DEVELOPMENT AND VALIDATION OF AN NCAP SIMULATION USING LS-DYNA3D

Dhafer Marzougui, Cing-Dao Kan, and Nabih E. Bedewi

FHWA/NHTSA National Crash Analysis Center
The George Washington University
Virginia Campus
Ashburn, Virginia 22011
USA

ABSTRACT

This paper describes a finite element computer simulation of a New Car Assessment Program (NCAP) full scale crash test. The full scale test selected for this study is a *30mph* frontal impact of a 1993 Ford Taurus vehicle into a rigid flat wall. Finite element models of a Ford Taurus vehicle, a Hybrid III dummy, and a driver side airbag were combined to simulate the test. The vehicle, dummy, and airbag models are described in detail. The combined model validation against the full scale test is also presented. The combined model validation focuses on the comparison of the test and simulations in terms of crush depth in the front of the vehicle, the acceleration at different locations of the vehicle as well as the head and chest accelerations and the femur loads of the dummy. The simulation results are found to be consistent and in good agreement with the crash test data. Recommendations for further improvements of the model are also included in this paper.

INTRODUCTION

Computer simulations of vehicle collisions have improved significantly over the past few years. With advances in computer technology and non-linear finite element (FE) codes, full scale models and simulations of such sophisticated phenomena are becoming ever more possible. Finite element crash simulations have been primarily focused on the vehicle models and their crash characteristics. Recently, refined FE models of airbags and dummies have been added to the simulations. This allows direct evaluation of occupant risks and injuries using simulation data.

In this paper, the development and validation of a FE model that incorporates a Ford Taurus vehicle, a *50th* percentile Hybrid III dummy, and a driver side airbag models are presented. This FE model is used for the simulation of a frontal impact with a full rigid barrier using LS-DYNA3D crash code. A full scale crash test of a Ford Taurus impacting a frontal flat barrier at *30mph* speed was used to validate the combined model. The model validation concentrated on the quantitative comparison of the test and simulation in terms of crush depth in the front of the vehicle, the acceleration at different location of the vehicle as well as the head and chest accelerations and the femur loads of the dummy. The simulation results are found to be consistent and in good agreement with the crash test data. Some deficiencies of the model were discussed and recommendations for the future improvement were also provided.

FINITE ELEMENT MODEL

The combined FE model used for the simulation of the NCAP full-scale crash test consists of a frontal-impact vehicle model, a Hybrid III dummy model, and a drive side airbag model. The following section describes the development and details of the three models. While the validations of the individual models are only discussed in brief, the full description of the validations can be found in their corresponding references.

Vehicle Model Description

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]. Figure 1 shows the vehicle model of the Ford Taurus. The hood was removed in Figure 1 to display the detailed mesh of the front portion of the vehicle. Initially, the model was developed and validated for a frontal impact into a flat rigid wall, consequently the frontal portion of the vehicle was modeled in details while the center and rear of the portions were modeled with a coarse mesh or beam elements. Since the central and rear portions of the vehicle do not undergo significant deformations in such impact, modeling these parts of the vehicle with coarse mesh or beam elements does effect the accuracy of the results as long a the overall mass distribution and inertia of the model are consistent with those of the actual vehicle.

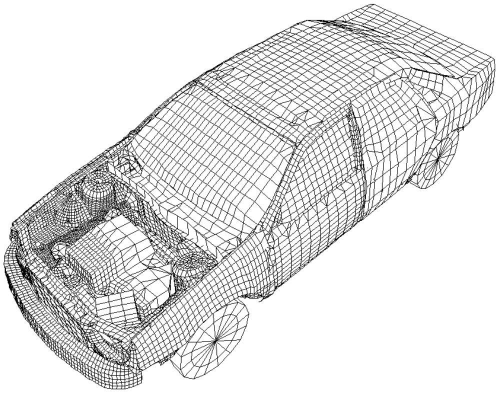

Figure 1. Vehicle Model.

The vehicle model is subdivided into *123* parts. Each part represents a component in the vehicle. Out of the *123* parts, *104* parts are used with shell elements to model the sheet metal components, *18* parts are assigned beam elements to represent the steel bars in the vehicle along with some of the connections between the sheet metal components, and one part is used with brick elements to model the radiator. Two types of shell elements are used in the model, quadrilateral and triangular. The formulation for both shell types is based on the Belytschko-Tsay shell theory which is the default shell formulation in LS-DYNA3D. The material model assigned to these shell elements is a general isotropic elastic-plastic material (material *type 24* in LS-DYNA3D). The stress-strain relation for the isotropic elastic-plastic material is defined with eight stress versus strain points. The beam elements in the vehicle model are based on Hughes-Liu beam formulation and use the isotropic elastic material model (material *type 1* in LS-DYNA3D). The solid elements, are assigned material *type 26*, metallic honeycomb, and use the constant stress solid element formulation. 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*; also known as automatic contact for beam, shell, and solid elements with arbitrary segment orientation. A summary of the FE vehicle model is listed in Table 1.

Table 1. Vehicle Model Summary.

Number of Parts	*123*
Number of Nodes	*26,741*
Number of Shell Elements	*27,874*
Number of Brick Elements	*341*
Number of Beam Elements	*140*

Several modifications were made to improve the vehicle model crash characteristics. The modified model was validated for several crash configurations using full scale crash test data[2,3]. In this application, the model was further modified in order to incorporate the dummy and airbag models. The first modification involved adding a driver side seat and a dashboard for the purpose of accommodating the dummy. The geometry of these two components were obtained from a Ford Taurus vehicle. The newly added components were connected to the rest of the model components by merging coincident nodes and using rigid body constraints. The second modification consisted of replacing the original steering column, which was modeled with beam elements, with a more realistic one. The new steering column was connected to the rest of the model using the rigid body constraint option. The third modification entailed adding seven accelerometer models at the same locations and with the same orientations as of full scale test accelerometers.

<u>Dummy Model Description</u>

The dummy model used in the simulation is based on a *50th* percentile Hybrid III. The model was developed and validated at the National Crash Analysis Center for application of injury assessment[4] as shown in Figure 2. The components of the thorax and neck assemblies are modeled with flexible parts. The other components of the dummy are modeled with rigid body parts. The geometry of the different components of the dummy was obtained from design drawings of the Hybrid III. The overall mass as well as the mass and inertia of each component of the dummy match those of the 50th percentile Hybrid III.

The joints at the ankles, knees, hips, shoulders, elbows, and wrists are modeled with spherical or cylindrical rigid body joints. Each rigid body joint is given properties that simulate those of the actual dummy. Table 2 shows the model information of the dummy.

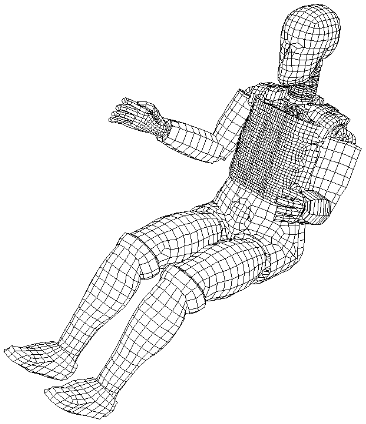

Figure 2. Dummy Model.

Three experimental tests were used to validate the dummy model. The first test was a chest pendulum test and was used to validate the thorax assembly. In this test the dummy thorax was impacted with 150 mm diameter pendulum. The pendulum mass was 23 kg and its initial speed was 6.7m/s. The chest accelerations and deflections from the test were compared to the simulation and were found to be in good agreement. The second test was a head-neck pendulum test and was used to validate the dummy neck. In this test, the head an neck of the dummy were mounted at the free end of the pendulum. The pendulum was given an initial speed then subjected to a sudden stop. The neck and head responses were recorded. The third test was a full scale frontal impact into a flat wall test. This test was used to validate the overall kinematics as well as the chest and head accelerations of the dummy model.

Table 2. Dummy Model Summary.

Number of Parts	*50*
Number of Nodes	*14,200*
Number of Shell Elements	*7,576*
Number of Brick Elements	*4,479*

<u>Airbag Model Description</u>

The airbag FE model was based on a Ford Taurus airbag. It was developed at the National Crash Analysis Center for the purpose of evaluating the interaction between the airbag and out-of-position occupants[5]. The airbag chamber, the flaps, and steering wheel were digitized from a Ford Taurus vehicle. The airbag fabric was folded in a similar manner as the actual airbag using the PATRAN pre-processor. The airbag fabric folding pattern included 12 folds and a total of 56 layers of fabric stacked on top of each other . Special care was taken to ensure that there is no penetration between these layers especially at the corners where two or more folds intersect. Figure 3 shows the airbag model at three different stages of deployment.

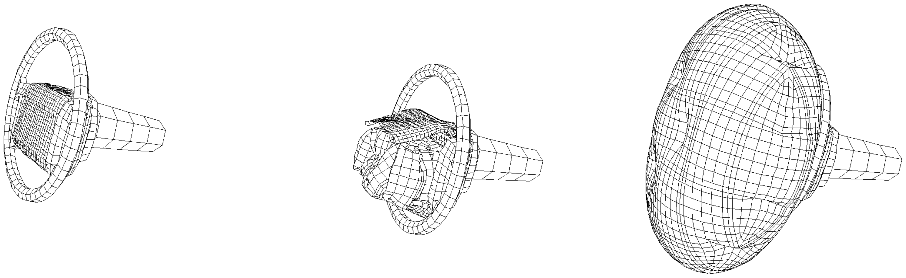

Figure 3. Airbag Model.

The airbag fabric was modeled with membrane shell elements. The elements were assigned LS-DYNA3D material *type 34* which is specifically formulated for fabric material under large deformations. The steering wheel and airbag chamber where modeled with shell elements and were assigned elastic materials. The airbag gas dynamics are modeled with the simple control volume formulation in LS-DYNA3D which is based on Cauchy's first law of motion and ideal gas laws. The inflator characteristics are defined by a curve of the input mass flow rate versus time.

The FE airbag model was validated against a test of an actual deployment of a Ford Taurus airbag. The validation consisted of comparing simulation pictures to the test high speed camera pictures at different stages of the deployment. In addition, motion analysis of the high speed film was performed to approximate the airbag volume at different times of the deployment. The motion analysis data and the simulation results correlate well.

Table 3. Airbag Model Summary.

Number of Parts	*8*
Number of Nodes	*4,724*
Number of Shell Elements	*4,588*

VEHICLE CRASH TEST

The finite element model presented in this paper was validated against a frontal impact full scale crash test. The test was performed by the Transportation Research Center Inc. for the U.S. DOT National Highway Traffic Safety Administration under contract No. DNTH22-90-C-21003[6]. The test involved a 1993 Ford Taurus 4-door sedan vehicle impacting a flat rigid wall. The vehicle gross weight was *3799 pounds* (*1725kg*) and its initial speed was *30mph* (*48.3km/hr*). Seven accelerometers were mounted on the left rear seat cross-member, right rear seat cross-member, top of the engine, bottom of the engine, right brake caliper, left brake caliper, and center of the instrument panel to measure the longitudinal accelerations of the vehicle at these locations. The vehicle included two *50th* percentile Hybrid III dummies in the front driver and passenger seats. The dummies were restrained with only the airbags, the seat belts were unbuckled. Both dummies were instrumented with accelerometers at the head and the chest and with load cells at the right and left femurs. A SAE *60Hz* filter was used to process all the measurements of the crash test.

COMPUTER SIMULATION

The simulation was performed on a Silicon Graphics Power Challenge XL super computer with *10* parallel processors. The Symmetric Multi-Processor (SMP) version of LS-DYNA3D, version 936, was used. The case was run for *150* milliseconds of simulation time on 4 processors with a fixed time step of 1 microseconds. The total computation time required for the run was *35* hours. LS-DYNA3D accelerometer models were placed at seven locations in the vehicle as well as the dummy head and chest similar to the full scale crash test. In addition, the loads from both dummy femurs were outputted for comparison with the full scale load cell measurements. The simulation output data were also processed using a SAE *60Hz* filter. Figure 4 shows a side view of the model at different stages of the simulation. A far and close-up views are shown side by side at each simulation stage.

MODEL 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, and post-crash rigid body motion of the vehicle and dummy, while quantitative evaluation focuses on the comparisons of acceleration and impact load of various positions of the vehicle and dummy. The qualitative and quantitative comparisons are discussed in the following two sections. In the first section, the vehicle data collected during the test is compared to the corresponding results from the simulation. In the second section, the dummy full scale test data are compared to those obtained from the simulation.

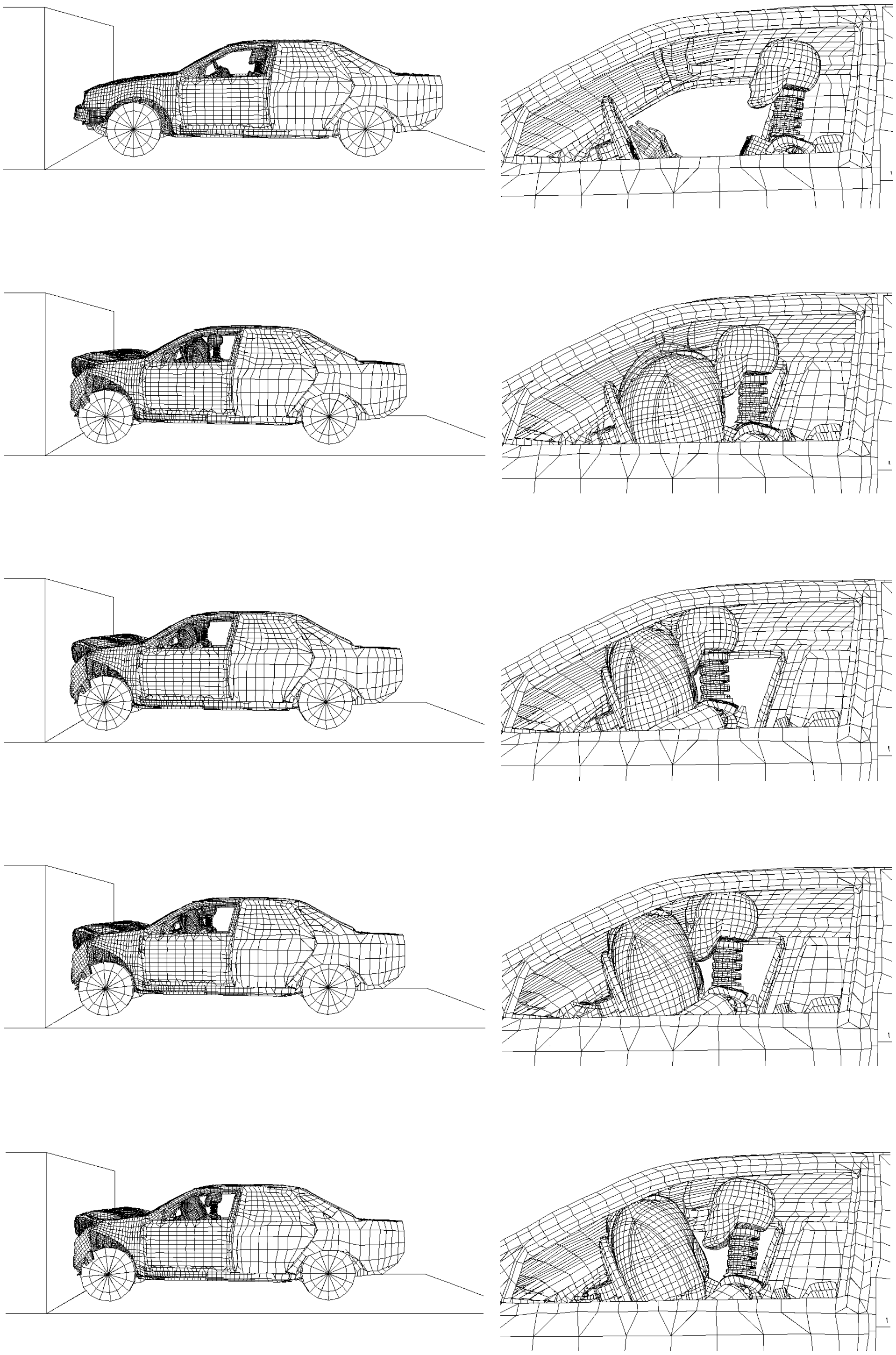

Figure 4. Crash Output Results at Different Stages of the Simulation.

Vehicle Data Comparisons

The frontal crush depth of the vehicle is measured at six different locations along the frontal profile of the vehicle as shown in Figure 5. The comparison between the test and the simulation output of crush distance for these respective positions is listed in Table 4. It can be observed that while the crush distances of the front for simulation is a bit longer than that of the test the overall crush profile of the simulation shows consistent good results in comparison with the test. The comparisons of acceleration of the test and simulation at the locations of test instrumentation are showed from Figures 6 to 12, respectively. For seat and break caliper positions (both left and right), the simulation shows excellent agreement with the test with respect to peak values and timing of the curves. For the engine, both test and simulation shows good agreement for the general trend of the curves while the first peak values (for both top and bottom position) of the simulation are higher than the test which may suggest that the model is a bit rigid then it should be. As for the comparison at the center dash panel, the magnitude is compatible but the peak and the shape of the curve from the simulation lacks of consistency. This may be attributed to the fact that the model lacks of detail and addition material parts in the interior dash panel area.

Table 4. Crush Depth Comparison.

	Full Scale Test Crush Depth (mm)	Simulation Crush Depth (mm)
C1	274	288
C2	320	340
C3	312	381
C4	305	389
C5	310	339
C6	280	308

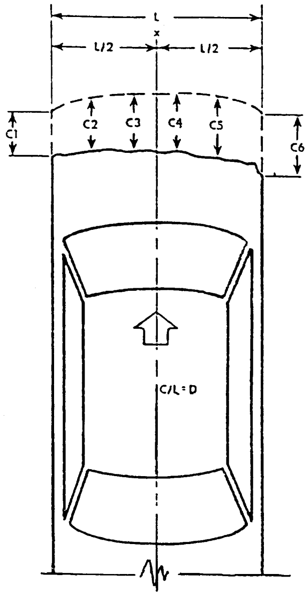

Figure 5. Crush Depth Locations.

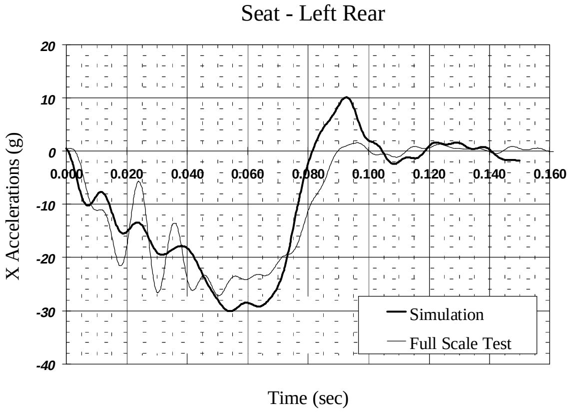

Figure 6. Comparison between the test and simulation for left seat.

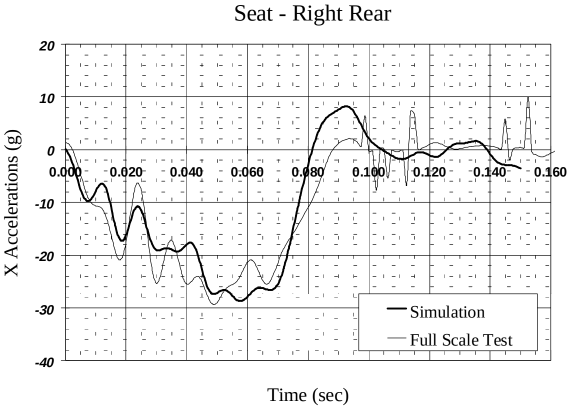

Figure 7. Comparison between the test and simulation for right seat.

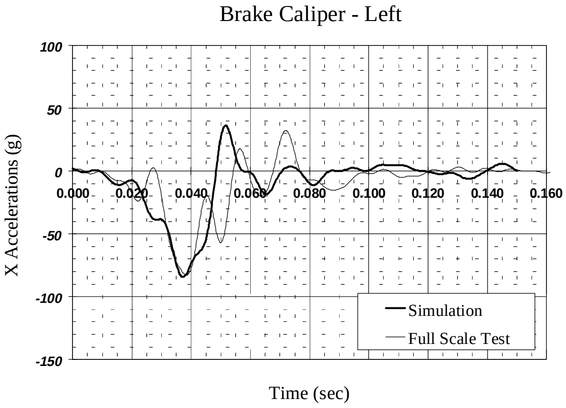

Figure 8. Comparison between the test and simulation for left break caliper.

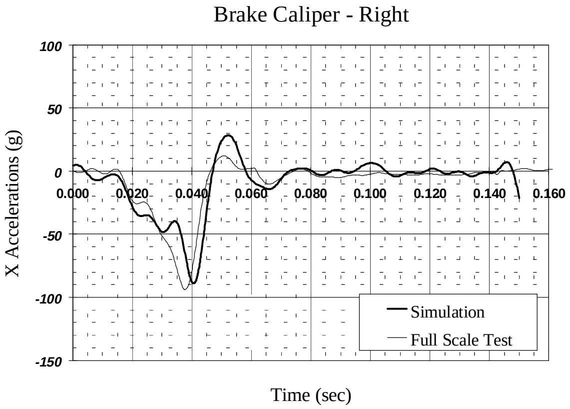

Figure 9. Comparison between the test and simulation for right break caliper.

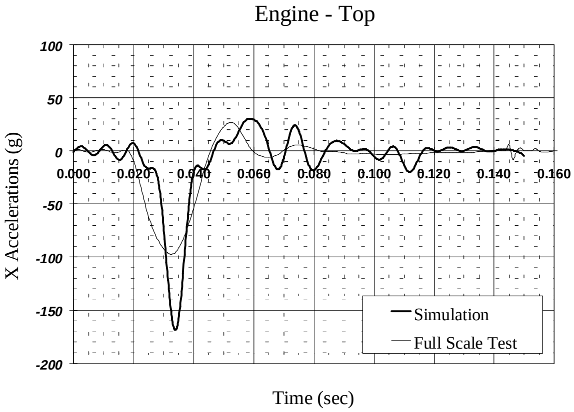

Figure 10. Comparison between the test and simulation for engine top.

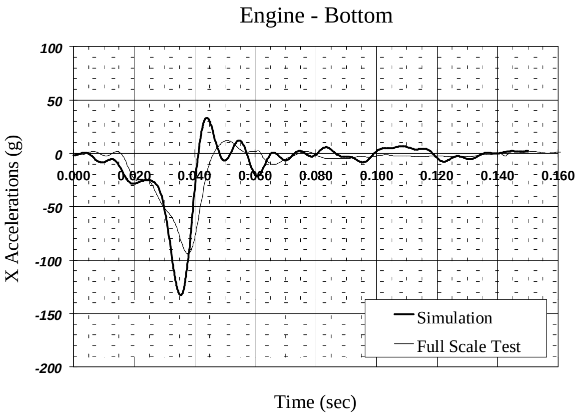

Figure 11. Comparison between the test and simulation for engine bottom.

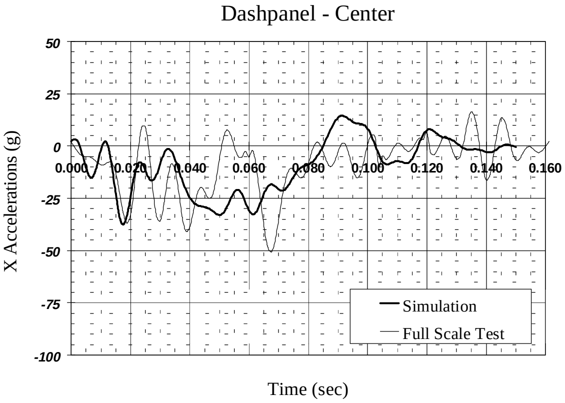

Figure 12. Comparison between the test and simulation for center dash-panel.

Dummy Data Comparisons

For the NHTSA's NCAP test, the head and cheat accelerations and femur loads of the dummy are measured. The head acceleration value is then converted into the head injury criteria (HIC) value. The comparison between the test and simulation for the peak values of HIC values, maximum chest acceleration and femur loads are shown in Table 5. Their corresponding acceleration curves are shown through Figures 13 to 16. The test and simulation showed good agreement for the dummy's head acceleration and thus the HIC values. From Table 5 and Figure 13, it can be observed that the magnitude of the simulation is lower than that of the test. The simulation curves of both left and right femur loads lead those of test and also have a lower peak values. The lead of the simulation pulse and lower magnitude in these comparisons may be caused by the placement of the dummy on the its seating position. In the current model, the seat is made of rigid material and the angle between the horizontal level and seat surface is zero. This may contribute to the earlier contact of the leg with the dash panel which resulted the lead in the simulation curves.

Table 5. HIC Value, Max Chest *g*, and Femur Loads.

	Full Scale Test	Simulation
HIC	427	422
Chest Max g	48.2 (g)	37.9 (g)
Left Femur Force	7470 (N)	5500 (N)
Right Femur Force	6490 (N)	5780 (N)

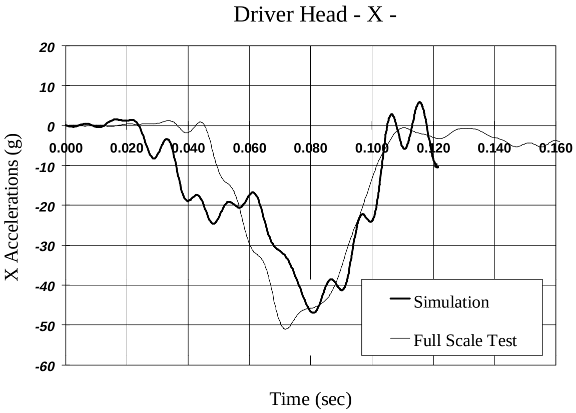

Figure 13. Comparison between the test and simulation for dummy head.

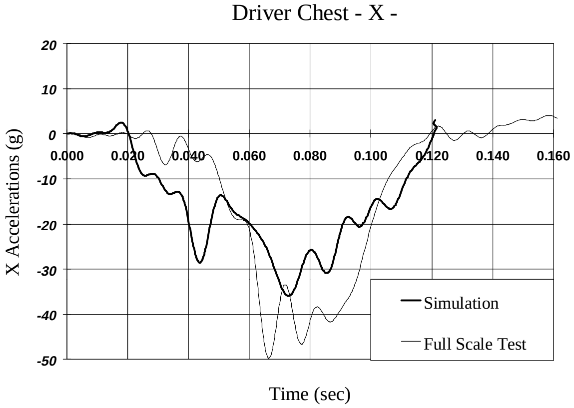

Figure 14. Comparison between the test and simulation for dummy chest.

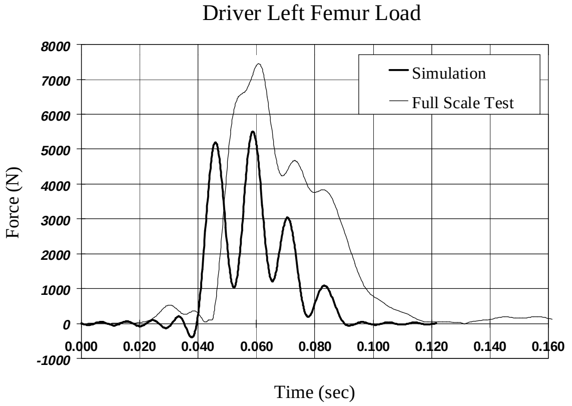

Figure 15. Comparison between the test and simulation for left femur load.

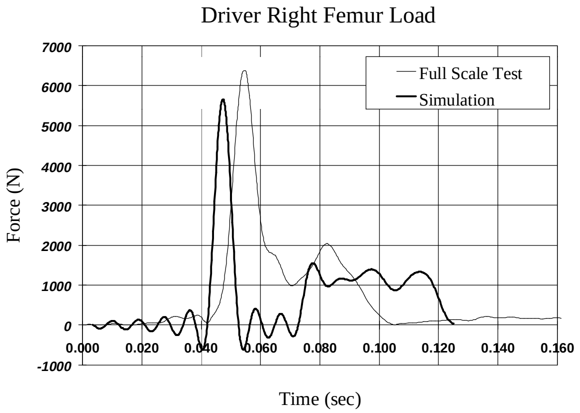

Figure 16. Comparison between the test and simulation for right femur load.

SUMMARY

The simulation shows consistent results compared with the crash test. The simulated overall crash profile of the impacted frontal section matches that of the test very 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. Observations of the crash test film and rendered simulation playback (not included in the paper) indicate that the motion and the crash characteristics of the vehicle were captured by the model. The HIC value, head and chest accelerations also show good comparison results between test and simulation. The timing of the airbag deployment in simulation was also in agreement with the test film.

However, for the model to be used for accurate injury assessment, further improvements are needed in several areas. The seat surface orientation and its contact with the dummy should be reexamined. The material model of the seat needs to be reassigned to replace the rigid material currently used. In addition, detailed parts and material model should be added around the dash panel, knee boost, and steering column area to better represent the dummy and interior crash characteristics.

As the computer technology and code development become more advanced, the used of combined vehicle, dummy and airbag models that includes detailed mesh, sophisticated material models, contact algorithms, and parts connectivity for the prediction and assessment of occupant injury and the vehicle crash performance is within the of reach of engineering community.

ACKNOWLEDGMENT

The authors wish to thank Mr. Hans Schinke for his assistant in developing the airbag model.

REFERENCES

[1] Varadappa, S., Shyo, S.C., and Mani, A., “Development of a Passenger Vehicle Finite Element Model,” Final Report DOT HS 808 145, Department of Transportation, November 1993, Washington, DC.

[2] Bedewi N.E., Kan, C.D, Summers, S., and Ragland C., “Evaluation of Car-to-Car Frontal Offset Impact Finite Element Models Using Full Scale Crash Data,” Issues in Automotive safety Technology, SAE Publication SP-1072, pp. 212-219, February, 1995.

[3] Reichert, R., “Validation of a Ford Taurus Finite Element Model for Frontal Offset Impacts” Masters thesis, The George Washington University, May 1995.

[4] Noureddine A., Digges, K.H, and Bedewi, N., “An Evaluation of Deformation Based Chest Injury Criteria Using A Hybrid III Finite Element Model,” International Journal of Crashworthiness, Vol. 1, 1996, pp. 181-189.

[5] Bedewi, N.E., Marzougui D., and Motevalli V., “Evaluation of Parameters Effecting Simulation of Airbag Deployment and Interaction with Occupants,” Accepted for publication, International Journal of Crashworthiness.

[6] Markusic, C.A., “Vehicle Safety Compliance Testing for Occupant Crash Protection, Windshield Mounting, Windshield Zone Intrusion, and Fuel System Integrity,” Final Report of FMVSS NOS. 208, 212, 219, and 301 Compliance Testing, NHTSA No. CP0203, Department of Transportation, November 1992, Washington, DC.