Finite Element Simulation of Guardrail Impact Using DYNA3D

A comparison of the simulation results of software using the finite element method with data obtained by performing tension tests indicates that the model can anticipate the behavior of guardrails accurately.

ALA TABIEI

In the past decades the design of highway hardware under impact loads was performed experimentally through an iterative process — consisting of design, build, test, redesign and retest — until the product met certain criteria. In the last decade many computer programs based on the finite element method have been developed for impact problems. One of the most comprehensive and successful programs is DYNA3D.[1] Recent advances in computer technology and the availability of inexpensive and efficient computational power have made it possible to tackle many current engineering problems numerically, thus permitting the refinement of railing designs by computer in preparation for the final physical crash test.

The numerical implementation and application of constitutive models using the finite element method can be used to analyze real structures and compare the results to physical tests. Using finite element analysis to simulate an impact phenomenon requires proper constitutive models for each material considered. The necessary parameters for the material model were obtained by performing tension tests of the guardrail material.

DYNA3D is used to simulate the full-scale impact tests of guardrails conducted at the Federal Outdoor Impact Laboratory (FOIL) in MacLean, Virginia. Figures 1 and 2 show the test set-up. This study presents the results of impact tests conducted on steel guardrails and the corresponding computer simulation. A test fixture was built at FOIL for center impact of poles and wooden posts (see Figure 1). This fixture was redesigned and modified for studying guardrail impacts.

A finite element model of the test fixture and the pendulum was developed. The model was used to simulate the impact of 850 kg (1900 lb) mass (pendulum) into a guardrail section. The weight of the pendulum was selected (as noted above) to reflect National Cooperative Highway Research Program (NCHRP) requirements for small vehicle impact.

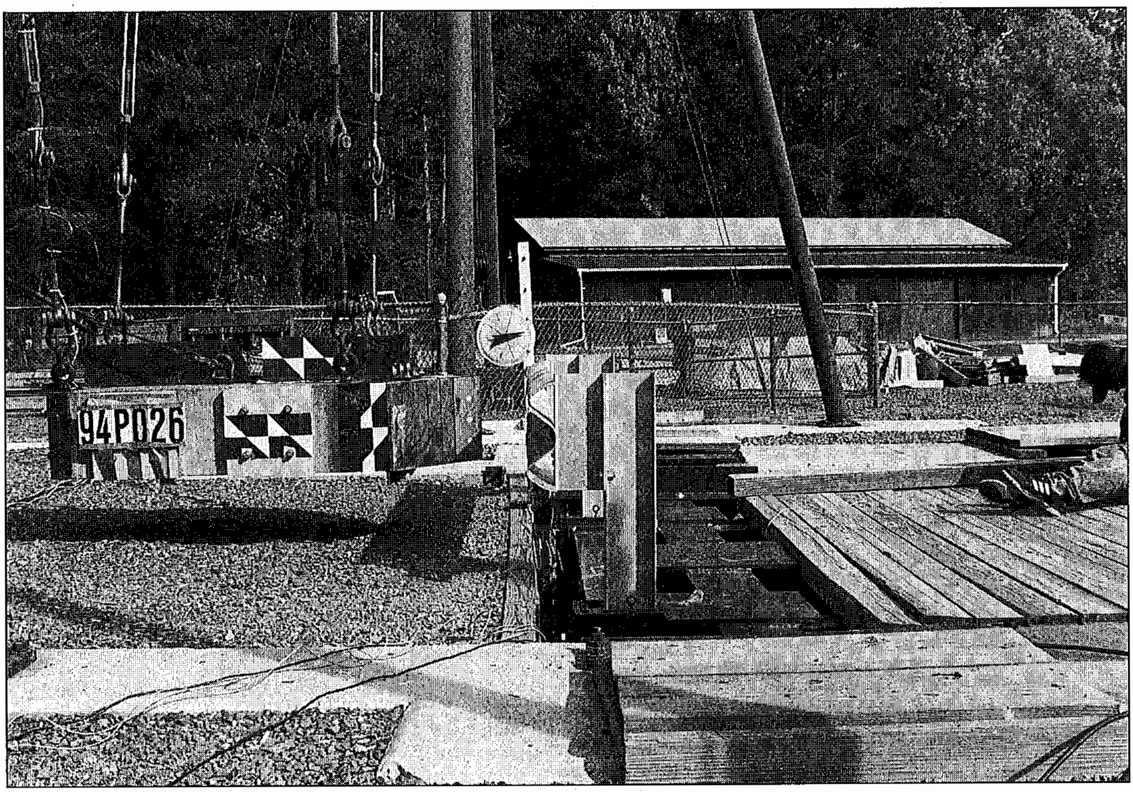

FIGURE 1. FOIL pendulum test fixture.

The objectives of this study can be summarized:

- Determine the feasibility of simulating full-scale impact tests of guardrails made of isotropic materials; and,
- Identify the critical parameters governing a successful simulation of test fixture pendulum impact.

A numerical simulation was successfully performed and the finite element model captured the impact behavior of the guardrails tested.

Finite Element Model

The finite element model for DYNA3D was developed using the preprocessor TruGrid as shown in Figure 3. The pendulum-fixture model consisted of:

- 6,217 nodes;
- 200 beam (truss) elements;
- 3,615 shell elements; and,
- 1,082 solid elements.

Table 1 (on page 42) lists the pendulum-fixture model components, element type and material type used for each part.

The finite element model consisted of a single post-supported guardrail section mounted on a blockout. The guardrail post assembly was housed in a box beam that was designed to hold steel as well as composite posts. Composite posts were larger in dimension to hold the steel posts, and spacers were necessary around the posts. The spacers provided full support to the posts inside the box beam.

The pendulum consisted of three components. The nose of the pendulum was made of wood and the body was made of concrete. Both were modeled by solid elements. The third component was the cable system that holds the pendulum that was modeled by beam (truss) elements. The truss elements that were used could resist tension only with no resistance to axial compression. Gravity was applied to the entire structure.

Two types of contact surfaces were used in the finite element model. The two surfaces con-

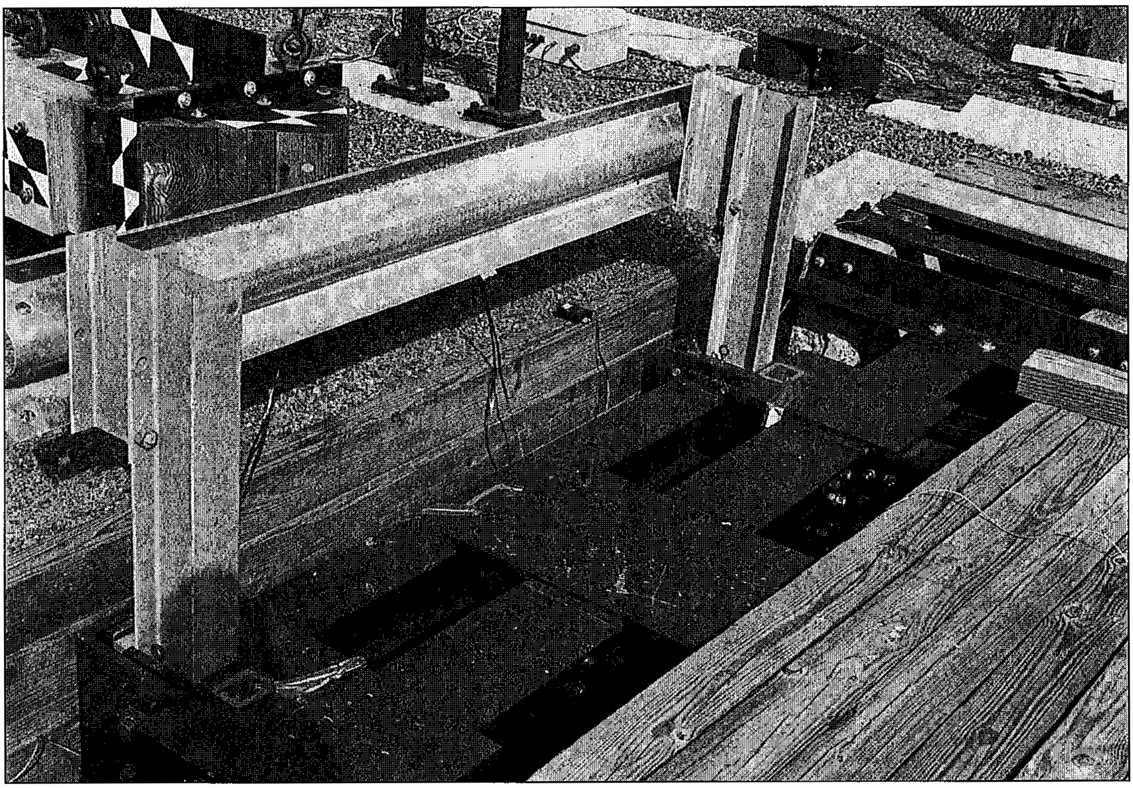

FIGURE 2. Impact test in progress at a velocity of 35 kilometers per hour.

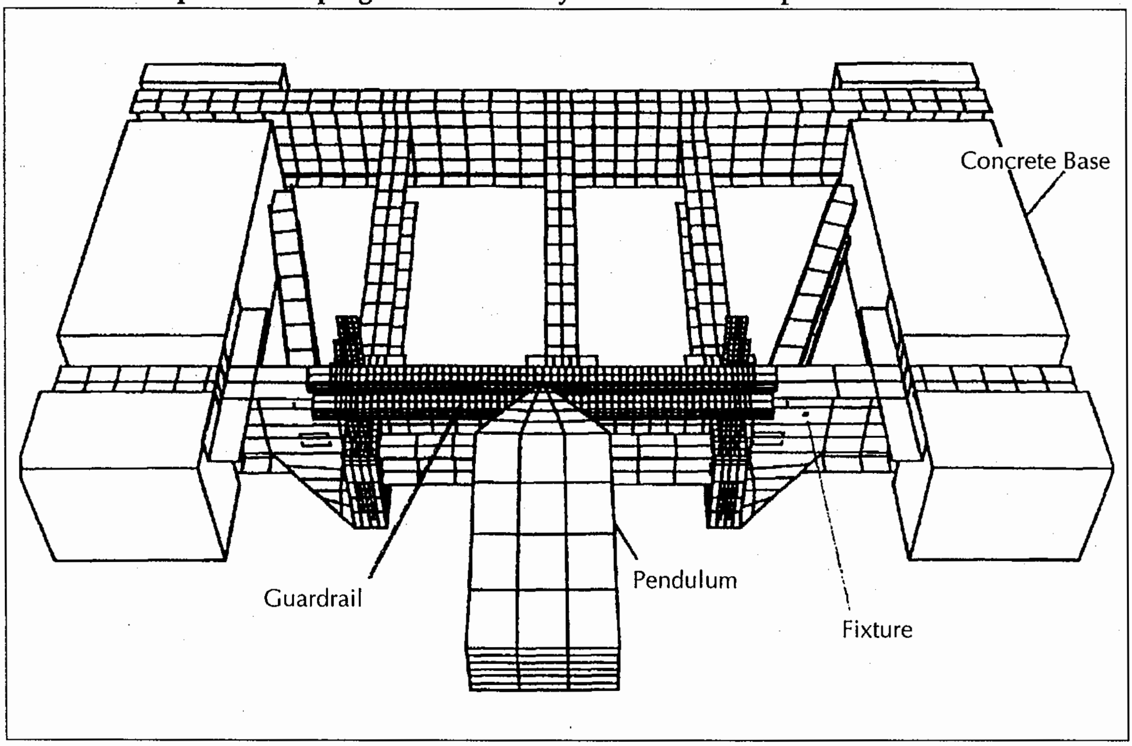

FIGURE 3. Finite element model of the pendulum test fixture, with its concrete base.

TABLE 1.
Components of the Finite Element Model

Component	Element Type	Material Type
Front & Back Large I-Beam	Shell	Elastic-Plastic
Longitudinal Braces	Shell	Elastic-Plastic
Bottom Gussets	Shell	Elastic-Plastic
Vertical Braces	Shell	Elastic-Plastic
Post Holders (Box-Beam)	Shell	Elastic-Plastic
Top Plates	Shell	Elastic-Plastic
Cross Beam	Shell	Elastic-Plastic
Left and Right Cross Beams	Shell	Elastic-Plastic
Guard Rail	Shell	Rate Dependent Elastic-Plastic
Post & Blockout	Shell	Elastic-Plastic
Wooden Spacers	Solid	Elastic-Plastic
Front Adjusting Plates	Solid	Elastic-Plastic
Side Adjusting Plates	Solid	Elastic-Plastic
Side Fixed Plates	Solid	Elastic-Plastic
Pendulum Head	Solid	Elastic-Plastic
Pendulum	Solid	Elastic-Plastic
Cables	Beam	Elastic

sisted of a tied contact surface and a contact surface with friction. A set of assumptions were made in the development of the finite element model for the pendulum fixture:

- Parts are joined by merging adjacent nodes.
- Bolted joints are modeled by merging several nodes of the joined parts.
- The cables that hold the pendulum are modeled by truss elements.
- Tied contact surfaces are used in merging parts with incompatible meshes.
- Parts connected to the concrete base (ground) are assumed to be fully constrained.

All material models used were elastic-plastic (Type 3) except the guardrail and the ca-

TABLE 2.
Mechanical Properties of Materials Considered

Material	E (N/m^2)	v	Et (N/m^2)	SIGy (N/m^2)	EPS	Type
Wood	11.5 E9	0.2	11.5 E6	50.0 E6		3
Concrete	24.0 E9	0.15	24.0 E6	10.0 E6		3
Steel (EP)	200.0 E9	0.33	200.0 E6	260.0 E6		3
Steel (REP)	200.0 E9	0.33		(345-415) E6	0.0-0.66	24

Notes:
EP = Elastic Plastic
REP = Rate Dependent Elastic Plastic
EPS = Effective Plastic Strain
SIGy = Yield Stress
E = Elastic Modulus
Et = Tangent Modulus

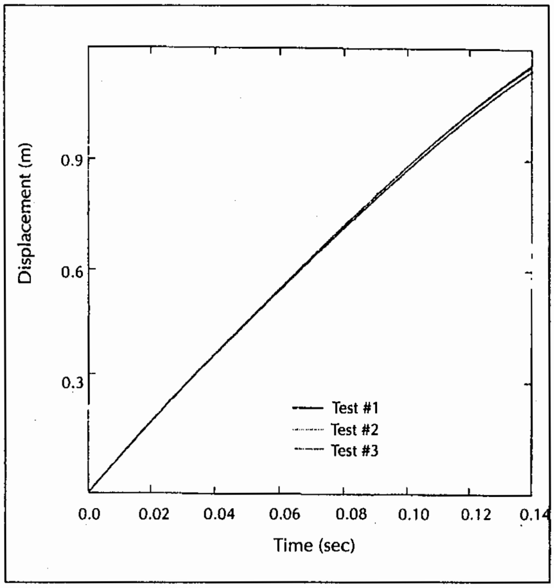

FIGURE 4. Pendulum displacement of three tests.

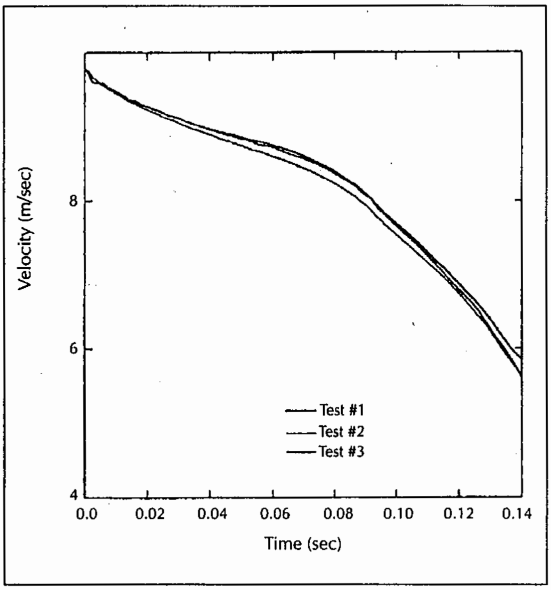

FIGURE 5. Pendulum velocity of three tests.

bles. The mechanical properties were obtained from published literature.[2] The material model for the guardrail was a rate-dependent elastic-plastic material model (Type 24) and the corresponding mechanical properties were obtained by testing four specimens.[3] The material model for the cable was an elastic material model (Type 1). Table 2 lists the mechanical properties for the material considered. Figure 3 shows the finite element model of the pendulum-fixture.

Tests

Three tests were conducted on a single rail section at FOIL. The pendulum was raised to a height of 4.82 meters, which yielded an impact velocity of 35 km/hr (21 mph). This velocity was selected to represent the velocity of a vehicle traveling at 100 km/hr (60 mph) and impacting a rail at a 20° angle. The pendulum was then released, allowing it to accelerate and impact the rail section at the midspan. Two accelerometers were positioned at the center back of the pendulum. Accelerometer data were collected a few milliseconds before impact and data collection continued until the pendulum comes to rest. A speed trap instrument was positioned just before impact to capture the speed of the pendulum at the moment of impact for verification.

As the pendulum hit the rail section and moved forward, it raised up due to the constraints of the cables. At 140 milliseconds, the lower edge of front surface of the pendulum raised to the center line of the rail section. At approximately 200 milliseconds, the pendulum climbed the rail section and was no longer in contact with the rail. In all tests the upper edge of the rail sections tore. However, the bolts that mounted the rail section to the blockout did not fail. Because of the upward motion of the pendulum, the loading was no longer symmetric on the rail. Therefore, only 140 milliseconds of the event were considered to be appropriate for the numerical simulation. The accelerometer output was filtered at 300 Hz. The filtered data were then imported to a spreadsheet.

All numerical integrations were performed to obtain the corresponding velocity and displacements. Figures 4 and 5 show the displacement and the velocity, repectively, as functions of time of the three tests of the guardrail sections.

Simulation

The DYNA3D finite element model was run on a personal computing workstation. Four nodes on the rear of the pendulum were used to collect kinematics variables (displacement, velocity and acceleration). The postprocessor TAURUS was used to obtain the above variables. The time interval for the output of these variables was the same as for the test data aquisition. The locations of these nodes in the finite element model were identical to the positions of the accelerometers at the back of the pendulum. The kinematics results were filtered, at 300 Hz, using the same filter used for the raw data from the impact tests.

Figures 6 and 7 show plots of the impact event obtained from the postprocessor. These plots show the progressive impact events. The deformed shape of the rail in the finite element model simulation was identical to that of impact tests. The finite element model indicated that there would be a failure of a few elements at the center upper edge of the rail. The failure location in the model was the same as observed in the tests. The average displacement of the three impact tests is compared to the DYNA3D simulation in Figure 8 (on page 47). The average velocity of the three tests and the velocity obtained from the simulation is shown in Figure 9 (on page 47). In general, the velocities and displacements obtained from the simulation were identical to the ones obtained from impact tests, as in this case. A good indication of a successful simulation are the acceleration curves. Figures 10, 11 and 12 (on pages 47 and 48) show the acceleration obtained from both the tests and the simulations for Tests #1, #2 and #3, respectively.

Model Validation

The objective of model validation is to reproduce (numerically) the measured quantities or observed behavior of some realistic event. For the event to be validated, a set of guidelines are used that specifies the limits for each measurable quantity to satisfy validation. The calculated time-dependent parameters are compared using the same sampling rate of the measured parameters. The validation procedure used is qualitative and quantitative in nature.

In this case, the qualitative validation was obtained by comparing the deformation of the components from a full-scale test and a simulation. They were in good agreement. The quantitative validation was obtained by comparing acceleration data, which also were in good agreement (see Figures 10-12). Therefore, the finite element simulation of the conducted test could be used as a substitute of full-scale impact for further studies. It is economically impossible to perform full-scale field testing on a wide range of parameters. Therefore, impact simulation utilizing nonlinear finite element analysis can be used as an effective tool in designing and evaluating guardrail systems.

Discussion & Conclusion

One of the objectives of this investigation was to determine the feasibility of using numerical methods to predict the behavior of real structures under impact loads. The finite element numerical simulation was successfully performed. The finite element model captured the impact characteristics of the guardrails used in this study. The deformed shapes of the guardrails in the simulation and the tests were identical. The elastic plastic rate-dependent material model is appropriate for predicting the impact behavior of such structural systems. As a result, it can be concluded that numerical simulation can be used as a supplementary tool for the evaluation and design of highway structures that meet the safety requirements of NCHRP Reports 230 and 350.[4]

The first peak in the acceleration curves is an indication of the inertial load and of the stiffness of the guardrail. The variation in the magnitude of the first peak is not clearly understood. This variation is believed to be due to material imperfection and the friction between the guardrail and the blockouts since each side of a guardrail section is connected to a blockout with a bolt that may slide in a slot cut in the rail. Once a guardrail bends, a significant reduction in the stiffness of the rail occurs. As the pendulum penetrates the rail further, the posts and the blockouts start to deform. The deformation is a combination of inward bending and twisting.

In a complete guardrail system (where there are an infinite number of guardrail sections), the stiffness is mainly governed by the tension in the rail section. From the impact simulation of the pendulum-fixture with a single rail section, it was observed that the posts twist significantly. A guardrail section connected to many sections (from the left and right) upon impact

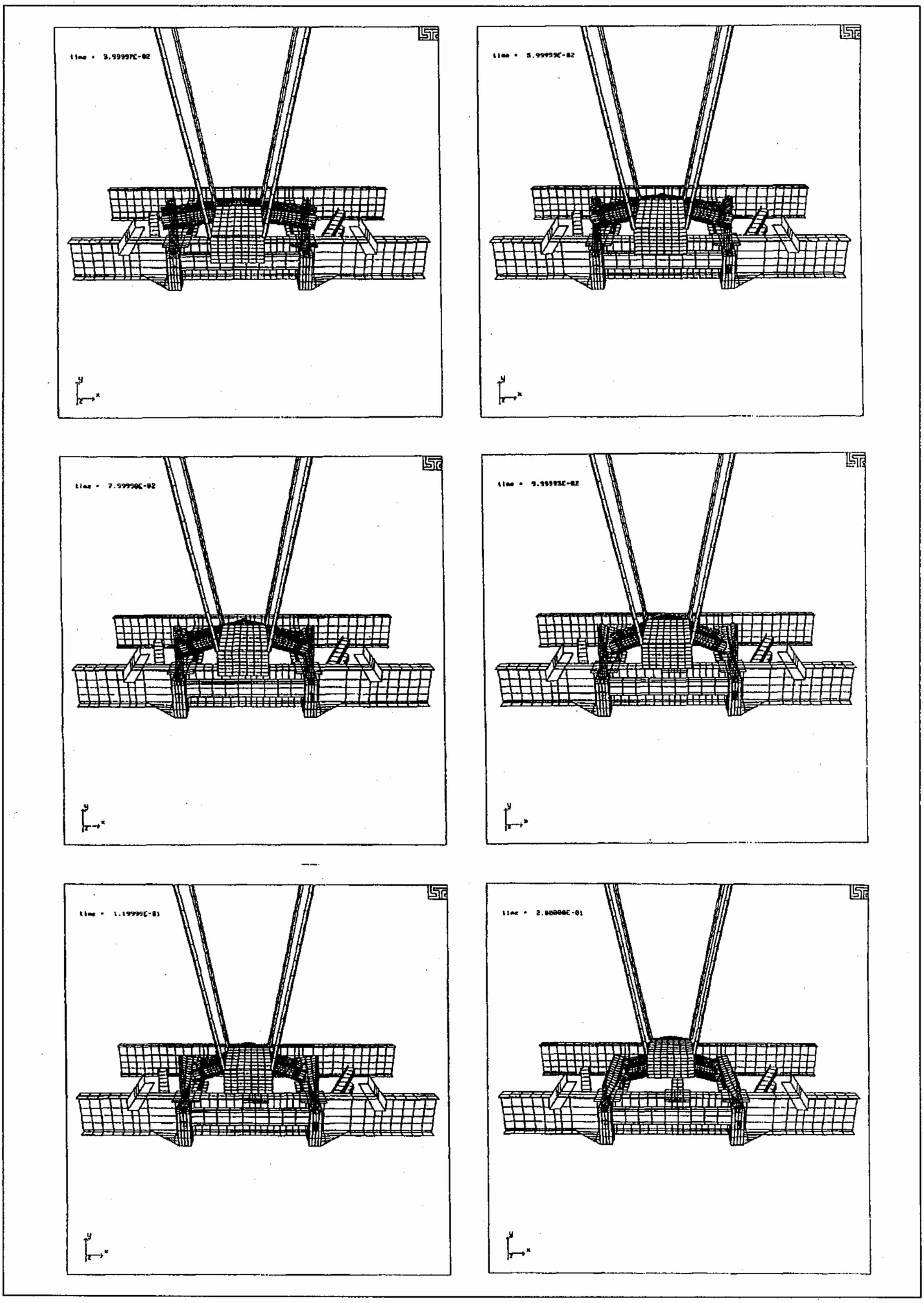

FIGURE 6. A front view of the progressive impact simulation.

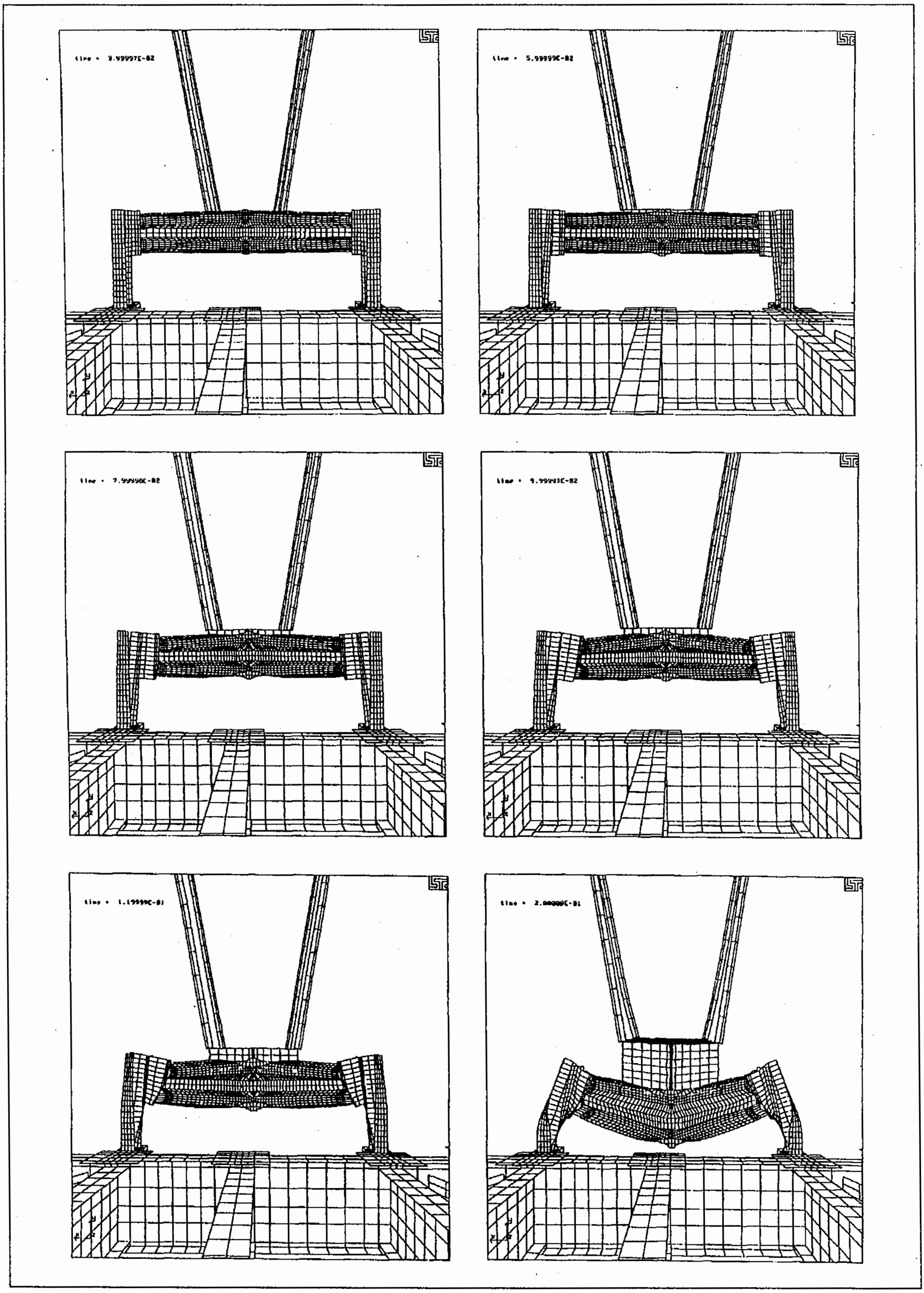

FIGURE 7. A back view of the progressive impact simulation.

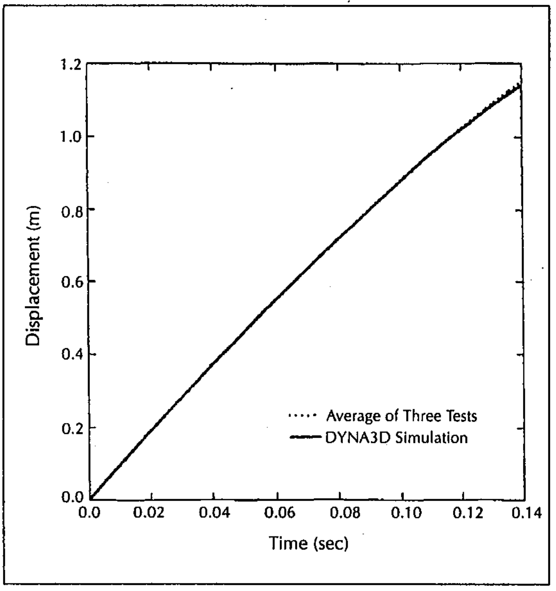

FIGURE 8. Average displacement of the three tests and the simulation predictions.

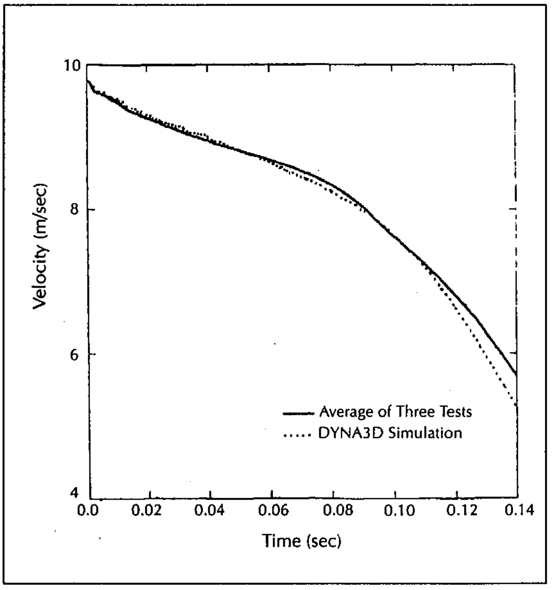

FIGURE 9. Average velocity of the three tests and the simulation predictions.

will cause less twist in the posts. Most of the kinetic energy in the case of infinite sections is absorbed by axial tension in the guardrail sections since most of the deformations are linear elastic.

Full-scale tests must be conducted very close to the actual service conditions of the guardrails. Therefore, it is essential that multiple sections be tested and numerically simulated in the future to reduce any unrealistic twist and deformation in the posts. Having done that, a parametric study can be conducted numerically to optimize for the design parameters.

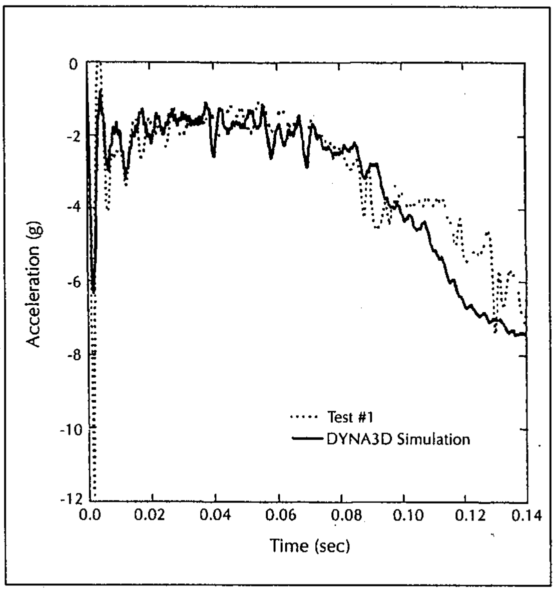

FIGURE 10. Pendulum acceleration of Test #1 and the simulation prediction.

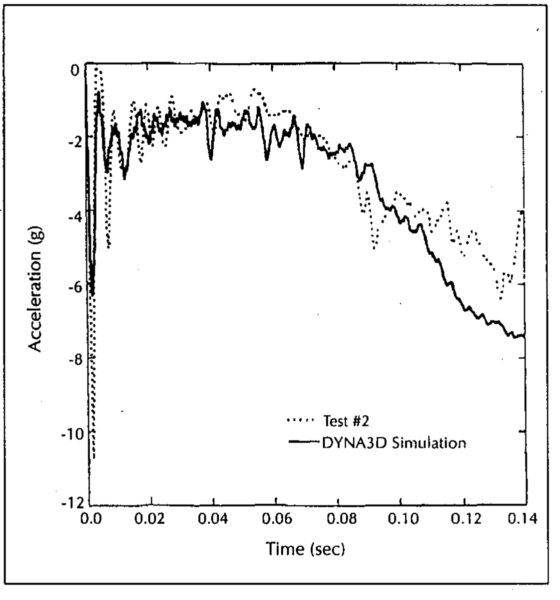

FIGURE 11. Pendulum acceleration of Test #2 and the simulation prediction.

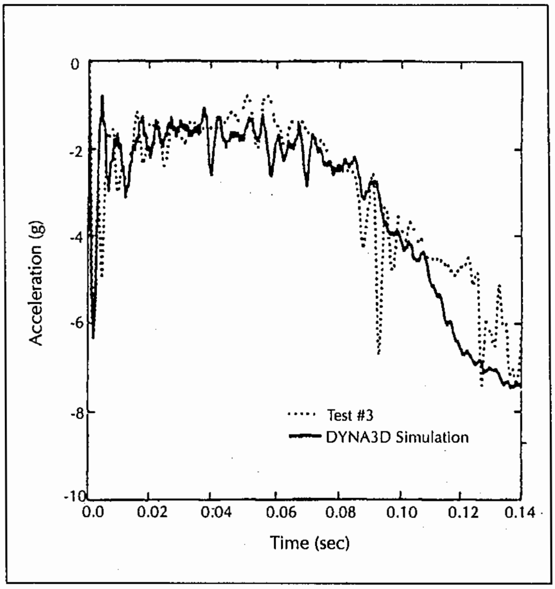

FIGURE 12. Pendulum acceleration of Test #3 and the simulation prediction.

NOTES — *The experiments described herein were actually used to verify DYNA3D's ability to model and predict the behavior of a guardrail's performance through the simulation of controlled physical tests. Therefore, the physical tests described are not crash tests in accordance with NCHRP 230 and 350. This article was based on work supported by the Federal Highway Adminstration (FHWA) under a graduate research fellowship grant. Any opinions, findings and conclusions or recommendations expressed in this publication are those of the author and do not necessarily reflect the views of the FHWA.*

ALA TABIEI *is an Assistant Professor in the Department of Aerospace Engineering and Engineering Mechanics at the University of Cincinnati. He is the director of the Center of Excellence in DYNA3D FE Analysis for Crashworthiness at the University of Cincinnati. Dr. Tabiei's areas of expertise are in finite element simulation and computational mechanics. He teaches four courses in the area of linear and nonlinear static and impact dynamics finite element techniques.*

REFERENCES

1. Hallquist, J., & Whirley, R., *DYNA3D User's Manual*, University of California, Lawrence Livermore National Laboratory, Report UCID 19592.

2. AASHTO, *Mechanical Testing of Steel Products*, T-244, 1990.

3. Ray, M., "DYNA3D Material Properties Based on Laboratory Tests," Simulation Technical Meeting, FHWA, August 1994.

4. Ross, Jr., H.E., Sicking, D.L., Zimmer, R.A., & Michie, J.D., *Recommended Procedures for the Safety Performance Evaluation of Highway Features*, NCHRP Report 350, Transportation Research Board, Washington, D.C., 1993.