Development & Validation of a 1994 Chevrolet C2500 Pick-up Truck FE Model

Background
A finite element (FE) model based on a 1994 Chevrolet C2500 pickup truck was developed through reverse engineering at the National Crash Analysis Center (NCAC) of The George Washington University (GWU). This vehicle was selected for modeling because it satisfied the requirement of a 2000 kg test vehicle under the crashworthiness evaluation requirements of NCHRP Report 350 [1]. While Report 350 specifies a generic 2000 kg pick-up vehicle, the C2500 pickup truck became the primary test vehicle for roadside hardware evaluation and certification crash tests. This model provides a linkage between crash testing and simulation activities. The finite element model of this vehicle is expected to serve a variety of safety analyses over the long term.

Modeling
The NCAC undertook development of an FE model of the 1994 Chevrolet C2500 pickup truck using reverse engineering procedures. A production version of this vehicle was purchased as the basis for the model. The vehicle was an 1828 kg standard cab pick-up truck with a 4.3 liter, V6 engine and an automatic transmission. Through reverse engineering, the vehicle was systematically disassembled part by part following traditional processes [2, 3]. Each part was cataloged, scanned to define its geometry, measured for thickness, classified by material type, and components meshed to represent elements. Data for each element was entered into a computer file with information on the connectivity of the elements as the basis for the computer representation or finite element model of the vehicle. The model reflected all of the structural and mechanical features in digital form.

The resulting FE vehicle model has 57,820 elements without the interior components or restraint systems. This FE model was constructed to include functional representation of the vehicle’s suspension and steering subsystems. A digital image of the resulting model is shown in Figure 1. Parts were broken down into elements considering the needs to represent critical features and recognizing the implications 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 strain rate values were determined to include in the model for the analysis of stress and strain behavior in crash simulation.

This set of elements 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 [4]. The resulting FE model had the following characteristics:

<table>
<thead>
<tr>
<th>Description</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of Parts</td>
<td>248</td>
</tr>
<tr>
<td>Number of Nodes</td>
<td>66,050</td>
</tr>
<tr>
<td>Number of Shells</td>
<td>54,028</td>
</tr>
<tr>
<td>Number of Beams</td>
<td>153</td>
</tr>
<tr>
<td>Number of Solids</td>
<td>33,695</td>
</tr>
<tr>
<td>Number of Elements</td>
<td>57,850</td>
</tr>
</tbody>
</table>
The modeling effort detailed all components of the vehicle. Figure 2 shows the details of the model for the frame and power train for this vehicle. The engine was not modeled in detail as simulation experience has found that it reacts as a large rigid mass in crashes.

![FIGURE 2 – Details of the modeled vehicle frame and drive train](image)

**Model Validation**

After general verification of the model, efforts were initiated to simulate a crash of this vehicle into a wall at 35 mph as required under the NHTSA New Car Assessment Program (NCAP). The simulation was expected to replicate the frontal impact from NCAP Test 1741 [5]. Table 1 provides specific data for key parameters of the FE model and the vehicle used in the NCAP test including vehicle weight, attitude, center of gravity (CG) location, and other data. Other details about the vehicle and the test results are published in NHTSA Test Report 1741.

**TABLE 1 – Comparison of parameters for FE model and vehicle used in NCAP Test 1741**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>FE Model</th>
<th>Test Vehicle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Weight (kg)</td>
<td>1828</td>
<td>2023</td>
</tr>
<tr>
<td>Engine</td>
<td>4.3 L V6</td>
<td>4.3 L V6</td>
</tr>
<tr>
<td>Tire size</td>
<td>P225/75R15</td>
<td>P225/75R15</td>
</tr>
<tr>
<td>Attitude (mm)</td>
<td>F – 890, R – 942</td>
<td>F – 825, R – 888</td>
</tr>
<tr>
<td>Wheelbase (mm)</td>
<td>3370</td>
<td>3340</td>
</tr>
<tr>
<td>CG (rear of FW)</td>
<td>1230 mm</td>
<td>1557 mm</td>
</tr>
<tr>
<td>Body Style</td>
<td>2-door, std cab</td>
<td>2-door, std cab</td>
</tr>
</tbody>
</table>

The first step in the validation was the comparison of the overall test and simulation results. The overall global deformation pattern of the FE model was noted to be very similar to that of NCAP Test 1741 (Figure 4).

![FIGURE 4 – Left side view of the test versus simulation](image)

The second stage in the validation was the comparison of key crash metrics from the simulation and crash test. For these comparisons, it was necessary to include accelerometers in the simulation at the same locations as in the NHTSA NCAP test (Figure 5). The left rear seat and right rear seat accelerometers are used to measure the deceleration response and velocity of the vehicle after impacting the rigid wall.

The simulation of the NCAP wall test was performed using the LS-DYNA non-linear explicit finite element code Version 970 on a single precision SGI Altix Itanium 2 platform. The FE model response would be expected to vary for other facilities depending on hardware, LS-DYNA version, and precision used. Total duration of the simulation was 150 milliseconds to capture the initial impact until the rebounding of the vehicle from the NCAP load cell wall. Approximate computation time to run 150 milliseconds using 4 processors on a single precision SGI workstation was 9 hours.
The global response of the vehicle was further benchmarked against the NCAP test data by comparing the response data from the accelerometers on the seat cross member and top and bottom of the engine.

The data from the accelerometers mounted on the left rear and right rear seat cross member were averaged and compared to the accelerometer response from the test. The timing and shape of the peak acceleration in the test was reasonably matched in the FE simulation results (Figure 6). The velocities of the seat cross member were also compared. The data generated in the simulations closely matched the test data, as shown in Figure 7.

The global response of engine top and engine bottom accelerometers also tracks the response from the test vehicle, as shown in Figures 8 and 9. The test and simulation show similar acceleration pulse magnitudes of about 35 g and pulse durations of 90 ms. These similar peaks and timing were the case for both the engine top and engine bottom accelerations.
Data from the load cells on the test wall allow comparisons of displacement metrics between the crash test and simulation. Figure 10 shows the comparison of the total force exerted by the vehicle on the load cell wall. The plots show good correlation between the test and simulation results. Overall the two curves have very similar pulse profiles, but some differences in the magnitude of force. A maximum force of 1100 kN was noted in the simulation, which was about 300 kN greater than the test. This difference might be attributed to greater frontal rigidity in the model associated with the engine engaging with the wall.

Figure 11 shows the total vehicle displacement, which tracks closely in both test and simulation over time, but particularly well up to 70 ms. This is considered important as displacement is a critical metric that is less subject to acceleration noise.

Figure 12 shows the comparison of the force-displacement curves over the duration of the crash event. The cross plot of force and displacement shows similar behavior of the vehicle in the test and simulation.

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.
Model Enhancement
This model has undergone continuous improvements and detailing in subsequent years to increase model performance and capabilities. One improvement involved adding details and functionality to the suspension system because pick-up trucks have been noted to have stability problems in impacts with roadside hardware. The characteristics of the pick-up truck, including the higher center of gravity, higher bumper, and greater mass, tend to make it less stable and more susceptible to rollovers. Therefore, the suspension and steering systems were the focus of several modifications to the finite element models. These improvements and the supporting validation efforts are fully reported in other documents [6, 7, 8].

Summary & Conclusions
A finite element model of the 1994 Chevrolet C2500 pickup truck was created using a reverse engineering process. This vehicle was selected by the FHWA to support barrier evaluations under the crashworthiness criteria in NCHRP Report 350. The initial modeling effort led to a model that consisted of 57,850 elements. The initial model did not represent the functions of the steering and suspension components, but these were later added. The C2500 FE model was validated by comparison to images and data derived from the NHTSA NCAP Test 1741, which involved frontal impact into a rigid wall at 35 mph. Comparisons of data from the test and the model included:

- Views of side deformations,
- Comparisons of accelerometer data for the seat cross member and engine top and bottom,
- Force-displacement plots, and
- Total crash energy balance analysis.

Both the vehicle kinematics and the accelerometer output results were compared and the simulation results showed overall good correlation with the physical test results. The FE model was found to be stable in full frontal flat rigid wall simulations. The model was also run at 25, 30, 35, and 40 mph to ensure stability.

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.