Strength Research and Structure Improvement of a Semitrailer Based on Finite Element Method

Yajuan Chen
Institute of artificial Intelligence, Chongqing Real Estate College, Chongqing, China
e-mail: 365379562@qq.com

Abstract. Based on the finite element theory, the analysis model of semitrailer is established. On this basis, the limit condition method is applied to simulate the stress level of the structure under four typical working conditions including bending, torsion, turning and emergency braking. The weak areas with insufficient strength are found out. Meanwhile, it proposes corresponding improvement measures based on the analysis results to provide a theoretical basis for the structural design engineer.

1. Introduction
The frame of semitrailer is important transport machinery and load-bearing part. This structure not only can bear the static load of the whole vehicle, but also can bear all kinds of dynamic loads in the driving of semitrailer. Therefore, its reliability is related to the safety and reliability of semitrailer driving. Therefore, it is very important to analyze and study the strength and reliability of the body structure in the product development stage by using the finite element method to improve the stability, reliability and safety of the product quality.

In this paper, the semitrailer of a recreational vehicle is taken as the research object. Based on the simplification of its structure, a reasonable mechanical model is established using the finite element technology. The stress conditions of the structure are analyzed under the four limit working conditions of bending, torsion, turning and emergency braking. The relatively weak area of strength is found out, and the corresponding improvement measures and optimization schemes are put forward to improve the structure Strength performance, to meet the user's use requirements.

2. Theory of finite element analysis
Finite element method is one of the most widely used numerical calculation methods in engineering analysis, which can be used to analyze the stress, deformation and dynamic of various structures with irregular geometry, complex loading and supporting conditions [1-2].

The essence of the mechanical analysis of the element is to establish the relationship between the element strain and the element node displacement, the relationship between the element stress and the node displacement by using the geometric and physical equations, so as to further establish the element stiffness matrix by using the virtual work principle. In this paper, the mechanical properties of elements are mainly considered from the following aspects.

The relationship between element strain and element node displacement can be established by using geometric equation.

\[
\{\varepsilon\} = [B]\{\delta\}^e
\]  (1)
In which, \( \{ \varepsilon \} \) is the strain matrix of any point in the element, and \( [B] \) is the strain matrix.

Based on the physical equation, the relationship between element stress and element node displacement can be established.

\[
\{ \sigma \} = [D]\{ \varepsilon \}
\]

(2)

\[
\{ \sigma \} = [D][B]\{ \delta \}^e = [S]\{ \delta \}^e
\]

(3)

In which, \( \{ \sigma \} \) is the stress array of any point in the element, and \( [D] \) is the elastic matrix related to materials, and \( [S] \) is the stress matrix.

Using the principle of virtual work or variational method, the stiffness matrix of the element can be established, and its stiffness equation is as follows.

\[
\{ F \}^e = [K]^e\{ \delta \}^e
\]

(4)

In which, \( \{ F \}^e \) is the nodal force matrix of the element, and \( [K]^e \) is the stiffness matrix of the element.

In the static analysis, according to the principle of static equivalence, the load of each element can be shifted to the node. The sum can be obtained. Thus, the equivalent node load array \( \{ F_p \} \) of the structure can be obtained.

According to the total stiffness matrix \( [K] \) of each node's related element set structure, the balance equation of the whole structure can be established.

\[
\{ F \} = [K]\{ \delta \}
\]

(5)

According to the node displacement, the stress of each element can be calculated. According to the calculated stress value, it can analyze and draw a conclusion.

3. Modeling frame finite element modeling

3.1. Simplification of frame structure

According to the requirements of the analysis, the model is processed into the geometric elements needed for mesh generation in 3D software. When establishing the geometry model of the finite element analysis of the semitrailer, only the important parts of the structure concerned in this analysis are established, and the non-important parts are ignored or simplified.

The semitrailer is divided into the underframe part and the suspension part. According to the structural characteristics, the left and right beam, tail beam, front beam and traction beam in the underframe components have adopted the method of extracting the middle plane for structural geometry processing. The support leg components are simplified, which are used the inertia mass points such as equal mass. The connecting plate in the suspension part has adopted the method of extracting the middle plane. The front beam, damping rubber, rocker shaft and rubber pad are adopted solid structure. Thus, the geometric analysis model is shown in Figure 1.
3.2. Finite element discretization of frame structure
Because the cross-sectional dimensions of the left and right beams, tail beams, front beams, traction beams and suspension connecting plates are made of steel plates of various sections or connected by bolts are very small relative to the length of the members, they are suitable for discretization with shell elements, and the shell elements are used to simulate the welding between members. The model mesh is composed of the most quadrilateral shell elements and a few triangular elements. The suspension front beam, damping rubber, rocker shaft welding combination and rubber pad are adopted tetrahedron and hexahedron elements according to the structural irregularity.

The support leg is simulated by mass points, and the connection relationship with beam structure is fitted by rigid element MPC. Rod element and rigid element MPC are used to simulate shock absorber components and bolts. In the modeling, nine pairs of contact pairs are established. The resulting finite element model is shown in Figure 2.

3.3 Material properties
The structure mainly uses steel and rubber materials. The main structure of the frame is made of ordinary carbon steel Q345. Its material properties are following. Modulus of elasticity is $2.1 \times 10^5 MPa$, Poisson's ratio is 0.3 and density is $7.85 \times 10^{-9} t/mm^3$.

The damping rubber is defined as hyper-elastic material, which constitutive parameters are $C_{10}$ and $C_{01}$. In which, $C_{10}$ is 0.5621 and $C_{01}$ is 0.04751. Density is $0.9 \times 10^{-9} t/mm^3$.

3.4 Boundary condition
When using the finite element method to carry out the static analysis of the structure, the results obtained are unique. From the mathematical point, it can be understood that the structure stiffness matrix composed of all the element stiffness matrix is non-singular. So, the structure must be constrained to eliminate the displacement of the rigid body in the static analysis [3-5].

In different motion states, the corresponding constraints of the frame are different. In the simulation calculation, the constraints are applied at the front traction, the front and rear rocker arm assembly holes respectively.

In this paper, the forward direction of the vehicle is the X-axis; the left side of the driver is the Y-axis, and the Z-axis is determined according to the right-hand rule, and the vertical ground is upward.
3.5. Selection of analysis conditions
In the driving process of real vehicle, there are four typical driving conditions, which are uniform straight driving, one-wheel suspension, emergency braking and sharp turning. Thus, the basic deformation of the frame includes bending deformation, torsion deformation and bending and torsion composite deformation.

In this paper, the stress of frame under each working condition is analyzed. During the finite element analysis and calculation, from the safety point of view, the applied external load is the worse situation in actual use, in order to ensure the safe driving ability of the vehicle to the maximum extent.

4. Structural strength analysis and result judgment

4.4. Strength analysis

4.4.1. Full load bending condition. The bending condition of the frame structure is mainly to analyze the stress of the frame under full load or uniform speed, and to evaluate the static strength of the frame structure. The load distribution under full load is that the total load is distributed to the front plate, middle plate, rear plate, front crossbeam and rear crossbeam of longitudinal beam.

In the analysis, the translational degrees of freedom in the X, Y, Z direction of the front traction of the frame are constrained, and the translational degrees of freedom in the Y, Z direction of the left and right assembly holes of the front rocker arm and the rear rocker arm are constrained. The stress calculation results of the frame are as shown in Figure 3.

![Figure 3. Cloud chart of frame stress distribution under bending condition](image)

It can be seen from the analysis results that under this working condition, the maximum stress on the longitudinal beam is about 220MPa. The longitudinal beam structure corresponding to the suspension beam connecting plate and the surrounding area of the following two bolt connection holes have stress concentration which is about 520MPa.

4.4.2. Full load torsional condition. If the semitrailer passes through the rough road at low speed during driving, the frame will generate static torsion distance under the action of asymmetric load or under the condition of asymmetric support, which will cause the frame to twist. The frame torsion bending is a dangerous working condition during driving. This condition concludes single side suspension torsion condition of the first axle and second axle. In the project analysis, only single side suspension torsion condition of the first axle is analyzed.

In the analysis, the translational degrees of freedom in the X, Y and Z directions at the front traction of the frame are constrained; the translational degrees of freedom in the y and z directions at the assembly holes on the right side of the front rocker arm and the left and right sides of the rear rocker arm are constrained to release the degrees of freedom on the left side of the front rocker arm. The stress calculation results of the frame are as shown in Figure 4.
Under this working condition, the maximum stress on the longitudinal beam of the main structure of the car body is about 235MPa. The longitudinal beam structure corresponding to the suspension beam connecting plate and the surrounding area of the following two bolt connection holes have stress concentration which is about 550MPa.

4.4.3. **full load braking condition.** In case of emergency braking during the driving process of semitrailer, the frame is affected by the weight of the load and the gravity of each component, as well as the inertial force generated along the longitudinal direction.

In this case, the frame is subjected to the horizontal bending load and the longitudinal dynamic load caused by the inertia force during braking. The inertia force mainly depends on the deceleration generated by the frame when braking. In this paper, the static and dynamic analysis method is used to analyze and calculate the braking condition of the frame with the maximum braking acceleration of 0.7g. The restraint condition is the same as the full load bending condition. The stress calculation results of the frame are as shown in Figure 5.

Under this working condition, the maximum stress on the longitudinal beam of the main structure of the car body is about 250MPa. The longitudinal beam structure corresponding to the suspension beam connecting plate and the surrounding area of the following two bolt connection holes have stress concentration which is about 550MPa.

4.4.4. **full load turning condition.** In the emergency turning condition, the frame not only carries the weight of goods and components, but also the whole frame structure is subjected to centrifugal force which produces side load. The analysis of the emergency turning working condition mainly studies the influence of centrifugal force when a lateral acceleration of 0.4g is applied.

In the analysis, the translational degrees of freedom in the X and Z directions of the traction at the front end of the frame are constrained, and the translational degrees of freedom in the Z direction of the assembly holes on the left and right sides of the front rocker arm are constrained, and the translational degrees of freedom in the Y and Z directions of the left and right sides of the rear rocker arm are constrained. The stress calculation results of the frame are shown in Figure 6.
Under this working condition, the maximum stress on the longitudinal beam of the main structure of the car body is about 240MPa. The longitudinal beam structure corresponding to the suspension beam connecting plate and the surrounding area of the following two bolt connection holes have stress concentration which is about 680MPa.

4.5. Result analysis and improvement

4.5.1. Theoretical basis of result analysis. In the strength check, the judgment of the analysis results is mainly based on the mechanical properties of the selected materials and the strength check criteria. In this paper, the static strength method and linear analysis method are used to simulate the stress distribution of the frame under various limit conditions. The premise of the analysis is that the material of the frame is in the elastic range under the action of various external forces, and no plastic deformation occurs. Therefore, the fourth strength theory, which is Mises equivalent stress yield criterion, is selected as the strength judgment criterion for the analysis results [6].

In the static analysis of the frame, the Mises stress at any point in the structure can be expressed as equation (6).

$$\sigma_r = \frac{1}{\sqrt{2}} \sqrt{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}$$

(6)

In which, $\sigma_1$, $\sigma_2$ and $\sigma_3$ represent three main stress values at any point.

The strength of the structure without fracture or failure within the allowable service life shall meet the following requirements:

$$\sigma_r \leq [\sigma]$$

(7)

$$[\sigma] = \sigma_s / n_s$$

(8)

In which, the $\sigma_s$ is the allowable stress limit value of the material. The $\sigma_s$ is the yield strength limit of the material. The $n_s$ is the safety factor of the material, which is usually taken as 1.3 on the base of the analytical experience. The frame is made of Q345 and its yield strength is 345MPa. So, the allowable stress under the ultimate load is 265.4MPa.

4.5.2. Strength judgment results. From the results of frame strength analysis under the above four common driving conditions, it can be obtained that the maximum stress of frame main structure longitudinal beam is 255MPa, which is less than the allowable stress of the material and can meet the strength use requirements.

The maximum stress of the longitudinal beam structure corresponding to the suspension beam connecting plate and the surrounding area of the following two bolt connection holes is 680MPa, which is far more than the allowable stress of the material.
Through the above analysis, the longitudinal beam structure corresponding to the suspension beam connecting plate and the surrounding area of the following two bolt connection holes have large stress area, which is the weak strength area, and this part will produce early fatigue fracture. It is necessary to strengthen the frame and adjust the structure to meet the users’ requirements.

4.5.3. **improvement opinions.** Through the strength finite element analysis of the frame, the above conclusions with guiding significance are obtained. Here, we further discuss the improvement measures for the frame structure.

- increasing the section size of longitudinal beam properly;
- changing the turning length of longitudinal beam;
- adding connecting plates at the connecting bolts of left and right longitudinal beams and axle connecting plates;
- changing the location of bolt connection holes;
- to be made of high-grade materials for the frame pipe shall;
- designing stiffeners or weld stiffeners in the high stress area of the frame.

Of course, the above improvement measures are only from the point of view of local improvement of static strength, the optimization of the frame is a system engineering. We do not advocate the continuous pursuit of high strength, affecting the comprehensive performance indicators of the frame system. Therefore, it is necessary to optimize the whole structure from the performance point of view and take the most practical and reliable scheme in the later stage.

5. Conclusion

In this paper, the semitrailer of a caravan is taken as the research object. The static finite element analysis is carried out to simulate the frame stress of the structure under the bending, torsion, turning and emergency braking conditions respectively. According to the analysis results, the weak area of the frame strength is found out, and the improvement suggestions are put forward, which provide theoretical support and guidance suggestions for the overall optimization of the structure. It may occur in the current independent development and use of vehicles, we must use advanced and scientific analysis methods to predict and solve problems, so as to reduce losses and ensure the high quality of vehicles.

Acknowledgments

This research is supported by the scientific and technological research projects of Chongqing Education Commission (NO. KJQN 201805202).

References

[1] Gao Yunkai. (2006) Analysis of Automobile Body Structure [M]. Beijing University of Technology Press, Beijing.

[2] Yu Zhisheng. (2006) Automobile Theory [M]. Mechanical Industry Press, Beijing.

[3] Zhang Guofen, Zhang Wenming, etc. Structure Design and Strength and Stiffness Analysis of an SGA92150 Semitrailer Frame[J]. Journal of Beijing University of Science and Technology, 2007(7):744-749.

[4] Wu Wei, Li Shoucheng. Finite Element Analysis and Experimental Research on Special Semitrailer Frame. Mechanical Strength, 2016(6):1361-1365.

[5] Zhong Peisi, Zhao Dan, Sun Xueyan. Modeling and Analysis of Truck Chassis Based On FEM[J]. Machinery Design and Manufacture, 2008(6):52-53.

[6] Theoretical Mechanics Teaching and Research Office of Harbin University of Technology. (1997) Theoretical Mechanics[M]. Higher Education Press, Beijing.