



Working Paper

NCAC 2004-W-002

October 2004

Finite Element Modeling and Validation of a 3-Strand Cable Guardrail System

Pradeep Mohan
Dhafer Marzougui
Nabih Bedewi

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

Leonard Meczkowski

Turner Fairbank Highway Research Center
Federal Highway Administration, USDOT
6300 Georgetown Pike, McLean VA 22101-2296 USA

Email: Kenneth.opiela@fhwa.dot.gov

This working paper is a compilation of recent efforts and findings intended to solicit feedback on the approach, scenarios analyzed, findings, interpretations, conclusions, and implications for practice resulting from the efforts of the research team. 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

The primary purpose of longitudinal safety barriers, such as cable barriers, is to contain and/or redirect errant vehicles that depart the roadway, hence keeping them from entering opposing travel lanes or encountering terrain features and roadside objects that may cause severe impacts. In this study, a detailed finite element model of a three-strand cable barrier was developed and validated against a previously conducted full-scale crash test. The full-scale crash test and simulation were setup for an impact of the cable barrier with a 2000 kg pickup truck at an angle of 25 deg and an initial velocity of 100 km/hr. This setup is in accordance with the National Cooperative Highway Research Program (NCHRP) Report 350 guidelines for Test Level 3 safety performance. This paper provides guidelines for simulating cable barrier guardrail systems. Detailed methods for system simulation involving dynamic interactions of soil/post, post/hook bolts, cable/ hook bolts and cable/truck are discussed. Results from the simulation and comparisons with the full-scale crash test are presented.

Finite Element Modeling and Validation of a 3-Strand Cable Guardrail System

INTRODUCTION

Three-strand cable guardrail is a longitudinal barrier used to contain and/or redirect errant vehicles that depart the roadways. This guardrail gradually redirects an impacting vehicle by elastically stretching the cables, minimizing forces on the vehicle occupants. During an impact, the kinetic energy of the vehicle is dissipated by breaking and bending of the posts and stretching of the cables. These barriers are becoming more and more popular than the more rigid systems like W-beam guardrails, Portable Concrete Barriers and Median Concrete Barriers. This is attributed to low installation costs, effective vehicle containment, and redirection over a wide range of vehicle sizes. Figure 1 shows a typical 3-strand cable barrier installed on one of the interstate highways. These cable guardrails are recommended for use in locations where there is sufficient space for lateral deflections. Due to an abundance use of these 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.

The objective of this study is to develop a finite element model that can accurately capture the crash behavior of 3-strand cable guardrails. The finite element model is validated using full-scale crash test data of a 2000 kg Chevrolet C2500 pickup truck impacting the length of need (LON) of the barrier at a nominal speed and angle of 100 km/h (60 mph) and 25 degrees, respectively. This test was intended to evaluate the structural strength of the barrier and its ability to contain and redirect the vehicle. The 3-strand cable guardrail used in this test is the Washington 3-strand cable barrier anchored with New York cable terminal ends. This approach enables us to identify the crash sensitive components of the 3-strand cable guardrail safety structure under investigation. Identification of the sensitive parameters can be used as feed back in an optimization process to design new and improved guardrail system that will effectively contain and redirect cars and trucks.

The 3-strand cable guardrail consists of three 19 mm round wire cable (7 wires per strand) having a minimum tensile strength of 110 KN. The cables are connected to S75 X 8 rolled steel posts through hook bolts. The hook bolts, as installed develops an ultimate pull open strength in the range of 2 KN to 4.5 KN applied in a direction normal to the longitudinal axis of the post. The ends of the cable are anchored to the ground through concrete blocks. The cables are pre-tensioned using spring cable end assemblies having a spring rate of 80 ± 8 N/mm and a minimum available throw of 150 mm. The tension in the cables is set depending on the atmospheric temperature and is adjusted every few weeks.

METHODOLOGY

To develop an accurate 3-strand cable guardrail model, several key features of the barrier design must be understood. Once known, these features may be incorporated into the FE model, resulting in a realistic

representation of the barrier system. Simplification and assumptions can be made to reduce computational costs; however, special care must be taken not to introduce inaccuracies into the model.

Four major issues are important when modeling the 3-strand cable guardrail for impact simulation. These issues are listed as follows: 1 - Soil/post interaction, 2 - Post/hook bolts interaction, 3 - Cable/hook bolts interaction and 4 - Cable/vehicle interaction. These issues will be treated in detail in separate sections below.

Simulation of Soil/Post Interaction

The simulation of the soil/post interaction, which has a significant influence on the response of the guardrail during the impact event, is a complex and important issue. Soil/post interaction was modeled using lagrangian mesh. Figure 2 shows one such soil/post model. The soil in the model is represented by a cylindrical block. The dimensions of the cylindrical block were chosen based on the embedment length of the steel post. These dimensions were chosen such that the behavior of the soil and soil/post interactions is accurately captured with reasonable computation time. The outer boundaries of the soil model were constrained using the boundary-spc option. The soil block is made up of solid elements surrounding the post. The shape of the post is incorporated in the soil model to represent the soil/post interaction. The material constitutive model used for the soil is the “soil and crushable foam model” [5] (type 5). The parameters used for soil material model were obtained from literature [4]. A “surface to surface” [5] sliding interface is defined between the outer faces of the post and inner faces of the soil block to simulate the contact between them. Component simulations were conducted using a single post and soil model to study the soil/post interaction. These component simulations consisted of impacting the post in lateral and longitudinal direction with a surrogate vehicle (bogie) model. These component simulations were compared to physical tests in terms of dynamic deflection of the posts and the acceleration data from the test.

Simulation of Post/Hook Bolts Interaction

Figure 3 shows one of the posts to hook bolt connection. It is important to accurately model the post/hook bolts connection as it influences the cable pull out from the hooks during the impact event and therefore the cable resistance to the vehicle. The hooks were modeled with discrete beam elements and the bolts were connected to the hook ends with discrete linear springs. The stiffness of the springs was selected in such a way to represent the correct clamping force of the hooks to the posts. The material constitutive model used for the hook is the “piecewise linear plasticity model” (type 24) [5]. The material properties were obtained by correlating the hook strength with standard pull tests in lateral and vertical direction [1]. Cowper-symonds strain rate effects were included for the material model of the hooks as the cable pull out from the hooks is a highly dynamic event. The material model used for the bolts is the “rigid” model. The true shape of the bolt was modeled using lagrangian mesh as the element size had no influence on the model computational time step. Automatic single surface contact was defined between the bolts and the posts. The hooks are mounted to the posts through holes on the edge of the posts. Elements with “null” material properties were created with these holes and were tied to the posts using the tied contact. This approach made it possible to achieve a good contact interaction between the hooks and the post holes, while keeping a reasonable element size on the posts, thus controlling computational costs.

Simulation of Cable/Hook Bolts Interaction

The angular displacements of a vehicle impacting these 3-strand cable barriers are greatly influenced by the cable/hook bolts interaction. During the impact, the cable pulls out of the hooks in a controlled manner allowing lateral deflection of the struck vehicle. This makes it very important to accurately model the cable to hook bolt pullout. Figure 4 shows the cable to hook bolt connection. Cables are modeled using discrete beam elements. Elastic material model is used for the cables. Null elements are modeled on the cables and are tied to the nearest beam elements using nodal rigid bodies. Similarly null elements are modeled on the hook beams and an automatic single surface contact is defined between the cable and the hook null elements. This approach allows for accurate contact modeling with friction between the cable and hooks which significantly influence the cable pull out.

Simulation of Cable/Vehicle Interaction

Simulating the contact interaction between the cable and vehicle is the most challenging issue in these impact scenarios due to contact instabilities. In the initial simulations the most common LS-DYNA contact was used between the cable and the vehicle. In almost all iterations dyna core dumped due to contact instabilities. This is mainly because of significant amount of sliding between the vehicle and the cables. Automatic_single_surface contact uses segment based projection which leaves gaps between two segments when projected. The slave nodes in contact with these segments enter the gap during sliding leading to unrealistic snagging between the cable and the vehicle. This also leads to distorted elements which causes the simulation to terminate before the final simulation time is reached. The best approach for contact modeling in such situations is to use nodal based projections such as nodes_to_surface and surface_to_surface [5]. This eliminated the snagging and element distortion problem and allowed for good contact interactions between the vehicle and the cable. The type 26 “automatic general” contact was also used between the cables and the side of the truck. This contact does additional penetration checks such as edge to edge and eliminated penetration problems.

MODEL VALIDATION

The finite element (FE) model of the 3-strand cable guardrail system was developed using the preprocessors Hypermesh and Easycrash. Specific details and dimensions for the 3-strand cable guardrail were obtained from the test report [2]. The FE model of the C-2500 pickup truck [3] was imported into the guardrail model to generate a full FE system model and the full scale crash test conditions were replicated. Figure 5 shows the full FE model of the system. After all design features of the 3-strand cable guardrail were implemented in the FE model, simulations were performed to validate the model using a previously conducted full-scale crash test.

The test performed (test designation 3-11) involved a 2000 kg Chevrolet C2500 pickup truck impacting the length of need (LON) of the barrier at a nominal speed and angle of 100 km/h and 25 degrees, respectively [2]. The test was intended to evaluate the structural strength of the barrier and its ability to contain and redirect the pickup truck. This test was conducted at the Texas Transportation Institute’s Facility. The 3-strand cable guardrail used in this test is the Washington 3-strand cable barrier anchored with New York cable terminal ends.

Figure 6 shows the full scale crash test results on the left and the simulation results on the right at different impact stages. The comparative figures indicate that the finite element simulation reasonably captures the basic sequence of events. The bottom cable engages the lower part of the bumper, slips

under and the truck under-rides the lower cable. The top two cables engage the truck just above the bumper and effectively redirect the truck. Cable pull out soil/post dynamic deflection correlated well with the full-scale crash test. The angular displacements of the pickup truck in the simulation were similar to those in the full-scale crash tests. Recorded test data such as maximum dynamic deflection allowed by the cable barrier and vehicle center of gravity accelerations compared well with the simulation results. These findings show that the FE model accurately represents the 3-strand cable guardrail. The friction between the ground and the vehicle tires has considerable effect on the vehicle angular displacements. The ground friction is modeled in the simulation using a simple linear friction formulation. Realistic friction modeling will further improve the model fidelity in accurately predicting the vehicle angular displacements. Overall, however, the finite element simulation replicates the basic behavior of the actual full-scale crash test.

CONCLUSIONS

This paper describes the methodology for modeling and simulation of the 3-strand cable guardrail system. This methodology can be used for modeling and simulation of similar guardrail systems. The use of various FE options to represent the physical behavior of 3-strand cable guardrail was examined. Detailed component simulation was done to better represent the soil/post and cable/hook interaction. A special modeling technique was used to connect the hooks to the posts. The simulation results presented in this paper demonstrate a prediction mode for highway barrier impacts based on a model validation with a full scale crash test. These results are preliminary and show a first attempt at such prediction. The model correlated well with the full-scale crash test. This model will be used in future studies to analyze, evaluate and improve the safety performance of cable barriers.

ACKNOWLEDGEMENT

This research is funded by the US Department of Transportation (DOT), Federal Highway Administration (FHWA), Turner-Fairbank Highway research center.

APPENDIX-1



Figure 1: Three-strand cable guardrail.

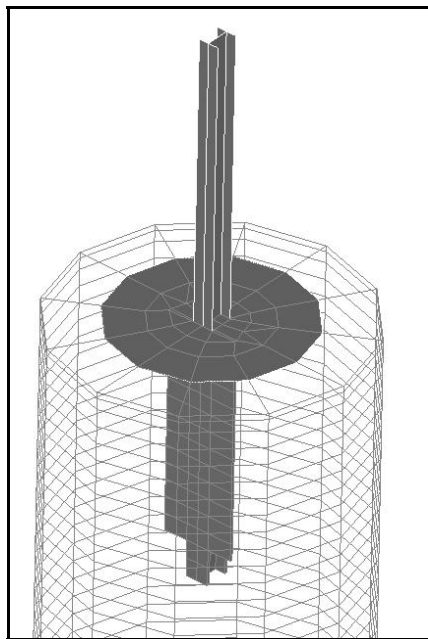


Figure 2: Soil/Post model

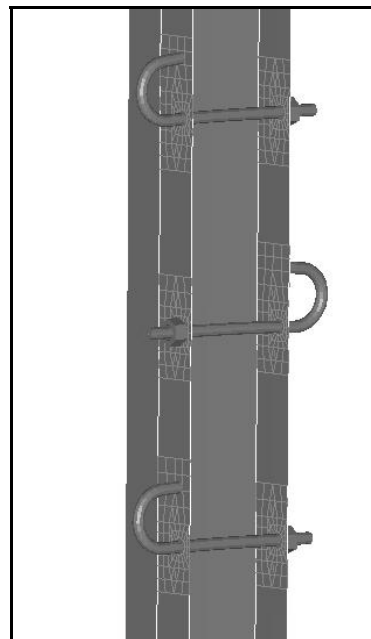


Figure 3: Post and Hooks model..

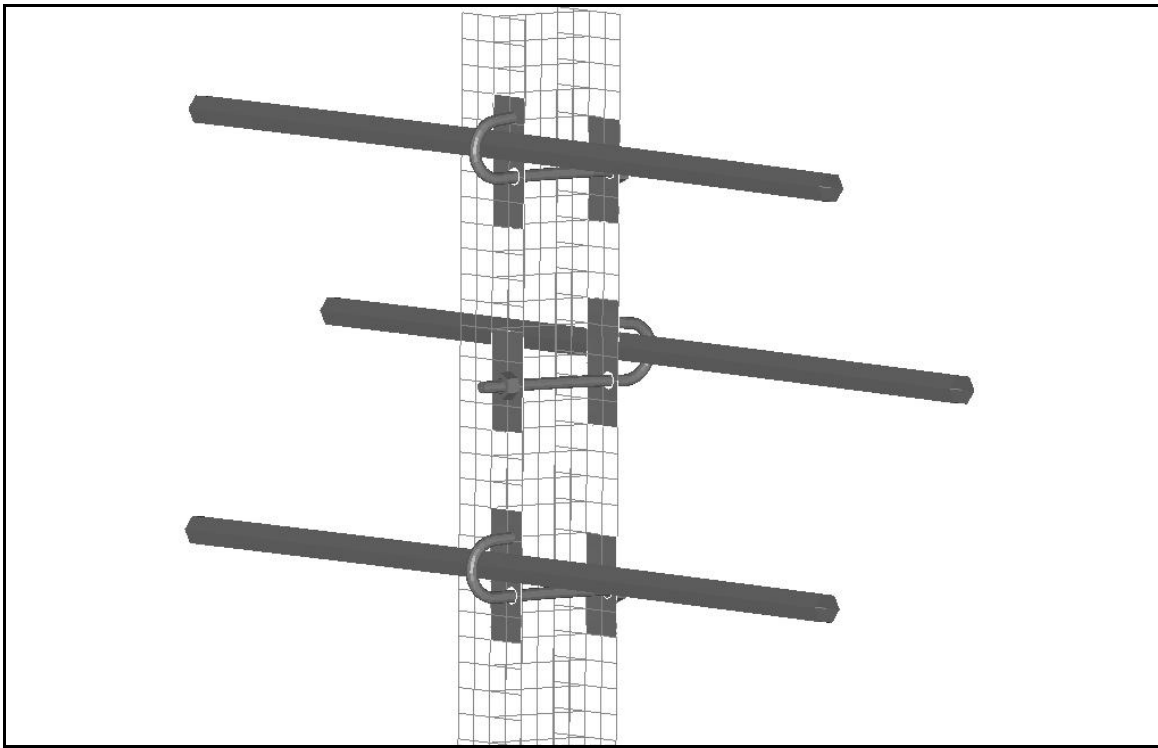


Figure 4. Post, Cable and Hooks setup.

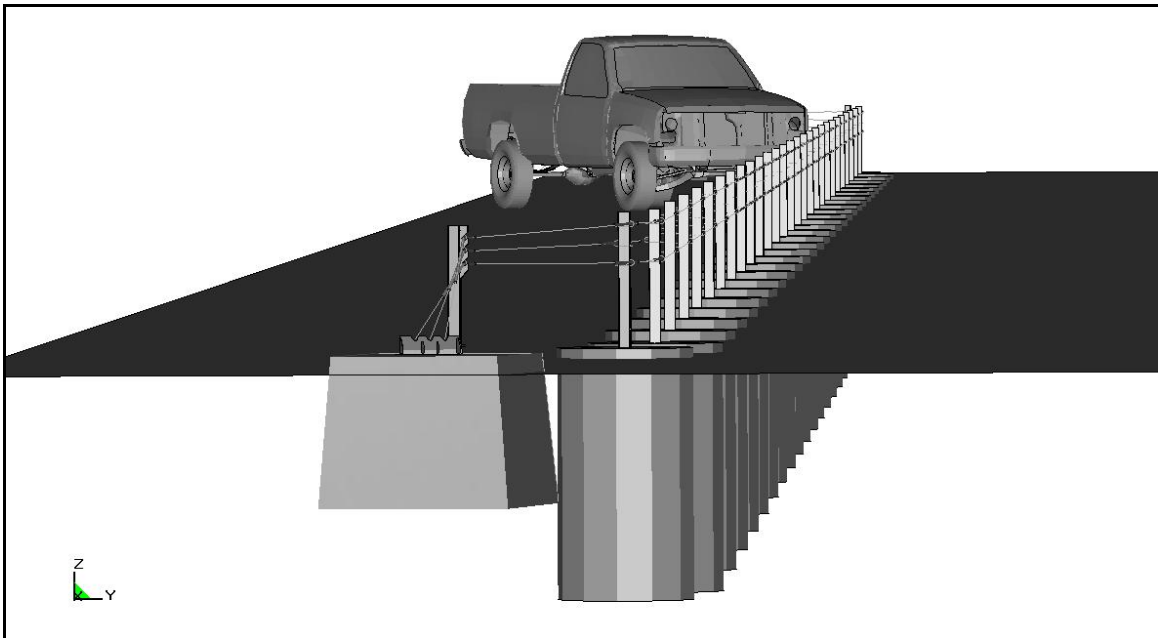


Figure 5: FE system model of 3-strand cable guardrail with the C2500 pickup truck.

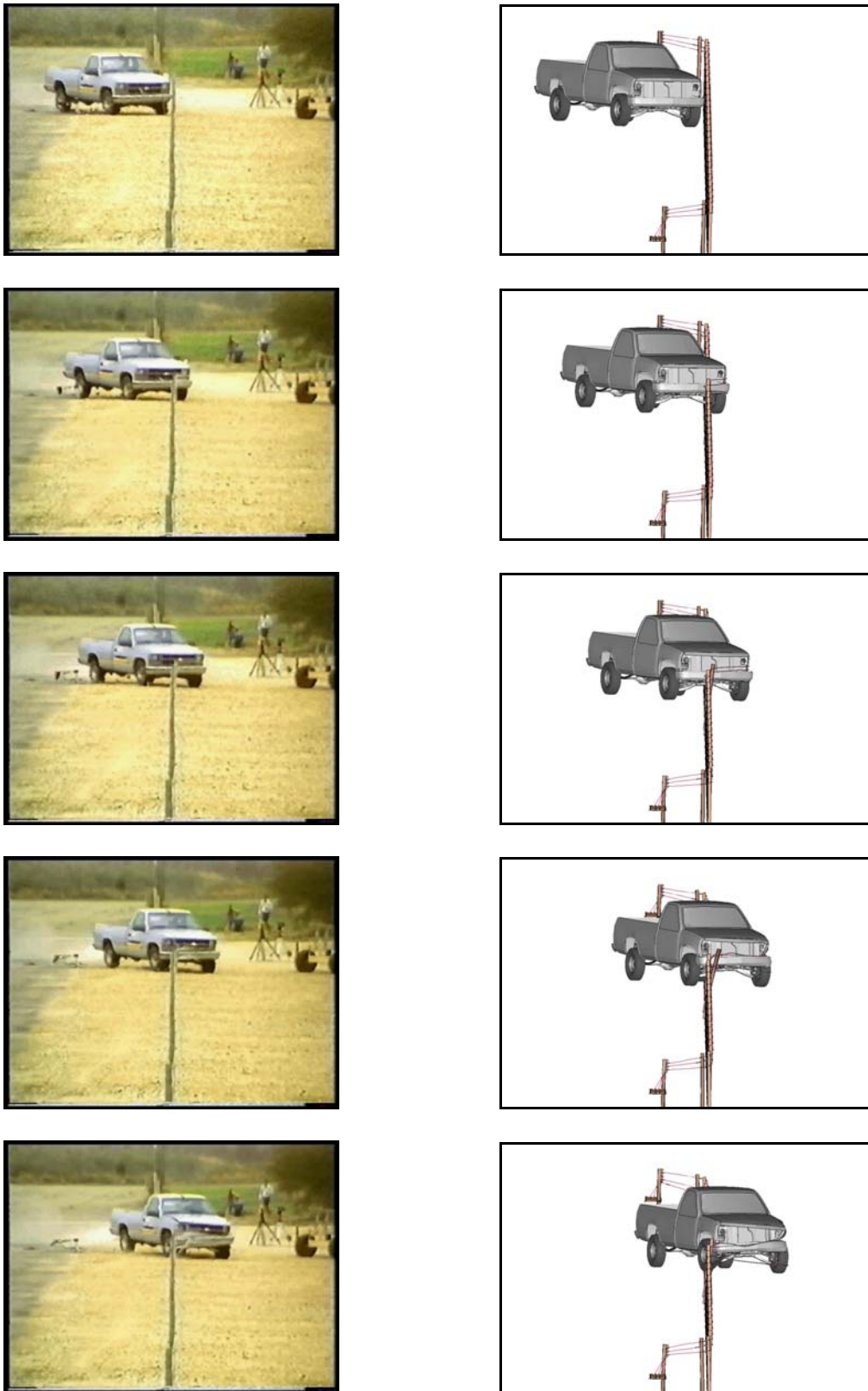


Figure 6: Full-scale crash test/simulation comparisons (continued on next page).

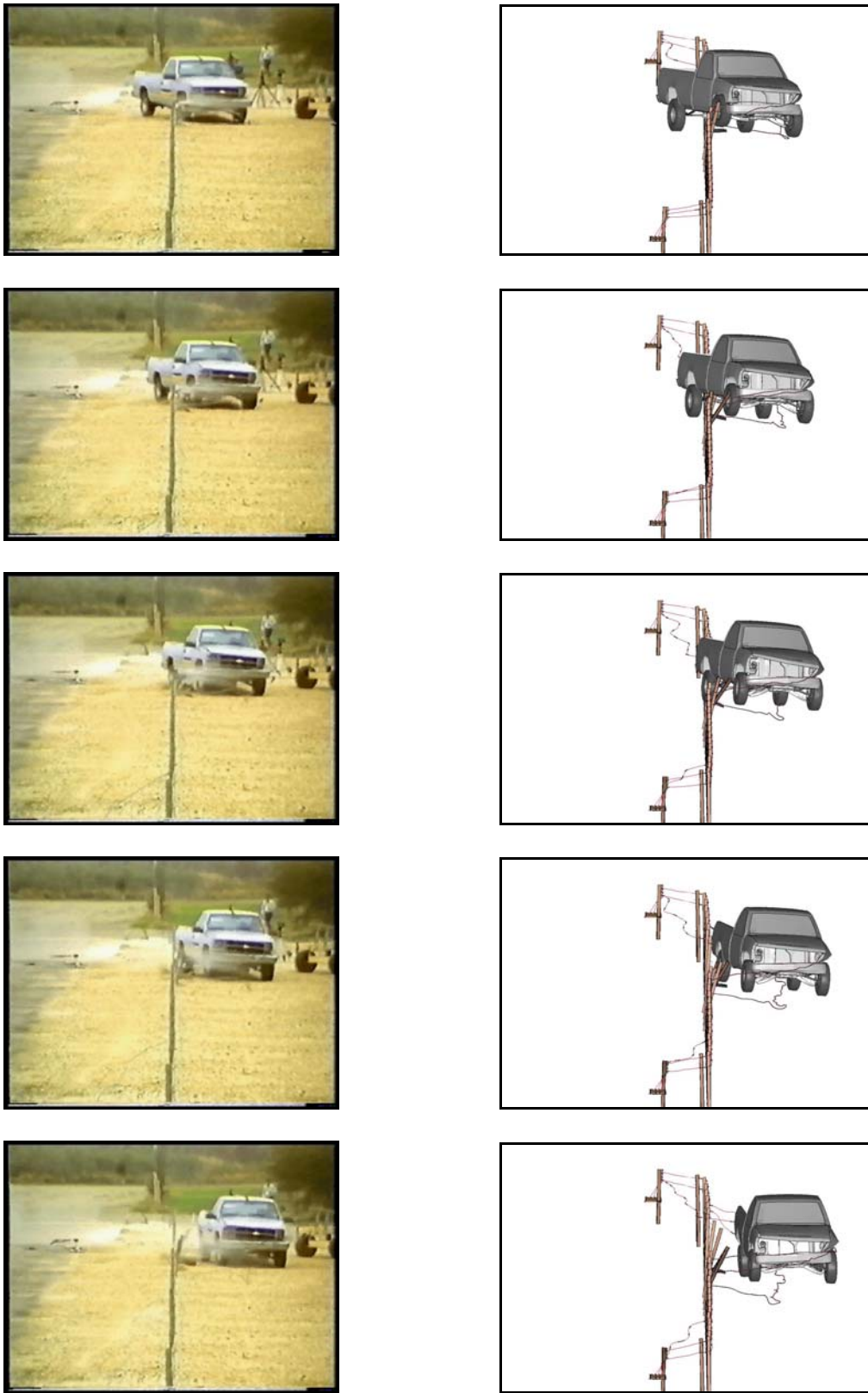


Figure 6: Full-scale crash test/simulation comparisons (continued from previous page).

REFERENCES

1. Reid, John and Coon, Brian, "Finite Element Modeling of Cable Hook Bolts", 7th International LS-DYNA conference,
2. Bullard, Lance. Menges, Wanda., and Buth, Eugene, "NCHRP Report 350 Test 3-11 of the Washington 3-strand cable barrier with New York cable terminal", Contract No. DTFH61-97-C-00039, Project No. 404211-8, (March 2000).
3. Marzougui, D., A. Zaouk and C.D. Kan, "Development of a C-2500 Pick-Up Truck Model for roadside hardware impact", Transportation research board annual conference, (1996)
4. Marzougui, D., Kan, C.D and Eskandarian, A, "Safety Performance Evaluation of Roadside Hardware Using Finite Element Simulation", ASCE Fourteenth Engineering Mechanics Conference, Austin, Texas, May 2000.
5. Hallquist, J. O., LS-DYNA User's Manual, Livermore Software Technology Corporation.