Development & Initial Validation of a 2007 Chevrolet Silverado
Finite Element Model

Background
A finite element (FE) model based on a 2007 Chevrolet Silverado 1500 2WD quad cab pickup was developed through reverse engineering at the National Crash Analysis Center (NCAC) of The George Washington University (GWU) under a Federal Highway Administration (FHWA) contract. This vehicle was jointly selected for modeling by the FHWA and the National Highway Traffic Safety Administration (NHTSA) because it:
- Satisfies the requirement of a 2270 kg test vehicle under the Manual for Assessing Safety Hardware (MASH) adopted in 2009 [1] and
- Allows analyses of the unique Safety Energy Absorbing Structure (SEAS) incorporated in the design of this vehicle in New Car Assessment Program (NCAP) frontal impact tests.

The model is expected to serve a variety of other analyses over the long term. This technical summary describes the process used to develop the model, the basic features of the model, and the initial efforts to validate the FE model.

Modeling
In March 2007, the NCAC began development of an FE model of the 2007 Chevrolet Silverado quad-cab pick-up following established procedures [2]. A production version of the truck was purchased as the basis of the model (VIN 2GCEC13C771511793). The vehicle was a 2298 kg, 4-door crew cab, short box, pick-up truck with a 4.8 liter, V8 engine and a 4-speed, automatic transmission.

Under the reverse engineering process, the vehicle was systematically disassembled part by part following traditional processes [2,3]. Each part was cataloged, scanned to define its geometry, measured for thicknesses, and classified by material type. Connectivity with other elements was defined. All the data was entered into a computer file and then meshed to create a computer representation of the vehicle as a finite element model. The model reflected all of the structural and mechanical features of the vehicle in digital form.

The resulting FE vehicle model has 928,932 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 picture of the vehicle and the digital representation of the resulting model are shown in Figure 1. Parts were broken down into elements considering the needs to represent critical features and the implications on simulation processing times.

Material data for the major structural components was obtained through coupon testing from samples taken from vehicle parts after teardown. 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:

- Number of Parts: 676
- Number of Nodes: 942,491
- Number of Shells: 872,960
- Number of Beams: 2,654
- Number of Solids: 53,286
- Number of Elements: 928,932

The modeling effort detailed all components of the vehicle. Figure 2 shows the details of the model for the frame and power train. The engine was not modeled in detail as simulation experience has found that it reacts as a large rigid mass in crashes. Figure 3 is a close-up of the front and rear suspension system. These moving parts were detailed to provide the capability to reflect the suspension response in the simulation analyses. The interior parts were not included in the initial modeling effort.

Model Validation

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 under the NHTSA NCAP testing to replicate frontal NCAP Test 5877 [5]. Table 1 provides specific data for key parameters of the FE model and the vehicle used in the NCAP test including vehicle weight, vehicle attitude, center of gravity (CG) location, and other data.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>FE Model</th>
<th>Test 5877</th>
</tr>
</thead>
<tbody>
<tr>
<td>Weight (Kgs)</td>
<td>2622</td>
<td>2622</td>
</tr>
<tr>
<td>Engine Type</td>
<td>4.8 L V8</td>
<td>4.8 L V8</td>
</tr>
<tr>
<td>Tire size</td>
<td>P245/70R17</td>
<td>P245/70R17</td>
</tr>
<tr>
<td>Attitude (mm)</td>
<td>F – 1016</td>
<td>F – 929</td>
</tr>
<tr>
<td></td>
<td>R – 1043</td>
<td>R – 1002</td>
</tr>
<tr>
<td>Wheelbase (mm)</td>
<td>3660</td>
<td>3664</td>
</tr>
<tr>
<td>CG (mm) rear of front wheel C/L</td>
<td>1670</td>
<td>1664</td>
</tr>
<tr>
<td>Body Style</td>
<td>4-door, crew cab</td>
<td>4-door crew cab</td>
</tr>
</tbody>
</table>

The overall global deformation pattern of the FE model was very similar to that of NCAP Test 5877 (Figure 4). Additionally, the crush mode of the rails and the structural members showed good correlation between the test and simulation when the underside views were compared (Figure 5). The before and after the deformation of the truck frame rail is similar for the test and simulation.
For this simulation, accelerometers were positioned in the same locations as in the NHTSA NCAP test (Figure 6). 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 971 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 16 processors on a single precision SGI workstation was 10 hours.

The global response of the vehicle was further benchmarked against the NCAP test data by comparing the acceleration and velocity responses from the left and right rear seat cross member accelerometers and the acceleration responses from the engine top and bottom accelerometers.

The data from the accelerometers mounted on the left rear and right rear seat cross member were compared to the accelerometer response from the test. The timing and shape of the peak acceleration in the test was matched in the FE simulation results (Figures 7 & 8). The timing and shape of the peak acceleration in the test was matched in the FE simulation through 120 ms into the crash event. The
simulation, however, shows higher peaks in the first 20 ms of the impact. This is attributed to the fact that the body mounts failed later in the simulation than in the test. The cab remained attached to the rails longer in the simulation and consequently saw higher deceleration. Further calibration eliminated this difference between the test and simulation.

The velocities of the right and left seat cross member were also compared. The data generated in the simulations closely matched the test data as shown in Figures 9 and 10, although the simulated velocities were consistently higher.

Figure 11 shows the comparison of the total force exerted by the vehicle on the load cell wall between the simulation and test. The plots show good correlation between the test and simulation results. The two curves have very similar pulse profiles and magnitudes. A maximum force of 1000 kN and impact duration of 150 ms and was observed in the simulation and test.

Last, in Figure 12 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.

FIGURE 12 – Comparison of test and simulation data for energy balance

Summary & Conclusions

A finite element model of the 2007 Chevrolet Silverado pick-up truck was created using a reverse engineering process. This vehicle was selected to allow NHTSA to study the behavior the vehicle’s Safety Energy Absorbing Structure (SEAS) in frontal NCAP tests and for FHWA use to support barrier evaluations under MASH. The initial modeling effort led to a detailed model that:

- Consisted of 942,491 elements,
- Represented the functions of the steering and suspension components, and
- Did not include interior parts.

The model was validated by comparison to images and data derived from the NHTSA NCAP Test 5877, which involved frontal impact into a rigid wall at 35 mph. Comparisons of data from the test and the model included:

- Views of side and underside deformations,
- Comparisons of accelerations and velocities for the left and right rear 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 data 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.

The Silverado FE model was initially released on February 27, 2009. A future release of the model will include the interior components to allow use for occupant risk analysis.

In addition to the 2622 kg version of the FE model, the NCAC has developed a 2270 kg version, which specifically meets the requirements of the MASH.

References

1. AASHTO, Manual for Assessment of Safety Hardware, the American Associations of State Highway & Transportation Officials, Washington, DC, 2009.

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
  edkan@ncac.gwu.edu
  Dr. Dhafer Marzougui (Director, Highway Safety and Infrastructure Research)
  703-726-8532
  dmarzoug@ncac.gwu.edu