Application of finite element method in mechanical design of automotive parts

Suohai Gu
Armour College of Engineering, Illinois Institute of Technology, Chicago, IL 60616, USA

Abstract. As an effective numerical analysis method, finite element method (FEM) has been widely used in mechanical design and other fields. In this paper, the development of FEM is introduced firstly, then the specific steps of FEM applications are illustrated and the difficulties of FEM are summarized in detail. Finally, applications of FEM in automobile components such as automobile wheel, steel plate spring, body frame, shaft parts and so on are summarized, compared with related research experiments.

1. Introduction

With the continuous progress of science and technology and the rapid development of social economy, automobiles have become one of the most commonly used and most important transportation in daily life. The automobile industry has also become a pillar industry of national economy. However, automotive safety performance and energy saving problems have become increasingly prominent with the continuous growth of car ownership. It is a universal concern around the world. The design and manufacture of automobile parts has a direct impact on the safety performance of automobile vehicles. In the premise of ensuring the safety, the reduction of the automotive structure weight and energy consumption and the improvement of structural safety in accidents are key issues of automotive industry.

The cooperative operation of each vehicle component is a necessary condition to ensure the security. It can be seen from the collision test data that the safety performance of the vehicle body is important as the main force structure. How to use efficient and reasonable design methods to reduce manufacturing costs and weight, but also with the right material to ensure the body structure stiffness, strength and stability is the biggest problem that automotive manufacturers should solve now. The parts of household automobiles can reach more than ten thousand pieces, and most of them are non-standard parts. Structures, materials, functions of different part are not the same. As a result, the traditional structural design methods not only cause a large amount of work, but also the unreasonable structural margin design due to larger calculation error. Traditional methods can not effectively improve the structure safety, as well as the incensement of production costs and structural weights.

FEM have been widely used in the mechanical design of automobile parts, such as the lightweight design of automotive frame and the analysis of frame and body vibration characteristics, which can effectively solve the problem of the whole deformation and stress distribution of complex parts 0. In addition, FEM also provides a strong support for the strength and stiffness analysis design of the transmission shell, suspension systems, brake systems, wheels and other auto parts. FEM is used to analyze stress condition under different working conditions, which effectively improves the design efficiency and reduces the calculation error. Automobiles are composed of three parts: body, chassis
and engine. The body is involved in the material, manufacturing process, modeling and so on. It is the most important and the most complicated part, and its strength and stiffness are closely related to safety performance. FEM can introduce the virtual simulation into traditional design methods, greatly reduces the test cost and shortens the manufacturing period.

In Section 2, we briefly introduce the development process of FEM. Section 3 illustrates the basic steps of FEM and the difficulties at the present stage. Section 4 will summarize and analyze the application of FEM in the auto parts.

2. Overview of FEM

2.1 Development history of FEM

In 1941, A. Hrennikoff proposed elastic mechanics problems can be solved by the framework method, known as the discrete element method, but it was limited to the bar structure to construct the discrete model. In 1943, Richard Courant first tried to solve the St.Venant torsion problem by combining the piecewise continuous function and the minimum bit-energy principle defined on the triangular region. In the middle years of the 20th century, Boeing Company used three-node triangular unit, made the matrix displacement method applied to the plane problem for the first time. In the early 1960s, Clough proposed the concept of finite element, but because there was no computer at the time, it was not taken seriously by all walks of life.

With the birth and rapid development of the computer technology, FEM began to apply to all walks of life, including machinery manufacturing, aerospace, automobile, ship, electronic, Engineering and so on in the early 1970s, from the relatively simple two-dimensional problem to the development of complex 3D problem, from the static problem extended to the dynamic problem, from the linear problem to the nonlinear problem. Because FEM has the advantage of simulating the structure of vehicle components and the analysis of the impact environment in the test stage, it can provide reasonable optimization plan in the design stage to meet all the needs. So FEM occupied the dominance of manufacturing design. Currently the popular FEM software is listed as follows: NASTRAN, ADINA, ANSYS, ABAQUS, MARC, COSMOS and so on.

2.2 Basic steps of FEM

2.2.1 Grid quantities. The number of grids directly affects the computational accuracy and the size of the computational workload. Increasing the number of grids can improve the accuracy, but also increasing the computation load. When the grid number is small, increasing the number of grids will greatly improve the accuracy without significantly increasing the amount of computation.

However, when the number of grids increases to a certain extent, increasing the number of grids will not get a great sense of the upgrade for the final calculation accuracy, but will make a substantial increase in computing volume. Therefore, accuracy and computation amount should be both taken into account when determining the number of grids.
2.2.2 Unit type. How to choose the correct unit type is closely related to the results accuracy. Rod unit can only bear the pulling force or pressure in the direction of the rod, but cannot bear bending moment. Beam unit can withstand both tensile and bending moment. The shell element is the most suitable for thin-walled structures, which can reduce the computational complexity to a certain extent. The solid elements have more calculation, including the common hexahedral elements, which are often used for simple structural analysis. For complex structures, using a tetrahedral unit with an intermediate node is a good choice.

2.2.3 Meshing. The solution domain is approximated as a discrete domain consisting of finite units of different sizes, also known as meshing. How to disperse the model into the appropriate number of cells in this process is a critical and difficult problem. Smaller grids correspond to higher accuracy, but calculation amount will increase. In the case of stress concentration, that is, when the data change ladder is larger, we should use denser grid to simulate the data change law. On the other hand, if the gradient of change is small, we should use more evacuation network to reduce the workload.

2.2.4 Unit order. The units are divided into linear, quadratic, and cubic forms, where the quadratic and cubic units are called higher order units. High-order form of the unit has a better precision advantage. When analyzing a more complex or non-uniform stress distribution structure, we usually use high-order unit, because its surface or surface boundary can fit the analyzed structure’s surface or surface boundary. However, the corresponding use of high-order unit will cause the incensement of the number of grids and nodes dramatically. So the unit order should be balanced to consider the accuracy and time consumption.

Because increase the number of grids or increase the unit order can effectively improve the accuracy of the calculation, so in order to optimize the program, we will use different unit order in the unified structure. If structure is complex or stress is concentrated, we will choose high order unit, if the low precision of results is required, we will choose low order unit 0.

2.2.5 Unit shape. The element shape of the grid plays a very important role in the calculation results, and unreasonable shapes can lead to poor accuracy or even termination.

3. Applications of FEM in the design of automobile parts

3.1 Wheels
Wheels are important parts of automobiles, and its strength is critical to safety. During the moving of the car, wheels do circular motion, so it not only take the radial load of the vehicle, but also take the bending load. Fatigue failure is the main failure form of wheels 0. The general design method is to analyze the force of the wheel by FEM. After determining the load and establishing the model, the force division situation and dynamic response are obtained, the extreme position of the failure position and dangerous point can be determined. As the most important element in driving time, wheels have to be strong enough to run on the rough roads during the cycle time. At the same time, we need to minimize the cost to meet the profit. As rotating, wheels will get two different impacts from radial and bending 0.

3.1.1 Radial load fatigue analysis. Radial load fatigue analysis refers to the radial load of the wheel under normal operating conditions, and then it is required to carry out the rotation test. It is required to maintain the standard air pressure after the wheel has undergone some cyclic fatigue test.

Many scholars have also done lot researches of the radial load of FEM, through different loading range or load function. However, in order to ensure the accuracy of the results, the tire inflation pressure on the wheel structure stress distribution cannot be ignored 0. The results show that the fatigue life of aluminum wheels is closer to the experimental results than other traditional methods by using the local stress-strain method combined with the Simth-Waston-Topper damage formula.
3.1.2 Bending fatigue analysis. The bending moment of wheels is caused by the mounting plate and the loading shaft attached to the wheel from bolts. Therefore, in order to simulate the actual situation, installation disk and other components should be installed in the wheel bending fatigue tests. In the bending fatigue test conditions, we also need to consider impacts of two forces on the wheel structure 0.

1) Bolt preload. First, find the node displacement, and then determine the unit strain. In this force, the local stress concentration of the bolt hole and nut on the contact surface beyond the elastic range, but does not affect the structural displacement, so the data is reliable. The final results show that the influence of the bolting force is negligible in the limited area outside the bolt hole position.

2) Rotational centrifugal force When the wheel rotates at high speed, it will produce centrifugal force. Calculated at 1700rpm as the load boundary condition, the stress distribution is more uniform, the wheel rim is slightly stressed and the spoke stress is the smallest.

3.2 Leaf spring
For the suspension system on cars, the most important part is spring, it is used for minimize the vibrations and impacts due to road irregularities and also create a comfortable ride. Rigidity is the most important parameter in springs 0. In the traditional calculation method, we generally assume that spring is straight, and use linear principles for the strength calculation. Since this calculation is based on the assumptions, so the results vary widely and the accuracy is low.

1) The common curvature method, proposed by Palhilovsky. It assumes that the leaf springs are loaded with the same curvature in any section and the entire leaf spring as a variable cross-section beam. The stiffness is obtained by the length of the blade, the reciprocal of the moment of inertia, the modulus of elasticity, the deflection and the load.

2) The concentrated loading method assumes that the blades of the leaf spring are only in contact with each other at the ends and that the deflections of the two adjacent blades at the contact point are equal. By the deflection of the system to solve the force, known to be the first piece of the load obtained by the end of the deflection, and then find the spring stiffness.

It is a really complex between the connected sheets of the spring 0, when the load changes, the contact situation changes, because the friction can change its mechanical properties, so how accurate and effective simulation of friction is very important. FEM not only analyzes the stiffness characteristics of the leaf spring, but also can analyze the impact of friction, improve the final calculation accuracy.

3.3 Body and frame
Frame as a vehicle skeleton, has an important impact for vehicle, not only requires a sufficient stiffness to ensure that the collision will not make too much damage deformation to hurt the car staff, and need to meet the conditions in the driving will not appear fatigue damage. Structure analysis and dynamic analysis of the frame can be carried out by FEM, and the optimization design of the strength and the weight cause by different materials can be compared, so as to shorten the design and development cycle and improve the efficiency.
3.3.1 *Horizontal bending conditions.* This situation is mainly considered the full-load vehicle with a uniform speed in the horizontal pavement road $0$. The results show that the maximum displacement occurs in the rear section. The front and rear suspension is constraint and connected with frame mechanism, the vehicle force and torque are transmitted from here, so the connection area must be high-bending and high-twisting. This displacement situation is decided by the vehicle structures of their own characteristics.

3.3.2 *Emergency braking conditions.* This condition takes into account the impact of the ground breaking force on the frame with maximum braking acceleration of $-0.7g$. As the air suspension can only withstand the vertical force, longitudinal force and lateral force will be transmitted through the rod mechanism to the frame, it must ensure that the strength of the rod mechanism $0$. The results show that the stress level of the vehicle frame is still low when the emergency braking is carried out. Due to the influence of the longitudinal acceleration, the frame will have a little bit leaning forward. The front suspension has a large stress due to the large part of the load. The rest parts’ stress is small and can be neglected.

3.3.3 *Emergency turning conditions.* This condition takes into account that the vehicle lateral inertial forces with a centripetal acceleration of $0.4g$. The rest parameters are similar to the emergency brake condition. The results show that more serious roll occurred, due to the larger body quality and sharp turn by the centrifugal force, and the maximum displacement occurs in the roof cover. Due to the impact of lateral force, the vehicle prone to flick phenomenon, rear suspension torque moment are significantly increased, resulting in rear section deformation increases, in line with the actual situation.

3.3.4 *Extreme bending and twisting conditions.* Extreme bending and twisting conditions is the most dangerous condition for vehicle in the daily life, mainly to consider the vehicle driving in the bump road, even a side of the vehicle’s wheel off ground. Practice shows that most of vehicle is at a low speed under this condition, this time the dynamic change is very slow, body torsion characteristics can be seen as static.

The results show that the great stress of left front suspension part of the vehicle chassis due to vacant right wheel, the maximum displacement occurs at the left front cover, which are in line with the actual situation.

3.4 *Analysis of shaft parts*

The car axle is divided into drive axle and slave bridge $0$. The front beam in the steering axle takes the main bearing role, also known as the front axle. During the travel of the vehicle, the front axle needs to bear both the vertical load and the side slip torque. And most of steering bridges with non-independent suspension are I-structure; most of steering bridges with independent suspension are shell structure. Static and dynamic analysis of the front axle can be carried out by FEM to simulate the deformation and stress in the working conditions, and also the defects are found and corrected at the design stage.
3.4.1 **Vertical static load condition.** This situation mainly analyzes the force of the vehicle front axle at the horizontal road surface. Results show that the maximum deformation of the front axle occurred in the front axle under the swing arm under the hinge hole and the front fender welding area. The maximum stress occurs at the bottom of the front axle side of the coil spring sleeve and the front axle body is welded.

3.4.2 **Emergency braking conditions.** In this condition, front axle not only takes the vertical load but also take a braking force from wheel forward direction. As the vertical load and horizontal braking force happened at the same time, it will produce bending deformation, by the bending effect. The result shows that the maximum deformation occurs at the hinge hole of the lower arm of the front axle, and the maximum stress occurs at the fixed area between the front axle and the frame.

3.4.3 **Slip condition.** When the vehicle is turning, if the speed is too fast, centrifugal force is obvious, once it is more than the adhesion force, the vehicle will slide in the direction of axis. The result of finite element analysis shows that the stress is mainly concentrated in the shaft diameter of the front axle, because not only bear the body load, but also bear the role of sliding moment. And the maximum displacement occurs at the left end of the front fender, consistent with the actual situation, the final result to meet the first fourth strength theory.

4. **Conclusions**

Automobiles as one of the most common transportation in the world, the quality of automotive parts is directly related to the safety of life and property of drivers and passengers. This paper integrates domestic engineering technicians and scholars about FEM of automotive parts. The conclusions are listed as follows:

- Fatigue damage is the main failure mode of wheels, so it is critical to take the radial load fatigue test and bending fatigue test. Local stress-strain method combined with Smith-Watson-Topper damage formula is more accurate than other traditional methods. In the bending fatigue test conditions, the bolt preload and rotational centrifugal force should be considered for the wheel structure.

- By comparing the traditional method, the common curvature method and the concentrated load method with FEM, FEM can not only analyze the strength and stiffness characteristics, but also can take into account the actual use of friction in the process of analysis, further improving the accuracy.

- For the body frame, this paper discusses the finite element analysis under four different working conditions, which are horizontal bending, emergency braking, emergency turning, ultimate bending and tearing. The results show that the maximum stress and the maximum displacement are within the allowable range of the actual situation.

- Shaft parts mainly analyze the front axle of the car, also divided into three different conditions, vertical static load, emergency braking, and sliding conditions.

**References**

[1] Zhu Yijing. Review of applications of finite element method in automobile design[C]. Automotive Engineering Technology Seminar of Henan Province, the Proceedings of the 9th Henan Automobile Engineering Technology Seminar, 2012:3.

[2] Gu Chengzhong, Wu Xinyue. A review of FEM and trend of development [J]. Journal of Frontiers of Computer Science and Technology, 2008, 2(3): 248- 259.

[3] Du Ping’an. Basic principles of finite element meshing [J]. Machinery Design & Manufacture, 2000(1):34-36.

[4] Zhong Cuixia. Design and structural analysis of aluminum alloy wheel [D]. Hangzhou: Zhejiang University, 2006.

[5] Chang C L, Yang S H. Simulation of wheel impact test using finite element method [J]. Engineering Failure Analysis, 2009, 16(5):1711-1719.
[6] Shokrieh M M, Rezaei D. Analysis and optimization of a composite leaf spring [J]. Composite Structures, 2003, 60(3):317-325.

[7] Zheng Yinhuan, Zhang Zhongfu. Nonlinear finite element analysis of the mechanics characteristic of a leaf spring [J]. Mechanical Science and Technology for Aerospace Engineering, 2008, 27(7):914-916.

[8] Li Chuanbo. Finite element modeling and analysis of static and dynamic characteristics for coach-body frame [D]. Hefei: Hefei University of Technology, 2010.

[9] Chen Yujie. Research on finite element modeling and optimization analysis for coach-body frame [D]. Hefei: Hefei University of Technology, 2007.

[10] Wang Zhangxiang. Static and dynamic strength analysis and test research on heavy-duty truck front axle [D]. Huaian: Anhui University of Science and Technology, 2016.

[11] Xiang lin. Finite element analysis about front-axle vehicle of steady and dynamic performances [D]. Nanning: Guangxi University, 2008.