Lightweight design of truck frame

Wei Luo, Zhongcai Zheng*, Fengliang Liu, Dongyue Han, Yifan Zhang
Shandong Jianzhu University, School of Mechanical and Electronic Engineering, Jinan, China

*Corresponding author: zhengzx@sdjzu.edu.cn

Abstract—Trucks are an important branch of the automotive field, and their market demand has been and will maintain a stable level for a long time. In this paper, a truck with a load capacity of 6t is selected as the research object. The frame of the truck is a side beam frame, which is riveted by two longitudinal beams and eight cross beams. The solid model of the truck frame is established by three-dimensional modeling software, and then the finite element software is used for static analysis of the frame, and the maximum deformation, the position of the maximum deformation, and the maximum Stress, maximum stress point. The analysis results show that the structural strength of the frame is far greater than the load-bearing strength, which meets the requirements of use, and there is a large space for lightening the structural materials. The structural optimization design of the frame is carried out, and the weight of the optimized frame is reduced by 8.4% under the premise of ensuring the strength requirements. The maximum stress under the full-load bending condition is reduced from 207.3MPa to 177.86MPa, achieving the lightweight goal.

1. Introduction
The lightweight design of automobiles is an important trend in the development of automobiles. Under the background of energy crisis and environmental protection, making automobiles lighter and more fuel-efficient while ensuring strength can effectively reduce energy consumption. Automobile lightweighting mainly starts from three aspects: structural design, selection of new materials and optimization of technology, and reasonable structural design is the most direct and simple method of lightweighting. There are many kinds of parts on the car. Among them, the frame is the base of the entire car, like the skeleton of the human body, used to support the cab, engine, gearbox, axle, drive shaft, cargo box, control system and other related components. And bear various loads from inside and outside the car to ensure the safety of the car. The total weight of the automobile frame accounts for a large proportion of the weight of the entire vehicle. The lightweight design of the frame is of great significance to the lightweight of the entire vehicle.

Hu. [1] conducted a linear static analysis on the light truck frame assembly, and conducted a topology optimization analysis on the frame assembly. After the optimization, the weight of the frame was reduced by 5%.

Wang et al. [2] took the lightweight optimization design of a domestic light truck frame as an example. The finite element strength, stiffness and modal analysis of the lightweight optimized and improved frame are carried out. At the same time, the results are verified by bench test, which provides a basis for the lightweight evaluation of trucks.

Wang et al. [3] carried out lightweight design research on side beam frame of truck. The ANSYS finite element analysis software was used to statically analyze the strength and stiffness of the frame
before and after the improvement. After the lightweight design, the strength and stiffness of the frame meet the design requirements.

Zhao et al. [4] explained the characteristics of the existing structure and process of a truck frame, and based on this structure, through the use of new materials, new structures and new processes, using NX Nastran software to design the front and rear of the lightweight design. The frame is simulated and analyzed.

Shi. [5] took a certain SUV frame as the research object, analyzed the sensitivity of each component of the frame, and selected optimized components; then according to the theory of structural optimization design, respectively defined optimization design variables, constraint functions and objective functions; finally established the lightweight design of the frame structure. The mathematical model of the vehicle frame is optimized for the size design.

Ding et al. [6] used the finite element method to establish a three-dimensional model of the truck frame in the CATIA software, and used the Hypermesh software to perform structural static analysis and modal analysis on the frame. The analysis results show that the strength, stiffness and natural frequency of the truck frame basically meet the requirements, but there are still some parts that can be optimized.

Ma et al. [7] took the domestic Dongfeng heavy-duty truck frame as the research object, used ANSYS Workbench to establish a finite element model, carried out static and dynamic analysis on the frame, and optimized the design of the frame longitudinal beam. Taking the lightweight of the frame and the minimum deformation under the external load as the optimization goal, the cross-sectional size of the longitudinal beam channel steel as the design variable, the optimal structural size parameters of the frame longitudinal beam are obtained, and the lightweight of the frame is realized.

2. Establishment of 3D model of truck frame

2.1. Establishment of 3D solid model of truck frame

Generally, the overall form of the truck frame is trapezoidal, consisting of two mirror-symmetric and parallel longitudinal beams and several cross beams. The longitudinal beams of the frame are stamped and formed from steel plates at one time, and most of the sections are grooved, which can reduce the weight of the frame and ensure the strength of the frame; the beams are also stamped and formed parts with different shapes. The longitudinal beams and the cross beams are connected together by riveting. The opening of the longitudinal beam faces inward, the opening of the front beam faces the rear, and the opening of the rear beam faces the front. The overall frame of the frame is trapezoidal, narrow in the front and wide in the rear. Figure 1 shows the frame model used in this study.

![Figure 1. Side beam frame model of truck](image)

Before establishing the 3D solid model, we split the frame assembly into parts, firstly establish the 3D solid model of the individual parts, and then use the assembly function of the 3D modeling software to assemble the frame assembly. The dimensional parameters of the main parts of the frame are shown in Table 1.
TABLE I. Dimensions of truck frame parts

| Components | Length | Width | Height | Thickness |
|------------|--------|-------|--------|-----------|
| Stringer   | 6700   | 80    | 250    | 129       |
| Beam 1     | 800    | 200   | 226    | 12        |
| Beam 2     | 776    | 80    | 202    | 12        |
| Beam 5     | 800    | 240   | 60 x 2 | 12        |
| Beam 6     | 800    | 200   | 226    | 12        |
| Beam 7     | 798    | -     | -      | φ60       |

The beams are numbered as beam 1, beam 2, ..., beam 7 in accordance with the direction of the frame from front to back.

The three-dimensional solid model of the frame assembly established by the three-dimensional modeling software is shown in Figure 2:

Figure 2. Three-dimensional model of truck frame assembly

2.2. Import of finite element model of truck frame
Before importing the finite element model of the truck frame, first convert the three-dimensional model file of the frame into a file type that can be read by the finite element software, and then import the file into the finite element software. The finite element model of the frame is shown in Figure 3.

Figure 3. Finite element model of truck frame

3. Finite element analysis of truck frame

3.1. Boundary condition setting
In the finite element analysis of the truck frame, the frame must be meshed first, and the automatic division method is used to mesh the frame. The grid size is 30mm. The grid diagram is shown in Figure 4.
After the frame is divided into meshes, there are a total of 82,556 nodes and 37,950 meshes. The quality of this grid is good, and it has the prerequisite for subsequent optimization design.

3.2 Material parameters of truck frame

The truck studied in this paper has a load capacity of 6t and belongs to a light truck. There are three types of materials used in its frame parts, namely, automotive beam steel 510L, ductile iron QT450-10, and cast steel ZG310-570. The materials and performance parameters of each part of the frame are shown in Table 2.

| Part name          | Material                  | Density (kg/m³) | Tensile strength (Pa) |
|--------------------|---------------------------|-----------------|-----------------------|
| Left and right stringer | Automobile beam steel 510L | 7850            | $5.1 \times 10^8$    |
| Stringer spring bracket 1 | ZG310-570              | 7800            | $5.7 \times 10^8$    |
| Stringer spring bracket 2 | QT450-10               | 7300            | $4.5 \times 10^8$    |
| Beam 1-7             | Automobile beam steel 510L | 7850            | $5.1 \times 10^8$    |
| Beam support         | Automobile beam steel 510L | 7850            | $5.1 \times 10^8$    |

3.3 Load calculation and constraint conditions

3.3.1 Full load bending condition

3.3.1.1 Load setting

The load of the truck studied in this paper is 7t, and the total weight of the loaded truck under full load is about 10t. The load acting surface is the upper wing surface of the left and right longitudinal beams. According to the relationship between force and pressure, the pressure is calculated as: $10 \times 1000 \times 10 \div 0.536 = 186567$ Pa. The loading diagram is shown in Figure 5.
3.3.1.2. **Boundary condition setting**
According to the installation position of the frame on the truck, the contact surface of the eight spring brackets on the longitudinal beams is selected to limit, and the boundary conditions are set as shown in Figure 6.

![Figure 6. Boundary condition setting of full load bending condition](image)

3.3.2. **Full load torsion conditions**

3.3.2.1. **Load setting**
The load setting is the same as the setting of full load bending conditions.

3.3.2.2. **Boundary condition setting**
The full-load torsion condition is to simulate the state that one wheel is lifted without touching the ground when the car is driving on the rugged road. Therefore, the situation where the right front wheel of the truck is off the ground is selected as the research condition. The boundary conditions are set as shown in Figure 7.

![Figure 7. Boundary condition settings for full load torsion conditions](image)

3.4. **Finite element simulation analysis results of truck frame**

3.4.1. **Full load bending condition**
Through the finite element software simulation analysis, the deformation cloud diagram and stress cloud diagram of the frame under full-load bending conditions are shown in Figure 8 and Figure 9.

![Figure 8. Deformation cloud diagram of frame under full load and bending](image)
Figure 9. Stress cloud image of frame under full load and bending

It can be obtained from the deformation cloud diagram that the maximum deformation of the frame under full load and bending conditions is 0.90875mm, and the maximum deformation is generated at the upper wing surface at the front end of the left and right longitudinal beams; from the stress cloud diagram, the frame is fully loaded. The maximum stress under bending conditions is 207.3MPa, and the maximum stress point is near the left and right longitudinal beams near the cross beam 2.

3.4.2. Full load torsion conditions
Through the finite element software simulation analysis, the deformation cloud diagram and stress cloud diagram of the frame under the full load torsion condition are shown in Figure 10 and Figure 11.

Figure 10. Frame deformation cloud diagram under full load torsion condition

Figure 11. Frame stress cloud diagram under full load torsion condition

It can be obtained from the deformation cloud diagram that the maximum deformation of the frame under the full-load torsion condition is 7.0949mm, and the maximum deformation is generated at the upper wing surface at the front end of the right side member; from the stress cloud diagram, the frame is under torsion and bending conditions. The maximum stress below is 292.74MPa, and the maximum stress point is near the right longitudinal beam 3 near the cross beam 3.

Combining the two working conditions, it can be concluded that in the ANSYS finite element analysis environment, the maximum stress of the frame under full load is 292.74 MPa, and whether it is bending or torsion conditions, the maximum stress point is between the longitudinal beam and near the contact of the beam.

4. Lightweight design of truck frame
Through finite element software analysis, we found that the maximum stress on the frame of a truck under full load is 292.74MPa. The materials used for the longitudinal beams and the cross beams are both automobile-specific beam steel 510L, with a tensile strength of $5.1 \times 10^8$Pa, which is much larger than the maximum stress on the frame. It shows that there is a large waste in the amount of materials for the frame, that is, there is still a lot of room for lightweighting in the structural design of
the frame. In this paper, the lightweight optimization of the frame is carried out from the perspective of increasing the weight reduction hole. The specific scheme is shown in Table 3.

### TABLE III. Optimization scheme of frame parts

| Part name              | Weight reducing hole size(mm) | Weight loss holes |
|------------------------|-------------------------------|-------------------|
| Left and right stringer| 120×160(Oval hole)           | 5                 |
| Beam 1                 | 125×160(Oval hole)           | 4                 |
| Beam 1                 | 130×160(Oval hole)           | 2                 |
| Beam 2-4               | 120×140(Oval hole)           | 6                 |
| Beam 6                 | 94×140(Oval hole)            | 4                 |
| Beam 6                 | 40×60×140×160(Cross hole)    | 1                 |
| Assembly              | 120×140(Oval hole)           | 6                 |

4.1. The establishment of a three-dimensional model of the optimized frame

After the optimization model is established, follow the previous steps to assemble the frame assembly. The assembly model is shown in Figure 12.

![Figure 12. The optimized three-dimensional model of the frame assembly](image)

Weight comparison before and after optimization: Use the finite element software to find the material properties of the query function to obtain the weight of each component of the frame before and after optimization. The specific numerical comparison is shown in Table 4.

### TABLE IV. Comparison of weight before and after frame optimization

| Part name | Quantity | Weight before optimization (kg) | Weight after optimization (kg) | Weight reduction (kg) |
|-----------|----------|---------------------------------|-------------------------------|-----------------------|
| Stringer  | 2        | 416.02                          | 391.86                        | 24.16                 |
| Beam 1    | 1        | 27.202                          | 22.541                        | 4.661                 |
| Beam 2    | 1        | 21.455                          | 17.628                        | 3.827                 |
| Beam 3    | 1        | 21.455                          | 17.628                        | 3.827                 |
| Beam 4    | 1        | 21.455                          | 17.628                        | 3.827                 |
| Beam 6    | 1        | 29.55                           | 24.889                        | 4.661                 |
| Assembly  | 1        | 537.137                         | 492.174                       | 44.963                |

According to the comparison of the data, the optimization scheme has reduced the weight of the frame assembly by 44.963kg, and the weight reduction ratio is about 8.4%.

4.2. Finite element analysis of optimized frame

4.2.1. Optimize the analysis results of fully loaded bending conditions of the frame

The deformation cloud diagram and stress cloud diagram of the optimized frame under full-load bending conditions are shown in Figure 13 and Figure 14.
Figure 13. Deformation cloud diagram of optimized frame under full load bending

Figure 14. Stress cloud image of optimized frame under full load bending

It can be obtained from the deformation cloud diagram that the maximum deformation amount of the frame under full load and bending conditions is 1.1579mm, and the maximum deformation amount is generated at the upper wing surface at the front end of the left and right side members; from the stress cloud diagram, the frame is fully loaded. The maximum stress under bending conditions is 177.86 MPa, and the maximum stress point is near the left and right longitudinal beams near the cross beam 2.

4.2.2. Optimize the analysis results of fully loaded torsion conditions of the frame

Figure 15 and Figure 16 show the deformation cloud diagram and stress cloud diagram of the optimized frame under full load torsion conditions.

Figure 15. Deformation cloud diagram of optimized frame under full load torsion conditions

Figure 16. Stress cloud image of optimized frame under full load torsion conditions

From the deformation cloud diagram, the maximum deformation of the optimized frame under the full load torsion condition is 7.6749mm, and the maximum deformation is generated at the upper wing
surface at the front end of the right side member; from the stress cloud diagram, the frame is twisting. The maximum stress under bending conditions is 315.67 MPa, and the maximum stress point is near the right longitudinal beam 3 near the cross beam 3.

In view of the two working conditions, after adding weight reduction holes on the frame, the maximum stress of the frame under full-load torsion conditions has increased, but it is still less than the strength limit of the material, which is sufficient for practical use; After the heavy hole, the maximum stress of the frame under full-load driving conditions is reduced from 207.3MPa to 177.86MPa, which improves the service conditions of the truck and effectively extends the material life.

5. Conclusions and prospects

In this paper, the frame of a truck with a load capacity of 6t is selected as the research object. First, a three-dimensional solid model of the frame is established using three-dimensional modeling software. During the modeling process, some parts are simplified in detail so that the follow-up is limited. The feasibility of meta-analysis. After the model was built, the finite element analysis of the frame was carried out. The study selected two working conditions of the truck with normal full load and full load on rugged roads as the analysis environment. The two conditions were simulated by setting the boundary conditions in the finite element software. In this working condition, the final result is that the strength of the frame is much greater than the maximum stress experienced during driving, and there is a large space for lightening the frame in the use of materials.

This article draws on other related papers, and uses the method of adding weight reduction holes to carry out the lightweight design of the frame. After performing finite element analysis on the optimized model, the total weight of the optimized design frame is reduced by 44.963kg, and the weight reduction ratio is about 8.4%; the maximum stress of the optimized frame under full load bending conditions is determined by 207.3MPa is reduced to 177.86MPa, but the maximum stress of the frame under the full-load torsion condition has increased, which is still within the strength range and meets the actual conditions of use. The analysis results show that the optimized design of this time has achieved the purpose of lightweight trucks, and the design scheme is feasible.

As an important research direction of automobile development, automobile lightweight is only increasing with the development of technology. Although this article has realized the lightweight optimization of the truck frame from one aspect, there are still many imperfections in the specific research, which can be mainly reflected in the following points:

1. The establishment of the frame model simplifies the details of many parts, which is different from the real model;
2. The operating conditions of the car only choose the full-load bending and torsion conditions. In reality, there are many other operating conditions, such as acceleration conditions and vibration conditions;
3. In addition to adding weight-reducing holes, there are many other methods for lightweighting the frame, such as selecting new materials and redesigning the spatial layout and shape of parts.

REFERENCES
[1] C. Hu, “Analysis and Optimization of Frame Assembly Structure of Light Truck,” Jiangsu University, 2016.
[2] Q. Wang, J. Zhao, F. Yang, Z. Mou, “Evaluation method and optimization design of lightweight truck frame,” Forging Technology, vol. 42, no. 9, pp. 174-181, 2017.
[3] H. Wang, P. Xiao, P. Shi, “Lightweight design of a truck side beam frame based on ANSYS,” Journal of Shaanxi University of Technology (Natural Science Edition), vol. 33, no. 3, pp. 5-9, 2017.
[4] J. Zhao, Q. Wang, J. Chen, “Structure improvement and simulation analysis of a truck frame,” Journal of Chengdu University of Technology, vol. 21, no. 2, pp. 5-11, 2018.
[5] W. Shi, “Research on lightweight technology of frame based on finite element method.” Internal Combustion Engine and Parts, vol. 23, pp. 21-24, 2018.

[6] Z. Ding, Q. Liu, “Finite element analysis of truck frame,” Equipment Machinery, vol. 4, pp. 15-20, 2018.

[7] H. Ma, B. Liu, X. Li, T. Tao, “Static and dynamic analysis and optimization design of Dongfeng heavy-duty truck frame,” Mechanical Engineering and Automation, vol. 5, no. 2, pp. 105-107+110, 2018.