Development & Validation of a Finite Element Model for the 2010 Toyota Yaris Passenger Sedan

Background
A finite element (FE) model based on a 2010 Toyota Yaris passenger sedan was developed through the process of reverse engineering at the National Crash Analysis Center (NCAC) of The George Washington University (GWU). 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 an 1100C test vehicle [1].

Modeling
A production 2010 Toyota Yaris four-door passenger sedan was purchased as the basis for the model [VIN JTDBT4K37A4067025]. The reverse engineering process systematically disassembled the vehicle part by part as in past efforts [2]. 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 974,383 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 and FE Model of a 2010 Toyota Yaris 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 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 properties of the various materials identified were determined by testing of specimens taken from the actual parts. The test data and/or standard material...
types were assigned to each component of the model.

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: 771
- Number of Nodes: 998,218
- Number of Shells: 950,560
- Number of Beams: 4497
- Number of Solids: 19,314
- Number of Elements: 974,383

The modeling effort detailed all components of the vehicle. Figure 2 shows the details of the model for the frame and drive train 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 3 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.

All inner components of the front and rear doors were included in this version of the model as shown in Figure 4. Coarse representations of interior components of this vehicle are included in this version of the model as shown in Figure 5. The remaining interior components will be added in the next version of the model.
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. 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.

This model was validated by comparing the simulation of the NCAP frontal wall impact with actual data from NHTSA Tests 5677 and 6221 for a comparable vehicle.

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

<table>
<thead>
<tr>
<th></th>
<th>Actual Vehicle</th>
<th>FE Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Weight, kg</td>
<td>1078</td>
<td>1100</td>
</tr>
<tr>
<td>Pitch inertia, kg-m²</td>
<td>1498</td>
<td>1566</td>
</tr>
<tr>
<td>Yaw inertia, kg-m²</td>
<td>1647</td>
<td>1739</td>
</tr>
<tr>
<td>Roll inertia, kg-m²</td>
<td>388</td>
<td>395</td>
</tr>
<tr>
<td>Vehicle CG X, mm</td>
<td>1022</td>
<td>1004</td>
</tr>
<tr>
<td>Vehicle CG Y, mm</td>
<td>-8.3</td>
<td>-4.4</td>
</tr>
<tr>
<td>Vehicle CG Z, mm</td>
<td>558</td>
<td>569</td>
</tr>
</tbody>
</table>

After general verification of the model using LS-DYNA, efforts were initiated to simulate a crash of this vehicle into a wall at 35 mph as required for NHTSA NCAP testing. For this simulation, accelerometers were positioned in the same locations as the NCAP test (Figure 6). The most commonly benchmarked accelerometers for NCAP performance are 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.

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

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

FIGURE 6 – 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 milliseconds to capture the initial impact until the rebound of the vehicle from the NCAP load cell wall. Approximate computation time to run 150 ms using 24 processors on the Intel was 2 hr 15 min.
Table 2 provides specific data 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 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’s report for Test Numbers 5677 and 6221 [4,5].

<table>
<thead>
<tr>
<th>TABLE 2 – Comparison of Parameters for FE Model &amp; Vehicle Used in the NCAP Test</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Weight (kg)</strong></td>
</tr>
<tr>
<td><strong>Engine Type</strong></td>
</tr>
<tr>
<td><strong>Tire size</strong></td>
</tr>
<tr>
<td><strong>Attitude (mm)</strong></td>
</tr>
<tr>
<td>F – 668</td>
</tr>
<tr>
<td>R – 673</td>
</tr>
<tr>
<td><strong>Wheelbase (mm)</strong></td>
</tr>
<tr>
<td><strong>CG (mm) Rear of front wheel C/L</strong></td>
</tr>
<tr>
<td><strong>Body Style</strong></td>
</tr>
</tbody>
</table>

The overall global deformation pattern of the FE model was very similar to that of the NCAP test as noted in the pictures in Figure 7.

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 engine top and engine bottom acceleration. The seat cross member acceleration plots are shown in Figure 8. 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 9, 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 10. The tests and simulation show similar acceleration pulse magnitudes. This was the case for both the engine top and engine bottom accelerations.

Figure 11 shows the comparison of the total force exerted by the vehicle on the load cell wall between the simulation and the tests. The plots show good correlation between the NCAP tests (5677 and 6221) and simulation results. The three curves have very similar pulse profiles and magnitudes. A maximum force of 550 kN and impact duration of 100 ms was observed in the simulation and tests.
To compute the vehicle stiffness (force vs displacement) plot, the displacements are computed from the seat cross member velocity pulses shown in Figure 9. Figure 12 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 both tests.

Last, in Figure 13, 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 since no external work was applied to the vehicle.
Summary & Conclusions
A finite element model of the 2010 Toyota Yaris 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. This vehicle conforms to the Manual for the Assessment of Safety Hardware (MASH) requirements for an 1100C 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 974,383 elements,
- Represented the functions of the steering and suspension components,
- Included all interior door components, and
- Included partial vehicle interior components.

The model was validated by comparison to images and data derived from the NHTSA NCAP Tests 5677 and 6221, which involved frontal impact into a rigid wall at 35 mph. Comparisons of data from the tests and the model included:

- View of side deformations,
- Acceleration and velocity changes for the rear seat cross member,
- Accelerations of the top and bottom of the engine,
- Total forces over time,
- 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 showed overall good correlation with the physical test results. Validations using other tests, including: speed bump, sloped median, IIHS ODB offset, NHTSA side impact, IIHS side impact, and roof crush, are currently in progress and an updated version of the model will be made available as soon as the validations are completed.

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

For More Information
See the NCAC website (www.ncac.gwu.edu) for more information, or contact:
- FHWA Office of Safety R&D
  Dr. Kenneth Opiela, PE
  202-493-3371
  kenneth opiela@dot.gov
- NCAC Staff
  Dr. Steve Kan (Director, NCAC)
  703-726-8511
cdkan@ncac.gwu.edu
  Dr. Dhafer Marzougui (Director, Highway Safety and Infrastructure Research)
  703-726-8532
dmarzoug@ncac.gwu.edu

This Technical Summary was produced under FHWA Contract DTFH61-09-D-00001 “Advanced Crash Analyses to Improve Safety & Security” with the National Crash Analysis Center of The George Washington University, Ashburn, Virginia.