Optimum Design of Central Console Structure for an All-Terrain Vehicle Based on Topography Optimization

Yajuan Chen¹*, Li Gu², Xianglin Li³

¹Institute of Artificial Intelligence Chongqing Real Estate College, ChongQing, 401331, China
²Product R & D department Chongqing Jialing Global Terrain Vehicle Co., Ltd. ChongQing, 402760, China
³Institute of Artificial Intelligence Chongqing Real Estate College, ChongQing, 401331, China

*chencaihua@cqcf.com
*365379562@qq.com

Abstract—The mechanical properties of plastics are obtained through material tests. The strength characteristics of the plastic middle console structure of a terrain vehicle under the vibration loads on the pavement are simulated by using the finite element analysis theory. On this basis, the morphology optimization technology is used to improve the structure. Meanwhile, the model is redesigned and verified according to the optimization results, which solves the problem of insufficient strength of the original structure. It provides an effective reference for the design of plastic structural parts of similar vehicles.

1. INTRODUCTION

As an integral part of the terrain vehicle, the plastic center console not only connects the left and right seats and carries a certain amount of weight, but also relates to the appearance of the vehicle, product quality and reliability. At present, the design of the plastic center console structure is mainly based on the designer's engineering experience and the design scheme of other relevant models. This traditional design method is blind to some extent. It can't give full consideration to the optimization of structural strength, weight, process feasibility and other indicators. According to the structural characteristics of the product, the combination of shape optimization and finite element simulation is introduced in the research and development stage, which can reach the optimal performance under the relevant constraints, and help the engineer to find the optimal structural design scheme quickly and accurately under the given design space. Thus, it can shorten the development cycle and reduce the test times of single product and whole vehicle.

The plastic center console structure of an all-terrain vehicle is taken as the research object. The finite element technology is used to simplify the structure and to discretize the element. The mechanical characteristic parameters of the common plastic materials of the structure and the vibration load of the road surface are obtained through the test. The stress distribution of the structure under the excitation load is simulated to find out the weak area of the structure. On this basis, the shape optimization technology is used to improve the structure and find out the stiffener layout trend of the center console. The optimization results provide a clear idea for designers to design safe and reliable products. At the
same time, the new structure is verified to meet the strength requirements.

2. MATERIAL TEST
PP-GN9 and PA6-PPE are commonly used in the central control console. The mechanical properties of plastic parts are unstable during the processing. The mechanical properties of different parts in different batches or different parts in the same batch are different. The mechanical properties of materials are one of the important input parameters and result evaluation criteria in the whole finite element analysis. So, it is necessary to test the mechanical properties of two commonly used materials. According to GB/T1040.2 and GB/T9341, the tensile strength, elongation at break, bending strength and bending module of the material are tested [1-2]. The test results are shown in Table 1. It can be seen from the table that the mechanical properties of PA6-PPE are better than that of PP-GN9. PP-GN9 is used in the original structure design.

| testing content       | unit | measured data |
|-----------------------|------|---------------|
| tensile strength      | MPa  | 60            |
| elongation at break   | %    | 37            |
| bending strength      | MPa  | 63            |
| bending module        | MPa  | 1870          |
|                       |      | 32            |
|                       |      | 1302          |

3. FINITE ELEMENT MODEL AND ANALYSIS OF CENTRE CONSOLE

3.1. Establish Finite Element Analysis Model
The object of this paper is the structure of the plastic center console connecting the left and right seats in the all-terrain vehicle, as shown in Figure 1. The structure is assembled into a whole with the surrounding parts through six mounting holes. The original design structure of the center console appears early fracture phenomenon in the road test of the sample vehicle. According to the analysis of the fracture, it is caused by the insufficient strength of the structure. So, it is necessary to analyze the strength characteristics of the structure in actual driving.

Because the center console is a plastic part with irregular structure, this paper mainly researches whether the strength of the structure meets the requirements of strength use. Therefore, the solid element tetra4 is used in the analysis for grid discretization. The finite element analysis model established is shown in Figure 2.
3.2. Load Boundary Conditions for Finite Element Calculation

After the finite element analysis model is established, the initial conditions of analysis, including material, element attribute and load need to be set to facilitate the later calculation and solution.

The main vibration sources of the all-terrain vehicle are the vibration caused by the uneven road surface and the vibration caused by the engine running. This paper analyzes the strength of the center console under the combined action of two vibration excitation sources. In order to obtain the vibration signal of the structure in actual driving, the road driving vibration test is carried out [3-4]. The main purpose is to obtain the vibration signal of the all-terrain vehicle under the action of the uneven excitation of the typical road surface and the engine excitation at the same time under the common speed. After the data processing, the vibration signal data is obtained, which provides the load boundary condition for the structural strength analysis. In the analysis, the driving direction of the vehicle is defined as x-axis, the vertical ground direction as z-axis, the left and right sides of the driver as y-axis. The vibration acceleration test results of the test point of the center console under the combined action of two vibration excitation sources are shown as figure 3, figure 4 and figure 5.
In this paper, the limit analysis method is used to analyze the strength characteristics of the structure under the maximum vibration load excitation. From the test data processing results, the maximum vibration acceleration of the structure in three directions is 14.6g, 10.5g and 10.1g respectively.

3.3. Strength Analysis Results of Original Structure

The stress calculation results of the original center console structure under the above conditions are shown in figure 6, figure 7 and figure 8.

From the analysis results, the maximum stress of the original center console structure in the X, Y and Z directions is 5MPa, 24.8Mpa and 5.5Mpa respectively. The stress under the vibration excitation in the Y direction is the largest. There is stress concentration phenomenon in the local part. The safety factor is low. And early fracture phenomenon will occur in the actual driving, which is the place where the sample vehicle breaks in the road test. Therefore, it is necessary to optimize the structure to meet the
use requirements.

4. SHAPE OPTIMIZATION OF CANTER CONSOLE

In the design process of the center console, due to the constraints of the assembly and installation position of the parts, it is impossible to simply determine the strengthening area and obtain the best strengthening layout plan.

Shape optimization is a kind of conceptual design method to find the optimal reinforcement distribution for thin-walled structural parts. It is a special shape optimization method. By specifying the movement of element nodes in its normal direction, the structure shape of the finite element mesh model is continuously adjusted until the optimal combination of the moving node areas meets the design objectives. This optimization method optimizes the design process, and provides effective reference for designers to design safe and reliable products according to the results [5-7].

In order to optimize the shape of the center console, it is necessary to determine the design space and non-design space of the structure. The design space is the part that can optimize the structure, that is, the area where the stiffeners can be arranged. The non-design space is the part that does not change the structure. In this paper, the fixed installation area is defined as non-design space, and the rest is design space. At the same time, PA6-PPE is used to optimize the structure.

On the basis of the optimization analysis and modeling of the center console, the mathematical model of the shape optimization is established, including design variables, objective functions and constraints. Here, the basic parameters including height and width of the stiffener are taken as the design variables. The minimum mass is taken as the objective function of the shape optimization. And the maximum stress of the structure under the vibration excitation is less than 20MPa as the constraint condition. After iterative calculation, the shape optimization results of the center console are shown in Figure 9.

According to the results of the shape optimization analysis of the center console, and combined with the manufacturing process factors, the layout of the stiffener structure is adjusted, and the center console model is redesigned. The new structure obtained is shown in Figure 10.

![Figure 9. Topography optimization results of central console](image1)

![Figure 10. Optimized structure of central console](image2)
5. **STRUCTURAL FEASIBILITY ANALYSIS AFTER OPTIMIZATION**

The simulation results of the mechanical characteristics of the optimized center console under three directions of vibration excitation are shown in figure 11, figure 12 and figure 13.

![Figure 11](image1.png)

**Figure 11. Stress distribution of center console under X-direction vibration excitation after optimization**

![Figure 12](image2.png)

**Figure 12. Stress distribution of center console under Y-direction vibration excitation after optimization**

![Figure 13](image3.png)

**Figure 13. Stress distribution of center console under Z-direction vibration excitation after optimization**

From the analysis results, the maximum stress of the optimized center console structure in X, Y and Z directions is 3.85mpa, 15.6mpa and 4.18mpa respectively. At the same time, PA6-PPE material is used in the optimized structure, which improves the safety factor of the structure. The structure is assembled to the whole vehicle for road test. It is verified that the structure can meet the strength requirements within the specified driving mileage.

6. **CONCLUSION**

In this paper, the theory of finite element analysis and the method of shape optimization are applied to the optimization design of the plastic structure center console. On the basis of the experiment of the mechanical properties of plastic materials, the strength characteristics of the component under the action of the road uneven excitation and the engine vibration excitation are simulated. Then the weak area of the plastic structure is found out. On this basis, the structure of the center console is optimized to find the optimal layout of the stiffeners, and the feasibility of the final optimization structure is verified. This
method can not only effectively improve the structural strength of the plastic center console components to meet the user's requirements for the static characteristics of the structure in actual use, but also optimize the design and development process of the plastic structure components to shorten the product development cycle.

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

REFERENCES
[1] Gao Cunsheng, Xu Zexi, Cao Zhiku. Discussion on the Application of Plastics in Automobile Industry. China Plastics Industry, 2012(8):7-9.
[2] Zhang Shiyou, Li Lihua, Lu Qingyun. Study on the Testing Technology of Small Samples for the Mechanical Properties of Plastics. China Plastics Industry, 1984(6):1-11.
[3] Yang Ping, Shi Xiaohui, Zou Xihong, Zhang Junjie. Sampling and Analyzing for Typical Road Loading Spectrum of Motorcycle. Machinery Design and Manufacture, 2011(11):193-195.
[4] Jiang Zhengfeng, (2007) Fundamentals of testing technology [M]. Peking University Press, Beijing.
[5] Zhou Chuanyue, Teng Wanxiu, Zhang Juntang,(2005) Engineering Finite Element and Optimization Analysis Application Case Course [M]. Science Press, Beijing..
[6] He Yanfei, Ding Jie. Cover and Lug Optimization of Auxiliary Converter Based on OptiStruct Software. Electric Drive for Locomotives, 2017(2):72-75.
[7] Cai Feng, Zhang Liangliang, Liu Jingyang. Application of Optimization Design in the Lightweight of Automotive Components. Automotive Technology, 2010(4):25-29.