SP-88-1

Strength Evaluation of the Manotick Bridge Piers

by Gilbert A. Hartley, Gerhard T. Suter, Heinz Keller, and David C. Marett

Synopsis: A structural investigation of a bridge pier using the finite element method is presented. For the pier, supported on three caissons with loads applied at the top, the structural action is similar to that of a deep beam. The purpose of this investigation is to examine possible causes of existing cracks in the pier. The simplicity of the application and the accessibility of the computer program are emphasized. Once the finite element model is prepared, experimentation is possible with the effects of various loading conditions, all feasible support conditions and the application of external posttensioning to prevent further cracking and to relieve internal tension stresses during remedial work.

Keywords: bridge piers; caissons; computer programs; cracking (fracturing); deep beams; evaluation; finite element method; reinforced concrete; stress analysis

Gilbert A. Hartley is an Associate Professor of Civil Engineering at Carleton University, Ottawa, Ontario, Canada. His research interests are mainly in the analysis of building frames and computer aided analysis and design of building frame structural components.

Gerhard T. Suter is Professor of Civil Engineering at Carleton University, Ottawa, Canada. His wide research interests include reinforced concrete, masonry as well as timber and these interests often are tied to his work as a structural engineer. The present paper is based on one of his recent consulting assignments.

Heinz Keller is a Research Engineer in the Department of Civil Engineering at Carleton University. His major research activities are in the area of engineered masonry design, masonry construction methods and performance evaluation of existing structures.

ACI Member, David C. Marett is chief structural engineer in the Transportation Department of the Regional Municipality of Ottawa-Carleton, Ontario, Canada. He has been involved in numerous bridge design, and evaluation projects. His current interests are in the area of bridge management systems.

INTRODUCTION

The theoretical basis for applying the finite element method in a structural investigation has been around for a long time. The availability of computers and appropriate software in the market has made it feasible for anyone, with a computing budget and a knowledge of finite element theory, to perform such an analysis. Since many papers have been published in this field, it is essential to preview without delay those aspects of the present work which are of special interest.

The computer program used is available in a published textbook (Ref. 1) and consists of less than 500 FORTRAN statements. It is inexpensive to run and produces excellent results in applications similar to the one described here. A brief explanation of the theory behind the simple finite element used in this paper is provided for the interested reader.

The pier investigation pertains to an orthogonally reinforced concrete deep beam structure which is approximately 30 feet (9 m) wide by 18 feet (5.5 m) deep, and is 2 feet (0.61 m) thick. The beam is supported on 3 caissons. A series of finite element stress analyses has been carried out to examine possible causes of existing cracks in the beam; to determine the caisson loads; and to investigate the effect on the beam of external posttensioning during repair work.

DESCRIPTION OF MANOTICK BRIDGE

The Manotick bridge, constructed in 1955, carries County Road No. 7 over the east channel of the Rideau Canal at Manotick, Ontario, Canada. The structure is a three-span continuous steel girder bridge with an 8 in. (20 cm) reinforced concrete deck. The two reinforced concrete piers are arranged symmetrically about the midspan of the bridge as shown in the general view of Fig. 1. Each pier is supported by three caissons topped with a reinforced concrete pier cap as indicated in Fig. 2. The caissons consist of rectangular concrete filled sheet piling.

In October 1978, a structural investigation of the piers was initiated because of the existence of pier cracking and the deterioration of the caisson pier cap system. Only the typical major pier cracking pattern is shown in Fig. 2. The major cracks, in general, did extend through the piers. The widest crack widths varied between 0.005 in. (0.13 mm) and 0.010 in. (0.25 mm). A severe case of concrete deterioration had taken place in the pier cap and caissons. Concrete had eroded in the pier cap in a pattern similar to that indicated in Fig. 2. The concrete in the piers, however, was not in an eroded state, and by visual inspection, it appeared to be generally of good quality, including that in the region where the cracking had taken place. In addition, at the time of this investigation the soundness of the caissons and the supporting bedrock were not known. The exact state of support of the pier at the time of pier cracking was also unknown.

This paper relates to the finite element stress analysis of the pier in its own plane. The key questions to be answered by the structural analysis were these:

(i) Could the pier cracking have been caused by the application of normal dead and live loading or else by an extra heavy live loading ?

(ii) What are the effects of different pier support conditions on pier stresses and possible pier cracking ?

(iii) Could the pier cracking be the result of the central caisson having settled and therefore being structurally ineffective ?

(iv) Do caisson forces vary substantially between the outer caissons and the central caisson, a situation that may lead to unequal settlements ?

(v) While remedial work would be carried out on the pier foundations, what magnitude of external posttensioning of the piers would be required to keep the piers from further cracking ?

FINITE ELEMENT ANALYSIS

Finite Element Method

The finite elements used in this study are triangles having 3 nodes each, a single node in each vertex. A node is a point of connection with adjacent elements. For the particular element used here, the in-plane x and y displacements are linear functions of the corresponding nodal displacements. The x displacement function u, for example, has the form

$$u = a + bx + cy$$

where a, b and c could be calculated for the element from a set of nodal values of the displacements. In finite element analysis, the problem is formulated so that the nodal displacements are calculated from a set of simultaneous equations.

Different linear displacement functions apply in each finite element. This means that mathematical 'corners' in the displacement functions occur across the boundaries. These 'corners' tend to smooth out as more elements are used in the finite element analysis.

Strains are derivatives of displacements, and for the linear displacement model used here, these strains are constants in each element. In the computer program which is listed and documented in Ref. 1, linear elastic behaviour is assumed.

Finite Element Model

The following points relate to reasons for selecting the finite element mesh shown in Fig. 3.

Only one-half of the symmetrical pier has been modelled. Nodes 1 through 10 are constrained for both symmetrical and antisymmetrical loading cases.

Most nodes are on a 2 foot (0.61 m) by 2 foot grid permitting most of this data to be produced automatically. Some deviations occur to allow nodes to be placed at load points.

The union jack pattern formed by a 4 foot by 4 foot square of elements has not been maintained throughout in order to keep triangle aspect ratios as large as possible.

A more detailed stress pattern immediately under load points and above points of support could have been achieved by concentrating elements in these areas, however, these were not regions of interest in this investigation.

The adequacy of the finite element grid refinement was also investigated, and this is reported in a later section of the paper.

The analysis pertains to a linear elastic homogeneous 2-dimensional solid. There is no attempt to model the cracking in the lower central region. Numerical analysis is still some years away from modelling, in a practical way; the crack interface transfer of force in reinforced concrete, i.e., the combination of friction, bearing and reinforcement forces at a crack. This does not, however, necessarily invalidate the use of linear elastic homogeneous models in this type of investigation.

The cracking in the pier is very limited, and unless the numerical analysis indicates a state of high tension stress, which tends to open the cracks, then no advantage is gained in using a cracked model. The uncracked analysis also demonstrates whether a static state of stress caused the existing cracking. This was one of the requirements of the investigation.

The orthogonal reinforcement, consisting of a single grid layer at each face of #5 horizontal bars and #7 vertical bars 12 inches on centres, is too light to be considered in the analysis and is not modelled here.

The concrete design cylinder strength, fc', of 4 ksi (27.5 MPa) was assumed together with a modulus of elasticity of 3620 ksi (25,000 MPa) and a Poisson's ratio of 0.15. The tensile strength, f_t', was taken as 0.41 ksi (2.8) MPa.

Three different support configurations were selected for study. In one, the stiffening effect of the pier cap is modelled by a fully supported base. In another, the caisson support is applied directly to the pier. In view of the nature and extent of pier cap deterioration, this may or may not be a more appropriate model. In the third model, the center caisson is not present.

<u>Loading Cases Investigated</u>

In the course of the structural investigation, the pier was analyzed for three different conditions of loading:

(i) The factored design loading consisting of five 252 kip (1120 kN) point loads were applied as shown in Fig. 2. Each of these loads represents a 228 kip dead load and a 24 kip live load. For the symmetrical dead and live loading, all three support conditions were investigated.

(ii) A heavy lane loading of 200 kips (890 kN) was investigated as shown in Fig. 4(a), as a feasible severe loading condition on the structure. The two 100 kip loads were placed directly on the finite element model as shown in Fig. 4(b). This approximation to the true loading introduces fictitious local distortions of the pier directly under the loads and therefore stresses are not correct in these areas. In order for the symmetrical finite element mesh to be usable for this case, symmetrical and antisymmetrical cases were superimposed as shown in Fig. 5.

(iii) The purpose of applying a posttensioning system to the pier is to reduce or eliminate horizontal tension in the pier and to strengthen the pier against the possibility of damage during subsequent remedial work on the caissons and foundation. The tendons are applied at top and bottom pier levels as shown in Fig. 6. Three point loads of 52.2 kips (232 kN) were applied top and bottom as shown in Fig. 7. Each set of 3 point loads is equivalent to 3 pairs of 5/8 in. tendons tensioned to 0.6 f_{pu}, and applied over a 2 foot depth. It was necessary to convert these to nodal load equivalents for analysis as shown in Fig.7.

FINITE ELEMENT RESULTS

Principal Stresses

The principal stresses at each element centroid were plotted on a calcomp plotter using a program written in an APL graphics language. Each principal stress was plotted as a scaled line, with appropriate orientation, and arrowhead turned inward for compression, and outward for tension.

The results for dead and live loading under three different support conditions are presented in Fig. 8. The pier outlines, load vectors and support symbols were added to the computer plots later by hand. Note that the highest compression stresses of 9.55 ksi (3.8 MPa) result with outer caisson support only in Fig. 8(a). Small horizontal tension stresses exist in the lower half of the pier with partial base support as shown in Figs. 8(a) and 8(b). With full base support in Fig. 8(c) these stresses are reduced to compression values. The highest tension stresses occur at the top of the pier between the applied loads. The tension stresses at the top are basically the same in each of the three cases and have a maximum value of 0.20 ksi (1.4 MPa).

In summary, the results for all three support conditions indicate that for 4 ksi (27.6 MPa) concrete all compressive stresses are localized. High tensile stresses are less than 50% of ultimate and do not occur in the cracked region. In the cracked region, in Fig. 8(a), the tensile stresses are a maximum of 0.09 ksi (0.62 kPa) at the centerline bottom and decrease to 0.03 ksi (0.21 MPa) in the next stack of elements. These values are not regional maxima since they occur not on the bottom edge, but rather at bottom element centroids. The actual tensile stresses in the bottom edge could be about 20% higher as indicated by a deep beam study reported later in this paper. Even with this 20% increase the maximum, highly localized, tension stresses in the cracked region is less than 30% of ultimate. All of these results indicate that for 4 ksi concrete, the given loading, and any of the feasible support conditions, pier cracking would not be expected to occur.

The principal stress results for the lane loading showed moderate compression and tension stresses. The tension stresses in the cracked region became smaller suggesting no pier cracking under a severe lane loading. Hand calculations were required for this loading, since the rectangular stresses from the two finite element models for the two support conditions shown in Fig. 5, had to first be combined. The stress results calculated in the cracked region were found by combining directly the results represented by Figs. 5(a) and (b) for right of centerline; and combining the results represented by Fig. 5(a) and the negative of Fig. 5(b) for left of centerline.

The principal stresses resulting from the application of the posttensioning system, in combination with dead and live loading and caisson support, are given in Fig. 9. The top and bottom tendon forces have eliminated the bottom tension stresses and reduced the top stresses.

The total experimentation with the finite element model is reported in detail in Ref. 2.

Caisson Forces

Bearing stresses at the caisson points of support are proportional to the caisson forces. By balancing these forces with the total gravity load, the caisson reactions were calculated. From the numerical finite element output, the results of these bearing force calculations are as follows:

Dead load plus live load-- Center caisson: pier bearing stress = 0.19 ksi (1.3 MPa); caisson reaction = 328 kips (1459 kN) Outer caisson: pier bearing stress = 0.27 ksi (1.9 MPa); caisson reaction = 466 kips (2073 kN)

Lane load plus dead load-- Center caisson; pier bearing stress = 0.21 ksi (1.4 MPa); caisson reaction = 299 kips (1330 kN) Outer caisson: pier bearing stress = 0.36 ksi (2.5 MPa); caisson reaction = 521 kips (2317 kN).

The difference in caisson forces clearly indicates the potential for differential settlements of the caissons.

VALIDATION OF RESULTS

A verification of the validity of the results of a finite element analysis is especially important when a problem type is solved for the first time using a particular element. A convergence study is carried out by subdividing the mesh, and monitoring the convergence of some aspect of the structural model's response, for example the strain energy content, the deflection at a point, and so on. Strain energy was not used as a basis for comparison here, because of the fact that the singularities at the points of support were not modelled. Since the areas of interest were somewhat remote from these points, centerline stress was considered an appropriate basis for convergence.

The verification of the model is provided by both a direct comparison with results published by Leonhardt (Ref. 3) for a similar structure; and also by mesh subdivision. The comparison example, having a shear span similar to that of the Manotick Bridge pier, is shown in Fig. 10.

A comparison of horizontal stresses at the centerline is given in Fig. 11. As expected, the Navier solution is not valid but the finite element analysis and the Leonhardt solution are in close agreement. Note that the finite element results are not taken from the exact centerline of the beam but from a stack of elements close to the centerline. These element centroids plot on a zig-zag line from the bottom to the top of the beam.

A comparison of results along the centerline is given in Table 1 for 2 successive mesh subdivisions and the results from Leonhardt (3). A further subdivision to 360 elements was attempted, but evidence of numerical roundoff errors appeared, and it was necessary, therefore, to maintain an upper limit on the number of elements. The finite element stresses for the 90 element mesh are considered to have converged and are expected to be slightly different from Leonhardt's because they are not exactly on the centerline. A comparison with results, such as those of reference 3, should be sought whenever possible, since these do provide a basic correct solution to the homogeneous linear elastic problem.

CONCLUSIONS

Based on the total results of this study, outlined in general in this paper, the following conclusions and recommendations are made:

(i) The results of the analyses for dead plus live load indicate that the pier would not crack under any of the three assumed support conditions. This statement holds even for the most critical situation where the pier cap and center caisson are structurally ineffective.

(ii) The application of a severe lane loading produced no major tensile stresses which could lead to pier cracking.

(iii) The structural analysis results show that substantially higher forces exist in the outer caissons which in turn could lead to significant differential settlements. The relatively rigid piers would be sensitive to even small differential foundation movements. In combination with vibration brought about by loose seating of the caissons, these factors could account for the present cracked state of the piers.

(iv) Posttensioning cables applied at the top and bottom of the pier are recommended as a safeguard against additional cracking before remedial work on the caisson and pier cap is implemented.

This paper has outlined a numerical investigation of a structural element for which a simple rational theory does not exist. The finite element analysis and the code design checks given in detail in Ref. 2 indicate that the pier was well designed. However, when problems occur in such a structure some of the more rugged procedures available for design are not suitable for investigation. It is fortunate that a simple finite element program is capable of producing cheap, good quality results. This procedure enabled a degree of experimentation with several loading cases and all feasible support conditions. It also made possible the accurate calculation of caisson forces and a study into the effect of posttensioning.

The finite element program used here, and several others having the same capabilities are readily available and can be run on personal computers with a FORTRAN compiler. This type of structural investigation is thus within the means of the smallest structural consulting offices.

REFERENCES

(1) Bowes, W. H. and Russell, L. T., Stress Analysis by the Finite Element Method for Practicing Engineers, Lexington Books, Massachusetts, 1975.

(2) Suter, G. T. and Hartley, G. A., "Structural Analysis of the Manotick East Channel Bridge Piers", a report submitted to the Transportation Department of the Regional Municipality of Ottawa-Carleton, February, 1979.

(3) Leonhardt, F. and Walther, R., Wandartige Traeger (wall-like elements), Deutscher Ausschuss fuer Stahlbeton, Nr. 178, 1966.

Table 1 Finite Element Convergence Study

	Photoelasticity*	180 elements	90 elements
Position of centerline neutral axis	0.28	0.27 h	0.27 h
σ_x value at bottom centerline	1.60	$1.56\ \frac{q}{b}$**	$1.55\ \frac{q}{b}$

* Leonhardt's results

** Uniform load intensity, q divided by out of plane beam width, b.

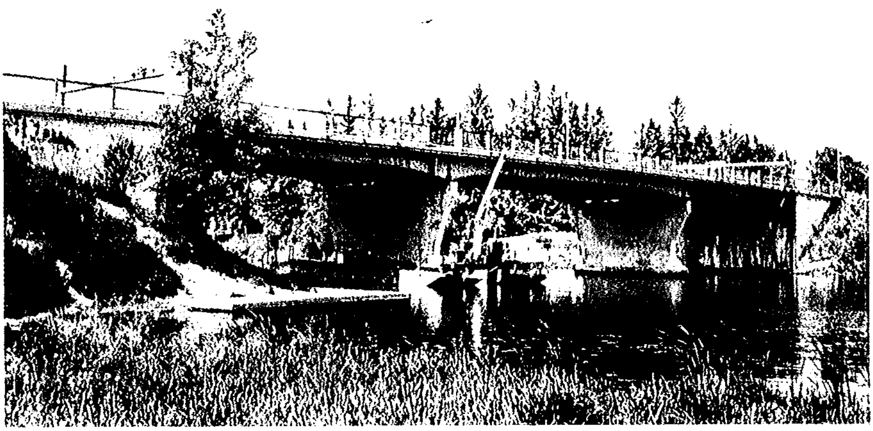

Fig. 1--Manotick bridge