George Mason University

Tech Summary

doi:10.13021/G8N88

May 2016

Development & Validation of a Finite Element Model for the 2012 Toyota Camry Passenger Sedan

Background

A finite element (FE) model based on a 2012 Toyota Camry passenger sedan was developed through the process of reverse engineering by the Center for Collison Safety and Analysis (CCSA) researchers under a contract with the Federal Highway Administration (FHWA). The model can be downloaded from the library of FE vehicle models developed to support crash simulation efforts [1]. The model was validated against the National Highway Traffic Safety Administration (NHTSA) frontal New Car Assessment Program (NCAP) test for the corresponding vehicle. 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 Assessing Safety Hardware (MASH) requirements for a 1500A test vehicle [2].

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. Each part was cataloged, scanned to define 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 of the structural and mechanical features in digital form.

The resulting FE vehicle model has 2,257,280 elements. 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 and FE Model of a 2012 Toyota Camry Sedan

Parts were broken down into elements such that critical features were represented consistent with the implications of element size on simulation processing times. Material data for the major structural components was obtained from manufacturer specifications or determined through coupon testing from samples taken from vehicle parts. The material information provided appropriate stress and strain values for the analysis of crush behavior or failures in crash simulation.

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 model with the following characteristics:

Number of Parts	- 1,086
Number of Nodes	- 2,255,361
Number of Shells	- 2,032,594
Number of Beams	- 5,901
Number of Solids	- 218,785
Number of Elem.	- 2,257,280

The modeling effort detailed all relevant components of the vehicle. Figure 2 shows the details of the model for the structural parts for this vehicle. 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.

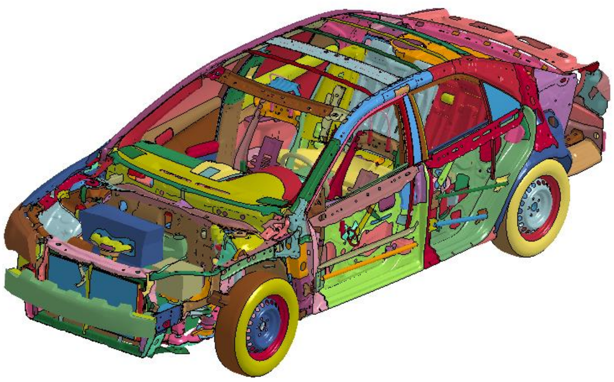
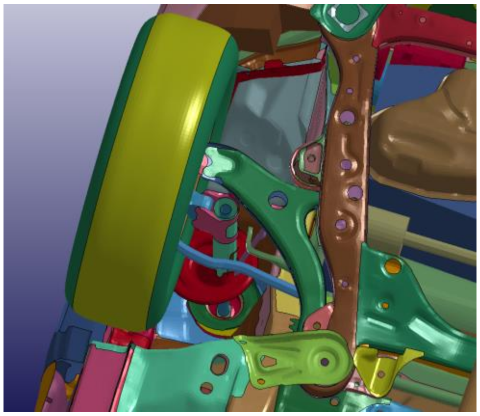


FIGURE 2 – Details of the Modeled Vehicle Structure

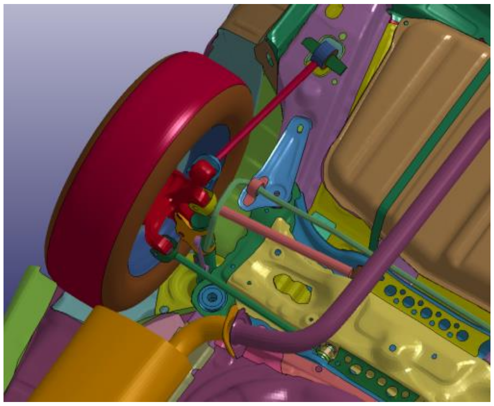
All inner components of the front and rear doors were included in this version of the model as seen in Figure 3.

FIGURE 3 – Details Door Components

Figure 4 provides a close-up of the modeled front steering and suspension system. These moving parts were detailed to provide the capability to simulate suspension and steering response in the simulation analyses.



Front Suspension



Rear Suspension

FIGURE 3 – Details of the Modeled Steering and Suspension Subsystems

Detailed representations of interior components of this vehicle are included in this version of the model as shown in Figure 5. This allows for dummy model integration with this model.

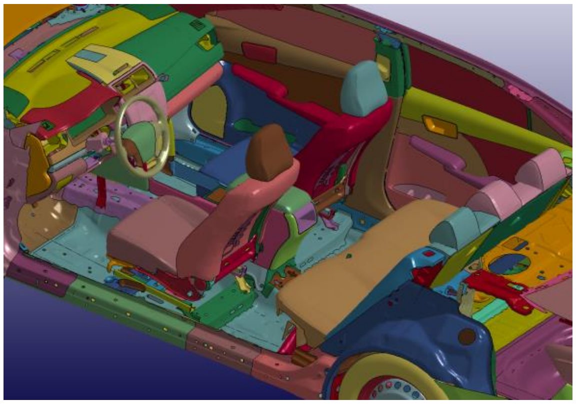


FIGURE 5 – Representations of Vehicle Interior Components

Model Validation

The FE model was verified and validated in several ways to assure that it was an accurate representation of the actual vehicle. These 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, as measured at the SEAS, Inc. lab, and vehicle model were compared [4]. 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.

TABLE 1– Actual Vehicle to Model Mass, Inertia, and CG Comparisons based upon Data from Testing at SEAS, Inc.

	Actual Vehicle	*FE Model*
Weight, kg	1452	1462
Pitch inertia, kg-m^2	2519	2524
Yaw inertia, kg-m^2	2796	2807
Roll inertia, kg-m^2	560	572
Vehicle CG X, mm	1063	1086
Vehicle CG Y, mm	-9	-1
Vehicle CG Z, mm	561	560

This model was validated by comparing the simulation of the NCAP frontal wall impact with actual data from multiple NHTSA NCAP tests to demonstrate the accuracy and versatility of the model.

After general verification of the basic integrity of the model using LS-DYNA, efforts were initiated to simulate a crash of this vehicle into a wall at 35 mph (56 kmh) as required for NCAP full-wall testing. For this simulation, accelerometers were positioned in the model at the same locations as in the NCAP test at the left rear seat, right rear seat, and engine top and bottom. The left rear seat and right rear seat accelerometers are used to measure the deceleration response and velocity of the vehicle cabin in the wall impact.

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 MPP-S R8.0.0 Revision 95890 on an Intel-MPI 4.1.3 Xeon64 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 120 milliseconds to capture the initial impact until the rebound of the vehicle from the NCAP load cell wall. Approximate computation time to run 120 ms using 16 mpp processors was about 11 hours.

Table 2 provides specific data for key parameters of the FE model and the vehicle used in NCAP Test 8545 [5]. It is easily noted that all were very similar. More information on the NHTSA's NCAP test vehicle information, like vehicle weight distribution, vehicle attitude, center of gravity (CG) location, and the fuel tank capacity, are published in the NHTSA report.

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

	FE Model	**Test 8545**
Year & Model	2012	2014 LE
Weight (kg)	1674	1664
Engine Type	2.5L 4 cyl	2.5L 4 cyl
Tire size	P205/55R16	P205/55R16
Attitude (mm) (As delivered)	F – 714	F – 742
	R – 680	R – 703
Wheelbase (mm)	2790	2769
CG (mm) aft of front wheel axle	1196	1194
Body Style	4 Door Sedan	4 Door Sedan

The overall global deformation pattern of the FE model was very similar to that of the NCAP full wall test as noted in the pictures in Figure 6. These visual images suggest that the FE model provides a reasonably accurate representation of an actual vehicle in a 35 mph impact scenario. Other views of deformation patterns of the FE model for simulated engine compartment and steering/underside damage was also noted to be very similar to that of the NCAP test. Similar patterns and extent of crush are noted.

FIGURE 6 – Comparison of the Global Deformation for NCAP Test and Simulation

The global response of the vehicle was further benchmarked against the NCAP test data by comparing the dynamic responses from the left and right rear seat cross member acceleration and average velocity, and engine top and engine bottom acceleration. The seat cross member acceleration plots are shown in Figure 7. 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 8, indicating that the test vehicle velocities also compared well to the simulation results.

The global responses of the engine top and engine bottom accelerometers also track the responses from test vehicles as shown in Figure 9. The tests and simulation show similar acceleration pulse magnitudes. This was the case for both the engine top and engine bottom accelerations.

Figure 10 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.

Last, in Figure 11, 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.

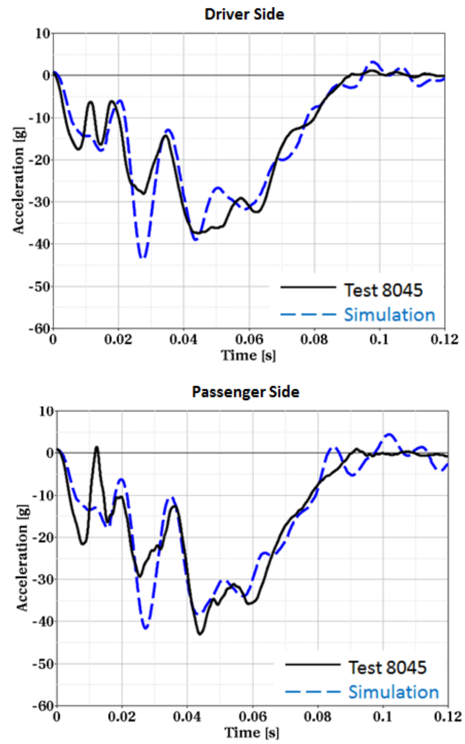


FIGURE 7 – Comparison of Tests & Simulation for Left and Right Rear Floor Accelerations

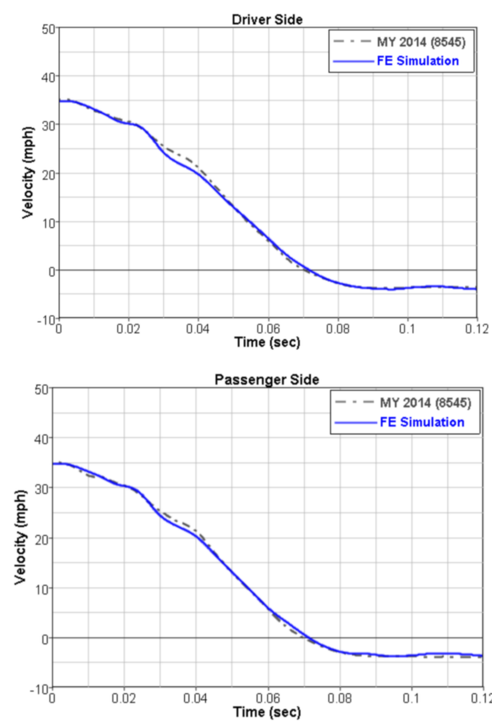


FIGURE 8 – Comparison of Tests & Simulation for Left and Right Rear Seat Velocities

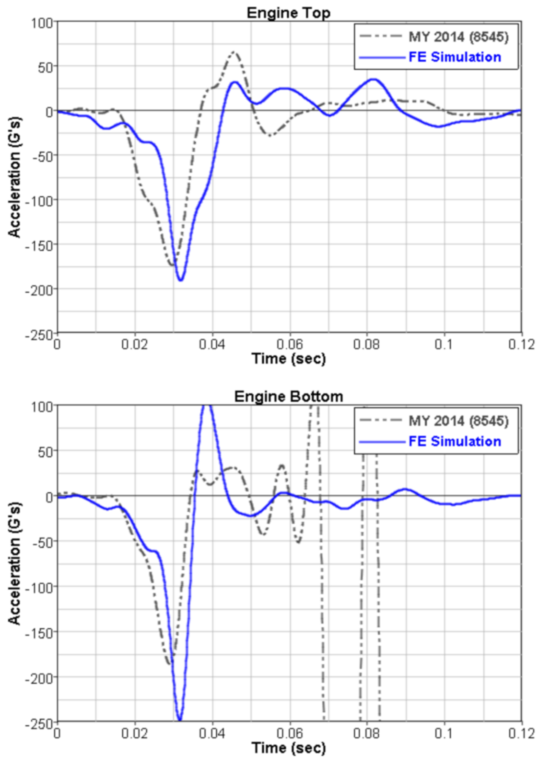


FIGURE 9 – Comparison of Tests & Simulation for Engine Top and Bottom Accelerations

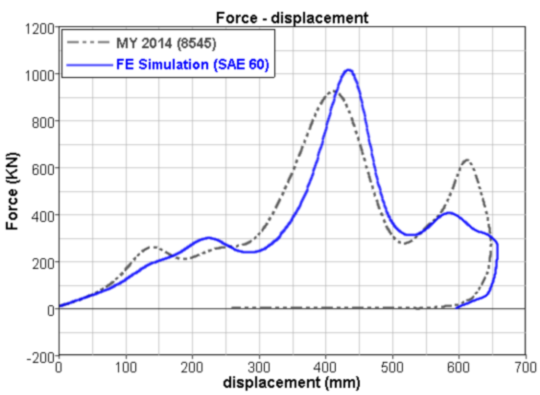


FIGURE 10 – Comparison of Tests & Initial Simulation Data for Force Displacement

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 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 over 2 million elements, included representation of all vehicle structural components, and represented the functions of the steering and suspension systems.

The model was validated by comparison to images and data derived from the NHTSA NCAP Test 8545, 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. 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.

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] Model available for download at: https://www.ccsa.gmu.edu/models/2012-toyota-camry
[2] AASHTO, Manual for Assessing Safety Hardware, published by the American Associations of State Highway & Transportation Officials, Washington, DC, 2009.
[3] Hallquist, J.O., “LS-DYNA Keyword User’s Manual,” Livermore Software Technology Corporation, 2015.
[4] SEAS, Inc., “Inertia Measurement Report for 2012 Toyota Camry,” VMF#4372, Columbus Ohio, May 30th 2012.
[5] NHTSA, “NCAP Frontal Barrier Impact – 2014 Toyota Camry,” NHTSA Test Report 8545.