Optimization and Experiment of Vehicle Frontal Impact Model

Sun Xilong[1, 2], *Wang Dengfeng*[1*]

(1.State Key Laboratory of Automotive Dynamic Simulation, Jilin University, Changchun City, 130025, caewdf@jlu.edu.cn)

(2. FAW-Volkswagen Auto.Co.,Ltd. Changchun City, 130011, xilong.sun@faw-vw.com)

Abstract: The finite element analysis model of vehicle frontal impact was built using PAM-CRASH software. The simplification of vehicle structure, the mesh quality, the definition of the material characteristics, the contact types and the simulation method of whole vehicle physical connections were discussed. The crashbox and longeron were optimized based on the crashworthiness of the vehicle. The reliability and accuracy of the frontal impact model were validated according to the impact energy curves and the comparison of B-pillar acceleration and longeron deformation with the corresponding experimental data.

Keywords: Automobile, Frontal Impact, Passive Safety, PAM-CRASH

1 Introduction

Automotive passive safety is defined as the ability of occupant protection and pedestrians from being hurt in the traffic accident, preventing the vehicle fire and evacuating occupants immediately afterwards.With the rapid development of the economy in China, demand of people to vehicle is increasing rapidly. Different kinds of traffic accidents not only bring injuries to the occupants, but also the pedestrians, so more and more attentions are paid to automotive crash safety. To improve the automotive passive safety and reduce the passenger casualties, the crashworthiness of vehicle structure and occupant protection should be studied at the vehicle developing stage [1].

In this paper the finite element impact model of a car was built. The key technology contents about mesh quality of the model, the material properties, the contact definition, the physical connection, the mass distribution and so on were discussed. The crashworthiness of the vehicle structure was simulated and the crashbox and the longeron were optimized. To verify the reliability of the optimization model and the validity of the modelling method, the initial conditions and boundary conditions of the model were set as the test requirements of China-NCAP front 100% rigid fixed barrier impact at the speed of 50km/h. 120ms impact response was solved using PAM-CRASH software and validated with the testing results [2].

2 Vehicle Modelling

When building a vehicle impact model, not only the vehicle structure should be reflected, but also the correctness of the connections among assemblies and parts should be ensured, such as the accuracy of the material characteristics, the density, the mass distribution, the rationality of the initial conditions, the boundary condition and the control parameters[3].

2.1 Finite Element Mesh Division

Mesh quality is one of the important factors that influences the accuracy of the simulation results. Since most parts of the vehicle body are punched with the sheet metals, the four-node quadrilateral and three-node triangular elements are used to discrete the vehicle body structure. The amount of the three-node triangular elements as one of the overall vehicle elements is controlled within 10%. Also the control parameters of the elements, such as the maximum and minimum of the element's interior angles, the warping angles and the element aspect ratio must meet rigid requirements. Structural features with small size do not have significant effect on the vehicle stiffness and strength, for example, the holes are all ignored when diameter is less than 7 mm. Structural characteristics having significant influence on the vehicle stiffness and strength, which deterimined the impact deformation mode and load transfer path were discreted into meshes. This was done according to the actural structure, such as the front floor plate, the front longeron, the front bumper, the inner plate of the door, the inner plate of the bonnet, the stiffening ribs of A-pillar and B-pillar, the groove and big opening. One element mesh was applied if the fillet curvature radius of the structure was less than 5mm, otherwise two elements were applied. The models of the flanging structures, for example, the door and the bonnet were all built according to the actual structure [4]. The outer and inner plates were connected with a shared-node to ensure the bending stiffness of the flanging structures was consistant with the actual value (Figure 1). The engine and transmission were defined as the rigid bodies because of their high stiffness and small deformation. There were 860,000 nodes and 850,000 elements in the whole vehicle model including body-in-white (BIW), the doors, the power train, the suspension system, the steering system, the braking system and the seats.

* Corresponding author: Wang Deng-feng, caewdf@jlu.edu.cn

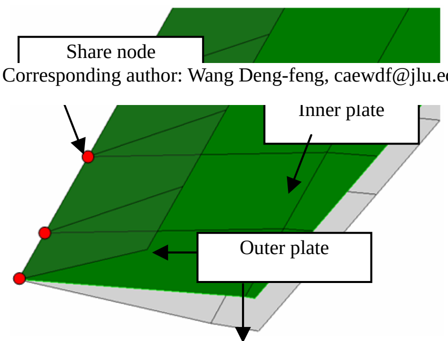

Figure 1 Flanging structure

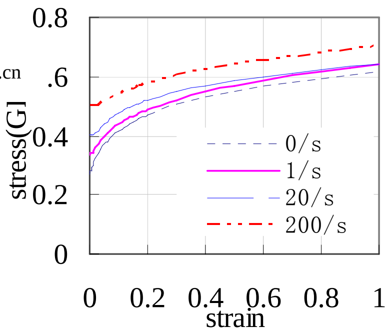

Figure 2 Stress-strain curve of H260LAD

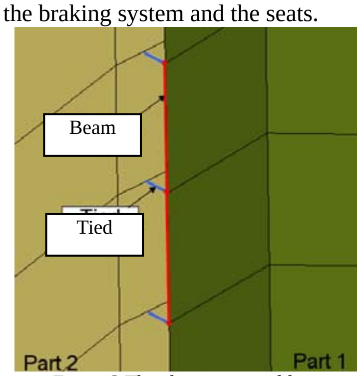

Figure 3 The electric arc welding

2.2 Material

Most metal parts of a vehicle are made of low-carbon steel, which behaves differently under the action of the dynamic load and static load. When high-speed impact was applied, the yield limit of the material was improved significantly. If the strain rate was ignored, the deformation in simulation will be larger than the test value which resulted in a larger error [1]. Moreover the definition accuray of the material properties will influence the precision of the model directly, so the strain rate must be taken into the consideration [5].

PAM-CRASH provides two methods defining the strain rate of the materials. One is the formula definition which has five strain rate models: Cowper-Symonds, Johnson-Cook, Modified Jones, Left Shifted Stress-Strain, and Modified Krupkowsky strain rate model. Another is defined based on the high-speed stress-strain curve of the materials. The second definition was applied in this model [6]. Figure 2 is the stress-strain curve of H260LAD material in different strain rate.

2.3 Contact Types

There are different kinds of contact types in vehicle impacting process, which are classified into three categories: the node-to-node contact, the face-to-face contact, and the one-side automatic contact. The above three types of contacts correspound to the contact of the deformation body to the deformation body, the deformation body to the rigid body and the one-side deformation body respectively [7]. Because the normal impact force and the tangential friction force on the contact interface are the main reasons of the elastic-plastic deformation, it needs the different kinds of contact interface model to define based on the contact types to avoid penetrateing. The static and dynamic friction coefficient needs to be considered to ensure the accuracy of the model in the contact simulation [5]. The four-types of contact were chosen in this study included: No.10 contact in the back and cushion system; No.33 in the vehicle and the rigid wall, the tire and the ground, the front wheel and body, the front wheel and the rigid wall; No.36 in the vehicle's automatic contact; the lock front and rear doors, and No.46 in the mudwing and the front door.

2.4 Connection

There are many kinds of physical connections in a vehicle: the spot weld, the electric arc welding, the hinge connection, the bolt fastening and the glue. These make the vehicle parts integrate into the whole vehicle and have significant effectsig on body stiffness and impact results. The connection of the spot weld, the electric arc welding, the articulation, the bolt fastening, and glue were used in this model to simulate correspondence to the combination element, the rigid connection for spot-weld[8]; the combination of Beam and Tied elements for the electric arc welding [9]; the pillar hinge for the articulation; the combination of solid and tied elements for the glue connection of the bonnet and the inner plate, the windshield and the body, the roof and the cross member; endowing solid element with the properties of glue (Figure 3). Tied elements were used to simulate the connection between the assembly and the body.

2.5 Model Mass

The model mass will be reduced due to the simplification of the model structure and the ignorance of the interior trim parts and electrical parts. In order to keep the model mass inconsistent with the actual value, the actual mass of the assembly and the key parts were consulted and the mass distribution and centroid position in the model were adjusted. Non-structure mass in the model, such as the dummy, the oil and the spare wheel, were added to the assembly, which made the centroid position and the total mass of the model match the actual [10]. The total mass of the impact model after adjustment was 1541 Kg, including the vehicle weight，two Hybid III 50% male dummies, one Hybid III 5% female dummy.

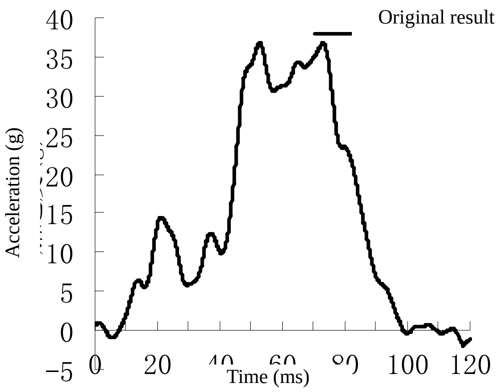

Figure 4 The acceleration time history of the original vehicle

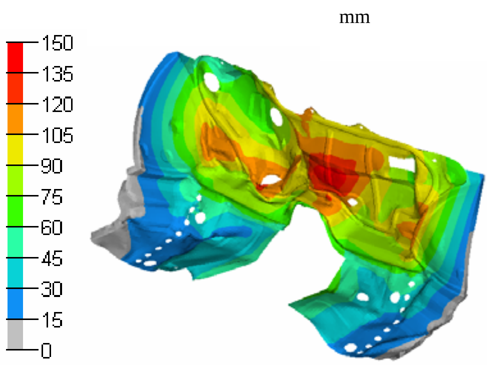

Figure 5 The deformation of fire wall in X direction

3 Simulation Model Analysis

Figure 4 shows the acceleration time history of the vehicle, where the acceleration reaches the peak value 36g at 53ms. The longeron was divided into the three segments: AB segment, BC segment and CD segment according to the analysis requirement (Figures 5-7). The depth of the crush deformation of AB segment is small due to the longitudinal stiffness of BC-segment is smaller than that of AB and CD segment. Before AB segment is deformed sufficiently, BC segment had deformation greatly in many places, which made the firewall intrusion bigger in X diredtion towards passenger compartment. In order to reduce the injury of the occupant, decrease the intrusion of firewall in X direction, the frontal stiffness of the vehicle needs to be increased.

During the impact process, although crashbox I (Figure 8 and 9) is crashed and absorbed the partial impact energy, the structure whose the three guiding grooves in the position of each edge don't fully work, only has secondary fold-deformation, which isn't enough and doesn't deform as desired.

4 Optimization

4.1. Crashbox Structure Optimization

Crashbox I doesn't deform as designed in the impact process, so the depth and the opening width of the structural deformation of the groove-oriented is optimized, which keeps the material and the thickness unchanged, with an opening width 24mm, the depth of the guide groove 5mm (Figure 10).

4.2. Longeron Optimization

Material H260LAD is used in the original longeron which is equal–thickness 2.0 mm. According to impact-deformation mode (Figure 7), AB segment can absorb more energy, so the stiffness of AB segment needs to be reduced, the stiffness of BC segment increased. If AB segment absorbs more energy and the deformation of BC segment is reduced, the intrusion of the firewall is reduced either. The longeron is optimized according to ideal characteristics of longitudinal impact. The longeron adopted were unequal –thickness structure, AB segment with a thickness 2.0mm, BC segment 3.0mm, and CD segment 1.5mm.

5 Model Validation

Optimization results are validated and analyzed based on the vehicle impact test.

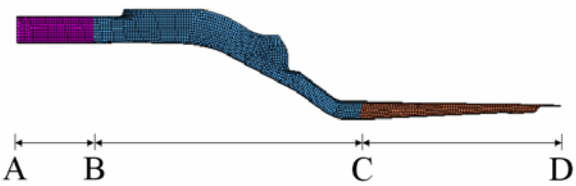

Figure 6 Longeron

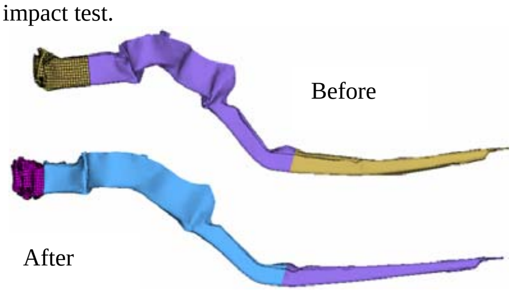

Figure 7 The deformation of the longeron

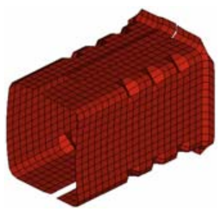

Figure 8 Crashbox Ⅰ

Figure 9 Deformation of crashbox Ⅰ

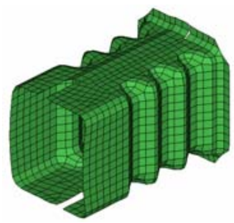

Figure 10 Crashbox Ⅱ

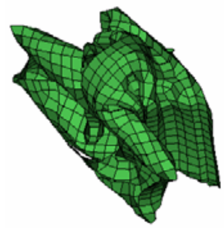

Figure 11 Deformation of crashbox Ⅱ

5.1 Body Acceleration

In the acceleration curve of optimization results, the values of the 4th and the 5th peak (arrange with the value) are obviously improved compared to the original simulation results, because of the adjustment of the crashbox structure and the thickness of longeron which make the frontal stiffness larger (Figure 12). The maximum peak value is 38g, increasing by 2g, and the 1st, 2nd and 3rd peak value tends to decrease.

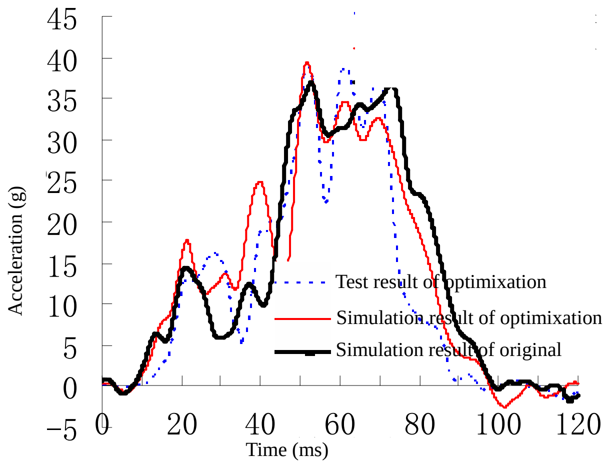

Figure 12 The acceleration time course of B pillar

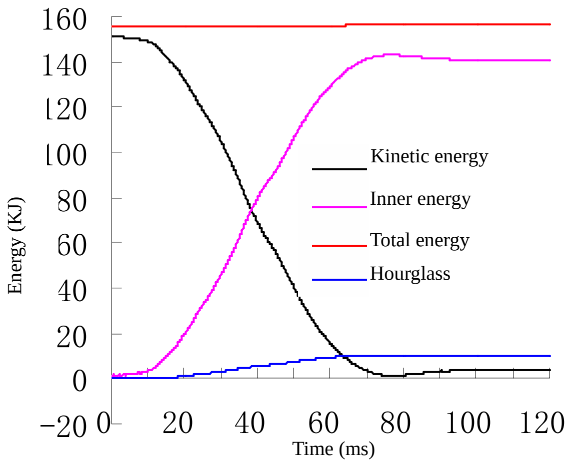

Figure 13 The curve of crash energy in optimization model

When comparing the optimization results with the test results, 51ms is the moment that the maximum acceleration occurs with the peak value showing the small difference of 0.6g in the comparison. The trend of the acceleration waveform curve is also the same with the test curve. Among the curves of the impact simulation, the time interval of the next two peaks is simulated better, moreover the peak value trend to decrease.

5.2 Energy Curve

Vehicle impact is a transient process of energy conservation and momentum exchange, where most of the kinetic energy is converted into deformation energy. Figure 13 is the energy curve of the optimized vehicle model in impact process, which shows that total energy of the system, 155KJ, keeps unchanged in impact process. When the car contacts the rigid fixed barrier, contact deformation occurres in the frontal of the car, kinetic energy of the system gradually decreased nonlinearly and car body absorbs impact energy. This happens because of crushing and bending deformation that makes the internal energy increase.Within 76ms the deformation of the body reaches maximum and that makes internal energy reaches maximum and kinetic energy of the system and impact velocity tend to be zero.After 76ms, which the deformation energy realeased converts into kinetic energy of the system makes that kinetic energy of the system increases and the internal energy decreases in the presence of the curve, because of the rebound of the structure. Actual energy change trend matches that of the simulation in the impact process. Although the hourglass consumes some of the energy, which makes sum of the kinetic and internal energy of the system less than the total energy of the system, the hourglass(the energy of which accounts for 6% of the internal energy) has little impact on the computing results, so the simulation result is acceptable[11].

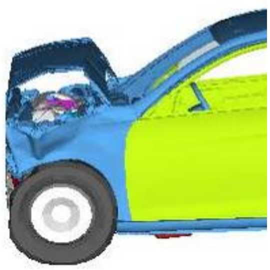

Figure 14 Front deformation

Figure 15 Front deformation of test

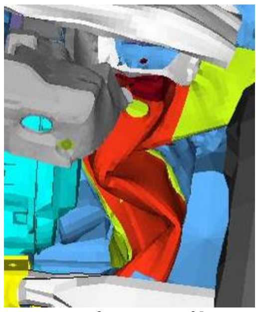

Figure 16 Deformation of longeron

Figure 17 Longeron deformation of test

5.3 Deformation Mode

The front-end of the crashbox II, not only has three fold-deformations adequately, but also ensure the AB-segment of the longeron is crushed to deform in X direction in impact process, playing a part in deformation guidance and making deformation mode reasonable, has been optimized, compress in X direction.

Between 20ms and 40ms, AB-segment of the optimized longeron is crushed to deform fully. Compared with the original longeron, the optimized one absorbs more energy (Figure 7). BC-segment deforms after AB-segment has absorbed the energy.Then the kinetic energy of the whole vehicle has reduced to 20KJ, which converts into internal energy through the structures such as longeron, the front floor and roof, A-pillar, all sharing the impact energy and reducing the amount-intrusing of the front wall angle.

Figure 14 and Figure 15 are body deformation graphs from simulation and test at 80ms. It can be seen that the contact state of the tire and the rigid wall, the deformation of the engine bonnet resembles that of the test vehicle. This shows that in the simulation, the articulation of the the engine bonnet and body, the glue of the inner and outer plate, reflects the physical connection of the test vehicle well. Figure 16 and Figure 17 are deformation graphs of the longeron simulation and test results which show that the deformation mode of the longeron, the bending position, and angle match well.

6 Conclusions

1. A method to simplify structrure geometry is proposed, including the requirement of the element's quality, the methods of structure connection, the definition of contact, which is correct and valid and can be adopted to building the crash simulation model later.

2. The simulation results of the model correspond well with the test results, and meet the requirement of the engineering design. Factors are considered comprehensively which take into effect the simulation accuracy, and the accuracy of the results of the simulation model.

3. Since the crashbox is optimized well and its deformation is adequate in crash, the guiding effect is realized well.

4. Longitudinal stiffness of the structure is distributed reasonable with an unequal-thickness, which makes the deformation of AB segment, low-stiffness position occur firstly, then BC-segment after absorbing vehicle kinetic energy adequately to reduce the intrusion of the firewall in X direction.

Reference

[1] 钟志华，张维刚，曹立波，等. 汽车碰撞安全技术[M]. 北京：机械工业出版社, 2003

[2] 王宏雁，高卫民，贾宏波. 轿车正面碰撞计算机数值模拟的分析[J]. 安全与环境学报, 2001, 1(5): 1-3

[3] 杨华. 电动汽车正面碰撞仿真分析[D]. 武汉：武汉理工大学, 2008

[4] 郝春鹏，范子杰，桂良进，等. 微型客车车身结构正面碰撞特性的数值模拟[J]. 汽车工程, 2004, 26(5): 274-275,995

[5] 高彬. 汽车正面撞击的结构耐撞性研究[D]. 哈尔滨：哈尔滨工业大学, 2006

[6] 刘军，李玉龙. PAM-CRASH 应用基础[M]. 西安：西北工业大学出版社, 2008

[7] 黄世霖，张金换，王晓东，等. 汽车碰撞与安全[M]. 北京：清华大学出版社, 2000

[8] 高卫民, 王宏雁, 徐敦舸. 碰撞模拟过程中焊点的影响[J]. 同济大学学报, 2001, 29(7): 870-872

[9] Mahmood H F, Paluszny A. Analytical Technique for Simulating Crash Response of Vehicle Structures Composed of Beam Elements. Sixth International Conference on Vehicle Structural Mechanics, Detroit, 1986, SAE Paper No.860820

[10] Cheva W, Yasuki T, Gupta V, Mendis K. Vehicle Development for Frontal/Offset Crash Using Lumped Parameter Modeling. 1996, SAE Paper No. 960437

[11] 徐文岷. 汽车碰撞过程的有限元数值模拟[D]. 哈尔滨: 哈尔滨工程大学, 2007