Viser Technology Pte. Ltd.
Mechanical Engineering Science
DOI: 10.33142/mes.v7i1.16750

Research Article Open Access

Lightweight Design of a Certain Type of Dump Truck Frame

Chuanqi LI, Jian ZOU [*], Yongjie LI, Yingwen REN, Xinxing YU
Yingkou Institute of Technology, Yingkou, Liaoning, 115014, China

[*]***Corresponding Author:*** *Jian ZOU, E-mail: zoujianqw@163.com*

Abstract
With the rapid growth of China's economic strength, the demand and market share are also constantly increasing. The number of cars is rapidly increasing, and a large amount of fuel is consumed as a result. The massive release of vehicle exhaust seriously damages the natural environment, and the environmental crisis is becoming increasingly serious. This article follows the principles of improving fuel efficiency, reducing emissions, and enhancing vehicle performance. Using NX 12.0 software, a three-dimensional model of a certain type of dump truck frame is constructed based on actual parameters. ANSYS Workbench is used to simplify the geometric model, mesh division, and material definition, and a finite element model is constructed. Obtain the structural performance and natural vibration characteristics of the original chassis under four typical working conditions: bending, torsion, lifting, and unloading, through static analysis and modal analysis. On this basis, the dimensions of the components that bear less load on the original frame were optimized, and the topology of the second crossbeam and rear end corner of the subframe that bear less load on the original frame was optimized to obtain a new frame. The new frame of the dump truck underwent secondary static analysis and modal analysis, and it was found that the weight of the new frame decreased by 41.03 kg, successfully reducing the weight of the frame by 4.38%, improving the vehicle's handling and stability, and extending its service life.
Keywords: *Dump truck frame; Lightweight design; static analysis*

1 Introduction

With the rapid development of the automotive industry and the continuous improvement of China's transportation system, the transportation industry in China has also shown a clear trend of development. Compared with ordinary trucks, dump trucks not only can load goods, but also have the advantage of mechanized cargo loading and unloading, effectively reducing the time spent on cargo loading and unloading, saving the labor force of workers, and improving worker efficiency. dump trucks have gradually become an important tool for cargo transportation.

With the increasingly strict environmental regulations, the market's demand for efficient and environmentally friendly transportation vehicles has increased, and enterprises need to constantly innovate to meet these needs. The exploration of lightweight optimization of vehicle frames often involves various aspects such as analysis and optimization of the frame structure. Through experimental simulation methods such as static analysis and modal analysis, the optimal optimization scheme is selected to improve the frame stiffness and durability, ensuring the safety and reliability of the vehicle under various working conditions.

The lightweight design of dump truck frames not only improves the performance and economy of vehicles, helps enterprises improve their technological level, meet new market requirements, enhance their market adaptability and innovation capabilities, but also has important significance for environmental protection, technological progress, and industry competition.

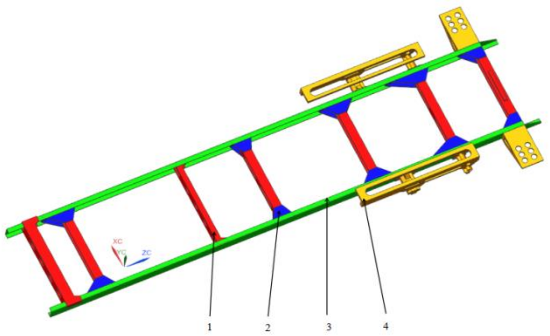
2 Creation of Finite Element Model for Vehicle Frame

2.1 Creation of 3D model of vehicle frame

This article selects a certain model of dump truck frame, which is mainly composed of two parts: the main frame and the sub frame. The main frame is usually fixed above the wheels by the suspension device, front axle, and rear axle support, playing a role in supporting and connecting various components of the vehicle. It not only plays a key role in maintaining the relative positional accuracy of all components, but also bears the impact of various loads from internal and external sources. Therefore, the frame must have sufficient strength and rigidity to withstand the heavy weight of the vehicle and the impact force transmitted through the wheels. As shown in Figure 1, the main frame mainly consists of seven crossbeams and two longitudinal beams, with the longitudinal beams being the main load-bearing parts. The crossbeam is mainly fastened to the corresponding

Received on February 7th, 2025; Accepted on March 4th, 2025

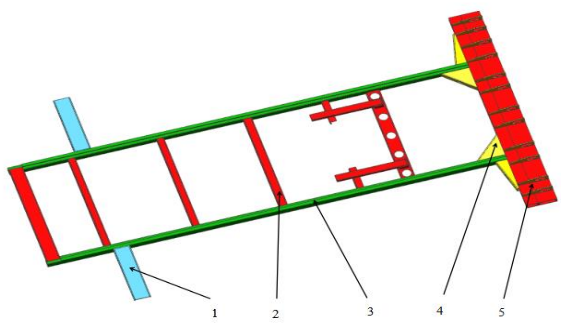
position of the longitudinal beam through fixed seats at both ends, helping the longitudinal beam to share various loads and increase the torsional strength of the frame. In addition, there are carriage lifting mechanisms installed at both ends of the main frame, which are connected to the subframe. When the lifting mechanism of the carriage rises or falls, it will drive the subframe to rise or fall, in order to achieve the purpose of self dumping of the vehicle.



1-Beam; 2-Fixed seat; 3-Longitudinal beam; 4-Car lifting mechanism

Figure 1 Simplified Model of Main Frame

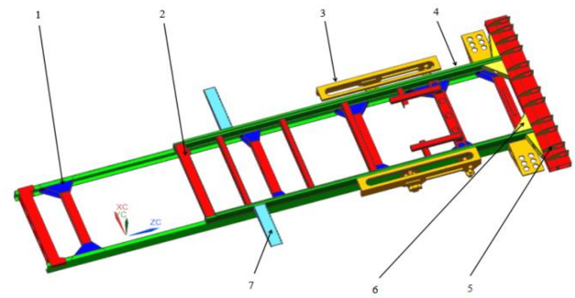
The subframe is mainly composed of two longitudinal beams and six transverse beams. The connection method between the transverse beams and the longitudinal beams is the same as that of the main frame, and they are fixed to the corresponding positions of the longitudinal beams through fixed seats, as shown in Figure 2. Unlike the main frame, the longitudinal beam and crossbeam at the rear end of the subframe, as well as the fastening of the left and right structures, use corner crossbeam components to make the rear end structure more tightly connected. The rear crossbeam of the car is welded with six sets of connecting buckles and suspended supports at both ends, which are used to connect other hanging structures of the car.



1-Support; 2-Beam; 3-Longitudinal Beam; 4-Corner Crossbeam Component; 5-Connecting Ring Buckle

Figure 2 Simplified model of subframe

When conducting static and modal analysis on the frame model of this type of dump truck, the difference between the constructed 3D model and the model used for finite element analysis is a key factor that causes the analysis results to be inconsistent with the actual situation. Therefore, this article uses NX 12.0 3D drawing software to construct the 3D model of the dump truck frame, following the principle of reasonable simplification, that is, to accurately establish the 3D model of the parts that have a greater impact on finite element analysis as close as possible to the actual situation, ignoring small details such as small holes or groove structures; Discard components that do not bear the force on the frame or have a small impact on the frame load; Neglecting unnecessary rounded corners and small chamfers can reduce the number of elements and nodes in the finite element model, improve the calculation speed of the model, reduce the difficulty of finite element analysis of the frame, and reduce the computation time for obtaining results. The simplified overall structural model of the dump truck frame is shown in Figure 3.



1-Fixed Seat; 2-Beam; 3 -Lifting Structure; 4-Longitudinal Beam; 5-Connection; 6-Corner Crossbeam Component; 7-support

Figure 3 Frame model of chassis

2.2 Grid division and material assignment

In the finite element analysis of this article, considering the size and structural details of the frame, a shell element size of 8 mm is assigned to the frame. For complex parts such as the contact between the crossbeam and longitudinal beam, taking into account various detailed features such as the edge position of the frame and the contact points of each component, the unit size for the above positions is set to 4 mm. In addition, in order to ensure the accuracy and computational efficiency of the finite element analysis results, this paper omitted the lifting device of the frame when meshing the frame, and only retained the main frame and sub frame. Based on this, the various components of the frame were meshed, and the corresponding connection relationships between each component were established, constructing a mesh meshing model for the frame. According to the above parameters, a total of 23345 grid cells and 59406 nodes were generated in this article's grid division.

The material of the selected dump truck frame in this article is B610L, which is a low-alloy steel known for its high strength and good weldability. A series of physical properties are shown in Table 1.

Table 1 Material parameter table

Material	Elastic modulus(MPa)	Poisson's ratio	Density(t/mm^3)	Yield limit(MPa)
B610L	2.1×105	0.3	7.89×10-9	500

3 Static Analysis of a Aertain Aype of Aump Aruck Arame

In the selection of load data for the original chassis, the total weight of the cab assembly is 450 kg, with a gravity of 4410 N; The powertrain integrated with engine, clutch, etc. has a mass of 660 kg and a measured gravity of 6468 N; When the mass of the cargo container is combined with the maximum load specified in its design, the accumulated mass obtained is 9530 kg, and the gravity is 93394 N. Among them, in order to make the calculation results more realistic, ensure its functional capability and operational safety, the gravity acceleration value is selected as 9.8 N/kg during gravity calculation.

3.1 Bending condition

3.1.1 Load and constraints

The gravity of the cab assembly and the gravity exerted by the powertrain cab assembly and powertrain are applied as concentrated loads at their respective centers of mass. On the contrary, the influence of gravity from the cargo box assembly and the goods it carries on the integrity of the frame structure is more dispersed, and is loaded onto the left and right longitudinal beams of the frame in a uniformly distributed load form. By adopting a card based approach, the gravity of the chassis itself was included in the analysis, which improved the accuracy of the simulation.

As the frame is connected to the axle through a suspension assembly located on the side of the main frame longitudinal beam, its translational motion in the X, Y, and Z directions can be restricted by designated nodes along the symmetrical centerline. Therefore, displacement constraints are applied at the six pairs of axles on the main frame, and loads are applied at the center of mass of the corresponding positions of the cab, power cab assembly, and powertrain. Apply constraints to the original frame model as shown in Figure 4.

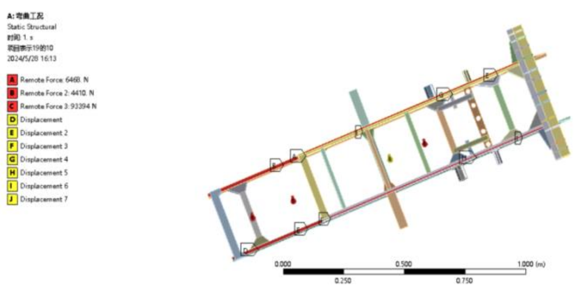


Figure 4 Original frame load and constraints under bending conditions

3.1.2 Result analysis

As shown in Figure 5, after observing and studying the stress distribution of the original frame under bending conditions, it is evident that the stress is mainly concentrated in the middle suspension position of the frame and at the connections with various suspension components. The maximum recorded stress value is 229.1 MPa, while the stress values in other important areas are mainly 203.6 MPa. Considering the smooth operation of dump trucks under these conditions, the probability of variability, transient properties, and load uncertainty is relatively small, therefore the safety factor is set to 2. The allowable stress value of the original frame under this specific condition is determined to be 250.0 MPa. Since the maximum stress value is lower than the allowable stress value, it can be concluded that the original frame meets the loading requirements of the dump truck, thus proving its sufficient strength.

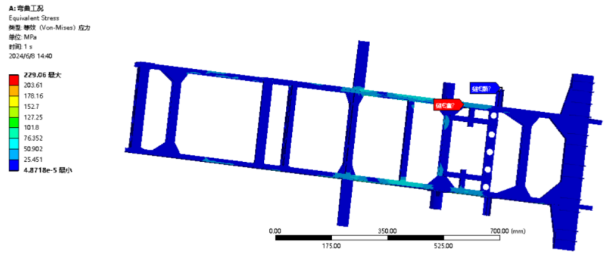


Figure 5 Stress distribution diagram of the original chassis under bending conditions

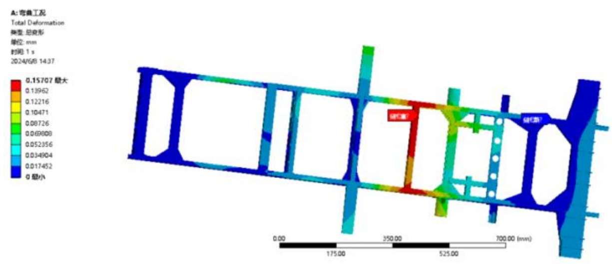


Figure 6 Stress distribution diagram of the original chassis under bending conditions

By observing and studying the dimensional deformation diagram of the original frame under bending conditions, it can be seen that the area with large deformation is located in the middle and surrounding areas of the subframe, with the greatest impact on the fourth crossbeam of the subframe, with a displacement of approximately 0.1571 mm. The rest of the frame has basically no deformation, so the frame stiffness is good.

3.2 Twisting condition

Twisting condition refers to the situation where a fully loaded dump truck travels through rugged and complex road environments at a certain speed. In this case, the wheels of the dump truck may encounter protruding obstacles or concave pits, causing at least one of the wheels to be suspended in mid air. The occurrence of this event will result in uneven force distribution on the original frame when affected, leading to a certain degree of twisting of the vehicle. When the vehicle is fully loaded with goods, the main load on the original frame comes from the weight of the goods, and the overall center of gravity is mainly located at the rear of the vehicle. The suspension of the rear wheels of the dump truck has a significant impact on the frame. Therefore, this section selects the case of left rear wheel suspension for the strength analysis of the frame.

3.2.1 Load and constraints

As shown in Figure 6, the left rear wheel is not constrained, and maintaining the same constraints as the bending condition at other positions can ensure the consistency of the simulation, thus enabling a more comprehensive evaluation of the performance of the frame under different operating conditions. According to the above description, displacement constraints are applied to the first four pairs of axles and the last two pairs of right axles.

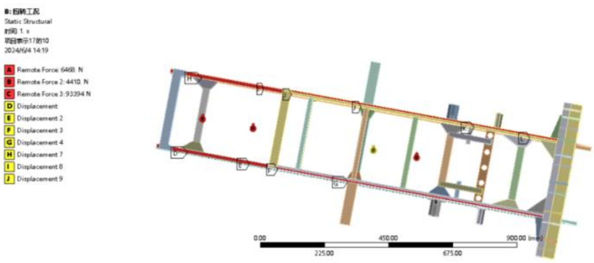


Figure 6 Load and constraints of the original frame under torsional conditions

3.2.2 Result analysis

As shown in Figure 7, after observing and studying the stress distribution of the original frame under torsional conditions, it can be seen that when the frame undergoes torsion, the sixth crossbeam of the main frame and its surrounding area, as well as the left side carriage support part and its surrounding area of the subframe, are greatly affected, with a maximum stress of 332.1 MPa. Considering the high variability, transience, and uncertainty of the load on the dump truck frame under this operating condition, in order to provide higher safety assurance, the safety factor is set to 1.5. Therefore, the allowable stress value of the original frame under this specific condition is determined to be 333.3 MPa. Since the maximum stress value obtained from the calculation is lower than the allowable stress value, it can be concluded that the original frame meets the torsional load requirements of the dump truck, thus proving its sufficient strength.

As shown in Figure 8, through the analysis of the deformation trend chart of the original frame under torsional conditions, it can be seen that when the left rear wheel of the dump truck is suspended, the deformation of the left rear part of the frame is the largest, with a deformation value of about 2.16 mm. This deformation gradually decreases towards the periphery, while the deformation of other parts is relatively small. Overall, under torsional conditions, the overall torsional stiffness of the chassis is relatively high, which means that when the model dump truck is fully loaded and driving on rough roads, the load-bearing capacity of the chassis is relatively weak.

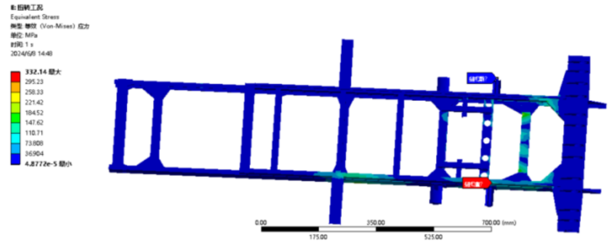


Figure 7 Stress distribution diagram of the original frame under torsional working condition

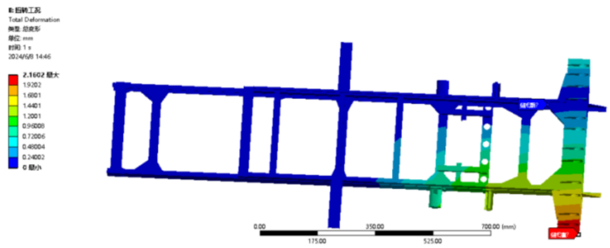


Figure 8 Deformation diagram of the original frame size under torsional condition

3.3 Lifting condition

During the lifting process of the dump truck, significant changes occur in the dynamics and forces acting on the frame. At the beginning of the lifting phase, load conditions different from those in the static or motion state of the vehicle were introduced. When the cargo box starts to lift, its weight and the force related to the lifting action are transmitted to the frame through specific connections, and the position and type of these connections determine how the lifting force is distributed on the frame. Similarly, a tilting mechanism that allows the container to rotate and tilt is also connected to the frame and container. By understanding the precise connection points and methods between the lifting device, tilting mechanism, and frame, necessary loads and constraints can be accurately applied to the frame model. This section selects the initial state of the frame during lifting for research.

3.3.1 Load and Constraints

When conducting structural analysis on the frame of this model of dump truck, loads that simulate real-world conditions are usually applied. In this case, the cab assembly consisting of the cab and related structures, the powertrain consisting of the engine and other power transmission components, and the assembly of the cargo box and its payload are considered as concentrated forces. These forces are applied at specific points corresponding to the center of mass of these components within the frame model. By using a card based method, the gravity of the frame itself was included in the analysis, and gravity acceleration was applied in the negative Y-axis direction of the frame model, which improved the accuracy of the simulation.When the cargo box is lifted, specific nodes representing the interaction with the ground on the frame model need to be constrained to prevent the frame from translating in different directions.

Thus, displacement constraints are applied at the center of mass of the six pairs of axles on the main frame, the cab, and the powertrain cab assembly, as shown in Figure 9.

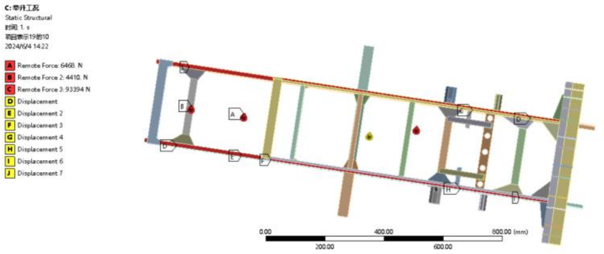


Figure 9 Load and constraints of the original frame under lifting conditions

3.3.2 Result analysis

As shown in Figure 10, after observing and studying the stress distribution map of the original chassis under lifting conditions, it can be found that during lifting, the structural integrity of the dump truck chassis is subjected to the highest stress in specific areas, and the maximum stress is mainly concentrated in the middle part of the chassis and the area leading to the rear, with a maximum stress value of 229.1 MPa. Considering the high variability, transience, and uncertainty of the load on the dump truck under this operating condition, in order to provide higher safety assurance, the safety factor is set to 1.5. Therefore, the allowable stress value of the original frame under this specific condition is determined to be 333.3 MPa. The maximum stress value obtained from finite element analysis is lower than the allowable stress value calculated theoretically. Therefore, it can be concluded that the original frame meets the requirements of this type of dump truck under lifting conditions, thus proving that the frame strength is sufficient.

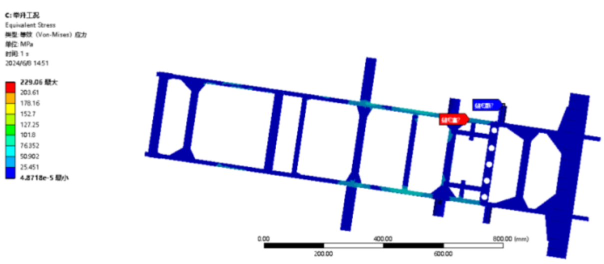


Figure 10 Stress distribution diagram of the original frame under lifting conditions

During the lifting process, the deformation of the dump truck frame is not uniform throughout its entire length. The middle part of the frame usually bears the load transmitted from the cargo box through the lifting mechanism, and its degree of deformation is also the highest. The maximum deformation occurring at the fourth crossbeam of the subframe indicates that this area is a critical point in the structure, with a maximum deformation of approximately 0.1571 mm. This may be due to the concentration of forces or the relative weakness of the crossbeam compared to other components in the frame.

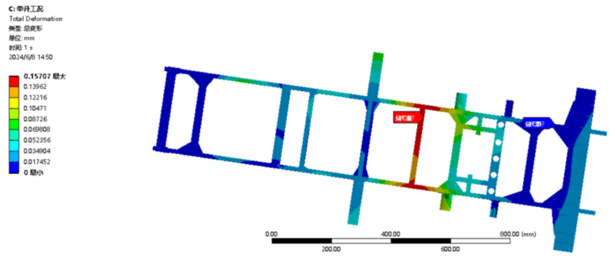


Figure 11 Deformation diagram of the original frame size under lifting conditions

3.4 Unloading condition

Unloading condition refers to the situation where, when the cargo reaches a critical state where it is about to slide, the weight of the cargo exerts a lever effect at the connection point between the frame, lifting device, and tilting mechanism due to the movement of the cargo's center of mass, applying a unique load to the frame. The specific angle for unloading the static angle of the dump truck frame of this model is 45 degrees., The typical situation where goods are ready to slide out of the cargo box. Therefore, this section will simulate the real-life scenario of the dumping process based on the above theoretical basis.

3.4.1 Load and Constraints

The cab assembly, powertrain, cargo box, and their payload assembly are considered as concentrated forces. These forces are applied at specific points corresponding to the center of mass of these components within the frame model. By using a card based method, the gravity of the frame itself is included in the analysis, and gravity acceleration is applied in the negative Y-axis direction of the frame model.When in the unloading condition, the dump truck should be stationary on the road surface, with the same constraints as the lifting condition. Displacement constraints should be applied at the six pairs of axles on the main frame, the center of mass of the cab, and the powertrain assembly, as shown in Figure 12.

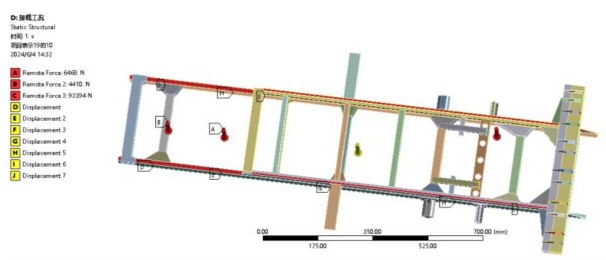


Figure 12 Load and constraints of the original chassis under unloading conditions

3.4.2 Result analysis

As shown in Figure 13,according to the analysis of the stress contour diagram of the original frame under the unloading condition, when the dump truck is in the unloading condition, the stress is mainly concentrated in the middle of the frame and some areas later, and the maximum stress value is 269.0 MPa. Considering the variability, transience and uncertainty of the load on the dump truck under this working condition, in order to provide higher safety guarantee, the value of the safety factor is 1.5, so that the allowable stress value of the original frame under this specific condition is determined to be 333.3 MPa. The maximum stress value obtained by finite element analysis is lower than the allowable stress value calculated theoretically, so it can be concluded that the original frame meets the requirements of this type of dump truck under unloading conditions, so as to prove that the frame strength is sufficient.

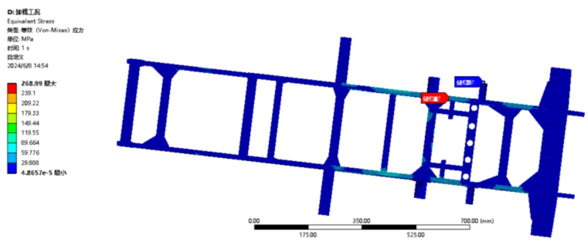


Figure 13 Stress distribution diagram of the original frame under unloading conditions

In the unloading condition, the container is only connected to the lifting device and the tilt seat, and the center of mass of the load is shifted backwards. As shown in Figure 13,through the observation and study of the frame deformation diagram under this working condition, it can be found that the main deformation area of the frame is concentrated in the fourth cross beam and later parts of the subframe, and the maximum size deformation is about 0.1524 mm, and the deformation extends backward and gradually decreases, so it can be analyzed that the stiffness of the frame is appropriate.

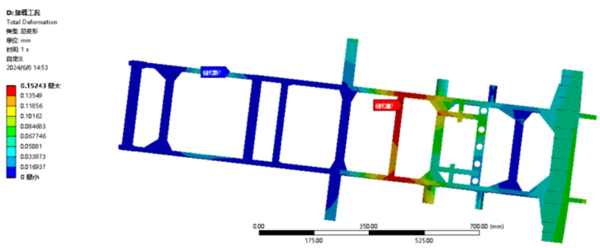


Figure 14 Deformation diagram of the original frame size under unloading conditions

As shown in Table 2, according to the actual working conditions of a certain type of dump truck, four typical working conditions of bending, torsion, lifting and unloading were selected, and the original frame of the dump truck was analyzed by finite elements. The bending and torsional conditions test the frame's ability to withstand stresses and handle complex road conditions, while the lifting and unloading conditions simulate the direct forces encountered during loading and unloading operations. The results of the analysis show that the strength and stiffness of the frame meet the standard regardless of the working conditions. It is important to note that the maximum stress under torsional effect is close to the allowable stress limit. If this continues for an extended period of time, the frame may operate within the range of material fatigue or structural failure, creating a safety hazard.

Table 2 Analysis results of the frame under different operating conditions

Operating conditions	Allowable stress (MPa)	Frame maximum stress (MPa)	Safety	Strength check
Curved	250.0	229.1	2	Qualified
Torsion	333.3	332.1	1.5	Qualified
Lifting	333.3	229.1	1.5	Qualified
Unload	333.3	269.0	1.5	Qualified

4 Optimized Design of a Certain Type of Dump Truck Frame

4.1 Frame topology optimization design

After completing the stiffness evaluation of the frame under four working conditions, it is found that the front end of the frame, that is, the cab assembly, the powertrain range is less stressed and the deformation is not obvious, so the second cross member of the subframe and the two parts of the rear side end angle in this area are selected for topology optimization design.

4.1.1 The second crosshead is optimized

From the stress analysis and deformation analysis of the above four working conditions, it can be seen that the stress and deformation of the second cross beam of the subframe are the largest under the lifting condition, and the stress value is 25.0 MPa, so the deformation and stress simulation results of the second cross beam of the subframe in this working condition are input into the Workbench structural optimization module, the two ends of the subframe are applied fixed constraints, the upper end is stressed, the optimization target is set to retain 60% of the front axle mass, and the restriction area and the optimization area are shown in Figure 15.

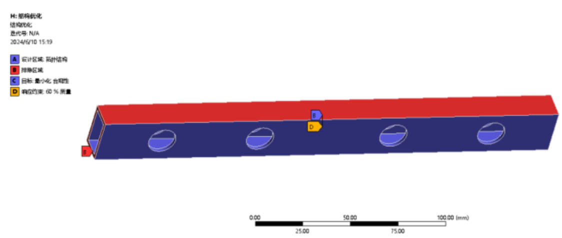


Figure 15 Second beam structural constraints

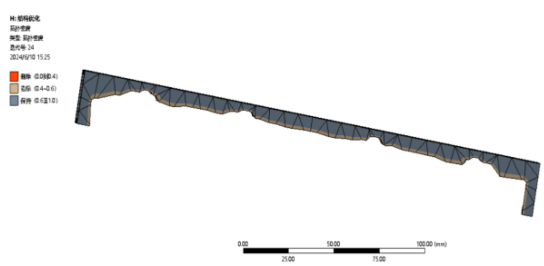


Figure 16 Optimization results of the second crossbeam

The second cross member of the subframe is placed under the defined load and constraint conditions, and then the topology optimization results of the structure in a specific scenario are calculated, as shown in Figure 16.According to the topology optimization result of the second cross beam of the subframe, the structural design of the original component is carried out again, as shown in Figure 17.

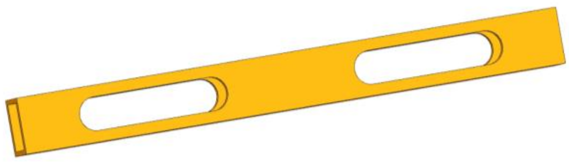


Figure 17 The structure of the second beam is optimized

4.1.2 Optimized rear end angle

Therefore, the simulation results of the rear end angle in this case are input into the Workbench structural optimization module, and a fixed constraint is applied to one end and a stress is applied to the upper end, and the optimization goal is set to retain 60% of the front axle mass, and the restricted area and optimization area are shown in Figure 18.

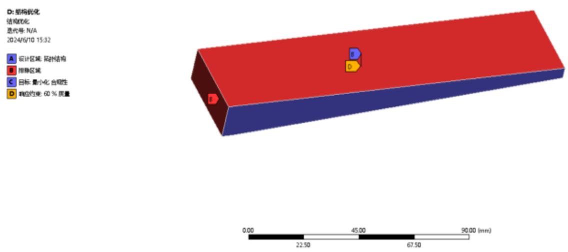
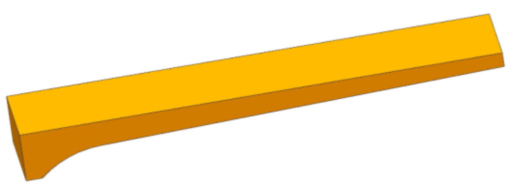


Figure 18 Rear end angle structure constraints

The virtual model of the rear end angle is placed under the above-defined loads and constraints, and then the topology optimization results of the structure in a specific scenario are solved, as shown in Figure 19.According to the results of the topology optimization of the rear end angle, the structural design of the original component is re-designed, as shown in Figure 20.



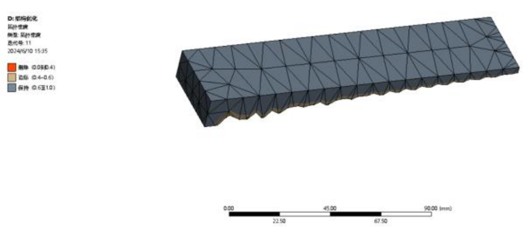


Figure 19 Result of the optimization of the rear side end angle

4.2 Optimised frame size

Through the query of literature and data, it can be obtained that the thickness of the plate of the frame of this type of dump truck produced by the manufacturer selected in this paper is 4 mm~12 mm. Combined with the above theory and analysis results, some parts of the main frame and subframe were selected for size optimization design, and the specific parameter changes are shown in Table 3.

Table 3 Size parameter optimization table

Optimize parts	Initial value(mm)	Optimize the value(mm)
Main frame rails	7	6
Main frame beam	6	4
Subframe rail	5	4.5
Subframe beam	6	4
Liners	6.5	5
Connecting plates	8	6

After the optimized design of the frame, it was found that the mass of the new frame was 895.77 kg, which was 41.03 kg less than the 936.80 kg of the original frame, a decrease of about 4.38%, which indicated that the improvement applied to the frame structure had successfully realized the lightweight design and did not affect the integrity of the frame structure of the dump truck.

4.3 Optimized static analysis of the rear frame

The stress distribution of the improved frame is viewed according to the above dimensional optimization and component optimization, and the results are shown in Figure 21～24.

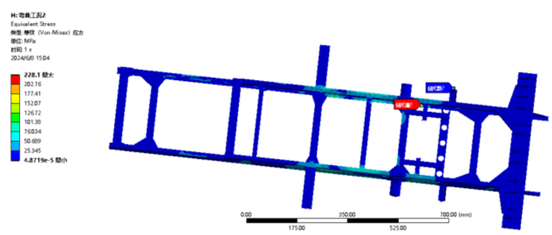


Figure 21 Stress contour of bending condition after optimization

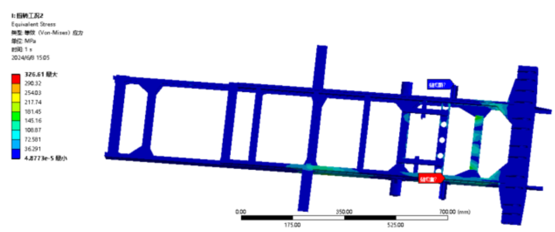


Figure 22 Stress contour diagram of torsion condition after optimization

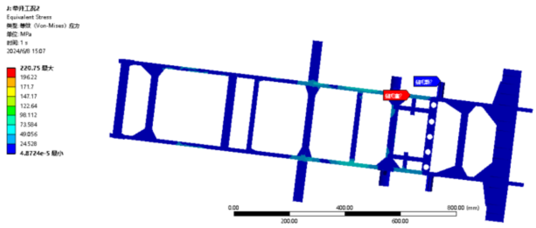


Figure 23 Stress contour diagram of the optimized lifting condition

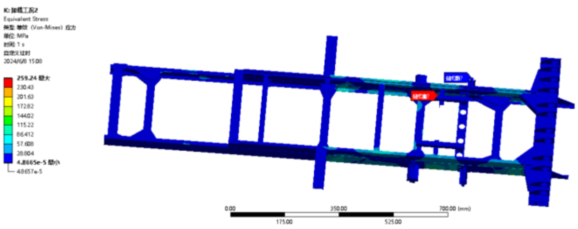


Figure 24 Stress contour diagram of unloading condition after optimization

Table 4 Comparison table of stress values under different working conditions before and after improvement

Operating conditions	Before optimizing the maximum stress(MPa)	Maximum stress after optimization(MPa)	Safety
Curved	229.1	228.1	2
Torsion	332.1	326.6	1.5
Lifting	229.1	220.8	1.5
Unload	269.0	259.2	1.5

By observing and analyzing the stress analysis diagram of the improved frame under different working conditions, it can be seen that the maximum stress area under the four working conditions is basically unchanged compared with before optimization, but the values are reduced to different degrees, the value under the bending condition becomes 228.1 MPa, the value under the torsion condition changes from 332.1 MPa to 326.6 MPa, the value decreases by 8.3 MPa under the lifting condition, and the maximum stress value decreases to 259.2 MPa under the unloading condition. And the improved frame meets the strength check under different working conditions. Table 4 compares the optimization parameters.

5 Conclusion

In this paper, a dump truck frame was taken as the research object, NX 12.0 3D software was used to construct a 3D model of the frame according to the actual parameters, and the static analysis was carried out by importing ANSYS Workbench analysis software, and the original frame structure was optimized based on this. The static analysis of the optimized frame was carried out, which successfully reduced the weight of the frame, improved the handling and stability of the vehicle, and extended its service life.

(1) The 3D model of the frame was constructed using NX 12.0 3D software according to the actual parameters, and the geometric model was reasonably simplified after importing the ANSYS Workbench analysis software, and then the meshing was carried out, and a total of 23,345 mesh elements and 59,406 nodes were generated, and the material was defined as B610L, and the finite element model of the frame was established.

(2) The static analysis of the frame was carried out under four typical working conditions: bending, torsion, lifting and unloading, and the stress contours and deformation contours under each working condition were obtained. The results show that the original frame has better strength and stiffness under four working conditions: bending, torsion, lifting and unloading, while the stress concentration is obvious under torsional conditions, and the maximum stress is close to the yield limit of the material.

(3) Based on the results of static analysis and modal analysis of the original frame, the size of the components with small load on the original frame was optimized, and the structure of the original frame was optimized by improving the connection mode and adding weight reduction holes. According to the above description, the 3D design of the optimized frame was completed, and the static analysis of the optimized frame was carried out. The results show that the structural performance of the optimized frame is more superior, and the maximum stress under each condition is significantly reduced, and it is lower than the material yield limit. At the same time, the total mass of the optimized frame has been significantly reduced by 4.38%, and the fuel economy of the whole vehicle has been improved.

References

[1] Chen Y, Zhu F. The Finite Element Analysis and the Optimization Design of the Yj3128-type Dump Truck's Sub-Frames Based on ANSYS[J]. Procedia Earth and Planetary Science, 2011(2):133-138.

[2] Ren Y, Yu Y, Zhao B, et al.Finite Element Analysis and Optimal Design for the Frame of SX360 Dump Trucks [J]. Procedia Engineering, 2017(174):638-647.

[3] Chao G, Zhiyuan L, Tianfeng M, et al.Strength analysis and optimization of three-bridge rigid frame for mining dump truck using finite element stress linearization method[J]. Journal of Physics:Conference Series,2022,2343(1):165-168.

[4] Huang Lijun, Wan Mingjun, Gao Zhigang. Lightweight design of a truck frame based on ANSYS Workbench [J]. Science & Technology Wind, 2021(19):18-21,24.

[5] Liu Qiang, Geng Jinping. Research on lightweight design and numerical simulation of electric drive dump truck frame [J]. Coal Mining Machinery, 2018,39(3):84-86.

[6] Mi Chengji, Gu Zhengqi, Jian Haigen, et al. Fatigue resistance and lightweight design of frame structure of electric wheel dump truck[J]. China Mechanical Engineering, 2017, 28(20):2455-2462.