FINITE ELEMENTS IN PRACTICE: A REVIEW OF ENGINEERING'S MOST VERSATILE METHOD

Oagile J. Kanyeto
Department of Civil Engineering
University of Botswana
Gaborone, Botswana

Advantages of using the Finite Element Method (FEM) in structural engineering practice are presented in this paper. A brief description of the method is given with the aim to illustrate the rich and solid mathematical basis that makes its foundation. The paper then presents some guidelines on how to build a good model for Finite Element Analysis (FEA) purposes. The guidelines emphasize the need for the analyst to have a clear understanding of the physical problem and of the behaviour of the elements he employs to carry out the analysis. A practical example is presented in order to illustrate the type of analysis and results that can be obtained from a commercial FE software package. This example forms part of the research work that was carried out by the author. And finally, it is recommended that all practicing structural engineers must learn the basics of the finite element method, and be equipped with some FE software packages, since it is the most commonly used and trusted method in the world of engineering today. In order to keep up with the rest of the world in engineering advancements, the author finds it vital to equip all engineers with the latest engineering software packages and to elevate continuing engineering education.

1 INTRODUCTION

The theory of structures developed in the 19th century focused on truss analysis methods, such as the force method. Non-frame-type structures could sometimes be analyzed by solving the governing differential equations [1]. However, complex structures with high order of redundancy, triggered by the introduction of reinforced concrete at the beginning of the 20th century, could not be analyzed with the classical methods of the 19th century. This prompted development of more powerful methods of analysis such as; the slope-deflection method, the method of moment distribution and the matrix methods of analysis.

The finite element method (FEM) was developed in the 1940s as an extension of the already established matrix analysis techniques, particularly for analyzing complex structural systems for which exact solutions do not exist. The method is a numerical procedure that produces many simultaneous algebraic equations, which necessarily implies a considerable amount of computation. For this reason, the use of a digital computer for data processing is implicit. Results are rarely exact. However, errors are decreased by processing more equations, and results accurate enough for engineering purposes are obtainable at reasonable cost.

During the 1960s, the FEM flourished because of the simultaneous developments in the field of computer science. Since then computer technology and cost have changed rapidly. It is now possible to buy cheap powerful systems that can support very sophisticated graphics and massive mathematical computations. Problems that previously were utterly intractable are now solved routinely.

Also contributing to the rapid growth of the FEM from the late 1960s onwards was the realization that it could be applied to problems other than those of solid and structural mechanics. The method was seen as applicable to all field problems that can be cast in a variational form. Consequently, finite elements are also used to analyze problems of fluid mechanics, aerodynamics, electromagnetic theory, metal forming, soil mechanics, heat transfer, and so on.

2 THE FINITE ELEMENT METHOD (FEM)

Modern finite element theory had its recognizable beginnings in the displacement (or stiffness) method of structural analysis. Initial work on finite elements (FE) was based on a combination of elementary theory plus intuitive reasoning. It was later shown that this early work could be developed from the variational principles of elasticity, and also from 'weighted residuals' techniques [2].

The underlying idea of the method consists of sub-dividing the structure being analyzed into a discrete number of sub-regions of finite dimensions, and of locally applying, under certain conditions, an approximate solution. The observation that an integral of a measurable function over an arbitrary domain can be broken into the sum integrals over an arbitrary collection of almost disjointed sub-domains, whose union is the original domain, is a vital one. The problem is greatly simplified in the sense that, over each finite element, it is possible to adopt simple functions (shape functions) to represent the local behaviour of that element. An analysis of a problem can be made locally, over a typical sub-domain, and by making the sub-domain sufficiently small it can be argued that polynomial functions of various degrees are adequate for representing the local behaviour of the solution. The true solution, however, may be represented by a series of partial differential equations whose solution may not be that apparent.

FEM, therefore, offers a unifying approach to the solution of diverse engineering problems, so that it is considered as a general numerical technique for the solution of systems of partial differential equations subject to appropriate boundary and initial conditions. The Rayleigh-Ritz method [3](one of the available FE formulation methods), for example, consists in seeking a function φ that minimizes integrals of the form

$$I = \int F[x, u(x), u_x(x)]dx \quad (1)$$

where u_x denotes the first derivative of u with respect to x. A trial function must be selected so that it satisfies the displacement boundary conditions prescribed for the problem itself. The task is to seek an approximate solution in the form of a finite series

$$u_N = \Sigma c_j \varphi_j, \quad \text{for } j = 1, 2, \ldots, N \quad (2)$$

in which the parameters c_j are determined by minimizing the functional I, and φ_j are a suitable complete set of linearly independent basis functions (approximation functions). N is determined by the number of degrees of freedom (DOF) associated with the element. After substituting u_N from Eq.(2) for u in Eq.(1) and integrating, the functional I becomes an ordinary function of the parameters $c_1, c_2, \ldots, c_N$. Then the necessary condition for the minimization of $I(c_1, c_2, \ldots, c_N)$ is that its partial derivatives with respect to each of the parameters is zero:

$$\partial I / \partial c_1 = 0, \partial I / \partial c_2 = 0, \ldots, \partial I / \partial c_N = 0 \quad (3)$$

Thus, there are N linear algebraic equations in N unknowns, c_j (j = 1, 2, ..., N). These equations can be written in the form

$$\mathbf{ku} = \mathbf{f} \quad (4)$$

where **k** is the element stiffness matrix of size N by N, **u** represents a vector of nodal displacements for the element (N by 1), and **f** is a vector of nodal forces (N by 1). Equations of this type are derived for each element in a mesh, and are later assembled and appropriately manipulated to form the **K** matrix, **U** and **F** vectors for the entire structure. The sizes of **K**, **U** and **F** are determined by the number of DOF of the structure. In most physical problems the DOF are usually in the thousands. The solution of these equations yields **U**, the nodal displacements of the structure. Strains and stresses are computed from the nodal displacements of each element, which are extracted from **U**. The degree of approximation that can be reached is dependent on the number of elements in the mesh and on their type. The finer the mesh and the higher the degree of shape functions, the better will be the numerical solution obtained. However, the computation time needed will also increase, eventually to undesirable proportions.

3 WHY FINITE ELEMENTS

Several properties of the finite element method have contributed to its extensive use in all fields of engineering and physics. The most distinctive feature of the method that separates it from others is the division of any given domain, regularly or irregularly shaped, into a set of sub-domains. Irregularly shaped boundaries can be approximated by using elements with straight sides or matched almost exactly with curved elements. A few examples of some commonly used elements are shown in Fig.1. The size of the elements can also be varied, thus allowing the element grid to be expanded or refined as the need arises, see Fig.2. Properties of adjacent elements of the boundary do not have to be the same, which allows the method to be applied to bodies composed of several materials.

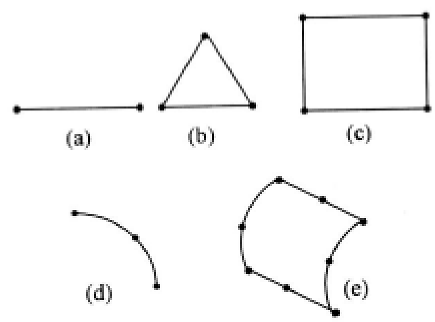


Fig.1 : Straight-edge (a, b, c) and curved (d, e) elements. **(a)** linear bar element, **(b)** linear triangle, **(c)** bilinear quadrilateral, **(d)** quadratic bar element, and **(e)** quadratic quadrilateral element.

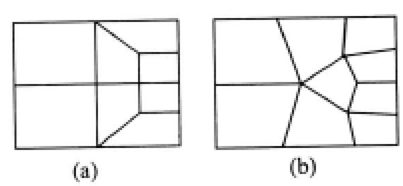


Fig.2 (a, b): Transitions from coarse to finer mesh that avoid abrupt size changes.

The success of the method is largely due to the ease with which the analysis of a class of problems, without regard to a specific problem, can be incorporated into computer programs. The steps involved in the analysis of a general class of problems are systematic, which makes them readily implementable on a digital computer.

4 MODELING CONSIDERATIONS

A good choice of elements can save computational cost while giving accurate results. The size of elements in a mesh depends on the problem to be solved and on the accuracy desired. A balance must be reached between the accuracy of the solution and the computational time. In any given problem, one might begin with a finite element mesh that is believed, based on experience and engineering judgment, to be adequate to solve the problem. Then as a second choice, one selects a mesh that consists of a large number of smaller elements to solve the problem once again. If there is a significant difference between the two solutions, one sees the benefit of mesh refinement, and further refinement may be warranted until the difference is negligibly small. In a case where the computational costs are the prime concern, the analyst must use one's own judgment concerning what is reasonably a good mesh.

Particular attention must be paid to regions of stress concentration, such as areas around holes and regions of load application, and also at areas where there are high rates of change of stress. Smallest elements are usually placed at such regions to track these with reasonable accuracy. When moving from areas of high element-density to low density areas, the elements should be graded in size, as shown in Fig.2, so that, ideally, the strain energy content of each is of the same order.

The shape of the elements should be as regular as possible, that is, triangles as near equilateral as possible and quadrilaterals as near square as possible. Elements with large aspect ratio (the ratio of the shortest side to the longest side of the element) should be avoided because such elements can cause ill-conditioning of the coefficient matrices. Fig.3 gives examples of elements with undesirable shapes.

The order of an element, e.g. linear, quadratic or cubic, should be chosen so as to get results of greatest accuracy while keeping computational cost reasonably low. A high-order element should be avoided in regions where a low-order element can yield acceptable results. Moreover, combining elements of different order should generally be avoided since it violates continuity along element interface, as shown in Fig.4. On the other hand, combining elements of different types, e.g. connecting plate bending elements to beam elements, and connecting elements of different shapes like triangles and quadrilaterals, naturally arises in structural problems. Where practical, it is advisable to use elements that have the same number of degrees of freedom at the connecting node.

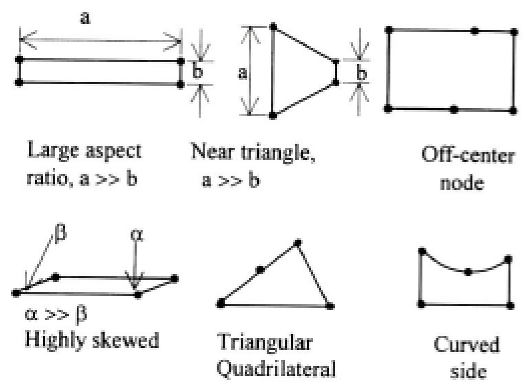


Fig.3 : Elements having shape distortions that tend to promote poor results.

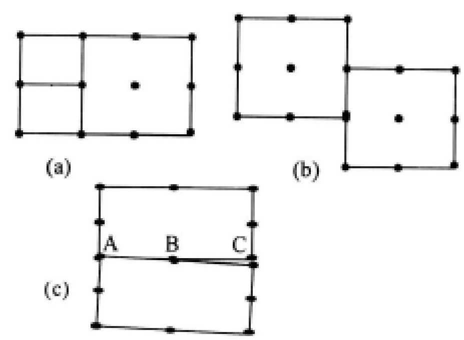


Fig.4: Poor element connections; **(a)** two bilinear elements and one quadratic element, **(b)** two quadratic elements, **(c)** two quadratic elements connected at A and B but not at C (as if to model a crack from B to C).

5 PROCESSING OF FEM DATA

The finite element computer program essentially consists of three basic parts: Pre-processor, Processor and Post-processor. The pre-processor part of the program consists of discretization of the structure, whereby the structure being analyzed is idealized as an assemblage of a number, often a large number, of discrete elements connected at the nodes. During this process the analyst decides on the number, shape, size and topology of the elements with the purpose of simulating the structure as closely as possible. The discretization process is essentially a task that requires engineering judgment and experience. The type of element chosen, for example, will depend on the type of structure and the expected pattern of deformation, or the desired accuracy of the results.

The remaining parts, processor and post-processor, are automatically processed by the computer. In the processor part, most of the steps in the finite element method are performed. Several subroutines are employed to perform different tasks. These subroutines typically involve; selection of displacement function and, numerical evaluations of the element matrices (the mass matrix, stiffness matrix, and the element force vector) from the element geometric and elastic properties. Other subroutines will involve the assembly of the overall stiffness matrix from individual stiffness matrices, and the modification of the stiffness matrix to take account of the boundary conditions. The resulting equilibrium equations are then solved to yield the nodal displacements of the elements. There are several procedures for solving these equations, such as; Gaussian elimination, LU (Upper-Lower) decomposition, Gauss-Seidel iteration, gradient techniques, etc.

Post-processing involves computation and display of the element resultants such as stresses and strains, and any other quantities associated with the element that are of importance. Much of the output can be represented graphically in a form more easily interpreted by the analyst, rather than analytically examining and comparing huge amounts of data. Displacements can be shown by amplifying and superimposing the deformed shape on the original structure. Stress levels can be easily shown on graphs of 'stress plotted against element number', or on contour plots displaying values of stress as coloured regions on the exterior surface of a model.

6 A PRACTICAL EXAMPLE

6.1 Problem Definition

FEM was used to investigate the response of a pedestrian bridge structure to both static and dynamic loads. The bridge, called The Merchants Bridge and located in Manchester, U.K., was designed by Whitby and Bird Consulting Engineers. The plan and elevation of the bridge are shown in Fig.5. The bridge's principal features are a sickle arch and a curved aerofoil box deck. The arch is tied by the deck, where the tie is not straight but curved in plan (Fig. 5(b)), thus creating a crescent shape. The arch is inclined outwards, as can be seen from Fig.6, and curves in the opposite direction to that of the deck. The deck, with its top and bottom steel plates and steel tubes on the edges, is a closed torsion box. Fig.7 shows the cross-section of the deck. Some steel plates have been used as transverse deck-stiffeners, while universal T-sections, welded onto the inside of selected top and bottom deck plates, stiffen the deck in the longitudinal direction. The support arch and the deck are connected to each other by means of tapered steel I-beams welded at both ends.

The structural behaviour of the bridge relies on a number of separate but interrelated actions. The hangers are fabricated from tapered steel I-sections, as opposed to the conventional steel cables because of the fact that, instead of transmitting loads primarily by tensile action, they serve many other equally important structural purposes. Acting as cantilevers anchored to the torsionally stiff deck, these tapered hangers, because of their bending stiffness, help to restrain the out-of-plane buckling of the arch. The hangers also help to transfer force into the plane of the arch by bending action, and this bending of the hangers is resisted by the torsional rigidity of the deck. The arch and the deck react to, and counterbalance, each other. Because of the non-uniform width of the deck, the variation of torsion along the deck is non-uniform. The transverse diaphragm stiffeners inside the deck reduce the effects of warping and shear forces caused by the non-uniform torsion in the deck.

6.2 Finite Element Models

Considering the brief account of the bridge's behaviour, given above, it is quite apparent that it would be very difficult, if not impossible, to predict the stress levels on different parts of the bridge using any of the traditional methods of structural analysis. A three-dimensional, linear elastic finite element analysis of the bridge was performed using a commercial program called ABAQUS, version 5.4, developed in the U.S.A. by Hibbitt, Karlsson and Sorensen. Its companion pre-processing software called PATRAN was used for data preparation.

Two different models of the bridge were built and analyzed. The models differed in the type of elements used to build the bridge. The first model employed simple linear elements such as two-noded bar elements and four-noded quadrilateral elements. The second model employed quadratic elements, whether straight-edged or curved. The types and behavior of these elements are described below.

In model 1 the curved shape of the arch was modeled as a sequence of short straight segments rigidly connected at the ends. Two-noded, linear pipe, beam elements, referred to as PIPE31 in ABAQUS, were used. A PIPE31 element is a one-dimensional line element in space that has a stiffness associated with the deformations of the line (the beam's axis). These deformations consist of axial stretch, bending and torsion. Each hanger was modeled as a series of four short segments of varying cross sectional properties.

Each segment was a uniform, two-node bar element with "I" cross section, and assigned properties of 'beams in space' of type B31 in ABAQUS. All of the deck components and the plate stiffeners were modeled with 2-dimensional, thin shell quadrilateral elements, ABAQUS type S4R5. This meant that the curved circular tubes had to be modeled as six-sided prisms, made of rectangular flat-surface elements as shown in Fig.8. This model comprised a total of 1220 elements and 927 active nodes, representing more than 4000 degrees of freedom.

The elements in model 2 were selected with the aim to improve the accuracy of the results by simulating the real structure as closely as possible. In order to achieve this goal, the use of more complicated elements was inevitable. However, the drawback in using such elements was very well understood, and a balance between accuracy and cost had to be reached. After a few tests, quadratic elements were found suitable to yield acceptable results within a reasonable time.

The arch rib in model 2 was modeled with three-noded quadratic beam elements in space, ABAQUS type B32. These elements were able to give the arch a curved profile which could not be obtained from two-noded beam elements which were selected for the first model. B32 elements use quadratic interpolating polynomials and were, therefore, expected to give more accurate results as compared to two-noded elements. The penalty in using these elements was the increase in computational time (and therefore higher costs) resulting from the higher number of degrees of freedom of the system. The top and bottom deck plates, and the plate stiffeners, were modeled with eight-noded, doubly curved thin-shell elements, referred to as S8R5 in ABAQUS. The shell elements in ABAQUS provide full three dimensional analysis capabilities, with three translational and three rotational degrees of freedom at each node. Their formulation accounts for bending, transverse deformation, membrane stretching and shear, but ignores transverse shear deformation [7]. Like in the case of the arch, these quadratic elements were expected to predict stress levels with better accuracy than the linear elements used in the first model. The deck-tubes were also modeled with eight-noded doubly curved thin-shell elements, S8R5. These elements, because of their mid-side nodes, were able to give the curvature required to describe the shape of the tubes, as shown in Fig.9. The hangers were assigned properties of 'beams-in-space', type B31 in ABAQUS. In total, this model comprised 1333 elements and 3067 active nodes, representing more than 15000 degrees of freedom.

Relevant geometric and material properties were assigned to each element. And finally, the boundary conditions were imposed at the appropriate notes to represent the support conditions in the real structure.

6.3 Results of the Work

The FE analyses gave an insight into the stress distribution on the bridge components and identified critical regions. Static analysis showed that wide ranges of torsion and shear stresses develop in certain components of the bridge. A pattern of the von Mises stresses on the surfaces of the elements (when the bridge was loaded with the most critical live load) was produced [4]. The figure of von Mises stresses cannot be shown here because it relies on different colours to display stress levels. Values of deflections of different points of the bridge were also obtained. Fig.10 shows the displaced bridge structure superimposed onto the original structure. The original and displaced structures were shown in different colours though it is not the case in Fig.10 because colour is not available here. From the dynamic analysis of the model, the lowest ten predicted values of resonant frequencies and the corresponding displacement mode shapes were obtained, these are shown in Fig.11[(a) to (j)]. Again, the original and displaced structures were shown in different colours for clarity. As can be seen from the figures, the analysis suggested the possibility of several different modes of resonant vibration of the bridge, most of which involved vertical movement of the deck. Torsion of the deck and lateral vibration of the arch were suggested by higher modes of vibration, modes 6 and 7 respectively. The highest modes, that is, modes 8, 9 and 10, suggested some localized plate vibrations in the deck elements.

6.4 Comparison of the Models

Table 1 shows the values of maximum deflections obtained in the analysis of the two models. It is apparent from this table that the values obtained from model 1 were lower than those obtained from model 2. Table 2 shows the maximum values of element stresses for both models. Similarly, this table gives larger values of stress levels in model 2 than in model 1. However, the critical values of both deflections and stresses occurred at corresponding points of the bridge, which suggested that the overall behavior of the bridge was consistent.

The differences in the values obtained were attributed to the differences in the elements used to build the two models. While both models were built with elements of similar shapes, their orders were different. Model 1 was built with linear elements while model 2 was built with quadratic elements. The linear elements incorporated in model 1 underestimated both the deflections and stresses. While it is not easy to ascertain which model gives the more accurate results, it would be safer to base design on the more critical values. In general, high-order elements approximate the deformations and element resultants with better accuracy than the lower-order elements because of more variables in their interpolating polynomials.

Consequently, the capacity to select the right elements for the right problem cannot be more emphasized. In this problem, only the order of the elements was varied. Elements of other shapes, e.g. triangles, could have been incorporated.

7 CONCLUSION

A brief description of the finite element method and modeling guidelines regarding choice of elements were given. Advantages of the method as a major tool of analysis and design in complex structural systems have been presented. A practical example was employed to illustrate the ease of using FEM programs to carry out stress and modal analyses of structures in practice. The example has demonstrated that the FEM is effective in identifying critical regions in structures, and in giving insight into the stress distribution over the entire structure. The importance of selecting the most suitable elements for the given problem was also demonstrated by the example. The FEM was also efficient in predicting the mode shapes and their resonant frequencies. All these analyses were carried out in a relatively short period of time, and at very low cost.

8 RECOMMENDATIONS

The rapid development of computer hardware and the widespread use of commercial finite element analysis software have greatly increased the expectation of engineers worldwide to solve large-scale structural analysis problems and, hence, to ascertain optimal design. It is recommended here, therefore, that all practicing structural engineers should learn the fundamentals of this method. Botswana Institution of Engineers, the University of Botswana's Faculty of Engineering, and companies that sell relevant engineering software, should arrange seminars and workshops aimed at training practicing engineers in this powerful method of analysis and design. It is vital that the analyst has a clear understanding of the method before attempting to use it to solve engineering problems. It is also important, however, that engineers are familiar with alternative methods of analysis and should be able to use them to predict the behavior of structures.

9 REFERENCES

1. Xanthakos, P.P., "Theory and Design of Bridges", John Wiley and Sons Inc., New York, 1994.

2. Zienkiewicz, O.C. and R.L.Taylor, "The Finite Element Method: Volume1 Basic Formulation and Linear Problems", Forth Edition, McGraw-Hill Book Company, London, 1967.

3. Astley, R.J., "Finite Elements in Solids and Structures", Chapman & Hall, London", 1992.

4. Oagile Jomo Kanyeto, "*Static and Dynamic Load Analysis of the Merchants Bridge, Castlefield, Manchester*", M.Sc. Dissertation, Department of Civil and Structural Engineering, UMIST, U.K., October 1996.

5. "ABAQUS/Standard User's Manuals, volumes 1 and 2", Hibbitt, Karlsson and Sorensen, Inc. U.S.A., 1995.

6. "ABAQUS/Post Manual", Hibbitt, Karlsson and Sorensen, Inc. U.S.A., 1995.

7. "ABAQUS Theory Manual", Hibbitt, Karlsson and Sorensen, Inc. U.S.A., 1995.

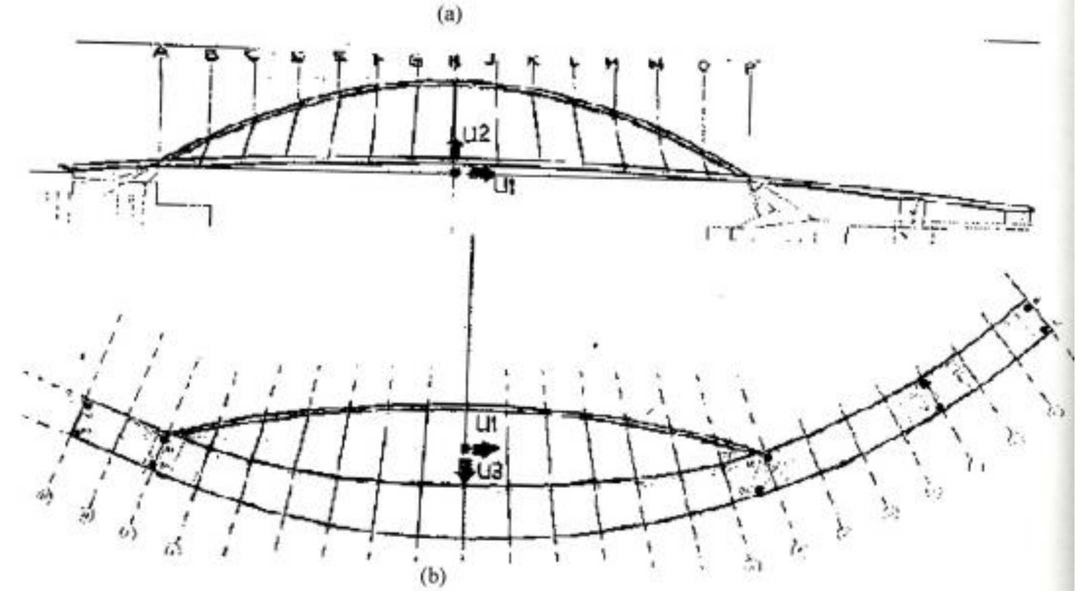


Fig.5: **(a)** Elevation and, **(b)** Plan view of the Merchants Bridge

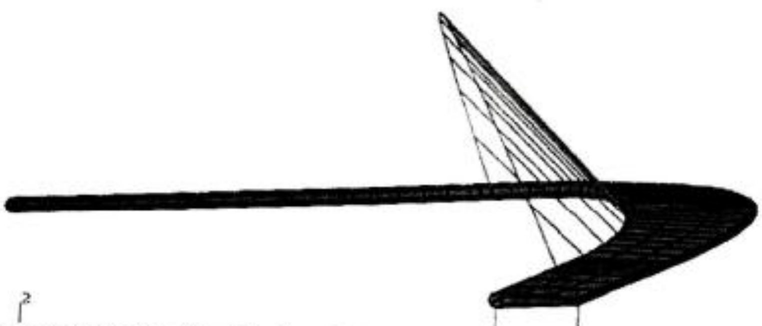


Fig.6 : Side elevation of the Merchants Bridge showing curvatures of the Arch and the Deck.

Elements	Normal Stress		Shear Stress		Torsion	
	Mod 1	Mod 2	Mod 1	Mod 2	Mod 1	Mod 2
Deck	108	196	51	91	1500	1830
Hangers	186	204				
Arch	95	99	17	9		

Table 2: Maximum Element Stresses (N/mm^2). Torsion in units of kNm.

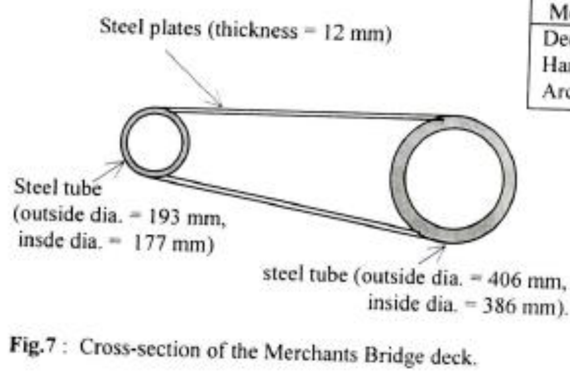


Fig.7 : Cross-section of the Merchants Bridge deck.

	Axes	Model 1	Model 2
	U1	2.8	3.0
U2	25	31	
U3	21	30	

Table 1: Maximum Nodal Deflections (cm)
***See Fig.5 for orientation of the axes.*

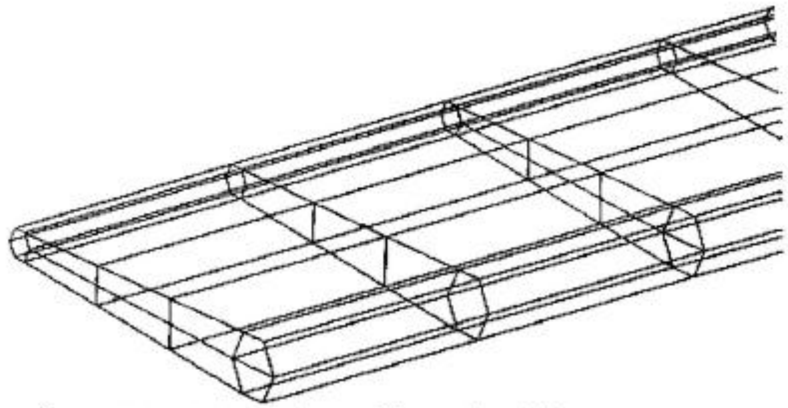


Fig.8: Part of the deck showing shapes of elements in model 1.

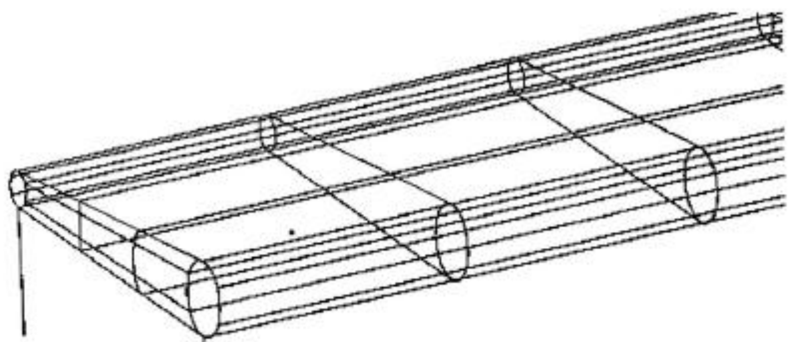


Fig.9: Part of the deck showing element shapes in model 2

Fig.10(a): Longitudinal view of the Bridge showing the deflected shape (due to working loads) superimposed onto the original structure.

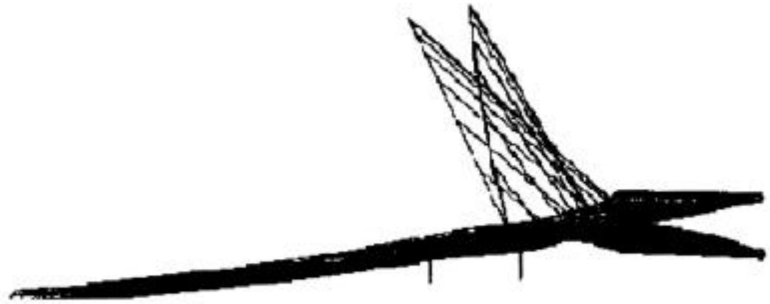
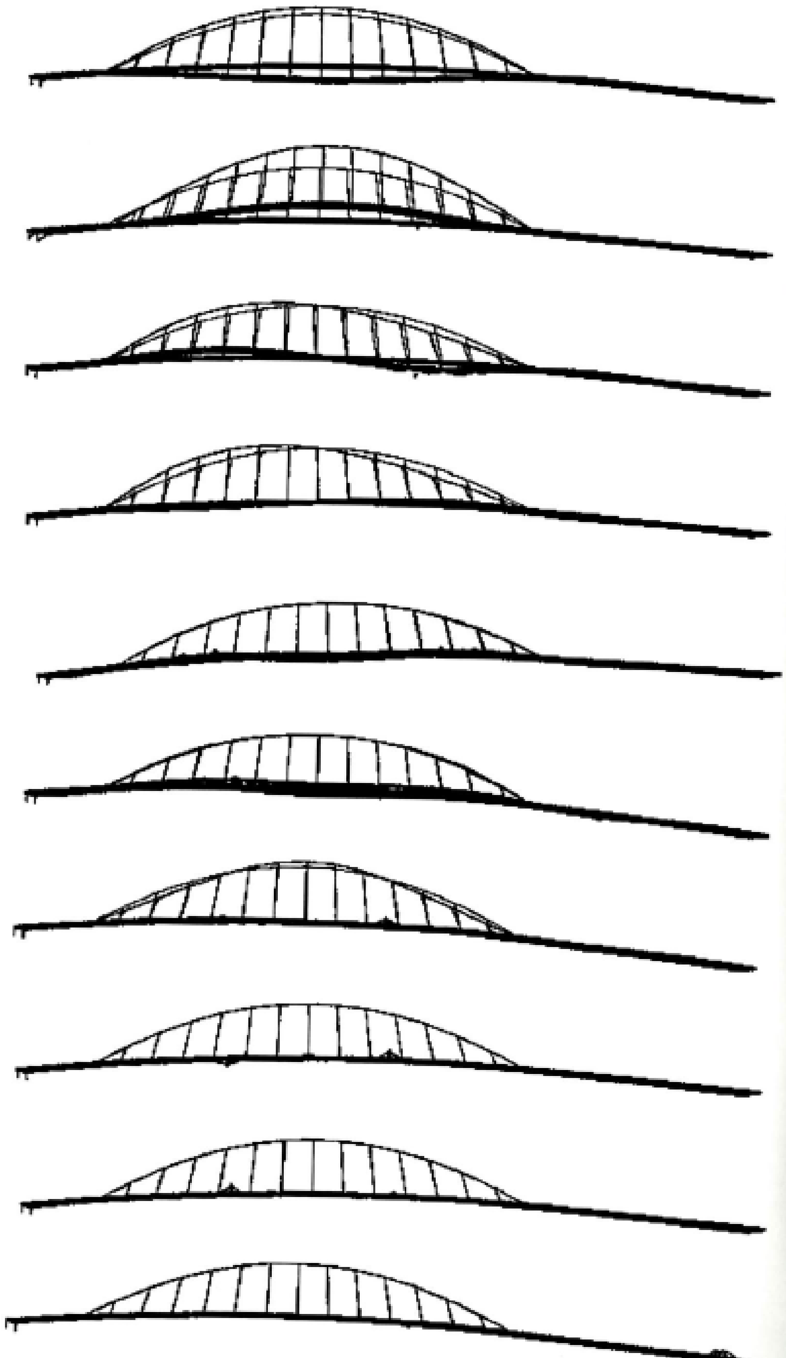


Fig.10(b): Side view of the Bridge showing the deflected structure superimposed onto the original one.



(a) Eigenmode 1, Frequency = 1.30

(b) Eigenmode 2, Frequency = 3.25

(c) Eigenmode 3, Frequency = 3.67

(d) Eigenmode 4, Frequency = 4.08

(e) Eigenmode 5, Frequency = 6.78

(f) Eigenmode 6, Frequency = 7.48

(g) Eigenmode 7, Frequency = 8.49

(h) Eigenmode 8, Frequency = 8.53

(i) Eigenmode 9, Frequency = 8.60

(j) Eigenmode 10, Frequency = 8.78

Fig.11(a) to **(J)**: Mode shapes and the corresponding resonant frequencies.