Development & Validation of a Finite Element Model for the 2006 Ford F250 Pickup Truck

Background:
A finite element (FE) model based on a 2006 Ford F250 pickup was developed through the process of reverse engineering at the National Crash Analysis Center (NCAC) of the George Washington University (GWU). This model was validated to the National Highway Traffic Safety Administration (NHTSA) frontal New Car Assessment Program (NCAP) test for the corresponding vehicle. The objective of this study was to develop a finite element model of the 2006 Ford F250 that would allow analyses of the Safety Energy Absorbing Structure (SEAS) incorporated in the design of this vehicle in the NCAP frontal impact test.

Modeling
A production 2006 SD Ford F250 4x4 Supercab pickup truck was purchased as the basis for the model (VIN 1FTSX21516E A73254). The reverse engineering process involved systematically disassembling the vehicle part by part. 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 meshed to create an FE representation or model that reflected all of the structural and mechanical features of the physical vehicle in digital form [1, 2].

The resulting FE vehicle model has 726,759 elements and does not include 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 is shown in Figure 1. 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 and properties for the major structural components were obtained through coupon testing from samples taken from vehicle parts. From the material testing, appropriate strain rate values were determined and included in the model for the analysis of stress and strain behavior in the crash simulation. Standard material types were assigned to any parts for which no test data were available.

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. The result of these efforts was a finite element model with the following characteristics:

- Number of Parts: 871
- Number of Nodes: 738,165
- Number of Shells: 698,501
- Number of Beams: 2,353
- Number of Solids: 25,905
- Number of Elements: 726,759

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 modeled with a coarser mesh as simulation experience has found that it reacts as a large rigid mass in crashes. The engine was modeled as 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 is a close-up of the front steering and suspension system. These moving parts were detailed to provide the capability to simulate suspension and steering response in the simulation analyses.

Figure 2 – Details of the modeled vehicle frame and drive train

Figure 3 – Details of the modeled steering and suspension subsystems

Interior elements of this vehicle were not included in the initial version of the model.

The model was validated by comparing the simulation of the NCAP frontal wall impact test with the data from NHTSA NCAP Test 5820 for a comparable vehicle [3].

Validation Results

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 New Car Assessment Program (NCAP) frontal testing. For this simulation, accelerometers were positioned in the same locations as the NHTSA NCAP test (Figure 4). The most commonly benchmarked accelerometers for NCAP performance are the left rear seat, right rear seat, engine top, engine bottom, and instrument panel top. 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.

Figure 4 – 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 971 on a single precision Itanium 2 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 rebounding of the vehicle from the NCAP load cell wall. Approximate computation time to run 150 milliseconds using 8 processors on a single precision SGI parallel machine was 31 hours.

Table 1 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, such as test vehicle weight distribution, test vehicle attitude, center of gravity (CG) location, and the fuel tank capacity, are published in the NHTSA’s report for High Resolution Test Number: 5820.
TABLE 1 – Comparison of parameters for FE model and vehicle used in the NCAP test

<table>
<thead>
<tr>
<th></th>
<th>FE Model</th>
<th>Test 5820</th>
</tr>
</thead>
<tbody>
<tr>
<td>Weight (Kgs)</td>
<td>3016</td>
<td>3054</td>
</tr>
<tr>
<td>Engine Type</td>
<td>5.4L EFI V8</td>
<td>5.4L EFI V8</td>
</tr>
<tr>
<td>Tire size</td>
<td>LT245/75R17E</td>
<td>LT245/75R17E</td>
</tr>
<tr>
<td>Attitude (mm) (As delivered)</td>
<td>F – 1016</td>
<td>F – 1013</td>
</tr>
<tr>
<td></td>
<td>R – 1043</td>
<td>R – 1055</td>
</tr>
<tr>
<td>Wheelbase (mm)</td>
<td>3610</td>
<td>3610</td>
</tr>
<tr>
<td>CG (mm) Rearward of front wheel C/L</td>
<td>1499</td>
<td>1489</td>
</tr>
<tr>
<td>Body Style</td>
<td>Extended Cab</td>
<td>Extended Cab</td>
</tr>
</tbody>
</table>

The overall global deformation pattern of the FE model was very similar to that of NCAP Test 5820 (Figure 5). In the test, the body mounts connecting the cab to the rails failed. Component tests were conducted on the body mounts of the F250 vehicle and used to calibrate the model. Simulations of these component tests were conducted to ensure the model can reflect the body mount failure. Upon completing the calibration, the improved body mount models were incorporated into the vehicle model. Full-scale simulations of the NCAP case replicated the body mount failures observed in the test.

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

Data from the accelerometers mounted on the left rear seat and right rear seat (inside the cab) were averaged and compared to the accelerometer response from the test. The seat cross member acceleration plot is shown in Figure 7. The timing and shape of the peak acceleration in the test was matched in the FE simulation up to 80 milliseconds into the crash event. The simulation, however, shows a higher peak 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 remains attached to the rails longer in the simulation and consequently its sees higher deceleration. Further calibration eliminated this difference between the test and simulation.
Following the 80 ms, the FE model decelerates faster than the test vehicle. This is also reflected in the average velocity comparison in Figure 8, which shows that the test vehicle slowed down more quickly compared to the FE model. Possible reasons include a coarser mesh used in the rear of the vehicle or the absence of failure criteria in the drive shaft connection to the rear differential.

The global response of engine top and engine bottom accelerometers also tracks the response from test vehicle as shown in Figures 9 and 10. The test and simulation show similar acceleration pulse magnitudes of about 35 g and pulse durations of 90 ms. 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 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 were observed in the simulation and test.

![Figure 11](image1.png)

**FIGURE 11 – Comparison of test and simulation data for total force**

To compute the vehicle stiffness (force vs. displacement) plot, the displacements are computed from the seat cross member velocity pulses shown in Figure 8. The computed displacement plots are shown in Figure 12. The figure shows that the vehicle displacement tracks closely between the test and simulation.

![Figure 12](image2.png)

**FIGURE 12 – Comparison of test and simulation data for total displacement**

Figure 13 shows the vehicle stiffness plots extracted from the test and simulation. The figure shows that, overall, the vehicle stiffness from the simulation is similar to that of the test. The plots show a vehicle stiffness of about 5 kN/mm in the first 150 mm of crush followed by a constant force of 600 kN between 300 and 700 mm of crush.

![Figure 13](image3.png)

**FIGURE 13 – Comparison of test and simulation data for force displacement**

Last, in Figure 14 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.

![Graph showing energy balance comparison]

**FIGURE 14 — Comparison of test and simulation data for energy balance**

---

**Summary and Conclusions**

A finite element model of the 2006 Ford F250 pickup truck was created using a reverse engineering process. This vehicle was initially selected to study the behavior the vehicle’s Safety Energy Absorbing Structure (SEAS) in frontal NCAP tests. The modeling effort led to a detailed model that:

- Consisted of 726,756 elements
- Represented the functions of the steering and suspension components
- Did not include interior parts

The model was validated by comparison to images and data derived from the NHTSA NCAP Test 5820, which involved frontal impact into a rigid wall at 35 mph. Comparisons of data from the test and the model included:

- View of side and underside deformations
- Acceleration and velocity changes for the seat cross member
- Acceleration of the top and bottom of the engine
- Total wall forces over time
- Force displacement plots
- 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.

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](http://www.ncac.gwu.edu)) for more information, or contact:

- **FHWA Office of Safety R&D**
  Dr. Kenneth Opieia, PE
  202-493-3371
  kenneth.opieia@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 Cooperative Agreement DTFH61-02-X-00076 “Operation & Maintenance of the FHWA/NHTSA National Crash Analysis Center” between the FHWA and the George Washington University.