第29卷 第6期 湖南大学学报(自然科学版) Vol. 29, No. 6
2002年12月 Journal of Hunan University(Natural Sciences Edition) Dec. 2002

Article number: 1000-2472(2002)06-0131-09

Finite Element Analysis of a Passenger Car-to-Highway Guardrail Crash*

MO Jin-xiang[1], YANG Ji-kuang[1,2], HUANG Hong-wu[1]

(1. College of Mechanical and Automotive Engineering, Hunan University, Changsha, China;

(2. Department of Machine and Vehicle Systems, Chalmers Univ of Technology, Sweden)

Abstract: With the rapid growing incidence of highway roadside crash accidents there is an increasing need to enhance highway guardrail safety. The objective of this paper is to study dynamic responses of vehicle-to-guardrail impacts and to understand the influence of guardrail design to protective mechanisms during crashes, which forms a basis for development of safer guardrail. In this study, the finite element method was used to build a passenger car-guardrail system model. The computer simulation of car-to-guardrail impacts was carried out by using the model at speed of 100 km/h with an impact angle of 25 degrees. The responses of the car-guardrail impact were analyzed with focus on the displacement and deformation mode of the W-beam, the failure stress distribution on the splice hinge, the influence of ground friction, as well the kinematics of the car model. The results indicated that the finite element analysis is an efficient method to imdrail design for development of safer guardrail system.

Key words: vehicle impact; computer simulation; finite element method; guardrail design

1 Introduction

With the rapid development of highway in our country the highway plays a more and more important role in transport industry, but the highway traffic accidents increased also from 2870 in 1994 to 16916 in 2000 (Fig. 1), of which about 30% of the reported accidents were the vehicle-to-guardrail crashes. These accidents resulted in a large number of casualties and huge economic loss. So that improving highway safety and minimizing the risks of injuries in the accidents is a priority in vehicle traffic safety area. The safer guardrail has been one important subject and the systematic research on guardrail has been widely conducted in the research institutes, university, transportation and vehicle industries. Researchers have found that the protection performance of

* Received 2002-10-29, revised 2003-03-21

fence depends on big deformation character. Since determining this kind of large deformation mechanisms will be inevitably concerned with big displacement, turn, warp and unknown contact interfaces which have overstepped the category of theoretical analysis, so experimental study and computer simulation analysis should be employed.

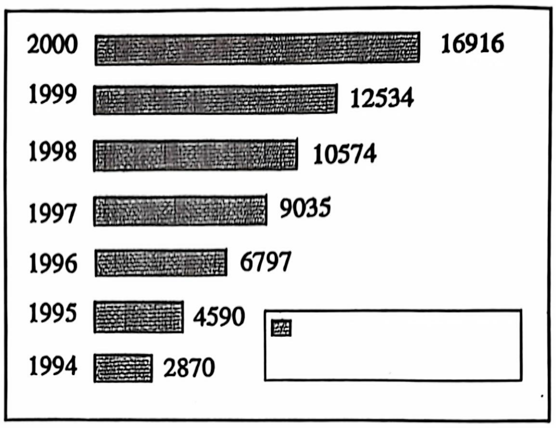

Fig. 1 The highway traffic accidents in China from 1994 to 2000

Conducting large-scale tests to gain insight into car-guardrail impact is still the most important approach. But this method is expensive and time consuming. These kinds of experiments which should be conducted after the guardrails have been turned out. The test results have been mainly used to verify whether guardrails have proper big deformation characters. At the same time, because of the restriction of time and capital, it is hard to consider various schemes to solve design problems and make the optimization of products design difficult. On the other hand, finite element analysis of impact behavior under different impact scenarios is cheaper and efficient. It is economically possible to perform impact tests on a wide range of parameters.

In the 1970's, Lawrence Livermore National Laboratory (LLNL) began to develop DYNA3D, a kind of explicit and nonlinear finite element program, to solve burst and highspeed impact problem. Not only limited experiment data but also the more detailed value answer can be obtained with more few investment by value analysis. On the other hand parameter study or sensitivity analysis can be carried out promptly on this base. Value analysis also make it easy to realize the optimization design of guardrail. This years with computer performance rise, price drop as well as military technology civil melt, nonlinear finite element analysis have become a kind of practical tool to verify and design roadside hardware.

Today, under the computer tech support, in the field of vehicle—guardrail collision safety research , the method of traditional collision test → design improvement → test again has been being step by step replaced by computer value simulated. Plenty of related researches have been carried out abroad. Computer simulated research of collision safety has been major technical scheme, and has gotten rapid progress.

2 The Method of Computer Simulation

2.1 The theory of finite element analysis

The first step of finite element analysis is to divide complex structure into many little elements then exert the boundary condition and condition of dynamic balancing on elements to calculate stress and the displacement of every element. In accidents guardrails have big deformation, geometry and material nonlinear characters. These characters make it difficult to forecast the dynamic response of guardrails. Until 1980', nonlinear finite element analysis technology has just been used to solve these collisions problem. In collection the trajectory of car and the kinetic state of occupants are very complex, therefore the establishment and the solution of kinetic equation are also very complex.

In colliding process, the motion law of vehicle and person is difference. In addition, the big deformation of car body will cause the instrument panel and the front board shrink into passenger room. As a result the distance between passenger and indoor object will be reduced. All these will make the colliding problem more complex. The study of computer simulation technology generally include the content as follows:

(1) Employ the method of finite element analysis to study the deformation of car body and frame and the dynamic response in collision.

(2) Study the dynamic response of human body in various collision condition

(3) Dynamics calculation of multi-body

What is related to it, the most basic and also the most complex work, is to develop the mathematics models that can reflect actual collision of process truly.

2.2 The software of finite element analysis

LS-DYNA developed by Livermore Software Technology Corporation development is one general explicit finite element analysis program. It can be used to analysis nonlinear dynamic responses of three-dimensional structure. Hallquist published the first version of DYNA3D in 1976. Initial purpose is to solute explode problems afterwards it had been modified several times. In 1988, Hallquist published the software LS-DYNA that is specially used in collides structural analysis of vehicle.

The basic equation of LS-DYNA is based on Lagrange equation. The Algorithm is mainly based on explicit time integral. Program can calculate the time step to meet steady condition. At the same time it can also determine the contact area voluntarily and complete the contact analysis. User may set different type of contact conditions such as adhesion force, contact, Slide and friction. There are many material models in LS-DYNA and it have still combined plenty of advanced functions including complete automatic contact handling and wrong detection, HybridIII dummy model, adaptive grid partition etc. Today, LS-DYNA can be directly supported by the third-party after- treatment and the pre-treatment software system.

3 Description of FF Model

3.1 Vehicle FE model

The vehicle FE model used for the simulation is based a 1991 Ford Taurus. It was developed by EASi Engineering for the National Highway Traffic Safety Administration (NHTSA)[1]. Fig. 2 shows the vehicle model of the Ford Taurus. Table 1 shows the calculation formulation of various types of elements used in the model.

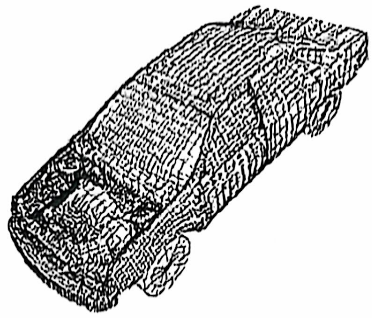

Fig. 2 Ford FE model

Tab. 1 Formulation

Element Type	Formulation
Shell	Belytschko-Tsay
Beam	Belytschko-Schwer
Solid	Constant stress

Tab. 2 Overall parameter

Weitht	1.56T
Length	4600mm
Width	1800mm
Height	1400mm

Table2 shows the overall parameter of the vehicle model. The FE vehicle model components are connected to each other using the spot weld and rigid body constraint options in LS-DYNA3D. A total of 142 spot welds and 178 rigid body constrains are used in the model. The contact and friction between the components are modeled with one single surface sliding interface type 13.

3.2 Guardrail FE model

The guardrail models are based on the models of standard guardrail components developed by Ray and Patzner[2]. Fig. 3 shows the guardrail FE model. Each individual 3810-mm long guardrail section was modeled as a separate part and was attached using nonlinear clamping spring and slot slip-springs at all the bolt locations. The FE model of the guardrail system consist of six w-beam rail section with eleven posts at 1905mm center-to-center spacing resulting in a total guardrail length of 22.9m. The w-beam component was attached to the posts using the nodal-rigid-body spot weld option in LS-DYNA. Linear springs were attached to the upstream end of the farthest upstream section of w-beam rail and to the downstream end of the farthest downstream section of w-

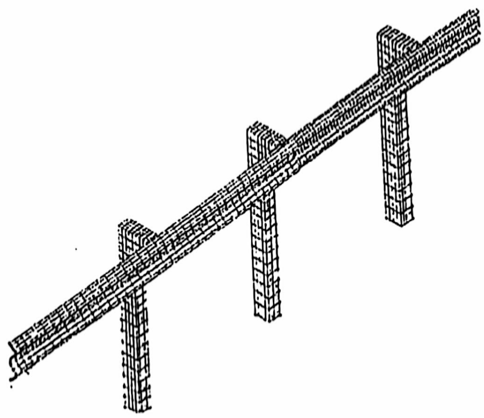

Fig. 3 Guardrail FE model

beam rail to simulate an anchored system. The stiffness of the end spring corresponds to the stiffness of the unmodeled section of w-beam and is calculated from the relationship:

$$K=\frac{AE}{L}$$

Where K, A and E are the elastic stiffness of the un-modeled guardrail, the cross-sectional area of a w-beam and the Young's modulus of steel, L is the un-modeled length.

The post-soil interaction is modeled using spring attached directly to the face of each post below the ground surface as described by Plaxico, Patzner and Ray[3].

4 Crash Simulation

Fig. 4 shows the FE model of car-guardrail impact system. What is corresponding in the figure is the relative location between car and guardrail at initial time. The initial velocity is 100km/hr and the impact angle is 25 degree. After 0.283s, the deflection angle between car and guardrail is zero. At that time the car and the guardrail were parallel. The velocity was about 68km/hr. Finally the car separated from the guardrail with 14 degrees angle and the speed of car is 63.0 km/hr.

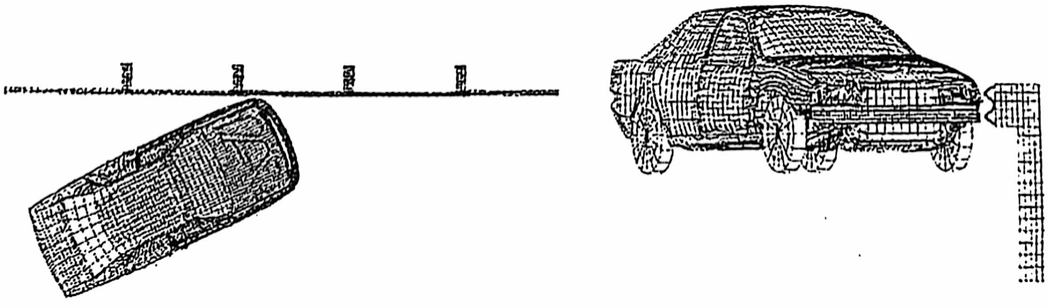

Fig. 4　Car-Guardrail Crash Simulation System

5 Analysis of the Result of Simulation

5.1 The friction retween car and ground

The friction between ground and vehicle has remarkable influence on vehicle's overturn in collision process. The instant force can be decompounded into lateral force and longitudinal force. Lateral force that is perpendicular to guardrail's vertical plane keeps balance with reaction of guardrail. On the direction of vehicle movement, because of the proliferation of deformed wave the movement of the vehicle will be resisted. Longitudinal force will take effect along the vertical plane of the guardrail. And its magnitude depends on the friction between car and guardrail. Under the action of the two forces, the velocity of the car will be reduced. At the same time the car begin to deflect with a certain angle. The magnitude of angular velocity is direct ratio with the above-mentioned force and is inverse ratio with initial moment of car. If moment caused by lateral force is larger than that caused by longitudinal the vehicle will leans to the guardrail and the track of the vehicle will be rectified. Otherwise vehicle will turn Lateral or roll.

5.2 Response of guardrail

Fig. 5 shows the damage degree of guardrail. During the process of colliding, none of the posts were broken but there were significant deflections of some of the posts as they were pushed back in the soil. Because nonlinear springs with no elastic unloading of the springs after deformation were used to simulate the post-soil interaction so groundline deflections measured after the impact in the simulation are actually the maximum dynamic groundline displacements of the posts. As a result, the deflections deduced from the simulation were expected to be slightly higher than those from the physical test.

5.3 Splice failures

The guardrail occasionally ruptures at the splice since a plastic hinge[4] is developed at the downstream column of splice holes where the rail is bent around the post. When the post-rail connection failed, the w-beam started to slide up against the edge of the post flange. As the rail is being bent around the post, stress concentrations develop on the back layer of w-beam around the column of bolts on the down-stream side of the splice connection (Fig. 5).

A plastic hinge was developed at the cross-section through the four right splice bolts and the w-beam was somewhat folded around the post at this location. The sharp edges of the splice bolt and splice nuts or the sharp edge of the post flange initiates a tear at the stress and effective plastic strain concentration. The tear propagates through the cross-section of the w-beam. The problem is the splice bending around the post. The most straightforward way to solve this problem is to move the splice to the mid-span between posts where it will be loaded more in tension than bending.

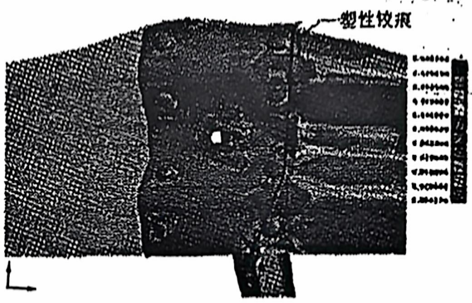

Fig. 5 Plastic Hinge

5.4 Post-rail connection

The post-rail connection is supposed to fail in the early stages of the crash so that the guardrail is released from the post instead of being pulled down with the post. When the rail detaches early in the impact event it will remain in contact with the fender of the vehicle and be more effective in redirecting the vehicle. If the connection fails too late, the rail will be pulled to the ground along with the post and this may result in the vehicle vaulting over the barrier.

The desired performance of this connection is for the bolt to fail at a low force level and for the failure to happen quickly. The failure mechanism also needs to be consistent and not sensitive to the direction of loading. The existing connection exhibits several different failure mechanisms (e. g. , nut stripping threads, pulling the washer through the slot and bolt failure). The loads that cause failure are variable and the displacement that causes failure is relatively large. For these reasons, the existing connection is not considered to result in good impact performance.

6 Results and Validation

The fidelity and accuracy of the simulation can be evaluated both qualitatively and quantitatively. In general, qualitative evaluation examines the comparison between the test and simulation in terms of general crush profile in the impact zone, crash characteristics of main components in the model. While quantitative evaluation focuses on the comparisons of acceleration and impact load of various positions of the vehicle. Fig. 6 shows the results of simulation and experiment. The simulation and test showed good agreement for the deformation in Fig. 6.

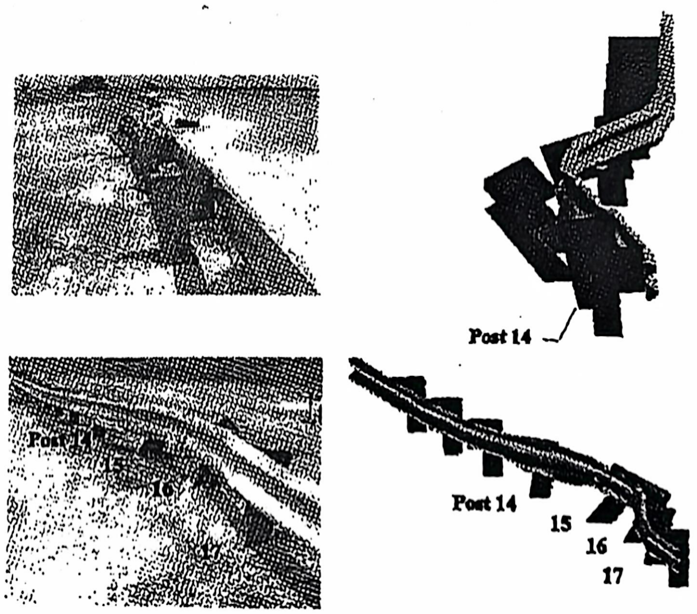

Fig. 6　Deformation

The acceleration curves are shown through Fig. 7 to Fig. 9. The simulation shows consistent results compared with the crash test. The simulated overall crash profile of the impacted section matches that of the test well. The magnitude of the accelerations and general trend and timing of the acceleration curves of simulation on the vehicle compared very favorably with those obtained from the test.

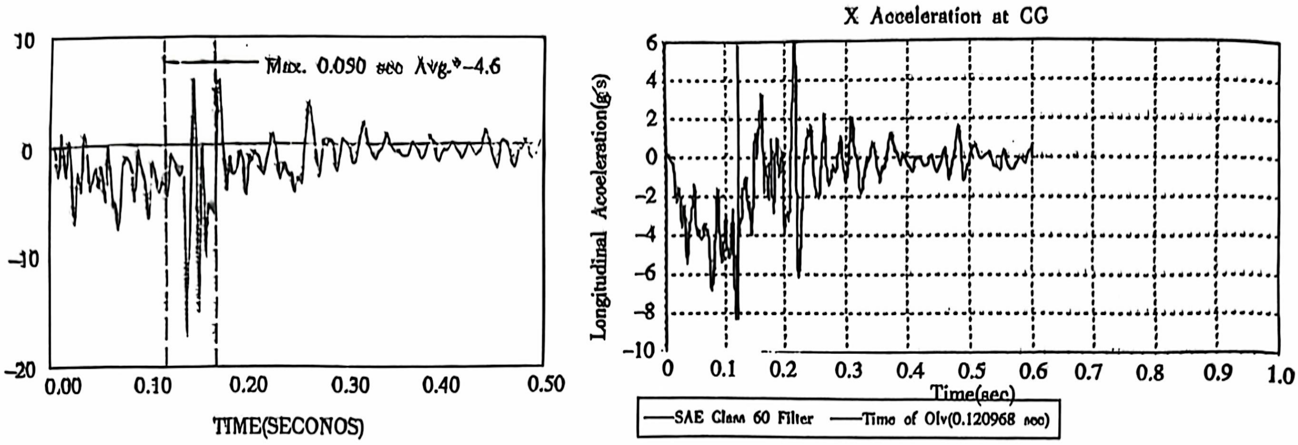

Fig. 7 X Acceleration at CG (Left-Test Right-Simulation)

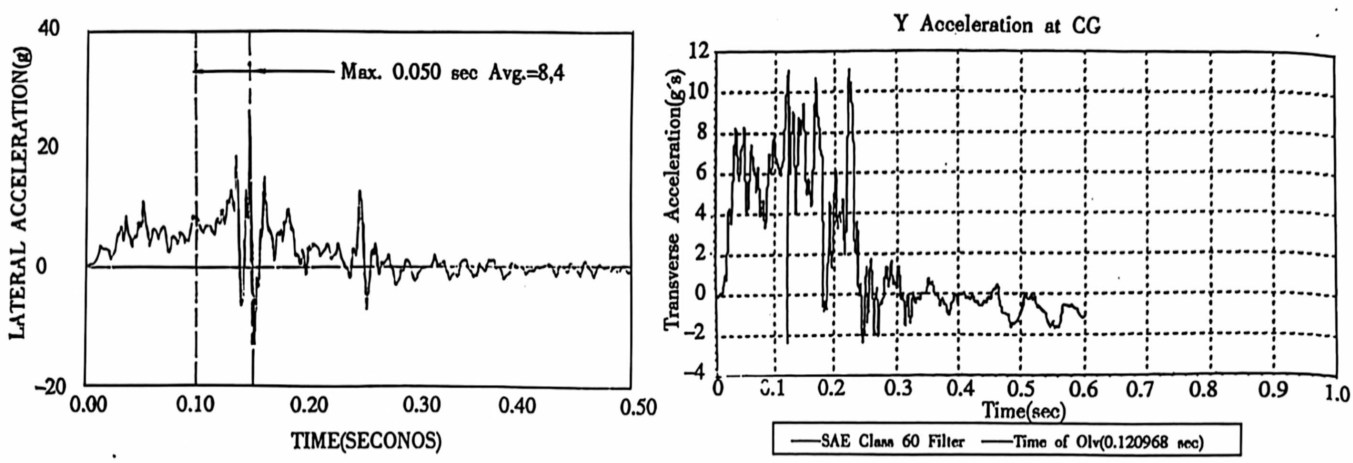

Fig. 8 Y Acceleration at CG (Left-Test Right-Simulation)

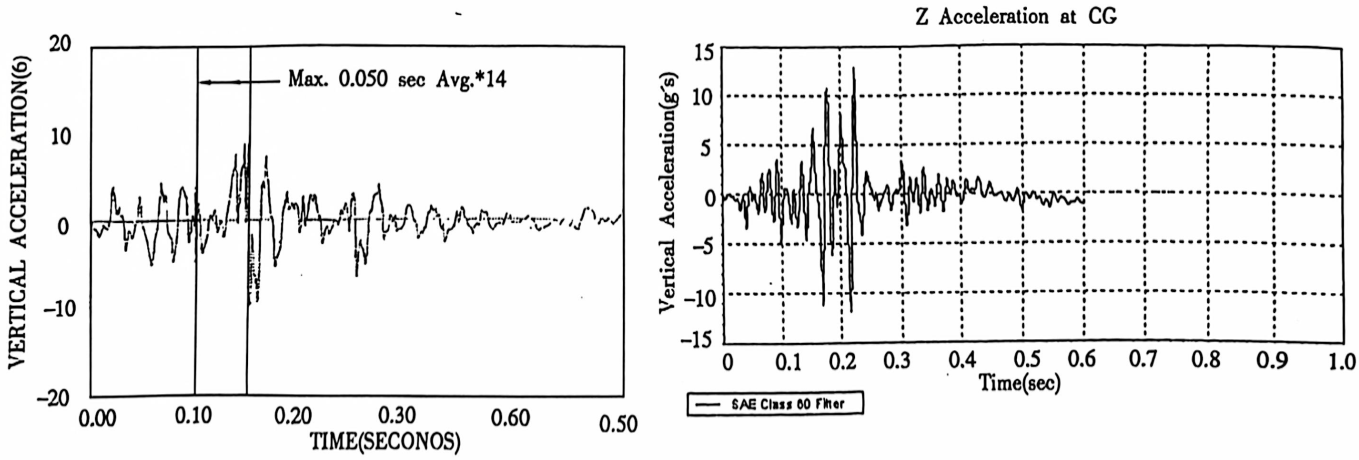

Fig. 9 Z Acceleration at CG(Left-Test Right-Simulation)

7 Remarks and Conclusions

The full-size experiment is still the most important method used to verify roadside establishments, but it is impossible to do experiments under all conditions.

With the improvement of the precision and the stability of computer simulation, it is valuable for researchers to use computer models for identifying the problems that exist in original design and thus for further finding the corresponding solution.

This paper carried out a preliminary study for the establishment of finite element model and for investigation of dynamic responses of car-guardrail crashes.

The results have demonstrated efficiency of the method that could be used to improve the guardrail design and to perfect the structure and performance of highway roadside furniture.

The purpose of guardrail design research is to prevent occupants from injuries in accidents. At present, some of the relationships between the impact dynamic responses and the injuries are still not clear (e. g. the relationship between the dynamic response of head and the injury mechanism and severity). The correlation of injury related parameters with occupant injuries found in accidents should be identified in further study. The production and R&D of automobile and roadside facility can be directed with more information from such investigations in the future.

References

[1] Varadappa, S., Shyo, S. C., and Mani, A. Development of a passenger vehicle finite element model [R]. final report DOT HS 808 145, Department of TransportationNovember 1993.

[2] J. O. Hallquist, D. W. Stillman. Nolinear Dynamic Analysis of Structures in Three Dimensions [C]. Livermore Software Technology Corporation, April 1994.

[3] Chuck A. Plaxico, fausto Mozzarelli, Malcolm H. Ray. Tests and simulation of a w-beam rail-to-post connection[C]. 80^{th} Annual Meeting of the Transportation Research Board Washington, D. C. January 2001.

[4] Klas Erik Engstrand. Improvements to the weak-post W-beam Guardrail[D]. 80^{th} Annual Meeting of the Transportation Research Board Washington, D. C. January 2001.

[5] Malcolm H. Ray, Chuck A. Plaxico and Klas Engstrand. Performance of W-beam splices[C]. 80^{th} Annual Meeting of the Transportation Research Board Washington, D. C. January 2001.

[6] Als Tabiei. Validated crash simulation of the most common guardrail system in the USA[C]. International LS-DYNA Conference Crashworthiness, 2000, (6): 3—13.