Analysis and Optimization of automotive seat structure based on finite element method

GengDongliang1*, GaoFengling1, BuXiaobing1, YuBayi2, YanTang1, WuYuan1.
1. Collision test research department, CATARC Automotive Test Center (Tianjin) Co., Ltd, Tian Jin, China
2. Factory inspection department, China Quality Certification Center, Bei Jing, China
* Corresponding author: gengdongliang@catarc.ac.cn

Abstract—According to the rigid design standard of automotive seat frame structure, the strength and stiffness of a vehicle seat is analyzed and optimized in this paper. The finite element analysis results show that the initial design of the seat can basically meet the stiffness requirements under most standard test conditions. However, in the front crash test condition, there is a risk of strength failure in the seat rail. Therefore, it is necessary to optimize the structure. Based on the engineering experience, local characteristics of the seat rail is modified. After conducting simulation of the improved seat structure, it is demonstrated that the seat meets all the design requirements of stiffness and strength.

1. INTRODUCTION
As the most direct component connecting people and vehicle body, the automotive seat plays an important role in the safety, comfort and driving experience of passengers. Among all the performance, ensuring the safety of drivers and passengers is the most basic and important for seats. High performance seat, in parallel with the airbag, has more ideal protection effect. Comfort and good driving experience of seats are meaningful only when passenger safety is ensured. Some countries in Europe, America and Japan have published their own seat standards since the longtime research and development of seat products. At present, Chinese design standards for automotive mostly are referred to some European standards. By modification, they have formed national standards that meet the safety conditions of domestic passengers. In the early years, most of the research and development process of seats are based on the experience of engineers to form the first version of seat products. After tests, whether the seats meet the standards are verified. This kind of development method has a long period of development, and the production cost of the first version of the seats for test is higher. To deal with these defaults, CAE analysis technology can be introduced. In the model design stage, the seat structure is checked. If the calculation results meet the requirements, the test products are produced. Therefore, some test risks can be avoided and the R & D cycle can be shortened[1]. Through the experimental analysis, some damage or unqualified places on the outside of the seat mechanism are shown, but some potential risk points can not be found. Through CAE analysis, the strain and stress of the whole structure can be observed, and the high strain risk area is clear[2].

The safety analysis of seat structure includes active safety and passive safety. Active safety focuses on strengthening the vehicle personnel to predict the dangerous situation in advance and respond to the danger signal in time for the driver. Passive safety focuses on the stiffness and strength of the seat
structure, as well as the performance of the adjustment device. This paper focuses on the passive safety of the seat structure, according to the requirements of industry standards and regulations, the stiffness and strength of the front seat frame structure are optimized.

2. MODEL DESCRIPTION

2.1. Seat geometry model features
As the weight of the vehicle affects the fuel consumption of the vehicle to a certain extent, and the seat as a part of the vehicle structure, the reduction of seat weight will also help the vehicle weight reduction. The traditional seat structure uses more sheet metal stamping structure, which results in heavier overall seat weight. At the same time, the stamping structure can be produced with the help of mould, and the R & D and production cost of automotive mould is high, and a large number of stamping structural parts will cause the rise of vehicle R & D cost. Therefore, according to the position and bearing capacity of each part, a large number of pipe structures are used in the seat frame structure. The steel pipe material is used here. If the weight does not meet the requirements, the steel pipe can be replaced with aluminum alloy material [3]. The mixed use of tube and sheet metal stamping parts brings enough space for the design and adjustment of the seat frame model.

The frame structure of the front seat is shown in Fig. 1. The whole frame structure is composed of tubular structure and sheet metal stamping parts. The seat frame is the main part of the whole seat weight. Through the actual measurement and comparison, the overall weight of the seat can be reduced to a certain extent through the application of pipe materials on the premise of meeting the structural stiffness and strength.

![Figure 1. Front seat frame model](image)

2.2. Establishment of finite element model
The shell element is used to divide the sheet metal structure and the large diameter thin-walled tube, and the beam element is used to simulate the small-diameter pipe and some spring components.

The shell element is selected as the modeling element of sheet metal structure and large diameter thin-walled pipe fittings from the calculation accuracy, calculation efficiency and the accuracy of geometric model discretization. In the shell element, the quadrilateral element is the main element, and a small number of triangle elements are used. The main reason for using quadrilateral element is that the integration characteristic of quadrilateral element is better than that of triangle element, and the calculation accuracy is higher, which can effectively ensure the accuracy of model calculation results. The existence of a small number of triangular elements is because the triangular elements can better discretize some special areas of the geometric model. The triangular element has better approximation
property to the transition position of complex surface than the quadrilateral element. However, the triangular element itself is a constant stress element, that is, no stress averaging is done inside the element. Therefore, the stress distortion often occurs where there are more triangular elements. In view of the shortcomings of triangular elements, the proportion of triangular elements must be strictly limited in the whole modeling process. In this paper, we take the proportion of triangular shell elements less than 5% as a criterion to evaluate the mesh quality. On the whole, the whole seat frame structure should be divided by quadrilateral shell elements in the structural parts which pay more attention to the stress-strain region.

Shell element, beam element and a few rigid element are used to model the whole seat frame model. Among them, the shell element is used to model the sheet metal stamping structure and the thin-walled thick pipe structure, the beam element is used to simulate the steel spring and pipe with small diameter [4], and the beam element is used to simulate the solder joint structure. In addition to the use of beam element and shell element, some solid geometric parts are simulated with rigid body element [5]. The premise that the rigid body element can be used for simulation is that the stress and strain of the part are not investigated and paid attention to. The part mainly plays the role of transmitting force and moment.

2.2.1. The division criteria of key elements are as follows:

2.2.1.1. The basic size of shell element is 5mm × 5mm;

2.2.1.2. The maximum size of shell element is less than 8 mm (the key parts are controlled about 5 mm), and the unit with 5-8 mm is controlled within 5%;

2.2.1.3. The minimum element size of shell element is greater than or equal to 3 mm;

2.2.1.4. The minimum size of the solid unit is greater than or equal to 4 mm, and the maximum size is less than or equal to 8 mm. (less solid unit parts)

2.2.2. Element Quality

2.2.2.1. Quality inspection criteria:
- warpage≤15
- aspect ratio≤5
- skew≤60°
- jacobian≥0.6
- $45^\circ \leq \text{angle (quad)} \leq 135^\circ$
- $20^\circ \leq \text{angle (tria)} \leq 120^\circ$

2.2.2.2. The number of triangular elements cannot exceed 5% of the total number of elements.

Through the control of element size and element quality parameters, the finite element mesh with high quality can be guaranteed, thus ensuring the correctness of the calculation results.

After the mesh element division is completed, the connection of the whole finite element model is established according to the connection mode of the geometric model of the seat frame, including welding, bolt connection, binding contact and some rotating pair connection. According to the installation mode of the seat, the boundary constraints are set [6].

3. Selection of Working Conditions

3.1. Structural performance analysis of seat under front crash
3.1.1. Loading condition

This working condition standard comes from 4.2.7 of QC/T740-2005, which stipulates that the front crash test shall be conducted to verify and analyze the seat condition equipped with dummy. It is required that the dummy shall not slide out of the safety belt during deceleration, the seat parts, stop and adjustment devices shall not be broken or broken, and bending is allowed [7].

According to the front crash regulations, Hybrid III dummy is used, and the dummy model is fixed on the seat with safety belt. The seat is installed on the pulley, and the specific loading setting is carried out according to any one of the following two situations:

3.1.1.1. The speed of the pulley is 50 km/h, and the deceleration process is in accordance with the actual deceleration of the vehicle;

3.1.1.2. In the process of deceleration within 110 ms, the acceleration of 20 g must be maintained within 35 ms with the peak acceleration of 35 g.

In this collision condition, the second sine wave curve loading method is used. The 95th dummy model of the seat is selected for simulation calculation. At the same time, in order to simulate the frontal collision of the vehicle, the rear acceleration is applied to the connection part between the seat bottom and the body. The specific loading conditions are shown in Fig. 2 and Fig. 3.

3.1.2. Simulation result

The deformation diagram of seat frame structure obtained by observation and calculation is shown in Fig. 4. It can be seen that the upper guide rail on the right side of the seat completely separates from the lower guide rail, and the structure fails due to severe plastic deformation. The plastic strain nephogram of the structure is shown in Fig. 5. From the CAE analysis results, the structure can not meet the design requirements.
3.2. Structural performance analysis of rear crash seat

3.2.1. Loading condition

This working condition comes from the regulation of 4.2.8 in the standard QC/T740-2005, which requires that the seat condition with dummy shall be verified and analyzed through vehicle rear crash test. During collision, the back of vehicle seat can be bent, but collision with other objects is not
allowed. The seat parts are not allowed to be broken or broken, the fixing device between the seat and the floor is not allowed to be broken, and the seat stop device is not allowed to be released, and its function should be kept in good condition [7].

For rear crash requirements, the seat assembly shall be installed on the pulley, and the Hybrid III dummy shall be used, and the dummy shall be fixed on the seat with safety belt. The sled acceleration is set as follows: the total collision process is within 110ms, in this process, 8g acceleration must be maintained within 35ms, and the peak acceleration reaches 10g.

The loading mode of rear crash condition is similar to that of front crash condition. The 95th dummy model is also used for calculation, and the forward acceleration is applied to the connection part between the seat bottom and the body. The acceleration curve is shown in Fig. 6.

![Figure 6. Rear crash acceleration loading curve](image)

### 3.2.2. Simulation result

![Figure 7. Deformation of seat structure](image)

![Figure 8. Deformation of seat frame](image)

![Figure 9. Strain diagram of seat frame](image)
Fig. 7 and Fig. 8 show the deformation diagram of the seat and seat frame in the calculation results, and Fig. 9 shows the strain nephogram of the seat frame. After looking at the deformation diagram of the seat frame in the calculation results, it can be seen that although the structure has a large deformation under the rear crash condition, there is no structural failure. The strain values of the whole seat frame are in a reasonable range and meet the requirements of the industry standard for rear crash test.

4. **Risk Summary**

4.1. **Risk conditions**

4.1.1. **Front crash conditions**

In the condition of frontal collision, the right guide rail of the seat is seriously damaged.

4.1.1.1. At the rear end of the seat guide rail, serious bending occurs, and the upper and lower guide rails are detached;

4.1.1.2. The plastic deformation of the upper opening of the lower guide rail of the seat is serious, and the upper guide rail cannot be fixed well.

The seat and safety belt constitute a system to protect the human body. If the seat structure fails, the whole system will no longer have the complete protection function. Therefore, the seat guide rail part needs to be optimized, so as to ensure that the vehicle will not be damaged when it is in front of the collision.

5. **Optimization Scheme**

5.1. **Structural optimization**

5.1.1. **Seat upper guide rail structure reinforcement**

According to the analysis, the most direct cause of the serious bending deformation of the upper guide rail of the seat is an opening feature on the lower side of the middle part of the upper guide rail. When the rear end of the upper guide rail is subjected to a large tensile force, the stress concentration will occur rapidly at the opening position, which makes the position reach the stress value of plastic deformation. When the structure continues to be stressed, the guide rail will bend, resulting in structural failure.

In order to improve this problem, we adjust the local shape of the gap of the upper guide rail. Because the position of the notch has certain function, it can not be cancelled directly. On the premise of not affecting the structural function of the guide rail, the outer edge of the notch position is strengthened to form a left-right connected square hole. The specific adjustment method is shown in Fig. 10. After adjusting the structure, the anti bending performance will play a very good role in strengthening.

![Figure 10. Model structure of reinforcing plate](image-url)
5.1.2. Material optimization of seat guide rail
Another factor for the separation of the upper and lower guide rails is the plastic deformation of the opening position of the lower guide rail, which can not be well constrained. To solve this problem, we adjust the material parameters of the lower guide rail from the original SAPH440 material to S550MC material. The ability of bearing plastic deformation of the lower guide rail is further strengthened. This will help the whole seat rail to resist deformation and maintain normal working state.

6. Optimization results

6.1. Seat rail optimization results
After the calculation, the deformation diagram and stress nephogram of the structure are extracted, as shown in Fig. 11 and Fig. 12. After observing the structural deformation diagram, it can be seen that after strengthening the seat rail, the seat rail position will no longer be detached under the condition of frontal collision. The stiffness of this part has been obviously strengthened, and the risk of structural failure has been effectively controlled.

By observing the stress nephogram of the structure, it can be seen that the material used in the high stress area in the figure is S550MC. The yield strength and ultimate strength of the material are 550MPa and 760 MPa respectively. The maximum stress in the cloud picture is 700.4 MPa < 760 MPa, so the structural strength meets the requirements.

![Figure 11. Structural deformation of seat frame](image)

![Figure 12. Stress nephogram of seat frame structure](image)

| Model                  | Is the structure invalid | Added mass (g) |
|------------------------|--------------------------|----------------|
| Before optimization    | invalid                  | 0              |
| After optimization     | no                       | 55             |

From the comparison of the data in TABLE 1, it can be seen that the ability of the seat frame to resist deformation has been greatly improved after adjusting the dangerous area of plastic deformation of the seat rail. The weight added in this structural adjustment is only 55g, which has little impact on
the overall weight of the seat. It is a relatively reasonable optimization way to improve a more serious risk problem by adding a small amount of quality.

7. CONCLUSION
The stiffness and strength of a seat frame structure were checked by two test conditions. The seat frame in the initial state can meet the requirements of the rear crash condition. However, it cannot meet the requirements of the front impact condition, and the seat rail has serious slippage deformation. By adjusting the local characteristics of the gap position of the upper rail, modifying the material of the lower slide rail. Finally, the seat frame meets the test requirements of industry standards with increasing a little weight, which provides a certain reference for the design and optimization of the subsequent seat structure.

REFERENCES
[1] Ye Fang, Xu Zhongming, Zhai Xicheng. Strength simulation analysis and research of automobile seat. Electromechanical engineering, 2019, issue11, vol. 36, pp. 1188-1193
[2] Zeng Hua, Liu Jing. Strength analysis and structural optimization of automobile seat frame. Internal combustion engine and accessories, 2019, issue21, pp. 29-31
[3] Bai Xiangzhong, Tan Wenfeng, Zhang Wenfei, et al. Mechanics of materials. Beijing: Science Press, 2007
[4] Fu Yalan, Xie suming. Structural finite element modeling technology based on HyperMesh. Beijing: China water resources and Hydropower Press, 2015
[5] Gao Guangjun. Study on the combined modeling of three-dimensional solid and shell elements by finite element method. China Railway Science, 2002, issue23, vol. 3, pp. 52-54
[6] Hu Yuanzhi, Zeng biqiang, Xie Shugang. Simulation and analysis of vehicle safety based on LS-DYNA and HyperWorks. Beijing: Tsinghua University Press, 2011
[7] QC/T740-2005 .passenger car seat assembly.