# Application of Finite Element Method to Problems in Power Plants Construction

*by*

**R. Natarajan***

## Introduction

In the design of power plants one may encounter situations wherein the simple method of stress analysis will not be adequate due either to complexity of loading and/or to irregular geometry. Finite element method provides solutions for these type of situations. In the present paper application of this method to two specific problems encountered in the construction of a power plant is discussed. These examples serve to illustrate the versatality and usefulness of the method in solving complex problems.

The two problems in which the finite element method is used for analysis are as follows :

Normally in the design of the lining in a power tunnel the reinforcements required are calculated using the simple theory of elasticity solutions. In the present context the analysis becomes complex due to the presence of a band of weak material at certain cross-sections of the rock through which the tunnel passes. Because of this weak band of material, it becomes essential to analyse these cross-sections so that the correct amount of reinforcements can be used in the lining of the pressure tunnel at these cross-sections.

It has been observed that the foundation of a power house consists of large number of alternate layers of clay stone and sand rock. It becomes necessary to ascertain whether there will be any differential settlement between the centre line of the penstock anchor after the completion of *the first stage of concreting* the foundation and, the centre line of the inlet to the turbine after the completion of the second stage build up of the power house. During the second stage, the super structure load, turbine load, load due to water hammer, pressure, crane load etc., have to be considered. Thus the problem involves the analysis of deformation of a medium which is nonhomogeneous.

## General Description of the Method

In short, the basis of the finite element method is the representation of of a body or a structure by an assemblage of subdivisions called finite elements. These elements are considered to be interconnected at joints which are called nodes or nodal points. Simple functions are chosen to

---

*Assistant Professor, Applied Mechanics Department, Indian Institute of Technology, Delhi, New Delhi-110029

*The paper is open for discussion till the end of August 1976.*

approximate the distribution or variation of the actual displacements over each finite element. Such assumed functions are called displacement functions or displacement models. The unknown magnitudes or amplitudes of the displacement functions are the displacements at the nodal points.

A variational principle of mechanics, such as the principle of minimum potential energy, is usually employed to obtain the set of equilibrium equations for each element. This principle states that of all possible displacement configurations a body can assume which satisfy compatibility and the constraints or kinematic boundary conditions, the configuration satisfying equilibrium makes the P.E. assume a minimum value.

The *equilibrium equations for the entire continuum* are then obtained by combining the equations for the individual elements in such a way that continuity of displacements is preserved at the inter-connecting nodes. These equations are modified for the given displacement boundary conditions and then solved to obtain the unknown displacements. In many types of problems, the desired solution is in terms of the strains or stresses and so additional calculations may be necessary.

*Iso-Parametric Elements*

It has been frequently demonstrated in the past that for a given total number of degrees of freedom in the structure, accuracy is increased for larger elements with a greater number of degrees of freedom. The process of idealisation could be extended without limit but for the fact that the larger elements can no longer follow the boundary (when curved boundary exists) in the problem to the same extent as the smaller ones. To overcome the geometric difficulties presented by large elements, curved sides are essential. If complex elements with sides capable of taking on the boundary curvatures are possible then indeed progress can be made. Such a step forward has been achieved by the introduction of various isoparametric element families.

The isoparametric concept, that is, to use the same interpolation functions for both the coordinates and the unknowns facilitates the formulation of curved elements. The important step as in the case of all finite elements, is to choose a shape function prescribing the variation of displacements (or coordinates) in terms of appropriate nodal values. Shape functions should also be (1) continuous within the element and across interelement boundaries and (2) reproduce conditions of constant strain exactly. In most cases these are derived from a suitable polynomial. The polynomial is chosen so that the dispalcements along the edges are linear, parabolic, cubic and so on, giving a unique variation of displacements in terms of the nodal values on that edge. The number of terms in the polynomial usually equals the number of nodes in element.

Let $\phi$ be the quantity to be interpolated, this may be a coordinate $x$, $y$, a displacement $u$, $v$, the temperature or any other quantity prescribed over the element, in terms of its nodal values. The quantity to be interpolated can be written down as :

$$\phi = N_1\, \phi_1 + N_2\, \phi_2 + \cdots\cdots\cdots \qquad ...(1)$$

where $N_i = N_i\,(x, y)$ is the interpolation shape function taking a value of unity at the node $i$ and zero at all other nodes.

Using precisely the same polynomial terms but now with the local natural non-dimensional $\xi$ and $\eta$ coordinates, with a range of +1 and −1with-in such elements, instead of the $x$ and $y$, the general shape elements as in Figure 1b can be formulated. The basic shape of the sides of these elements are straight and since the more general shaped elements are derived from this, these are called 'parent' elements. The parent element with the cartesian system oxy as its reference is shown in Figure 1(a).

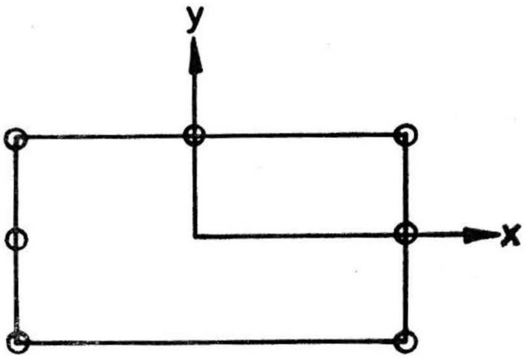


**FIGURE 1(a). Rectangular Parent element**

The position within the new element is determined by the curvilinear coordinates $\xi$ and $\eta$ (Figure 1b). This by itself does not define the mapping relationship needed. This will be provided by the basic 'isoparametric' definition. We can thus write for a deformed element,

$$u = N_1 u_1 + N_2 u_2 + \dots\dots\dots$$
$$v = N_1 v_1 + N_2 v_2 + \dots\dots\dots \qquad \dots(2)$$

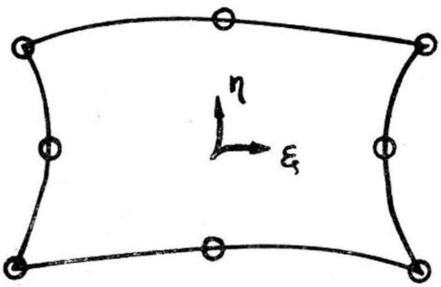


**FIGURE 1(b). Curvilinear Quadrilateral element**

together with

$$X = N_1 X_1 + N_2 X_2 + \dots\dots\dots$$
$$Y = N_1 Y_1 + N_2 Y_2 + \dots\dots\dots \qquad \dots(3)$$

with, in two-dimensional cases

$$N_i = N_i (\xi, \eta) \qquad \dots(4)$$

If the shape functions are based on the parent element definition then not only will the compatibility of displacements be satisfied on element interfaces but an original fit of these surfaces will be ensured Eragatoudis et. al. (1968).

Similarly it can be shown that if constant strain conditions were obeyed by the original parent functions this will be preserved in the distorted elements.

The formulation of displacement model element characteristics such as stiffness etc., are easily available (Zienkiewicz, 1971). The stiffness matrix of an elemenl is defined, for instance, as

$$\left[ K \right]^e = \int_v B^T D B \, dv \qquad \ldots(5)$$

in which the $[B]$ matrix relates the strains $\{\epsilon\}$ to the element nodal displacements.

$$\left\{ \epsilon \right\} = \left[ B \right] \left\{ \delta \right\}^e \qquad \ldots(6)$$

and $[D]$ is the elasticity matrix giving stress-strain relationship. All integration of the element, Legendra-Gauss points which facilitates the speedy formulation and programming of a complicated element like the curved sided elements, for which closed form integration is complicated, in a relatively short time.

In the present context, the elements need to be curved sided, from the point of view of easy generation of data. Further in both the problems, the continuum is made up of more than one material and hence the stress distribution need not be uniform. The continuum considered for analysis in both the problems is quite large. To represent this small number of higher order elements will always be best. Thus from these considerations, quadratic iso-parametric elements have been chosen in idealising the problem. A 3 x 3 Gauss integration rule has been chosen for the integration.

**Solution Method**

The next step in the finite element method is the assembly of all stiffness equations. The stiffness matrix $[K]$ of the complete assemblage relates the nodal loads $\{F\}$ acting on the structure to the corresponding nodal displacements $\{\delta\}$ by

$$[K] \cdot \{\delta\} = \{F\} \qquad \ldots(7)$$

The stiffness matrix, may be characterized in general as :

(1) symmetric, (2) banded, (3) positive definite, and (4) sparsely populated.

Algorithms which utilize either iterative methods or direct methods for the solution of equations with these properties are well known. Earlier applications of the finite element procedure were based predominately on iterative methods in which property 4 was utilised so that the solution could be obtained while working entirely in the high speed core memory. In the more recent years, however, direct solutions based on Gaussian elimination have become the preferred solution procedure due to their overall economy and ease of applicability.

With the development of iso-parametric elements, which have nodes along sides, the size of stiffness matrix of each element becomes large and hence cannot be fully assembled and stored in core. Further the assembled matrix will be sparsely populated. Front solution technique is used to solve such type of equations. In this method, due to sparsness of the coefficient matrix, only a small amount of the matrix has to be calculated before forward elimination of a variable corresponding to a row in Equation 7. This method is geared to elimination based upon elements. A variable becomes active on its first appearance and is eliminated immediately after its last. After the elimination process, data pertaining to the variable is stored on a file and the row is freed. Consequently the total core required for the method is very small compared to the conventional methods.

Figure 2 shows a typical case, where each node is coupled to seven others in a rectangular element. When the element stiffness matrix of element 1 is assembled, the equations pertaining to the nodal variables at nodes 1, 2 and 8 are stored on the magnetic tape since they have appeared for the last time. At the same time these variables are eliminated from rest of the equations. Now while assembling the stiffness of element 2, the spaces occupied in the core by variables at nodes 1, 2 and 8 are also used. Further after the assembly of element 2, equations pertaining to nodes 12, 13, 19, 23, 24 are stored on to the tape and corresponding area in the 'assembly' matrix

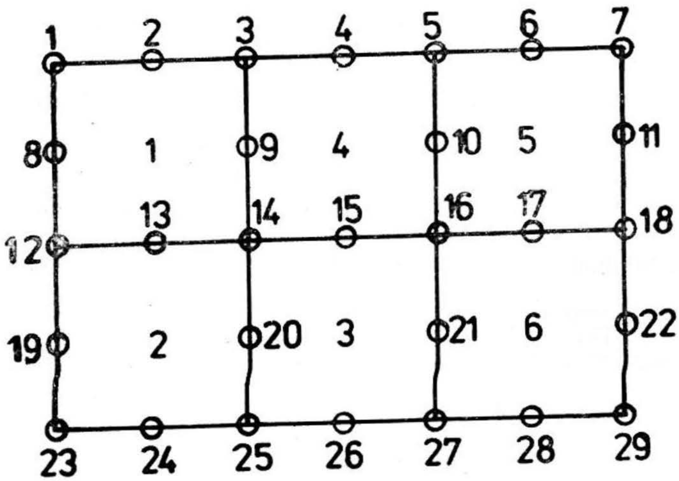


FIGURE 2. Front Solution Method

is freed. The equations pertaining to nodal variable at 3, 9, 14, 20, 25 are also modified at the same time. Thus as element by element is taken into consideration the front advances from 3, 9, 14, 13, 12 to 3, 9, 14, 20, 25 etc. Finally when stiffness of element 6 is assembled, the equations corresponding to nodal variables are stored one by one until by back substitution, one of the variables can be calculated. Now the stored equations are called back into memory from the tape unit in the reversed order in which it has been stored and the unknown variables determined.

### Practical Aspect of the Computer Algorithms

The program for the front solution method given by Irons (1970) was not used but a program based on a description contained in Irons (1966) was programmed and developed independently (Natarajan, 1975)

Suitably partitioning the Equation 7 as

$$\begin{bmatrix} M_{KN} & M_U \\ M_U^T & M_{UN} \end{bmatrix} \begin{Bmatrix} \delta_{KN} \\ \delta_{UN} \end{Bmatrix} = \begin{Bmatrix} F_{UN} \\ F_{KN} \end{Bmatrix} \quad \ldots (8)$$

where $\delta_{KN}$ and $\delta_{UN}$ are known (prescribed) and unknown nodal variables. The corresponding unknown and known generalised forces are $F_{UN}$ and $F_{KN}$. Expansion of Equation 8 gives

$$M_{KN}\, \delta_{KN} + M_U\, \delta_{UN} = F_{UN}$$

and $M_U^T\, \delta_{KN} + M_{UN}\, \delta_{UN} = F_{KN}$ ... (9)

Re-ordering $M_{UN}\, \delta_{UN} = F_{KN} - M_U^T\, \delta_{KN}$ ...(10a)

and $F_{UN} = M_U\, \delta_{UN} + M_{KN}\, \delta_{KN}$ ...(10 b)

From Equation 10a the unknown nodal variables can be calculated by an extension of Gaussian elimination process with the modified right hand side vactor $F_{KN} - M_U^T\, \delta_{KN}$. In the backward elimination stage, as the elements of $\delta_{UN}$ are explicitly known, the unknown reactions $F_{UN}$ can also be found.

### Example 1

This example shows an interesting large scale application of the isoparametric elements. The idealisation of curved boundaries, treatment of materials of different property, the simple method by which the continuum under analysis can be extended with minimum extra input data, are brought out in this example.

The problem dealt with here, is the stress analysis for the design of the lining of the power tunnel in a region where the tunnel crosses a fault zone existing in the rock as shown in Figure 3. Since the continuum is made up of three different materials : concrete, rock, and the fault zone materials, closed form solutions are not possible. Finite element method offers an easy and accurate solution to this problem.

The continuum considered for the analysis is limited up to one and half times the diameter of the tunnel from the centre of the tunnel. This limit imposed on the extent of the continuum analysed has been arrived at from the study of the stresses near the external boundary where the effect of tunnel is negligible. This is divided into a number of quadratic isoparametric elements as shown in Figure 4. While dividing the continuum into number of elements, care is taken to effect a smooth change in displacement when two elements of different materials are connected together. The nodes on the extreme outer boundary are assumed fixed.

The stiffness of the elements are computed and the solution of the equations is done using the front solution method. The load on the tunnel is due to the unit internal pressure. The displacements are computed at all nodes in the continuum under these conditions. Using this, the stress $\sigma_X$,

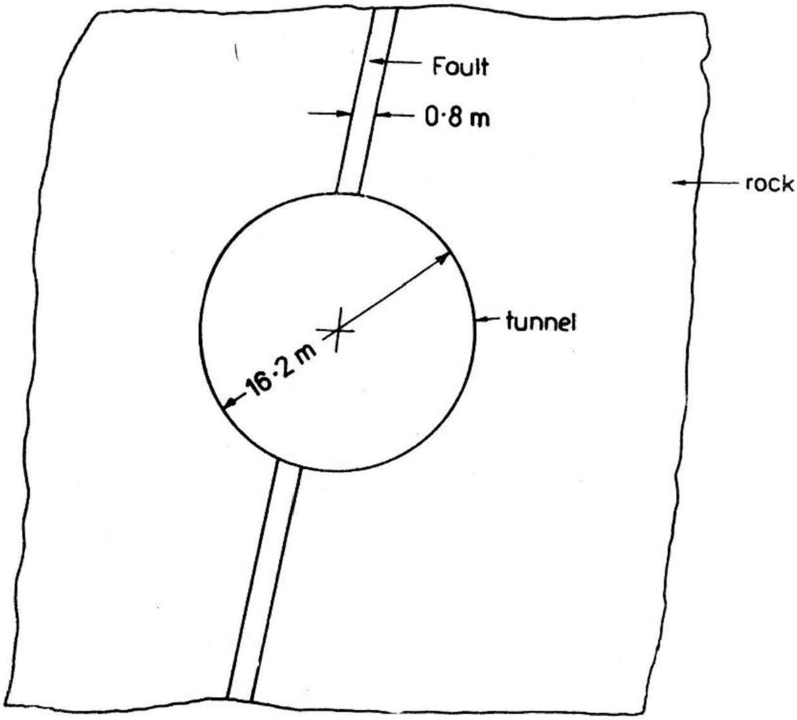


FIGURE 3. Tunnel in a fault zone

$\sigma_Y$ and $\tau_{XY}$ are computed at the integrating points and plotted and a representative distribution of the stresses is as shown in Figure 4.

From the stress plot it is concluded that the maximum tensile stress occurring in the concrete lining is near the place where the fault crosses it and is 40 per cent more than the one which occurs when the materials surrounding the tunnel is a homogeneous one. The maximum tensile-stress in the concrete lining making 90° with the fault zone is only about 10 per cent more than the homogeneous case. A further study is conducted by radially extending the continuum analysed by 10 metres. Due to the front solution method, the data for the new elements alone have to be prepared and the core required for the solution is increased only marginally. The results show that the maximum tensile stress in the concrete lining near the fault zone is 60 per cent more than the one which occurs when no fault exists. In this study the maximum stress at the extreme boundary elements is found to be about 18 per cent of the internal pressure in the tunnel. The corresponding maximum stress in the first analysis is 33 per cent. Since the stresses at the extreme boundary elements are to be zero, by interpolation the maximum tensile stress values in the lining can be obtained.

The number of elements used is 261. The core required is approximately 25 K word length and the computer time taken is about 1 hour 43 mts in IBM 360/44 system.

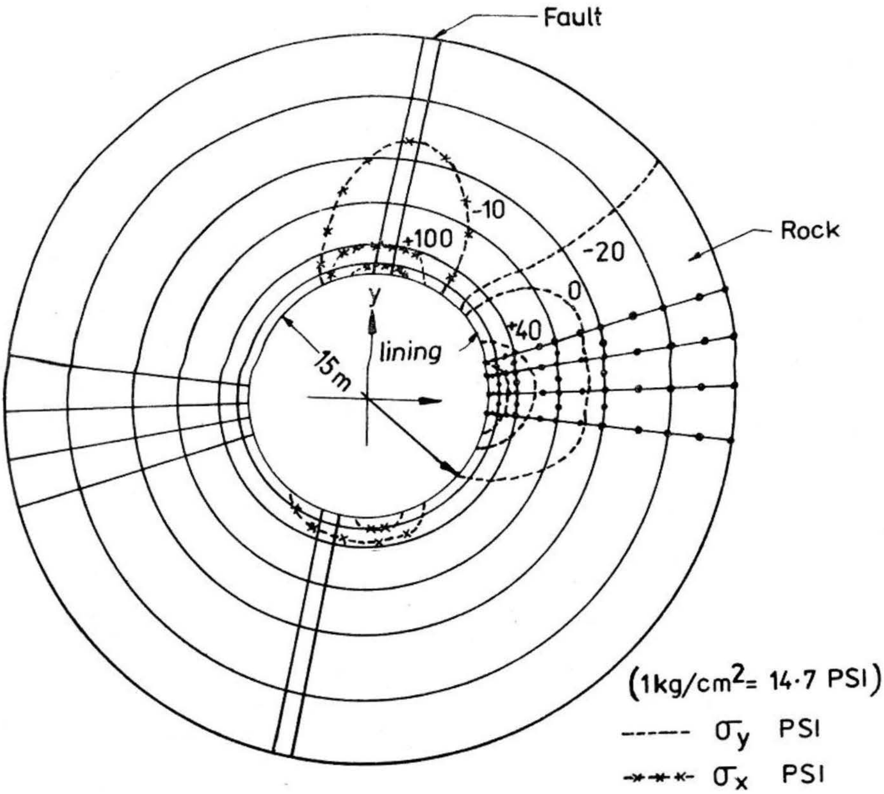


FIGURE. 4. Finite Element idealisation and stress distribution

**Example 2**

This deals with the differential settlement between a power house and penstock anchors due to the foundation characteristics. Over the completed first stage of concrete, the loads due to second stage of concreting, dead weight of turbines and generators, fully loaded crane weight, load due to water pressure etc. will be acting. Since the foundation is made up of layers of clay stone, sand stone, sand rock and silt stone, it is expected that there will be a differential settlement between the power house block and the penstock anchors.

This study has been done using two dimensional isoparametric elements. The distance between two power units of the generator has been given as 18.3 metres, and this represented the thickness of the elements idealising the foundation. Thickness of elements representing the superstructure has been taken equal to the thickness of the concrete at the corresponding point. A smooth change in the thickness of the elements has been adopted to avoid discontinuity in the geometry. A depth of about 45 metres below the first stage concrete has been considered in the analysis so that the loads acting on the foundation would have dispersed evenly within this depth. The various loads considered were: dead weight of the structure, crane loads at different positions of the crane, load due to water in penstock and turbine, load due to maximum tail water level. A representative sketch of the finite element distribution is shown in Figure 5.

The analysis of the structure with the foundation has been conducted with two types of loading which represents the extreme conditions of loading on the super-structure. In the first case, the dead weight of the structure and machine loads have only been considered with the main crane load acting at its extreme right position. This creates a clockwise moment on the super structure. In the second study in addition to the above loads, the load due to water in the penstock and turbine have been considered. Further the load due to maximum tail water level and the load of main crane occupying its extreme left position have been considered. Thus these loads create an anticlockwise moment on the main structure.

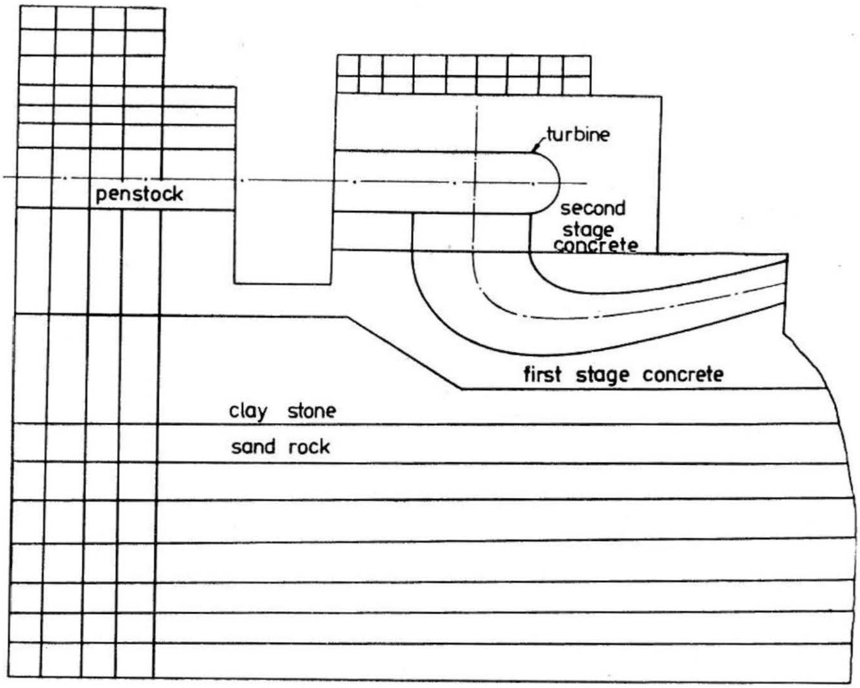


**FIGURE 5. Representative diagram**

From the study of the displacements from these two analyses the differential settlement between the main structure and the anchor block is estimated and found to be negligibly small. In the present study 321 elements are used. The core requirement is of the order of 28 K word length and the time taken is 1 hr 50 mts in IBM 360/44.

### Advantages in the Front Solution Method with Reference to the two Problems

If the original mesh pattern is too coarse in some region, a soft rubber or a black pen is sufficient to alter the drawing. For example in the problem of the pressure tunnel analysis, when larger area was included in the analysis of the continuum data was prepared only for the additional elements included in the analysis. But in the case of banded algorithm, extensive re-numbering may be necessary to preserve a small band width. A cylindrical network as in the first problem requires some manipulation in numbering the nodes for an optimum bandwidth, while using a banded solution techique. The frontal solution in its nature takes care of this situation. In frontal technique the elements are presented in a certain order, which is critical, just as the node numbering is critical in a band algorithm. In fact the ordering of the variable is irrelevent to the frontal technique. If the same structure is to be analysed for different loading conditions for example in problem 2, the analysis of the structure was conducted for (*i*) extreme positions of the crane loading, (*ii*) with and without water loads—the completed equations of all the variables in the structure are available in a tape unit. It is only necessary to update the right hand sides of the equations, and solve for the unknowns by back substitution.

## Conclusions

The two problems analysed are quite difficult to handle using conventional strength of material approach. It is seen that the finite element method—using isoparametric elements with front solution method—offers a powerful tool for the solution of such type of complex problems. The operations encountered in the method are systematic and well defined thus amenable to easy computer programming.

Using this method for the first example, it has been shown that the reinforcements in the tunnel linings near the fault zone have to be more than normally required. From the study of the second problem, it has been shown how to obtain the differential settlement in a huge structure like a power house using a relatively simple technique.

## Acknowledgement

The two problems studied were done under a consultancy scheme of C.W.P.C. New Delhi. The author wishes to thank C.W.P.C. for allowing to publish this matter.

## References

ERGATOUDIS, I, B. M. IRONS and O. C. ZIENKIEWICZ (1968) 'Curved, isoparametric quadrilateral elements for finite element analysis'. *International Journal of Solids Structures*, Volume 4, pp. 31-42

IRONS, B.M (1970), "A frontal solution program for finite element analysis'. *International Journal for Numerical Methods in Engineering*, Volume 2, pp, 5-32.

ZIENKIEWICZ, O.C., (1971), "*The finite element method in Engineering Science*", Mc-Graw Hill.

IRONS BM, (1966) "*The GSS front Solution*", systems and programming specifications 0021, Rolls-Royce Ltd., England.

NATARAJAN, R. (1975), "Front solution program for transmission tower analysis", *International Journal of Computers and Structures*, Volume 5, pp. 59-64.