Optimization and verification of fatigue life simulation analysis of the front hatch cover of a certain vehicle type

Ruoyu Zhao, Bin Wang

CATARC Automotive Test Center (Tianjin) Co.,Ltd, Tianjin 300000, China

Abstract

In view of the problem of sheet metal and welding point cracking in the fatigue durability of the front hatch cover, this paper takes the front hatch cover of a model as the research object in the development stage, establishes the open-closed durability simulation finite part model according to CAD geometry data, uses ABAQUS/DYNAMIC, EXPLICIT nonlinear display dynamics algorithm for simulation analysis, combined with Miner linear fatigue accumulation injury theory and EN fatigue analysis method. Using the Design-Life module in nCode software to predict fatigue life, the structural optimization of components is carried out according to the problem of substandard damage to welding joints, and the test verification of open-closed durable bench is carried out in the optimized and improved prototype. Switch fatigue durability simulation analysis and optimization of the front hatch cover can provide the basis for automobile design, shorten the development cycle and save costs.

Keywords

Front hatch cover; Fatigue durability; Simulation optimization; Test verification.

1. Introduction

The front hatch assembly is one of the important systems that make up the car. Due to improper design, poor use conditions, etc., long-term opening and closing may cause fatigue failure of the front hatch body sheet metal, solder joints and hinges. In the process of automobile design and development, the durability performance of automobile opening and closing parts has become an important indicator for evaluating the quality of automobiles[1].

Usually, the switch durability performance of automobile opening and closing parts is mainly evaluated through its switch durability test. However, the durability test is time-consuming and expensive, and if durability performance problems are found, it is necessary to optimize the design and change the sample, thus further increasing the number of automobiles. R&D cycle and cost. With the development of computer simulation technology, switch durability virtual simulation has become an important design method in the automobile development process. For example, Shen Jia et al. analyzed the switch impact strength of a certain car door based on LS-Dyna nonlinear display calculations, conducted fatigue analysis, and predicted the fatigue life danger area on this basis; Xing Zhiwei et al. [2] conducted a transient dynamics analysis on the rear door of a car, predicted the fatigue life on this basis, and conducted experimental verification; Feng Changkai et al. [3] used MSC.Fatigue software to conduct sheet metal analysis on an SUV door. Fatigue damage analysis was conducted and compared with the bench opening and closing durability test, and the weak locations were optimized.

This paper establishes a finite element simulation model based on the CAD data in the development of a certain vehicle model. According to the test-related specifications of the durability performance of the front hatch system, the boundary conditions and load input of the corresponding finite element simulation model are determined. The ABAQUS /Explicit solver is used to perform the front hatch design. The transient dynamics analysis of the cover impact process was carried out to obtain the energy time history during the impact. Based on the EN fatigue analysis method and Miner cumulative damage theory, nCode software was used to conduct fatigue analysis on the impact of the car's front hatch switch to predict the fatigue life danger area, and Carry out structural optimization design of dangerous areas to improve the durability performance of the front hatch, provide a basis for the early development and research of the car's front hatch, and conduct bench durability tests on later prototypes.

2. Front hatch slam analysis

2.1. Finite element model of front hatch slam analysis

2.1.1. Model interception

Part of the body model including the front hatch assembly can be intercepted, as shown in Figure 1. The simulation model includes components such as the front windshield, fenders, front-end modules, headlights, strip lights, headlight beams and front anti-collision beams.

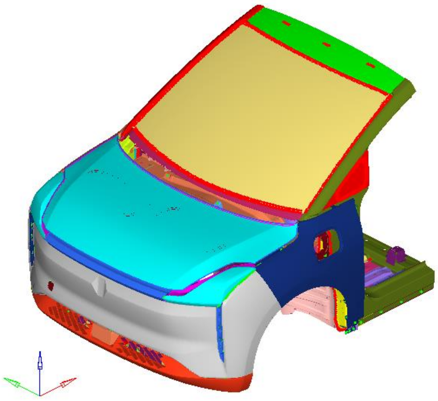

Fig. 1 Slam analysis finite element model

2.1.2. Buffer block simulation

The front hatch cover buffer block is simulated using a rigid body unit, and the rigid body reference point is set at the intersection of the rigid body unit and the buffer block axis.

The buffer block is simulated using the translator unit, and different stiffness values must be assigned to different buffer blocks. The stiffness curve of the buffer block is obtained from experimental testing. The stiffness curve of the main buffer block of the front hatch cover is shown in Figure 2.

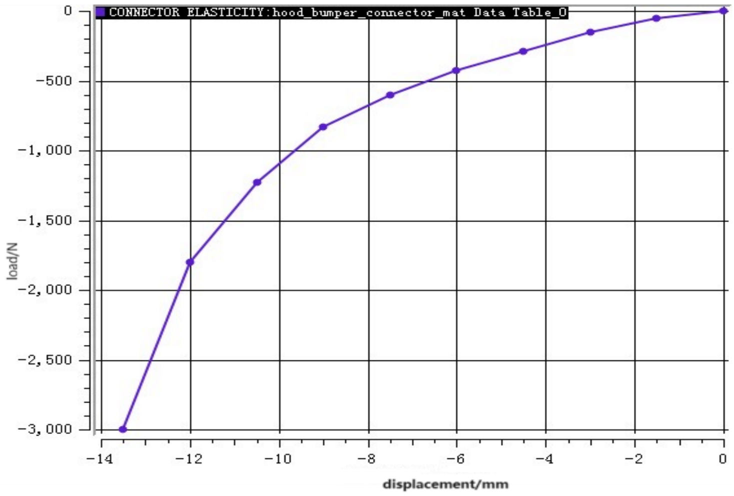

Fig. 2 Buffer block stiffness curve

Establish a local coordinate system at the buffer block position. The buffer block installation point position is the local coordinate origin. Define the buffer block axis direction as the x-axis of the local coordinate system to control its compression direction. Use CONN3D2 to connect the lower end point of the buffer block axis line and the rigid body reference point to establish The translator unit and buffer block model are shown in Figure 3.

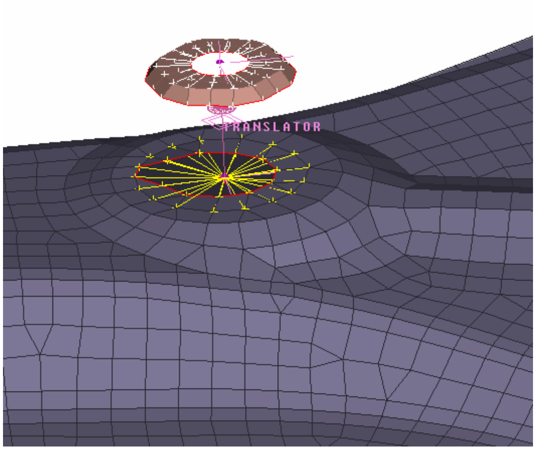

Fig. 3 Buffer block model

2.1.3. Lock mechanism simulation

The simplification of the lock mechanism is a key step in simulating the closing and impact process of the front hatch cover. The lock ring is simulated to fall and the lock mechanism is locked. The lock ring and stop pawl are simulated with solid units. The spring devices on the left and right sides determine the spring head and tail according to the spring geometric size. The elastic center is simulated by the SPRING unit, and the rotating pin is simulated by the HINGE unit to establish the contact relationship. Finally, a simplified model of the front hatch lock body structure is obtained, as shown in Figure 4.

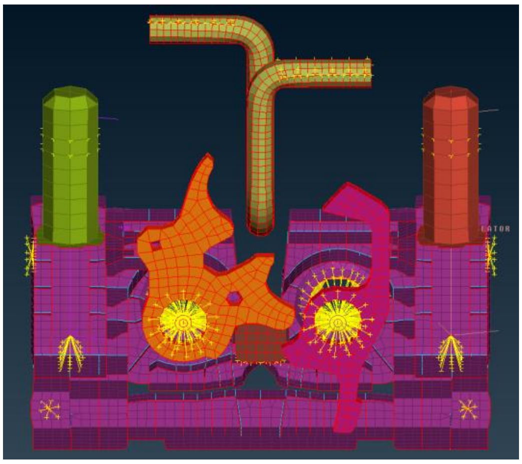

Fig. 4 Front hatch lock body model

2.2. Loads and boundary conditions

According to the relevant specifications of the durability performance test of the front hatch system switch, the opening angle is determined to be 1/3 of the maximum opening position. Rotate the front hatch cover to the opening angle with the hinge center point as the axis. Its closing angular speed is 2.58rad /s based on the conversion relationship between gravitational potential energy and rotational kinetic energy. Material nonlinearity and geometric nonlinearity are considered, and a gravity field is applied to the front hatch and hinges to constrain the body section surface to 123456 degrees of freedom.

2.3. Impact process simulation results

The front hatch is closed and impacted at a rotational angular velocity of 2.58 rad/s. The energy variation curve with time during the closing process is shown in Figure 5. It can be seen from

Figure 5 that the total energy during the impact process is conserved, and the changes in kinetic energy and internal energy are consistent with the actual physical process. The mechanism locks at about 23.4ms, with the smallest kinetic energy and the largest internal energy.

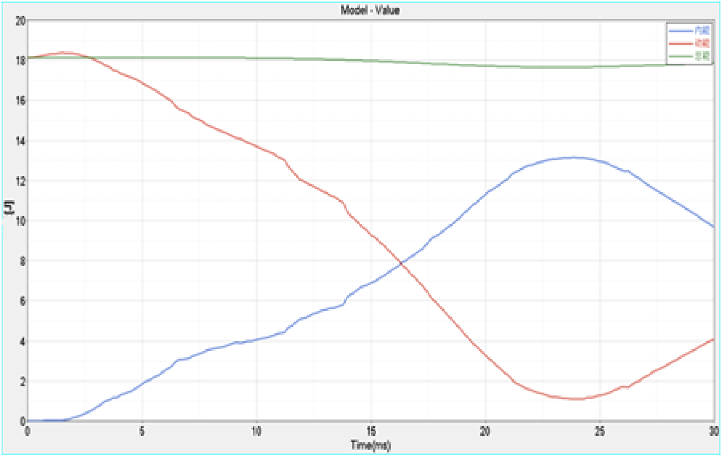

Fig. 5 Energy time history curve

The transient stress distribution was obtained from the slam simulation analysis results of the front hatch cover, and the maximum stress of the hinge was 170.9MPa, as shown in Figure 6; the maximum stress of the front hatch cover was 186.7MPa, located at the front end of the outer panel of the front hatch cover, as shown in Figure 7; the body The maximum stress is 180.4MPa, located at the left flume connecting plate, as shown in Figure 8-9, which does not exceed the yield strength of the material.

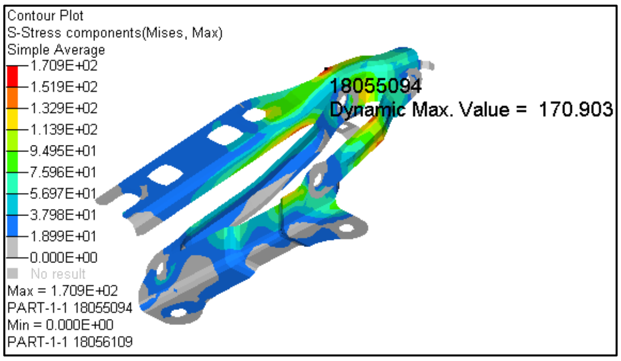

Fig. 6 Hinge stress distribution cloud chart

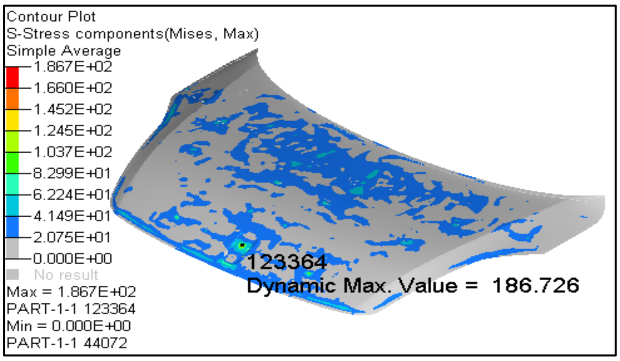

Fig. 7 Stress distribution cloud chart

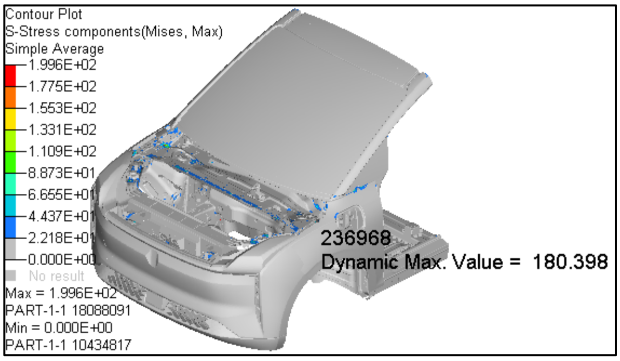

Fig. 8 Body stress distribution cloud chart

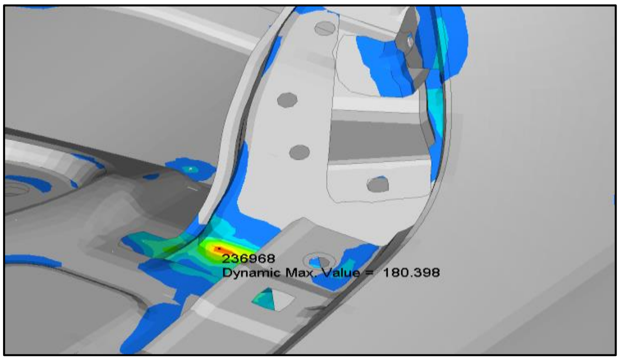

Fig. 9 Body stress distribution cloud chart

3. Fatigue life assessment of front hatch cover

3.1. Miner linear cumulative damage theory

In order to accurately estimate the fatigue damage of the front hatch during the opening and closing endurance test, Miner's linear cumulative damage theory is used. [4,5] This theory believes that the fatigue damage of materials is caused by the repeated action and accumulation of cyclic loads; the damage is proportional to the number of cycles. The energy absorbed when the fatigue damage accumulates to failure has nothing to do with the sequence of load application, and is only determined by the linear accumulation of damage. [6] Assume that the energy limit absorbed by the material before failure is E, the total number of cycles before failure is N, and the energy absorbed at a certain number of cycles is $E_{1.}$ The energy absorbed by the material is proportional to its cycle number $n_{1:}$

$$\frac{E_1}{E} = \frac{n_1}{N}$$

Then, when corresponding to N_i the stress cycle at the i-th stress level level, the cumulative fatigue damage of the material is:

$$D = \sum D_i = \sum \frac{n_i}{N_i}$$

In the formula: n_i is the number of stress level S_i cycles; N_i is S_i the fatigue life of the structure under the stress level. Failure occurs when the cumulative damage reaches 1.

3.2. nCode fatigue analysis process

Based on slam dynamic analysis, the finite element model.inp file used for stress analysis and the stress analysis results.odb file are imported into nCode to perform fatigue calculations on sheet metal and solder joints. The calculation flow chart are shown in Figure 10 and Figure 11 .

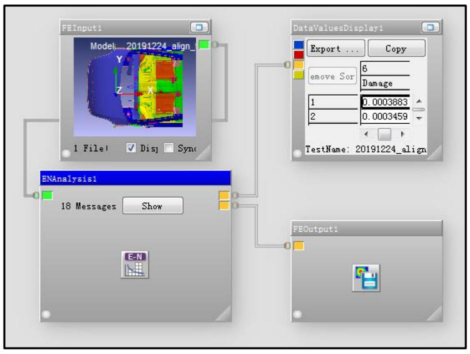

Fig. 10 Sheet metal fatigue analysis flow chart

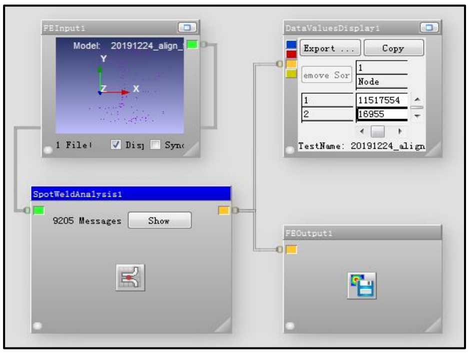

Fig. 11 Solder joint fatigue analysis flow chart

Fatigue failure types can usually be divided into low-cycle fatigue and high-cycle fatigue. The impact fatigue of the front hatch switch is mainly low-cycle fatigue failure. In the sheet metal fatigue simulation calculation, it is necessary to input the EN curve of the material of the automobile component to be investigated. The EN curve of a low carbon steel used in the front hatch cover is shown in Figure 12 below.

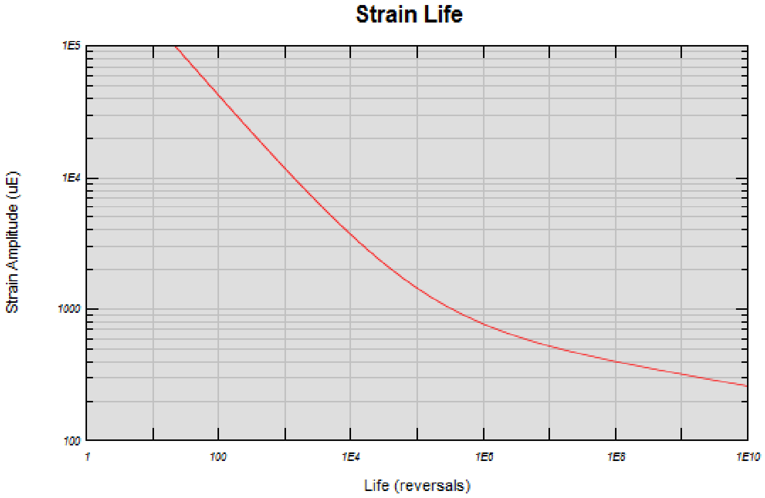

Fig. 12 EN curve of a certain low carbon steel

According to the relevant specifications of the durability performance test of the front hatch system, the number of switch cycles is determined to be 4500 times. The fatigue damage results of the front hatch sheet metal and solder joint switches calculated by nCode are imported into Hyperview for viewing, as shown in Figure 13 and Figure 14 below.

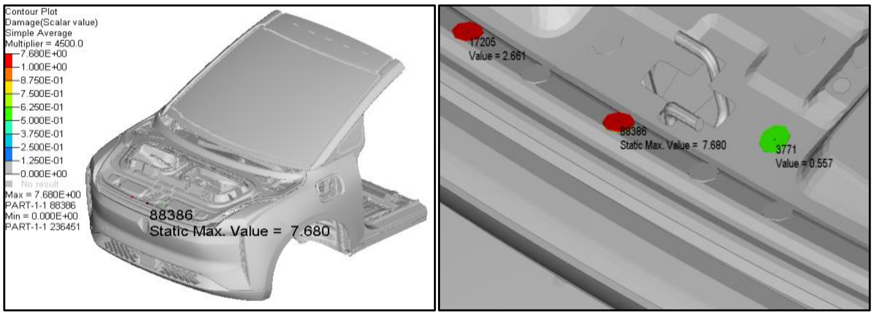

Fig. 13 Solder joint fatigue damage cloud diagram

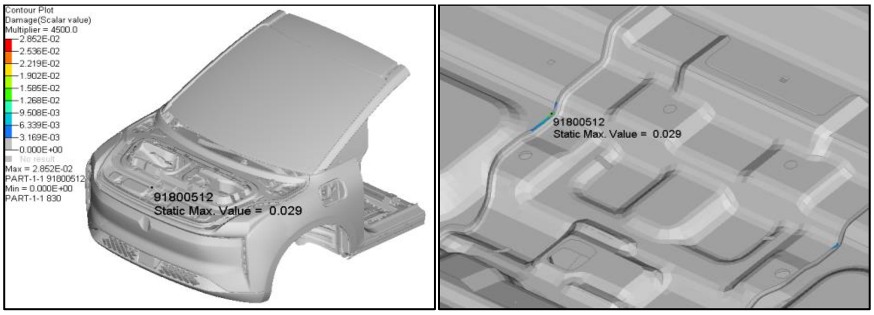

Fig. 14 Sheet metal fatigue damage cloud diagram

It can be seen from the results in the above figure that after 4500 switching cycles, the maximum fatigue damage value of the front hatch welding point is 7.680. The location is the overlap between the outer plate support plate and the inner plate at the front end of the front hatch, which does not meet the requirements. The maximum value of sheet metal damage is 0.029, and its location is the flange of the front hatch lock ring mounting plate. The result is less than 1, which meets the requirements.

3.3. Fatigue optimization of front hatch cover solder joints

Since the damage value of the front hatch cover solder joints is greater than 1, the positions of the solder joints that do not meet the requirements need to be optimized. At the overlap between the front hatch cover outer panel support plate and the inner panel, the original support plate connection surface is enlarged and a welding point connection is added for improvement, as shown in Figure 15 below.

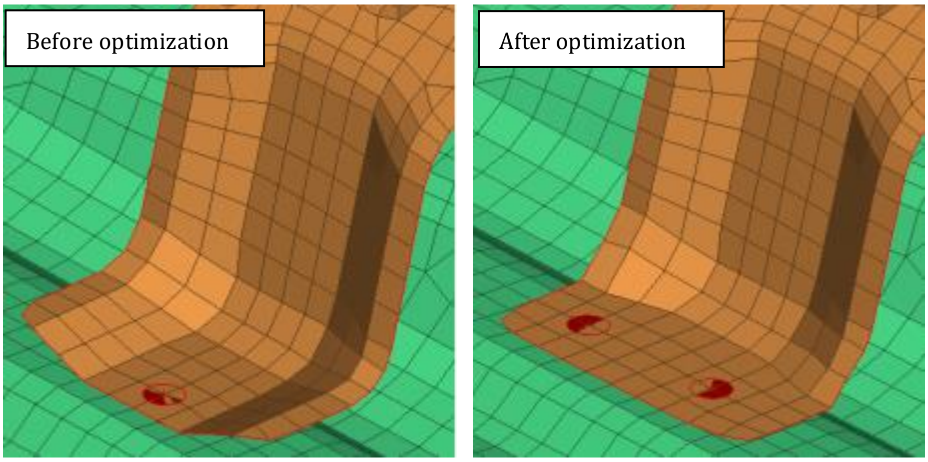

Fig. 15 Support plate connection

Submit the optimized model to calculate the maximum fatigue damage result of the solder joint, which is 0.394. The optimization result is qualified, as shown in Figure 16 :

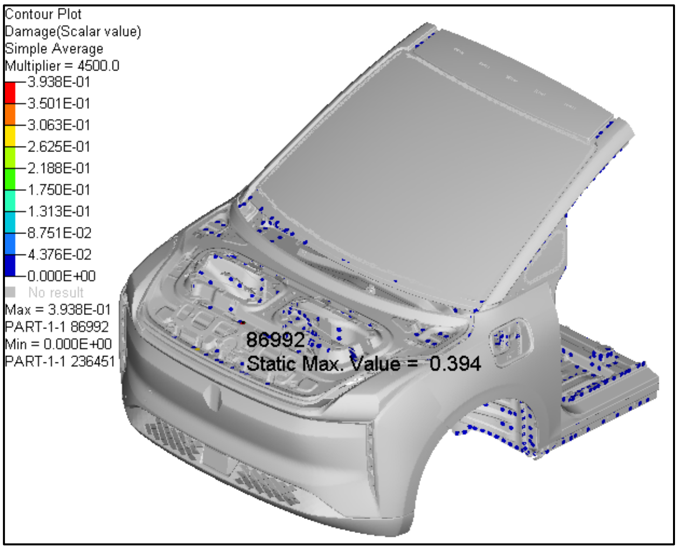

Fig. 16 Solder joint damage distribution cloud chart

The statistics of fatigue durability damage results of the front hatch switch are shown in Table 1 :

Table 1 Fatigue damage results of front hatch cover

Model status	Part	Maximum damage value	
		Damage value	Target value
Original plan	sheet metal	0.029	<1
	solder joint	7.680	
Optimization	sheet metal	0.058	
	solder joint	0.394	

4. Front hatch switch durability test verification

Suggestions were made for designers to change the partial structure of the front hatch, and the fatigue durability of the front hatch switch was tested on the improved prototype in the subsequent stages. Bench test, the bench test opens and closes 4500 times. The test process is strictly monitored. After the test is completed, carefully check whether there are any cracks in the sheet metal parts and solder joints. If there are any, it is determined that the durability of the front hatch switch is not good. If there is no failure phenomenon, the front hatch structure optimization is qualified.

It was verified by the switch fatigue durability bench test that after 4,500 switching cycles, there was no cracking in the sheet metal and solder joints of the body and front hatch, and all components were in good condition. No abnormalities occurred during the test, which met the design objectives.

5. Conclusion

This article takes the front hatch of a certain model in the development stage as the research object, establishes the front hatch slam transient dynamics finite element model, and imports the stress simulation results into nCode for fatigue analysis. Through the entire simulation process, the following conclusions are drawn:

Use the ABAQUS/ EXPLICIT display dynamics algorithm to perform a transient dynamics analysis on the car's front hatch, and obtain the transient stress distribution during the closing process and the maximum stress values on the hinges, front hatch and body. Pay attention to these high stresses Whether the location of the hazardous area will cause fatigue failure.

Based on the Miner cumulative damage theory and the EN fatigue analysis method, the opening and closing durability fatigue analysis of the front hatch cover was carried out. The result was that the sheet metal damage value was less than 1, and the result was qualified; the solder joint damage value was greater than 1, which did not meet the requirements. Structural optimization and improvement were carried out to target the unqualified solder joints. After 4500 opening and closing cycles of the optimized front hatch, the maximum damage value was 0.394, which met the requirements.

In order to verify the fatigue durability performance of the optimized and improved front hatch cover, a switch fatigue durability bench test was conducted on the prototype vehicle according to the relevant test specifications. During the test, no cracking of the sheet metal or solder joints was found. The results showed that the front hatch cover The fatigue durability performance of the switch is qualified.

The method applied in this article can accurately identify problems and solve them quickly during the product design stage. It can effectively guide structural design, improve design quality, shorten the development cycle, and save development costs. It has certain engineering practical value. At the same time, this method also Suitable for switch slam fatigue analysis of front and rear doors and rear doors.

References

[1] Deng Saibang, Tan Dongsheng, Liu Xiangzheng, Yuan Huanquan. Car door fatigue analysis and optimization based on finite element theory. Modern Manufacturing Engineering, (2019) NO.6, p.50-54.

[2] Xing Zhiwei, Hui Yanbo, Feng Lanfang. Based on MSC. Fatigue analysis and optimization of door fatigue of a microbus. Mechanical Research and Application, (2013) No.4, p.61-63.

[3] Feng Changkai, Wang Jun, Sheng Shouzeng. Simulation analysis of vehicle door opening and closing durability based on finite element. Auto Parts, (2016) No.5, p.54-56.

[4] Wang Wenwei, Cheng Yuting, Jiang Weiyuan, et al. Random vibration fatigue analysis of electric vehicle battery box structure. Transactions of the Chinese Society of Automotive Engineering, Vol. 6 (2019), No.1, p.10-14.

[5] MINER M A. Cumulative Damage in Fatigue. Journal of Applied Mechanics Review, (1945) No.3, p.159-164. (in Chinese).

[6] Xu Chengmin, Lian Zhibin, Li Tianbing. Research on fatigue optimization of exhaust pipe bracket considering welding nut. Journal of Automotive Engineering, Vol. 9 (2019), No.1, p.46-48.