Application of LS-DYNA and MADYMO Coupled Model for Simulating an Offset Frontal Crash Scenario

Gunasekar T. J.
Mostafa Rashidy
EASi Engineering

Norman Ludtke
Ludtke and Associates

Bruce Spinney
NHTSA

ABSTRACT

A full vehicle finite element model of the 1995 Chevrolet Lumina validated in full frontal impact was used to simulate a frontal offset impact. The tibia index injury criteria set forth in the European Directive 96/79 were used to assess the injury indices of the occupant in frontal offset computer simulation. Based on the injury predictions from the baseline model, a structural modification to vehicle was made in order to reduce the tibia index. In the modified vehicle model simulation, the tibia index of the occupant was found to be reduced from 1.13 to 0.73 for the right tibia and from 0.62 to 0.45 for the left tibia.

INTRODUCTION

Computer Aided Engineering (CAE) analysis of crashworthiness and occupant protection involves employing full vehicle finite element (FE) models and either FE or rigid body occupant models. LS-DYNA and MADYMO coupled CAE models are two of the most popular and comprehensive tools for development of automotive crash models involving both the vehicle and occupant. Several other types of CAE models have been developed for analyzing full vehicle impacts. Some of these tend to employ FE models for representing the vehicle and the occupant (Maruthayappan, 1999), while others tend to employ rigid body representations for both the dummy and the vehicle (Deshpande, 1999). The disadvantage of the first type of models is in the computational effort involved. Typical full vehicle models can range between 100,000 to 150,000 elements, which when combined with the FE representation of the dummy can easily increase the number by another 30 to 50 thousands. In the second type of models, structural details of the vehicle have to be simplified and consequently local deformation effects are lost. The advantage of performing coupled analyses is that there is saving in the computational effort compared to those models that employ FE dummy and vehicle. At the same time, there is an added benefit of a well-established MADYMO dummy database.

Coupled analyses have been used in the past (Guha, 1997), but a major hurdle has been the lack of a pre- and post-processing tool that can handle both DYNA and MADYMO components of the model. This paper describes the application of coupled analysis to study an offset frontal impact scenario and to propose design changes to the vehicle based on the occupant injury and vehicle deformation predictions made by the analysis. It also presents some useful techniques and software tools that can rapidly create coupled models. The same tools have been used to post-process the results generated both by LS-DYNA and MADYMO.

The main objective of this study was to perform an offset frontal computer crash simulation analysis of a mid-sized car with a 50th percentile Hybrid III dummy in the driver seat. The results predicted by the analysis were assessed relative to the European Union (EU) Directive 96/79 injury reference values for the head, head/neck interface, chest, femur, knee and tibia. Structural revisions were then developed so that the EU 96/79 directive injury criteria were met.

APPROACH

Finite element (FE) model of 1991 Ford Taurus, 1995 Chevrolet Lumina and 1994 Dodge Intrepid were developed earlier (Gupta, 1999). Of these three, only the 1995 Chevrolet Lumina was still a production model and was therefore selected for offset frontal impact crash simulation. The model did not include the instrument panel, seat, steering wheel, pedals, and floor mats, airbag and restraint system. Subsystem level modeling was carried out to test the airbag, dummy coupling, seat and the restraint system. The pedals and floor mats were not modeled due to the unavailability of packaging and characteristic data as well as to eliminate variability caused by foot to pedal interaction.

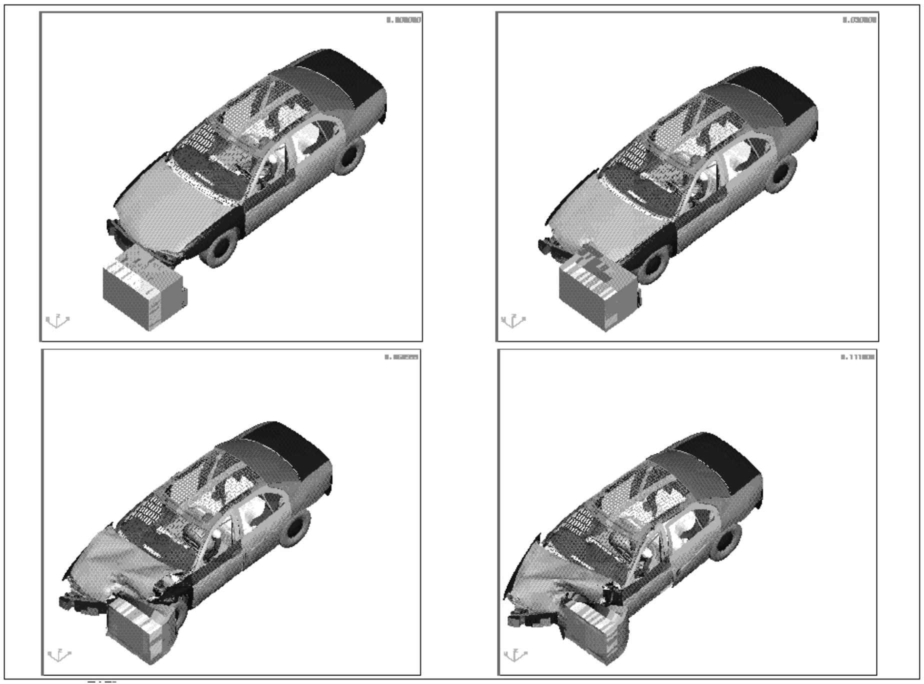

Figure 1. Animation Sequence.

EASi-CRASH™ DYNA (ECD) (developed by EASi Engineering, USA) was used to pre & post process the DYNA model. It is a fully integrated environment specifically designed for crash engineering requiring large model manipulation capability. It can read in both MADYMO and DYNA input data and output files. It combines both modules under a single graphical user interface. Input data and simulation results are read in, without the need for translators or creation of temporary files. The full vehicle simulation at discrete times for the offset impact is shown in Figure 1.

In the baseline run, because of the interaction between the deforming toepan and the foot, it was observed that the injury to the foot of the occupant was more pronounced. To reduce the lower injury criteria, specifically the tibia index, a doubler part was included in the modified vehicle simulation (henceforth termed Iteration 1 - see Figure 2) at the toepan. This decreased the intrusion in to the occupant compartment and reduced the tibia index.

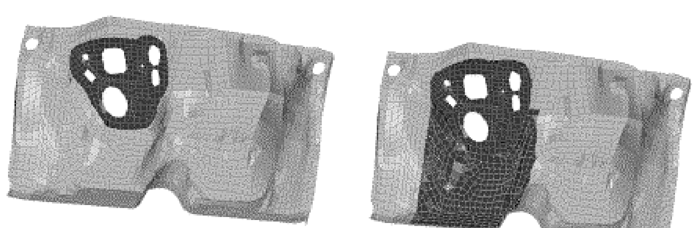

Figure 2. Change from baseline(left) to Iteration 1 (right) in the Toepan/Firewall

DISCUSSION OF RESULTS

In the frontal impact mode according to NCAP (New Car Assessment Program) regulations, vehicle crashes into a flat rigid wall at 35 mph. Hence, the deformation is spread across the parts from one side to the other. Compared to that, in the offset impact mode, the contact area is relatively smaller (40% overlap of front-end structure with the offset deformable barrier). Due to a smaller front-end contact area and higher closing speed (40 mph), the deformation is more severe in the offset frontal impact mode, which increases the intrusion into the occupant compartment. Hence, the occupant is exposed to higher risk of lower extremity injuries. In this study, coupled DYNA and MADYMO simulation has been used to predict the toe pan / firewall intrusion into the occupant compartment and compute the tibia index.

In Figure 3, the intrusion in the firewall and toepan from the baseline and iteration 1 are depicted. Although the floorpan deforms more in iteration 1, the footwell intrusion itself is reduced. Observation of the dummy foot rotation also indicates the reason why the tibia torque on the foot is higher in baseline run than in Iteration 1. In Table 1, the tibia injury number (forces, torques and tibia index) is shown. Clearly the tibia index has reduced in iteration 1 compared to the baseline run.

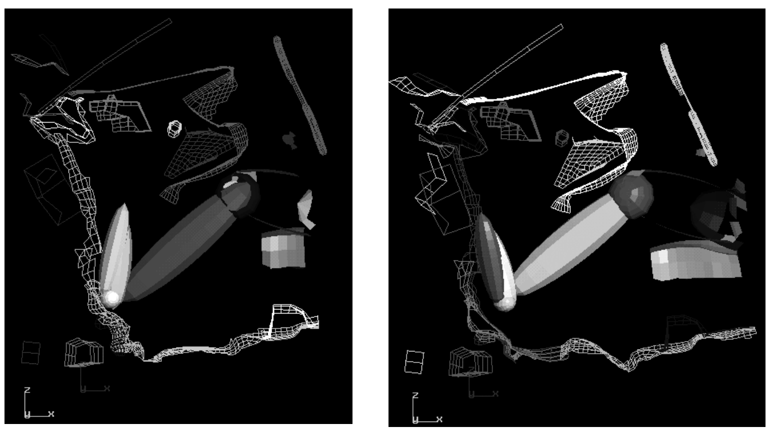

Figure 3. Animation results showing the dummy lower extremity and footwell interaction. *Left*: Baseline *Right:* Iteration 1.

Table 1. Tibia force, Tibia torque and tibia index of the occupant

	Tibia Force (N)		Tibia Torque (N.m)		Tibia Index	
	Left	**Right**	**Left**	**Right**	**Left**	**Right**
Baseline	**3308**	**5500**	**118**	**220**	**0.62**	**1.13**
Iteration 1	**3000**	**2375**	**83.4**	**150.4**	**0.45**	**0.73**

CONCLUSIONS

In this paper, a full vehicle finite element model of Chevrolet Lumina 1995 validated in full frontal impact was used to simulate a frontal offset impact situation. The injury criteria set forth in European Directive 96/79 was used to assess the injury indices of the occupant in frontal offset computer simulated crash using LS-DYNA and MADYMO. A structural modification to vehicle was made, after analysis of the baseline simulation in order to reduce the occupant injuries. In the modified vehicle model simulation, the tibia index of the occupant was found to reduced by 35% on the right and 27% on the left.

Recommendations and Future Scope

There are other factors to be looked into, which increases the scope of the analysis to come up with effective counter measures for this mode of impact.

1. Dummy Kinematics: More attention needs to be paid to the overall dummy behavior, particularly the upper extremity.

2. Shoes worn by the occupant, pedals & floor mat could influence the foot injury.

ACKNOWLEDGEMENTS

This work was sponsored by the DOT/ National Highway Traffic Safety Administration under contract No. DTNH22-96-D-12003, Project No. NPS-30-8-02069, Task Order 002.

REFERENCES

DESHPANDE, B., GUNASEKAR, T.J., MORRIS, R., PARIDA, S., RASHIDY, M., and SUMMERS, S. (1999). “Methodology Development for Simulating Full Frontal and Offset Frontal Impacts Using Full Vehicle MADYMO Models.” 1999 Intl. Mech. Eng. Congress and Exposition.

GUHA, S. (1997). “MADYMO and LS-DYNA Coupling”. North American MADYMO Users Meet, October 1997.

GUPTA, V., KAMARAJAN, J., GUNASEKAR, T.J., RAO, A., SUMMERS, S. (1999).“Reverse Engineering Method for Developing Passenger Vehicle Finite Element Models”. SAE1999-01-0083, SAE Intl. Congress & Exposition, March 1999,

MARUTHAYAPPAN, R., SAMAHA, R.R., RAO, A., GUPTA, V. (1999). “Improved Finite Element SID for In-Vehicle Simulation”. SAE1999-01-0716, SAE Intl. Congress & Exposition, March 1999,

European Directive 96/79/EC OF THE EUROPEAN PARLIAMENT AND THE COUNCIL on the protection of occupants of motor vehicles in the event of a frontal impact and amending Directive 70/156/EEC