Finite Element Modeling of Impact between Bus and Roadside Guardrail

ZHANG Fan[1,2], WANG Dazhi[1], ZHOU Qing[1], ZHANG Jinhuan[1], WANG Chengbiao[2], ZHANG Ying[3], BAI Shufeng[3], LIANG Yaping[3]

[1]. *State Key Laboratory of Automotive Safety and Energy, Tsinghua University, Beijing, 100084*

[2]. *China University of Geosicences (Beijing), Beijing, 100083*

[3]. *Beijing Zhonglu'an Traffic TechnologicaCo. Ltd, Beijing, 100071*

Abstract – Roadside guardrail on highway is an essential protecting device for preventing vehicles from accidentally running off the road. The guardrail should provide protection for both passenger cars and large vehicles such as buses and trucks. Finite element modeling is an effective and efficient design tool for simulating impact between guardrail and vehicle. A finite element model of 10 m long bus was developed for simulating the impact between the bus and a concrete guardrail. The required regulation test is an angular impact, in which the bus impacts with the guardrail at 20 degree and 80 km/h. Unlike a car crash test that typically lasts about 0.1 seconds, the vehicle-guardrail crash test needs to be monitored for one second or longer. This is a great challenge to the bus modeling in terms of balancing the structural details and computational efficiency. To better capture the impact response, the bus suspension, the tire rotation, the steering, and the tire pressure are also modeled. The simulation correlates well with the test results in the local deformation of the bus, and more importantly, in the post-impact trajectory of the bus.

Keywords: Highway safety, Bus, Guardrail, Impact, Finite element modeling

1 Introduction

As highways and vehicles grow rapidly in the recent years in China, fatality is increasing due to serious accidents involving with buses that breakthrough highway guardrail and run off the road. Roadside guardrails on highway are essential protecting devices and they should provide protections for both passenger cars and large vehicles such as buses and trucks. There have been several studies for development and improvement of the roadside guardrail, especially for blocking errant heavy vehicles [1] [2] [3] [4].

Although full-scale crash testing is a primary method in evaluating the performance of roadside guardrail, the cost associated with the test is extensive, test conditions are not well controlled, and test repeatability is usually not good. To accurately replicate response of full-scale crash test, nonlinear finite element analysis methods have been used as a cost effective and efficient design tool for simulating impact between guardrail and vehicle under different crash conditions. LS-DYNA [5], a nonlinear explicit finite element code, is widely used to model the dynamic impact response behavior of roadside guardrails and the involved vehicles, and its accuracy has also been proven by many projects of roadside guardrail research.

In this study, a finite element model of a 10 m long bus was developed for simulating the impact between the bus and a concrete guardrail with LS-DYNA. The CAD data of the bus is built by dismantling an actual bus and measuring its structural details. The material properties used in the model are obtained from quasi-static tensile tests using specimens cut from the dismantled bus. To better capture the impact response, the bus suspension, the tire rotation, the steering, and the tire pressure are also included in the full bus model. Meaningful impact duration may be as long as 3

Contact Email: zhouqing@tsinghua.edu.cn

seconds, and simulating such a long impact process renders a great challenge to the bus modeling. So what structural details should be included in the model and computational efficiency must be balanced. The simulation correlates well with the test results in the local deformation of the bus, and more importantly, in the post-impact trajectory of the bus.

2 Structural Model

The bus geometry is reproduced with UG NX2. Hypermesh is used for creating the finite element model of the bus and defining material models and boundary conditions. The full bus model consists of 53,970 nodes and 40,818 elements, including 26,252 shells, 12,799 beams, and 1767 solid elements. Figure 1 shows finite element model of the bus. There are three element types used in the model. The bus body frames are modeled with beam elements instead of shell elements in order to reduce computational consumption. The bus chassis and the body skin covers are modeled with shell elements. The minimum size of the shell elements is about 38mm for controlling the smallest time step. Large mass blocks, such as the engine, the gear box, the radiator and the gas tank, are modeled with solid elements as their geometrical details are not as critical as their masses in terms of their influences to the impact performance.

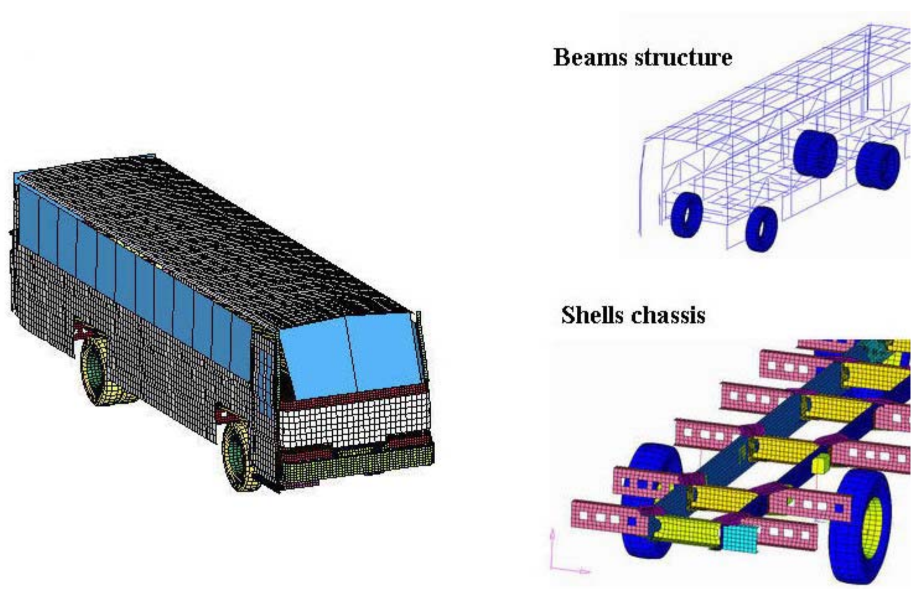


Fig. 1 Finite element model of the bus

Most materials of the bus are steels. Several specimens are cut from different characteristic parts of the bus, including the bumper, the main crossbeam of chassis and the body skin cover. Standard quasi-static tensile tests are carried out. Table 1 shows these material properties.

Table 1 Material properties of some metal materials used in bus model

Source of specimen	Thickness (mm)	Young's modulus E (GPa）	Yield stress (MPa)	Tensile strength (MPa)
Front bumper	2.2	168	201.7	310
Chassis longitudinal beam	7	203.7	470	535
Chassis 1# crossbeam	5.1	192.7	225	315
Chassis 2# crossbeam	5.1	191.4	260	305

Floor board	2.2	196.6	305	400
Side body skin cover	1.3	149.7	226.7	280

MAT_PIECEWISE_LINEAR_PLASTICITY (Type 24) material model is used for metal shell parts. Some non-essential components are not tested for material properties. In the model, properties from one of the tested materials are assigned to each of un-tested components based on similarity. For example, other chassis parts use the same material parameters as chassis 2# crossbeam material's. MAT_RESULTANT_PLASTICITY (Type 28) material model is used for beam elements as it allows for arbitrary shape beam sections. MAT_ELASTIC (Type 1) material model is assigned to those solid elements that are considered as undeformed parts.

It is assumed that bolts and spotwelds provide rigid links between deformable sheet metal parts. And therefore most connections are modeled with shared nodes between parts. MAT_RIGID material model (Type 20) is assigned to those connections that do not share common nodes. Some spotwelds in the impact zone are defined by CONSTRAINTED_SPOTWELD with failure and the joints are released if its force exceeds the failure force (normal force $S_N = 36kN$ and sheer force $S_S = 18kN$).

The shape of the concrete guardrail is L shape, which is widely used in china. It is modeled with rigid shell elements and MAT_RIGID (Type 20) is used.

3 Steering Model

To realize synchronous steering of the front wheels, two revolute joints and two sphere joints are established to connect four parts of steer machines called "steering trapezoid", as illustrated in Figure 2. Two revolute joints are at the junctures between each turning half axis and front shaft. This joint allows two parts rotate along one revolute axle that is nearly vertical to the ground with one rotation degree-of-freedom. Two other sphere joints are at the location between each turning half axis and turning link. This joint allows two parts rotate at one point with three rotation degrees-of-freedom. The front shaft fixed to the chassis links to left and right turning half axis. When left turning half axis rotates along the axis to the left, right turning half axis also rotates to the same direction as turning link keeps two half axis moving in the same way.

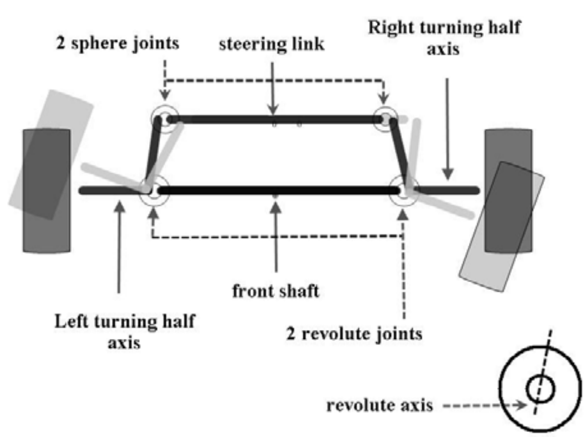


Fig. 2 Synchronous steering function

4 Suspension Model

The leaf springs are installed in the bus to connect the front and rear shafts to the chassis. They can stabilize the bus and provide resistance to pitching and rolling. Although their detail structures are complex, each leaf spring can be simplified as a combination of a linear spring and a damper. MAT_ELASTIC_SPRING_DISCRETE_BEAM (Type 74) material model allowing elastic spring combined with damping is used in suspension model. Linear stiffness and damping coefficients can be defined. Figure 3 illuminates the simplified suspension structure. To obtain the stiffness, a compression test, compressing the mid point of the leaf spring with two ends fixed, is carried out to record its force versus displacement curve. Front spring stiffness is set at 0.232 kN/mm, smaller than the rear one 0.354 kN/mm, and damping coefficients are set at 0.5 kN · s/m for both.

Another factor considered is the degree-of-freedom of the suspension movement. Cylinder joints are defined to constrain the movement between two ends of the simplified leaf spring. As a result, the shaft (part B in fig.3) can only move up and down relative to the chassis (part A in fig.3) by deforming the discrete beam.

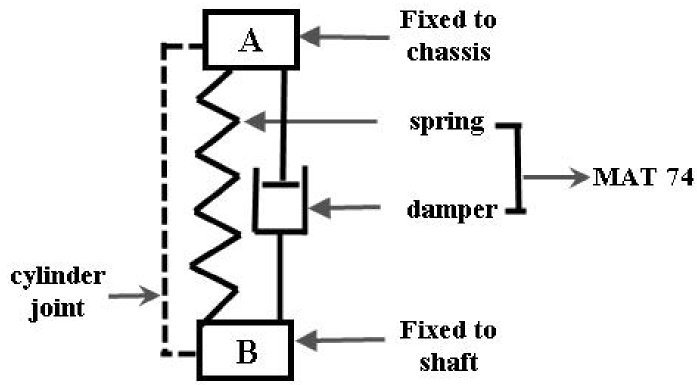


Fig.3. Simplified leaf spring

5 Tire Model

To accurately simulate tire performance, the tire pressure is represented by using SIMPLE_PRESSURE_VOLUME keyword in LS-DYNA. In this control volume card, CN is a key parameter to define the pressure of tires. To define an appropriate CN value corresponding to real tire pressure, a tire compression test is carried out to obtain the relationship between tire deformation and load force. In the test, as shown in Figure 4, standard pressure tire fixed to a setting is compressed by a jack. The average stiffness of the tire is derived from the curve of the applied load versus the tire deformation increment. The tests are then simulated repeatedly in the same loading condition using finite element tire model with different CN parameters (right picture of Figure 4). Comparing the tire responses between the test and the simulation, a CN value of 2.5e-4 (if BETA=1) is determined.

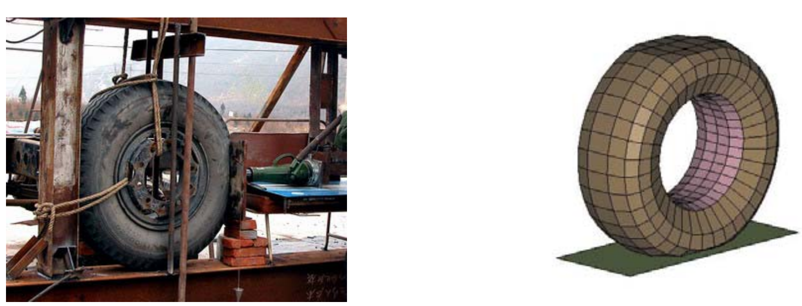


Fig. 4 Tire Compression Test and simulation

6 Description of the Crash Test

Shen Hua Da Traffic Engineering Technological Co. Ltd has performed a full-scale crash test on concrete guardrail system to evaluate its crashworthiness. In the test, a 14,000-kg bus impacted with the guardrail at 20 degree and 80 km/h. The concrete guardrail was 1000mm high and 60 m long. Within the first 0.24 seconds, the bus maintained its original traveling direction. Front structures of the bus were damaged as it absorbed some impact energy by fractures and deformation. The bus then changed its direction from the 20 degree impact and kept parallel to the guardrail till 0.48 seconds. After the tail of the bus hit the rail and deformed, the bus tilted to the left contacting the guardrail on its side. As the deformation and movement of the concrete guardrail are small, the bus was then redirected and stabilized.

7 Discussion of Results

In the simulation, boundary conditions are defined according to that of the test, including the bus C.G. location, the total mass, the impact speed and angle, and the shape of guardrail. While the termination of the simulation is set to 3 sec, the simulation results of the first second are studied and compared with that of the test in the following aspects:

a. energy analysis of the crash simulation;
b. trajectory of the bus;
c. impact velocity of the marked points on the bus top; and
d. local deformation of the bus.

These comparisons also serve as a validation of this bus model. Energy analysis is an essential way to evaluate the model accuracy. The energy balance is shown in Figure 5, including the kinetic energy, internal energy, sliding energy and hourglass energy. The hourglass energy is less than 10% of the internal energy, demonstrating a good quality of the bus model. In the test, the post-impact trajectory of the bus was recorded by high-speed cameras from the top view. For comparison, Figure 6 shows some time-sequential pictures of the full-scale crash test and the corresponding simulation. They show a good correlation in the bus trajectory between the simulation and test results. The bus tilts more in the test than in the simulation. This is in part due to that the concrete guardrail is simplified as a rigid slope in the model and thus it limits some tilt of the bus. In order to compare the reduction of kinetic energy, the velocity of the bus is used. The films from the high speed camera are analyzed using motion analysis software to obtain the time history of the displacement, from which the velocity of the marked points on the bus can be calculated. Good correlation in velocity between the test and the simulation is shown in Figure 7. Figure 8 shows that the simulated deformation in the bus impact zone is similar to that found in the test.

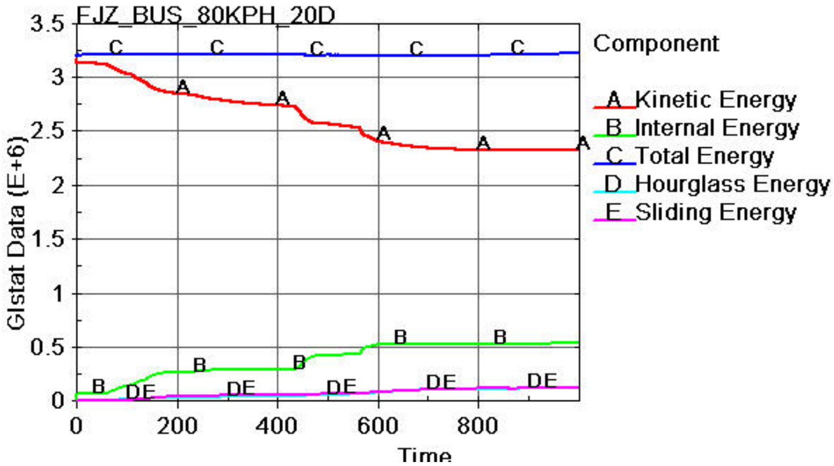
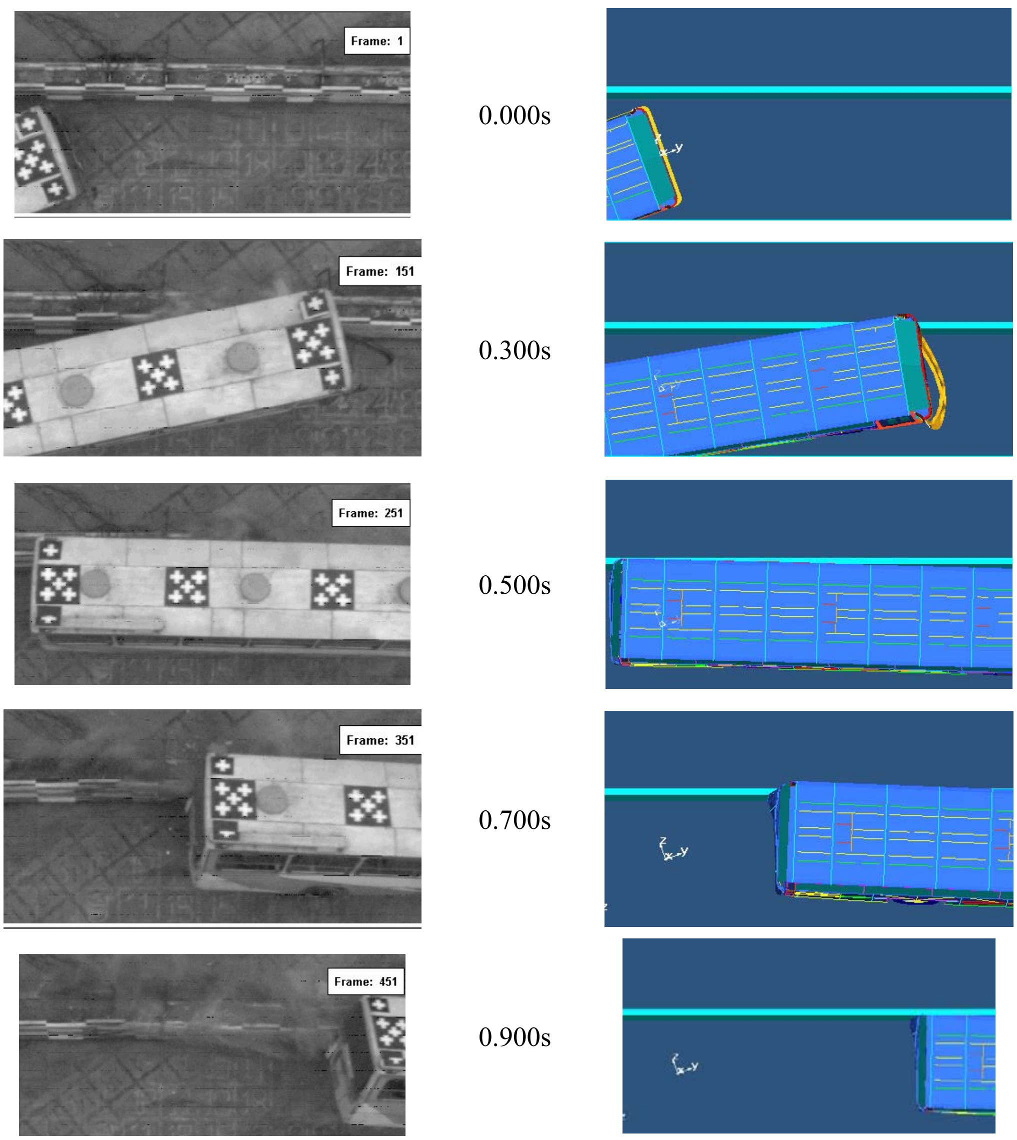


Fig. 5 Energy balance



(a) Full-scale Crash test (b) Simulation results

Fig. 6 Time-sequential picture comparison for full-scale crash test

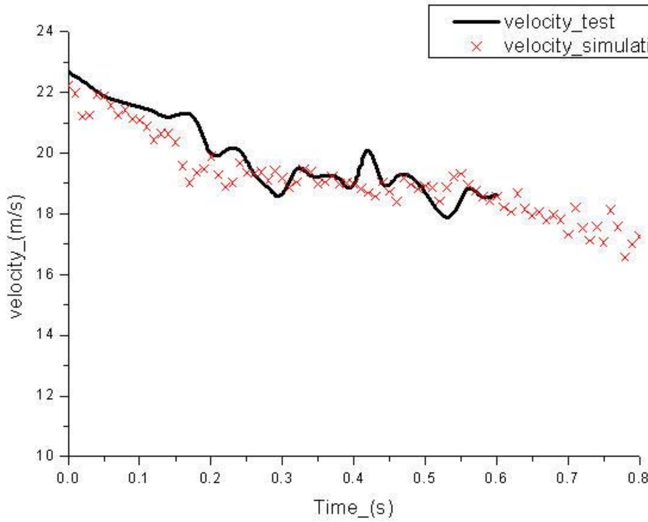
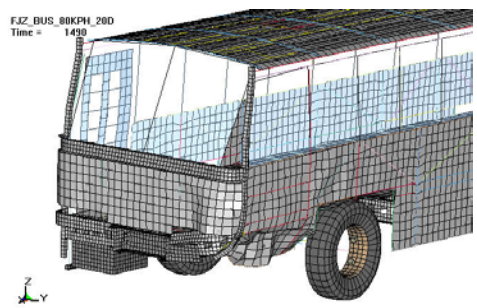


Fig. 7 Comparison of horizontal resultant velocity at bus top board



Post-test Simulation

Fig. 8 Comparison of local deformation

8 Conclusions

This paper presents methodology of building a finite element bus model for simulating a long duration impact with roadside guardrail. The model incorporates key characteristics such as steering, suspensions, tire rotation and tire pressure that are critical to the impact behaviors.

Unlike a car crash test that typically lasts about 0.1 seconds, the vehicle-guardrail crash test needs to be monitored for one second or longer. To increase computational efficiency, a main technique is to use beam elements to model the frames of the bus body. Size of the shell elements for modeling the bus chassis and other parts is also properly controlled. As a result, it takes about 8 hours running time to complete one second long simulation on a 8-processor PC cluster.

The bus model is validated using the test results of a full-scale crash test. A satisfactory correlation between the test and the simulation is found. This bus model can be used as a computational tool in further studies of roadside guardrail.

With more test results available, the model should be validated under more test conditions to gain a higher confidence level for model reliability and quality. The accelerations at the bus gravity center should also be compared.

9 References

1. Tabiei A and Wu J, "Roadmap for crashworthiness finite element simulation of roadside safety structures", Finite elements in Analysis and Design, 34, pp 145-157, 2000.
2. Tiso P, Plaxico C, Ray M, and Marzougui D, "An improved truck model for roadside safety simulations: Part II—Suspension modeling", Transportation Research Record, No. 1797, 02-3013, pp 63-71, 2002.
3. Whitworth HA, Bendidi R, Marzougui D, and Reiss R, "Finite element modeling of the crash performance of roadside barriers", International Journal of Crashworthiness, Vol. 9 No. 1 pp 35-43, 2003.
4. Vesenjak M, and Ren Z, "Improving the roadside safety with computational simulations", Forth European LS-DYNA User Conference, B-II, p 21, 2003.
5. LSTC, "LS-DYNA Keyword User's Manual", Livermore Software Technology Corporation, CA, 2003.