2012

FEM Simulation for INDOT Temporary Concrete Anchored Barrier

Efe G. Kurt
ekurt@purdue.edu

Amit H. Varma
Purdue University, ahvarma@purdue.edu

Sangdo Hong
Indiana Department of Transportation, shong@indot.in.gov

JOINT TRANSPORTATION
RESEARCH PROGRAM

INDIANA DEPARTMENT OF TRANSPORTATION
AND PURDUE UNIVERSITY

FEM SIMULATION FOR INDOT TEMPORARY CONCRETE ANCHORED BARRIER

Efe G. Kurt
Graduate Research Assistant
School of Civil Engineering
Purdue University

Amit H. Varma
Associate Professor of Civil Engineering
School of Civil Engineering
Purdue University
Corresponding Author

Sangdo Hong
Research Engineer
Office of Research and Development
Indiana Department of Transportation
Corresponding Author

SPR-3406
Report Number: FHWA/IN/JTRP-2012/21
DOI: 10.5703/1288284314982

RECOMMENDED CITATION

Kurt, E. G., A. H. Varma, and S. Hong. *FEM Simulation for INDOT Temporary Concrete Anchored Barrier*. Publication FHWA/IN/JTRP-2012/21. Joint Transportation Research Program, Indiana Department of Transportation and Purdue University, West Lafayette, Indiana, 2012. doi: 10.5703/1288284314982.

CORRESPONDING AUTHORS

Professor Amit H. Varma
School of Civil Engineering
Purdue University
(765) 496-3419
ahvarma@purdue.edu

Dr. Sangdo Hong
Research Engineer
Office of Research and Development
Indiana Department of Transportation
(765) 463-1521
shong@indot.in.gov

JOINT TRANSPORTATION RESEARCH PROGRAM

The Joint Transportation Research Program serves as a vehicle for INDOT collaboration with higher education institutions and industry in Indiana to facilitate innovation that results in continuous improvement in the planning, design, construction, operation, management and economic efficiency of the Indiana transportation infrastructure.
https://engineering.purdue.edu/JTRP/index_html

Published reports of the Joint Transportation Research Program are available at: http://docs.lib.purdue.edu/jtrp/

NOTICE

TECHNICAL REPORT STANDARD TITLE PAGE

1. Report No.	2. Government Accession No.	3. Recipient's Catalog No.
FHWA/IN/JTRP-2012/21		

4. Title and Subtitle	5. Report Date
FEM Simulation for INDOT Temporary Concrete Anchored Barrier	September 2012
	6. Performing Organization Code
7. Author(s)	**8. Performing Organization Report No.**
Efe G. Kurt, Amit H. Varma, Sangdo Hong	FHWA/IN/JTRP-2012/21
9. Performing Organization Name and Address	**10. Work Unit No.**
Joint Transportation Research Program Purdue University 550 Stadium Mall Drive West Lafayette, IN 47907-2051	
	11. Contract or Grant No. SPR-3406
12. Sponsoring Agency Name and Address	**13. Type of Report and Period Covered**
Indiana Department of Transportation State Office Building 100 North Senate Avenue Indianapolis, IN 46204	Final Report
	14. Sponsoring Agency Code

15. Supplementary Notes

Prepared in cooperation with the Indiana Department of Transportation and Federal Highway Administration.

16. Abstract

Portable Concrete Barriers (PCBs) are used to redirect errant vehicles to keep them passing to opposing lanes and to ensure safety of the people and any objects behind the barriers. In the state of Indiana, increments to the PCBs, such as L-Shape steel plates, have been applied to enhance the safety performance of these barriers. In this study, Finite Element (FE) analyses are performed to evaluate the safety performance of PCBs with and without the increments and get thorough information about the increments applied. A full-scale crash test (INDOT, 2001) was executed for an impact to the PCBs with a 2000 kg pickup truck at an angle of 25 degrees and an initial velocity of around 100 km/hr in accordance with National Cooperative Highway Research Program (NCHRP) Report 350 guidelines for Test Level 3 safety performance. Aforementioned full-scale crash test data are used to validate the FE model constructed. Roadside Safety Verification and Validation Program (RSVVP) was used to compare the crash test and FE model results quantitatively. Validating the results of the initial FE Model leaded the way in confidence to implement the increments in the following FE Models.

17. Key Words	18. Distribution Statement
concrete barriers, vehicle impact analysis, LS-Dyna	No restrictions. This document is available to the public through the National Technical Information Service, Springfield, VA 22161.

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of Pages	22. Price
Unclassified	Unclassified	68	

Form DOT F 1700.7 (8-69)

EXECUTIVE SUMMARY

FEM SIMULATION FOR INDOT TEMPORARY CONCRETE ANCHORED BARRIER

Introduction

Longitudinal portable concrete barriers (PCBs) are used to keep errant vehicles on the roadway. By doing this, workers, work areas and construction equipment at highway worksites are protected, and separation of two-way traffic is achieved. PCBs used in the crash test and finite element model (FEM) are attached to each other by pin-and-loop connections. In this report, information about the crash test conducted by the Indiana Department of Transportation (INDOT), FEM simulating the crash test, validation process of the FEM and implementation of the L-shaped steel plates to the validated FEM are presented.

Findings

- Constructed benchmark FEM captures the results of the crash test successfully. The results were verified and validated using the Roadside Safety Verification and Validation Program (RSVVP) and the criteria of Appendix E of the NCHRP 350 report.
- The maximum deflection of the barriers was around 63 inches in both the benchmark simulation and the crash test.
- Implementing L-shaped steel plates reduced the maximum displacement of the barriers to 5.5 inches. No overturning of the barriers was observed. After the impact, the concrete barriers returned to close to their original positions.
- No significant damage in the concrete pavement was observed around the most critical steel plate. Minor damage around the critical plates was observed due to the anchorage–concrete pavement interaction.
- The steel plate located close to the impact bent over but no failure of the plates and anchorages was observed.
- The exit angle of the vehicle also decreased in the model with increment when compared to the benchmark model.
- The benchmark analysis can be used for further analysis for different types of increments.

CONTENTS

LIST OF FIGURES

1. INTRODUCTION

Longitudinal portable concrete barriers (PCBs) are used to keep errant vehicles in the roadway. By doing this, workers, work areas and construction equipment at highway work sites are protected, and separation of two-way traffic is achieved (*1*). PCBs used in the crash test and in the finite element model (FEM) are attached to each other by pin-and-loop connections. In the following sections of this report, information about the crash test conducted by the Indiana Department of Transportation (INDOT), FEM simulating the crash test, validation process of the FEM and implementation of the L-shaped steel plates to the validated FEM are presented.

2. FULL-SCALE CRASH TEST

The temporary precast concrete barriers with pin and loop connections were tested for Indiana Department of Transportation. The test was conducted, and data collected relative to relevant portions of the NCHRP Report 350 test level 3-11.

The test article for the tests is 32-inch height temporary precast concrete barriers with pin and loop connection. Each barrier has a length of 10 feet. The loop is located 9 inches measured from the top and 6 inches measured from the bottom of the PCB on one side and vice versa on the other end.

A 2000P impacting the critical impact point along the length of need section at a nominal speed of 100 km/hr and an angle of 25 degrees is needed for the NCHRP Report 350 test designation 3-11. The test is intended to evaluate the length of need section in containing and redirecting the 2000P vehicle.

In the actual test, a 1995 Chevrolet C2500 Cheyenne pickup was used. The weight of the vehicle was 2041 kg. The vehicle was directed into the installation using the tow system, and was released to be freewheeling and unrestrained just prior to impact. The vehicle travelling at a speed of 102.9 km/hr impacted the precast concrete barriers at 23.8 degrees. Post-test barrier diagram is shown in Figure 2.1. Maximum deflection was observed as 63 inches.

Most of the damage to the vehicle was to the right front corner. The right front outer rim was separated from the center hub section of the rim. The barriers cracked during the test, but they remained intact. Pre and post-test views of the vehicle and barriers are presented in

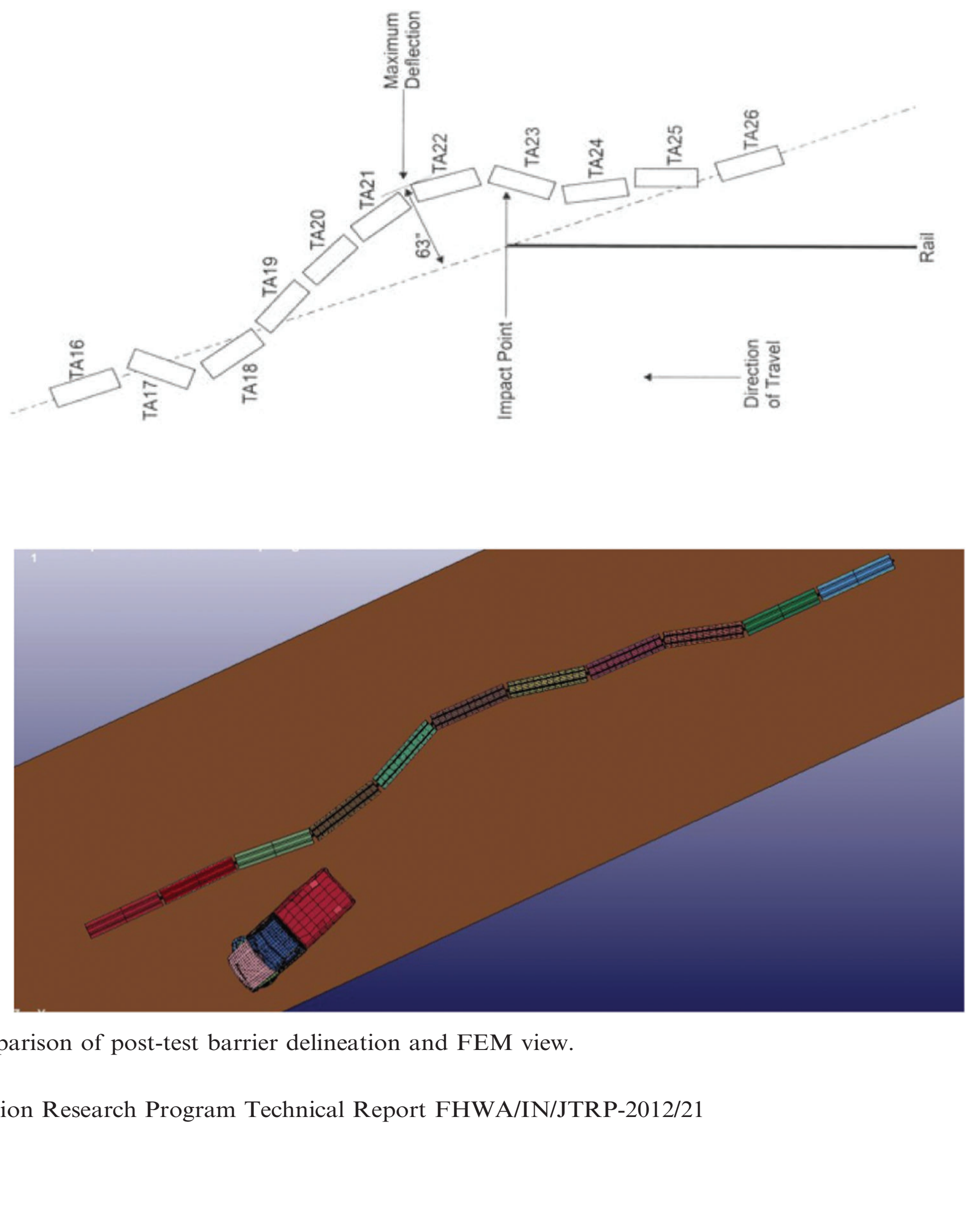

Figure 2.1 Comparison of post-test barrier delineation and FEM view.

Figure 2.2 General view of vehicle and barriers before and after test.

Figure 2.2. During the test, the vehicle was contained and redirected by the precast concrete barriers. The vehicle remained upright during and after the collision.

The vehicle remained upright during and after the collision. The exit angle at loss of contact was 8.4 degrees which was within the preferred limit of 60 percent of the impact angle.

3. FINITE ELEMENT ANALYSIS (FEA) USING LS-DYNA

3.1 General

LS-DYNA is a general-purpose finite element (FE) program capable of simulating complex real world problems. It is used by the automobile, aerospace, construction, military, manufacturing, and bioengineering industries. LS-DYNA's potential applications are numerous and can be tailored to various fields.

In this study, LS-DYNA software is used to simulate a vehicle impact with the barrier to quantify dynamic deflection characteristics, vehicle stability measures and occupant risk metrics as defined by NCHRP Report 350.

The National Crash Analysis Center (NCAC) is a successful collaborative effort among the Federal Highway Administration, the National Highway Traffic Safety Administration and George Washington University. The NCAC website provided much guidance during the construction of the FEM for this study.

FEM has the same number of barriers connected to each other as in the full-scale crash test. C2500 truck model is used to represent the impacting vehicle. The vehicle is impacted with the same location, velocity and angle. A general view of the FEM is shown in Figure 3.1.

3.2 Details

There are FEMs constructed for different purposes by NCAC. These models have been validated or verified for combinations of different types of barriers and connection details. This study is also a reflection of the models that NCAC "built" for analyses of precast concrete barriers.

The constructed model has 11 numbers of barriers as in the full-scale crash test. They are connected to each other by pin-and-loop connections. The C2500 pickup model is developed by NCAC, and it is widely used for this type of crash simulations. This model has proven itself in several simulations. Therefore, this C2500 model is also used in this study.

The concrete barriers were generated by using solid hexagonal elements. As in NCAC models, solid barriers were covered with shell barriers. This was done to gain a better contact between the truck and the barriers as shell elements are better in contact calculations compared to solid elements. Parts of the barriers in contact with the ground surface were modeled as elastic, and the rest was modeled as rigid. Modeling reinforcement bars inside the barriers were not considered as the inertia and mass of the barriers were based on the cross section and material properties. In order to have same inertia as in the experiment, barriers have the dimensions consistent with the experiment. General view of the barriers is shown in Figure 3.2.

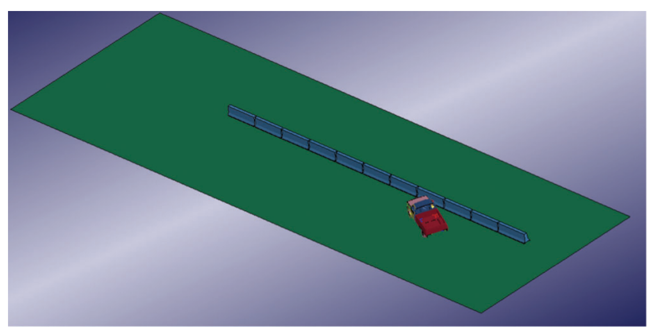

Figure 3.1 General view of FEM in LS-DYNA.

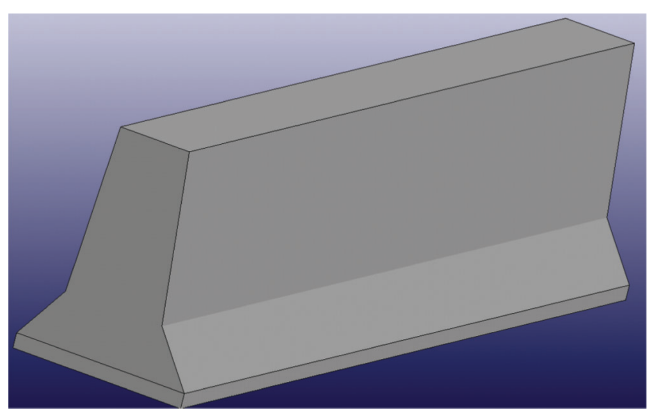

Figure 3.2 PCB model.

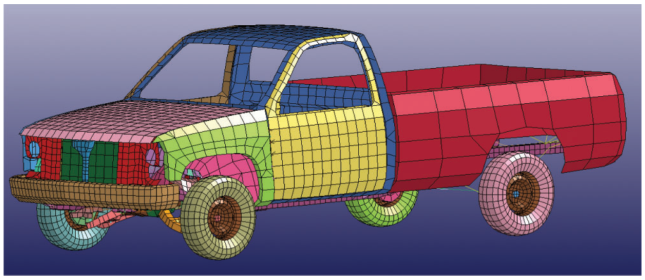

Figure 3.3 C2500 pickup model from NCAC.

Length of the barriers was 3050 mm. The widely used C2500 truck model can be seen in Figure 3.3. This model is developed by NCAC.

3.3 Results of the Benchmark Analysis

Maximum deflection during the study was also observed at the same location as in the experiment. It was measured as around 1600 mm (63 inches). Resultant displacement history of the node of interest for maximum deflection is shown in Figure 3.4.

In order to compare the simulation with the experiment visually, Figure 3.5 is constructed. General

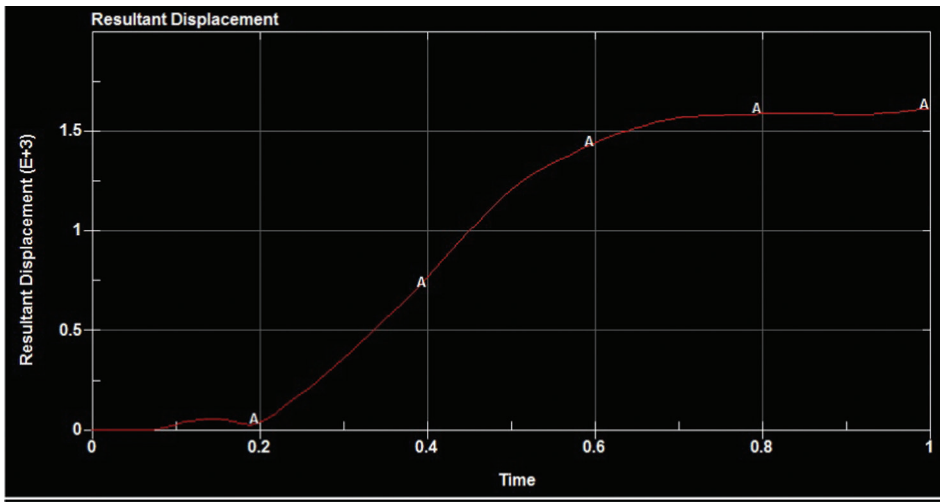

Figure 3.4 Displacement history of maximum deflection-observed node.

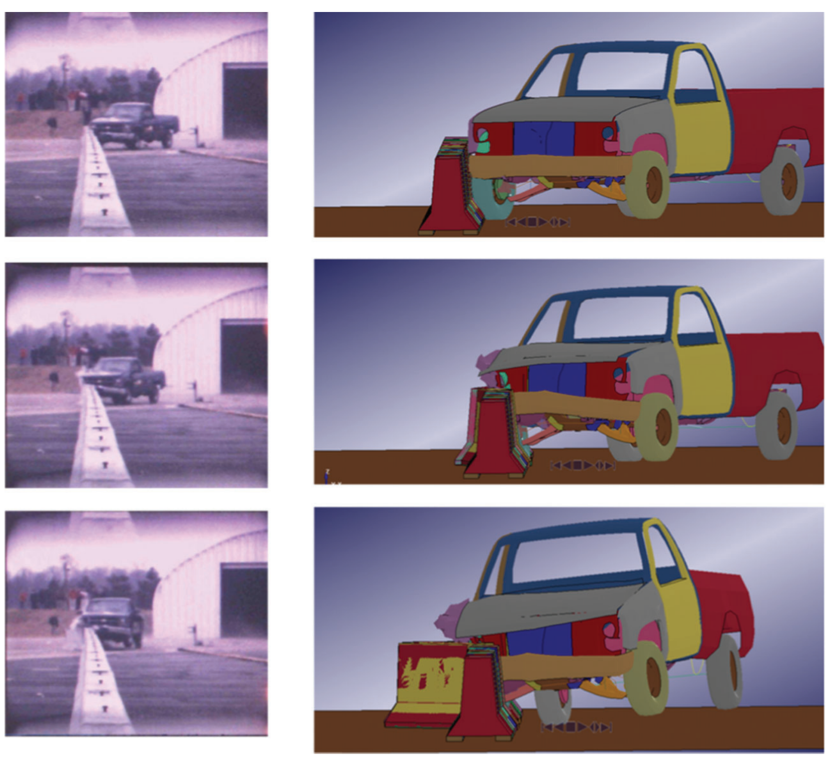

Figure 3.5 Comparison of experiment and simulation.

views of the barriers and truck are shown both during the experiment and analysis at nearly same time intervals.

4. RSVVP

The Roadside Safety Verification and Validation Program (RSVVP) (*2*) quantitatively compares the similarity between two curves, or between different pairs of curves, by computing similarity metrics. Similarity metrics are objective, quantitative statistical measures of the similarity between two curves. The similarity metrics calculated by RSVVP can be used to validate computer simulation results against experimental data, to verify the results of a simulation against the results of another simulation or analytical solution, or to determine the repeatability of a physical experiment. Although RSVVP has been specifically developed to assist in the verification and validation of roadside safety computational models, it can usually be used to provide a quantitative comparison of essentially any pair of curves.

In order to ensure the most accurate comparison between the curves, RSVVP allows the user to choose among several preprocessing tasks prior to calculating the metrics. The interactive graphical user interface of RSVVP was designed to be as intuitive as possible in order to facilitate the use of the program. Throughout each step of the program, RSVVP provides warnings to alert the user of possible mistakes in their data and to provide general guidance for making appropriate choice of the various options.

The interpretation of the results obtained using RSVVP is solely the responsibility of the user. The RSVVP program does not imply anything about the data; it only processes the data and calculates the metrics. The user must verify that the data input into the program is appropriate for comparison and that the appropriate options in RSVVP are used for their particular situation.

Available experimental data and results from the simulation are compared in RSVVP. At the first step, accelerations of x, y and z are compared in separate channels. Comparison of x and y accelerations did well. However, the results in z-direction could not pass the criteria of RSVVP. Then, a multichannel analysis is conducted and passed the required criteria of Appendix E of NCHRP 22-24. The results of Appendix E are presented in the appendix of this report.

5. L-SHAPED STEEL PLATE INCREMENT

5.1 General

General information about the L-shaped steel plate increment model is given in this part. Further discussion is in the following parts of this chapter.

After completing verification of the initial model with the available experimental data, L-shaped steel plates were implemented to the existing model. A general view of the plates is shown in Figure 5.1. Dimensions of the plate are 150 × 450 mm with a 200 mm height. Holes on the plates were also modeled. Distance between the holes was 280 mm with a diameter of 30 mm. Spacing of the plates was 3160 mm, and they were placed between the barriers on the opposite side of the traffic.

In Figure 5.2, steel anchorage bolts are shown. The length of the anchors was 660 mm. Anchors were passed through the holes on the plates and embedded to the concrete pavement and soil beneath the pavement.

The concrete pavement had a thickness of 300 mm. Finer meshing was applied near the holes on the pavement. The concrete was modeled with the material 159_CSCM_CONCRETE (*3*). A part of the concrete pavement is shown in Figure 5.3. The holes on the pavement can be seen clearly on the following figure. The plates were placed on the locations to fit perfectly over the holes on the pavement.

Figure 5.4 shows a part of the soil model. Material for soil is defined as 26_HONEYCOMB. The material is defined as it will not take any force/pressure when it is in tension as to represent the behavior of a "real" soil component. Soil is also fine-meshed around the holes in it as shown in the figure.

Interaction between steel plates, anchors, concrete pavement and soil are achieved by AUTOMATIC_SINGLE_SURFACE. After the collision, the anchors

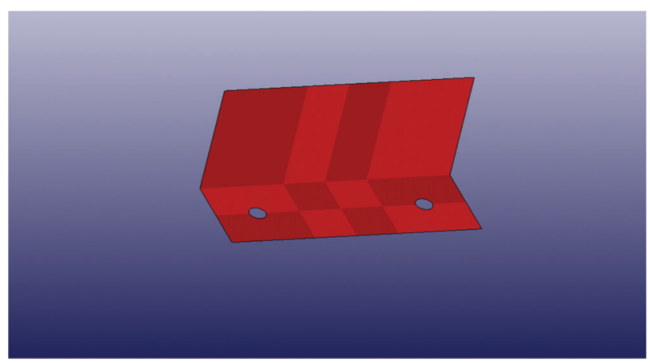

Figure 5.1 General view of steel plate.

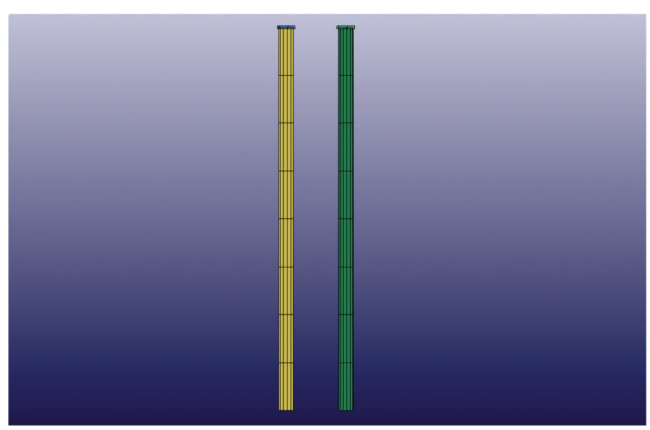

Figure 5.2 Anchor bolts used in model.

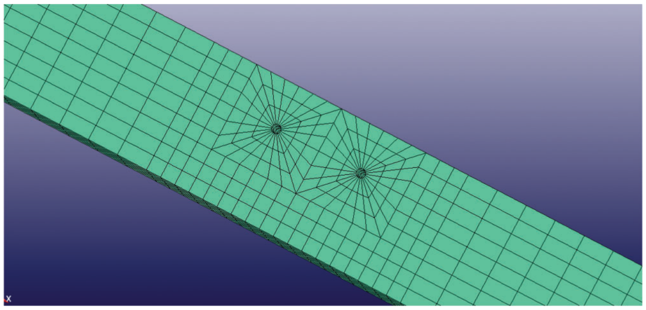

Figure 5.3 Concrete pavement used in model.

tried to get pulled out but prevented from the interaction with concrete pavement and soil. Figure 5.5 shows anchors passing through steel plate and steel plate placed over the concrete on top of soil part by means of anchors.

5.2 L-Shaped Steel Plates

Steel plates are modeled as shell elements. Dimensions of the plate are 150 × 450 mm with a 200 mm height. It has a thickness of 12.7 mm (approximately ½ inch). The elastic modulus for the plates is 200 GPA with a Poisson's ratio of 0.3. The failure strain is taken as 0.15 with a yield stress and ultimate stress of 290 and 470 MPA, respectively.

Belytschko-Tsay shell formulation is used for the shell elements of the plates. Belytschko-Lin-Tsay shell element is usually the shell element formulation of choice because of its computational efficiency. The formulation is based on a combined corotational and velocity-strain formulation. The efficiency of the element is obtained from the mathematical simplifications that result from these two kinematical assumptions. The corotational portion of the formulation avoids the complexities of nonlinear mechanics by embedding a coordinate system in the element. The choice of velocity-strain or rate-of-deformation in the formulation facilitates the constitutive evaluation since conjugate stress is the physical Cauchy stress (*4*).

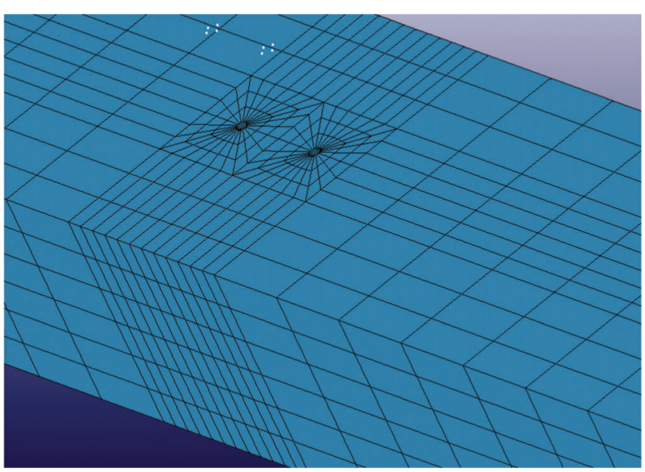

Figure 5.4 Soil used in model.

5.3 Materials

The thickness of the concrete pavement is 300 mm (approximately 12 inches). The holes on the concrete pavement are a bit larger than the anchors' diameter. So that an Auto_Single_Surface contact keyword could be defined between them. The friction coefficient between the anchors and the pavement is 0.18.

Material of the concrete pavement is assigned as CSCM_CONCRETE (Mat 159 in LS-DYNA). Concrete cylinder analyses are conducted in order to see whether the model gives the desired compression and tension strengths. In Figure 5.6, behavior of the concrete model is shown under uniaxial compression. The strength reaches to 6 ksi and starts to drop as shown in the mentioned figure. Then, modeled concrete cylinders were subjected to uniaxial tension. Figure 5.7 shows the behavior of the cylinder under pure tension. A stress of 420 psi was observed as the maximum strength which is also reliable. Figure 5.8 shows the stress–strain curve for the steel plates and anchors. Ultimate strain is assigned as 0.15 with a stress of 470 MPa.

6. RESULTS OF THE MODEL WITH INCREMENT

The same speed and location of impact were used for this analysis. In the following figure, the position of the truck, plates and the barriers are shown. The plates

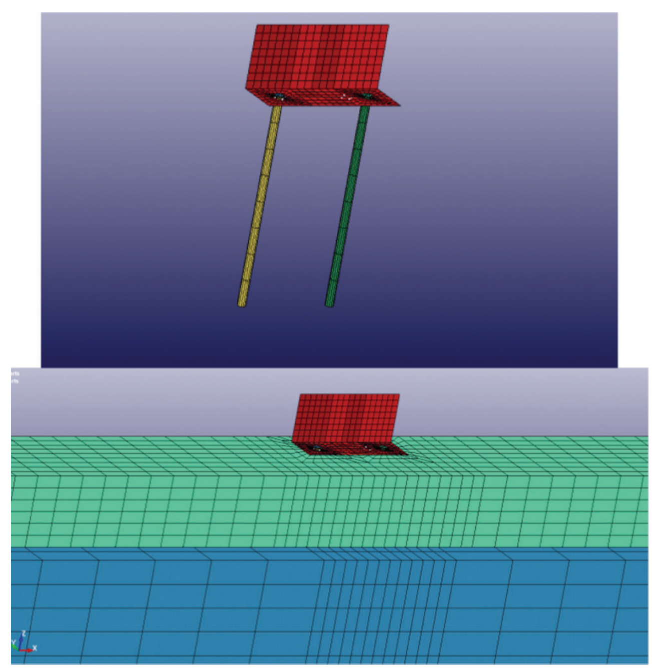

Figure 5.5 Steel plates embedded in concrete and soil by means of anchors.

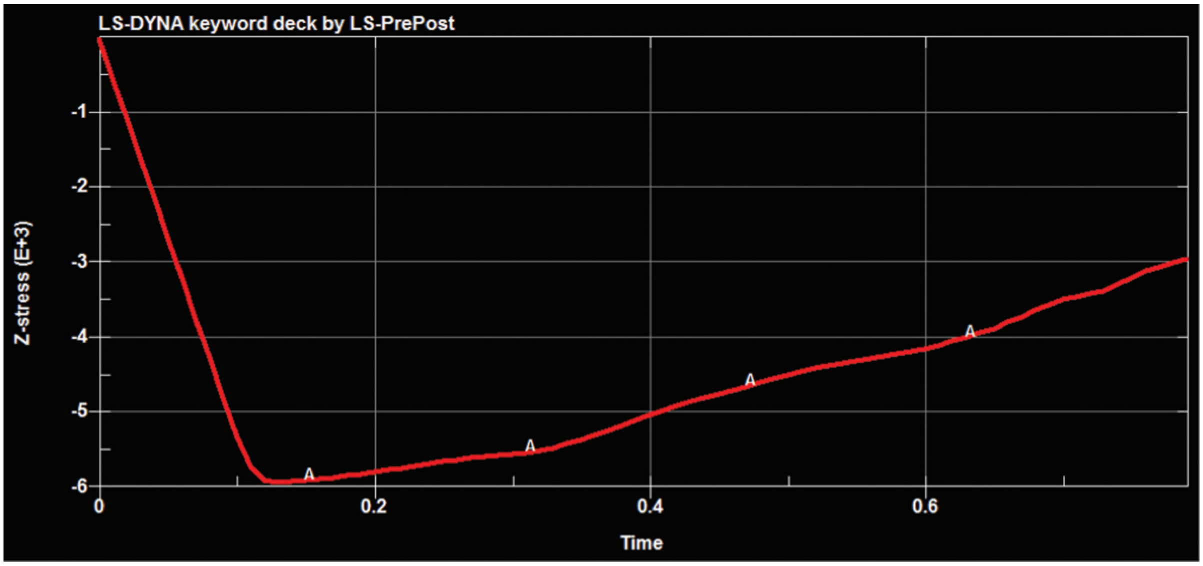

Figure 5.6 Typical behavior of concrete model in uniaxial compression stress.

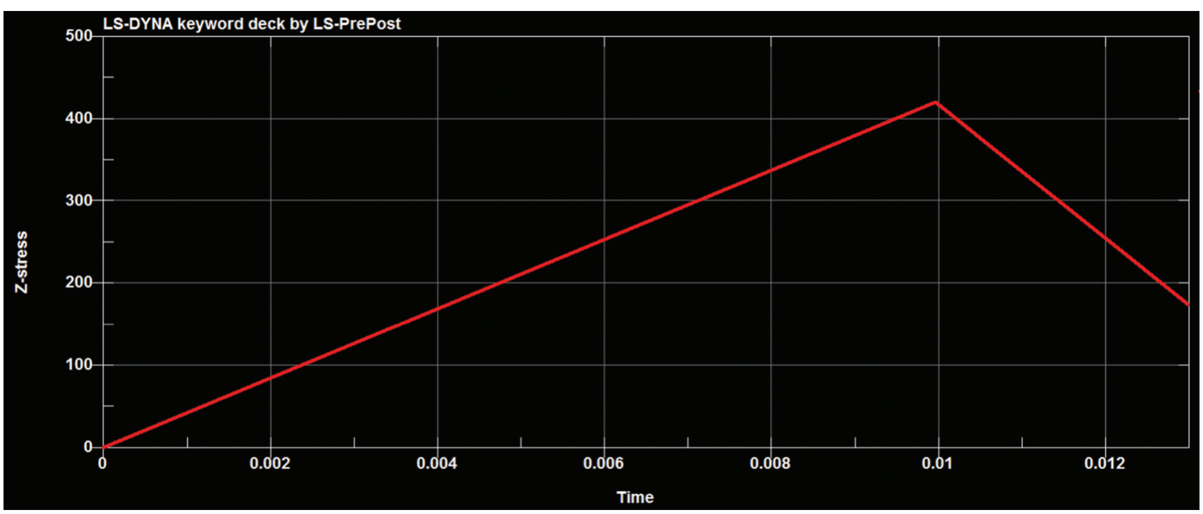

Figure 5.7 Typical behavior of concrete model in uniaxial tension stress.

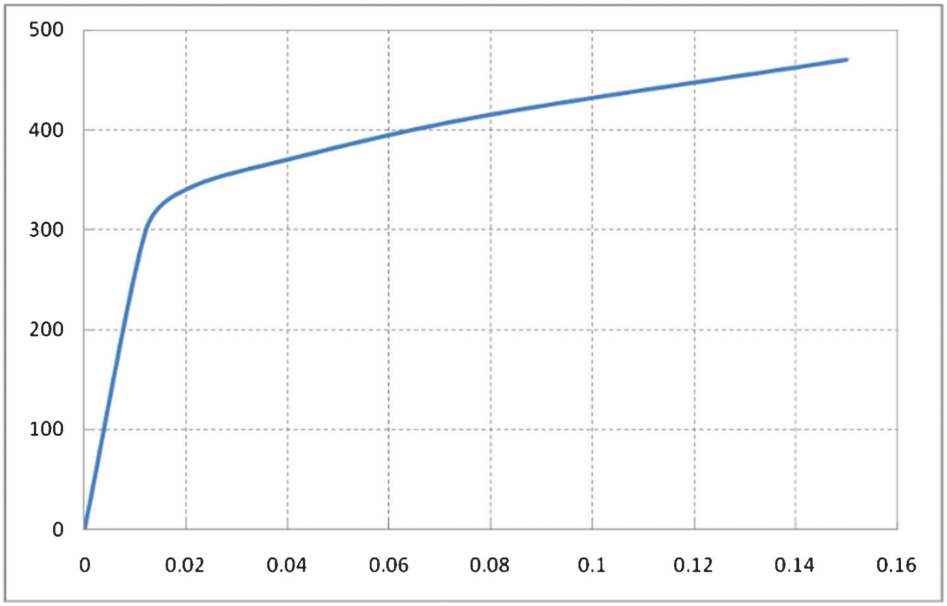

Figure 5.8 Steel strain curve for plates and anchors.

were placed opposed to the traffic. Another view of the model is shown in Figure 6.1.

Displacements of the barriers were relatively smaller than the barriers without steel plates added. The barriers tried to overturn over the plates, but the mass of the barriers and blockage from the plates prevented them to overturn (Figure 6.2) and landed almost to the same location before the impact. It is also seen that the vehicle redirected parallel to the barriers after the impact and decreased the probability of interrupting other lanes on its own traffic side.

What was observed after the impact was the plates would be flying away from the barriers if there were no anchorages interacting with concrete and soil. After the barriers relocated closer to their original position before the impact, the plates in the proximity of the impact point had permanent deformations. It was clear that the most critical deformation was observed at the plate located between the impacted barrier and its neighborhood barrier. The plate in Figure 6.3 shows the most critical plate, and it bent over as shown.

This report contains a study about the material type used for concrete. It is shown that the material model is successful in capturing the desired compression and tension strength with the specified values. Figure 6.4 shows the deformation around the most critical steel plate. There was not so much damage observed around the anchorages. There was some minor damage due to the anchorage forces around the holes in the concrete.

The displacement of the barriers decreased when compared to the situation without any improvements. It can be said that the steel plates resulted in a reduced displacement due to preventing the barriers from moving away from their original position. The most critical barrier, the one neighbor to the impacted barrier on downstream, experienced a maximum displacement of around 140 mm (approximately 5.5 inches), and it decreased after the vehicle started to get away from it. Figure 6.5 shows the displacement history of the most critical barrier.

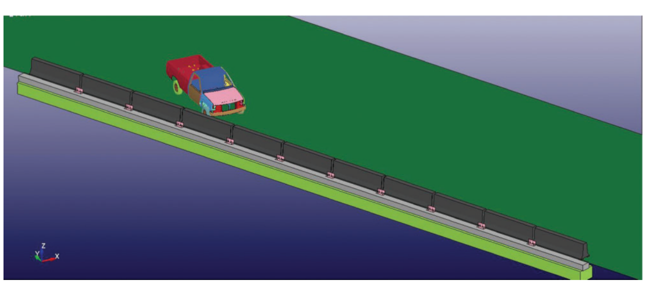

Figure 6.1 General view of model before impact.

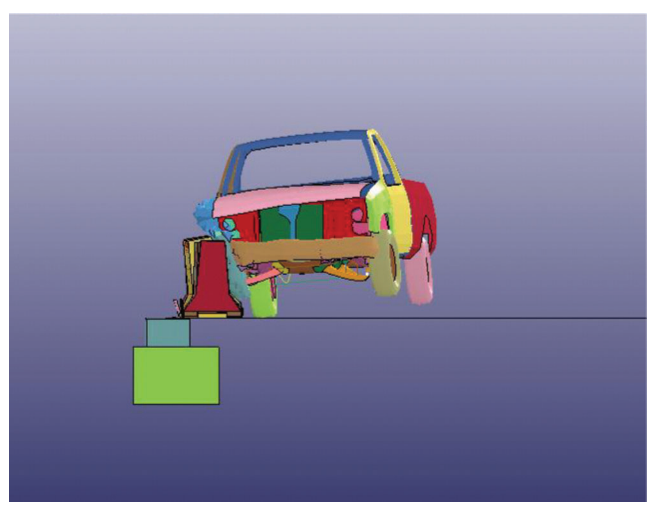

Figure 6.2 Impact of C2500 pickup to barriers with L-shaped plates.

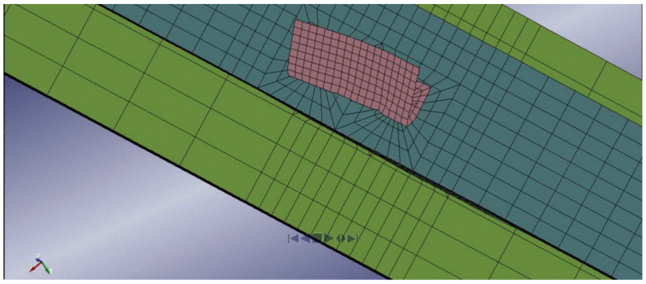

Figure 6.3 Bending of plate after impact.

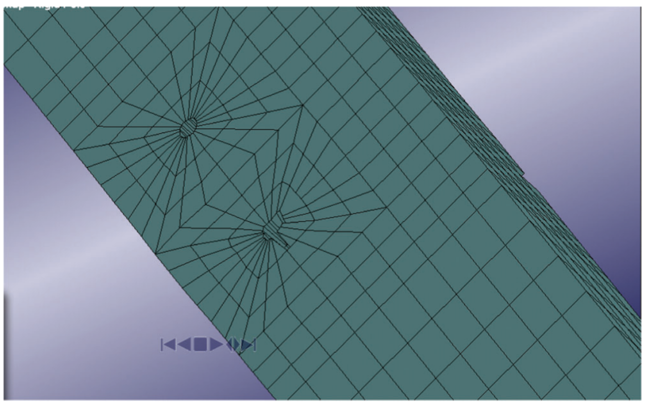

Figure 6.4 Concrete damage at location of most critical steel plate.

7. CONCLUSION

The primary objective of this study was to investigate the safety performance of the road side safety barriers. In order to achieve this goal, a benchmark analysis was conducted at the first step. This analysis was done in commercial FE analysis program LS-DYNA. The results of the analysis were compared both visually and analytically with the crash test conducted by INDOT. After verifying and validating the FEM, L-shaped steel plates were implemented to the model including the effects of concrete pavement and soil. Following conclusions were derived from this study:

- Constructed benchmark FEM captures the results of the crash test successfully. The results were verified and validated using RSVVP and criteria of Appendix E of NCHRP 350 report.
- The maximum deflection of the barriers was around 63 inches both in the benchmark simulation and the crash test.
- Implementing L-shaped steel plates reduced the maximum displacement of the barriers to 5.5 inches. No overturning of the barriers was observed. After the impact, the concrete barriers returned close to their original positions.
- No significant damage in the concrete pavement was observed around the most critical steel plate. Minor damage around the critical plates was observed due to the anchorageconcrete pavement interaction.
- Steel plate in the proximity of impact location bent over but no failure of the plates and anchorages was observed.
- The exit angle of the vehicle also decreased in the model with increment when compared to the benchmark model.
- The benchmark analysis can be used for further analysis for different type of increments.

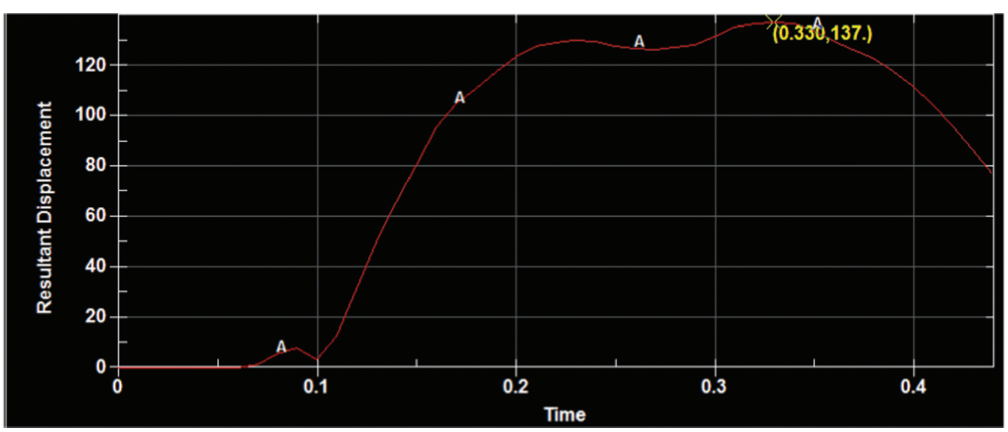

Figure 6.5 Displacement history for most critical barrier.

REFERENCES

1. Marzougui, D., M. Buyuk, and C.-D. Kan. *Safety Performance Evaluation for Combinations of Portable Concrete Barrier Elements*. National Crash Analysis Center Working Paper 2007-W-004. August 2007. Available at http://www.ncac.gwu.edu/research/pubs/NCAC-2007-W-004.pdf.
2. Ray, M. L., et al. *Guidelines for Verification and Validation of Crash Simulations Used in Roadside Safety Applications*. Report from NCHRP Project 22-24, Transportation Research Board of the National Academies, Washington, D.C., 2010.
3. *Users Manual for LS-DYNA Concrete Material Model 159.* FHWA-HRT-05-062. May 2007. http://www.fhwa.dot.gov/publications/research/infrastructure/pavements/05062/05062.pdf. Accessed May 1, 2012.
4. Hallquist, J. O. *LS-DYNA Theory Manual.* March 2006. http://www.dynasupport.com/manuals/additional/ls-dyna-theory-manual-2005-beta/at_download/file. Accessed May 1, 2012.

APPENDIX A. APPENDIX E OF NCHRP 350 REPORT

http://docs.lib.purdue.edu/cgi/viewcontent.cgi?filename=1&article=3008&context=jtrp&type=additional

APPENDIX B. DRAWINGS OF THE PLATES AND BARRIERS

http://docs.lib.purdue.edu/cgi/viewcontent.cgi?filename=2&article=3008&context=jtrp&type=additional