Development & Validation of a Finite Element Model for a Mid-Sized Passenger Sedan

D. Marzougui, D. Brown, H.K. Park, C.D. Kan, and K.S. Opiela

Center for Collision Safety and Analysis, George Mason University

Abstract

A Finite Element (FE) model of a mid-size passenger sedan was created by reverse engineering to represent that aspect of the fleet in crash simulation analyses. A detailed FE model of this vehicle was created to allow application for different types of crash scenarios. The initial version of the model includes detailed and functional representation of suspension and steering components. Material characteristics and thicknesses of the different components were determined from manufacturer's information and coupon tests so that the simulated crash behavior would reflect actual impact test results. The model mass and inertial properties were compared to measurements made on the actual vehicle. Initially, the model was subjected to a series of debugging and verification simulations to insure that all components of the vehicle are included and appropriately connected. A series of validation tests followed to compare simulated and actual crash tests. Comparisons to full-scale crash tests indicated that acceleration pulses at different locations of the vehicle, deformations in the occupant compartment, and overall vehicle kinematics are similar. Detailed representation of the vehicle interior components and restraint systems is currently being incorporated in the model to provide opportunities to use FE occupant models in the vehicle and assess injury risks.

Background

A finite element (FE) model based on a 2012 Toyota Camry passenger sedan was developed through the process of reverse engineering at the National Crash Analysis Center (NCAC). These efforts were conducted under a contract with the Federal Highway Administration (FHWA). This model will become part of the array of FE models developed to support crash simulation. The model was validated against the National Highway Traffic Safety Administration (NHTSA) frontal New Car Assessment Program (NCAP) test for the corresponding vehicle. This vehicle was selected for modeling to reflect current automotive designs and technology advancements for an important segment of the vehicle fleet. This model is expected to support current and future NHTSA research related to occupant risk and vehicle compatibility as well as FHWA barrier crash evaluation, research, and development efforts. This vehicle conforms to the Manual for the Assessment of Safety Hardware (MASH) requirements for a 1500A test vehicle [1].

Modeling

A production 2012 Toyota Camry four-door passenger sedan was purchased as the basis for the model [VIN 4T1BF1FK2CU079329]. The reverse engineering process systematically disassembled the vehicle part by part as in past efforts [2]. Each part was cataloged, scanned to describe its geometry, measured for thicknesses, and classified by material type. All data was entered into a computer file and then each part was meshed to create a computer representation for finite element modeling that reflected all structural and mechanical features in digital form.

Parts were broken down into elements such that critical features were represented consistent with the implications of element size on simulation processing times. Material characterization data for the major structural components was obtained through coupon testing from samples taken from vehicle parts. From the material testing, appropriate stress and strain values were determined to include in the model for the analysis of crush behavior in crash simulation

The resulting FE vehicle model has 1.7 million elements, without the interior components or restraint systems. This detailed FE model was constructed to include full functional capabilities of the suspension and steering subsystems. A representation of this model in comparison to the actual vehicle is shown in Figure 1.

Figure 1 – Actual Vehicle and FE Model of a 2012 Toyota Camry Sedan

The set of elements representing the vehicle was translated into an FE model by defining each as a shell, beam, or solid element in accordance with the requirements for using LS-DYNA® software [3]. The result of these efforts was a finite element vehicle model with the following characteristics:

Number of Parts - 972
Number of Nodes - 2 million
Number of Elements - 2 million

The average element size used was 6-7 mm with a minimum size of 4 mm. The modeling effort detailed all components of the vehicle. Figure 2 provides a bottom view of the vehicle model. Figure 3 shows the details of the model for the body frame and drive train for this vehicle. The uni-body frame and the door structures are shown. The engine was modeled with a coarser mesh, as simulation experience has found that it reacts as a large rigid mass in crashes. It was modeled with a solid block using hexa (brick) elements. The material density for the engine was defined such that the mass is similar to the one measured from the actual engine. The engine was assigned an elastic material (Type 1) in the model. The limp mass representation of the engine and elements of the suspension can also be noted in the model.

Figure 4 provides close-up views of the modeled front steering/suspension and rear suspension system. The moving parts were detailed to provide the capability to simulate suspension and steering responses.

All inner components of the front and rear doors were included in the initial version of the model as seen in Figure 5. Structural components of the vehicle interior were included in the initial version of the model as shown in Figure 6. The remaining interior components will be added in the next version of the model.

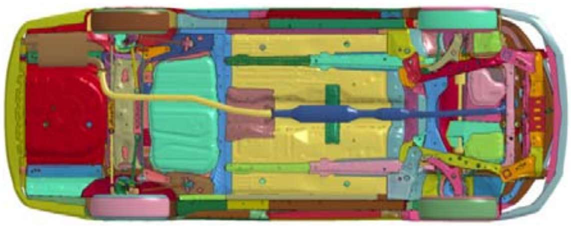


Figure 2 –Vehicle Model Bottom View

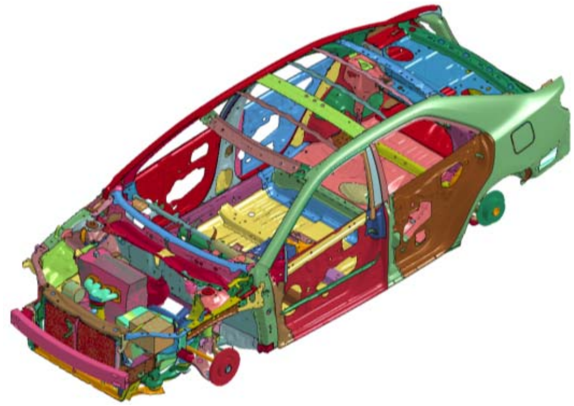
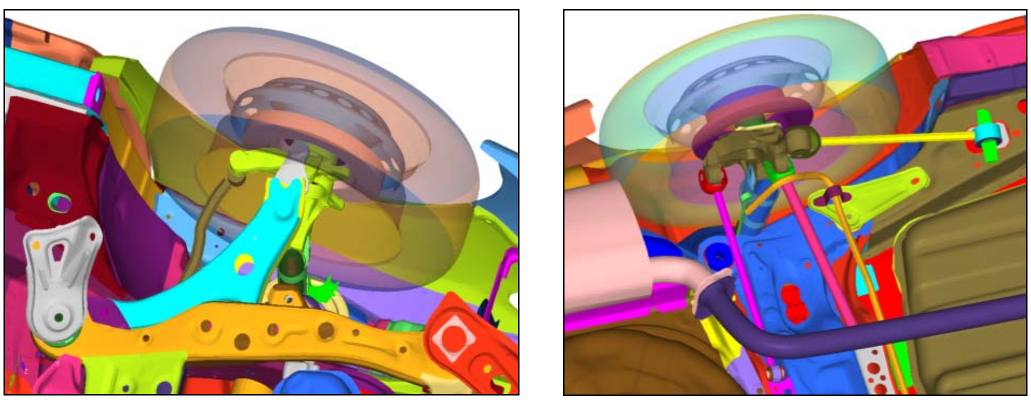


Figure 3 – Details of the Modeled Vehicle Body Structure and Drive Train



Modeled Front Suspension

Modeled Rear Suspension

Figure 4 – Details of the Modeled Steering and Suspension Subsystems

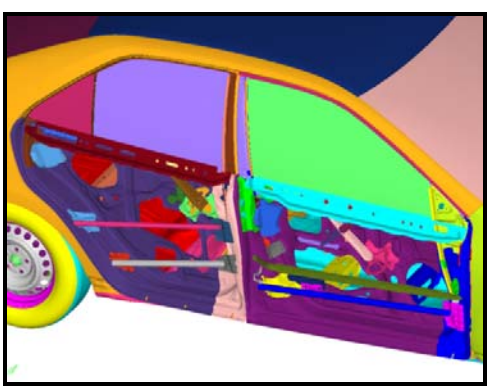


Figure 5 – Details of Interior Door Components

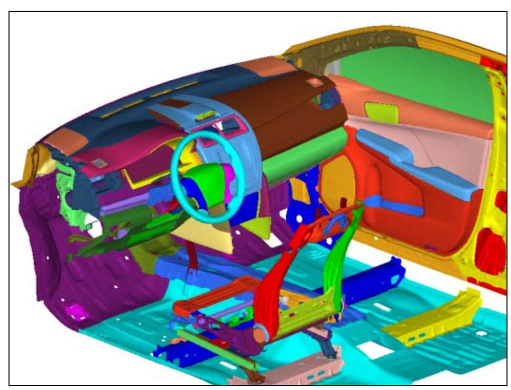


Figure 6 – Coarse Representations of Structural Interior Components

Model Validation

The FE model was verified and validated in several ways to assure that it was an accurate representation of the actual vehicle. The initial efforts included checks for completeness of elements and adequacy of connection details.

The mass, moments of inertia, and center of gravity (CG) locations of the actual vehicle were measured on a inertial table at the SEAS, Inc. lab as shown in Figure 7 [4]. The measured inertial properties for the vehicle were compared to those generated from the FE model. The results are shown in Table 1. The weight; pitch, roll, and yaw inertias; and x, y, and z coordinates for the CG were found to be similar and within acceptable limits.

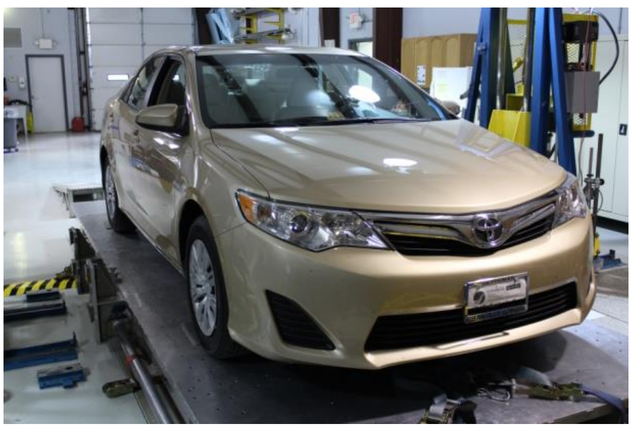
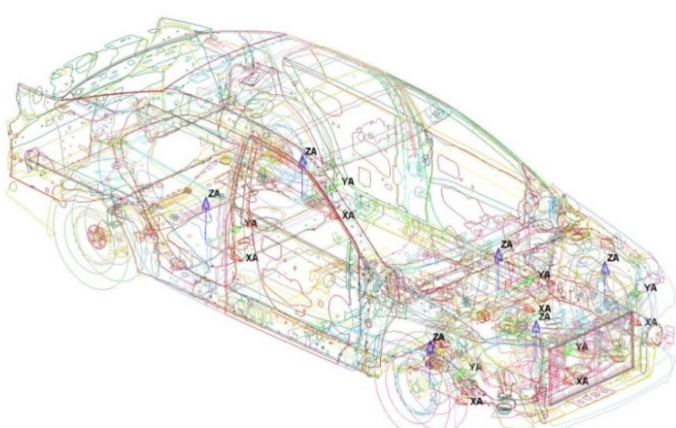


Figure 7- Actual Vehicle on Inertial Measurement Table

Table 1–Mass, Inertia, and CG Location Comparisons

	Actual Vehicle	FE Model
Weight, kg	1452	1453
Pitch inertia, kg-m^2	2519	2482
Yaw inertia, kg-m^2	2796	2752
Roll inertia, kg-m^2	560	560
Vehicle CG X, mm	1063	1088
Vehicle CG Y, mm	-9	-1
Vehicle CG Z, mm	561	559

The primary impact validation was accomplished by comparing the simulation of the NCAP frontal wall impact at 35 mph with actual data from a NHTSA NCAP test for a comparable vehicle [5]. For this simulation, accelerometers were positioned in the same locations as the NCAP test (Figure 8). The most commonly benchmarked accelerometers for NCAP performance are the left rear seat, right rear seat, engine top, and engine bottom.



Location	Node ID
Left Seat	319812
Right Seat	319820
Engine Top	319828
Engine Bottom	319836

Figure 8 – Accelerometer Locations in FE Model

The FE model NCAP simulation was performed using the LS-DYNA non-linear explicit finite element code. The FE vehicle model was run using LS-DYNA Code Version MPP971sR4.2.1 on an Intel MPI 3.1 Xeon 64 parallel computer platform. The FE model response would be expected to vary for other facilities depending on hardware, LS-DYNA version, and precision used. The variations are typically minimal and the results from the different versions are comparable. The total duration of the simulation was 150 msec to capture the initial impact until the rebound of the vehicle from the NCAP load cell wall. Approximate computation time to run 150 msec using 24 processors on the Intel cluster was less than 3 hours.

Table 2 provides specific comparisons for key parameters of the FE model and the vehicle used in the NCAP test. It is easily noted that all were very similar. More information on the NCAP test vehicle, like vehicle weight distribution, vehicle attitude, center of gravity (CG) location, and fuel tank capacity, are published in the NHTSA's test report [5].

Table 2 – Comparison of Parameters for FE Model & Vehicle Used in the NCAP Test

	FE Model	**Test 5100**
Weight (kg)	1662	1662
Engine Type	2.5L V4	2.5L V4
Tire size	P205/65R16	P205/65R16
Attitude (mm)	F – 715	F – 720
	R – 674	R – 675
Wheelbase (mm)	2774	2782
CG (mm) Rear of front wheel	1194	1184
Body Style	4 Door Sedan	4 Door

The overall global deformation pattern of the FE model was very similar to that of the NCAP test. Figure 9 shows the actual versus the simulated engine compartment damage. Figure 10 shows side view of the frontal deformations taken at three intervals during the impact. It can be noted that the actual and simulated views reflect similar deformations for the bumper, hood, fender, and wheel assembly at each time point. Figure 11 shows the underside view of the engine compartment and front suspension and steering assemblies. Similar patterns and extent of crush are noted. These visual images suggest that the FE model provides a reasonably accurate representation of an actual vehicle in a 35 mph frontal impact scenario.

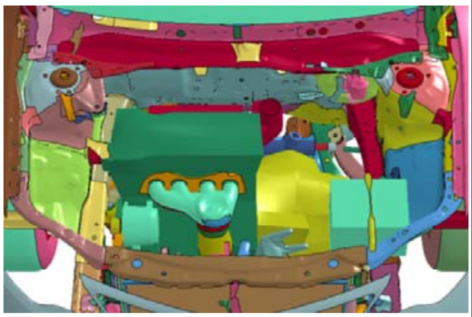
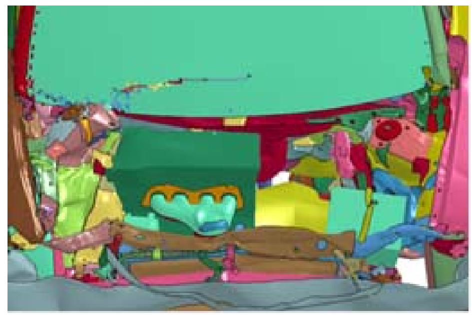
	Pre-Test	Post-Test
Actual		
Simulated		

Figure 9 – Comparison of Actual versus Simulated Engine Compartment Damage

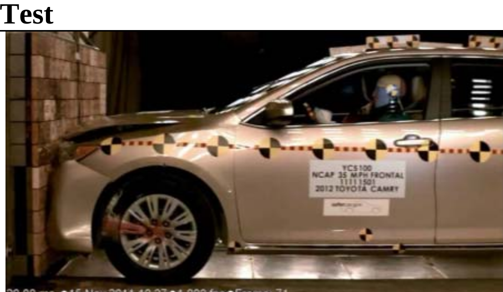
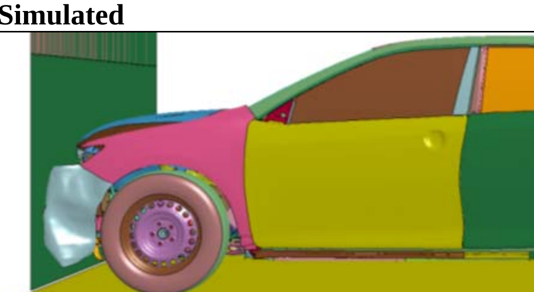
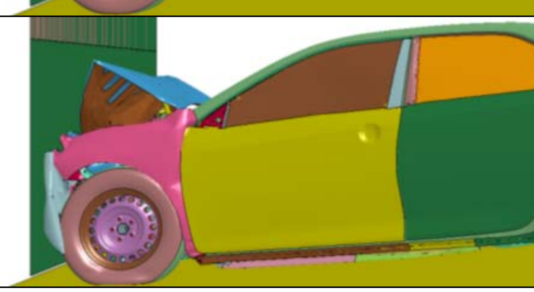
Test	Time	Simulated
	**20 msec**	
	**60 msec**	
	**100 msec**	

Figure 10 – Sequential Views of the Actual and Simulated NCAP Frontal Wall Test for the Camry

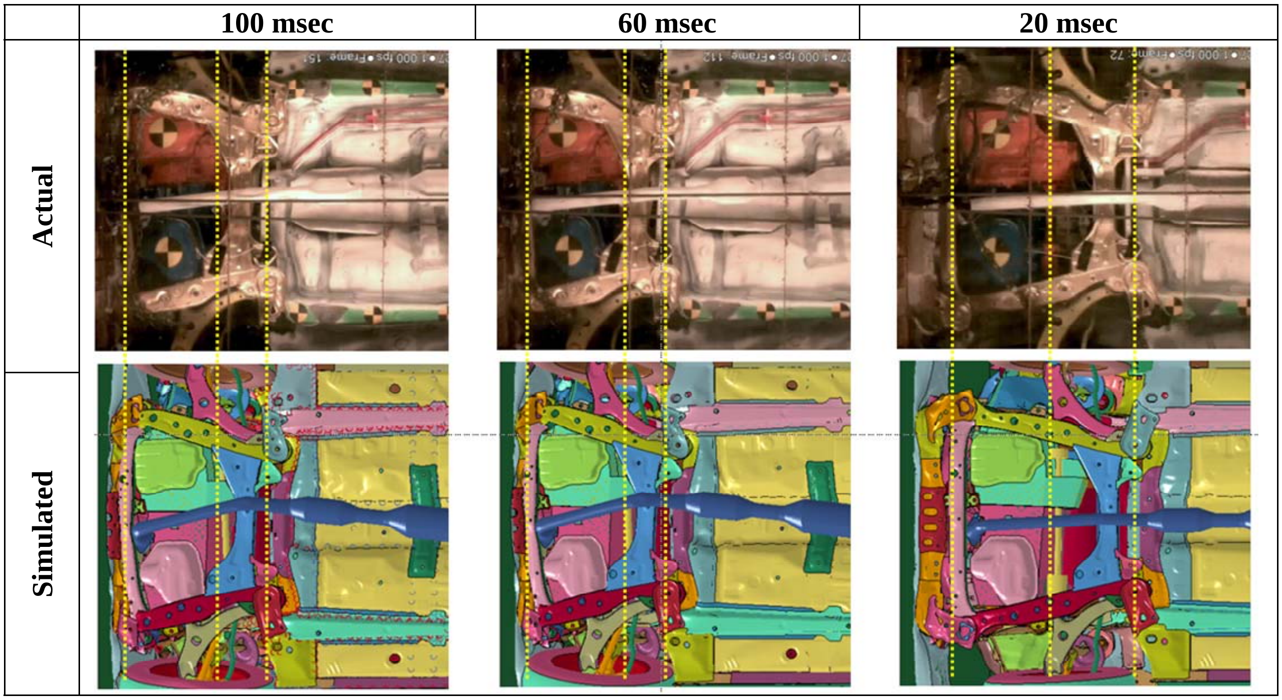


Figure 11 – Comparison of Actual versus Simulated Underside Engine Compartment and Steering Suspension Assemblies Damage

The global response of the vehicle was further benchmarked against the NCAP test data by comparing the acceleration responses from the left and right rear seat cross member accelerometer, average velocity of the vehicle, and displacement. The seat cross member acceleration plots are shown in Figure 12. The timing and shape of the peak acceleration in the tests were matched in the FE simulation. Velocity comparisons for the seat cross member are shown in Figure 13, indicating that the test vehicle velocities also compared well to the FE model. The global responses of the engine top and engine bottom accelerometers also track the responses from test vehicles as shown in Figure 14. The tests and simulation show similar acceleration pulse magnitudes. This was also the case for both the engine top and engine bottom accelerations.

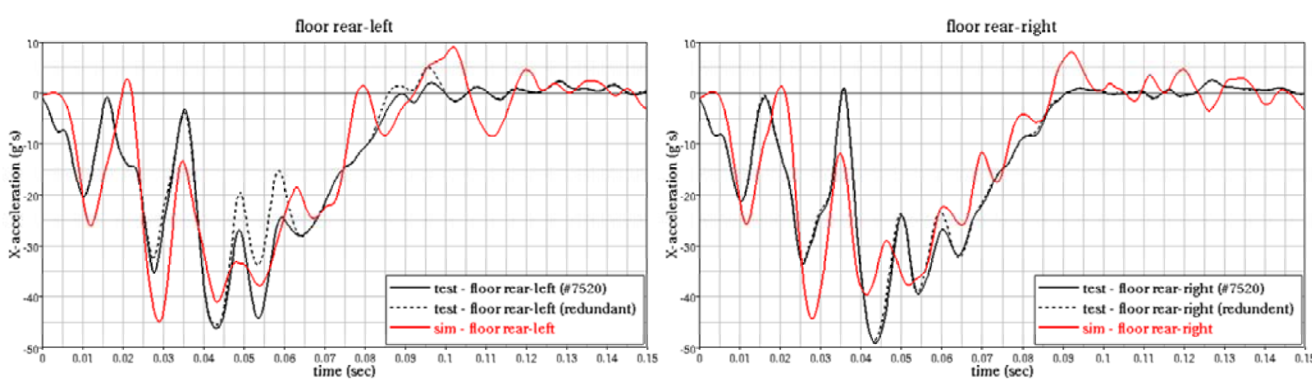


Figure 12 – Comparison of Tests & Simulation for Left and Right Rear Seat Accelerations

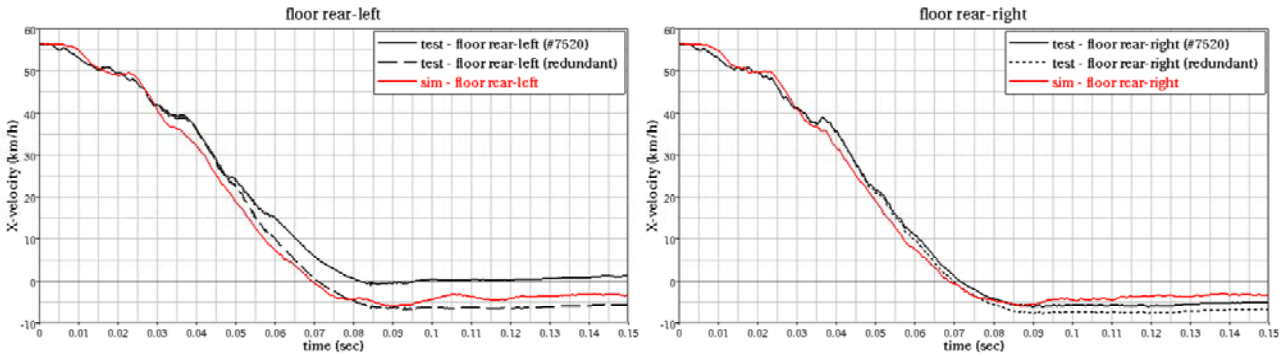


Figure 13 – Comparison of Tests & Simulation for Left and Right Rear Seat Velocities

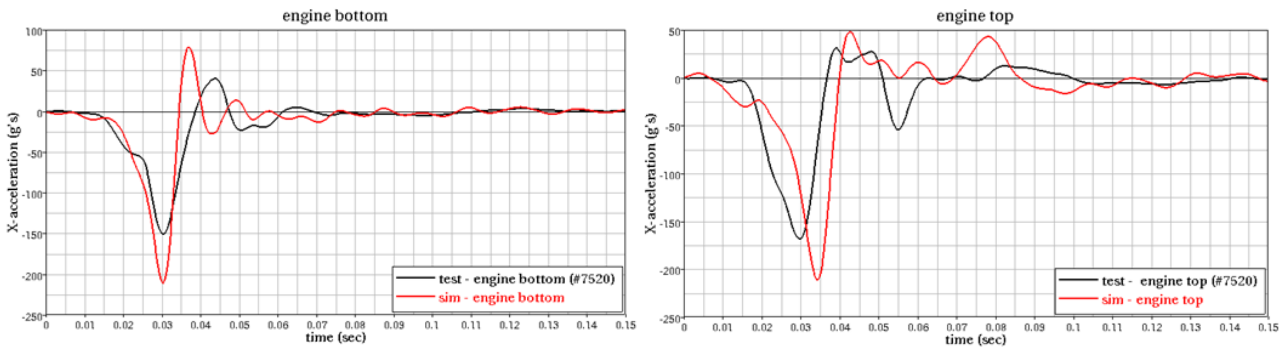


Figure 14 – Comparison of Tests & Simulation for Engine Top and Bottom Accelerations

Figure 16 shows the vehicle stiffness plots extracted from the tests and simulation. The figure shows that, overall, the vehicle stiffness from the simulation is similar to the test. Similar maximum force of ~900 kN and maximum crush of 650 mm were observed.

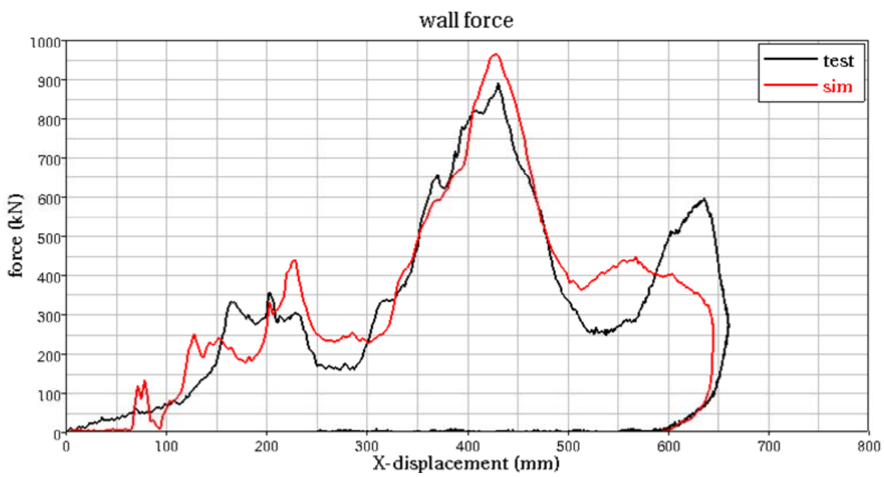


Figure 16 – Comparison of Tests & Initial Simulation Data for Force Displacement

Last, in Figure 17, the global energy plots from the simulation are provided. It can be seen that there is energy balance throughout the simulation. The simulation started with an initial kinetic energy and no external work was applied. As the simulation progressed, the kinetic energy decreased and the internal energy increased due to the impact into the wall. The total energy remained constant in the simulation as no external work was applied to the vehicle.

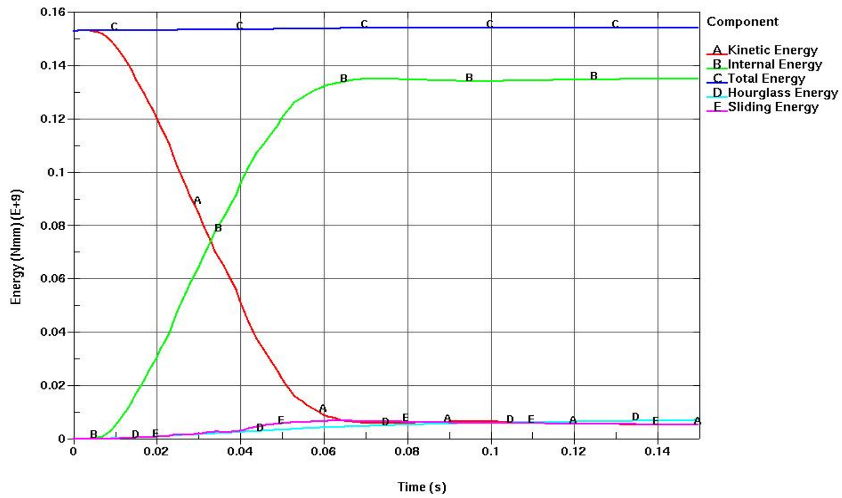


Figure 17 – Simulation Energy Balance Analyses

Extended Validations

Over the years, extended validations of the FE models were undertaken to demonstrate the capabilities of the models. Since suspension failure has been observed in several roadside hardware crash tests over the years, efforts to fully model the suspension assemblies in the detailed models was considered important to reflect impact behavior. This necessitated formulation of protocols for quasi-static and dynamic experimental tests to demonstrate similar responses to known impulses and determine the failure characteristics of key components of the suspension system (suspension joints and steering tie-rods). To validate that reasonable spring response behavior was built into the model, several "bump" and "terrain" tests are conducted with the vehicles to be modeled. As other impact tests become available, further impact comparisons are made and documented to extend the confidence in the FE model of the vehicle for various possible applications.

Summary and Conclusions

A finite element model of the 2012 Toyota Camry passenger sedan was created using a reverse engineering process by the NCAC under contract to the FHWA. This vehicle was modeled to support current and future NHTSA and FHWA research efforts. The vehicle conforms to the Manual for the Assessment of Safety Hardware (MASH) requirements for a 1500A vehicle, so it can be used for the design and evaluation of new roadside hardware. The modeling effort led to a detailed model that consisted of 2 million elements, included representation of all vehicle structural and interior components, and has functional representations of the steering and suspension systems.

The model was validated by comparison to images and data derived from the NHTSA NCAP Test , which involved frontal impact into a rigid wall at 35 mph. Comparisons of data from the test and the model included:

- View of side, engine compartment, and underside deformations,
- Acceleration and velocity changes for the rear seat cross member
- Accelerations of the top and bottom of the engine,
- Force displacement plots, and
- Total crash energy and energy balance.

Both the vehicle kinematics and the accelerometer output data were compared and the simulation results using the initial version of the model showed overall good correlation with the physical test results. These comparisons will be repeated when the final version of the model is complete. Extended validations using data from speed bump, and sloped terrain will be undertaken to demonstrate that the model can reflect these effects in impacts. Further impact comparisons including IIHS SOL, NHTSA/IIHS side impacts, and roof crush are expected to be undertaken, as well.

Acknowledgements

The authors wish to acknowledge the Federal Highway Administration (FHWA) and National Highway Traffic Safety Administration (NHTSA) of the U.S. Department of Transportation for supporting these modeling and simulation efforts.

References

1. AASHTO, Manual for Assessment of Safety Hardware (MASH), published by the American Associations of State Highway & Transportation Officials, Washington, DC, 2009.
2. NCAC, “Development & Validation of a Finite Element Model for the 2010 Toyota Yaris Passenger Sedan,” Working Paper 2011-T-001, Prepared by The George Washington University, Ashburn, VA, June 2011.
3. Hallquist, J.O.,“LS-DYNA Keyword User Manual,” Livermore Software Technology Corporation, 2007.
4. SEA, Limited, VIMF, Testing Report 4732, 2012 Toyota Camry 4 Door Sedan, Ohio, May 30, 2012.
5. MGA research Corporation, “NCAP Frontal Barrier Impact – 2012 Toyota Camry,” NHTSA Test Report YC5100, Burlington, WI, Dec 16, 2011.