World Journal of Engineering and Technology, 2014, 2, 92-99
Published Online May 2014 in SciRes. http://www.scirp.org/journal/wjet
http://dx.doi.org/10.4236/wjet.2014.22010

Analysis of Vehicle Seat and Research on Structure Optimization in Front and Rear Impact

Haining Chen, Hao Chen*, Liangjie Wang

College of Automotive Engineering, Shanghai University of Engineering Science, Shanghai, China
Email: pschenhao@163.com

Received 21 March 2014; revised 26 April 2014; accepted 3 May 2014

Abstract

This paper used the Hyper Mesh and LS-DYNA software to establish a dummy-seat finite element simulation model. The head, chest and neck injury of the dummy were analyzed respectively in the frontal impact and rear impact. It was indicated that modification of seat was needed to meet the requirements. The simulation results showed that the original model cannot provide effective protection for the occupants and need for structural improvements. According to the simulation results of deformation and stress conditions of the seat parts, the original seat structure was improved and optimized for four improvement schemes, including the structure optimization of the seat side panel, the center hinge, framework under the cushion and the backrest lock. The results indicated that the optimized seat improved the occupant protection performance by reducing occupant damage parameters compared with original seat, which illustrated that the optimization basically met the target.

Keywords

Front and Rear Impact, Seat, Occupant Injuries, Optimization

1. Introduction

With the development of auto industry, the safety of the car has increasingly become an important research field of modern automobile development design [1] [2].

As an important safety component, vehicle seat is a hot spot in the study of automobile safety and it provides a decisive protection for passengers [3]. In 2011, Jin Jingxu [4] systematically introduced the seat safety per-

*Corresponding author.

How to cite this paper: Chen, H.N., Chen, H. and Wang, L.J. (2014) Analysis of Vehicle Seat and Research on Structure Optimization in Front and Rear Impact. *World Journal of Engineering and Technology*, **2**, 92-99.
http://dx.doi.org/10.4236/wjet.2014.22010

formance requirements, pointing out that seat back should be strengthened, the cushion stiffness improved and headrest redesigned on low-speed crash protection. Yang Hongying [5] analyzed that insufficient stiffness of seat cushion is the cause of human body diving in rear collision, which result in great damage on the abdomen in 2012. As to the recent research status abroad, Nicolas [6] established finite element models of multi-body human body and seat to study the safety of the crew in rear crash. Masahide [7] *et al*. from Japan's Toyota motor crop, studied occupant protection when the cars crash.

On the whole, current domestic and foreign researches on the safety of the seat emphasize on seat and body connection strength and seat features during frontal crash, and headrest safety and backrest strength in rear impact.

Seat safety refers to the ability to prevent vehicle accidents effectively and to reduce the damage of occupants to a minimum at the time of the accident [8]. Research on vehicle seat in a front and rear collision mechanism of injury to the occupant can provide theoretical technical support for seat design, research and development. It can improve vehicle passive safety performance in a collision and have a great significance for traffic safety [9]-[11].

Based on a domestic car seat as research subjects, the HyperMesh and LS-DYNA simulation software were used in this paper to build a seat-occupant finite element model, and study the injury mechanism of the seat to occupant. The main purpose was to analyze the performance of the seat and the improvement and optimization of the structure, thus to improve dummy injury indicators, to provide effective protection, and provide a method for modern seat design and passenger protection evaluation.

2. Dummy-Seat Model Establishment and Analysis

2.1. Model Establishment

The CAD geometric model was imported to HyperMesh software and meshed according to engineering experience. In this model, the sheet metal parts using two-dimensional grid method and meshing by quadrilateral element and triangular element. Three-dimensional mesh was used in headrest, backrest, cushions and other special components. Belt model was the combination of one-dimensional multi-rigid-body seat belt element and two dimensional membrane elements [12]. Model grid size was controlled at about 10 mm. Final mesh model shown in **Figure 1**. A total of 30954 nodes and 93421 elements were in this model. And then, on the basis of this seat model to join Hybrid Ⅲ 50% male dummy completed the dummy-seat model, as shown in **Figure 2**.

2.2. Frontal Impact Analysis of Dummy-Seat Model

This model was analyzed using a low speed collision. In accordance with requirements of the low-speed colli-

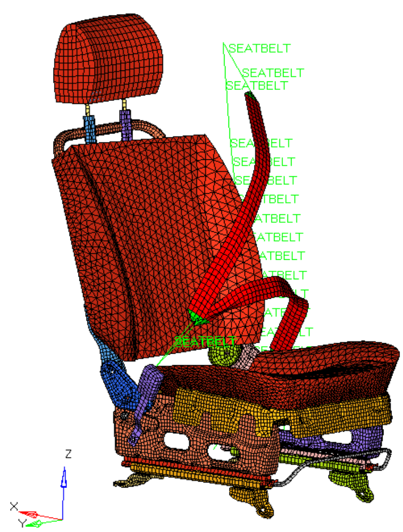


Figure 1. Finite element model of the seat.

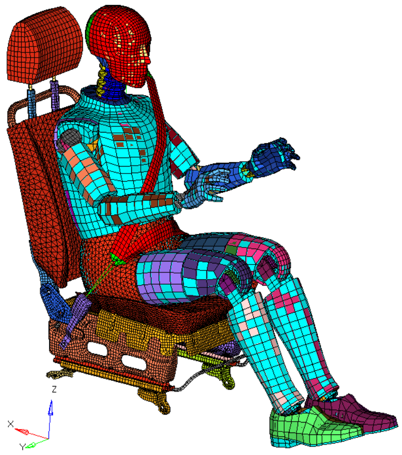


Figure 2. Analysis model.

sion, the collision speed was selected 50 km/h, X-axis negative direction. Collision time was 150 ms. Acceleration curve was shown in **Figure 3**, and test curve from vehicle collision test was compared. As can be seen in **Figure 3**, the overall trends of both curves were basically the same, and the only difference is that the maximum value. Maximum value of simulation acceleration was 47.1 g, while test acceleration is 50.2 g, and the error was 6% within 20%. And, occurrence of maximum moment was basically consistent. Through above analysis, the simulation model of input acceleration was effective, which can be used for further analysis of the dummy movement response and damage in frontal crash.

Using LS-DYNA software was used to calculate the model, and the results were viewed by HyperView software. In the process of vehicle frontal impact, dummy chest injury was the most common body parts. In this study, therefore, adopt the chest compression, chest 3ms synthetic acceleration as evaluation indexes. The calculated curves of the dummy chest and the chest 3ms synthetic acceleration were shown in **Figure 4** and **Figure 5**, respectively.

Figure 4 showed the maximum amount of chest compression was 63.84 mm, and chest compression volume of more than 50 mm period was about 50 ms. This phase last longer, which means that the dummy's chest in a long period of time under large external impact in the process of collision, and this situation led to a large amount of dummy chest compression, resulting in more serious injuries in chest. **Figure 5** was the curve of 3 ms synthesis acceleration of chest, and the value was 24.67 g, which represent the accumulation of head acceleration value within 3 ms. Since it took a shorter time interval, so they can be better described the peak acceleration of collision. The smaller the value, the lower the collision damage was.

2.3. Rear Impact Analysis of Dummy-Seat Model

The speed of rear collision was 50 hm/h, in X direction. The remaining boundary conditions were the same as the frontal collision simulation. Seat acceleration curve was shown in **Figure 6**. As can be seen in **Figure 6**, the overall trends of both curves were basically the same, and the only difference was that the maximum value. Maximum value of simulation acceleration was 18.9 g, while test acceleration was 22 g, and the error was 14% within 20%. And, occurrence of maximum moment was basically consistent. Therefore, the simulation model of input acceleration was effective, and it means the model can be used for further analysis of the dummy movement response and damage in rear crash.

The model calculated by LS-DYNA and the simulation results were analyzed by the post-processing software. In the process of rear impact, the neck was the most common site of injury to the body. Therefore, in this analysis, the dummy neck force in X direction, the torque at Z direction and the backrest angle variation were used as a reference index. Calculate the dummy neck X axis and Z torque curve were shown in **Figure 7** and **Figure 8**.

As can be seen from **Figure 7**, dummy neck X to a maximum force FX was 881N. The value was greater than zero start at around 104 ms, and has larger change. The neck X-force FX was over 730N, and had a longer duration with more than 15 ms. This showed that there were larger impact force act on the dummy neck over a long period of time, which can cause greater harm to the dummy neck. Z-torque MZ was 12.13 N·m, and this curve

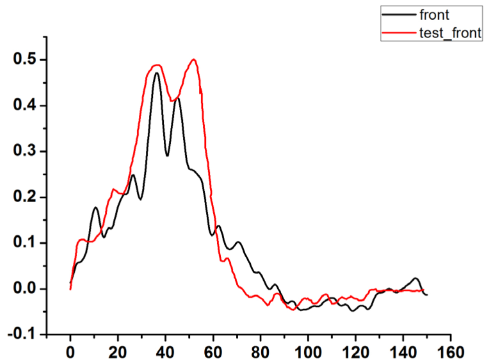


Figure 3. Acceleration curves of front impact.

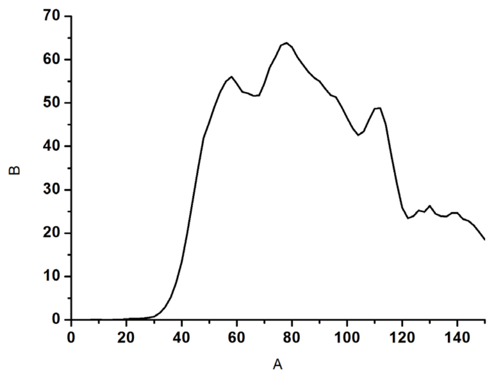


Figure 4. Chest compression of dummy (mm).

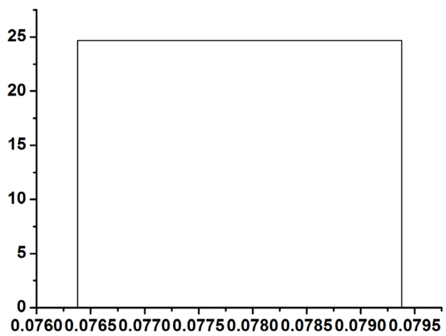


Figure 5. 3 ms synthetic acceleration of chest (g).

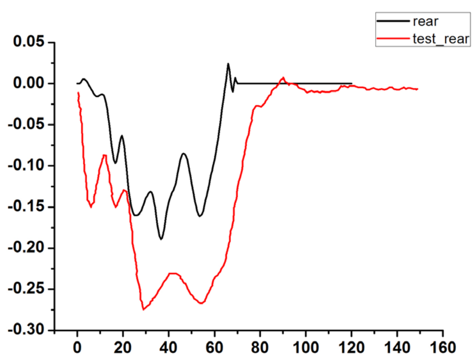


Figure 6. Acceleration curves of rear impact.

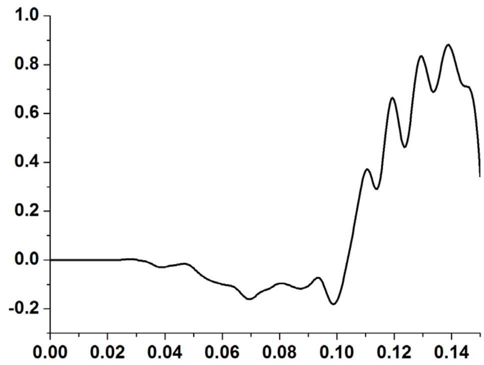


Figure 7. Curve of X force of dummy neck (N).

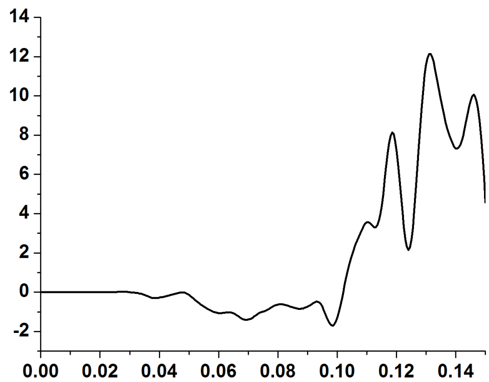


Figure 8. Curve of Z moment of dummy neck (N·m).

value was greater than 0 after 102 ms, which would cause some dummy neck injury.

The seat back angle change curve was shown in **Figure 9**.

As can be seen from **Figure 9**, with the increase of the force of the backrest, the backrest angle increased, the maximum change of the backrest angle was 22.97˚, which can be further optimized.

3. Suggestions and Research for Improvement Programs

3.1. Seat Problem Description

From the above analysis showed that in the process of frontal crash, the amount of the dummy's chest compression was poorer, and the seat structure should be improved.

The deformation and stress of the seats in the frontal and rear crash was analyzed, and the stress nephogram was shown in **Figure 10** and **Figure 11**.

From **Figure 10**, the stress mainly concentrated in the lateral plate and seat slide. Seat side plate strength was low, slippery course strength was poorer, deformation was bigger, and it can increase the seat strength by strengthening their strength. **Figure 11** showed that in the rear impact, the stress mainly concentrated in the backrest and cushion joints. Hinge joints exist stress concentration, weaken the lower seat strength, and result in poor safety performance, which reduced the protection for dummy during the impact.

3.2. Suggestions on Improvement

Given the large deformation seat side and side slabs seats had important support role, it must ensure that its strength was enough. This article increased side sheet thickness increased to 30mm, and removed the two non-mounting holes at the same time (shown in **Figure 12**).

The center hinged backrest was modified. As the backrest in a frontal collision force was greater than the center hinge endurance. Thus, reinforcement was added to the center hinge to strengthen the intensity of the

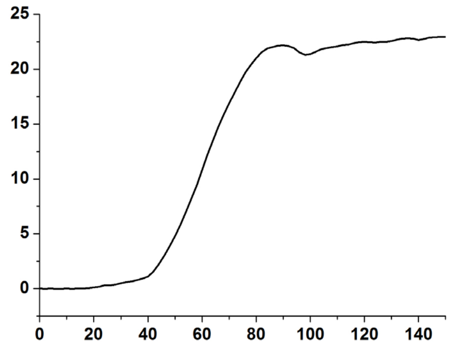


Figure 9. Angle change of backrest (˚).

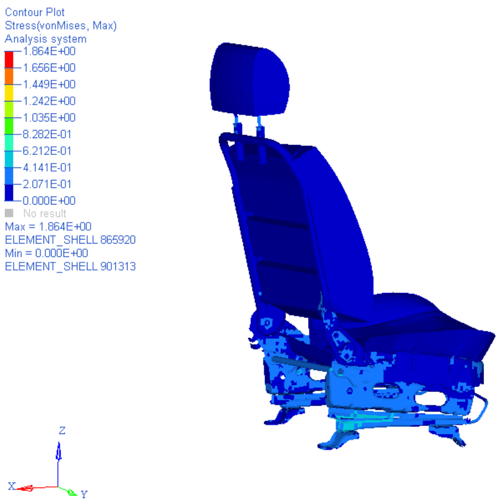


Figure 10. Stress nephogram of seat during front impact.

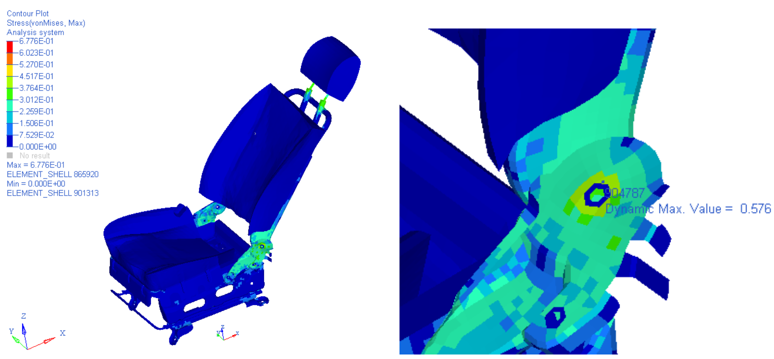


Figure 11. Stress nephogram of seat during rear impact.

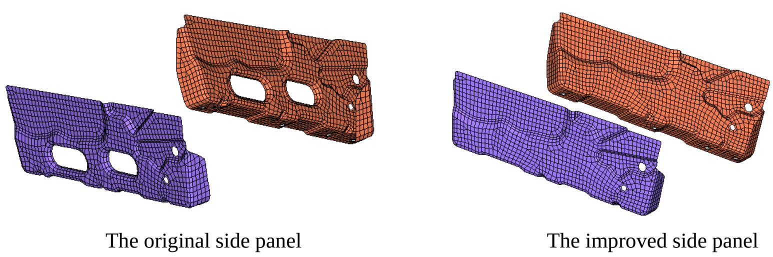


Figure 12. Structure improvement of side panel.

central hole, and increased the connection hole in the center hinge and back to make the connection become more firmly (shown in **Figure 13**).

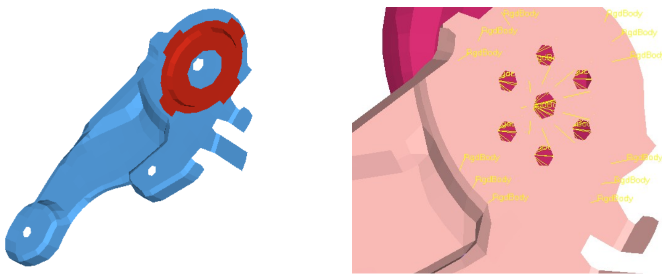
In the original structure, cushion was on both sides of the lower bracket transition for straight line, and there was stress concentration phenomenon, so stress prone to mutation. This will make parts bend or even break. Thus, the circular arc shape was rounded in the new model, and extended its bottom until connected to the side panel, to increase the intensity of the seat bottom at the same time (shown in **Figure 14**).

In the back of the lock member, along it elongation was shorter (56 mm), low intensity, and made the chair back strength was insufficient. Therefore, its length was increased to 269 mm. Meanwhile, considering the manufacturing craftmanship, changed the shape right side of the backrest lock to the left (shown in **Figure 15**).

3.3. Improved Model Analysis

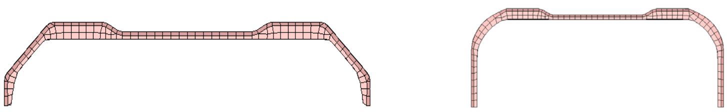
On the basis of the improvement structure, the optimized seat finite element model was established. Improved model analysis was done again in LS-DYNA, and comparing with the simulation results of the original model. Chest compression, 3 ms resultant acceleration, neck X-force, Z-moment and seat back angle were compared before and after optimization, which was listed in **Table 1**.

As shown in **Table 1**, the dummy injury indicators decrease obviously. Among them, the backrest angle, chest compression and 3 ms synthetic acceleration decreased apparently. This was due to the optimized sides strength increase, so that the seat can withstand a greater impact in front collision. Center hinge and back lock strengthen result in backrest strength increases, so backrest can withstand greater impact, and it angle variation decreased, the dummy rebound decreased when impacted by external force.



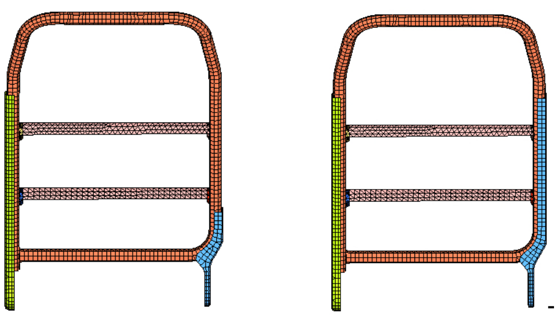
Adding reinforcement　　　　Increase the connection hole

Figure 13. Structure improvement of center hinge.



Original lower bracket　　　　Improved lower bracket

Figure 14. Structure of bracket under cushion.



Original chair back　　　　Improved chair back

Figure 15. Structure improvement of back lock.

Table 1. Comparison before and after optimization.

Index	Before optimization	After optimization	Change amount	Rate of change
Chest compression/mm	63.84	52.85	10.99	17.21%
3ms synthetic acceleration/g	24.67g	19.45g	5.22	21.16%
F_X/N	881	745	136	15.44%
M_Z/N·m	12.13	11.75	0.38	3.13%
Backrest angle/°	22.97	12.38	10.59	46.10%

Results showed that the optimized results had been effectively in strengthen, and it has provides an important protective to reduce the occupant injury during impact.

4. Conclusions

- Dummy-seat model was established by HyperMesh and LS-DYNA software. Analysis was conducted to determine the performance of seat in dummy protection during front and rear impact. Results showed that the structure was needed to be modified to ensure passenger protection.
- Based on the analysis of the seating stress nephogram, the seat structure was improved and optimized for four improvement schemes, including the structure optimization of the seat side panel, the center hinge, framework under the cushion and the backrest lock.
- Simulation results indicated that the optimized stricter can strengthen passenger protection. In the final optimization schemes, chest compression reduced 17.21%, 3 ms resultant acceleration reduced 21.16%, dummy neck FX decreased 15.44%, MZ value decreased 3.13%, backrest angle decreased 46.1%.

References

[1] China Communications and Transportation Association (2012) China Transportation Yearbook. People's Communications Press, Beijing, 83-115.

[2] Chen, H. (2010) Structural Design and Finite Element Simulation of Automobile Driver Seat. Northeastern university, Shengyang.

[3] Li, Q.H., Li, J.P. and Li, S.H. (2007) Finite Element Analysis and Test Study about Sled Impact Test of Automobile Seat. *Automotive Technology*, **5**, 30-34.

[4] Jin, J.X. (2011) Research on Headrest and Wield Stripes Performance and Improvement of a Car Seat. Jilin University, Changchun.

[5] Yang H.Y. (2012) Prevent and Stripes for the Optimization Design of the Safety Seat Research. Hunan University, Changsha.

[6] Bourdet, N. and Willinger, R. (2005) Human Body-Car Seat Coupling under Rear Impact. The 19th Enhanced Safety of Vehicles, Paper Number 05-0261.

[7] Sawada, M. and Hasegawa, J. (2005) Development of New Whiplash Prevention Seat. The 19th Enhanced Safety of Vehicles, Paper Number 05-0288.

[8] Wang D.F. (2011) A Study on the Impact Safety of Commercial Vehicle Cab and Its Improvement. *Automotive Engineering*, **1**, 34-36.

[9] Long, H.Q., Chen, J.H. and Wu, Y.X. (2011) Application of CAE in Development of Automotive Seat Frame. *Computer Aided Engineering*, **20**, 112-115.

[10] Shin, M.K., Park, K.J. and Park, G.J. (2003) Occupant Analysis and Seat Design to Reduce Neck Injury from Rear End Impact. *International Journal of Crash Worthiness*, **8**, 573-581. http://dx.doi.org/10.1533/ijcr.2003.0260

[11] Chen, D.J., Fu, D.C. and Wang, H.H. (2011) Lightweight Design of Car Seats. *Mechanical Science and Technology*, **30**, 26-32.

[12] Yang, H.Y. (2012) A Preliminary Study on Anti-Whiplash Seat. Hunan University, Changsha.