Development and Validation of a Pick-Up Truck Suspension Finite Element Model for Use in Crash Simulation

Dhafer Marzougui
Cing-Dao (Steve) Kan
Matthias Zink
Abdullatif Zaouk
Nabih Bedewi

The National Crash Analysis Center, The George Washington University
20101 Academic Way, Ashburn, VA 20147 USA
Email: dmarzoug@ncac.gwu.edu
Email: cdkan@ncac.gwu.edu

This working paper summarizes recent efforts and findings derived from NCAC research. It is intended to solicit feedback on the approach, scenarios analyzed, findings, interpretations, and implications for practice reported by the research team. The statements contained herein do not necessarily reflect the views or policy of the FHWA. Please forward comments or questions to the authors noted above. These efforts will ultimately be documented and made available to advance research efforts related to this topic and guidance for practice.

ABSTRACT

A finite element model of a Chevrolet C2500 pickup truck vehicle was developed at the FHWA/NHTSA National Crash Analysis Center (NCAC). This model has seen widespread use by transportation safety researchers to analyze vehicle safety issues as well as to evaluate and improve the performance of roadside hardware. Over the years after the initial release of the model, modifications and additional details have been incorporated into the finite element model to add capabilities, better represent dynamic response, and allow the use of the model in different impact scenarios. In this study, the rear suspension details of the C2500 pickup truck model were enhanced. Pendulum tests were conducted at the Federal Highway Administration Federal Outdoor Impact Laboratory (FOIL) and used to validate the dynamic response of the model. The focus in this study was on the rear suspension system of the vehicle. Simulations were conducted and the results are compared to the pendulum tests in terms of deformation, displacement, and acceleration at various locations. To ensure the accuracy of the newly upgraded vehicle model, previously conducted full-scale crash tests were simulated and the results from these simulations were compared to the test results. It was concluded that after the modifications to the detailed pickup truck model, there was a good correlation of dynamic response in simulated results to the pendulum and full-scale tests. This increases confidence in the results generated by crash simulations using the C2500 pickup model.
Development and Validation of a Pick-Up Truck Suspension Finite Element Model for Use in Crash Simulation

INTRODUCTION

Detailed and reduced multi-purpose finite element models of a 1994 Chevrolet C2500 pickup truck were developed at the FHWA/NHTSA National Crash Analysis Center (NCAC). The models were developed to address vehicle safety issues, including front and side performance, as well as roadside hardware design. The reduced model was developed to offer a downsized, simplified version of the detailed model, requiring less computation time [1].

The importance of the pickup truck for roadside safety research was elevated by National Cooperative Highway Research Program (NCHRP) Report 350, published in 1993 [2]. The report contains the recommended procedures for testing and evaluating roadside safety features, superseding the NCHRP Report 230, which was published in 1980 [3]. Of particular importance was the replacement of the 2044 kg passenger car test vehicles specified in NCHRP Report 230 by a 2000 kg pickup truck under NCHRP Report 350 procedures. This change was predicated by the need to have a large test vehicle more representative of the fleet of vehicles operating on US highways. While NCHRP Report 350 specifies a generic 2000 kg pickup vehicle, the C2500 pickup truck became the main test vehicle for the majority of roadside hardware evaluation and certification crash tests. To support a linkage between crash simulation and testing activities, there was a need to develop a finite element model of this vehicle. This model was first developed by reverse engineering in 1996 and has undergone continuous improvements and detailing in subsequent years to increase model performance and capabilities. One improvement was the creation of a reduced element version of the model to allow the use of the model on more computers and to reduce the computer time needed to execute a crash simulation. Other modifications involved adding details and functionality to the suspension system because pickup trucks often experience stability problems in impacts with roadside hardware. The characteristics of the pickup truck, including the higher center of gravity, higher bumper, and greater mass, tend to make it less stable and more susceptible to rollover, which can result in more severe crashes than for other vehicles. Therefore, the suspension and steering systems were the focus of several modifications to the finite element models.

The primary goal of the research reported here was to upgrade the back-suspension system of the C2500 pickup truck model. The upgrade has been incorporated in the detailed model, which should replace the reduced model as the primarily used vehicle model in roadside hardware crash tests and result in a higher accuracy of deformation and failure prediction as well as the representation of realistic dynamics and overall crush. To validate these changes, pendulum tests that were performed on the pickup suspension and full-scale crash tests were compared to simulation results. A summary of the validation results are presented in this paper.

DETAILED C2500 PICKUP TRUCK MODEL

In this research, Version 4f of the Chevrolet C2500 pickup truck detailed model was used. The model is based on a Fleetside, regular cab, long-box pickup truck with a total length of 5.4 meters and a wheelbase of 3.34 meters. The engine was a 4.3 liter Vortec V6 with electronic fuel injection coupled to a manual transmission with a rear wheel drive configuration. Even though several different versions of the truck existed, such as higher engine capacity, automatic transmission, and four wheel drive configuration, they had no significant change on the general geometry. The model was created using a systematic reverse engineering process, where the truck was first disassembled, grouped into several main groups and the geometry of each part of the groups was then obtained using a digitizing arm. The parts were then meshed and connected to each other using “spotwelds” and “nodal rigid body constraints.” Material properties taken from coupon specimens were then incorporated in the model to further characterize each part of the vehicle. The model consisted of approximately 55,000 elements and
211 components. Full-scale vehicle crash tests were used to evaluate and validate the model [1]. The detailed finite element model of the C2500 is shown in Figure 1.

Figure 1: Finite Element Model of the C2500 Pickup Truck

The original version of the model was developed for use in frontal impact simulations. Therefore, the front components of the vehicle were modeled in fine detail in order to obtain reasonable agreement with frontal crash tests that focused on overall motion and deformation profiles in the high crush regions. The rear suspension does not have a significant effect in frontal impacts and was therefore incorporated with less detail in the model. The rear suspension of the C2500 pickup truck consists of a leaf spring with a hydraulic shock absorber. Figure 2 shows a rear suspension system from an actual C2500 vehicle and its representation in the original model. The leaf spring is modeled using a series of beam elements with elastic material properties and constant cross-sectional properties. The beams are connected to the rails through “nodal rigid body constrains.” For the connection to the rear axle, a “constrained extra nodes added to rigid body” option is used. No damping system was included in the original model. Also, some of the rear components of the pickup truck, such as the rear cross members, were not modeled in detail because they have negligible effect on frontal impacts.

Figure 2: Rear Suspension of C2500 Pickup Truck and Finite Element Model

MODEL UPGRADES

To make the detailed pickup truck model more applicable for roadside hardware simulations, several details were added to the model. First, some of the missing or coarsely meshed components in the rear of the vehicle, such as the rear cross member and bed, were modeled with more detail. Next, all updates that were made to the reduced version of the C2500 model were incorporated in the detailed model. This included details to allow spinning wheels, features of the front and rear suspension, and steering system components. Model updates related to the rear suspension system are described in the following sections.

Rear Cross-Member

The last cross-member of the rail-connection, referred to as rail-constraint #5, was digitized and added to the C2500 model. The rail-connection consisted of 458 shell elements with a thickness of 3.3 mm. The material
definition is an LS-DYNA material Type 24 or “Piecewise Linear Isotropic Plastic” model [4,5]. Material properties of steel were assigned and a stress-strain curve was defined. The cross-member was connected to the pickup truck rail using the “nodal rigid body constraint” option. Figure 3 shows pictures of the actual C2500 pickup truck frame and views of the frame of the finite element model after adding the cross-member.

Figure 3: C2500 Pickup Truck Frame Cross-Member and its Finite Element Representation

Leaf Suspension and Shock Absorber
The reduced model of the C2500 pickup truck contained a crude representation of the leaf spring and shock absorbers. More detailed representations of these components were added to the detailed model. Figure 4 shows an isometric and a side view of the leaf spring model. The model of the complete leaf spring consists of 193 shell elements and four joints. It is subdivided into the components for the leaf springs (including the frontal mounting), the hinge, and the rear mounting. The material type used for the leaf spring is an LS-DYNA material Type 1 or “elastic” model. The material elastic properties are those of steel. The two components of the hinge and the rear mounting are modeled as rigid. The joints are modeled as revolute joints. The first joint is between the two components of the hinge and the second is between the hinge and the rear mounting. Figure 5 shows the hinge from an actual pickup truck and from the model.

Figure 4: Leaf Spring Model

Figure 5: Rear Suspension Hinges
The leaf spring is connected to the rail using a “nodal rigid body constraint” for the front mounting and a “constrained extra nodes added to rigid body” for the back mounting. The connection to the rear axle is also modeled using a “constrained extra nodes.” The shock absorbers are modeled with discrete elements between one node of the rear axle and one node of the rail. These nodes were selected such that they would represent the actual damper connection locations. The material type used for the damper is a linear viscous model, which provides a linear translational damper between two nodes. The damping constant is defined as the relation between the force and displacement rate. Figure 6 shows a view of the rear damper locations.

**REAR SUSPENSION VALIDATION**

To validate these upgrades, and more specifically the rear suspension model, simulations were performed and compared to pendulum and full-scale crash tests. The pendulum tests were conducted at the FHWA Federal Outdoor Impact Laboratory (FOIL). A total of three pendulum tests were performed, one on the front suspension and two, tests 02025 and 02027, on the rear suspension [6]. A previously conducted full-scale crash test of a C2500 pickup truck impacting a New Jersey shape concrete median barrier was also used to validate the updated model. The validations using the two pendulum tests on the rear suspension and the full-scale test are described in the following sections.

*Simulation for Pendulum Test 02025*

In this test, a pendulum with a mass of 2,014 kg impacted the rear right tire of the C2500 pickup truck from the bottom and compressed the leaf spring and the shock absorber (Figure 7). The rails of the truck were fixed at four points to avoid movement due to the pendulum impact. The pendulum was pulled by a quick release hook to a certain height such that its impact speed was 2.17 m/s.

![Figure 6: Rear Shock Absorbers](image)

![Figure 7: Setup for Pendulum Test 02025](image)
For the finite element simulation, parts were removed from the truck model in accordance to the test setup. The rails were fixed at four places using “single point constraint” boundary conditions in a similar manner to the way they were fixed in the test. Accelerometers were placed in the finite element model at similar locations to the accelerometer locations in the test. A previously developed finite element model of the pendulum similar to the test setup was imported for the simulations. Figure 8 shows an isometric and a side view of the finite element simulation setup.

Upon completing the model upgrades, simulations were performed and compared to the pendulum test results. Visual comparisons of the animations generated from simulation results and test videos were the first step in validating the finite element model. Figure 9 shows the comparison at different time stages. Comparing the slides for 0.1s, 0.25s, and 0.35s, it is observed that the displacement of the rear axle occurs faster in the physical test than in the simulation. In the test, the rear axle impacts the rail at 0.25s and is still in contact with it at 0.35 s, while in the simulation the rear axle is still moving at 0.25s and strikes the rail at 0.35s. A second significant deviation from the test is the bending behavior of the leaf spring. In the simulation, the leaf spring shows non-uniform bending at the time of the maximum displacement of the rear axle.

A comparison of accelerations measured at different positions during the simulation was the second primary method of validation. The acceleration sensors in the real test were modeled in the finite element simulation with rigid bodies mounted to the respective part of the finite element model. Acceleration measurements are very sensitive and are susceptible to high frequency noises. Thus, the data is filtered to get a clearer measurement. All vehicle structure accelerations were filtered with a 60 Hz SAE filter. The first comparison was the acceleration measured at the rear of the pendulum. This pulse is shown in Figure 10. The acceleration measurements taken from the simulation appear to be very noisy. A reason for this might be a missing damping between the pendulum and the tire. An obvious deviation is recognized for the time period between 0.2s and 0.7s. While the test deceleration reaches its maximum peak, the deceleration of the simulation oscillates around a constant value. Figure 11 shows the displacement of the rear axle measured with the string pot. This graph shows the same deviation between the physical test and the simulation, i.e. the displacement of the rear axle occurs slower in the simulation and the rear axle does not stay in contact with the rail as observed in the test. Comparing the maximum displacements, it was found that the rear axle impacts the rails in the simulation after a displacement of 262 mm whereas the physical test shows a maximum displacement of 323 mm. This leads to the conclusion that the initial distance between rear axle and rail was different for the simulation. In summary, three major differences between the test and the simulations were observed: 1) the initial distance between the rear axle and rail is different, 2) the displacement rate is slower, and 3) the leaf spring shows non-uniform bending at the maximum displacement.
After examination of the comparisons, modifications were made to the model to mitigate the differences between the simulation and test results. The first modification was made to the initial geometry of the leaf springs. The initial distance between the rear axle and the rail in the finite element model was found to be different from that in the physical model. The string pot measurements of Test 02025 show a distance of 323mm between the rail and the rear axle for the physical test. For the finite element model, the distance was measured to be 262mm. This difference is attributed to the fact that the geometry of the leaf-spring in the model was taken when the truck was at equilibrium under gravity loads. This was not the case in the test since the truck was tested with no compression loads applied on the leaf-spring (the truck frame was tested on its side). The geometry of the uncompressed leaf springs was obtained through simulation and incorporated in the model. This was achieved by applying a prescribed motion to the nodes in middle of the leaf spring to extend the leaf spring until the distance between the rear axle and the rail was 323mm. The final geometry from this simulation was used as the initial geometry in the pendulum simulations.

Another modification was applied to the hinges attaching the leaf springs to the rails (Figure 5). After examining the motion of the hinges in the simulation, it was found that the hinge rotates almost 90 degrees during the compression of the leaf spring. To investigate the motion of the hinge in the physical test, the video data of Test 02025 was analyzed. Even though it was not possible to have an explicit view of the hinge, some of the views showed smaller rotation than that from the simulation. The hinge rotation was below 45 degrees. The reason for this difference was attributed to the difference in geometry between the actual and modeled component as well as its location relative to the leaf springs. A difference of 25mm was found between the actual component and the
model. The hinge geometry and location was corrected by extending its length by 25mm. After this modification, the motion of the hinge in the simulation was found to be similar to the one in the test. The results from the simulation after the modifications are shown in Figures 12, 13, and 14. It can be seen from these figures that the behavior of the simulation and the test were clearly much more similar.

Time= 0.00s     Time= 0.10s  Time= 0.25s       Time= 0.35s    Time= 0.75s

Figure 12: Comparison of Pendulum Test 02025 and Modified Simulation

In the second test, Test 02027, the pendulum impacted the rear left tire of the C2500 pickup truck sideways and displaced the rear axle laterally (Figure 15). The rails of the truck, in this test, were also fixed at four points to avoid movement due to the impact of the pendulum.

Similar to the previous simulation, parts from the truck model were removed to reproduce this test setup. The model was rotated and positioned in front of the pendulum model in a similar manner to the test. Accelerometers were placed in similar locations to the accelerometers in the test. The pendulum initial velocity from the test was assigned to the pendulum model. Isometric and side views of the finite element simulation setup are shown in Figure 16.
Visual comparisons of the test film and initial simulation sequential views (see Figure 17) revealed significant differences in component behavior. The model showed a much stiffer response than the test. The duration of the impact was much shorter and the maximum displacement was significantly smaller in the simulation. The pendulum acceleration data, shown in Figure 18, also varied significantly between the simulation and the test. The results from the simulation appeared to be very noisy. The pendulum underwent a very high deceleration at the initial stages of the impact. The simulation acceleration after this peak shows some agreement with the test up to time 0.1s. The deceleration in the physical test, however, lasted until 0.25s, whereas the deceleration of the simulation ended early at 0.15s. The displacement of the rear axle measured with the string pot is visualized in Figure 19. The displacement of the rear axle in the simulation is significantly smaller than the results from the physical test.
Figure 18: Comparison of Test 02027 and Initial Simulation Accelerations

Figure 19: Comparison of Test 02027 and Initial Simulation Displacements

Based on the results from the initial simulations, additional changes were made to the rear suspension model. One of the changes was made to the front leaf spring mount. In the original model the front mounting is included in the leaf spring component. Figure 20 illustrates the connection between the leaf spring, the mount, and the rail. The circled points show the nodes that are part of a “nodal rigid body constraint,” fixing the mounting to the rail and the leaf spring to the mount. In the actual pickup truck, there is a joint between the leaf spring and the frontal mount. To improve the behavior of the finite element model and make the performance of the leaf spring more realistic, the connection between the leaf spring and the frontal mount was replaced by a joint. Figure 20 shows a view of the frontal mounting with the joint.

Another modification incorporated clamps for holding the leaf springs together. These clamps are modeled as “nodal rigid bodies” in the original model. Figure 21 shows a picture of a clamp and the connection in the finite element model. Here again the circled nodes are part of the “nodal rigid body constraint,” connecting the leaf springs together. In the actual truck, two clamps are mounted on each leaf spring, one in front of the rear axle and one behind it. The function of the clamps is to prevent a lateral motion between the leaf spring, but allow some longitudinal motion. The “nodal rigid body constraint” in the model prevents any motion between the constrained nodes. To improve the behavior of the leaf spring, the nodal constraints representing the clamps were deleted and four clamps were created and added to the model. Figure 21 shows the leaf springs with the new clamps. The clamps are connected to only the bottom leaf, as in the actual truck, with a “nodal rigid body constraint.” A contact definition was added between the clamps and the leaf springs to prevent the lateral motion between them.

A third modification was made at the joints connecting the leaf springs to the rear hinge. The joints in the finite element model are defined as revolute joints. Using this type of joint, the only relative motion between the leaf springs and the mount is a rotation about the joint axis. In the actual pickup truck, there is a rubber bushing at the joint which allows some translation and rotation between the leaf spring and mount due to deformation of this bushing. To allow for this motion in the finite element model, nonlinear springs were placed between the leaf springs and the hinge. The properties of these springs were chosen based on the stiffness of the rubber bushing. Figure 22 illustrates the joint at the hinge and the new connection incorporated in the model.
After incorporating the above mentioned modifications, the pendulum simulation was performed again. The results from the simulation are shown in Figures 23, 24 and 25. The figures show good correlation between the test and simulation results.
A full-scale crash test of a corner impact into a New Jersey safety shape barrier with the modified version of the detailed C2500 pickup truck model was also used to validate the rear suspension updates. Comparisons of the modified version of the detailed model to the crash test should show the ability of the new model to realistically represent dynamics and overall crush.

A compliance test in accordance with NCHRP Report 350 of the New Jersey safety shaped barrier was conducted for FHWA (Test 405491) in September 1995. The test vehicle was a 1991 Chevrolet C2500 pickup truck. The vehicle speed was 100 km/hr and impact angle was 25 degrees [7]. Figure 26 shows different stages from the crash test and corresponding simulation results.

Comparing the deformations of the simulation with the crash test, it is found that the simulation shows a reasonable agreement with the physical test. The deformations of the bumper and the right front side as well as the motion of the hood are similar in the test and simulation. In both test and simulation, the barrier contained and redirected the vehicle satisfactorily. For the time periods of impacting the barrier, becoming parallel with it, and separating from it, test and simulation show reasonable correlations. Few differences were observed between the test and the simulation. These differences could be attributed to the front suspension, which will be a subject of similar validation in future research.

CONCLUSIONS

The purpose of this study was to upgrade the rear suspension system of the detailed C2500 pickup truck model. To ensure this, several parts were added to the finite element model. A rail cross-member was digitized, modeled with shell elements and included in the pickup truck model, replacing the original rail constraint modeled only with beam elements. A leaf spring model was included, consisting of hinges and mountings modeled with shell elements and joints. Finally, shock absorbers at the rear axle were implemented in the model.

Simulations were conducted and the results were compared to pendulum tests in terms of deformation, displacement, and acceleration. The initial comparisons showed significant differences between the simulations and the physical tests relative to lateral and longitudinal displacement of the rear axle and bending behavior of the leaf springs. Several modifications were subsequently made to improve the behavior of the rear suspension system in the model. The major modifications included changing the leaf spring geometries, altering the representations of the hinges, adding detail for the frontal and back mountings, changing the leaf spring constraints, and adding new parts. New simulations were conducted and subsequent comparisons showed a reasonable agreement with the pendulum tests. The modifications significantly improved the model’s behavior. The lateral and longitudinal displacements of the rear axle as well as the bending behavior of the leaf springs and the accelerations at various locations showed good correlation with the physical tests.

An angled impact into a New Jersey safety shaped barrier following NCHRP Report 350 procedures was simulated to further validate the model. The comparison of the simulation and the crash test video showed a
reasonable agreement between the physical test and the simulation. A good correlation for the deformation and the general motion could be seen. After comparing the simulation results presented in this paper, it can be said that the modified version of the detailed Chevrolet C2500 pickup truck model shows significantly improved behavior in all simulations. Future studies will be conducted on the front suspension system of the pickup truck model to further improve the behavior and capabilities of the detailed C2500 pickup truck model.

Figure 26: Full-Scale Crash Test and Simulation Comparison at Different Stages of the Impact

ACKNOWLEDGEMENT

The authors wish to acknowledge the U.S. Department of Transportation (DOT) Federal Highway Administration (FHWA) for supporting this research study.
REFERENCES


