Finite element analysis of skew, curved box-girder bridge

Finite Element Analysis of Skew, Curved Box-Girder Bridge

Calcul à l'aide des éléments finis des ponts courbes, biais à section en caisson

Berechnung von schiefen, gekrümmten Brücken mit Kastenquerschnitt mit der Methode der finiten Elemente

R. G. SISODIYA
M. Sc., Graduate Student, Dept. of Civil Engineering, University of Calgary, Calgary, Alberta, Canada

Y. K. CHEUNG
Ph.D., MICE, Professor of Civil Engineering, University of Calgary

A. GHALI
Ph.D., Professor of Civil Engineering, University of Calgary

Introduction

In modern highways, many skew bridges are built, and very often these bridges are curved in plan. The box section has been favoured by many designers because of its aesthetic appearance and because of its high torsional rigidity.

In the past curved box-girders has been treated one-dimensionally as a curved beam, thus ignoring the distortions of the cross-section, and in many cases the skew effect as well.

CHEUNG et al. [1] use the finite strip method to analyse curved box-girder bridges. In this method, the curved plates should be circular and of constant width and the bridge should be ended by two radial cross sections. If all these conditions are satisfied, the finite strip method provides a solution which can be conveniently used in practical design, because it requires relatively short computer time and small computer storage.

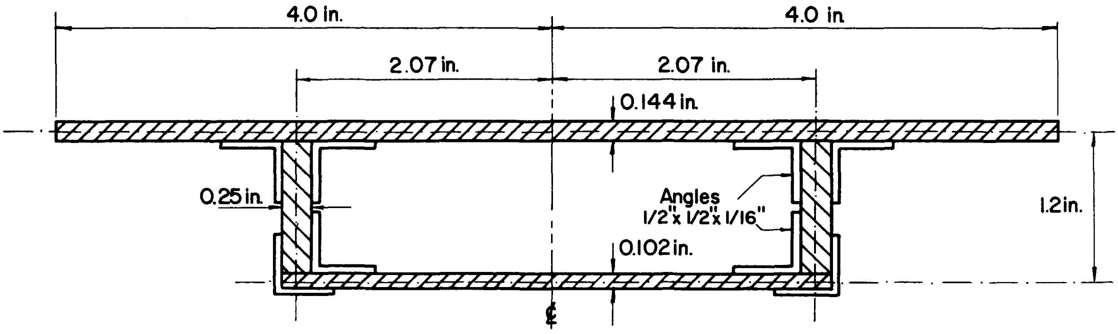
The present paper deals with finite element analysis of single box-girder skew bridges curved in any shape. The bridge may be of varying width and of any support conditions.

The procedure of the analysis and the types of the finite elements used to idealize the bridge deck is presented in a separate paper [2] which is limited to skew straight bridges. Results of the analysis of a curved bridge are presented here and they are compared with the results of experiments on a model 3.

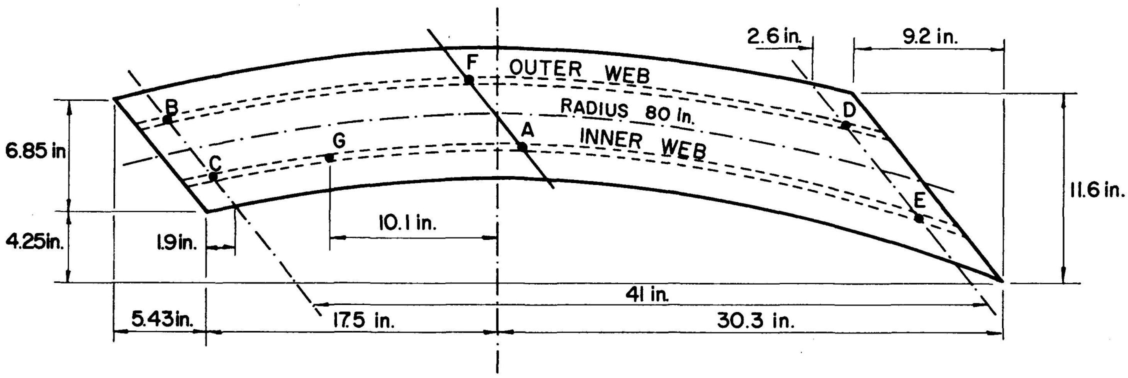
In the earlier paper [2] we recommended that the webs of the box be divided into rectangular elements. Here we will show how the ratio of the sides of the rectangular elements can affect the accuracy of the results.

Finite Element Analysis and Test Results

The general principles of the finite element method and the detailed formulation of the elements used in this paper can be found in a text by ZIENLIEWICZ and CHEUNG [4], and shall not be discussed here.



a) Cross-section.

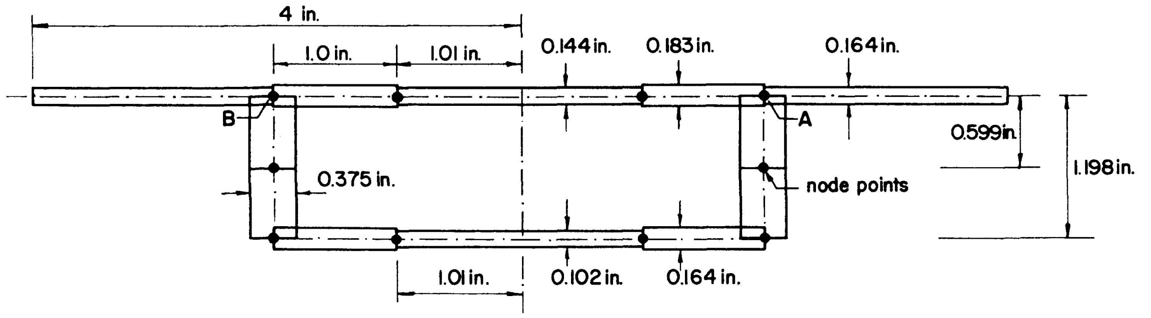


b) Plan.

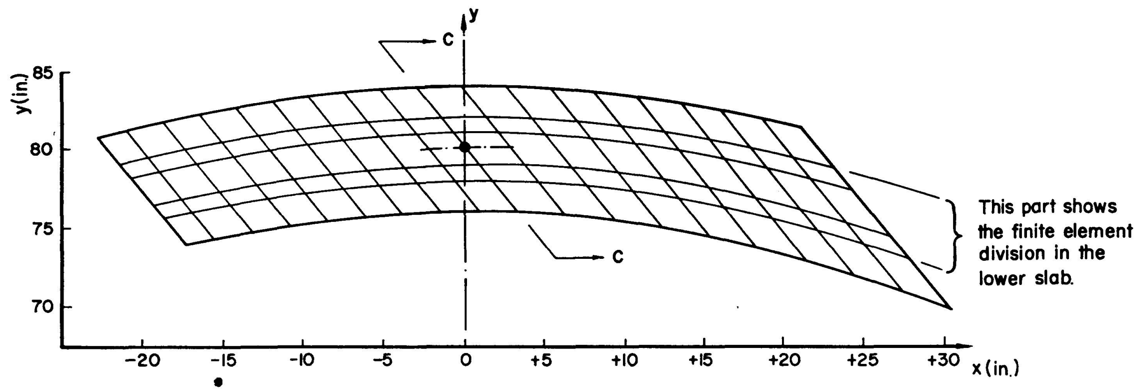
Fig. 1. Curved box-section bridge model.

Fig. 1a and b show the plan and the cross section of a bridge model made of aluminum alloy *) which was analysed by finite elements and tested for various loading cases. Fig. 2a and b show the finite element idealization used for the analysis, in which rectangular elements are used for the webs while the top and bottom slabs are divided into parallelogram elements. As an alternative the top and bottom slabs may be divided into triangles as shown in Fig. 2c. Both idealizations gave identical results and the computer time was shorter with the triangular elements. However the parallelogram element offers

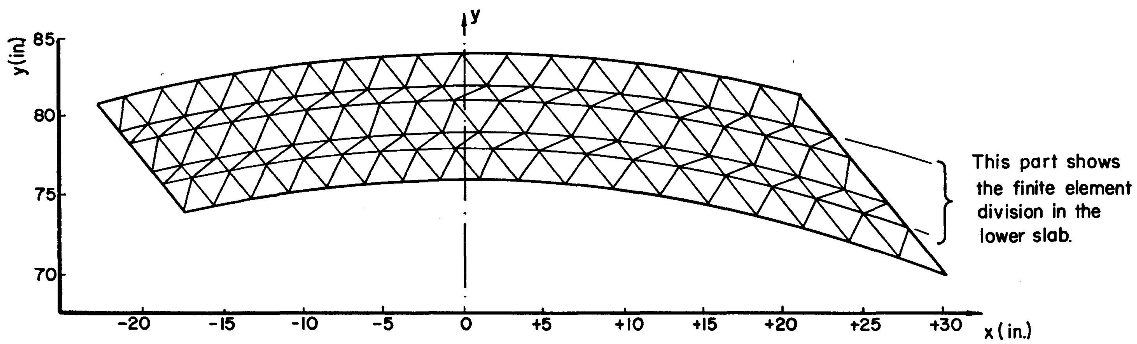
*) For details of testing procedure see Ref. 3.



a) Cross-section.

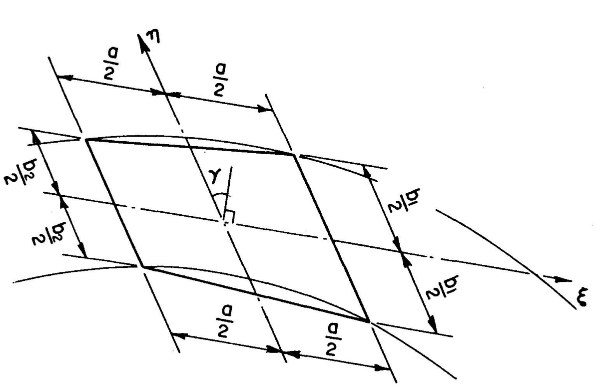


b) Use of parallelogram elements for top slab.

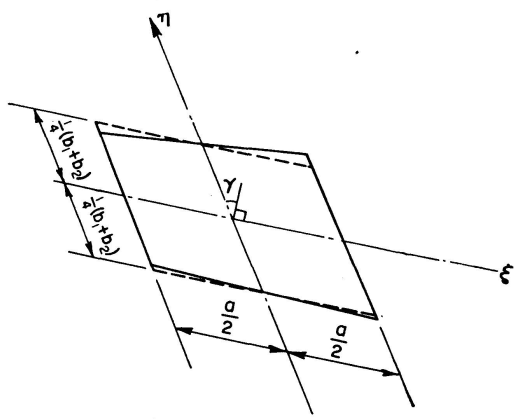


c) Use of triangular element for top slab.

Fig. 2. Finite element idealization.

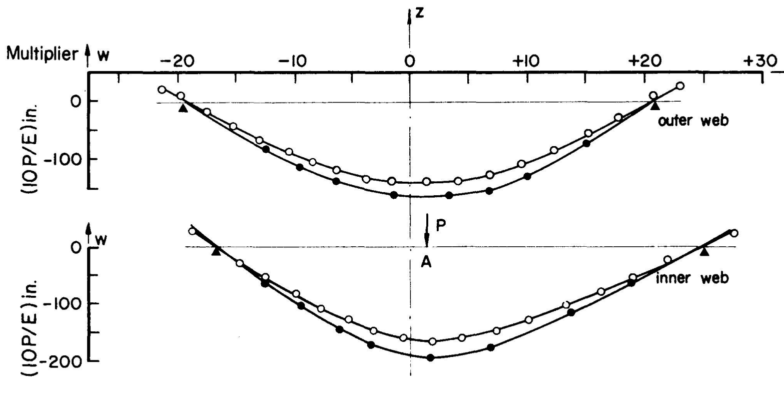


a) Quadrilateral defined by 2 arcs of circles with same centre and two parallel lines.



b) Parallelogram approximation of quadrilateral in a).

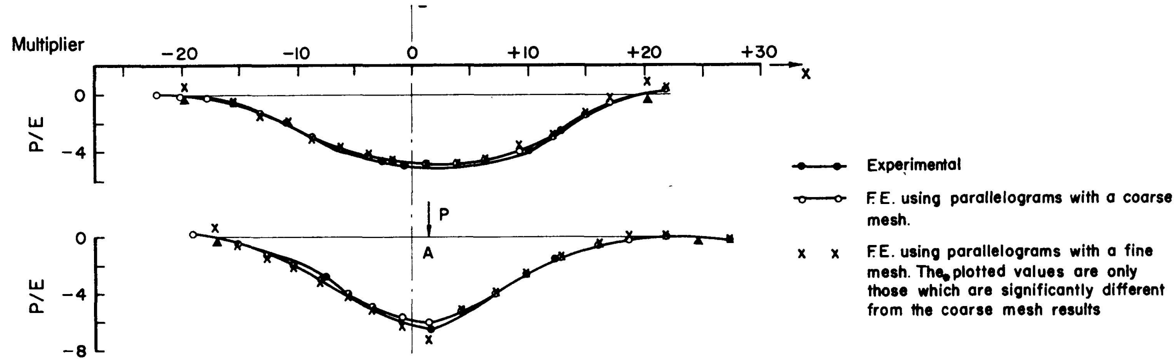
Fig. 3. Parallelogram approximations of the elements in top and bottom slabs (Fig. 2b).



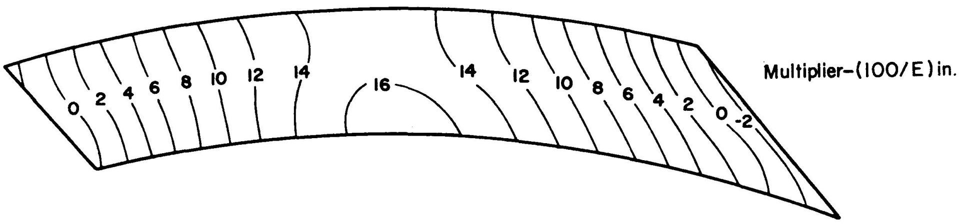
REACTIONS (Multiplier : P)

POINT	EXPERIMENTAL	ANALYTICAL Coarse Mesh	ANALYTICAL Fine Mesh
B	− 0.131	− 0.169	− 0.169
C	+ 0.606	+ 0.669	+ 0.669
D	+ 0.657	+ 0.647	+ 0.647
E	− 0.127	− 0.147	− 0.147

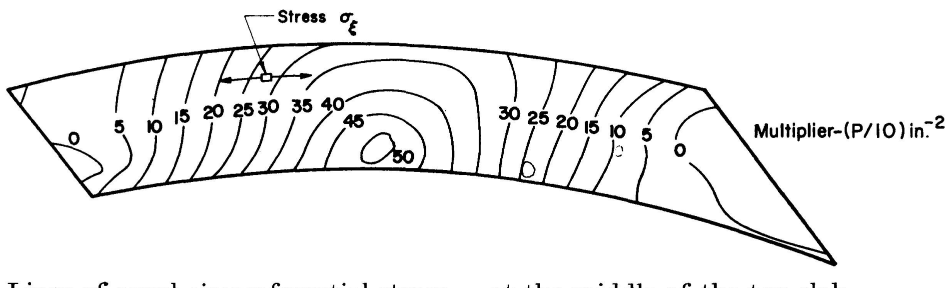
a) Vertical deflections of the webs.



b) Tangential strains at upper surface of the top slab.



c) Conturs of the vertical deflections of the top slab.



d) Lines of equal circumferential stress σ_ξ at the middle of the top slab.

Fig. 4. Deflections, reactions, strains, and stresses due to vertical load P at point A for one span bridge model (Fig. 1).

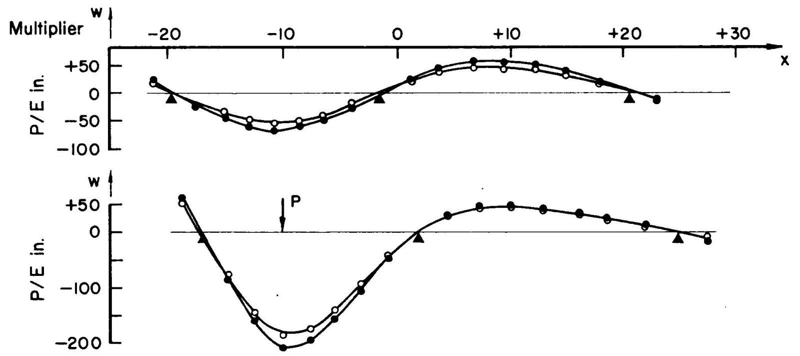
The deck is supported at four points B, C, D and E (Fig. 1b), and the reactions are assumed to have vertical component only.

some advantage in the interpretation of the results since the stresses as well as displacements are computed at the same nodal points.

It is obvious that the top and bottom slabs cannot be divided into elements which are perfect parallelograms, and thus some approximation is necessary. The lines of division in Fig. 2b are actually straight lines parallel to the lines of supports, and in the other direction they are in fact polygonal lines joining points on circular arcs. In the calculation of the stiffness matrix and the stresses in an element such as that in Fig. 3a, it is approximated into a perfect parallelogram element as in Fig. 3b.

Some of the results of the analysis and experiment are given in Fig. 4, which represent the effects of a unit vertical load applied at point A (Fig. 1b).

In Fig. 4a the variation of the vertical deflection and the strains are plotted along the centre line of the web, together with a table giving te reactions at the supports. Two different finite element mesh divisions are used in the analysis; a coarse mesh corresponding to the idealization shown in Fig. 2a and b, and a fine mesh obtained by subdividing into two all the elements of the coarse one in the span direction (except for the elements adjacent to the end cross-sections). The deflections obtained for the two different meshes are practically identical. The experimental values for the deflections are somewhat



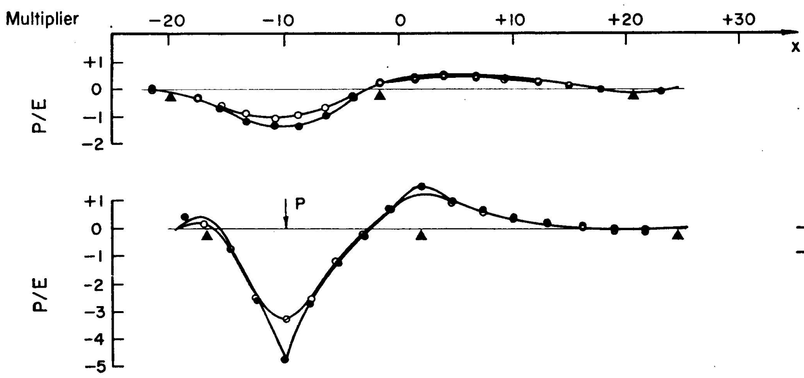
REACTIONS

POINT	ANALYTICAL	
	Coarse Mesh	Fine Mesh
B	−0.019	−0.029
C	+0.550	+0.560
F	+0.164	+0.183
A	+0.383	+0.362
D	−0.102	−0.110
E	+0.025	−0.033

Multiplier: P

Finite element using parallelogram
—○— Coarse mesh
—●— Fine mesh

a) Vertical deflections of the webs.



Finite element using parallelogram
—○— Coarse mesh
—●— Fine mesh

b) Tangential strains ϵ_ξ at points A and B in Fig. 2a.

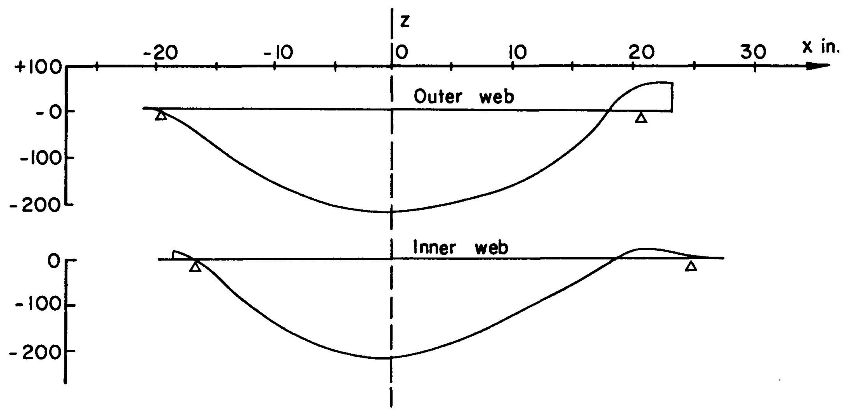
Fig. 5. Deflections, reactions and strains due to vertical load P at point G for two span bridge model (Fig. 1).

higher on the whole and this has been attributed to the fact that the stiffness of the model beam has been reduced by the imperfect epoxy bond between the web and flanges. The contours for the circumferential stress σ_ξ is shown in Fig. 4d.

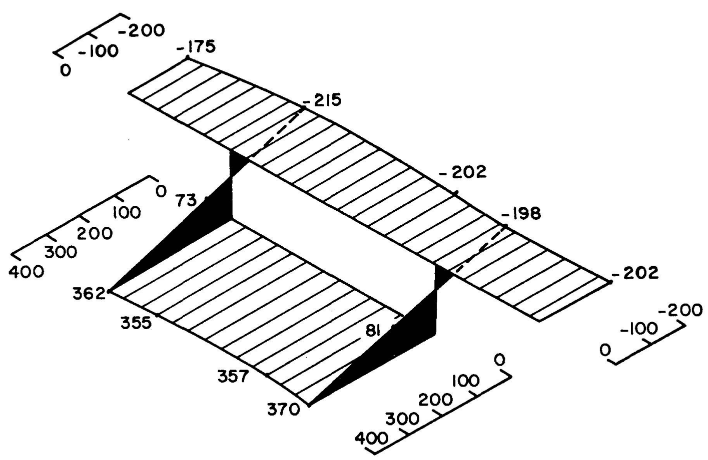
The same model was analysed as a continuous bridge over two spans by introducing the supports at A and F and applying a vertical concentrated load at G (Fig. 1b). The vertical deflections, strains and reactions are plotted in Fig. 5 for the coarse and fine meshes described above.

The number of node points in the idealized structure in Fig. 2 is 228 requiring the solution of $6 \times 228 = 1368$ equations. The time taken by an IBM 360-50 computer was 24 minutes to calculate the displacements, the reactions and the stresses due to one loading case, and the time is increased by 2 minutes for each additional case.

When the finer mesh described above was used, the number of node points and equations became 420 and 2520 respectively. Because of the large number of equations and the short word length (32 bits) of the computer, double precision had to be used in the equation solving routines.



a) Along span of the bridge at points A and B in Fig. 2a.



b) At section C-C in Fig. 2b·

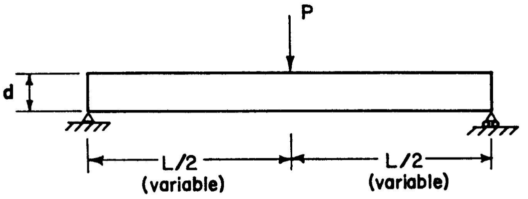
Fig. 6. Variation of stress σ_ξ (lb./in.^{2}) as defined in Fig. 4d due to self weight of specific weight of 1 lb./in.^{3}.

To analyse the bridge model for load representing its own weight, the weight of each element is distributed equally to the four corner nodes (Fig. 2b). The variation of stress σ_ξ along the span of the beam at the intersection of the web and the top slab (points A and B in Fig. 2a) are shown in Fig. 6a. In Fig. 6b, we give the variation of σ_ξ on Section C-C (see Fig. 2b).

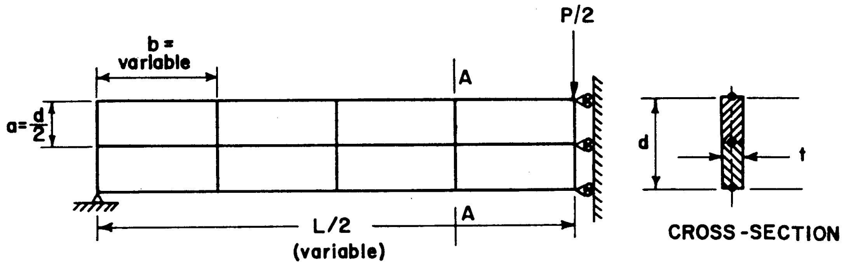
Accuracy of In-Plane Rectangular Element for Beam Problems

As previously mentioned, the agreement between the experimental and analytical deflections in Fig. 4a is not entirely satisfactory, and originally the discrepancies were suspected to be due to the mesh division used for the web, which has a 1 : 4 aspect ratio for the elements used in the coarse mesh, and a 1 : 2 aspect ratio in the fine mesh.

In order to obtain further information on the adequacy of the mesh divisions, the simply-supported beam in Fig. 7a was analysed using rectangular elements (Fig. 7b) with $b=a$, $2a$ and $4a$ respectively for three different cases.



a) Support and load.



b) Finite element model of half of beam.

Fig. 7. Simply supported beam.

The deflection at centre and stresses at Section A-A in Fig. 7 are compared with known values from beam theory (including shear deformation for Poisson's ratio = 0) in Table 1.

It is indeed surprising to note that the rectangular element which has been widely used in two-dimensional elasticity problems gives such poor results in beam analysis when the ratio of the sides a/b is smaller than unity, and that the deterioration in accuracy when the aspect ratio changes from 1 : 1 to 1 : 4 can be so drastic.

However, the above drawback of the in-plane rectangular element does not have such an important effect on the analysis of box sections, since the longi-

Table 1. Central Deflection and Maximum Stresses in a Test Problem (See Fig. 7)

Half Span Length $L/2$	Ratio $a:b$	Deflection at Mid-Span in terms of P/Et			Maximum Stresses at Section A-A in terms of P/td		
		F.E.M.	Beam Theory Including Shear Deformation [5]	F.E.M. Results as a Percentage of the Beam Theory Result	F.E.M.	Beam Theory	F.E.M. Results as a Percentage of the Beam Theory Result
$2d$	1 : 1	16.62	18.19	91.4 %	4.00	4.5	88.7 %
$4d$	1 : 2	88.38	132.59	66.7 %	6.00	9	66.7 %
$8d$	1 : 4	340.58	1033.29	33.0 %	5.94	18	33.0 %

tudinal forces are mainly resisted by the top and bottom slabs. This is confirmed by only a small improvement of the results achieved by the use of the finer mesh described earlier (see Fig. 4a and b) for the curved box-girder bridge.

Conclusion

The finite element method can be economically used to replace model testing for the analysis of curved skew box-girder bridges. More research is needed to develop elements which permit the use of smaller number of equations and to reduce computing time and programming effort, before the method can be widely accepted by bridge designers.

It is suggest that rectangular elements be used for the webs and either parallelogram or triangular element for the top and bottom slabs. In practice, the support of a skew bridge lie on parallel lines, and curved bridges have large radii of curvature such that it is often possible to approximate the top and bottom slabs as assemblage of parallelograms. The triangular element can be used in any general case.

Acknowledgement

This project was supported by grants from the National Research Council of Canada, which is gratefully acknowledged.

Notation

E Young's Modulus
L span length
P load
R reaction

a, b	sides of a rectangular element
d	depth of a beam
t	thickness of an element
w	vertical deflection
ξ, η	skew coordinates for a parallelogram element
σ	stresses
ϵ	strains
x, y, z	axes

References

1. M. S. Cheung, and Y. K. Cheung: Analysis of Curved Box-Girder Bridges by Finite Strip Method. In preparation.
2. R. G. Sisodiya, A. Ghali, and Y. K. Cheung: Finite Element Analysis of Skew Box-Girder Bridges. Submitted for publication.
3. R. G. Sisodiya: Analysis of Box-Girder Bridges by the Finite Element Method. M.Sc. Thesis, University of Calgary, 1969.
4. O. C. Zienkiewicz, and Y. K. Cheung: Finite Element Methods in Structural and Continuum Mechanics. McGraw-Hill Book Co., 1967.

Summary

Box-girder bridges having one or two spans and with curved layout and skew supports are analysed by the finite element method. The results of analysis is verified by tests on an aluminum bridge model.

The type of elements used to idealize the bridge are described and the effect of their choice on the accuracy of the analysis is discussed.

Résumé

On présente le calcul par la méthode des éléments finis des ponts à une ou deux travées, présentant un axe courbe et des appuis biais, avec une section en caisson fermé. On a vérifié les résultats du calcul à l'aide d'essais sur des modèles de ponts en aluminium.

Le présent article décrit le type des éléments utilisés et étudie l'incidence du choix de la forme des éléments sur l'exactitude du calcul.

Zusammenfassung

Kastenförmige Brücken über eine oder zwei Spannweiten mit gekrümmter Achse und schiefer Lagerung werden mittels der Methode der finiten Elemente gerechnet. Die Ergebnisse der Berechnung werden durch Versuche an einem Aluminium-Brückenmodell bestätigt.

Die Art der verwendeten Elemente zur Idealisierung der Brücke werden beschrieben und die Auswirkung ihrer Wahl auf die Genauigkeit der Rechnung diskutiert.

Leere Seite
Blank page
Page vide