Zeitschrift: IABSE reports of the working commissions = Rapports des commissions de travail AIPC = IVBH Berichte der Arbeitskommissionen

Band: 19 (1974)

Artikel: Development and application of advanced computer analyses in canadian concrete bridge design

Autor: McNeice, Gregory Malcolm

DOI: https://doi.org/10.5169/seals-17510

IABSE
AIPC
IVBH

SEMINAR on:
«CONCRETE STRUCTURES SUBJECTED TO TRIAXIAL STRESSES»
17th-19th MAY, 1974 - ISMES - BERGAMO (ITALY)

Development and application of advanced computer analyses in canadian concrete bridge design

Dévéloppement et application des méthodes avancées d'analyse de calcul pour le 'design' des ponts en béton au Canada

Entwicklung der theoretischen Analyse der endlichen Element und ihre Anwendung fuer den Entwurf von Betonbruecken in Kanada

GREGORY MALCOLM McNEICE
Associate Professor Solid Mechanics Division University of Waterloo
Waterloo, Ontario - Canada

1. INTRODUCTION

The ever-increasing demands by both the motoring public and the commercial transport community for increased traffic networks in Canada, have led recently to the development of bridge structures which no longer resemble conventional designs. In high density urban areas, especially in the capital cities, the spider-web of expressways, parkways and interchanges result in multi-level stacking of overpasses. These present particular problems since column spacing becomes irregular both in the longitudinal and transverse directions. In many cases this has led to single columns placed on the center line of multiple span curved decks. In the case of multi-level overpasses, depth of the superstructure is an important consideration. Although box girder design will allow longer spans and therefore less support congestion, it also leads to higher overall elevation of approach embankments which leads to further cost and other considerations. The increased vehicle speeds lead to more stringent requirements on geometrical layout with respect to both planar curvature and superelevation. User acceptance of these structures is directly dependent on the quality of ride, the ease with which one can enter and exit as well as the aesthetic characteristics of the structure when viewed from under, over or some distance away. The present awareness of our environment and the public demand for government preservation of it, has undoubtedly reached the eyes and ears of the designer. He no longer can view his structure primarily on the basis of strength and serviceability. Improved quality of ride especially at high speeds requires continuous span structures in order to avoid construction and expansion joint disturbances which can be quite annoying to those thousands of motorists who commute daily by automobile.

Although there are many other factors, in addition to the above that influence the choice of bridge structure, it is a fact that new and improved designs are required to meet the demands of the Canadian people. These designs usually involved irregular geometry, very long spans and irregularly spaced isolated supports. In addition to the long slender structures, there are increasing demands for exceptionally wide shorter span slabs with skewed end supports and skewed support lines with isolated columns. In many cases the width of the slab exceeds the span. Comparing slab type designs with conventional girder design leads one to question the validity of using approximate analysis techniques which are based on simplified behaviour. The tendency of many bridge designers who have wide experience in conventional girder slab design is to extend the simplified concepts of beam theory to structures that undoubtedly behave as plates. This has led to some embarrassment on occasions when the resulting structure cracked extensively, deflected beyond acceptable amounts and shifted off its supports. Not only is there a need for better understanding of mechanical behaviour but the effects of creep, shrinkage and differential temperature are very important items in the design of continuous slab type bridges.

In Canada, as in many other countries, academics who research methods of structural analysis and design are usually well versed in the theoretical aspects of two and three dimensional solutions of various boundary value problems. The same is not true for chief designers nor those directly under them. Consequently, it is not unreasonable to find some engineers applying inadequate theories to certain aspects of bridge design. This paper presents a report on what academics across the country have been doing to supply the designer with more appropriate tools for cases in which he must use them and to indicate implementation of various developments to the design office.

In collecting the data for this paper I was fortunate to meet with some of the researchers, both in the western and eastern regions of the country. Since, to the best of my knowledge, the most significant work has come from the University of Calgary in Western Canada, McGill University in Eastern Canada and the University of Waterloo in Central Canada, I have chosen to report the efforts in a similar regional order. These centers that have been mentioned are not the only places where bridge research is continuing, but rather, they are the establishments at which the advanced stress analysis systems are being developed. For purposes of reporting the regions are illustrated in Figure 1.

2.0 WESTERN REGION

The major contribution from western Canada was kindled by Professor Y.K. Cheung who co-authored the first textbook on the Finite Element method while lecturing at the University of Swansea in Great Britain. Upon his arrival at the University of Calgary, he set out to extend his Finite Strip approach to the analysis of slabs [1]. A specific application to bridges was made early in 1969 [2]. The finite strip approach has been the central theme of most of the computer analyses development at Calgary between 1969 and 1972. A brief outline of the method follows, after which a number of applications to bridges will be summarized.

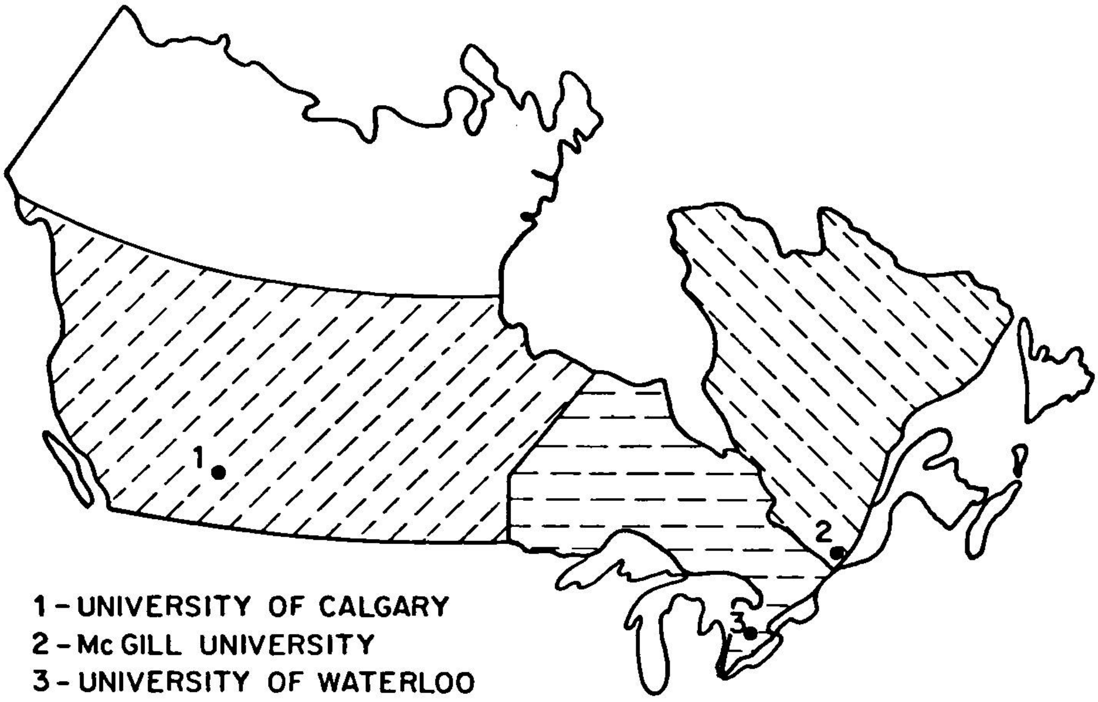


Figure 1 - Research Centers in Regional Subdivisions

2.1 Finite Strip Approach to Finite Elements

In the finite strip approach, the structure is considered to be divided into continuous strips in one direction. Each of these strips is given some approximate response function. Next the force-displacement or stiffness properties are derived and the strips are connected together such as to satisfy equilibrium and compatibility to a certain degree. To illustrate more clearly, consider the two cell box girder in Figure 2.

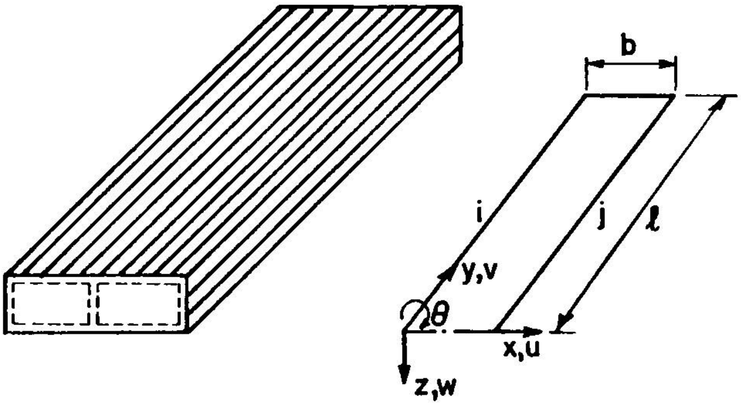


Figure 2 - Typical Finite Strips for Box Girder

The box girder, simply supported, is assumed to be divided into a number of strips in the longitudinal direction in both the webs and flanges. The properties of each strip are considered constant over that strip. Thickness variation can be approximated by using a number of constant thickness strips. Each strip will contain both in-plane or membrane stresses and bending stresses. Since small deflection theory is assumed, these two basic types of stress are considered to be uncoupled, allowing strip behaviour for membrane and bending to be dealt with separately.

The stiffness characteristics for in-plane response are derived by first selecting the end conditions for the strips. For example, the displacement and stress conditions at the ends of the strip can be selected as

$$u = 0 \qquad \sigma_y = 0 \tag{1}$$

for $y = 0$ and $y = \ell$. At $x = 0$ and $x = b$, the nodal line displacements are respectively,

$$u = u_i, \quad v = v_i$$

$$\text{and} \quad u = u_j, \quad v = v_j \tag{2}$$

The strip displacement functions which will satisfy these conditions can be given by a Fourier series such as,

$$u = \sum_{m=1,2,\ldots r} \left[(1 - \frac{x}{b})u_{im} + (\frac{x}{b})u_{jm}\right] \sin \frac{m\pi}{\ell} y$$

$$v = \sum_{m=1,2,\ldots r} \left[(1 - \frac{x}{b})v_{im} + (\frac{x}{b})v_{jm}\right] \cos \frac{m\pi}{\ell} y \tag{3}$$

In Equations 3, u_{im} and v_{im} are displacement parameters of the m^{th} term at the nodal lines. Proceeding with the two dimensional case and using the virtual work principle, one can derive the strip stiffness matrix. A full account of this derivation is given in reference [2].

For the case of strip bending behaviour, the strip end conditions, if simply supported are

$$w = 0 \qquad M_y = 0 \tag{4}$$

for $y = 0$ and $y = \ell$. The displacement functions which satisfy these conditions are

$$w = \sum_{m=1,2,\ldots r} \left[(1 - \frac{3x^2}{b^2} + \frac{2x^3}{b^3})w_{im} + (x - \frac{2x^2}{b} + \frac{x^3}{b^2})\theta_{im}\right.$$

$$\left. + (\frac{3x^2}{b^2} - \frac{2x^3}{b^3})w_{jm} + (\frac{x^3}{b^2} - \frac{x^2}{b})\theta_{jm}\right] \sin \frac{m\pi}{\ell} y \tag{5}$$

These functions can be used to establish the bending stiffness matrix for the strip. The applied loads on each strip are also resolved in terms of Fourier series. The structure is assembled by specifying common nodal line displacements between strips for proper compatability in any one plane. Also, equilibrium of nodal line forces and any applied loads must be established. Since the end conditions are automatically satisfied at the outset, the final set of assembled force-displacement equations can be solved to determine the nodal line displacements. Implied herein, of course, is the fact that a definite number of series terms (m = r) have been used. The more terms, the better will be the approximation. Once nodal line displacements are known, calculation of membrane stresses and internal bending moments is a straightforward procedure. One disadvantage of the strip model as outlined in reference [2] is the fact that vertical and horizontal strips are not fully compatible with respect to rotations along their intersection. One of the major advantages of the finite strip approach is that analyses can be performed on medium-sized computers quite readily. For application to continuous span structures, some difficulty was encountered when longitudinal strips were used. Consequently, for continuous spans, transverse strips were chosen. An example of this can be seen in reference [3] in which Cheung, Cheung and Reddy carried out frequency analysis of continuous span bridges. One example was a two span isotropic bridge of constant thickness shown in Figure 3. Figure 4 illustrates the thickness idealization that was used on a second example containing a haunch over the interior support. Comparison of frequencies and various mode shapes with known analytical results illustrated the good accuracy provided by the strip approach [3]. In addition to Professor Cheung's involvement in the development of the finite strip method, Professor Ghali became interested in the method since he had carried out a number of physical model studies of bridges and wished to compare these results with analytical values. After having co-authored a paper involving continuous span finite strip analysis [4], Ghali eventually changed the approach to continuous span analysis. In reference [4] superposition of two simply supported cases was used to model continuity of the structure over the interior support. Ghali's approach [5] was to first analyze each span as simply supported, determine the rotation and then superpose couples to produce the necessary compatibility. Of major importance in reference [5] was the introduction of the trigonometric basic functions into the transverse deflection approximation. That is,

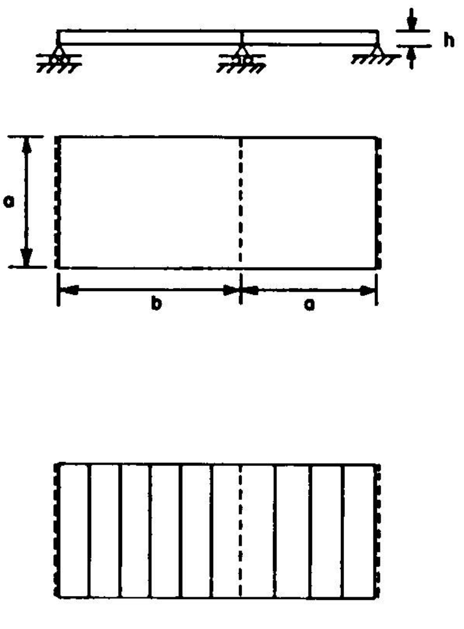


Figure 3 - Transverse Strips for Continuous Span Analysis

$$w = \sum_{m=1}^{r} f(x)\, Y_m \qquad (6)$$

in which $f(x)$ is a polynomial and Y_m the basic function required to satisfy the strip end conditions in the y direction (see Figure 5).

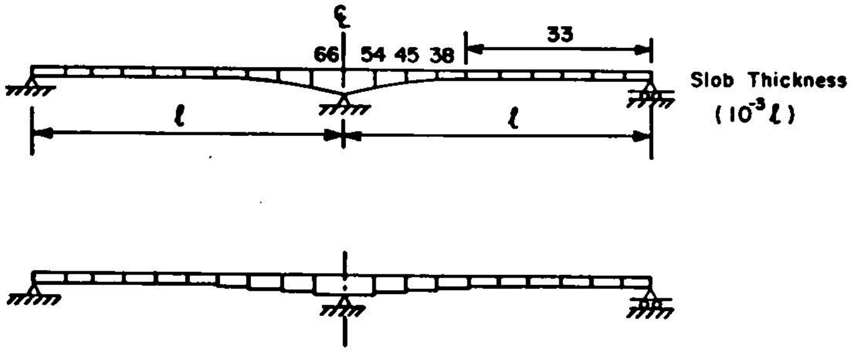


Figure 4 - Actual and Idealized Thickness Variation

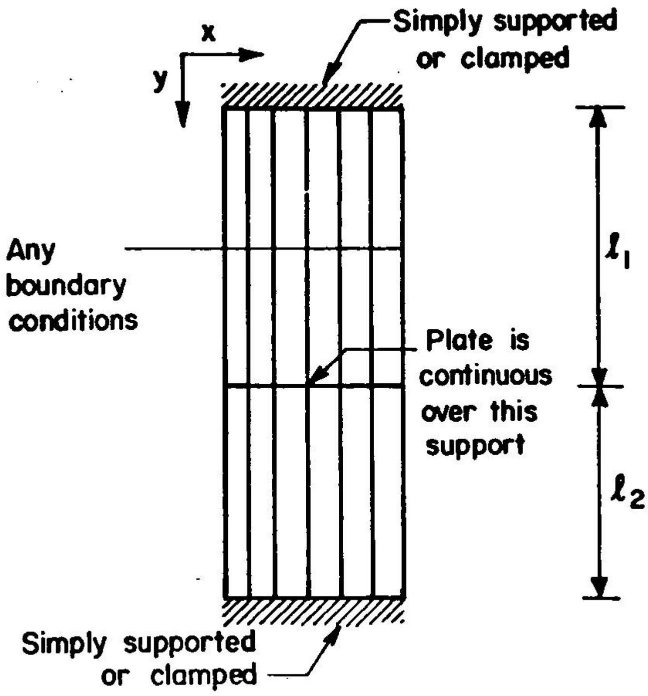


Figure 5 - Longitudinal Strips for Continuous Span Analysis

The orthogonal properties of the trigonometric basic functions allows the total number of equations to be uncoupled in such a manner that a number of smaller sets of equations can be solved. This is true with these basic functions for both simple supports and clamped conditions. The major disadvantage found in the work of reference [5] was that the connecting moment method of superposition required many terms in the series in order to obtain good accuracy. In an attempt to reduce the computational costs due to a high number of terms, an attempt was made to develop an extrapolation technique in order to predict results on the basis of those obtained using only a few terms. As outlined in reference [6], the finite strip approach using basic functions that have orthogonal properties consists basically of adding weighted solutions, one for each term in the selected trigonometric series. This evolves from the fact that the loading on the strip must be approximated also by a series approach. The solutions obtained from the uncoupled sets of equations are directly dependent on the type of load and the functions chosen to approximate them. From the contributions so established, the deflection at any one point can be plotted versus the number of terms. If a polynomial curve is passed through these values, it can be used to extrapolate results beyond that obtained using the terms chosen. This procedure certainly increases the possibilities of the finite strip approach.

Since many box girder bridges have skewed supports, Brown and Ghali [7] developed parallelogram strips having a fifth order polynomial in the transverse direction. The results reported indicate that good accuracy is obtained by using a small number of equations. However, only solid skew slabs were analyzed and no box girders were attempted. Although this work and the previously outlined finite strip developments have clearly defined advantages for certain types of plate analyses, the method is not convenient for irregular bridges. That is, in cases where diaphragms are used, or in which solid sections exist over support regions and in end anchorage zones. Also, there does not appear to have been any attempts to analyze slabs with isolated column supports, a common case in modern day bridge complexes.

2.2 Special Parallelogram and Quadrilateral Elements

Realizing the shortcomings of the finite strip method for general purpose bridge analysis, Sisodiya, Cheung and Ghali [8] developed a parallelogram finite element especially designed to model bending deformation in the webs of box girders. A second element used for the deck membrane response is a quadrilateral element and is presented in the same paper. In general, the element uses curvilinear coordinates, similar to the linear isoparametric elements [9]. However, for membrane behaviour, three degrees of displacement freedom were developed. That is, the in-plane displacements of u and v as well as a rotation θ_z in the plane of the element. This rotation is of fundamental importance in providing rotation compatibility at nodes that are common to two elements which are inclined to one another. These freedoms combined with the transverse displacement w and the bending rotations θ_x and θ_y constitute the required minimum number of degrees of freedom for point continuity. For bending response, the authors selected the Dawe element for use with the parallelogram element and two of the Bazeley et al. triangular elements for the quadrilateral element [9].

The parallelogram element is illustrated in Figure 6. The displacement v is assumed to vary as a cubic in ξ and linear in η.

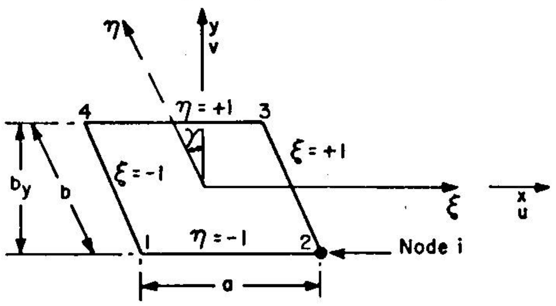
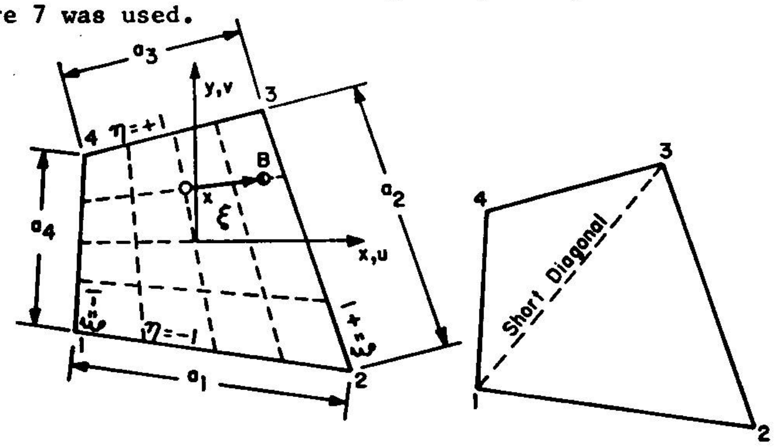


Figure 6 - Special Parallelogram Finite Element

The rotation normal to the element plane is simply $\theta_z = \partial v/\partial x$. The resulting element response is obviously biased for direct application to web deformation. The cubic response in v allows the bending deformation to take place in the plane of the web perpendicular to the ξ axis. However, v need only be linear in the vertical direction. The displacement u is quadratic in both ξ and η allowing a linear distribution of bending stress in the plane of the web as would be required.

For the deck of the box girder, the quadrilateral element of Figure 7 was used.



(a) Coordinates in the element

(b) Plate bending from two triangles

Figure 7 - Special Quadrilateral Finite Elements

For this element, u was chosen as a linear function of ξ and η with v cubic in both. This allows the element to be used as a web element if a variable thickness box girder is analyzed. A number of tests were performed using both elements, the results of which indicated that both elements model standard beam type problems very well. For skewed beam dimensions, the parallelogram element was clearly superior. The quadrilateral element had to be used in cases of haunched beams, of course. In these cases the results seemed quite good, sufficient for normal engineering practice. Application of these elements was made to the two span skewed box girder in Figure 8. Additional applications were also presented in reference [8].

A further application to single and double cell skewed box girders with diaphragms is presented in reference [10]. The results of that study showed that end diaphragms in skewed bridges produced undesirable effects on reaction distributions. It was also concluded that transverse stiffeners have little use and in some cases have a harmful effect. An interesting application of these special elements was also made to a skew vault bridge [11]. There, excellent results were produced when compared to other type of finite element models.

It should be stated here, as did the authors in reference [8], that these elements do not provide compatibility of displacements along common boundaries of all elements. This is since the x axis of two adjacent elements could have different directions in a mesh of quadrilaterals and the displacement functions for u and v are unsymmetrical with respect to the co-ordinates of each element. In addition, when parallelogram web elements are connected to quadrilateral deck elements, the bending and in-plane rotations do not conform to the same displacement functions and this leads to incompatibilities. Although the authors state that these

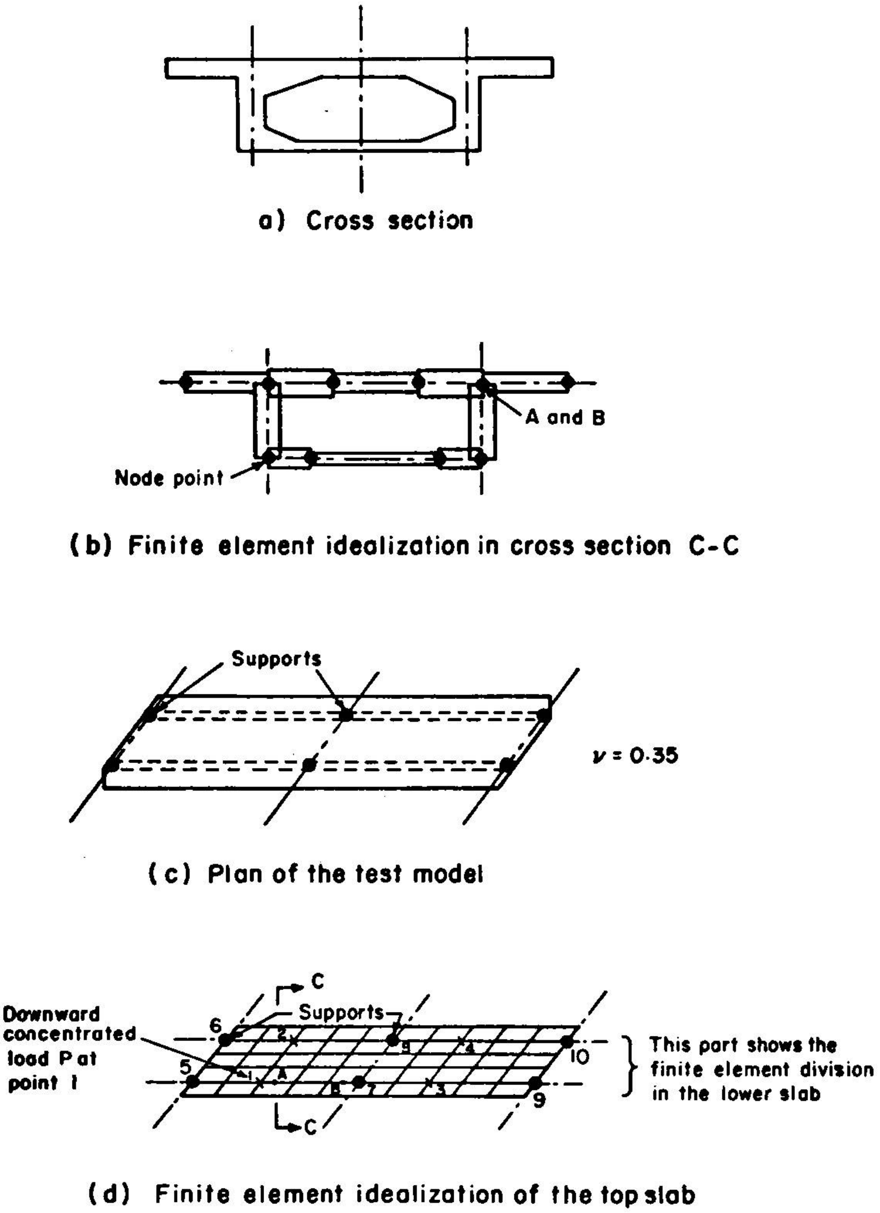


Figure 8 - Straight Skew Box Girder Model

effects are negligible, they lead to the requirement that an increased number of elements must be used for any one problem as compared to an analysis using fully compatible elements. Furthermore, if the designer wishes to view stresses in the regions of intersection between webs and slab or in the vicinity of the end anchorage zones, it is unlikely that much importance could be attached to these stresses. However, for general overall response, it is clear that these models provide substantially more information than simplified beam theory approaches.

To summarize the contribution from western Canada, perhaps it is fair to say that the work done at the University of Calgary under the direction of Professors Cheung and Ghali constitutes the major effort directed toward bridge analysis in western Canada using modern methods of structural analysis. To the best of my knowledge there has been no significant application of this work to bridge designs produced by western provincial governments. Researchers at Calgary have provided results for individual consulting firms on certain bridge structures in the west. Although there is evidence of an increasing awareness of the finite element method in design offices all across the country, there is no clear effort being made by western provincial ministries to co-ordinate analytical capabilities or provide any means of using existing knowledge.

I should add here that after having completed his post-doctoral studies at Calgary, Sisodiya was given an assistant professorship at McGill University. This leads me to report on the contribution from eastern Canada.

3.0 EASTERN REGION

Evidence of early application of the finite element method to metal box girder bridges in Canada exists in reference [12], in which Mehrotra et al. at McGill University used triangular finite elements with linear membrane response and a modification of Tocher's bending element. The elements were used to model webs and flanges in a fairly straight forward manner. The major drawback of the model was the fact that compatibility of displacement was not maintained, requiring many elements for accurate results. More recent results will form the main discussion to follow.

3.1 Circular Curved Box Bridges

A new in-plane annular finite element has been developed by Fam and Turkstra [13]. The element is illustrated in Figure 9. A polar co-ordinate system is used with each element having four corner nodes, each with four degrees of freedom u, v, $\partial u/\partial\theta$ and $\partial v/\partial\theta$. The displacement functions are cubic in θ and linear in r. The generation of the element membrane stiffness matrix is straightforward and is given explicitly in [13]. Comparison of results for simple structures having series solutions showed that the annular element gave accurate results and could therefore be used with confidence. Practical application consisted of the analysis of the horizontally curved box girder bridge illustrated in Figure 10.

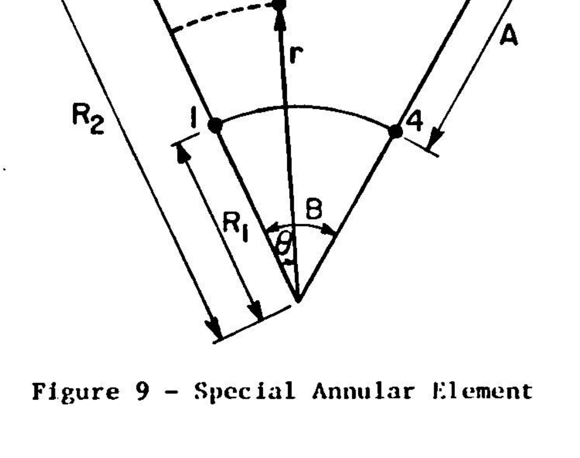


Figure 9 - Special Annular Element

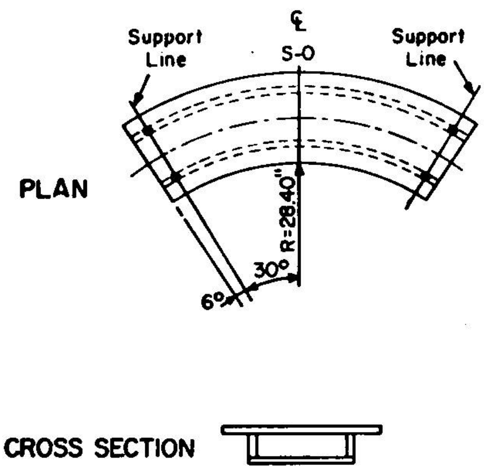


Figure 10 - Geometry of Box Girder Model

The in-plane behaviour of the top and bottom flanges were modelled using the above annular element. The bending behaviour of the flange elements was modelled using the Olson-Linberg annular bending element [14]. A new cylindrical element was used for the web regions. This latter element will be published in the near future. The particular bridge in Figure 10 was a plexiglass model which was tested and reported on by Aneja and Roll [15]. The superior results of the annular elements are clear from all the comparisons made between experimental, annular and regular flat type element results. One example is shown in Figure 11.

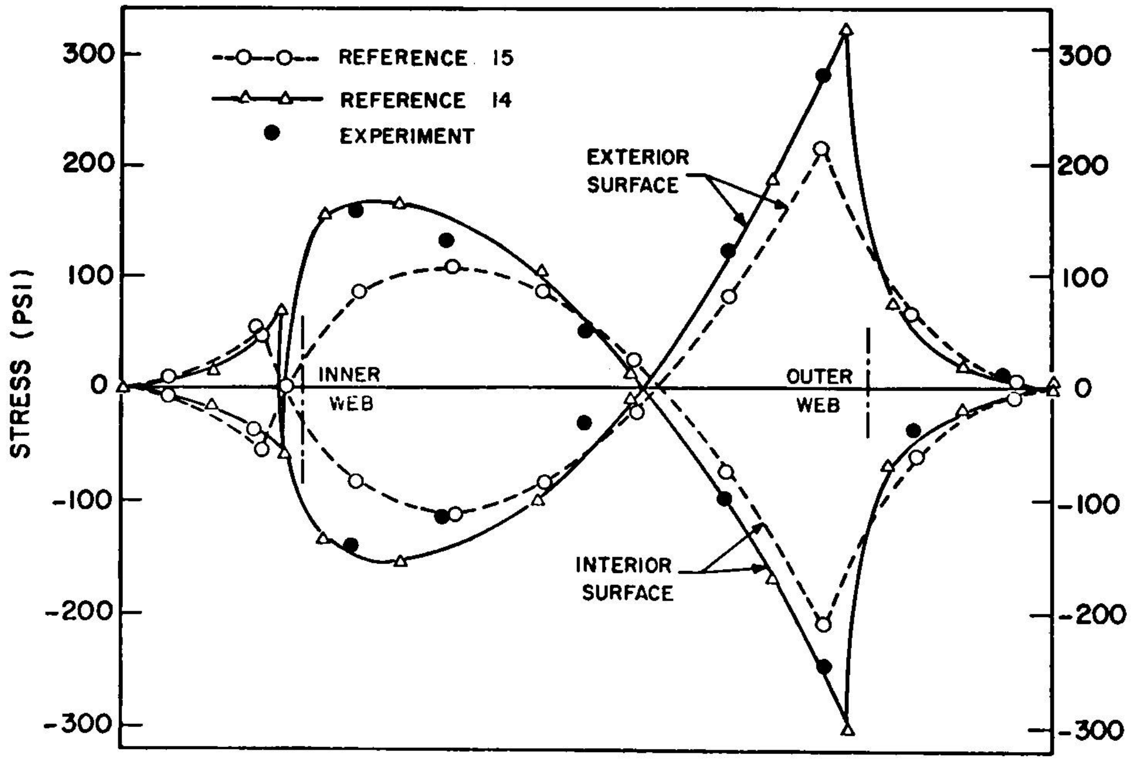


Figure 11 - Radial Stresses in Top Flange

These results indicate the importance of having the proper mathematical model for different types of box girder bridges.

3.2 Special Membrane Elements for Box Girder Bridges

Another approach to multi-cell box girder analysis was presented by Gurevich and Redwood [16]. In this analysis only membrane behaviour was used. In order to approximate the bending effects, fictitious transverse diaphragms were introduced. Therefore, the bending stresses in the model were assumed to be only membrane stresses in the top and bottom flanges of the boxes. No bending stresses were allowed through the thickness of any flange or web element. Although approximate, this approach can produce good results. For example, in cases in which all members, especially flanges, are thin and do not possess much bending strength, this model will perform adequately. However, for cases in which the flange thickness to span ratio approaches a medium thick plate, the flange bending stresses can be relatively high and this model will not likely provide adequate approximation. The authors used their model to analyze a five cell, 60° skew box girder which had previously been analyzed by usual finite elements [17]. The results are shown in Figure 12 and they are encouraging. Some of the elements developed for use on the webs are shown in Figure 13. The major drawback of a purely membrane type model is the complete lack of rotational compatibility, even at the nodes. The influence of such discontinuity between members can lead to suspicion of stress results at these joints. Also, fairly narrow cells must be available in order to allow the fictitious diaphragms to approximate the bending modes in the transverse span directions.

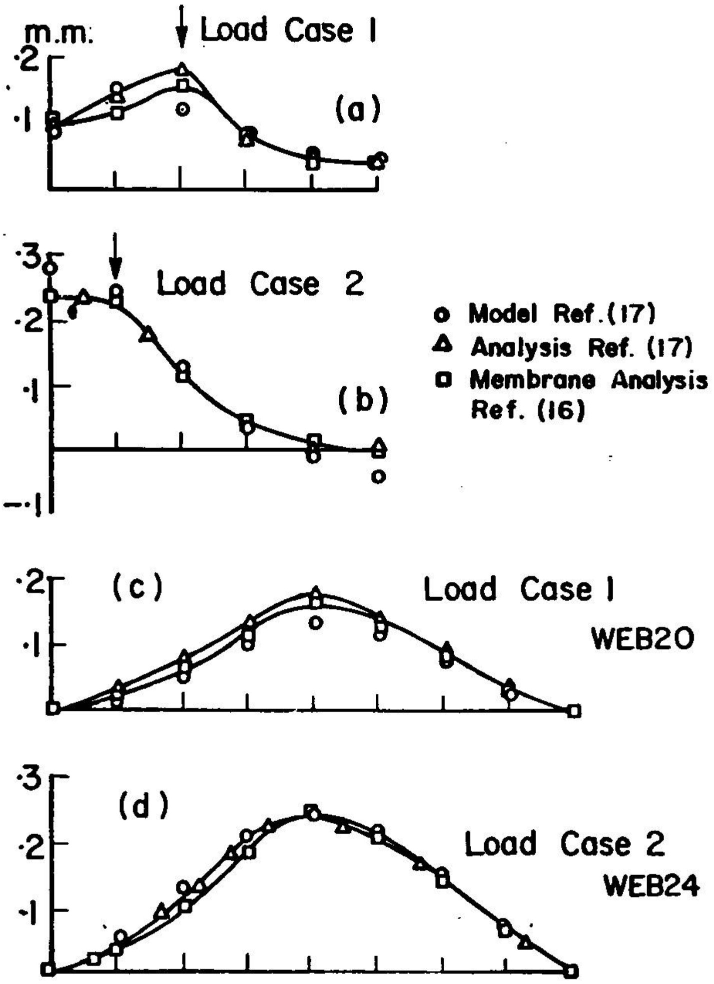


Figure 12 - (a) & (b) Centerline Deflection Point Loads on Ribs 2 & 3
(c) & (d) Loaded Rib Deflections

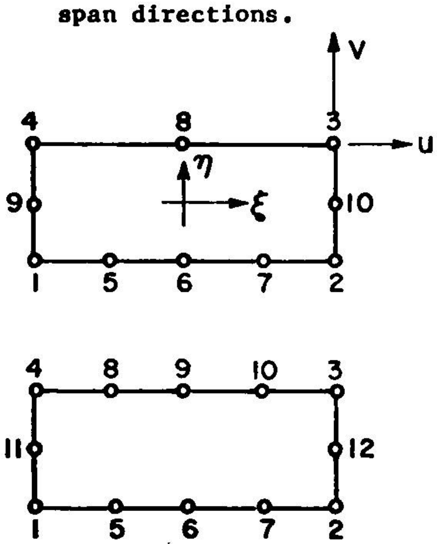


Figure 13 - Special Isoparametric Web Elements

Recently, Sisodiya at McGill University reviewed the state-of-the-art with respect to the analysis of box girder bridges [18]. Included in this review is the work of Scordelis at the University of California, U.S.A. and that by Cope at the University of Liverpool, England. The major point to be made is that most attempts have involved only 5 degrees of freedom at common nodes. This leads immediately to rotational incompatibility and for practical cases of stress analysis for which the designer does not have textbook solutions nor experiment to check the results from his computer, the effect of these discontinuities on the stresses will be impossible to assess. Indeed, the analyst cannot even perform a three solution convergence study of energy in order to assess the approximations. If, however, the 6 degrees of freedom (3 translations and 3 rotations) are used and the connecting elements are properly formed, certain types of box girder bridges will be fully compatible and the necessary numerical checks can be made. The models developed by Sisodiya include the 6 required degrees of freedom. Although computer costs are higher with use of more degrees of freedom, these costs are trivial compared to that taken to prepare the geometrical input data. Also, as computers and computer hardware systems become more efficient, these disadvantages become less important.

Summarizing the efforts from eastern Canada, there appears to be less direct bridge research being done than in the west. However, since Professor Sisodiya is now at McGill with the desire to further develop his work, I am sure that more developments will be coming from the eastern region.

Although I have devoted the discussion thus far to the analysis of box girder bridges, I should like to make note here of the substantial efforts made in the finite element field by the group at the National Research Council. In particular, the work done by Kosko, Cowper, Lindberg and Olson. Their research on higher order elements for shell, plate and membrane analyses has been given significant reference internationally. I should also like to mention the research by Professor Reddy of Memorial University. He has carried out analysis of various plates subjected to random excitations using the finite strip approach [19]. Once again, and to the best of my knowledge, there has not been any direct interaction between government design offices and researchers in eastern Canada; that is, with respect to coordination of efforts toward bridge research and design. I am aware, however, of such involvement in central Canada and it is to this that I address the remaining portion of this report.

4.0 CENTRAL CANADA

In the past, most Ontario highway bridges were designed such that the primary structural members were griders of either steel or prestressed concrete. For these structures, the well-known theories of beam bending and combined axial and flexural response were quite adequate in predicting prototype behaviour. However, when bridge slabs are used as the primary structural member, care must be exercised in using beam theory to check their design. The reason is that in many cases, such as in skewed slabs or in the vicinity or isolated supports, the longitudinal and/or transverse stresses are not the critical stresses. Instead, due to shear stresses the principal stresses far exceed the normal stresses.

In Ontario, the latest bridge design consists of voided prestressed slabs with multiple spans and in many cases, irregularly spaced single columns. For these slabs, beam theory provides a good estimate of longitudinal stresses but is inadequate in predicting transverse stress distributions which contain stress concentrations over the void regions. Also in the end anchorage zones, the stresses are very complex and of a three dimensional nature. Consequently, more refined methods of stress analysis are required for bridge slabs that behave as plates rather than as beams.

Recognizing the need for additional information concerning the behaviour of these structures, the Ministry of Transportation and Communications of Ontario, through the Structural Research Branch, undertook an extensive programme of study. It included analytical, numerical, physical and on-site investigations.

The particular bridge design currently used in Ontario contains voids with diameters as large as 80 percent of the depth of the slab. The effects of these voided regions, especially in a transverse direction are a major concern to the designer. The real structure is truly three dimensional in nature, since the concentration of stresses could occur over the voids along with a non-linear stress distribution across sections between voids. Since a full three-dimensional analysis would be too costly during the preliminary stages of design, a two dimensional model was chosen in order to assess its capability prior to any three dimensional analysis.

I will report firstly on the computer stress analysis system which uses two dimensional finite element uncoupled bending and in-plane stress analysis to analyze these bridges. This will be followed by a report on the three dimensional system currently being developed.

The programmes that make up the two dimensional Finite Element System are described briefly in the following. The system allows the engineer to perform the following types of stress analyses:

(a) Plane Stress
(b) Plane Strain
(c) Combined Bending and In-plane

4.1 Description of 2-D Finite Element Stress Analysis System

The two dimensional stress analysis system that was developed at the University of Waterloo is based on the displacement model of the finite element method and has been developed especially for the analysis of voided prestressed bridge slabs with typical cross-section as illustrated in Figure 14. However, the two main subsystems were constructed to work independently in order to provide additional stress analysis capability. The basic system flow chart is illustrated in Figure 15.

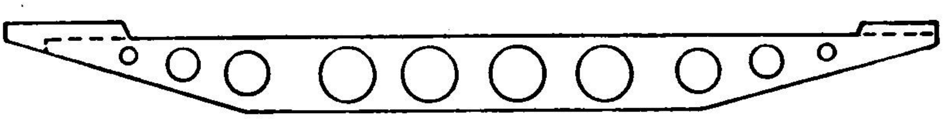


Figure 14 - Typical Cross Section with Circular Voids

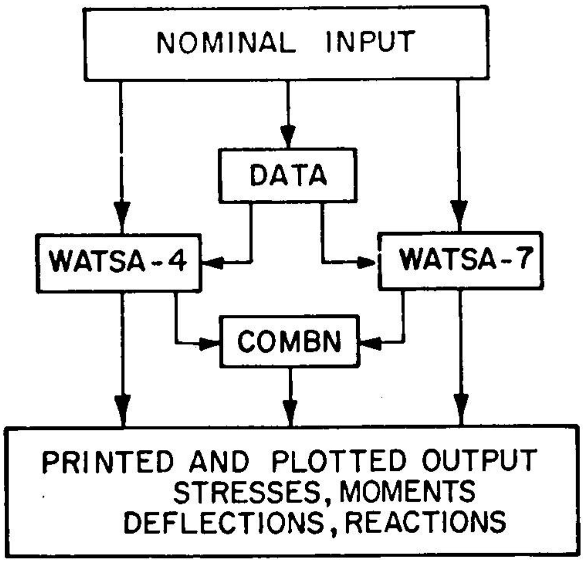


Figure 15 - Flow Chart of M.T.C. 2-D Finite Element System

<u>Programme WATSA-4</u>

This programme performs finite element plate bending analyses based on thin plate small deflection theory. The Clough-Felippa quadrilateral bending element [20] is used throughout. Material properties in each element can be orthotropic and with different orientations in each element. Individual element thicknesses can vary linearly over the element. Automatic element mesh generation is optional and the programme sorts identical elements such that unnecessary calculations are prevented.

The programme has a four level overlay structure requiring approximately 220K bytes core storage, 10 scratch files and allows the use of 1000 nodes and 900 elements. Plotting of element mesh as well as principal stresses are optionally available.

<u>Programme WATSA-7</u>

This programme performs finite element plane stress and plane strain analyses. There are two elements available, namely a quadrilateral isoparametric element [21] with or without a centroidal node. Material properties in each element can be orthotropic and with different orientations in each element. For the plane stress case, element thickness can vary linearly over the element. Automatic element mesh generation is optional and the programme sorts identical elements such that unnecessary calculations are prevented.

The programme has a three level overlay structure requiring approximately 220K bytes core storage, 9 scratch files and allows the use of 1000 nodes and 900 elements. Plotting of element mesh as well as principal stresses are optionally available.

Programme DATA

This programme creates the necessary input data required by WATSA-4 and/or WATSA-7 for typical voided prestressed slabs. The bridge is laid out in the usual manner by using a control line and a provincial coordinate system. Various measurements such as curb location, void positions, type and size of void, support region and skew angle as well as location of column supports must be provided as input to the programme. The user specifies the density of elements both longitudinally and in the transverse direction. The programme divides the slab into appropriate elements in the support regions as well as over the spans. These regions are matched, the supports are located, the moments of inertia and necessary cross-sectional areas are automatically calculated as input to the analysis programmes. In addition, a plot of the entire plan view of the bridge is provided in order to check on element subdivisions, etc. prior to analysis. This programme also orders and sorts elements into groups with similar geometry and material properties. It also calculates all dead load forces and prepares the loading to simulate prestressed effects for separate input to WATSA-4 and WATSA-7. The programme allows up to 8 spans and assumes that the cross section selected is constant throughout the length of the bridge. Both circular and rectangular voids can be positioned anywhere in the cross-sections.

Programme COMBN

This programme simply combines the stress and displacement results as produced from WATSA-4 and WATSA-7 and provides plots of top and bottom surface principal stresses. It allows any factored combination (up to 10 selections) of results from individual load cases, either from WATSA-4 and/or WATSA-7. The programme requires two direct access files and five scratch files.

Each of the above programmes performs a separate task. As far as voided prestressed bridge slabs are concerned, only DATA pertains exclusively to these structures. Each of WATSA-4 and WATSA-7 can be used entirely separate from all the other programmes. Indeed, WATSA-7 can be used effectively for studying transverse behaviour by analysing a cross-sectional portion of the slab. It can also be used to study other structures ranging from steel gusset plates to underground concrete culverts. WATSA-4 can be used for analysis of any irregularly shaped slabs, manhole covers, concrete pavement slabs, etc.

Printed output can be selected from one or more of the following: displacements, bending and twisting moments, principal moments, normal and shear stresses, principal stresses. Stresses from various load cases can be combined. Plotted output includes the element mesh layout generated by the data preparation programme as well as plots of principal stresses on top and bottom surfaces of the bridge.

4.2 Example of Bridge Analysis

A series of bridge analyses has been reported elsewhere [22,23] as well as comparisons with experimental results [24]. In order to illustrate the type of visual output possible from the Finite Element System, the five span bridge of Figure 16 is presented herein. The particular loading case was that of dead load. The principal stresses at the centroid of each element are presented such that the largest stress is directed in its proper orientation. The smaller stress is written beneath the largest. A glance at the stress plot gives the designer an idea of the distribution of stress around column heads and in the vicinity of cable anchorage zones.

The bridge in Figure 16 was fully subdivided automatically using DATA. Additional stress plots for each live load case, as well as for both top and bottom surfaces of the deck were produced during the one pass through the computer.

4.3 Applications to Various Problems

Gusset Plate Analysis (Plane Stress Analysis)

In plane stress analyses, stresses normal to the mid-plane of the plate are assumed to be zero while normal strain still occurs due to the Poisson effect. As an example of this type of analysis, the automatic mesh generation in WATSA-7 was used to produce the element mesh for the gusset plate in Figure 17. Nodes that were closest to the rivet holes were then selected as loaded points. Plotted output of principal stresses was selected for quick evaluation of the stress field.

Transverse Section Analysis (Plane Strain Analysis)

Plane strain analysis is used for structures that are of infinite extent in the direction perpendicular to its cross-section. Such is the case for a typical cross-section of a voided bridge slab. Figure 18 illustrates both a circular voided slab and a thick web box girder type. The element meshes seen here were custom made for these analyses using an electronic digital table. For complete manual preparation, they can be drawn to scale on graph paper and then the coordinates determined. It should be mentioned that this type of stress analysis performed on a section of the bridge gives a much better idea of the transverse behaviour of a slab design than by simply using a cantilever beam strip. Beam theory gives an erroneous distribution of stress in the vicinity of the void.

Effects of Shrinkage around Circular Voids

In order to assess the effects of shrinkage on stresses in the vicinity of the voids, a series of finite element analyses was carried out on a typical voided region. Shrinkage was approximated by introducing a temperature drop throughout the concrete. The restraint provided by the corrugated steel liner leads to surface tensile stresses which are very sensitive to the ratio of void diameter to slab thickness. Results of the analyses are summarized in Figure 19. As an example, a d/H ratio of 0.8 will give over 300 psi surface tensile stress over the void if a shrinkage strain of about 400 microstrain is selected.

These applications illustrate the varied use of the 2-D system and the analytical capability that now exists with the Ministry. Recently, the programmes have been used to study not only multi-lane skewed prestressed bridges, gusset plates and associated members, but also buried culverts having very large spans. The engineers in the Bridge Office are finding the system quite helpful in locating the principal stress regions of irregular members and wide slab bridges.

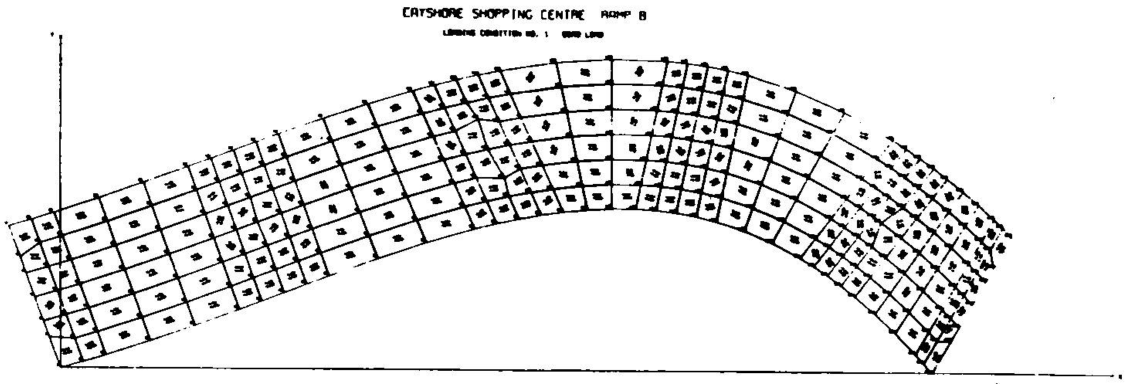


Figure 16 - Principal Stress Plot for a Curved Voided Bridge using WATSA-4

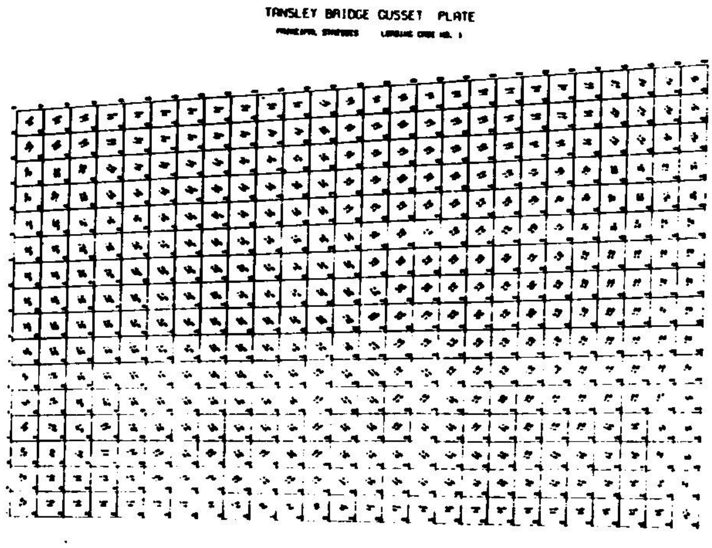


Figure 17 - Principal Stress Plot for Gusset Plate Analysis Using WATSA-7 (Plane Stress)

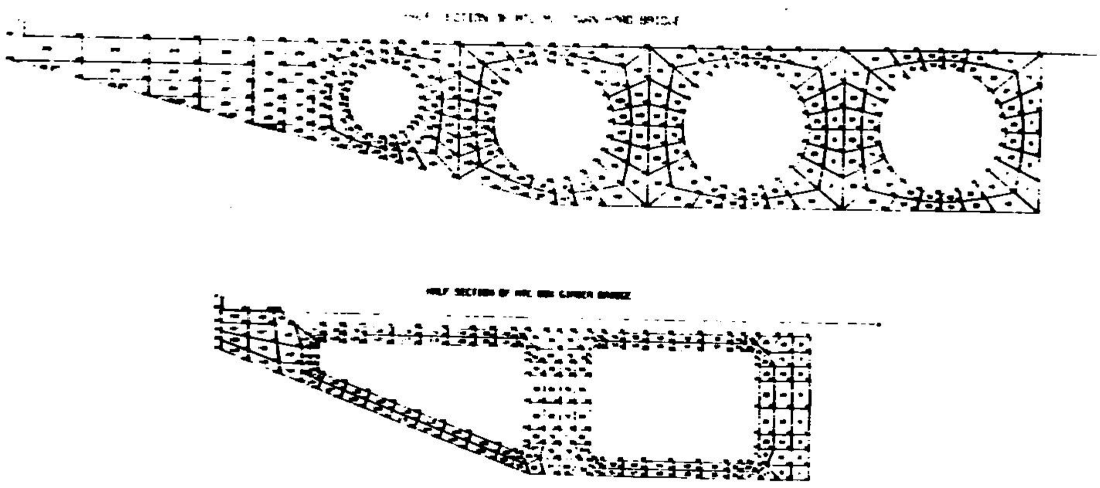


Figure 18 - Plots of Finite Element Meshed Used in Transverse Section Analysis Using WATSA-7 (Plane Strain)

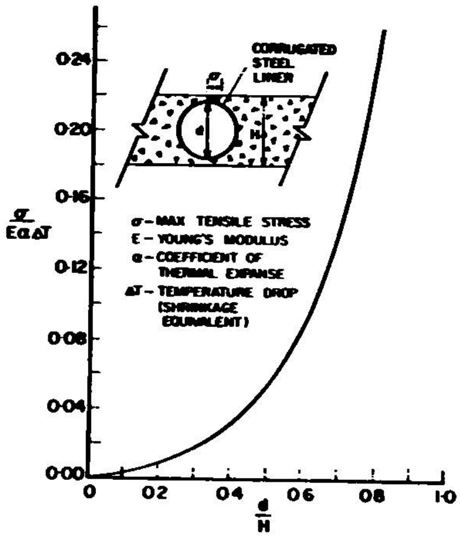


Figure 19 - Effect of Shrinkage Around Circular Voids

4.4 Experimental Model Investigations

In conjunction with the development of computer systems for bridge analysis, there are model studies being done on both rectangular and skew multi-span voided prestressed bridges at the University of Waterloo. These experiments are being conducted under the supervision of Professor Green. Through coordinated efforts, Professor Green and I are able to provide a set of computer-aided analysis systems along with physical verification. This determines both their capabilities and limitations. A three span 1/36 plexiglas bridge model with a rectangular planform is shown in Figure 20. The test results for this bridge were reported in reference [25]. Some of these results were compared with finite element analysis [24]. A second model of 1/24 scale 40° skew bridge was recently tested [26]. This model is shown in Figure 21, prior to testing. For completeness, I have included in the next section some of the results of reference [24].

4.5 Comparison between Numerical and Experimental Models

The primary purpose of the work reported in reference [24] was to assess the adequacy of the two-dimensional finite element system in analyzing the three dimensional stresses in the voided prestressed bridge slabs. Figure 22 contains a plan view of one half of the bridge, while Figures 23 through 27 indicate the accuracy obtained in predicting longitudinal and transverse stresses. The solid line is based on a neutral axis assumed to be at the mid-depth of the bridge cross-section. The dashed lines are finite element results based on a modified moment of inertia for the tapered exterior wings of the bridge. This is discussed in detail in reference [24]. From these results it is clear that where the stress state departs drastically from that assumed in two dimensional theory, a three dimensional model must be used.

The conclusion drawn in the original study [23] was that the two dimensional model was sufficiently accurate to predict stresses in regions other than the end anchorage zones where large concentrated cable forces caused complex three dimensional stress fields and in the outer wing regions where the assumption of the neutral axis cannot be accurately made. Also, in the transverse span direction, the stresses over the voids are not estimated properly. However, unless specific detail is required, the results from the two dimensional model can be appropriately factored in these regions and quite accurate estimates made. Although the 2-D system was more advanced than any similar system existing at the M.T.C. and could be used for most bridge designs, as well as a wide variety of other problems, it was the opinion of the engineers involved that a 3-D capability should be provided. Consequently, such a system was initiated and I should like to report on the status of this endeavour.

4.6 Outline of 3-D Finite Element Analysis System

The 3-D system consists of solid elements of the isoparametric family having quadratic displacement functions [27]. The introduction of a mixed displacement formulation [28] has also been added. This allows any combination of curved boundary elements to be used including those with 20 nodes along with others with any number less than 20. If 8 corner nodes are requested, then the element degenerates to a hexahedron with linear sides. Using these elements in combination will allow the higher order

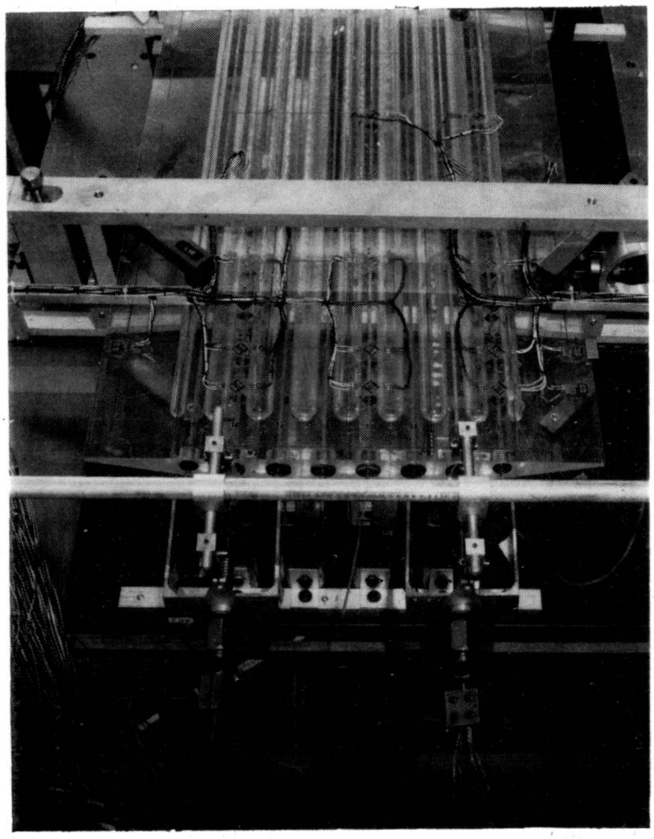


Figure 20 - Three Span Plexiglass Model of a Post-Tensioned Voided Bridge Slab (1/36 Scale)

Figure 21 - Two Span Plexiglass Model of a Post-Tensioned Voided Skew Bridge Slab (1/24)

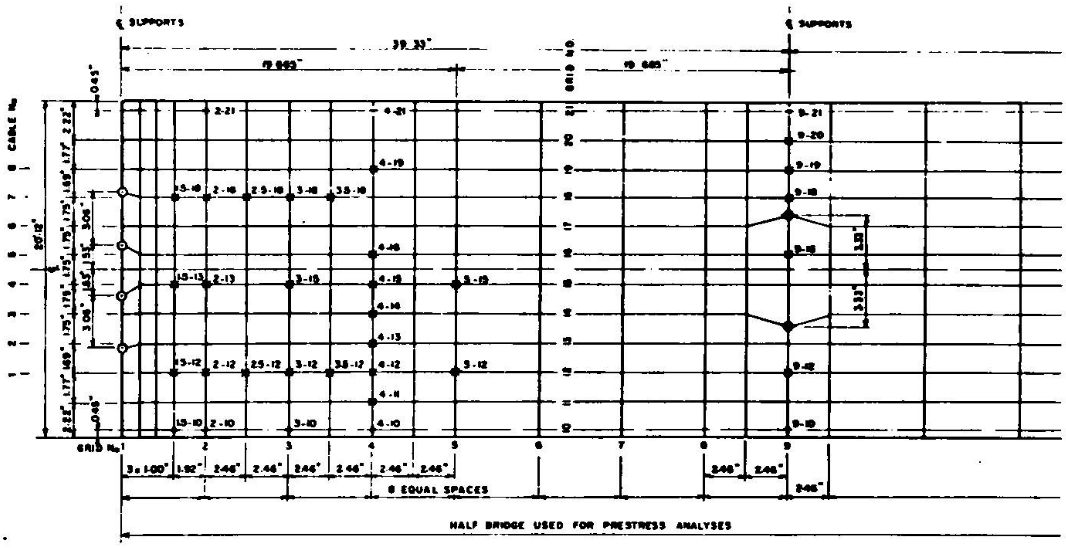


Figure 22 - One Half Plan View of 1/36 Scale Model Showing Gauge and Cable Locations

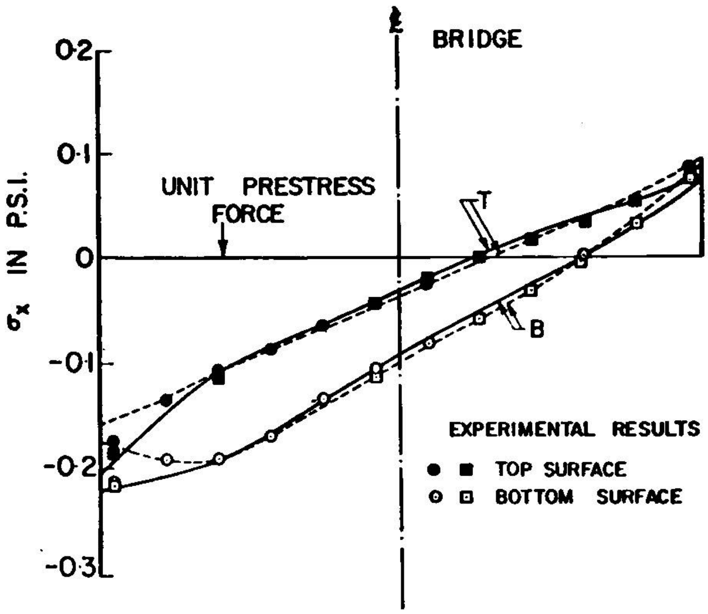


Figure 23 - Longitudinal Stresses Along Transverse Grid Line 4 Unit Prestress in Cable 1

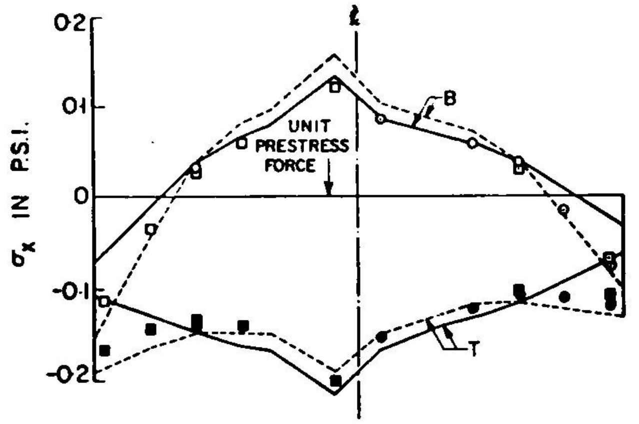


Figure 24 - Longitudinal Stresses Along Transverse Grid Line 4
Unit Prestress in Cable 4

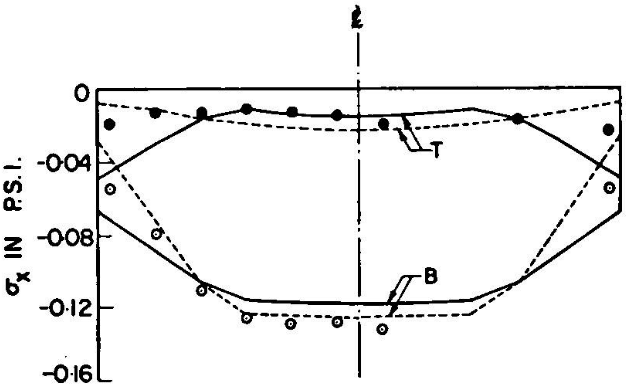


Figure 25 - Longitudinal Stresses Along Transverse Grid Line 4
All Cables Prestressed

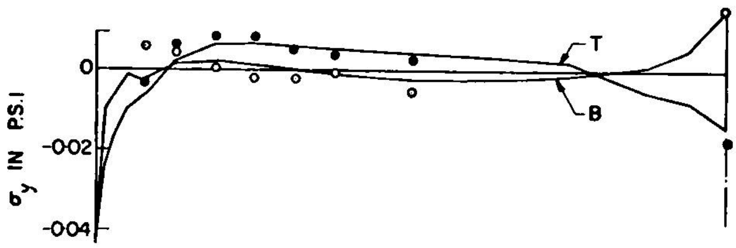


Figure 26 - Transverse Stresses Along Longitudinal Grid Line 12
All Cables Prestressed

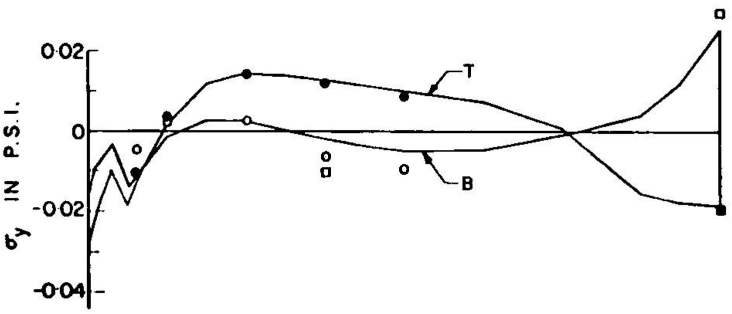


Figure 27 - Transverse Stresses Along Longitudinal Grid Line 15
All Cables Prestressed

quadratic elements to model high gradient strain areas as well as model the curved geometries better around voids, and the lower order elements to model less important areas of the structure. Such a gradation of elements combined with a non-banded equation solver allows much flexibility of analysis at reduced costs.

There are three major levels of activity associated with constructing computer based analyses systems. These comprise the data input level, the mathematical analysis level and the output level. For the 3-D system, the data input is very important since the bridge structure should be automatically described in a form available for analysis. The system being constructed at Waterloo has the capability of producing a complete bridge up to eight spans in length and having any cross-section. There can be both solid and voided regions throughout the length of the bridge. The only stipulation is that the geometry of the cross-section cannot change along the length of the bridge. Work is underway to allow variable width bridges with curved boundaries to be used. The system automatically calculates dead load and temperature and point loads can also be specified. Uniform loads can be positioned over any element and have any value. This is important in dealing with lane loads.

For the analysis section of the system, a large capacity out-of-core solver has been built using matrix partitioning schemes and sparse term storage. Substructuring is now being formulated since otherwise a complete bridge could not be attempted. Individual sections of the bridge can also be formed and analyzed separately in order to investigate local stresses without a full bridge analysis. Three principle stresses are produced at all nodes as well as the usual cartesian values.

The output options of the system will comprise a number of items. Computer plots of individual elements, sections of the bridge or the entire bridge can be selected and viewed from any chosen angle. Figures 28 and 29 are samples of three dimensional output. At the present time, no stress plotting has been developed. This will constitute the final aspect of the work.

Once again, experimental tests on plexiglas models are being done by Professor Green. So far, tests have been conducted on a transverse and a longitudinal section of a typical voided deck. These have been used to establish the element subdivision though the thickness of the slab and along the length of the span. Currently, model tests are being performed on a multi-cell box girder of scale 1/18. This will be used to assess both the 2-D and 3-D finite element systems results.

The system is scheduled to be completed late in 1974. This system, along with the 2-D system that is with the Ministry, should provide a substantial stress analysis capability for both the research and design offices. In this respect perhaps it is fair to say that the Ontario Ministry of Transportation and Communications through its research branch, is one of the most progressive governmental agencies doing structural research in Canada.

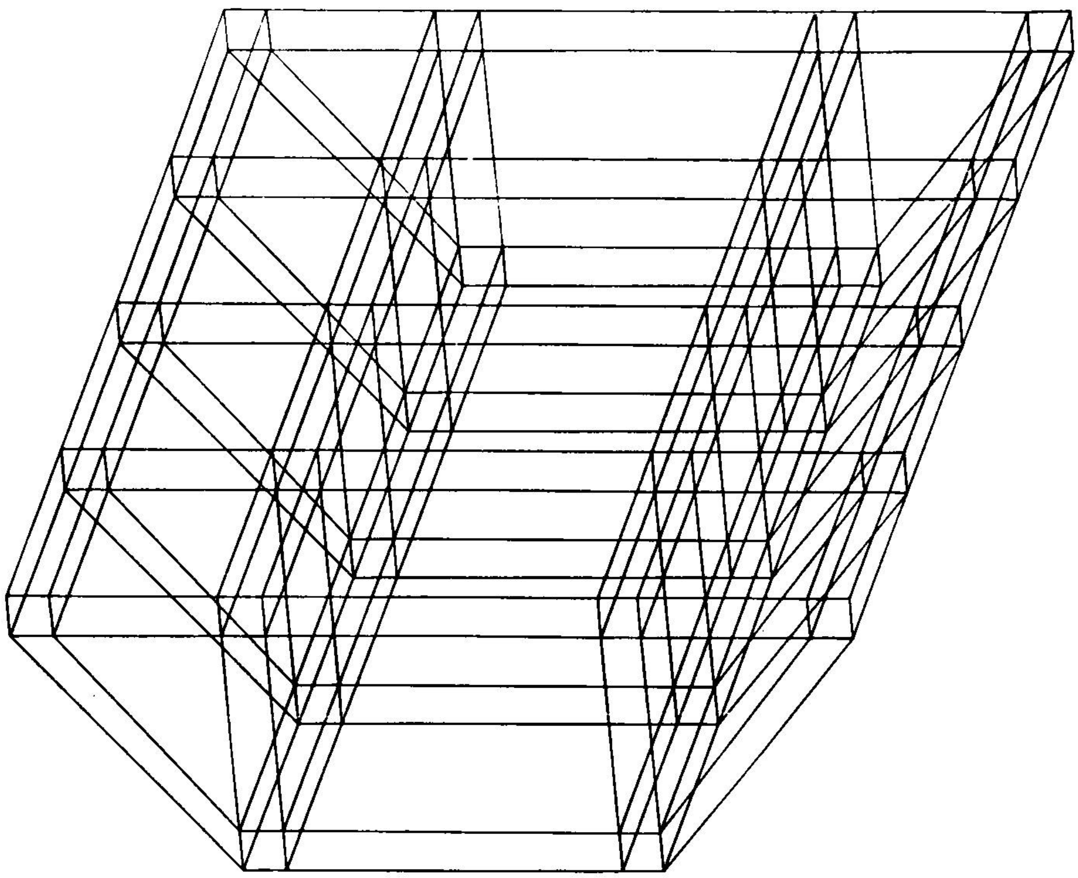


Figure 28 - Three Dimensional Plot of a Three Cell Box Girder Bridge

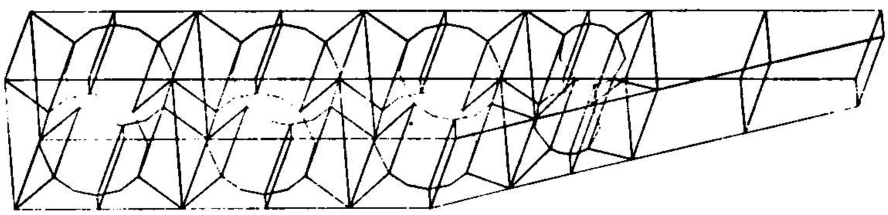


Figure 29 - Three Dimensional Plot of One Half of Transverse Section Test Model

5.0 CLOSURE

In this report I have attempted to summarize on a national basis, the efforts made by various researchers to provide methods of advanced structural and stress analyses for direct application to bridges. The regional subdivision selected was made arbitrarily and does not imply that other researchers in the areas subscribe to the regional work reported herein. Furthermore, the opinions expressed are entirely mine and I stand to be corrected if necessary.

From a national standpoint, there appears to have been substantial developments of computer stress analyses systems in the universities. However, coordination of efforts with potential users such as government design offices has not occurred except in Ontario. The potential now exists and hopefully other provinces will follow the Ontario example.

In closing I should like to thank everyone who has contributed to this report and especially those in the eastern and western regions who made my visits so pleasurable.

REFERENCES

1. Cheung, Y.K., "Finite Strip Method Analysis of Elastic Slabs", Journal of the Engineering Mechanics Division, A.S.C.E., Vol. 94, No. EM6, Proc. Paper 6267, December, 1968, pp. 1365-1378.

2. Cheung, Y.K., "Analysis of Box Girder Bridges by the Finite Strip Method", Proceedings of the Second International Symposium on Concrete Bridge Design, American Concrete Institute, Paper SP 26-15, April 1969, Chicago, Illinois, U.S.A.

3. Cheung, M.S., Cheung, Y.K. and Reddy, D.V., "Frequency Analysis of Certain Single and Continuous Span Bridges", Developments in Bridge Design and Construction, edited by K.C. Rockey, J.L. Bannister and H.R. Evans, Crosby Lockwood and Sons Ltd., London, pp. 188-199, 1971.

4. Cheung, M.S., Cheung, Y.K. and Ghali, A., "Analysis of Slab and Girder Bridges by the Finite Strip Method", Building Science, Vol. 5, Pergamon Press 1970, pp. 95-104.

5. Ghali, A. and Tadros, G.S., "On Finite Strip Analysis of Continuous Plates", Department of Civil Engineering Research Report No. CE72-11, March 1972, University of Calgary.

6. Tadros, G.S. and Ghali, A., "Convergence of Semi-Analytical Solution of Plates", Department of Civil Engineering Research Report, No. CE72-38, September 1972, University of Calgary.

7. Brown, T.G. and Ghali, A., "Semi-Analytic Solution of Skew Plates in Bending", Department of Civil Engineering Research Report No. CE72-46, December 1972, University of Calgary.

8. Sisodiya, R.G., Cheung, Y.K. and Ghali, A., "New Finite Element with Application to Box Girder Bridges", Paper No. 7479, Journal of ICE, London, 1972.

9. Zienkiewicz, O.C. and Cheung, Y.K., "Finite Element Method in Structural and Continuum Mechanics", New York, McGraw-Hill, 1967.

10. Sisodiya, R.G., Ghali, A. and Cheung, Y.K., "Diaphragms in Single and Double Cell Box Girder Bridges with Varying Angle of Skew", A.C.I. Journal, July, 1972, pp. 415-419.

11. Sisodiya, R.G., Cheung, Y.K. and Ghali, A., "Finite Element Analysis of Skew Vault Bridges", International Association for Bridge and Structural Engineering, Publications Vol. 32-II, Zurich, 1972.

12. Mehrotpa, B.L., Mufti, A.A. and Redwood, R.G., "Analysis of Three Dimensional Thin Walled Structures", Journal of A.S.C.E. Structural Division, Vol. 95, No. ST12, December 1969, pp. 2863-2872.

13. Fam, A. and Turkstra, C., "An Annular In-plane Finite Element", Proc. of the Specialty Conference on Finite Element Method in Civil Engineering, McGill University, June, 1972, pp. 1027-1047.

14. Olson, M.D. and Lindberg, G.M., "Annular and Circular Sector Finite Elements for Plate Bending", International Journal of Mechanical Sciences, Vol. 12, No. 1, Jan. 1970, pp. 17-34.

15. Aneja, I. and Roll, F., "Experimental and Analytical Investigation of a Horizontally Curved Box-Beam Highway Bridge Model", Paper SP26-16, Proceedings of the A.C.I. Annual Convention, Chicago, March 1969.

16. Gurevich, S. and Redwood, R.G., "Approximate Analysis of Multi-cell Skew Box Bridges", Proc. of the Speciality Conference on Finite Element Method in Civil Engineering, McGill University, June 1972, pp. 1027-1047.

17. Crisfield, M.A., "Finite Element Methods for the Analysis of Multicellular Structures", Proceedings of the ICE, Vol. 48, Paper No. 7373, March 1971, pp. 413-437.

18. Sisodiya, R.G. and Ghali, A., "Analysis of Box Girder Bridges of Arbitrary Shape", International Association of Bridge and Structural Engineering, Publications Vol. 33-I, Zurich 1973.

19. Sundararajan, C. and Reddy, D.V., "Response of Rectangular Orthotropic Plates to Random Excitations", Faculty of Engineering and Applied Science, Memorial University of Newfoundland, St. John's, Newfoundland, 1972 (private communication).

20. Clough, R.W. and Felippa, C.A., "A Refined Quadrilateral Element for Analysis of Plate Bending", Proceedings 2nd Conference on Matrix Methods in Structural Mechanics, Wright-Patterson Air Force Base, Ohio 1968.

21. Zienkiewicz, O.C., "The Finite Element Method in Engineering Science", McGraw-Hill, 1971.

22. McNeice, G.M. and Jofriet, J.C., "Finite Element Analysis of Bridge Slabs", Report submitted to the Ministry of Transportation and Communications, Downsview, Ontario, July, 1971.

23. McNeice, G.M. and Jofriet, J.C., "Finite Element Analysis of Post-tensioned Highway Bridges", Report submitted to the Ministry of Transportation and Communications, Downsview, Ontario, April 1972.

24. Jofriet, J.C., McNeice, G.M. and Csagoly, P., "Finite Element Analysis of Prestressed Concrete Voided Bridge Decks", PCI Journal, Volume 18, No. 3, May-June, 1973.

25. Green, R., "Prestressing Effects in Voided Slab Bridges", Report Submitted to the Ministry of Transportation and Communications, Downsview, Ontario, April 1972.

26. Green, R., "Skew Voided Slabs", Report Submitted to the Ministry of Transportation and Communications, Downsview, Ontario, April, 1972.

27. Ergatoudis, J.G., "Isoparametric Finite Element in Two and Three Dimensional Stress Analysis", Ph.D. Thesis, School of Engineering, Division of Civil Engineering, University College of Swansea, Great Britain, October, 1968.

28. McNeice, G.M. and Hunnisett, S.F., "Mixed Displacement Finite Element Analysis with Particular Application using Plane Stress Triangles", Journal of Strain Analysis, Vol. 7, No. 4, October 1972.

Summary

This report contains a summary of Canadian research efforts that are directed toward providing advanced analytical capabilities for the analyses and design of concrete bridges. Activities are reported on a regional basis arbitrarily selected as western, eastern and central. The corresponding centers from which most of the developments originate are respectively, the University of Calgary, McGill University and the University of Waterloo. Brief outlines of the theoretical aspects are provided in order to illustrate the varied approaches. Also, comparison with model studies are made wherever possible. With the exception of central Canada, most of the effort has been directed toward box girder analysis using two dimensional finite strip and finite element techniques. In Ontario both two and three dimensional computer systems are being developed. An account is given of a provincial government involvement in the creation of advanced stress and structural analyses systems.

Sommaire

Ce rapport contient un résumé des efforts de la recherche Canadienne dirigés vers l'avancement des capacités analytique de l'analyse et du 'design' des ponts en béton. Les activités sont rapportées sur une base régionale choisie arbitrairement comme celles de l'ouest, l'est, et le centre. Les endroits correspondante d'où originent la plupart des développements son L'Université de Calgary, L'Université McGill, et L'Université de Waterloo. De brefs sommaires des aspects théoriques sont fournis pour démontrer les diverses idées. De plus, des comparaisons avec des études en modèle sont faites chaque fois que c'est possible. A l'exception du centre du Canada, la plupart des efforts ont été dirigés vers l'analyse de poutres-caisson, en utilisant les techniques des bandes finies a deux dimensions et des éléments finis. En Ontario, des systèmes de calcul à deux et trois dimensions ont été développés. On se doit de mentionner d'aide accordé par le gouvernement provincial pour la création de systèmes avancés d'analyse de tension et de construction.

Zusammenfassung

Dieser Artikel beinhaltet einen zusammenfassenden Bericht ueber die kanadische Forschungsarbeit auf dem Gebiet der Anwendung theoretischer und numerischer Analysen fuer den Entwurf von Betonbruecken. Die Arbiet stuetzt sich auf Forschungen an verschiedenen Universitaeten Kanadas; haupsaechlich University of Waterloo, University of Calgary and McGill University. Ein kurzer Uebersicht der theoretischen Aspekte ist beschrieben, um die vielseitige Method der Analyse klar zu zeigen. Wo es moeglich ist, sind auch Vergleiche mit den Resultaten aus den experimentellen Modelluntersuchungen gegeben. Die meisten Arbeiten betreffen die Anwendung von zweidimensionalen endlichen Elementen an Balkan. An der University of Waterloo werden auch dreidimensionale systemen entwickelt. Auszerdem ist auch eine Auffuehrung der praktischen Anwendung der Analyse angefuehrt.

Leere Seite
Blank page
Page vide