles are constrained, the rotational degrees of freedom in the X, Y, and Z directions are released, the vertical degree of freedom Y and the longitudinal degree of freedom Z of the rear wheel shackles are constrained, and other degrees of freedom are released. The deformation cloud diagram of the vehicle frame under this working condition is obtained, as shown in Figure 11.

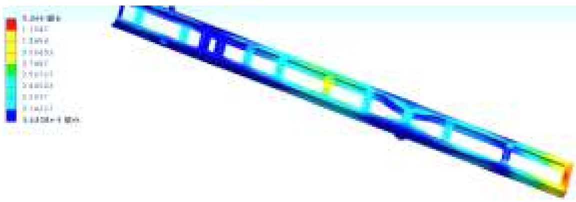

Figure 11: Displacement Cloud Diagram under the Braking Condition

In Figure 11, the position where the maximum displacement occurs is at the rear end of the vehicle frame, and the maximum displacement amounts to 1.344mm. Because the vehicle frame undergoes emergency braking, an inertial force in the negative Z-axis direction is generated, which causes the deformation at the rear of the vehicle frame to decrease to a certain extent compared with the turning condition. However, the deformation at the 7th crossbeam in the middle increases, but the overall force condition is relatively uniform and still complies with the design requirements.

The stress distribution cloud diagram of the vehicle frame is obtained, as shown in Figure 12.

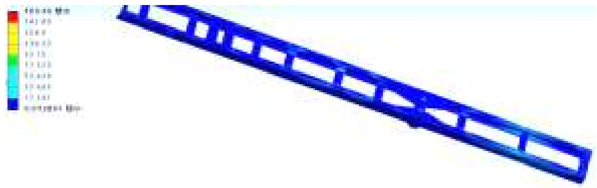

Figure 12: Stress Distribution Cloud Diagram under the Braking Condition

In Figure 12, the position where the maximum stress occurs is near the connection between the eighth and ninth crossbeams and the stiffeners and the longitudinal beams, and the maximum stress is 160.46MPa. The maximum stress occurs here because the cargo box and the cargo, under the combined action of gravity and longitudinal inertial force, cause the right rear suspension shackle to bear a greater load, resulting in the maximum stress at the stiffener bolt holes of the right rear suspension shackle. It can be calculated that the safety factor of the vehicle frame under the emergency braking condition is 2.49. From this, it can be inferred that the vehicle frame still meets the design requirements.

Through the above analysis, it is found that the force conditions of the vehicle frame under various typical working conditions all meet the strength requirements. Through comparison, it is found that the force condition of the vehicle frame is the worst under the torsional condition, but the safety factor reaches 1.72, which still complies with the design requirements. The safety factors of the other working conditions are all relatively high, indicating that the vehicle frame also has a favorable safety factor.

4. Modal Analysis of the Vehicle Frame

Modal analysis is an essential step in structural design [11]. In order to prevent damage to the vehicle frame caused by resonance, it is necessary to analyze the natural frequencies and vibration modes of the vehicle, which is also a critical aspect in the design and analysis of automotive products. Through modal analysis, the natural frequencies of the vehicle frame and their corresponding vibration modes are determined, and subsequently, the vehicle can be optimized and improved in terms of design to enhance its vibration characteristics and safety performance [12, 13].

The first 11 natural frequencies of the vehicle frame are obtained through modal analysis of the unloaded vehicle frame. Given that the first 3 natural frequencies are approximately 0, the statistics start from the fourth order, as presented in Table 3, and the vibration modes of each order are depicted in Figure 13.

Table 3: Modal Frequencies and Vibration Mode Descriptions of the Vehicle Frame

Order	Frequency (Hz)	Characteristic Description
4	2.729	Frame Vertical Bending
5	3.001	Frame Vertical Bending
6	14.93	Frame Vertical Bending
7	18.13	Frame Lateral Bending
8	21.259	Frame Torsional Bending
9	23.92	Frame Lateral Bending
10	32.121	Frame Torsional Bending
11	48.725	Frame Torsional Bending

The vibration patterns of the vehicle frame are presented in sequence in Figure 13.

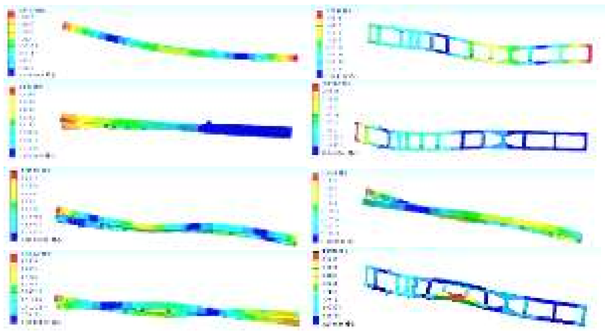

Figure 13: The 4th - 11th Order Vibration Pattern Diagrams of the Vehicle Frame

Analysis reveals that the vehicle's engine has an idle speed of 600r/min, and the idle frequency of the engine, calculated by a formula, is 10Hz. The rated speed of this engine is 2300r/min, and its vibration frequency is 38.3Hz.

Based on the modal analysis, although there is a certain difference between the natural frequencies of the vehicle frame and these two vibration frequencies, they are relatively close. It can thus be inferred that resonance may occur during the normal driving process of the truck. To avoid resonance, long-term idling of an empty vehicle should be avoided.

5. Optimization Design of the Vehicle Frame

From the frame strength analysis, it is known that the vehicle frame has a relatively high safety factor, which implies that the vehicle frame has ample safety margins. Meanwhile, since the natural frequency of the vehicle frame is relatively close to the vibration frequency of the engine, it is essential to optimize the vehicle frame structure.

When optimizing the vehicle frame structure, the frame strength should still fulfill the usage requirements. At the same time, in order to avoid resonance damage, it should be ensured that the strength of the frame material is considerably enhanced. High-strength steel is an indispensable material in the lightweight design of the vehicle frame, with parameters including a density of 7.85e3kg/m3, a yield strength of 500 MPa, an elastic modulus of 2.1e5 N/mm2, and a Poisson's ratio of 0.3. Therefore, in this optimization, both the vehicle frame and the stiffeners are replaced with high-strength steel, and two crossbeams are removed based on the relevant experience of topological research, and the positions of other crossbeams are readjusted [14-16]. The newly designed crossbeam structure is presented in Figure 14 below.

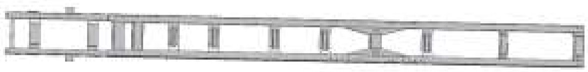

Figure 14: The Remodeled Vehicle Frame Model

After replacing the material of the vehicle frame, the weight of the vehicle frame has been reduced. The weight of the original vehicle frame was 1269 kg, whereas the weight of the redesigned vehicle frame is 1211 kg, achieving a weight reduction of 58 kg.

The finite element static analysis was carried out again on the vehicle frame, and the comparison of the strength and deformation of the vehicle frame under various working conditions with that of the original vehicle frame is presented in Table 4 below.

Table 4: Comparison Table of Stress and Displacement of the Optimized Vehicle Frame

Items	Original Vehicle Frame	New Vehicle Frame	Parameter Change
Mass(kg)	1269	1211	Reduction by4.6%
Maximum Stress under Bending Condition (MPa)	158.75	161.12	Increase by 1.5%
Maximum Stress under Torsion Condition (MPa)	231.92	245.92	Increase by 6%
Maximum Stress under Turning Condition (MPa)	224.32	239.36	Increase by 6.7%
Maximum Stress under Braking Condition (MPa)	160.46	162.52	Increase by 1.3%

From the above analysis, it can be inferred that the strength of the vehicle frame still fulfills the usage requirements after replacing the new material. Although the maximum value of the observed quantity does not change considerably, the overall mass of the vehicle frame decreases significantly, thereby achieving a 6% weight reduction design for the vehicle frame.

In addition, modal analysis was conducted again on the vehicle frame, given that the first 4 natural frequencies are approximately 0， the 4th-14th natural frequencies of the vehicle frame obtained are presented in Table 5.

Table 5: Modal Frequencies and Vibration Mode Descriptions of the Optimized Vehicle Frame

Order	Frequency (Hz)	Characteristic Description
4	2.681	Frame Vertical Bending
5	3.38	Frame Vertical Bending
6	14.665	Frame Vertical Bending
7	15.982	Frame Lateral Bending
8	23.192	Frame Torsional Bending
9	24.084	Frame Lateral Bending
10	28.993	Frame Torsional Bending
11	33.72	Frame Torsional Bending
12	43.1	Frame Torsional Bending
13	43.902	Frame Lateral Bending
14	55.888	Frame Vertical Bending

From the analysis, it can be inferred that the natural frequency of the vehicle frame is considerably different from the engine excitation frequency, and in comparison, with the natural frequency of the original vehicle frame structure, resonance problems can still be avoided. Since the material used for the new vehicle frame is high-strength steel, the safety margin has been enhanced.

6. Conclusions

Simulation analyses of the vehicle frame under four typical operating conditions were conducted, and thereby the distribution patterns of the frame's stress and displacement were derived. The results indicate that the locations where stress concentration occurs are predominantly concentrated in the first half of the vehicle frame's sub-longitudinal beams, and the strength complies with the design requirements.

Modal analysis was performed on the vehicle frame, and the natural frequencies and vibration modes of the 4th to 14th order modes of the vehicle frame were extracted. The results reveal that the modal frequency of the vehicle frame is relatively close to the engine's excitation frequency, and resonance problems may arise. However, the optimization results show that the natural frequency of the frame and the engine rotation frequency are far different, and resonance will not occur, so the optimization results are effective.

During the vehicle frame optimization process, high-strength steel was chosen to replace the original frame material, and the layout of the frame crossbeams was reconfigured. Specific optimization measures involve removing the 5th and 6th crossbeams and adjusting the positions of the 7th and 8th crossbeams. After optimization, a 4.6% weight reduction of the vehicle frame was accomplished. Subsequently, finite element analysis was carried out on the new vehicle frame again. The results demonstrate that the static strength indicators fulfill the usage requirements, and the modal analysis results essentially prevent the occurrence of resonance.

References

[1] Dongchen Qin, Yuheng Lin, Tingting Wang, et al. (2021). Finite Element Analysis and Lightweight Design of Sub-frame of Hybrid Electric Sanitation Vehicle Based on HyperWorks. Journal of Chongqing Institute of Technology, 35(10):9-15. https://DOI:10.3969/j.issn.1674-8425(z).2021.10.002.

[2] Yu-hong Zeng, Wei-ping Ding, Ming-liang Yang, et al. (2023). Tradeoff Between Accuracy and Efficiency of Subframe Constrained Modal Simulation. Machinery Design & Manufacture, 383(1):34-39, 44. https://DOI:10.3969/j.issn.1001-3997.2023.01.009.

[3] Shui-guang Tong, Jia-zhi Miao, Zhe-ming Tong, et al. (2019). Finite element analysis and optimization for static and dynamic characteristics of diesel forklift frame. Journal of Zhejiang University (Engineering Science), 53(9):1637-1646. https://DOI:10.3785/j.issn.1008-973X.2019.09.001.

[4] Li C, Kim I Y, Jeswiet J. (2015). Conceptual and detailed design of an automotive engine cradle by using topology, shape, and size optimization. Structural and Multidisciplinary Optimization, 51(2):547-564. https://DOI:10.1007/s00158-014-1151-6.

[5] Chengqiang Liu, Lianhua Lin, Haigang X. U., et al. (2017). Finite Element Analysis and Process Optimization Design of Automobile Frame. Hot Working Technology, 46(23):199-200, 204. https://DOI:10.14158/j.cnki.1001-3814.2017.23.054.

[6] Wei L. I., Chen Yang, Gui-lu Wang. (2023). Finite Element Analysis and Experimental Study on the Frame of Automotive Multifunctional Obstacle Breaking Vehicle. Machinery Design & Manufacture, 391(9):137-140. https://DOI:10.3969/j.issn.1001-3997.2023.09.030.

[7] Ye Zou, Baoshan Shen, Bing Zhong, et al. (2023). Structural optimization of tractor frame based on user operating conditions. Journal of Machine Design, 40(9):108-114. https:// DOI:10.13841/j.cnki.jxsj.2023.09.015

[8] Jinlong L. I., Xiang Gao, Yi Feng, et al. (2022). The Fatigue life analysis and optimization of subframe based on road load spectrum. Modern Manufacturing Engineering, (5):50-55. https://DOI:10.16731/j.cnki.1671-3133.2022.05.008.

[9] Xiang-chao Meng, You-ping Sun, Jiang-mei H. E., et al. (2023). Conceptual design of topology optimization for aluminum alloy front subframe of a vehicle. Manufacturing Automation, 45(10):71-75. https://DOI:10.16731/j.cnki.1671-3133.2022.05.008.

[10] Weixin Liu. Automobile Design [M]. Tsinghua University Press: Beijing, 2021.

[11] Wu Chenyu, Zhong Zipeng, Quan Shuanglu, et al (2024). Free Modal Analysis and Test Based on FSAE Frame. AUTOMOBILE APPLIED TECHNOLOGY, 49(17): 95-98. https:// DOI:10.16638/j.cnki.1671-7988.2024.017.019.

[12] Kai-jia W. U., Xiao-ping S. U., Qiang Wang. (2020). Structure Analysis and Optimization Design of Construction Machinery Frame. Machinery Design & Manufacture, (7):51-55. https://DOI:10.1016/j.engstruct.2011.07.028.

[13] Haoyao RAO, Yinhui A. O., Haochuan Zhang, et al. (2024). Structural optimization and stiffness analysis of truss frame. Journal of Machine Design, 2024,41(7): 28-35, https:// DOI:10.13841/j.cnki.jxsj.2024.07.006

[14] Su Yonglei, Zhang Zhifei. (2023). Correction Method of Static Stiffness and Multi-level Topology Optimization for Subframe. Automotive Engineering, 45(11):2157-2164. https:// DOI:10.19562/j.chinasae.qcgc.2023.11.017.

[15] Fan W, Yin L, Xu Z, et al. (2023). Two-stage surrogate-based optimization of a vehicle composite subframe with modified fast flexible space-filling designs. Mechanics of Advanced Materials and Structures,30(4): 724-737. https://doi.org/10.1080/15376494.2021.2023916

[16] Meng X, Sun Y, He J, et al. (2023). Multi-objective lightweight optimization design of the aluminum alloy front subframe of a vehicle. Metals, 13(4): 705. https://doi.org/10.3390/met13040705.