The Development of an Integrated Finite Element System for the Analysis of Problems in Soil and Rock Mechanics

by

J. A. WEBSTER, B.Sc.(Eng.), M.Sc., Ph.D., M.N.Z.I.E.

Senior Engineer, Ministry of Works and Development, N.Z.

and

P. B. CLOUSTON, B.E.

Assistant Engineer, Ministry of Works and Development, N.Z.

1 INTRODUCTION

The paper gives a progress report on the development of an integrated finite element system, which will analyse various aspects of soil and rock behaviour, based on a common discretisation of the structure concerned, and will evaluate the interactions between these aspects. The system components which are currently available include a sophisticated mesh generation program, a finite element program for the linear and nonlinear analysis of structures in a state of plane strain or plane stress, or of axisymmetric structures, a program for the steady-state analysis of two-dimensional field problems, and a comprehensive plotting program which produces a visual display of the finite element mesh and the parameters computed by the stress analysis and field analysis programs.

The basic programs assume that the structural stresses and displacements and the pore pressures exhibit only spatial variation although provision is made for the quasistatic simulation of the construction process where relevant. Pore pressures appear as prescribed values in the stress analysis program, while volumetric strain rates appear as prescribed values in the field analysis program. When the time variation of each of these parameters is taken into account, the stress and field analysis equations combine to form a coupled system of matrix differential equations. Work is in progress on the development of a control program to solve the combined problem by using the separate analysis programs in parallel. This approach, which offers a high degree of flexibility, is made possible by the adoption of a common discretisation and a common database. Each program is aware only of that area of the database which is relevant to the parameters under examination.

2 MESH GENERATION PROGRAM

The mesh generation program carries out two distinct functions, the specification of a set of nodal coordinates and the automatic generation of element incidences. The node generation process is controlled by a sequence of free-format commands. The NODE command allows node coordinates to be specified directly, and is useful for those nodes which do not conform to any pattern suitable for automatic generation. The LINE command specifies a line between two points, along which nodes may be located at equal intervals or at given distances or ratios. The ARC command is similar, except that the nodes are located on a specified circular arc. The REPEAT command selects a block of nodes, which is then translated, rotated and scaled as necessary to create new nodes. The PRINT, DELETE and CHANGE commands allow the generated data to be inspected and edited. The QUADRILATERAL command generates nodes to subdivide a quadrilateral, given the coordinates of the corner nodes and the desired spacing of nodes in each direction, while a similar LSHAPE command generates nodes within an L-shaped figure. The PENCIL command reads data from an electromagnetic tracing table, together with appropriate scaling and alignment information, and generates the recorded nodes. The node sequence numbers are controlled by the order in which the nodes are generated. This is of no consequence in the present application, as the stress analysis and field analysis programs incorporate a resequencing procedure which remains transparent to the user.

When node specification is complete, element generation can begin. Each node i is considered in turn. The nearest node j is found and a side ij established. A further node k is then selected so that the points i, j and k form a counter-clockwise sequence, and the angle ikj is as large as possible. Note that the node coordinates are stored in integer form, so that the largest angle may be precisely selected. When two or more nodes subtend equal angles, the node with the highest sequence number is chosen. The potential element ijk is then checked to ensure that there is no overlap with any previously defined elements. When no such conflict exists, and when both j and k are higher numbers than i, the element ijk is generated. Whether or not ijk has been generated, the side ik then becomes the baseline, and the process is continued until the node i has been completely surrounded. The next node in sequence then becomes the central node, and the procedure is repeated.

The end result of the generation process is a mesh comprising triangular elements with the lowest possible aspect ratios. A useful reduction in computational effort is possible when a restriction can be placed on the number of remote nodes which must be checked while a node is being surrounded. Fredrick, Wang and Edge (1) specified a maximum bandwidth for this purpose. In the present method, the search is restricted to those nodes which are located within a circle of specified radius centred on the current node. When points on the mesh boundary are to be surrounded, artificial points are automatically generated if no point k can be found which subtends a sufficiently large angle with i and j. This process may result in a few surplus elements, near re-entrant corners and in holes, which must subsequently be eliminated. In general, it is not possible to determine in advance the pattern of elements which will be generated. This may not be an important restriction when adequate plotting and editing facilities are available.

Procedures are under development which will allow elastic and other properties to be assigned to elements. These procedures involve the concept of polygonal zones defined by a list of vertex points given in cyclic order. Each vertex point may be described by its coordinates, or simply by a node sequence number if it happens to coincide with a generated node. Elements whose centroids are located within a defined polygon then belong to that zone, and the various properties can be assigned by zone rather than by element. Zones may be nested, and the properties of elements in such zones are controlled by those of the innermost zone for which each property is defined. Overlapping zones are permitted only when there is no conflict between the relevant properties.

Further development will involve the use of a storage-tube display, and additional facilities will be provided for the direct graphical input of nodes and zones. The availability of an interactive plotting capability should improve still further the flexibility of the system, and should permit rapid checking of input data.

3 STRESS ANALYSIS PROGRAM

The stress analysis program uses the finite element method to predict the stresses and deformations developed in a two-dimensional structure, which may be in a state of plane stress or plane strain, or which may be axisymmetric. The effects of stress and displacement boundary conditions, concentrated loads, inertial forces, pore pressures and temperature stresses may be included. Dynamic storage allocation is used in both the stress analysis and field analysis programs, so that no fixed limits are placed on the number of nodes, elements, materials and boundary conditions involved.

The node information which must be provided includes the global coordinates, the temperature and pore pressure, the applied loads or displacements parallel to the local axes, and the angle between the local and global axes. The definition of local and global coordinate systems provides a flexible mechanism for the description of skew boundary conditions. Missing nodes are generated at equal intervals between the available defined nodes, and the associated temperatures and pore pressures are determined by linear interpolation.

The element information which must be provided includes the nodes to which the element is connected, the material of which it is composed, the construction phases during which it is added to, and subsequently removed from, the structure, the angle between the local and global axes, the element thickness (for plane stress only) and the initial stresses. Missing elements are generated by numbering the nodes at equal intervals between those associated with the adjacent defined elements, and the various parameters are determined by linear interpolation. Two nodes identify a bar element, three nodes identify a triangular element and four nodes identify a quadrilateral element. Simulation of the construction sequence is an important feature of the program, and demands a nonlinear analysis. The response of those elements in existence when construction starts is determined by an initial linear or nonlinear analysis, during which the geometry is assumed to remain constant. When no measured values are available, this procedure offers an approximation to the stresses present in the undisturbed structure. During subsequent phases, the internal structural geometry is updated in accordance with the computed nodal displacements, but the external geometry of added elements is assumed to correspond to the original description.

The information required for each material can be divided into two classes, parameters which remain constant, and parameters which are stress-dependent, and which are grouped into property sets. Constant parameters include information relating to failure criteria, thermal expansion coefficients, and lists of the values of mean principal stress and octahedral shear stress for which property sets are tabulated. The stresses within the material may be limited by a Mohr-Coulomb failure criterion. Alternatively, the material may be assumed to be laminated, so that failure is restricted to shearing between adjacent laminations. A mechanism is thus available for the analysis of rock joints, and soil-structure interfaces. Each material property set includes the elastic modulus and Poisson ratio associated with each principal direction, together with the independent shear modulus where relevant (isotropic behaviour is assumed when data is provided for only one principal direction), and the coefficient of internal friction. The actual parameters applicable to a given stress situation are calculated by a double linear interpolation procedure.

Both nonlinear material properties, and the effects of finite deformations, are taken into account by an iterative solution based on the repeated computation and application of residual nodal forces within each construction phase.

The displacements or reactions at each node, the average stresses in each element, and the element factors of safety where relevant, are calculated and printed at the end of each construction phase. At the same time, any number of plots can be generated. Nodes can be plotted and labelled, incidences can be plotted as solid or broken lines, elements can be labelled with element numbers, material numbers, or safety factors, nodal displacements, element stresses and nodal reactions can be plotted in vector form, and the displaced structure can be plotted. Separate scale factors can be selected for the geometrical coordinates and for each of the computed parameters, and plotting can be restricted to any desired region of the mesh.

4 FIELD ANALYSIS PROGRAM

Consider the case of an incompressible fluid contained in a porous saturated elastic medium. The spatial variation in the fluid pressure is governed by the following quasi-harmonic partial differential equation:

$$\frac{\partial}{\partial x}(K_x \frac{\partial p}{\partial y}) + \frac{\partial}{\partial y}(K_y \frac{\partial p}{\partial y}) + Q = 0$$

where p is the pressure in the fluid, K_x is the permeability coefficient parallel to the x-axis, K_y is the permeability coefficient parallel to the y-axis and Q is the rate at which fluid is supplied per unit area. When the solid medium deforms elastically, the fluid supply rate can be expressed as follows:

$$Q = - \frac{\partial}{\partial t} (\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y})$$

where u is the displacement component parallel to the x-axis and v is the displacement component parallel to the y-axis.

Note that, in general, the permeability coefficients may be time-dependent. When the situation at some particular instant is considered, the equation reduces to a steady-state form which can be solved by the field analysis program. The program can, in fact, be applied to a wide variety of engineering problems, including those associated with heat transfer, the distribution of magnetic and electrostatic potential, seiche development in enclosed bodies of water, and torsion in prismatic shafts.

The physical constraints associated with each region and problem are expressed in the form of boundary conditions, the most common types of which are as follows:

(a) The value of p is specified on the boundary.

(b) The following equation must be satisfied on the boundary:

$$K_x \cos \Theta \frac{\partial p}{\partial x} + K_y \sin \Theta \frac{\partial p}{\partial y} + \alpha p + q = 0$$

where Θ represents the angle between the outwards normal to the boundary and the global x-axis.

The formulation of the problem in finite element terms means that isotropic and anisotropic regions can be analysed with equal facility, and that any desired boundary conditions can be applied.

The input data for the field analysis program is very similar to that for the stress analysis program, as both programs share a common database. The fluid pressure at each node, and the average stream velocity components in each element, are computed and printed, together with suitable explanatory matter. Plotting facilities similar to those associated with the stress analysis program are available.

As explained previously, the fluid pressure appears as a load component in the stress analysis from which the nodal displacements are determined. Hence, given initial values for these displacements, for the fluid pressures and for the various elastic and permeability parameters, the stress analysis and field analysis can be operated in parallel to trace the variations in displacements and pressures with time and space. A somewhat similar procedure was carried out on an experimental basis by Thoms, Arman and Pecquet (2), with encouraging results. While experience with the present programs is limited, and the above approach is in some respects more approximate than that described by Sandhu and Wilson (3), the fact that the system components are uncoupled offers a valuable measure of flexibility, particularly when creep becomes significant.

5 MATHEMATICAL FORMULATION

The discretisation of two-dimensional elastic problems in terms of finite elements has become commonplace, and full details have been given by Zienkiewicz (4). The central assumption in the present case is that the displacements parallel to the x-axis and parallel to the y-axis vary within a triangular element, in accordance with two linear polynomials whose coefficients can be evaluated in terms of the element geometry and the nodal displacements.

The strain components within the element can be related to the spatial variation in the displacements and hence, through the polynomial coefficients, to the nodal displacements:

$$E = B\delta$$

where E is the vector of strain components in the element and B is a matrix dependent on the element geometry. Because the chosen displacement functions are linear, the strain components within the element are constant. The stress components are in turn related to the strain components:

$$\sigma = DE$$

where σ is the vector of stress components in the element and D is the elasticity matrix, the composition of which depends on whether the material is isotropic or anisotropic and on whether the structure is in a state of plane stress or plane strain or is axisymmetric. The internal work done by the stresses and distributed loads can be computed by summation over the element area and, since displacement continuity between adjacent elements is ensured by the linear displacement functions, can be equated to the work done by the external nodal forces.

Consideration of the force equilibrium at each node allows the stress analysis problem to be reduced to the solution of the matrix equation:

$$K\Delta + MP + R = 0$$

where K is the structural stiffness matrix, M is the matrix which converts pore pressures to effective nodal forces, P is the vector of pore pressures at the nodes and R is the vector of all specified forces, including those related to thermal variation and initial stresses, but excluding those related to pore pressures.

The field analysis problem can be formulated in a very similar manner, and reduced to the solution of the matrix equation:

$$Hp + S \frac{\partial}{\partial t} (\Delta) = 0$$

where H is the field stiffness matrix and S is the matrix which converts the rate of time variation of the nodal displacements into an effective rate of fluid supply or depletion.

While triangular elements are simple and versatile, Doherty, Wilson and Taylor (5) consider that quadrilateral elements which are internally handled as sets of four triangular elements with a common node at the centroid of the original element offer certain economic advantages. The degrees of freedom associated with each such element can be divided into those which are in common with other elements and those which occur only in the current element.

The element equilibrium equation can then be partitioned as follows:

$$\begin{matrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{matrix} \quad \begin{matrix} \delta_1 \\ \delta_2 \end{matrix} = \begin{matrix} r_1 \\ r_2 \end{matrix}$$

where δ_1 and r_1 are respectively the displacement and force vectors at external nodes while δ_2 and r_2 are the displacement and force vectors at internal nodes. The internal displacements can be expressed in terms of the remaining parameters:

$$\delta_2 = k_{22}^{-1} (r_2 - k_{21}\delta_1)$$

and then eliminated from the equilibrium equation, which reduces to:

$$(k_{11} - k_{12} k_{22}^{-1} k_{21})\delta_1 = r_1 - k_{12} k_{22}^{-1} r_2$$

Assembly of the complete structural stiffness matrix follows. When the external and internal force vectors, and the external displacement vector, have been calculated for each element, the internal displacement vectors can readily be computed.

Consider now the nonlinear form of the equilibrium equation:

$$K\Delta + R = 0$$

where the stiffness matrix K is a function of the displacement vector Δ. Provided that the stress pattern which corresponds to any given strain pattern can be uniquely determined, the linear elastic relationship:

$$\sigma = \sigma_0 + D\ (E - E_0)$$

can be adjusted to reflect the actual situation by altering the initial stress vector. Suppose that the linear equation:

$$K_0\Delta_0 + R_0 = 0$$

has been solved, and that the necessary adjustments to the initial stress vector have been determined. These adjustments will effectively establish an incremental load vector R_1 and the following equation must be solved:

$$K_0\Delta_1 + R_1 = 0$$

where Δ_1 represents the corresponding incremental displacement vector. At each stage in the procedure, the difference between the actual stress pattern predicted by the current strain pattern and that generated by the elastic solution is computed. The stress pattern is then redistributed elastically to restore equilibrium. The incremental load vector can be interpreted physically as the vector of unbalanced residual loads, and, as indicated in Figure 1, the procedure is continued until these loads are reduced to an acceptable level.

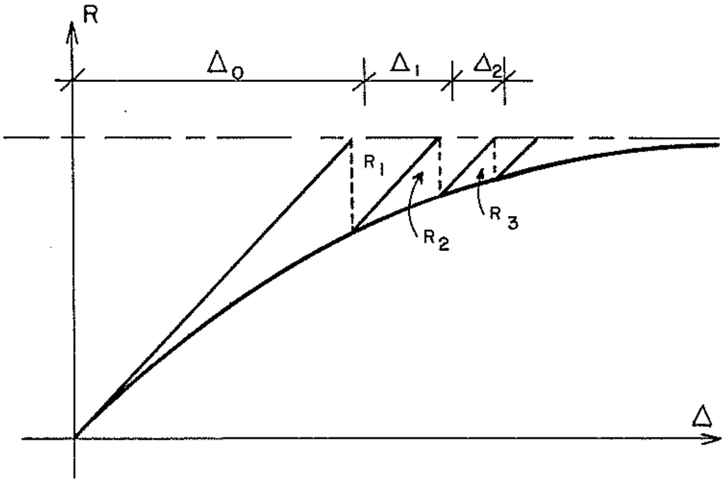

Fig. 1 Iteration pattern

The stiffness matrix is held constant during each construction phase, and is updated at the end of that phase to reflect the changes which have occurred in the structural geometry and the material properties. The procedure readily handles the situation where the stresses in certain elements are limited by prescribed failure criteria.

The typical stiffness matrix is sparse, containing many zero coefficients, banded, with the non-zero elements grouped around the leading diagonal, symmetric and positive definite. There would appear to be little doubt that, as advocated by Wilkinson (6), the Cholesky decomposition, in which a lower triangular matrix L_0 is computed such that:

$$L_0L_0^T = K_0$$

represents the most satisfactory elimination procedure for symmetric positive definite systems.

The method is particularly convenient for non-linear systems, as the repeated computation of incremental displacements can be achieved by the solution in sequence of the following equations:

$$L_0z_i + R_i = 0 \qquad \text{and} \qquad L_0^T\Delta_i = z_i$$

Since L_0 is the lower-triangular, very little computational effort is involved. The decomposition algorithm is compact, minimal data management is required, error bounds are low, full advantage is taken of symmetry and the lower-triangular matrix remains banded.

The above algorithm must be applied in a manner which takes maximum account of the matrix sparseness. One possible approach is the frontal solution technique described by Irons (7), in which the basic matrix is generated in compacted form, and related to the original matrix by a system of pointers. The procedure is directed towards an elimination based on the elements, and independent of the nodal sequence, and derives its name from the creation of an 'elimination front' which passes through the nodal system.

The usual band matrix approach is based on the assumption that all non-zero coefficients are located within a narrow band of constant width centred on the leading diagonal. The sequence in which the nodes are eliminated clearly determines the width of this band, and hence the efficiency of the procedure, and the selection of an optimal sequence involves considerable effort. A significant improvement can be made by recording the maximum offset from the leading diagonal of any actual or potential non-zero coefficient in each row, and storing the matrix coefficients, above and including the leading diagonal, which are located within the resultant variable band, in a linear array. Only a subset of the complete array need actually be resident in the main store of the computer at any given instant.

The procedure is more efficient than the constant band width approach, and the additional data management is insignificant. An important consideration is that the optimum band profile, initially narrow, then expanding monotonically, can be closely approximated by a simple algorithm. While the balance between the extra data management needed for a frontal solution and the resequencing activity needed for a variable band solution is fairly even when a single linear analysis is concerned, the fact that several steps of a non-linear analysis can share the benefits of a single resequencing step makes the variable band approach clearly superior for the present application.

6 CONCLUSIONS

The present stage of development of a finite element system with particular application to problems in soil and rock mechanics has been described in some detail. The most important system component is the common data base which serves a range of separate programs. These may operate in parallel, but continue to be available individually for problems in which only one aspect of the overall situation needs to be studied. The system is currently being adapted to the I.C.E.S. environment, where control will be achieved through the use of a suitable problem-oriented language.

The problems associated with the interaction of beam and slab elements and the currently available planar elements are now being examined. Such interaction must be taken into account in the analysis of pile groups, and foundation structures in general, and the initial facilities should become available in the near future. The current programs are so arranged that the computation of the structural response associated with an earthquake time-history is feasible, and the additional facilities needed for this purpose are included in the I.C.E.S. implementation. Provision has already been made for the assessment of element safety factors and, as the mesh generation program is developed to the point where the effort required to construct a finite element mesh becomes insignificant, the use of nonlinear elastic analyses may substantially reduce the present dependence on approximate limit-state methods for stability calculations.

The field analysis program will shortly be extended to cover the problem of partially saturated flow by the introduction of a further iterative procedure which allows for variations in the free surface profile, and which takes the storage capacity of the individual elements into account.

The present programs are restricted to two-dimensional problems, and, for various reasons, it is anticipated that such problems will continue to form an important part of the system workload. An obvious extension is to cover the case of a three-dimensional structure with a constant cross-section which sweeps a solid volume along a straight or curved generator. As discussed by Zienkiewicz and Too (8), such a structure can, with the aid of Fourier analysis, be reduced to an equivalent series of two-dimensional structures, and this approach seems likely to be useful for tunnel and slope analysis. Fully three-dimensional programs will be provided during the next phase of system development.

7 REFERENCES

1. FREDERICK, C.O., WONG, Y.C. and EDGE, F.W. Two-dimensional Automatic Mesh Generation for Structural Analysis, Int. J. Num. Meth. Eng., Vol. 2, No. 1, 1970, pp.133-144.

2. THOMS, R.L., ARMAN, A. and PECQUET, R.A. Finite Element Analysis of Embankments over Soft Soils, Symposium on Applications of the Finite Element Method in Geotechnical Engineering, Vicksburg, Mississippi, May 1972, pp.605-627.

3. SANDHU, R.S. and WILSON, E.L. Finite Element Analysis of Seepage in Elastic Media, Proc. A.S.C.E., Vol. 95, No. EM3, June 1969, pp.641-651.

4. ZIENKIEWICZ, O. The Finite Element Method in Engineering Science, London, McGraw-Hill, 1971.

5. DOHERTY, W.P., WILSON, E.L. and TAYLOR, R.L. Stress Analysis of Axisymmetric Solids utilising Higher-Order Quadrilateral Finite Elements, Report 69-3, Structural Engineering Laboratory, University of California, Berkeley, 1969.

6. WILKINSON, J.H. The Algebraic Eigenvalue Problem, Oxford, Clarendon Press, 1965.

7. IRONS, B.M. A Frontal Solution Program for Finite Element Analysis, Int. J. Num. Meth. Eng., Vol. 2, No. 1, 1970, pp.5-32.

8. ZIENKIEWICZ, O. and TOO, J.J.M. The Finite Prism in Analysis of Thick Simply Supported Bridge Boxes, Proc. Instn. Civ. Engrs., Part 2, Vol. 53, September 1972, pp.147-172.