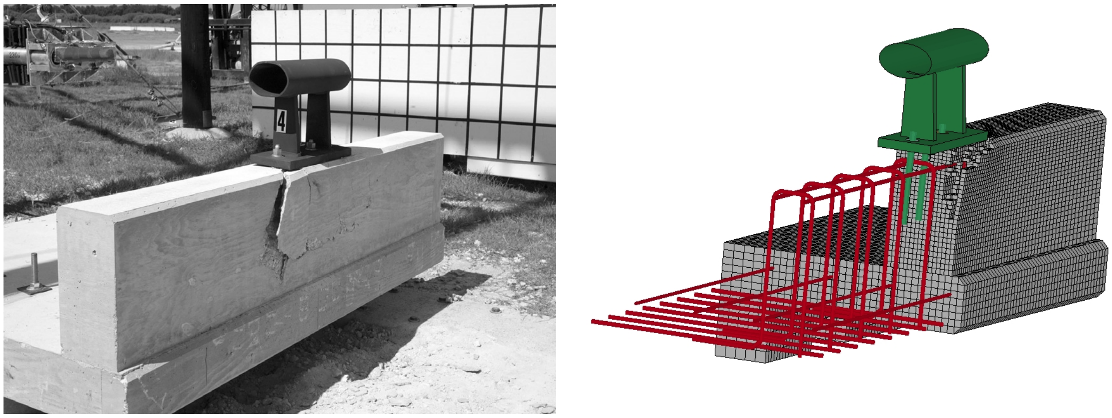
Application of New Concrete Model to Roadside Safety Barriers

Akram Abu-Odeh
Associate Research Scientist
Texas Transportation Institute
The Texas A&M University System
College Station, TEXAS 77843-3135

Abstract

*The subject of roadside safety has been seeing a healthy growth in the use of nonlinear finite element analysis in designing and analyzing roadside hardware systems. Some factors that helped researchers include the expanded capabilities of LS-DYNA commercial finite element code, the availability of several public domain vehicle models and the availability of material models explicitly built and modified for roadside safety applications. One of these new material models is *MAT_CSCM (and the short input version *MAT_CSCM_CONCRETE) incorporated in LS-DYNA version 971 as material type 159. Material model *MAT_CSCM which was developed by APTEK INC. is a continuous surface cap material model with the ability of capturing concrete material behavior using minimal input like the compressive strength of concrete and maximum aggregate size. In this paper two examples of using LS-DYNA to simulate impacts with concrete barrier are presented. They are two different pendulum simulations of a concrete parapet with a steel railing on top. The deformation and damage profiles along with the force time history were used as measures for comparing tests and simulations. Both examples indicate a reasonable correlation between tests and simulations. Figure 1a shows the response of the concrete parapet as tested and figure 1b shows the response calculated from the LS-DYNA simulation.*



(a) (b)

Figure 1 Test and LS-DYNA simulation of a concrete parapet and steel rail system.

Introduction

Concrete barriers have been extensively used in roadside safety applications. Traditionally, finite element analysts characterized these barriers are rigid for two main reasons. First reason has to do with computer hardware speed and thus analysts would save great time by assuming rigid material behavior for the concrete barriers. The other reason was the lack of suitable constitute model that captures the behavior of concrete while not being overwhelmingly difficult to use. In this paper, the new material model *MAT_CSCM (and the short input version *MAT_CSCM_CONCRETE) which is available in LS-DYNA version 971 is used to simulate pendulum impact of a concrete barrier. This paper presents the modeling and analysis steps used to archive such endeavor as well as compares test results with the results of the simulations.

Tests Description

The T4 railing consists of a 15-in. tall metal rail anchored to the top of an 18-in. tall concrete parapet wall, providing an overall rail height of 33 in. The metal rail consists of a short section of an elliptically shaped steel tube welded atop a post fabricated from steel plate. The steel post is welded to a steel base plate that is anchored to the concrete parapet.

In one design variation, the width of the steel-reinforced concrete parapet is 10 in., and the steel rail is attached to the parapet using four 7/8-in. diameter anchors. In the second variation, the width of the concrete parapet is 12 ½ in., and the steel rail is attached to the parapet using three 7/8-in. diameter anchors.

For each design variation of the T4 bridge rail, two “similar” specimens were constructed and tested. For the design variation with the 10-in. wide parapet, the test designations were P3 and P4 as shown in Figure 2. The observed failure mode in both tests was punching shear failure of the field side of the concrete parapet due to load applied through the anchor bolts and baseplate. In the 10-in wide parapet tests, the impact forces caused the concrete to fracture and spall off the field side of the parapet. Additionally, cracks in the top of the concrete parapet radiated outward from the outside anchor bolts of the base plate and pieces of concrete were pushed out on the field side of the parapet. Damage to the test specimen with a 10-in. wide parapet after test P3 and test P4 is shown in Figure 3.

On the other hand, on the 12½-in wide and 3-bolt parapet tests, the parapet did not break and there were no significant fracture of concrete in the parapet. The only visible damage was the crack lines radiating from the base plate area.

Figure 2 Test set-up for the 4-bolt parapet.

Figure 3 Parapet damage after test P3 and P4.

Simulation Methodology

The methodology followed to model the bridge parapets, simulate pendulum impacts, and evaluate the concrete material model consisted of the following steps:

1- Construct finite element model of the two variants of the T4 rail system.
2- Incorporate the new concrete material model for concrete parts (i.e., parapet and bridge deck).
3- Identify critical parameters of the model and investigate the performance of the model through finite element analysis (FEA).
4- Compare results with test data and identify any further investigation needed.

Overview of T4S Parapet/Deck Model

The finite element representation of the T4 bridge rail specimens consists of the following components:

1- Concrete parapet and deck,
2- Steel reinforcement and anchor bolts, and
3- Steel elliptical rail, post, and base plate.

Figures 4 and 5 show the model components for the 4-bolt parapet designs, respectively. The concrete parapet and bridge deck were modeled using solid elements, as were the steel base plate and the plates comprising the steel post. Shell elements were used to model the elliptical steel rail. Beam elements were used to model the anchor bolts and steel reinforcement inside the parapet and bridge deck.

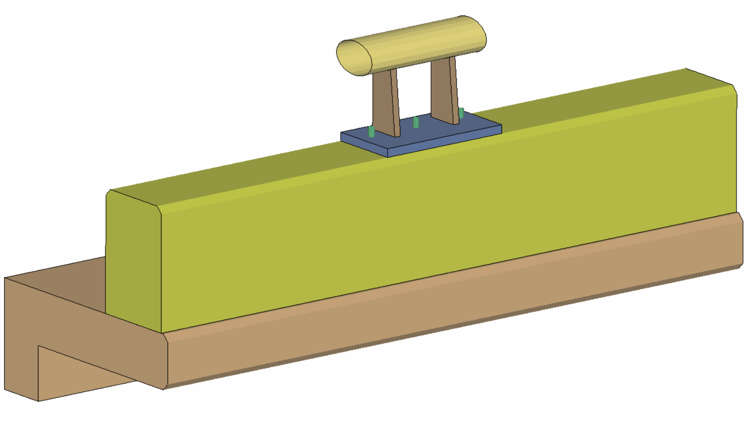


Figure 4 Model of the T4S rail.

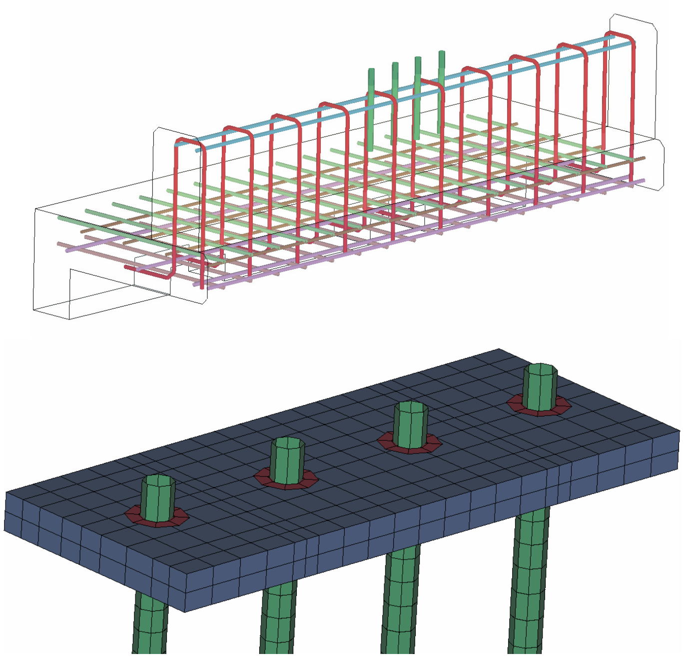


Figure 5 Reinforcement and baseplate component model of the T4S rail.

A constraint was used in to represent the interaction between the anchor bolts, steel reinforcement, and the surrounding concrete continuum. The steel reinforcement and anchor bolts are coupled (rather than merged) to the surrounding concrete continuum. This was achieved using the **`*CONSTRAINED_LAGRANE_IN_SOLID`** feature in LS-DYNA. In this constraint, the steel reinforcement and anchor bolts are treated as a slave material that is coupled with a master material comprised of the deck and parapet concrete. Using this methodology, the slave part(s) can be placed anywhere inside the master continuum part without any special mesh accommodation. Figure 6 shows two meshing schemes, one uses merged nodes between beams and solids while the other uses coupling through **`*CONSTRAINED_LAGRANE_IN_SOLID`** feature between solid and beam parts.

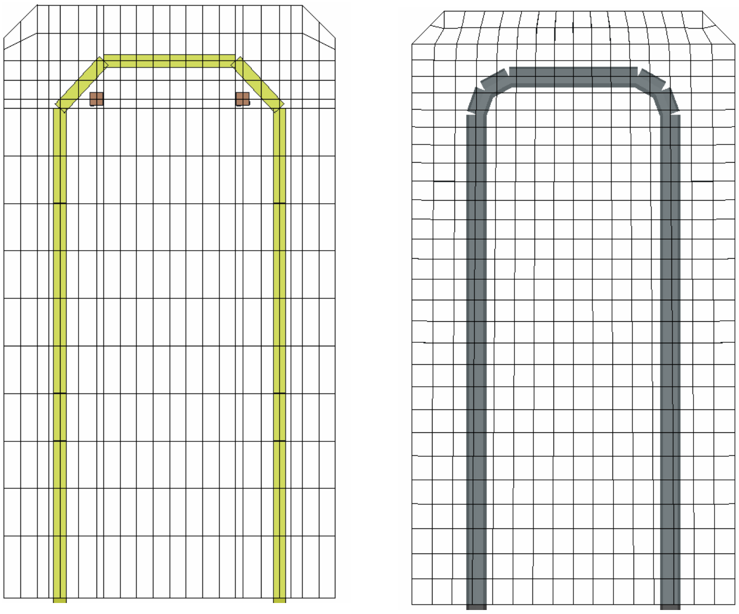


Figure 6 Merged beam and solid elements (left) and coupled beam and solid elements (right).

The parapet and deck were both modeled using the new concrete material model developed by APTEK. In the beta version of LS-DYNA 971, it is designated as material type 159. The model name is **`*MAT_CSCM_CONCRET`** or **`*MAT_CSCM`**. The "**`_CONCRET`**" suffix indicates a short input format that utilizes hard-coded default values for numerous variables, while the other name indicates a long input format for which the user must supply values for all the required input parameters. The short input format will work with default model parameters based on the user supplied compression strength of concrete (f'_C) and the maximum aggregate size (Dagg), while the long format requires the user to explicitly define all the model parameters.

For the T4 bridge rail analysis, the concrete had a maximum aggregate size of 25.4 mm (1 in.). The concrete used for the bridge deck had an average compressive strength of 35.5 MPa (5,152 psi) on the day of testing. The parapet concrete had an average compressive strength of 30.44 MPa (4415 psi).

Results

For the 10-in wide parapet, the parapet damage consisted of a failed region of concrete which was similar to that observed in tests P3 and P4 as shown in Figure 7. Moreover, velocity-time histories for the pendulum tests and simulation are within reasonable agreement as presented in Figure 8.

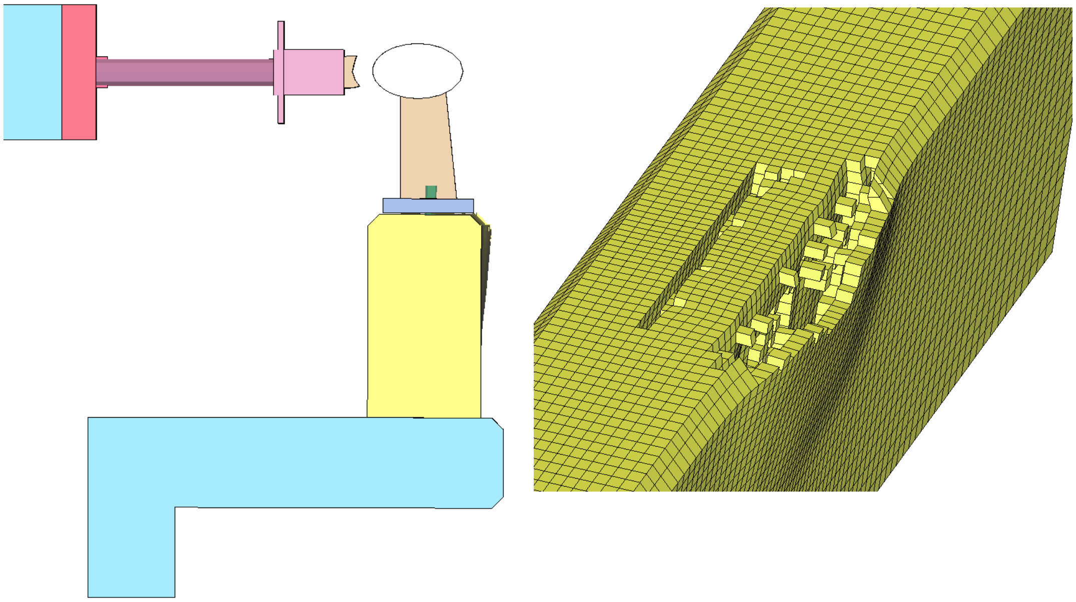


Figure 7 The 10-in wide parapet after impact simulation.

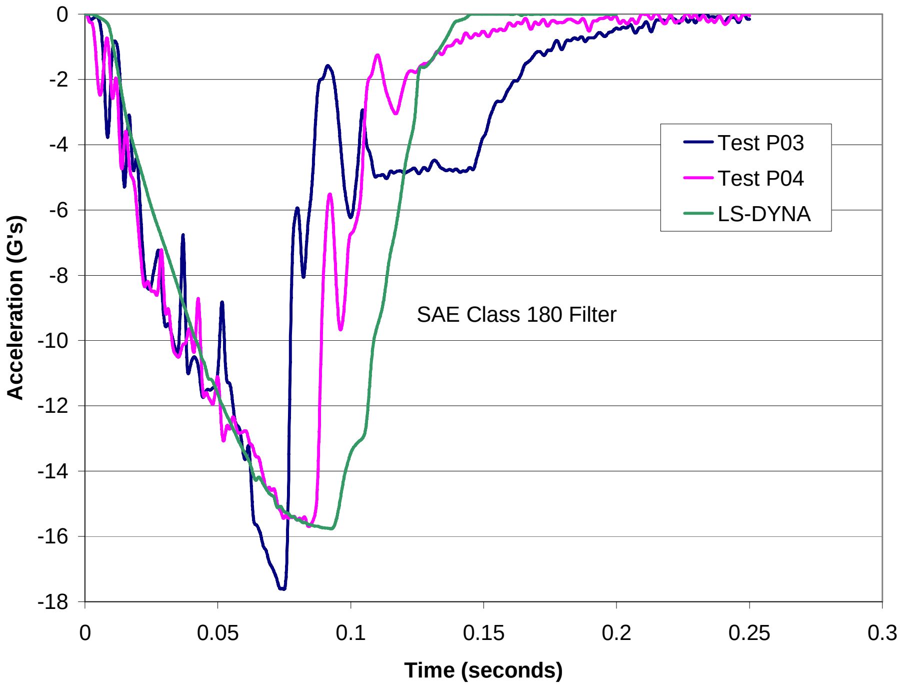


Figure 8 Pendulum bogie accelerations for the 10-in wide parapet impact.

As for the 12 ½ -in wide parapet simulation, the damage observed in simulation was limited to a group of elements indicating potential cracks similar to those observed in tests P5 and P7 as shown in Figure 9. Moreover, velocity-time histories for the pendulum tests and simulation are within reasonable agreement as presented in Figure 10.

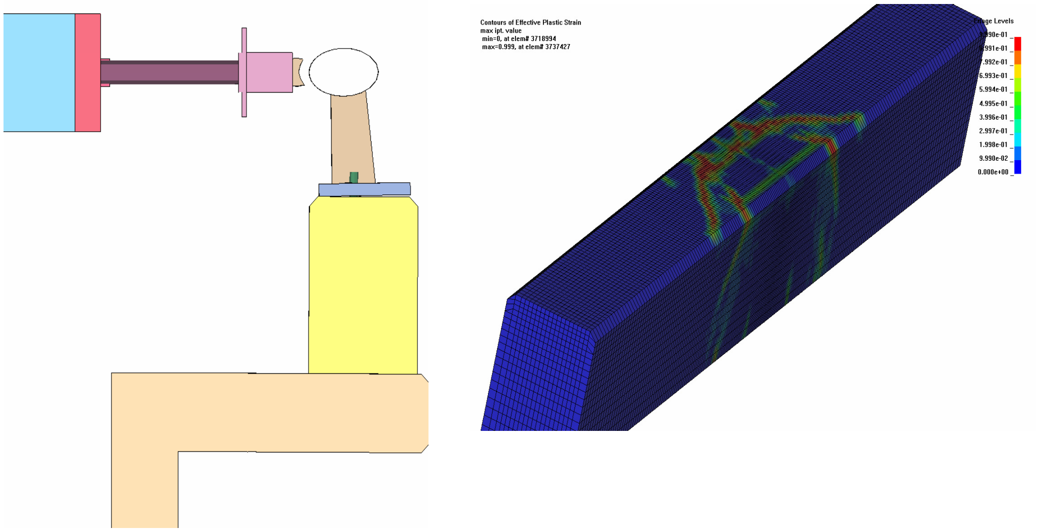


Figure 9 The 12 1/2-in wide parapet after impact simulation.

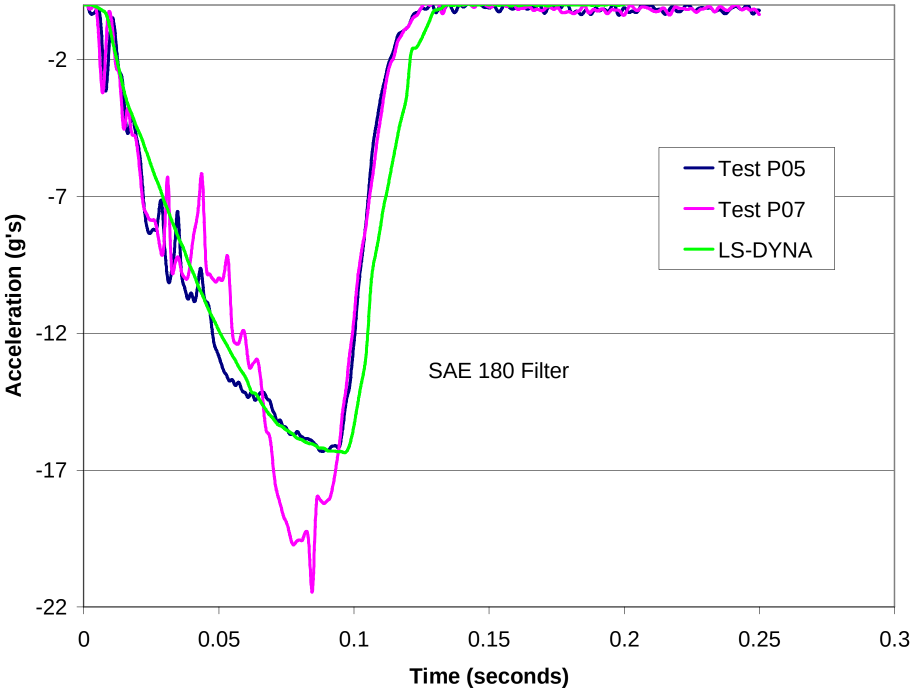


Figure 10 Pendulum bogie accelerations for the 12 ½-in wide parapet impact.

Summary and Conclusions

Numerical analyses were conducted using the newly developed LS-DYNA concrete material model. The analyses showed very good potential for useful application of the model in analyses of steel-reinforced concrete roadside safety structures. The T4 bridge rail study is a good example of the. The analyses of the T4 bridge rail alternatives provided a good benchmark for evaluating the strengths, sensitivity, and usability of the new concrete material model. Use of the pendulum tests provided more controlled impact conditions and eliminated many variables associated with the comparison of simulation to full-scale vehicle crash tests. Two specimens were tested for each design variation, providing some information regarding system variability. Further, the two design variations demonstrated different levels of damage to the concrete parapet which provided an opportunity to assess the sensitivity of the concrete model to small design changes under similar loading conditions.

Acknowledgment

This paper summarizes the results of a research task that was funded by the Federal Highway Administration (FHWA) through APTEK, Inc. The help of Mr. Martin Hargrave (FHWA) and Ms. Yvonne Murray (APTEK, Inc.) is greatly appreciated.

References

1- Schwer, L.E. and Murray, Y. D. "Continuous Surface Cap Model for Geomaterial Modeling: A new LS-DYNA Material Type", 7th International LS-DYNA Users Conference, 2002.

2- Murray, Y. D. "Theory and Evaluation of Concrete Material Model 159". 8th International LS-DYNA Users Conference, 2004.

3- LSTC. "Developments in Version 971", FEA Information Newsletter, January, 2004.