Transactions on Computer Science and Intelligent Systems Research
ISSN: 2960-1800, eISSN: 2960-2238 | Volume 8
AICT 2024

Simulation of Medium Truck Frame Based on ANSYS

Chang Xu[1, a]

[1]School of Mechatronics and Vehicle Engineering, Chongqing Jiaotong University, Chongqing, China

[a]chan9shun@163.com

Abstract. The frame is the important part of the car, as the understructure of assemble. the important assembly and structure on the car are based on the frame, and fixed to corresponding position, such as, engine, transmission, steering system and cab and so on. Force situation of the frame is very complex, it supports various load from road and loading, a large proportion of vehicle performance lie on structure performance of the frame. Therefore, the frame must have adequate intensity assure its fatigue life, and adequate rigidity assure its requirement of assemble and use. The requirements of modern automobile development are not limited to traditional mechanical analysis methods. This paper takes the frame of a medium truck as the research object using the finite element analysis software ANSYS to carry on the static mechanics. In this paper, the three-dimensional model is made by SolidWorks as the carrier according to the parameters of a selected frame, and leading-in the ANSYS to carry on finite element analysis.

Keywords: Frame, Finite Element Analysis, SOLIDWORKS, ANSYS.

1. Introduction

With the widespread use of automobiles in modern society, people have higher demands for vehicles, particularly regarding the economy, safety, and environmental performance of commercial trucks. The frame is a crucial component of a vehicle, making its analysis highly significant. By conducting finite element analysis on the frame to obtain deformation and stress under various conditions, designers can identify defects early, allowing for optimization and modification of the design, thereby ensuring the durability of the frame.

In 2014, Jilin University discussed the key research topic of the National Science and Technology Development Plan of Jilin Province, titled "Comprehensive Performance Matching and Lightweighting of Automobile Chassis Components. [1] " Sheng J et al[2] used the finite element optimal design module to conduct a lightweight design of a bus frame, discussing the method of finite element optimization design, which achieved a 5% reduction in vehicle body weight. By using FEM software, they simulated a car crashing into a wall at different speeds and conducted both static and dynamic analyses of the collisions at various speeds. The results of these simulations were then compared with actual crash test data, providing valuable reference for vehicle design and research.

2. Related Models and Their Improvement Methods

A. Establishing a Frame Model

The overall length of the vehicle is essentially the same as the total length of the frame. The front and rear overhangs should not be too long, as excessively long overhangs will increase the approach and departure angles of the vehicle beyond normal levels, thereby reducing its off-road capability. Along the entire length of the longitudinal beams, the front portion of the frame, which supports structures such as the cabin, carries a lighter load than the rear. Therefore, the cross-sectional area of the longitudinal beams in the front should be appropriately reduced. In contrast, the rear of the frame supports the cargo compartment and load, where the load is heavier, so the cross-sectional area of the longitudinal beams should be increased accordingly. The structure of the frame is shown in Figure 1.

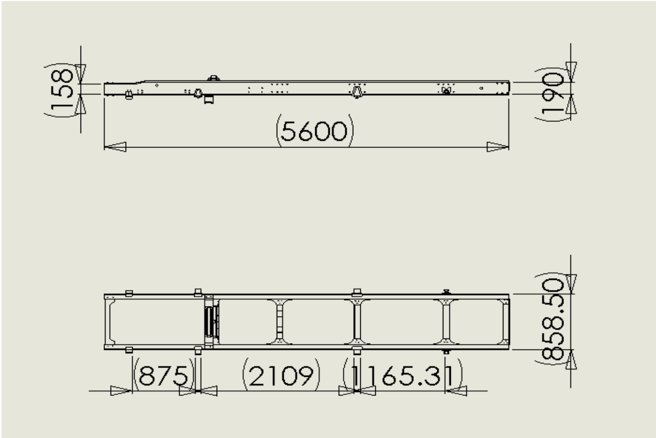

Figure 1. Frame Structure Diagram

Since this design is for a medium-duty truck frame, which bears significant loads, the material chosen for the frame is a low-alloy high-strength structural steel, Q345 [4]. The main parameters are shown in Table 1.

Table 1. Frame Material Parameters

Poisson's Ratio	Density	Yield Strength
0.31	7850 kg/m³	345 MPa

The frame's 3D model, created using SolidWorks, is saved in x-t format and imported into ANSYS Workbench software, as shown in Figure 2. The model is then converted into a solid in DesignModeler. This approach eliminates the need for additional modeling in ANSYS, achieving a seamless integration between CAD software and finite element analysis.

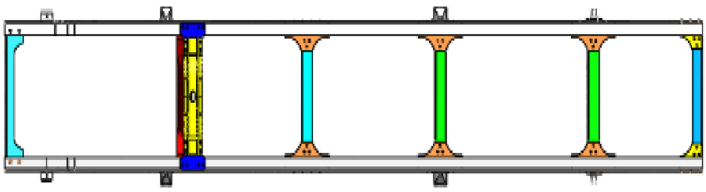

Figure 2. Top View of the Frame

B. Steps in Finite Element Analysis

When applying the finite element method to solve practical problems, the following steps are generally used:

1) Discretization of the Structure: Divide the entire structure into smaller elements of similar shape and size. Nodes can connect these subdivided elements. Finer mesh divisions lead to more accurate results [5]. Thus, discretization is a core technique for the reliable application of finite element technology.

2) Selection of Displacement Mode: Displacement, stress, strain, and other physical quantities are represented by the displacements of nodes. After determining the displacement mode of the elements,

compute the displacement at any point within the element based on the node displacements. The element's deformation mode is also known as the shape function.

3) Analysis of Element Mechanical Properties: Determine the relationship between the displacements and forces of the element nodes based on the material properties, dimensions, shape, position, and number of nodes of the element.

4) Assembly of Global Balance Equations: Combine the balance equations of all elements to obtain the global balance equations for the entire structure. Reconnect the individual elements based on the balance conditions.

5) Solve for Node Displacements: Use the balance equations to determine the displacements of the nodes.

6) Calculate Strain and Stress: Compute the strain and stress within the elements based on the node displacements.

C. Applying Loads and Boundary Conditions

When a vehicle is in motion, the frame's load mainly comes from the weight of the springs. The load sources are as follows:

1) Powertrain: Includes the mass of the engine, transmission, and other components.

2) Cabin Mass: Includes the mass of the cabin and its occupants.

3) Cargo Box and Load: The mass of the cargo box and the cargo it carries.

4) Fuel Tank and Battery: The mass of the fuel tank and battery.

During the analysis, small components such as the fuel tank and battery can be neglected.[6]

When applying loads, the masses of various components, including the powertrain, cabin, and cargo, are applied to their respective points of action on the frame. Since this study analyzes the frame of a medium-duty truck, the analysis conditions should be considered under fully loaded conditions. The masses of the components are listed in Table 2.

Table 2. Component Masses

Component Name	Powertrain	Cabin	Rated Load
Mass (kg)	800	600	4000

To apply the load, first open the Mechanical platform. In the Geometry section, select Point Mass to apply the mass. When applying the mass, first select the surface where the mass will be applied, and then add the mass.

In practice, due to the complex operating conditions of a vehicle, the forces on its components are also very complex, especially for the frame. The forces that the frame typically endures include: the weight of heavy components such as the powertrain, centrifugal forces during acceleration, deceleration, or turning, and forces from uneven road surfaces.[7] Various combined deformations, torsions, and bending of the frame are caused by these forces, placing high demands on the frame's stiffness and strength.

During the analysis, the frame's leaf spring seats should be constrained.[8] The constraints and their locations are shown in Table 3.

Table 3. Constraints on the Frame and Their Locations

Location	Front Spring	Front Spring	Rear Spring	Rear Spring
	Front Support	Rear Support	Front Support	Rear Support
Constraint	XYZ	YZ	Z	YZ
	Axial Translation	Axial Translation	Axial Translation	Axial Translation

When applying constraints to the frame, open the Mechanical platform, select Static Structure, and add Remote Displacement under Supports to apply the constraints. The details of the constraints are shown in Figures 3 and 4.

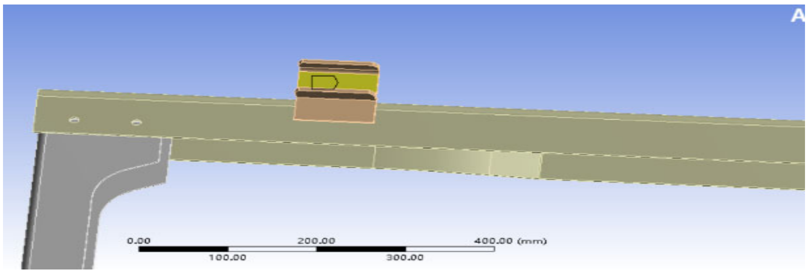

Figure 3. Select Constraint Facesre

Type	Remote Displacement
X Component	0. mm (ramped)
Y Component	0. mm (ramped)
Z Component	0. mm (ramped)
Rotation X	Free
Rotation Y	Free
Rotation Z	Free
Suppressed	No

Figure 4. Define Constraints

3. Experiment and Result Analysis

A. Fully Loaded Load Calculation

Vehicle Parameters are as shown in Table 4:

Table 4. Vehicle Parameters

Powertrain Mass	Cabin and Occupant Mass	Rated Payload Mass
800kg	600kg	4000kg

Full load conditions refer to the scenario where the vehicle is fully loaded with cargo, passengers, and fuel, and is either stationary or moving at a constant speed on relatively smooth roads. In this condition, the vehicle is only subjected to gravitational acceleration without any other forms of acceleration. Figure 5 shows the application of acceleration and load on the frame.

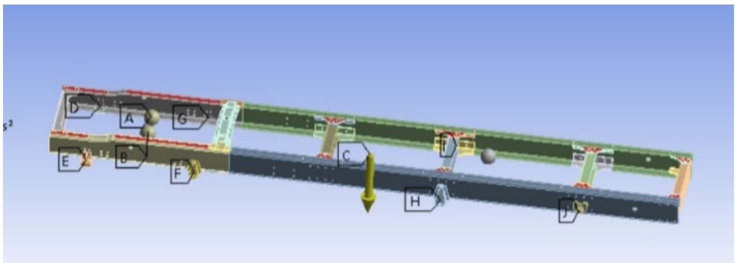

Figure 5. Acceleration and Load Diagram of the Frame Under Fully Loaded Conditions

After performing the finite element analysis, the frame's deformation and stress are obtained, as shown in Figures 6 and 7.

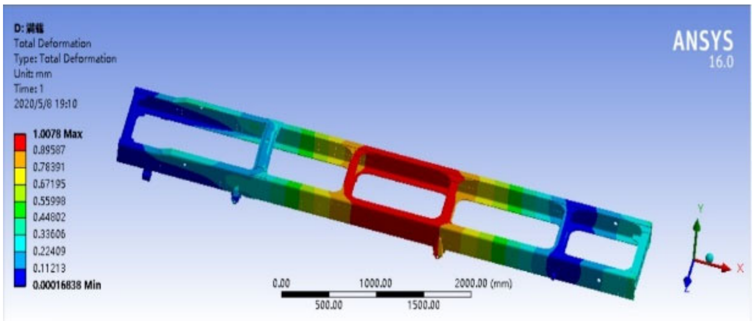

Figure 6. Deformation of the Frame Under Fully Loaded Conditions

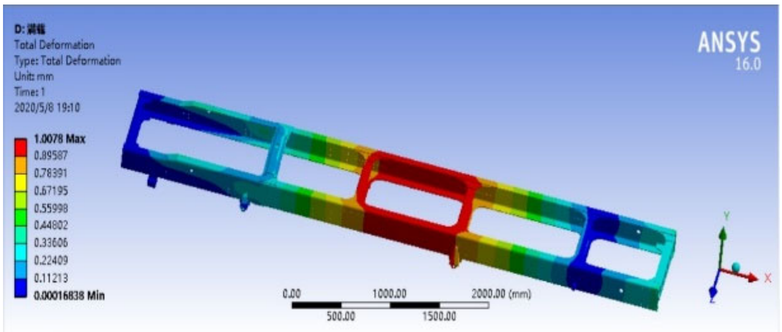

Figure 7. Stress in the Frame Under Fully Loaded Conditions

The results from the finite element analysis show that under fully loaded conditions, the maximum deformation of the frame occurs between the third and fourth cross beams and the longitudinal beams connecting them, with a value of 1.0078 mm. Since the forces acting on the frame are largely concentrated in the mid-rear part of the frame under fully loaded conditions, the deformation in this area is relatively larger compared to other parts. However, this value is quite small relative to the size of the frame. The maximum stress occurs near the left side of the second cross beam’s lifting lug, with a value of 139.96 MPa, which is well below the material’s strength requirements.

B. Calculation of Acceleration Under Maximum Traction Force

When the vehicle is under maximum traction force, which occurs when the engine is delivering its maximum torque, typically during startup or rapid acceleration [9], the load experienced includes the cargo mass. The maximum torque output from the engine is transmitted through the transmission system to the drive wheels. The drive wheels experience a tangential reaction force from the ground, which is then transmitted to the frame through components such as the suspension and thrust rods, causing tangential acceleration of the frame. At this time, the maximum tangential reaction force exerted by the ground on the wheels is as follows [10]:

$$F_x = \frac{T_{e\max} i_1 i_0 \eta_T}{r_r} \tag{1}$$

T_{emax} is the engine's maximum torque, with T_{emax} = 600 N·m; i_1 is the transmission's first gear ratio, with i_1 = 13; i_0 is the final drive ratio, with i_0 = 5; η_T is the transmission efficiency, with η_T = 0.8; r_r is the rolling radius of the wheel, with r_r = 0.4 m.

When the engine is outputting its maximum torque and the transmission is in first gear during acceleration, the maximum tangential reaction force F_x exerted by the ground on the wheels is given by:

$$F_x = \frac{600*13*5*0.8}{0.4} = 78000\text{N} \tag{2}$$

The acceleration of the vehicle under maximum traction force is:

$$a_x = \frac{Fx}{G} = \frac{78000}{11500} = 6.78\text{m}/\text{s}^2 \tag{3}$$

Under these conditions, the frame experiences both gravitational acceleration and acceleration due to maximum traction force. These are applied in the finite element analysis. Figure 8 shows the application of frame acceleration and load.

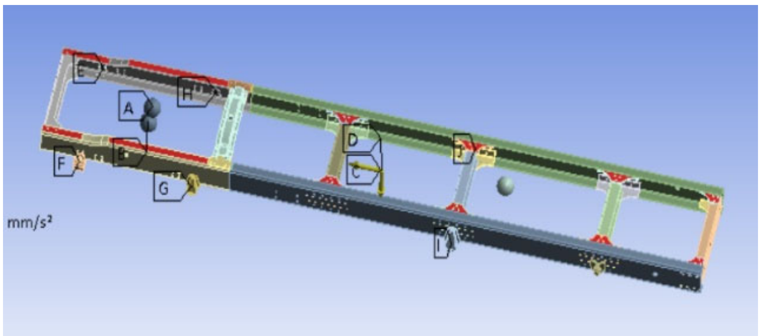

Figure 8. Acceleration and Load of the Frame Under Maximum Traction Force

The analysis results under maximum traction force, obtained through the above calculations, are shown in Figures 9 and 10.

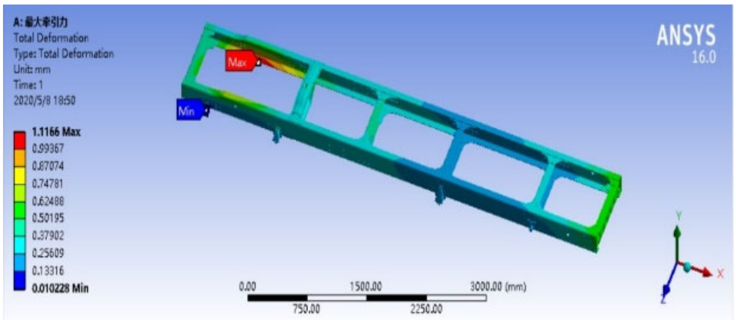

Figure 9. Deformation of the Frame Under Maximum Traction Force Conditions

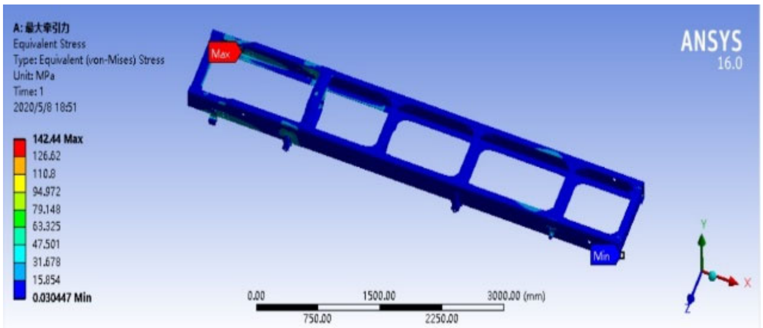

Figure 10. Stress in the Frame Under Maximum Traction Force Conditions

The results from the finite element analysis show that under maximum traction force, both the maximum deformation and maximum stress of the frame occur in the longitudinal beam between the first and second cross beams. Due to the high engine power and intense vibrations under maximum traction force, the deformation and stress in this area are greater compared to other parts.[11] The maximum deformation is 1.1166 mm, and the maximum stress is 142.44 MPa, which meets the material's strength requirements.

C. Calculation of Acceleration Under Maximum Braking Force

The maximum braking force of the vehicle depends on both the ground braking force and the brake system force, with the smaller of the two being the determining factor. Due to the large mass of the truck, wheel lock-up may occur during braking, so the maximum braking force should be chosen as the force when the wheels are locked up. At this time, the maximum braking force of the wheels is:

$$F_b = \varphi G \tag{4}$$

φ is the coefficient of friction for the wheels, taken as 0.85.

$$F_b = 0.85 * 112700 = 95795N \tag{5}$$

The acceleration of the vehicle under maximum braking force is:

$$a_b = \frac{F_b}{m} = \frac{95795}{11500} = 8.33m/s^2 \tag{6}$$

Under these conditions, the vehicle experiences both gravitational acceleration and rearward acceleration due to braking force. These are applied in the finite element analysis. Figure 11 shows the application of frame acceleration and load.

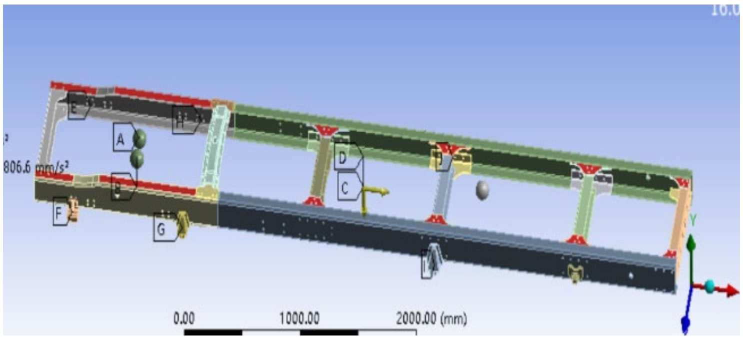

Figure 11. Stress in the Frame Under Maximum Traction Force Conditions

The results obtained from the calculations are shown in Figures 12 and 13.

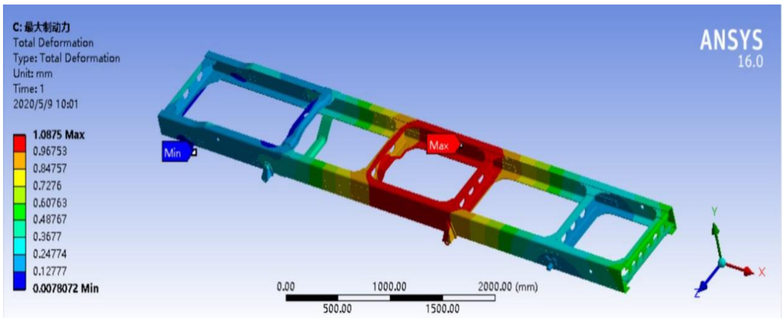

Figure 12. Deformation of the Frame Under Maximum Braking Force Conditions

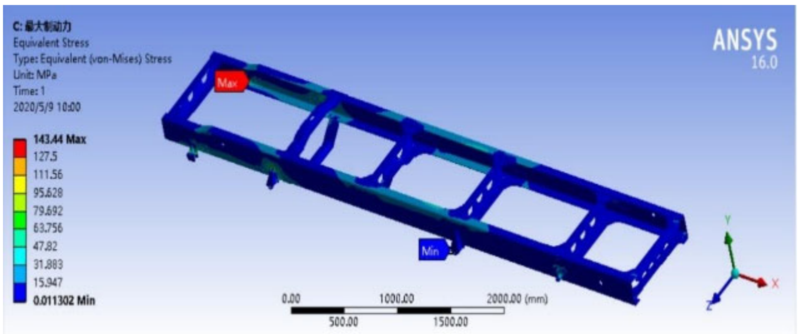

Figure 13. Stress in the Frame Under Maximum Braking Force Conditions

The results from the finite element analysis show that under maximum braking force conditions, the maximum deformation of the frame occurs in the longitudinal beam between the third and fourth cross beams, with a value of 1.0875 mm. This deformation is very small relative to the frame dimensions and does not pose a significant threat to the vehicle's operational safety. The maximum stress occurs at the front of the longitudinal beam, with a value of 143.44 MPa, which is still far below the material's yield strength, thus meeting the strength requirements.

4. Conclusion

In this study, we have developed a comprehensive finite element analysis of a medium-duty truck frame to evaluate its performance under various loading conditions. The process began with the construction of a detailed 3D model of the truck frame, which was subsequently imported into ANSYS software for further analysis. The focus was on understanding the structural behavior of the frame under different scenarios, including fully loaded conditions and maximum traction and braking forces.

The static analysis revealed crucial insights into the frame's stress and deformation characteristics. Under fully loaded conditions, the maximum deformation observed was 1.1166 mm, and the maximum stress was 142.44 MPa. These values, while higher than other regions of the frame, remain within acceptable limits relative to the material's strength, indicating that the frame is adequately designed to handle the loads it encounters during operation.

In scenarios involving maximum traction force, the frame experienced a maximum deformation of 1.0078 mm and a stress of 139.96 MPa. Similarly, these results demonstrate that the frame's design meets the necessary strength requirements while accommodating the significant forces exerted during acceleration. Under maximum braking force conditions, the frame's deformation reached 1.0875 mm, and the maximum stress was 143.44 MPa, both of which are still within the material's capacity, ensuring safety and reliability.

Overall, the finite element analysis confirms that the truck frame's design is robust and capable of sustaining the stresses and deformations encountered in real-world conditions. The study underscores the importance of using advanced simulation tools to validate structural integrity and optimize design for safety and performance.

References

[1] Wu Yinlong, structural parametric lightweight optimization design of car front subframe [D]. Jilin University, 2014.

[2] Sheng Jian, Dai Zuoqiang, Zhang Jipeng, et al. Optimization design of pure electric bus frame based on ANSYS Workbench [J]. Journal of Qingdao University, engineering technology edition, 2015.

[3] Wang J R, Chou H C, Chen D W, et al.Dynamic reacction analysis of delivery truck hitting rigid wall directly at different speed[J].Chung Cheng Ling Hsueh Pao/journal of Chung Cheng Institute of Technology,2010,39(1):13-22.

[4] Yu Zhixin, Cao Quande, Yang Shitong. Finite element analysis and size optimization of a truck frame [J]. Mechanical engineer, 2018 (01): 18-20.

[5] Dong Xiaolei, Li Dongmei. Finite element analysis of automatic delivery robot frame based on ANSYS Workbench [J]. Mechanical engineering and automation, 2019 (06): 52-53 + 56.

[6] Chen Yang. Finite element analysis of kd3900 dump truck frame based on ANSYS Workbench [J]. Equipment manufacturing technology, 2016 (09): 35-37 + 45.

[7] Hu Ruixue, Zhang Yuquan, Zhao Zhanliang, et al. Finite element analysis of side beam frame based on ANSYS Workbench [J]. Internal combustion engine and accessories, 2020 (18): 22-23.

[8] Zhi Shuya, Xu Mutian, Li Jiqiu. Finite element analysis of frame based on ANSYS [J]. Journal of Jinling University of science and technology, 2019,35 (02): 1-5.

[9] Zhou Mengmeng. Finite element simulation and optimization design of light truck frame [D]. Jiangsu University, 2020.

[10] Han Yang. Finite element analysis and optimization design of light truck frame [D]. Dalian Jiao Tong University, 2019.

[11] Liu Bingxiao. Finite element analysis of truck frame based on ANSYS Workbench [J]. Automotive practical technology, 2017 (12): 157-159.