TRC0605

Development of Design Improvements for Bridge Railings

Nathan B. Edgar

Final Report

2007

FINAL REPORT

DEVELOPMENT OF DESIGN IMPROVEMENTS FOR BRIDGE RAILINGS

Nathan B. Edgar, Ph.D.

(The opinions, findings, and conclusions expressed in this report are those of the author and not necessarily those of the sponsoring agency.)

Arkansas Transportation Research Council
April 2007
TRC 0605

ABSTRACT

Bridge railings in Arkansas are designed according to the AASHTO Specifications. The AASHTO LRFD (Load and Resistance Factor Design) Bridge Design Specifications include Test Level Criteria that increase the design forces for bridge railings when compared to the AASHTO Standard Specification for Highway Bridges 17th Edition, resulting in large increases in the required amount of reinforcing in the bridge railing and deck

The LRFD Specifications also recommend the railing systems and their connection to the deck be warranted by crash testing, but a crash test program is expensive and cannot fully test for effects of discontinuities and varying reinforcing patterns. The LRFD code does permit the use of finite element methods to model bridge railing and deck response.

The concrete bridge railing used in Arkansas needs was analyzed using the general purpose finite element code, ABAQUS. Finite element simulations were used to determine stresses, deflections, and crack pattern predictions in the parapet and deck system, and recommendations are made for optimal reinforcement in these systems.

DEVELOPMENT OF DESIGN IMPROVEMENTS FOR BRIDGE RAILINGS

Nathan B. Edgar, Ph.D., P.E.

Introduction

The incorporation of the finite element method as a design tool in structural mechanics dates back to the late 1960's. Since its inception, it has found widespread acceptance in modeling steel and aluminum support structures with structural design engineers. However, reinforced concrete does not provide the predictable constitutive properties of steel or aluminum. The differing material properties of concrete and steel and the fact that the concrete itself is a composite made up of mortar, aggregates of varying size, and air voids make predictive behavior very difficult.

Despite the maturity of the finite element technique, numerical modeling of reinforced concrete still lacks consensus among engineers and researchers. Comprehensive literature reviews of finite element techniques used to model reinforced concrete have been published by Darwin (1993) and Barbosa (1997) and illustrate the variety of approaches used to model this material.

Successful use of the finite element method to model reinforced structures including bridge decks and parapets have been presented by Biggs, et. al. (2000) and Thiagarajan and Roy (2005). In both studies, ABAQUS, a general purpose finite element code was used to predict displacements, strains, and stresses to model the nonlinear behavior of reinforced concrete due to externally applied loads.

Material cracking of the concrete and yielding of the steel reinforcement are the dominant failure modes for most reinforced concrete structures. Micro-cracks can develop during the curing process, under self-loading conditions, and due to temperature effects. Under externally applied loads, these micro-cracks will coalesce into finite sized cracks that continue grow and propagate under increased loads. The surfaces that surround the crack represent discontinuities that provide extensive challenges for the finite element technique to represent with any degree of accuracy.

To accurately predict the location, orientation, and growth of cracks in concrete requires an approach based on 'first principles' (i.e. three-dimensional elasticity/plasticity theory coupled with fracture mechanics). While 'first principle' methods show promise for ultimately understanding the detailed response of reinforced concrete to applied loading conditions, they have yet to yield an efficiently implementable formulation algorithm for use in commercial finite element codes. Most commercial codes employ cracking models that provide equivalent responses (e.g. displacements, strains, and stresses) of the structure due to externally applied loading conditions without accurately accounting for the cracking mechanism itself. These models often require user-defined properties that can be manipulated to produce satisfactory agreement with calibration tests and experimental results.

The most popular of these techniques was introduced by Scanlon and Murray (1974) and is known as tension stiffening. This technique attempts to account for the ability of cracked concrete to exhibit a significant stress-strain response due to its bond integrity with the steel reinforcement between large crack propagations.

Three strategies have emerged for representation of steel reinforcement within concrete structures. These include smeared crack models, embedded models, and discrete models. Investigations of these techniques have been conducted by Barzegar (1994), Ramaswamy et. al. (1995), and Jiang and Mirza (1997).

After cursory numerical investigations the smeared crack model was determined to be the most appropriate for the present study due to computational efficiency and the extensive literature on which to draw on.

Purpose and Scope

Bridge railings in Arkansas are designed according to the AASHTO Specifications. The AASHTO LRFD (Load and Resistance Factor Design) Bridge Design Specifications include Test Level Criteria that increase the design forces for bridge railings when compared to the AASHTO Standard Specification for Highway Bridges 17th Edition, resulting in large increases in the required amount of reinforcing in the bridge railing and deck

The LRFD Specifications also recommend the railing systems and their connection to the deck be warranted by crash testing, but a crash test program is expensive and cannot fully test for effects of discontinuities and varying reinforcing patterns. The LRFD code does permit the use of finite element methods to model bridge railing and deck response.

The concrete bridge railing used in Arkansas was analyzed using finite element techniques to determine the response of the proposed parapet design and the resulting response of the bridge deck. These simulations include a review of resulting forces in the barrier and deck system.

Four objectives were to be met in this study as follows:

1.) Develop finite element models of current bridge railing and deck system that account for discontinuities such as open joints, drain slots, skews, etc, with respect to specified level of service.
2.) Make recommendations for an empirical design procedure.
3.) Recommend efficient reinforcing pattern of the railing and deck based on finite element analysis of current bridge railing systems.
4.) Determine whether a crash-testing program is warranted based on the required changes to the current bridge railing system.

Initially, to meet these objectives it was planned to conduct a series of event simulations with impact pre-impact conditions specified by NCHRP 350 and conducted using a crude model (bogey) of an impact vehicle to test the performance of the current bridge railing system and its modified candidates. However, in consultation with bridge engineers at AHTD, it was decided that a more efficient approach would be to use the static loading requirements required for AASHTO Railing Test Level 4 (AASHTO LRFD Bridge Design Specifications 3rd Edition 2004 ,Section 13: Railings Table A13.2-1). These results could then be compared to AHTD in-house yield line calculations.

Due to the nonlinear behavior of the material model, this static load approach required the use of an explicit technique to slowly load the structure until the maximum design load was reached. The specified time history of the loading was conducted so that all inertial and wave propagation effects were negligible. The explicit formulation required the use of a mesh dependent time step that restricted computational efficiency and resulted in long solution times.

Validation cases were computed to determine the effectiveness of the various proposed solution methodologies in determining strains and stresses in the parapet and bridge deck. The results of these verification problems are briefly outlined in the following section.

Methodology

The mechanical constitutive behavior of reinforced concrete is very difficult to model due to the extensive nonlinearities that exist once the material starts to crack. The release of energy during the cracking process coupled with the complex bond interaction between the concrete and the rebar can lead to significant instabilities. To alleviate such difficulties, these effects can be modeled using a 'tension stiffening' assumption for the concrete to simulate the load transfer in this cracked concrete/rebar interface area.

At the onset of the project, a number of modeling approaches and formulations were investigated to determine their suitability for use in the present study. The primary candidates included the explicit codes LS-DYNA and ABAQUS to mimic the static loading conditions on the parapet. Ultimately, ABAQUS was chosen, and a brief overview of its cracking model is discussed here.

Concrete material model

The brittle cracking model in the ABAQUS/EXPLICIT finite element code was designed to model reinforced concrete applications that are dominated by tensile cracking, and assumes linear elastic compressive behavior.

Reinforcement is modeled as one dimensional elements exhibiting an elasto-plastic behavior. These elements can be defined individually or embedded in host elements to mimic the effect of rebar in plain concrete. This implementation detail for the various element types investigated (solid, shell, and membrane) proved to be most challenging.

Cracking is accounted for using a smeared crack formulation to model these discontinuities. Fixed, orthogonal cracks are assumed and a Rankine criterion is used to detect crack initiation. The crack surface is taken to be perpendicular to the maximum principle tensile stress.

Tension stiffening is used to model the post-failure stress-strain relationship. While this input parameter can be manipulated by the user, there is a physical basis for its use based on the assumption that for significantly reinforced structures, the strain softening after failure reduces the stress linearly to zero at a total strain about ten times the strain at failure (HKS, 2006). Calibration of this parameter is recommended. The results from the validation problems were used to evaluate the appropriate parameter used. Starting with a recommended value, evaluation of the converged solutions were used to determine the sensitivity of the final results on the tension stiffening parameter

Model validation

To validate the effectiveness of the finite element code to predict the reaction of a reinforced concrete structure to external loads, a series of studies were undertaken to address the following issues:

1.) Accuracy: The examples that were evaluated had either analytical or published experimental results to compare with. These two concepts of accuracy are different. Comparing a numerical technique with an analytical solution determines how well a particular formulation can approximate the mathematical model. Strong agreement is expected here. Comparison with experimental results determines the degree with which the numerical model (and by implicit extension the mathematical model) can represent or mimic the physical system being investigated. By evaluating the numerical solution in light of experimental results, one can adjust critical aspects of the numerical formulation.

2.) Element type: The use of shell, beam, and solid elements in ABAQUS have been used to successfully model bridge decks and parapets (Thiagarajan and Roy, 2005 and Biggs et. al., 2000). In the present study, it was determined that solid elements for both the parapet and the deck would be most appropriate. Validation problems were chosen to test the use of these solid elements with superposed rebar elements.

3.) Material model: The nonlinear material model requires specification of the tension stiffening parameter. In ABAQUS, this user defined stresss-strain curve controls the post-cracked behavior of the concrete, as shown in Figure 1.

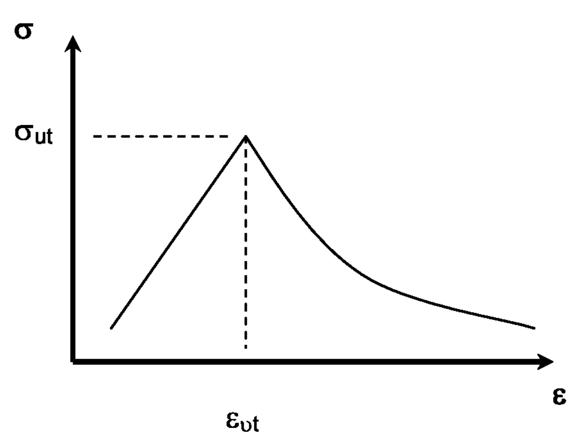

Figure 1. Post cracked behavior using tension stiffening model.

Singly reinforced beam

As a first validation problem, an under-reinforced beam with tension reinforcement was evaluated using analytical calculations (Nawy, 2000) and compared to the values predicted by the finite element model. The simply supported beam has a clear span of 20 ft, a width of 10 in., and a total depth of 16 in., the cross-section is shown in Figure 2. The concrete strength, f_c', was 4500 psi and the rebar yield strength was 60,000 psi.

The beam is simply supported and is subjected to a uniform load of 1050 lb_f/ft over a 2 ft. section in the middle of the span.

Beam properties

b = 10 in.
d = 13 in.
h = 16 in.
$A_s = 1.32$ in^2 (three #6 bars)
$f_c' = 4500$ psi
$f_y = 60,000$ psi
$E_c = 3.6 \times 10^6$ psi
$E_s = 29 \times 10^6$ psi

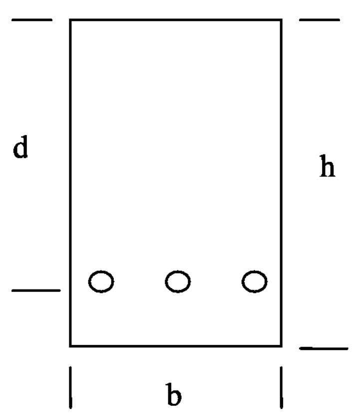

Figure 2. Beam Cross Section

The finite element model was used to determine the deflection at mid-span and the stress in the rebar. While beam elements are a natural choice for this model, it was determined that solid elements would be used to investigate their suitability in the present study. Eight-node linear solid, or continuum, elements (C3D8R) were chosen, with the rebar being superposed on the concrete solid elements. This allows the concrete and the steel to be represented separately, with their interaction mimicked using the tension stiffening model.

Reduced integration is used because the primary mode is bending and 3 layers of elements are used to model the beam thickness.

The beam deflection comparison for half the beam span is shown in Figure 3. Sixty uniform elements were chosen to model the beam. Investigation of element skewness on solution quality was also conducted. For elements that had face angles the varied up to 20^{o}, results varied from those below by up to 8% at the mid-span. Therefore, care was taken to keep the interior element angles as close to 90^{o} as possible.

The stress in the reinforcement was calculated to be 48 ksi and the finite element results showed the reinforcement stresses to be 46.6 ksi. Both the deflection and steel stress values showed a sensitivity to the tension stiffening parameter, loading increments, and element skewness. A series of investigations showed that the tensioning stiffening parameter of 0.0025 was most appropriate for the current study. The loading increment and the element quality along with the tension stiffening parameter used for this study will now be implemented for the slab study and comparison with its experimental results.

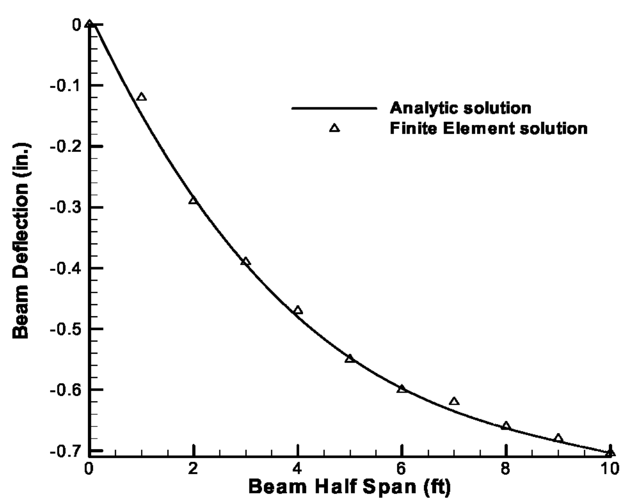

Figure 3. Beam deflection result comparison.

McNeice slab study

A two-way slab, simply supported at its corners, was tested by Jofriet and McNeice (1971) and has subsequently been studied by Hand et. al. (1973), Lin and Scordelis (1975), and Bashur and Darwin (1978). It provides a well-documented test case with experimental results.

The test slab was 36 inches square with a thickness of 1.75 in. The reinforcement material was wire mesh with resulting in a 0.85% steel ratio. The slab was loaded via a concentrated load, **P**, at its center as shown in Figure 4.

The concrete model parameters resulting from the beam study were used for the slab. A finite element model of 48 uniform elements (4 elements x 4 elements x 3 elements through the slab thickness) was used to model one-quarter of the slab by taking advantage of the slab symmetry. The mesh is shown in Figure 5.

A maximum load of 3 kip at the slab center is applied and the deflection is monitored at the midspan. At the maximum load, the finite element approximation for the deflection was 0.29 in. as compared to 0.3 in. from the experimental study.

The mesh was both coarsened and refined to determine the extent of mesh dependence on the numerical solution. The results of a 27 element uniform mesh and a 150 element mesh showed virtually no change in the monitored deflection. Some stress values showed a 3% discrepancy for the coarse mesh.

The ABAQUS documentation provides a way to decrease the computational effort by using mass scaling. This is accomplished by increasing the density of the concrete by a factor of 100 and thus increasing the stability limited time step by a factor of 10. This method was not utilized in this study.

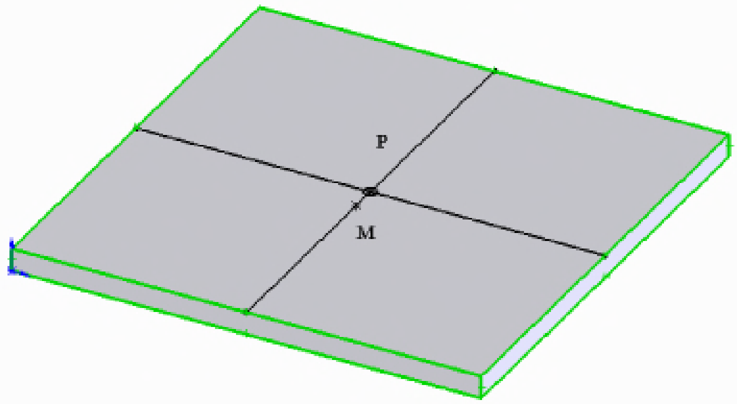

Figure 4. McNiece two-way slab

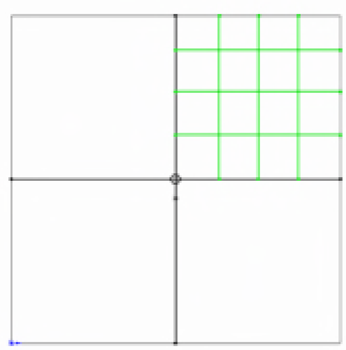

Figure 5. Finite element mesh of slab quarter.

Parapet/Deck Study

Preliminary parapet and deck designs, including reinforcement patterns, were received from the Bridge Division of the Arkansas Highway and Transportation Department (AHTD). A solid model of a 20 ft parapet, with New Jersey profile, and deck connection was created, and an initial study was conducted to determine appropriate boundary and connectivity conditions to be applied for support of the deck and also the interface between the deck top surface and the cast-in- place parapet.

It was determined that the deck would be modeled with simple supports at locations of 31 in. and 145 in. from the outer deck face to coincide with the longitudinal beam supports, and a cantilever condition implemented at 193 in to truncate the finite element mesh. Deck slope was not accounted for. Figures 6 through 8 show isometric, front, and side views of a 12 ft section of the solid model for the original deck and parapet design. Features in this model include a 4 ft drain slot centered along the length of the parapet.

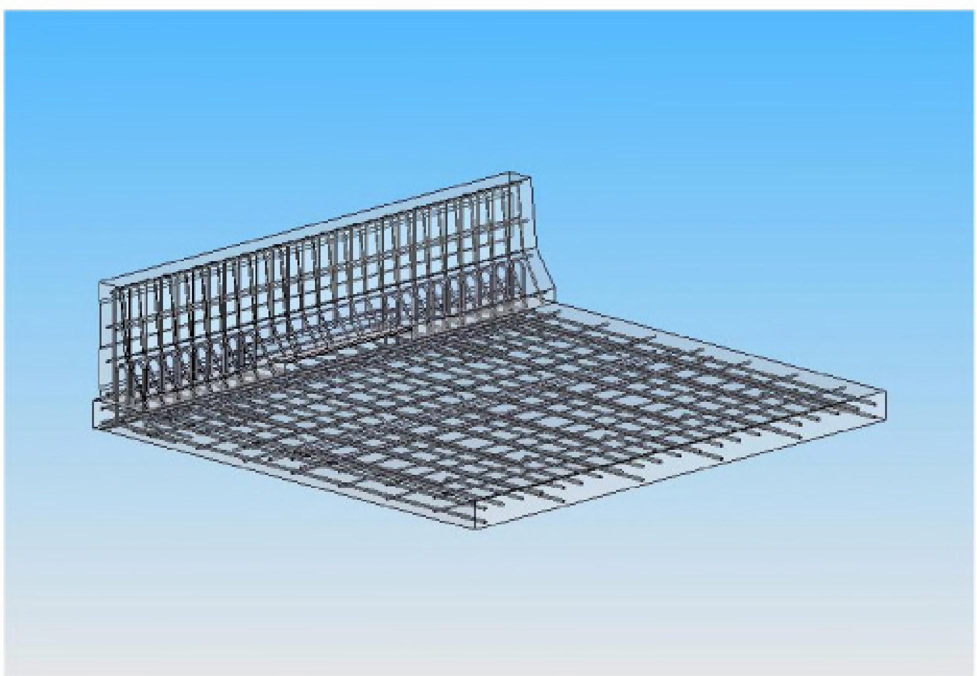

Figure 6. Isometric view of the original deck and parapet system

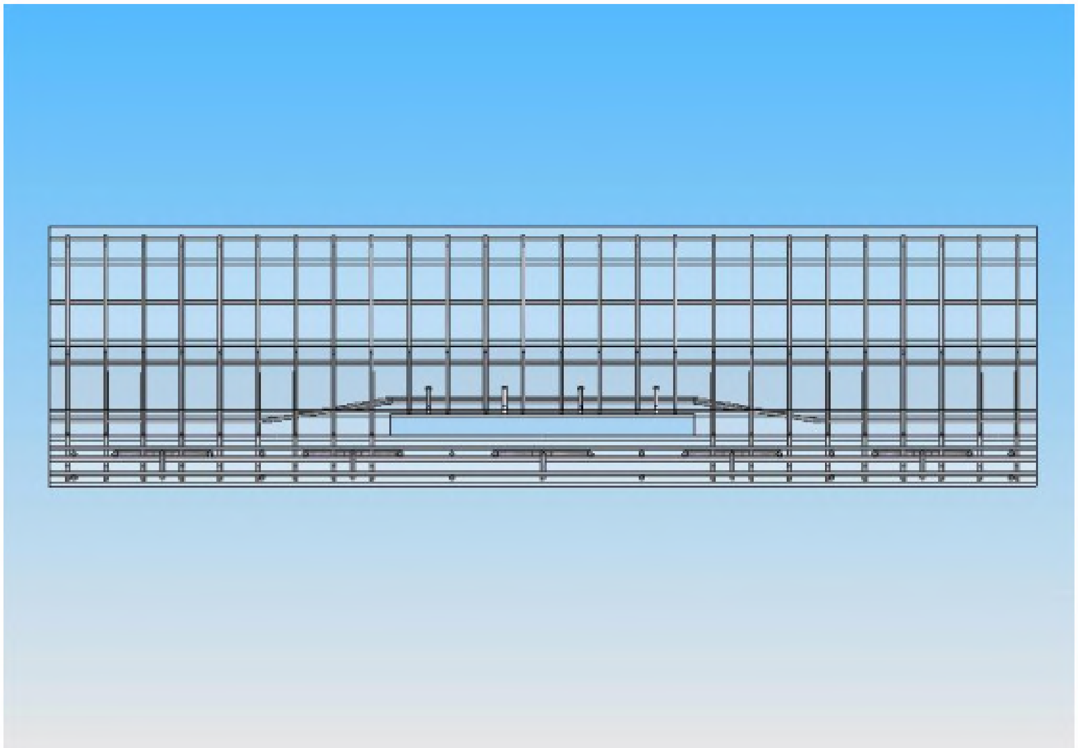

Figure 7. Front view of the original deck and parapet system.

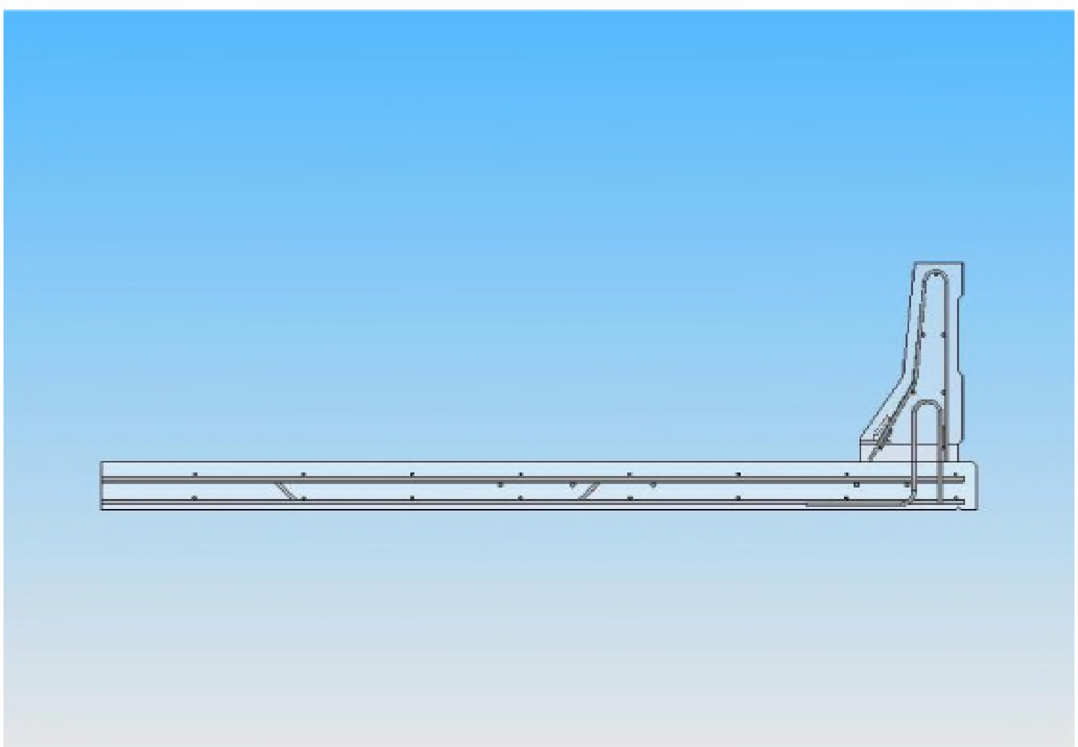

Figure 8. Side view of the original deck and parapet system.

Finite element study

AASHTO design forces and loading area designations for traffic railings are based on crash test observations. For the test level 4 (TL-4) designation of the proposed parapet, a transverse loading of 54 kip is applied at the top of the parapet over a length of 3.5 ft. as shown in Figure 9. Additionally, a longitudinal load of 18 kip is applied over the same contact area. The contact area coincides with the rear axle wheel diameter of a truck, and a 2.5 in. width was used to distribute the force over the inclined face of the parapet.

AHTD specified all concrete to have a minimum compressive strength, f_c', of 4000 psi and the reinforcing steel to be grade 60, with a yield strength of 60 ksi.

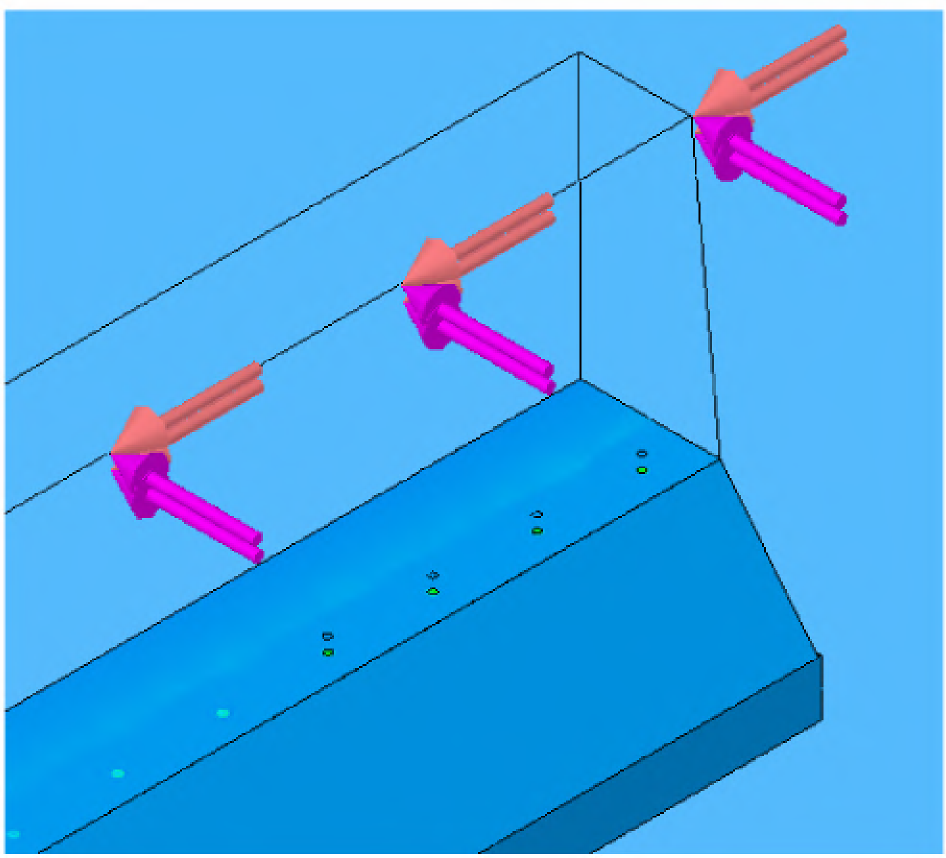

Figure 9. AASHTO loading designation for proposed parapet. (Top section of parapet shown as transparent to enhance viewing)

In consultation with AHTD bridge engineers, three critical loading points were determined for investigationof a parapet in service. A test section with an overall length of 20 ft. and a 4 ft. drain slot centered along the parapet length was designated for the study. These critical areas include the location of an open parapet joint, a partial depth parapet joint, and the drain slot.

The open parapet joint was assumed to be a 1 in. opening that starts at the top of the parapet and extends downward a distance of 29 in., stopping 4 in. from the deck's top surface. The partial depth joint starts at the top of the parapet and extends downward a distance of 19 in.

The connectivity between the parapet and deck is due to a number 5 bar that comes up from the deck structure and forms a vertical hook that rises 11 inches above the deck's top surface. This bar will be called a hatbar from this point forward in the report, and is shown in Figure 10. The bottom of the hatbar is located one inch from the bottom deck surface and has an overall height of 18 inches. Its front leg is fully developed via a hook with a 16 in. horizontal length and the back leg of the hat bar is undeveloped.

The connectivity between the parapet and deck was determined to be a weakness since the parapet was cast after the deck was poured and had solidified. In the finite element model, this connectivity is described by discrete rebar elements that connect the parapet and deck pieces of the solid model and by a contact surface that stretches from the back edge of the parapet inward by 2 in. and spans the entire length of the model as shown in Figure 11.

Loading of the parapet will cause the deck/parapet interface to separate, with rotation of the parapet being resisted due to the stresses developed in the hatbar legs and the contact area along the back of the parapet.

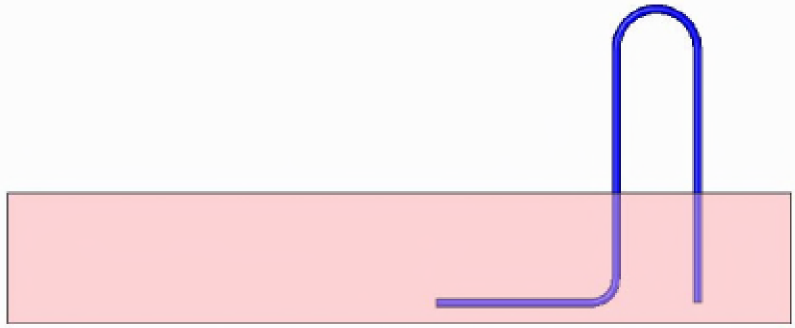

Figure 10. Original deck and hatbar.

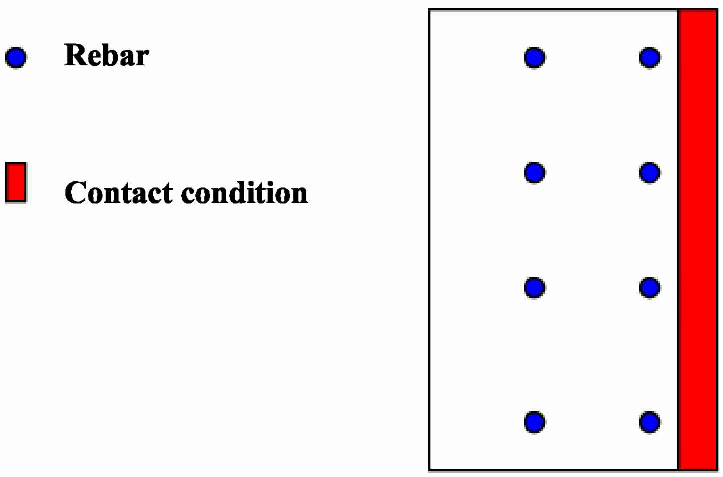

Figure 11. Connectivity for representative section of parapet bottom.

An initial study was conducted assuming a rigid deck response to act as a "worst case" scenario. The hatbars were uniformly spaced at 6.5 in. Finite element results showed that the placement of the hatbar with respect to the parapet and the leg spacing caused tension in both legs. The front legs of the three hatbars nearest the full depth open joint experienced yielding.

The hatbar configuration was changed to widen the base from 6.5 in. to 8.5 in. at the deck/parapet interface. The spacing at the bottom of the hatbar is 10.6 from outside edge to outside edge. The new configuration is shown in Figure 12.

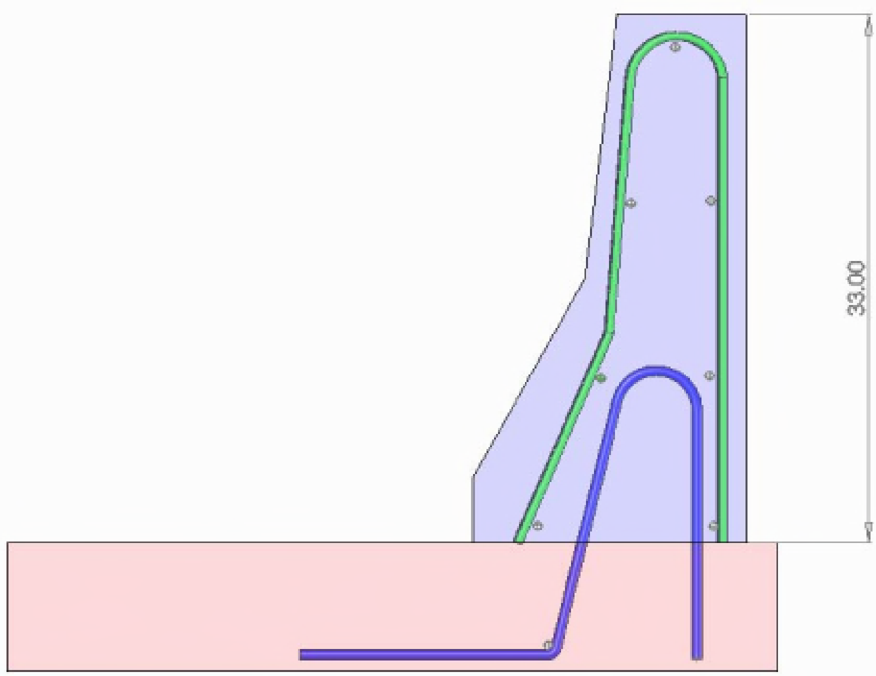

Figure 12. Modified reinforcement detailing.
(Deck reinforcement is not shown)

Results

At the three critical load locations; an open depth parapet joint, a partial depth parapet joint, and a drain slot, finite element studies were conducted to determine the response envelope for the required loading conditions. The monitored responses included rebar stresses, maximum parapet deflection, and crack location and configuration.

A baseline design for the reinforcement was available from yield line calculations conducted by AHTD bridge engineers. Stirrup and hatbar spacing was manipulated to induce varying responses to the loading. The open joint proved to be the most sensitive to reinforcement spacing and will be addressed first, followed by the drain slot and finally the partial depth parapet joint.

Open parapet joint study

A finite element model of the parapet was developed using 8-node tri-linear solid elements and reduced integration. Twelve elements were used to span the parapet thickness to ensure proper resolution of the bending behavior. The deck was modeled using a uniform mesh with 6 elements used to model the thickness. The ABAQUS/EXPLICIT finite element code was used to evolve the static solution using the prescribed loading conditions shown in Table 1, per AASHTO specifications for TL-4 certification.

Table 1

Design Forces and Designations	Test Level 4
Transverse Force	54 kip
Longitudinal Force	18 kip
Transverse loading length	3.5 ft.
Longitudinal loading length	3.5 ft.

For the specified loading conditions, 25 separate cases were run using varying reinforcement spacing with construction variances of up to 2 inches. The first initial cases were used to determine the sensitivity to material properties and to the tension stiffening parameter used to model the post-cracking behavior. A baseline value of 0.0025 was used for the tension stiffening.

It was found that there is little sensitivity to variations in material properties up to the ±15% variances used here. As expected, the tension stiffening parameter did provide significant variation in the results. This was especially true when the reinforcement spacing varied beyond the optimum design specification. When the reinforcement spacing was uniform, this played a somewhat lesser role.

Results of this study show that optimum spacing for reinforcement is 7.5 in. from outside bar surface to outside bar surface for the stirrups. The stirrups provide internal strength to resist inclined shear cracking. From Figure 13, it can be seen that this is needed for the open joint loading condition. The red fringes show areas of high probability for cracking. The case shown if for the optimal reinforcement spacing for the stirrups and hatbars. For this case, it is critical that the stirrup be located within 3 inches of the joint and that the hatbar be located at least 2 inches from the joint.

Figure 14 is an end view of the parapet and shows how properly placed reinforcement can arrest the inclined shear crack that appears when the stirrups are place too far from the parapet end. For the improperly spaced stirrup case, a crack pattern develops from the red zone on the front parapet face to the lower cutout on the back edge of the parapet with the nearest stirrup exceeding its yield strength. Table 2 shows representative results of the cases run including stirrup and hatbar spacing, maximum rebar stress, and maximum transverse deflection of the parapet. For the cases cited, the spacing of the stirrup and hatbar from the end of the parapet joint are 3 in. and 2 in., respectively.

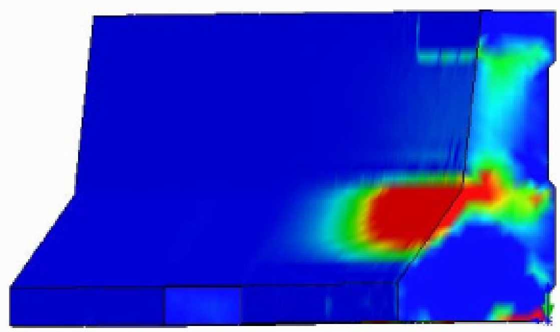

**Figure 13. Cracking predictions for open joint parapet case.
(Optimal reinforcement spacing used)**

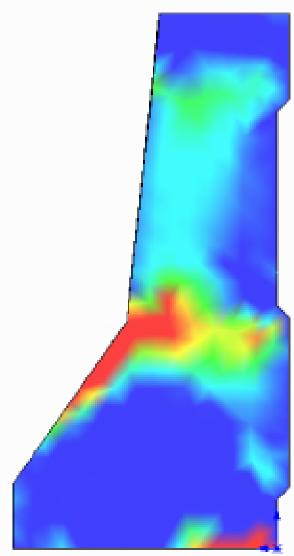

**Figure 14. Cracking prediction for open parapet case.
(End view)**

Table 2

Open joint results (All cases noted have stirrup located 3 in. from joint and hatbar 2 in. from joint)					
Case #	Stirrup spacing (in.)	Hatbar spacing (in.)	Maximum rebar stress (ksi)	Maximum transverse deflection (in.)	Location of maximum stress
4	6	6.0	53	0.65	Stirrup
7	6	8.0	yielding	0.69	Hatbar
8	9	6.5	yielding	0.78	Stirrup
12	9	7.5	yielding	0.82	Both
14	9	6.0	yielding	0.80	Stirrup
15	8	4.0	55	0.83	Stirrup
20	7.5	5.5	57	0.81	Stirrup
23	7.5	6.5	58	0.85	Stirrup

Drain slot study

Using the uniform spacing for the reinforcement as shown in Case 23 from the open joint study, loading above the center of the drain slot opening was studied to determine the parapet response. Unlike the previous case, the reinforcement spacing was kept constant and the loading location was moved from the center to both the upstream and downstream locations of drain opening. The center location produced the most critical results. Cracking predictions for this case are shown in Figure 15. The maximum rebar stress was determined to be 48 ksi and the maximum transverse parapet deflection was determined to be 0.53 in.

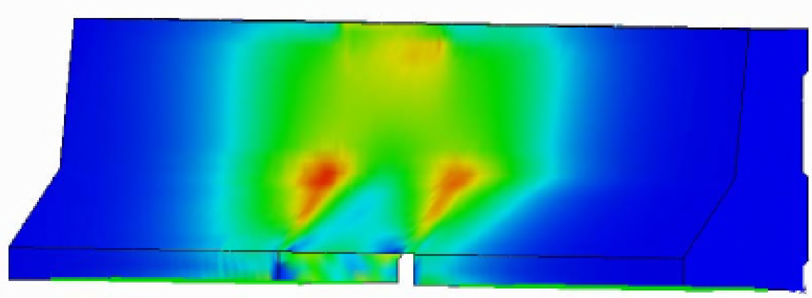

Figure 15. Cracking predictions for drain slot study.

Partial depth joint study

The loading condition for the partial depth joint proved to be the least critical of the three studies. Loading of the parapet immediately downstream of the joint showed that the maximum reinforcement stress was 39 ksi and the maximum transverse deflection of the parapet was 0.56 in.

The cracking prediction for this case is shown in Figure 16. It is noted that the support given in the lower portion of the parapet would appear to reduce the cracking probability in this region, even though it is pronounced in the drain slot study. Uncertainty regarding this behavior was not alleviated by looking at mesh quality, tension stiffening, or material properties.

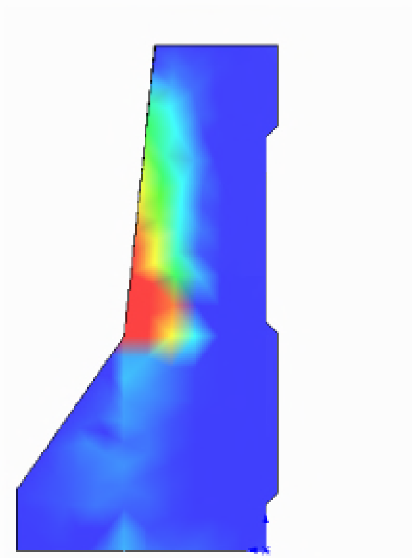

Figure 16. Partial depth joint cracking predictions.

Deck response

In all of the studies, the deck response has been unremarkable with respect to stresses, cracking, and deflection. The maximum deflection of the deck was 0.26 in. Caution was used in regarding predictions of deck response. The high stress area leading from the back edge of the parapet to the first support beam relies on tension reinforcement that is not fully developed. It is recommended that experimental study regarding this section of the deck be done to determine its response to impact loads.

Conclusions

At the onset of this study, four objectives were to be met. The first objective, to create a finite element model was successfully completed, with the results presented in the previous section. The use of the finite element method for this type of study is instructive, but much work is needed to make this tool available for the practicing structural engineer. Many of the models required days to reach a converged solution. While techniques such as mass scaling could be used to decrease the computational time, there is little published information to guide the user regarding this technique.

Over 40 cases were conducted to test the design envelope of the proposed parapet and deck structure. The critical design point was deemed to be the parapet section immediately downstream of the open joint. Spacing of the stirrup and the hatbar was critical in controlling cracking and the stresses in the reinforcement. With regard to Objective 3, it was found that the optimal reinforcement spacing was 6.5 in. for the hatbar and 7.5 in. for the stirrups. No recommended change in longitudinal reinforcement or deck reinforcement is warranted.

Objective 2 involved recommending an empirical design procedure. In the end, it was deemed that the yield line technique would be a superior method of design than the individual locations in the design space covered by the current study. It would take an enormous computational effort to develop a set of evaluation functions that would be superior to the yield line results.

Objective 4 was to make a recommendation regarding the need for crash testing of the proposed parapet. The agreement between the finite element results and the yield line calculations do not warrant the need for a crash test. This particular shape has been crash tested, and similar reinforcement patterns exist for other State DOT's.

Implementation

Based on the computed results from the finite element study, no change recommendations are warranted for the horizontal deck reinforcement. A change in the configuration of the # 5 bar that forms the vertical hook that provides connectivity between the deck and parapet is recommended with the detail shown in Figure 17.

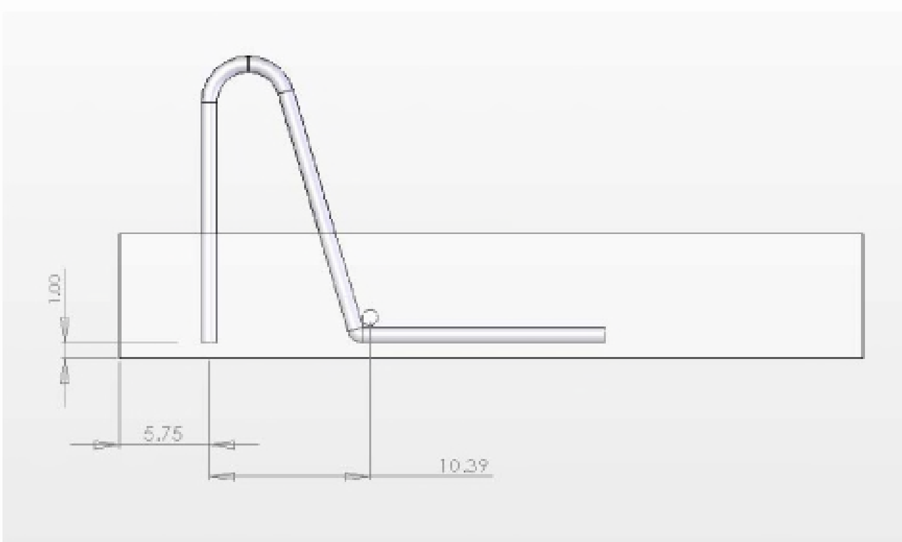

Figure 17. Hatbar detail.

Spacing of this feature was determined to be 6.5 in. with the requirement that it be within two inches of an open parapet joint.

Spacing of the # 4 bars that form the vertical stirrups was determined to be 7.5 in. with the requirement that it be within three inches of an open parapet joint

Acknowledgements

The author would like to acknowledge the use of the computational facilities provided by the Arkansas State University College of Engineering. Also, the author is indebted to Dr. Tom Parsons for guidance throughout the project. Special thanks to the TRC committee and the engineers from the Bridge Division at AHTD for their helpful oversight and recommendations during the project. Finally, thanks to Mark Bradley and James Key for their patience.

References

AASHTO LRFD Bridge Design Specification Manual 3rd Edition, 2004.

Barbosa, A. F. (1997) *A Study of Models for Nonlinear Finite Element Analisys of Concrete Structures*, Masters Dissertation, Federal University of Minas Gerais, Brazil.

Bashur, F. K. and Darwin, D., "Nonlinear Model for Reinforced Concrete Slabs," ASCE Journal of the Structural Division, vol. 104, pp. 157–170, January 1978.

Barzegar, F. (1994). Generating Reinforcement in FE Modeling of Concrete Structures. *ASCE Journal of Structural Engineering*, 120(5), pp. 1656-1662.

Biggs, M. R, Barton, F. W., Gomez, J. P., Massarelli, P. J., and McKeel, W. T. (2000), *Finite Element Modeling and Analysis of Reinforced-Concrete Bridge Decks*, Virginia Department of Transportation, Final Report VTRC 01-R4.

Darwin, D. (1993). Reinforced Concrete. *Finite Element Analysis of Reinforced-Concrete Structures II: Proceedings of the International Workshop*. New York: American Society of Civil Engineers, pp. 203-232.

Hibbitt, Karlson & Sorensen, Inc. (2005), ABAQUS User's Manual; Theory; Version 6.6, Rhode Island.

Jiang, J. and Mirza, F. A. (1997). Nonlinear Analysis of Reinforced-Concrete Slabs by a Discrete Finite Element Approach. *Computers and Structures*, 65(1), pp. 585-592.

Jofriet, J. C. and McNeice, G. M., "Finite Element Analysis of Reinforced Concrete Slabs," ASCE Journal of the Structural Division, vol. 97, pp. 785–806, March 1971.

Lin, C. and Scordelis, A. C., "Nonlinear Analysis of RC Shells of General Form," *Journal of the Structural Division*, ASCE, 101(ST3), pp. 523-538, March 1975.

Nawy, E. G.(2000), *Reinforced Concrete: A Fundamental Approach*, 4th Edition, Prentice Hall International Series in Civil Engineering and Engineering Mechanics, Prentice Hall, Inc., Upper Saddle River, NJ.

Ramaswamy, A., Barzegar, F., and Voyiadjis, G. Z. (1995). Study of Layering Procedures in Finite Element Analysis of RC Flexurla and Torsional Elements. *ASCE Journal of Structural Engineering*, 121(12), pp. 1773-1783.

Scanlon, A. and Murray, D. (1974) Time-Dependent Reinforced-Concrete Slab Deflections, *ASCE Journal of Structural Engineering*, 100, pp. 1911-1924.

Thiagarajan, G. and Roy, S. (2005), *Finite Element Modeling of Reinforced Concrete Bridge Decks with ABAQUS*, U.S. Department of Transportation, Final Report DTRS98-G-0021.