Journal of Engineering Research and Reports

Volume 24, Issue 10, Page 13-22, 2023; Article no.JERR.98034
ISSN: 2582-2926

Finite Element Analysis and Optimization of Medium Truck Frame Based on ANSYS

Jing Yinghao [a*]

[a] *North China University of Water Resources and Electric Power, Zhengzhou, Henan Province, China.*

Author's contribution

The sole author designed, analysed, interpreted and prepared the manuscript.

Article Information

DOI: 10.9734/JERR/2023/v24i10844

Open Peer Review History:
This journal follows the Advanced Open Peer Review policy. Identity of the Reviewers, Editor(s) and additional Reviewers, peer review comments, different versions of the manuscript, comments of the editors, etc are available here: https://www.sdiarticle5.com/review-history/98034

Short Research Article

Received: 21/01/2023
Accepted: 25/03/2023
Published: 29/03/2023

ABSTRACT

Frame is one of the most important parts of a car chassis. As the most basic part of a car, the research of frame plays a very important role in the safety of the vehicle. The research content of this topic is the statics and dynamics of a certain type of truck are analyzed, and CATIA is used for modeling. By building a model of the frame and analyzing it under two working conditions with ANSYS, the stress change diagram of the frame under two different working conditions is obtained. The first 10 free modes of the frame are calculated by modal analysis. The frequency and vibration pattern of the frame in each mode are obtained, and then the possibility of the cab forming resonance with the road surface is analyzed on this basis. On this basis, the size of the frame is optimized, and the easy fracture area is strengthened, and a feasible optimization scheme is proposed.

Keywords: Frame; finite element; structure; mode; light weight; statics.

**Corresponding author: Email: 595533634@qq.com;*

1. INTRODUCTION

The gap in automobile design technology at home and abroad cannot be ignored. Although the automobile industry will be protected by the World Trade Organization for vulnerable industries in a short time, domestic enterprises must form their own competitive advantages instead of only having a weak cost and price advantage [1]. Must catch up with and exceed the current developed country level on the performance of engineering machinery, which is the important branch of automobile industry. Industrial structure and technological innovation mechanism become the main factors that restrict the development of engineering vehicles in our country, and as the body of engineering vehicles, It not only carries the main parts of the whole paver, but also installs the chassis, engine, generator and diesel tank on it. Moreover, the main general Chengdu in the axle is carried on the frame, and the force under different working conditions is to be borne in the driving process. Therefore, for the reliability of the frame, scholars have carried out a lot of research. In the reference, the modal frequency is taken as the optimization objective to improve the vibration isolation ability of the frame. Literature [2] provides the basis for optimization design by comparing the strength analysis of the frame under two typical working conditions. In literature [3], parameterized modeling was carried out on the frame of medium bus. The static and dynamic characteristics of the built passenger frame model are analyzed. Under the condition that the strength and stiffness meet the requirements, the structure size of the frame is optimized. It not only realizes the lightweight of the frame, but also improves the vibration characteristics of the frame and improves the fuel economy of the bus.

2. FRAME 3D MODELING

2.1 Simplification before Modeling

In this paper, CATIA software is used to model the frame running of a medium truck of Dongfeng Motor. In order to facilitate grid division, the structure needs to be properly simplified before modeling, so as to ensure the efficiency of calculation. For model simplification, there are the following criteria;

1) Retain the main parts of the workshop and delete those that have little impact on the structure.
2) Holes and burrs with small diameters and chamfering are ignored to prevent non-convergence of results.
3) The joints of all structural parts, whether welded or riveted, should be regarded as a whole, and the influence of force on welding seams and process holes should not be considered. In order to ensure smooth surface, some chamferes and rounded corners are done at right angles.

2.2 Three-dimensional Model Establishment

The simplified processing of the three-dimensional model will bring some negative effects. For example, the removal of a part of the accessory structure will affect the strength and stiffness of the frame structure during the calculation and analysis, which will make the stress analysis results of the frame larger. But this will objectively ensure the strength and stiffness of the frame structure, conservative but guaranteed quality. It is safe and effective to adjust and strengthen the structure according to the analysis result of this treatment. According to the above requirements, the frame model was first drawn on CATIA 3D drawing software and the necessary simplification was done, as shown in Fig. 1 below.

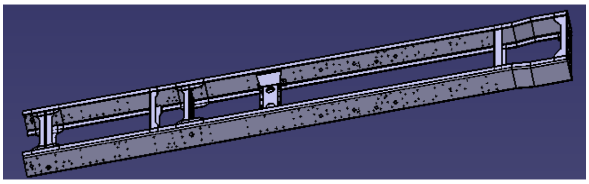


Fig. 1. Simplified 3D model diagram

Dimensions are marked in CAD as shown in Fig. 2.

2.3 Grid Division and Imposition of Constraints and Loads

The finite element method is a numerical method to discretize the continuum into a set of finite size elements to solve the continuous mechanics problem. With the popularization of large general finite element analysis software, the theory and application of finite element method have been developed rapidly. The core ideas of finite element analysis include structure discretization, element analysis and integral assembly.

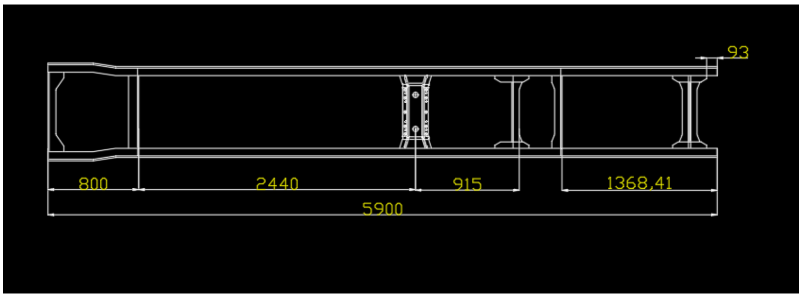


Fig. 2. Dimensional sketch

As a comprehensive finite element analysis software, ANSYS has its own pre-processing function to meet the needs. Its own powerful grid processing ability and the ability to interface with a variety of software greatly reduce the early work of engineering analysts. The geometric model built in CATIA was imported into ANSYS Workbench for grid division. It should be noted that when dividing cells, attention must be paid to avoid irregular shapes. Triangles with too large or too small angles will affect the calculation results. Regular tetrahedral mesh should pay attention to the aspect ratio to avoid excessive flat or thin height, which will cause matrix pathology in the analysis.

Because the simplified model of Dongfeng medium truck frame studied is still complex, solid grid division is selected here, and tetrahedral grid division is adopted. The information of the drawn finite element grid model of the frame is as follows: the number of nodes is 149,572, and the number of elements is 71,270. The model is shown in Fig. 3.

The frame is a kind of side beam frame, with two longitudinal beams on both sides and several beams in the middle. The material of the frame studied in this paper is Q345 structural steel, and its related mechanical properties are shown in Table 1 [4-5].

Before the calculation of the frame model, reasonable and complete constraint conditions must be established to ensure that there is no rigid displacement of the frame, but not excessive constraints, otherwise the calculation results will be distorted. The imported finite element model needs to be positioned in the coordinate system to facilitate the addition of subsequent constraints. In this paper, the forward direction of the frame is parallel to the X-axis, the Z-axis is parallel to the plane of symmetry of the frame, the Y-axis is parallel to the direction of gravity, and the three axes are perpendicular to each other. Make the connection between the frame and the rear axle connect with the ground, fix all parts have completed the overall positioning of the frame, and all constraints are set to fixed connection.

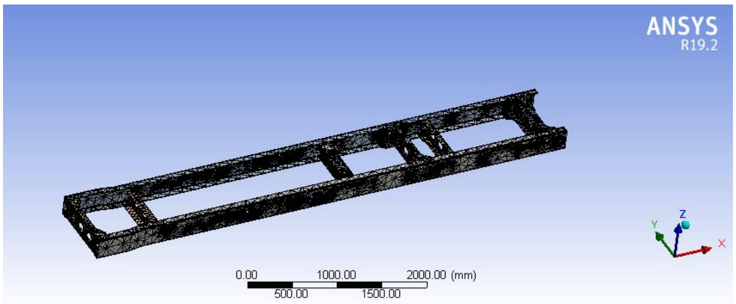


Fig. 3. Mesh model

Table 1. Material properties

Yield strength /MPa	Strength of extension / MPa	Elasticity modulus 10^{11}Pa	Density/kg.m^3	Poisson's ratioμ
345	450	2.1	7800	0.3

3. STATIC ANALYIZE OF THEFRAME

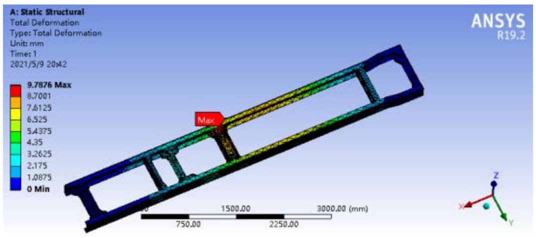
The static analysis of the frame structure is a kind of analysis method based on different working conditions. Static strength is the most basic requirement for the normal operation of frame structure. Mechanical parts under normal working condition are not allowed to appear structural fracture or plastic deformation, which will affect normal running; Surface damage is not allowed. Strength is the ability of a component to resist such failure. Therefore, this paper carries on statics analysis through several typical working conditions.

Working condition 1: constant speed driving. In this condition, the truck belongs to the pure bending condition. The analysis of this condition is mainly to investigate the ability of the frame to resist bending deformation. This condition is one of the most important conditions for testing the stiffness of the frame. The front and rear supports were fixed and restrained, and 35000N (7 tons at full load) concentrated stress was applied to the middle beam of the frame, in a vertical downward direction. The nephogram of stress and deformation of the frame is shown in Fig. 4.

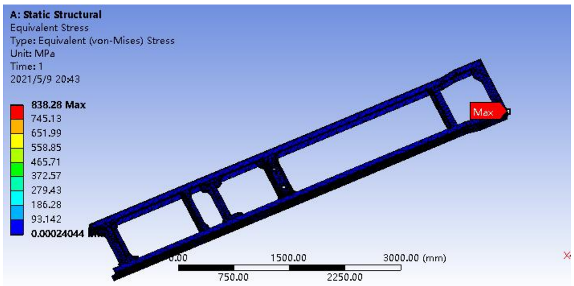
By analyzing the calculation results, the maximum stress appears on the rear leaf spring support of the frame. The maximum stress value is 838.28Mpa, which can be ignored in the corner stress concentration position. The maximum stress of the whole structure is 186.28 Mpa which is less than the yield limit of the material 235 Mpa. According to the structural deformation cloud map of the frame, the maximum deformation of the whole frame is 9.78mm, which appears on the third beam of the frame. This is the most dangerous part of the frame. Frequent unloading and loading will cause rupture in this area [6-9].

Condition two: torsion condition, this condition is mainly to consider the frame under torsion load, resistance to deformation. This condition is caused by the truck driving on the uneven road surface. When one wheel is not the same height as the other wheel, there is a certain torsional deformation. This condition is an important condition used to measure the torsional stiffness of the frame.

The loading method is to apply 35000N concentrated force in opposite directions at the front end of the left and right longitudinal beams respectively. At the same time, the freedom of rotation of the Y axis of the rear plate spring, the translational freedom of the position of the rear support in the x and y directions and the rotational freedom around the x and z axes are released. The calculation results are shown in Fig. 5.

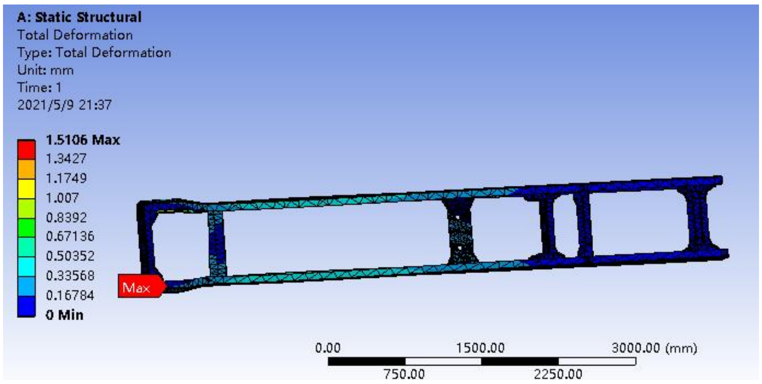


- Frame deformation diagram under full load pure bending condition

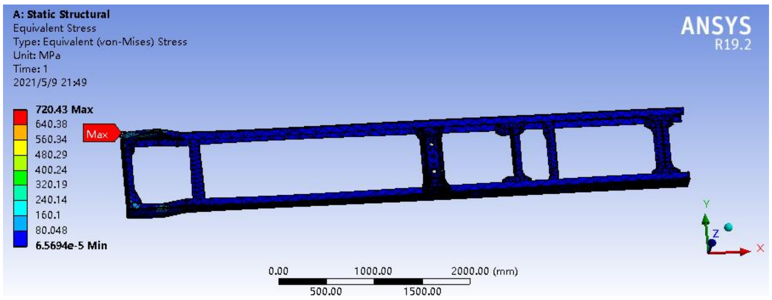


- Frame stress diagram under full load pure bending condition

Fig. 4. Condition I static finite element analysis



- Frame deformation diagram under full load pure bending condition



Frame stress diagram under full load pure bending condition

Fig. 5. Condition Ⅱ static finite element analysis

By analyzing the calculation results, it can be observed that the maximum deformation of the frame is concentrated in the front of the left and right beam, and the maximum deformation is 15.1mm. It can be seen from the finite element stress nephogram that the maximum stress of the frame is still concentrated in the front of the frame under torsion condition. Ignoring the stress singularity, the maximum stress is 161.4MPa. Well below the limit.

Through the statics analysis of the medium frame, it can be seen that the most easily broken part of the whole frame appears on the third beam, which can be thickened in the optimization process to enhance its bending resistance. It can be seen from the stress nephogram that the strength and stiffness of the side beam are good, and there is a certain optimization space, which can be properly lightweight [10].

4. MODAL ANALYSIS

One of the important fields of the application of finite element method is the modal analysis of the frame structure in the development of new platform models. The modal analysis of the frame structure is the main content of the structural analysis in the process of new product development. Particularly important is the low order elastic mode of the frame, which is not only the key means to control the conventional vibration of the vehicle, but also reflects the overall stiffness performance of the vehicle. Practice has proved that the modal analysis of the frame structure using finite element method can fully understand the structural stiffness and natural vibration mode of the frame in the initial stage of design, so as to avoid the design defects related to the frame to the maximum extent. The optimal scheme is adopted to ensure that the frame structure not only has enough static stiffness to meet the requirements of assembly and use, but also has reasonable dynamic characteristics to control general vibration and noise. The application of finite element analysis makes it possible to verify the design scheme in the initial stage of product development and judge whether the scheme can meet the requirements of the design product, thus greatly shortening the development and test cycle, saving the test cost, and improving the reliability of the product.

At present, in terms of the technical development of engineering projects, modal analysis based on finite element calculation has been applied to various engineering departments. The use of finite element for structural modal analysis has the following advantages.

1) In the development of new products, it can help engineers and technicians to carry out effective analysis of dynamic characteristics and optimize the original design scheme on this basis.

2) It can evaluate the dynamic characteristics of the existing structure to a certain extent. Through modal analysis, the parameters of each mode can be calculated in the software, and these parameters can evaluate whether the structure meets the requirements of the system.

3) It can help designers identify the load of structural system to some extent.

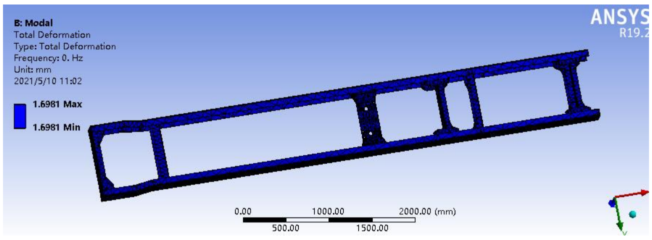
For the linear structure of N degrees of freedom, the differential equation of motion is expressed as follows [11]:

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

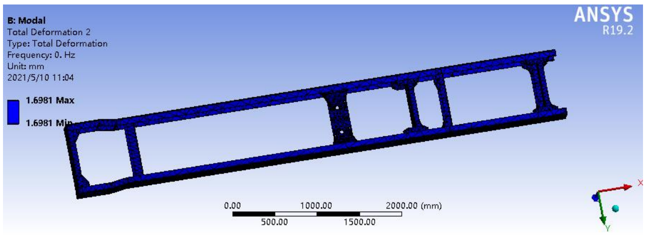
Where

[M] -- quality of the system;
[C] -- damping of the system;
[K] -- stiffness matrix of the system;
{x} -- displacement response vector of each point in the system;
{F} - excitation force vector.

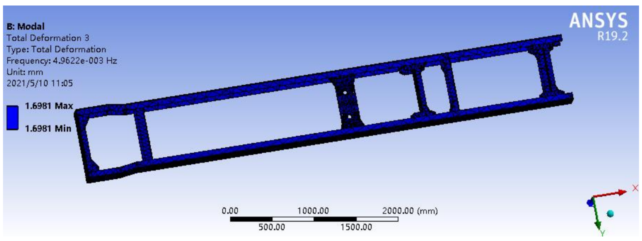
This equation (1) is a set of coupled equations. When the system has a lot of degrees of freedom, solving such equations is difficult. The algorithm of modal analysis is to change the coupled equations into uncoupled independent equations. The frame mode analyzed in this paper is free mode, that is, the mode without any constraint. The simplified CATIA model is imported into ANSYS dynamic module for calculation. The first ten free modes of this model frame are obtained through analysis, namely natural frequency and vibration mode, and then the output results are checked in ANSYS. The modes of each order are shown in Fig. 6 below [12].



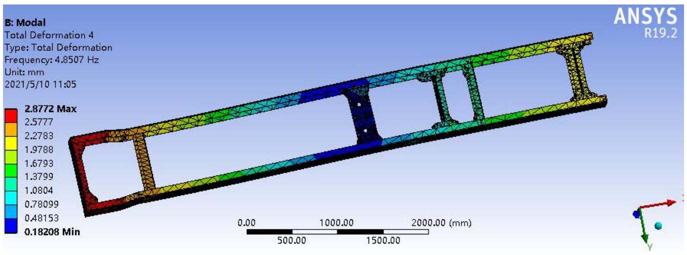
First order natural frequency of frame structure



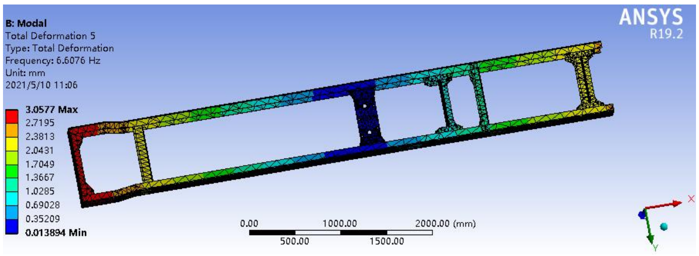
Second order natural frequency of frame structure



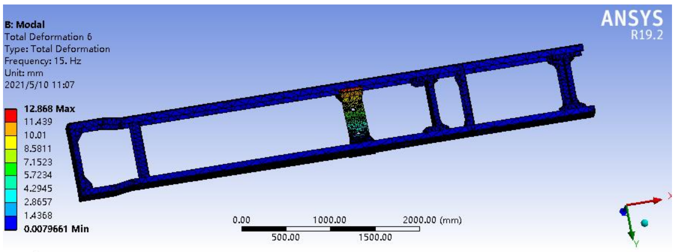
Frame structure of 3 natural frequencies



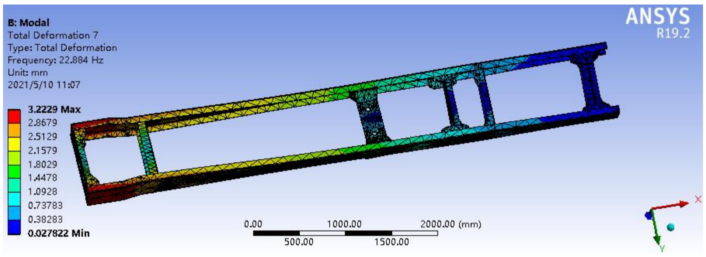
Frame structure 4 natural frequencies



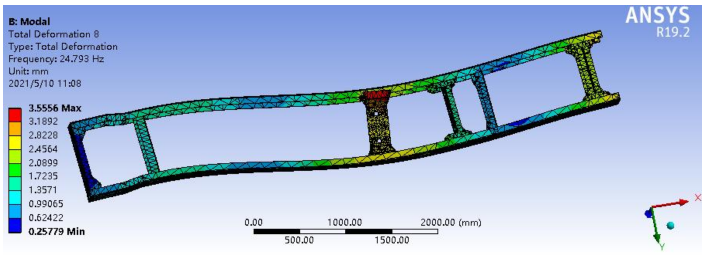
Frame structure 5 natural frequencies



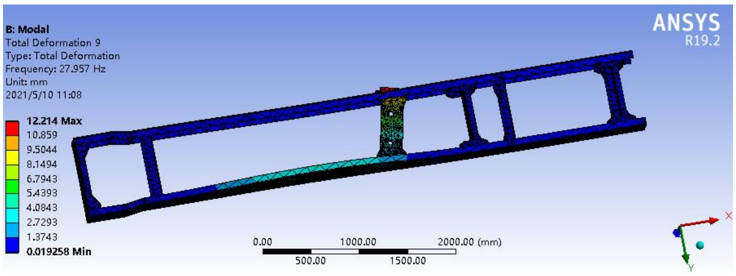
Frame structure 6 natural frequencies



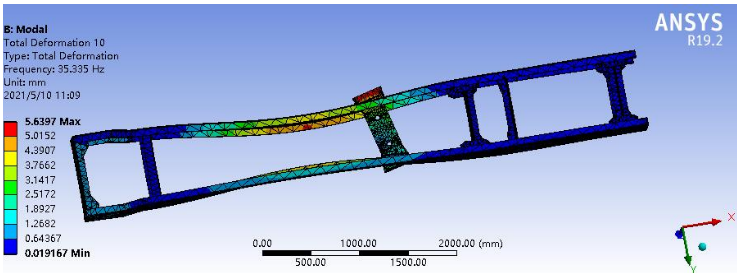
Frame structure 7 natural frequencies



Frame structure 8 natural frequencies



Frame structure 9 natural frequencies



Frame structure 10 natural frequencies

Fig. 6. First 10 order modal pattern of the frame

Where, the natural frequency of each modal frame is, the first order: 0; Second order: 0; Third order: 4.96e-003; Level 4:4.85; Fifth order: 6.60; Sixth order: 15; The seventh stage: 22.88; Rank 8:24.79; The ninth order; 27.96; Step 10:35.34.

In general, the vibration excitation of truck in the process of running is composed of road bump excitation, engine operation process excitation, wheel dynamic balance excitation and drive shaft dynamic balance excitation. Bumpy excitation of road surface is related to the smoothness of road surface. The highway, general national highway and provincial highway pavement in our country, the road surface excitation frequency range is generally 1 to 3Hz, the influence of low frequency vibration is more significant [13]; The excitation frequency caused by the unbalance of wheel rotation is generally no higher than 9Hz, and the further improvement of the manufacturing quality and detection level of tires and rims, and the constraint of the tolerance range of the rim and shaft head stop, this excitation occupies a smaller and smaller component, which can be avoided. The torsional vibration and vertical vibration caused by engine operation are above 10±1.67Hz and 20±1.67Hz respectively (the truck engine is a 4-cylinder engine, idle speed 650r/min), where the excitation component is relatively large [14]. In terms of driving the drive shaft, the speed of the drive shaft is generally within the range of 40 ~ 70km/h in low speed or urban road conditions, and within 90 ~ 110km/h on the highway. The frequency of vibration caused by the unbalance of the drive shaft at the speed of 40km/h is about 26Hz, and the frequency of the unbalance excitation of the drive shaft corresponding to the higher speed is higher. So the incentive is relatively small. It can be seen from the modal analysis results that the low-order modal frequency value of the frame is about 20Hz, which has relatively little influence on the cab, resulting in a very small possibility of resonance between the cab and the road excitation [15-20].

5. CONCLUSION

The static analysis and calculation of two working conditions of the frame are respectively torsion working condition and bending working condition. The main conclusions are as follows: 1. It can be seen from the finite element stress nephogram that the maximum stress of the frame is still concentrated in the front of the frame under torsion condition. Ignoring the stress singularity, the maximum stress is 161.4MPa. Well below the limit. The greatest deformation of the frame is concentrated in the front of the left and right beams. Under bending conditions, the maximum deformation occurs on the third beam of the frame. Which means the third beam is the most vulnerable part of the frame to breakage. 2. This paper proposes that the thickness of the longitudinal beam on both sides can be reduced appropriately under the condition of ensuring the strength. At the same time strengthen the thickness of the third beam. Rough estimate can lose more than 10% weight. In this way, the strength of the whole frame can be guaranteed, but also the weight of the frame can be reduced, and the fuel economy of the truck can be improved to some extent. 3. Through the calculation results of the first ten modes, the low-order modal frequency is found out and compared with the usual natural frequency of the cab, and the possibility of the cab forming resonance with the road excitation is analyzed. The analysis results are used to optimize the actual design of the frame structure.

The next research direction is to further study the finite element software, especially the meshing and dynamic analysis, to achieve more accurate simulation

COMPETING INTERESTS

Author has declared that no competing interests exist.

REFERENCES

1. Su Xiaoping, Zhu Jian, Xu Lian. Simulation study on structural dynamics of agricultural vehicle frame [J]. Agricultural Mechanization Research. 2010;(5): 36.
2. Yu Xian, Lin Huashan. Strength analysis of solar clean vehicle frame with different welding spot simulation forms [J]. Machinery Design & Manufacture. 2015;(11):1.
3. Sun Houhuan, Liu Chuang, Wang Hongnan. Frontal collision simulation and structure improvement of medium bus [J]. Journal of Nanjing University of Technology (Natural Science Edition). 2012;34(6):126.
4. Saeed Moaveni. Finite element analysis - Theory and application with ANSYS, second edition[D]. Beijing: Publishing House of Electronics Industry. 2005;8.
5. Zhou Chuanyue, Teng Wanxiu, Zhang Juntang. Engineering finite element and optimization analysis application example course [M]. Beijing: Science Press Press; 2005.
6. Xia Zongbao. Numerical simulation analysis and optimization design of a light electric truck frame [D]. Yangzhou University; 2020.
7. Wang Weiwei, Zheng Zaixiang, Xia Zongbao, Fang Jianyu. Finite element analysis and optimization of a light electric truck frame [J]. Southern Agricultural Machinery. 2021;52(01):23-25.
8. Zhou Mengmeng. Finite element simulation and optimization design of light truck frame [D]. Jiangsu University; 2020.
9. Han Yang. Finite element analysis and optimization design of light truck frame [D]. Dalian Jiaotong University; 2019.
10. Yu Dali. Study on structure analysis and lightweight of pure electric van frame [D]. Kunming University of Science and Technology; 2019.
11. Ma Xiyong. Finite element analysis and optimization design of TLD110 wide-body dump truck frame [D]. Chang 'an University; 2018.
12. Wang Lanthanum. Finite element analysis and optimization of YC1040 truck chassis [D]. Hubei University of Technology; 2017.
13. Zhou Wen, Wang Xiaorui, Xu Wenchao, Wang Yuchen. Finite element analysis of heavy truck frame [J]. Agricultural Equipment and Vehicle Engineering; 2017;55(01):65-68.
14. Zhao Yunan, Si Jingping, Wang Ermao, Wan Fangjun. Optimization design of frame structure of mining dump truck based on ANSYS [J]. Coal Mine Machinery. 2014;35(03):18-21.
15. Cao Lei. Frame design and finite element analysis of a pure electric bus [D]. Master Thesis of Soochow University; 2019
16. Wei Qing. Structure design and analysis of QD135 bridge crane with low clearance [D]. Master Thesis of Harbin Institute of Technology; 2019.
17. Zhou Xingliang. Finite element analysis and topology optimization of electric tour bus body frame [D]. Master Thesis of Wuhan University of Technology; 2008.
18. Zhang Daqian. Modeling and structural analysis of a bus body [D]. Master Thesis of Northeastern University; 2006.

19. Zhang Hongwei. Finite element analysis of a bus body structure. Master Thesis, Dalian University of Technology; 2005.

20. Frederic Dieu. Structural optimization of a vehicle using finite element techniques, SAE885135.225-226.

Peer-review history:
The peer review history for this paper can be accessed here:
https://www.sdiarticle5.com/review-history/98034