Validation of a Single Unit Truck Model for Roadside Hardware Impact

ABSTRACT

The objective of this research is to validate the Finite Element (FE) model of a Ford F800 Single Unit Truck (SUT). The FE Model of the SUT was developed at the FHWA/NHTSA National Crash Analysis Center (NCAC) at the George Washington University. The SUT model has been used by several transportation safety researchers to evaluate and improve roadside safety hardware. In this study, the characteristics of the SUT model were investigated and several modifications were incorporated in the model to accurately simulate its interaction with roadside safety hardware. A full-scale crash test of the Ford F800 SUT impacting an F-shape Portable Concrete Barrier (PCB) was conducted at The Federal Highway Administration’s (FHWA) Federal Outdoor Impact Laboratory (FOIL) and used in the validation. The response of any vehicle impacting a PCB is primarily governed by the mass and suspension characteristics of the vehicle. Detailed methods for modeling the truck’s suspension components are discussed. The comparison between the FE Model and the crash test in terms of overall impact kinematics, component failure modes, velocity and acceleration at various locations of the vehicle are presented.

Keywords: LS-DYNA, FE Model, SUT, Roadside Hardware, PCB.

INTRODUCTION

The importance of the Single Unit Truck (SUT) for roadside safety research is given in the National Cooperative Highway Research Program (NCHRP) Report 350, published in 1993 [1]. The SUT (8000S) is one of the vehicles used in roadside hardware evaluation and certification crash tests for test level 4. Due to an abundance of use of roadside barriers, their safety has been the subject of many studies and investigations. It is economically not feasible to perform full-scale crash tests on a wide range of parameters which influences the performance of these safety features. Impact simulations utilizing nonlinear finite element analysis have thus become effective tools in designing and evaluating these systems. Once successful in validating one or more finite element models to represent full scale crash tests, the model can be applied to new crash scenarios. Varying crash parameters, like impact angle and vehicle speed, or original design of the roadside safety feature, will lead to an optimization process of the design of the roadside hardware itself. Therefore, there was a need to develop a finite element model of this vehicle to aid in evaluating and improving roadside devices.

A FE Model of the 1996 Ford F800 18,000 lb truck was developed at the FHWA/NHTSA National Crash Analysis Center (NCAC) at the George Washington University (GWU). The model’s primary purpose was to be used as a bullet vehicle for computational evaluation of roadside safety hardware, and as such it does not warrant modeling complexity ordinarily expected in a vehicle crash analysis model. In fact, too much complexity in a bullet vehicle could be detrimental to its primary purpose because of an increased computational burden that may reduce the efficiency of the analysis.

When impacting roadside safety hardware, the vehicle often experiences stability problems. The suspension response is particularly important for truck impacts. The severity of the crash as well as the characteristics of the truck, especially the low static stability factor (track/2h) due to higher center of gravity, makes it less stable and more susceptible to rollover than other vehicles. Therefore, the suspension and the steering
system has been the major focus of this study. A more detailed front suspension system
of the SUT is incorporated in the model. The model was validated using a full scale
 crash test into a 42” F-shape concrete barrier. A summary of the validation results are
 presented in this paper.

**FORD F800 SUT MODEL**

The Ford F800 SUT is a multi-purpose truck. The truck is 8.5 meters long and
has a wheel base of 5.3 meters. The truck has a Diesel engine coupled to an automatic
transmission with a rear wheel drive configuration. Version F800-V01d was used in this
study. The FE model consists of 24,915 nodes, 1492 solid, 124 beam and 20109 shell
elements. Under today’s computing standards, the model can be handled by entry level
workstations, and as such meets its role as a bullet vehicle for roadside hardware safety
analysis. The Ford F800 SUT finite element model is shown in Figure 1.

![Finite element model of the Ford F800 SUT](image)

*Figure 1: Finite element model of the Ford F800 SUT*

An important aspect of a bullet vehicle is its ability to simulate its overall
kinematics, which, in case of a truck implies existence of accurate models for mass
distribution, global bending and torsion stiffness, and response of wheels and suspension
components [2]. For example, in the case of low impact angle of an SUT into a portable
concrete barrier (PCB), the spinning of wheels and suspension models should capture the
truck riding up the barrier. If such capability is missing, the simulation results will not
provide reliable information about barrier’s effect on the trucks kinematics. The original
SUT FE model had fixed tires, the climbing of the tire and entire vehicle along the slope
of the barrier was not correctly represented. Using fixed tires in head-on crash is
acceptable, but in the case of oblique impacts, especially when the tire is the first object
that comes in contact with the barrier, detailed wheel, suspension and steering models are
needed. The front leaf spring was connected to the front axle using rigid bolt, the failure
of the bolt was not incorporated. The front suspension in the original SUT FE model is
shown in Figure 2. The front suspension components are fully upgraded in this study and
a steering system has also been incorporated to better capture the kinematics of the SUT
for impacts into roadside hardware.

*orc*
Figure 2: Front Suspension of the Ford F800 SUT Finite Element Model

MODEL UPGRADES

To make the SUT model more applicable for roadside hardware simulations, several details have been added to the model. First, the missing components such as frame rail reinforcement, dampers for the front suspension and steering system were added to the model. For a vehicle impacting a roadside hardware, the angular displacements (i.e. yaw, pitch and roll) of the impacting vehicle is greatly affected by its suspension and steering characteristics. Hence accurate representation of the suspension and steering system is needed in the finite element model.

The detail view of the front suspension and its connections are shown in Figure 3 [2]. The picture of the front suspension taken from the test vehicle is shown in Figure 4. An important part of the suspension that was not implemented in the earlier model was the shock absorber (damper). The damper and its attachment between the frame rail and leaf spring play a significant role on the kinematics of the front suspension. This detail was added to the SUT FE model.

Figure 3: Front Suspension detail of the Ford F800 SUT
The failure mechanisms for the front suspension were not implemented in the earlier model. In most impacts with roadside hardware, failure of the front suspension of the SUT often occurs shortly after impact. To a large extent, the shape of the barrier base influences the severity of the suspension damage and mechanism of its failure. The failure occurs in connections between suspension parts like shearing of the U-bolt, spring pin etc., that then causes the connected parts to separate. The leaf spring connects the axle and the frame rail and is attached to the axle using U-bolts with a specified amount of torque. The ends of the leaf springs are connected to the frame rail brackets through spring pins. The leaf spring, spacer, mounting bracket and shim are all bolted to the axle. All of the above details were included in this study. The spring pin, shackle pin and the U-bolts were modeled as beam elements. The leaf spring, spacers, frame and shackle brackets were modeled as shell elements. The details added to the SUT front suspension is shown in Figure 5.

Figure 4: Front Suspension of the Ford F800 SUT

Figure 5: Front Suspension details incorporated in the Ford F800 SUT FE model
The U-bolts are SAE Grade 8 material with ¼" diameter. The U-bolts were modeled as beam elements. The material definition is an LS DYNA material type 24 or "Piecewise Linear Isotropic Plastic" model [4, 5]. Material properties of Grade 8 steel were assigned and a stress-strain curve was defined. In addition, plastic strain failure was incorporated in the material model to simulate failure of the U-bolts. Pre-tensioned non-linear discrete springs were used to connect the U-bolts to the axle with the specified amount of torque. This model was run for 20 ms to establish equilibrium. The nodal coordinates at the equilibrium state was extracted and used as the initial state. This approach ensures a constant clamping force on the U-bolts, which, ensures realistic frictional forces to be generated between the leaf spring and the axle. The bolt connecting the leaf spring, spacer, mounting bracket and shim to the axle was modeled as "CONstrained generalized WELD SPOT" in LS-DYNA [5]. This card provides the means to incorporate normal force and shear force leading to failure of the bolt.

The steering system was another important feature added in this study. The steering system plays a crucial role in determining the trajectory and the stability of the truck especially during redirection events. The steering system for the Ford F800 FE model is shown in Figure 6. This version incorporates a detailed model of the steering linkages. Rigid body elements are linked with joints in such a way to reproduce the actual mechanism of the steering system of the truck. Steering input was not modeled in this version. Linear rotational springs were placed between the steering linkages and also between the tire and the axle joint to keep the wheels from oscillating [3]. The numerical value for the rotational spring was established by trial-and-error matching of the steering wheel response between the test and the simulation. Revolute joints were used between the axle and the tires. The spinning of the tires was modeled using *INITIAL VELOCITY GENERATION [5].

![Figure 6: Front steering system of the Ford F800 SUT FE model](image)
CRASH TEST DESCRIPTION

To validate the above mentioned upgrades, and more specifically the front suspension and steering system, simulations were performed and compared to a full-scale crash test. A 25 degree corner impact of the Ford F800 SUT into a 42” tall F-shape Portable Concrete Barrier (PCB) was performed as part of this study at FHWA’s Federal Outdoor Impact Laboratory (FOIL). The pre-impact test setup is shown in Figure 7. The focus of this test was on understanding the behavior of the front suspension and steering system for oblique impacts into PCB’s. The box on the truck was removed to isolate any inertia effects of the box on the overall kinematics of the truck during impact. The front hood was removed for better high speed camera coverage on the front suspension. The details of the test vehicle and test impact conditions are listed in Table 1. Several accelerometers were mounted on the test vehicle, including the engine, front axle, frame rails, rear axle and at the truck’s center of gravity.

![Figure 7: Pre Impact test setup](image)

Table 1: Full Scale Crash Test Description

<table>
<thead>
<tr>
<th>Test Number 03007</th>
<th>August 12, 2003</th>
</tr>
</thead>
<tbody>
<tr>
<td>Test configuration</td>
<td>Vehicle into F-shape Barrier at a 25 degree Impact Angle</td>
</tr>
<tr>
<td>Vehicle</td>
<td>1995 Ford F800</td>
</tr>
<tr>
<td>Vehicle mass as tested</td>
<td>3762 kg</td>
</tr>
<tr>
<td>Engine Type</td>
<td>7.9 liter diesel</td>
</tr>
<tr>
<td>Impact Velocity</td>
<td>50 km/hr</td>
</tr>
</tbody>
</table>

For the finite element simulation, parts were removed from the SUT model in accordance to the test setup. Accelerometers were defined in the finite element model at similar locations as in the full scale crash test. An “F” shape PCB FE model similar to the one used in the full scale crash test was created for this study. The finite element simulation setup is shown in Figure 8. The mass and center of gravity comparison between the test vehicle and the FE simulation are listed in Table 2.
Figure 8: Simulation setup for test 03007

Table 2: Mass and CG comparison

<table>
<thead>
<tr>
<th></th>
<th>Test Vehicle</th>
<th>FE Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass as tested (kg)</td>
<td>3762.0</td>
<td>3762.5</td>
</tr>
<tr>
<td>X (cm)</td>
<td>312.7</td>
<td>310.3</td>
</tr>
<tr>
<td>Y (cm)</td>
<td>106.0</td>
<td>112.0</td>
</tr>
<tr>
<td>Z (cm)</td>
<td>Not measured</td>
<td>75.0</td>
</tr>
</tbody>
</table>

MODEL VALIDATION

Upon completing the model setup, simulations were performed and compared to the full scale crash test. The simulations were performed on a Silicon Graphics PowerChallenge system shared memory, SMP supercomputer consisting of 32 processors. The SMP version of the LS-DYNA, version 960 was used. The simulation was run for 1 second duration of impact using 4 processors. The CPU time for the complete simulation was 20 hours. A fixed time step of 3.5 microseconds with mass scaling was used. This resulted in a 1.4% mass increase at time zero and increased to 1.6% at the end of the simulation. The accelerometers were modeled in the finite element simulations by rigid bodies mounted on the respective part of the finite element model. Acceleration measurements are very sensitive and are susceptible to high frequency noises. Therefore, an SAE-Class 60 filter was used to reduce numerical noise effects in the simulation for nodal acceleration records, as well as for the crash test data.

The accuracy and fidelity of the simulation was studied with respect to general kinematics of the truck during impact and time history records at different locations. The general kinematics of the truck can be compared visually from the images captured with the high speed cameras. The comparison between the crash test and simulation at different instances in time during impact is shown in Figure 9. The global kinematics of the truck matches well with the crash test. The truck exhibits good rigid body motion; the images show the truck impacting, riding along and then leaving the barrier.
In the crash test, seven “F” shape barriers were connected together along the impact line. Two additional rows of barriers were positioned behind the impact point as shown in Figure 10, and filled with gravel to minimize the deflection of the barrier. The purpose of this setup was to impart the maximum force on the front suspension and steering components to study their failure limits. Even though every effort was made to rigidize the barrier, the impact severity caused the impacted barrier to deflect by a small amount, creating a ridge between the impact barrier and the subsequent barrier. The ridge created between the second and third barrier caused the right front tire to snag for a small duration. The tire marks on the barriers is shown in Figure 11. To capture this behavior in the simulation, the barriers need to be modeled as deformable and the gravel in between the barriers should also be included. The SUT FE model along with the “F” shape barrier consists of 27000 elements. The model size will increase significantly to incorporate the additional details of the soil, pins and gravel behind the impacted barriers. This increases model complexity, significantly increases the computation time and is beyond the scope of the current study. For simplicity, the seven “F” shaped barriers were assumed as one single rigid unit in this study.
Figure 10: Barrier Layout in the full scale crash test

![Barrier Layout](image)

Figure 11: Front right tire marks on the barrier

Comparing the truck deformations from the simulation to 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 front suspension are similar in the test and simulation. Upon impact, the bolt connecting the leaf springs to the front axle shears off and the front axle slides back along the leaf springs as the truck continues to ride up the barrier. This behavior is captured in the simulation. The post impact pictures of the front suspension from the crash test and simulation is shown in Figure 12.

![Post impact picture](image)

Figure 12: Post impact picture of test and simulation
For the next level of comparison, the acceleration and velocity time histories of the crash test and simulation taken from the accelerometers were compared. The time history between crash test and simulation is shown in Figures 13 through 18. The angular displacement of the truck derived from the gyroscope mounted at the center of gravity of the truck is shown in Figure 13. For the time periods of impacting the barrier, becoming parallel with it, and separating from it, test and simulation show reasonable correlations. The pitch of the truck correlates well with the test. The roll and yaw of the truck are a few degrees off compared to the test. This could be attributed to the tire snag between the second and third “F” shape barrier observed in the crash test. The tire snag slows the yaw and roll rate of the truck in the test. Since this phenomenon was not included in the simulation, the truck rolls a few degrees more and yaws faster compared to the test.

![Graph](image1)

**Figure 13:** Vehicle Angular displacements

![Graph](image2)

**Figure 14:** Z acceleration at Engine top
Figure 15: X velocity at vehicle CG

Figure 16: Z velocity at vehicle CG

Figure 17: Z velocity at front right rail

The accelerometer mounted on the engine top gives an insight into the engine kinematics during the impact. The Z acceleration of the engine top is shown in Figure 14. The initial peaks of the acceleration time history correlates reasonably well between the test and the simulation for the most part of the event. The higher peak at 70 millisecond observed in the simulation is attributed to the truck impacting the ground after separating from the barrier. The X and Z velocity from the accelerometer mounted at the truck center of gravity are shown in Figures 15 and 16. The forward velocity of the truck matches with the test for the first 5 millisecond. Between 5 and 10 millisecond the forward velocity decreases faster in the crash test than in the simulation. This is due to the front right tire snagging between the second and third barrier. Since the simulation assumed all the barriers as one rigid entity, the forward velocity of the truck does not show the velocity drop observed at 5 milliseconds.

The Z velocity at the front right rail is shown in Figure 17. The accelerometer time history show good correlation between the crash test and simulation. The frame rail velocity is similar between the test and simulation for the first 20 millisecond. After 20 milliseconds, the truck in the simulation separates faster from the barrier compared to the crash test. The reason is attributed to the friction modeling between tires – barriers, and tires – ground. The friction modeling between the tires and the barrier has a significant impact on the global kinematics of the truck. Several simulations were performed to find an optimal coefficient of friction between the tires and the barrier. In reality as the front right side tire impacts the barrier and starts riding on the barrier the normal force on the front left side tire should start increasing due to the weight shift. The simulation shows this behavior, but the force keeps oscillating due to spinning tires. This results in an
unrealistic friction modeling between tires and the ground. Further study the friction modeling effects on the global kinematics of the truc-

CONCLUSIONS

The purpose of this study was to update and validate the Force for use in roadside hardware simulations. To ensure this, more details in the finite element model. The frame rail reinforcement, front suspension system were added to the SUT model. The front suspension more detail to accurately capture the SUT kinematics in oblique im hardware.

Simulations were conducted and the results were compared to test in terms of overall deformation and time history plots. The simulation consistent with the crash test. The global kinematics of the truck were from the crash test. The acceleration time histories also show good correlation and the simulation. The simulation assumed the “F” shape barrier to this was not the case in the test. This implies that soil should be more that are driven through the barriers to the soil be included in the increases computational time and further work will be needed to v Friction between the tires-barrier and the tires-ground needs further v the yaw of the test. This would provide consistent normal force on the

ACKNOWLEDGEMENT

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

REFERENCES