A New Set of Single Element Tests for Accuracy Assessment in Finite Element Analysis

Wu, Yunling
Presented for the degree of M. Sc.

July 1989

Acknowledgements

The work described in this thesis was carried out in the Department of Mechanical Engineering at the University of Glasgow.

I wish to express my indebtedness to my supervisors, Dr. D. K. Brown and Mr. W. M. Mair for their valuable guidance, advice and constant encouragement throughout the work.

Many thanks are due to:

Professor Henry H. Y. Wong whose help and encouragement was most appreciated.

Dr. D. J. Burrows for his information and useful discussion.

Mrs. Yasmine Mather for her advice and help.

The staff of Robert Clark Centre of Technological Education for their help.

Many friends whose help and advice will never be forgotten.

Thanks are due to Dr. H. A. Kanol of CASA/GIFTS, Tucson, Arizona and Dr. N. Adams of NAP Engineering Microsoftware Ltd, London, for kindly making available the GIFTS software for this study.

My thanks are reserved for my family for their boundless patience, continual encouragement through the years.

Finally, many thanks are due to the grant in support from the Government of the People's Republic of China.

Summary

A set of single element tests has been devised which are based on assumed stress functions with constant, linear and quadratic stress fields. These three fields can be used in piecewise approximation of higher order stress fields. Element shape sensitivity has already been explored, but element size and location are also important sources of error and they are related to effective discretization of a structure.

A variety of errors are introduced depending on how elements are combined in a mesh. Even a very refined regular mesh in a subregion may produce poor results depending on the coupling to adjacent subregions. Several tests of different element combinations have been carried out in this work.

The technique proposed involves an initial crude model to obtain a rough assessment of displacement and stress fields. This is followed by a refined solution based on the results from the single element tests with particular attention to critical regions. An accurate assessment of the actual stress field can be attempted by the technique.

The practical application of the tests in structural analysis practice is discussed. Improved accuracy can be achieved either by mesh subdivision (so called h-extension) or the use of higher order elements (p-extension) but only the former technique is employed here. It is known that the errors are associated with the size effect and combinational effect although element shape and location are unchanged. Since errors increase rapidly with the increasing order of the stress field, the size of elements is reduced such that field representation is approximately linear in order to obtain more accuarate results.

To carry out the tests, it is necessary to know the features of available finite element software on microcomputers. In this work, the comparison of several FEA programs has been acomplished. Also, some guidelines on choosing a suitable package is given.

Contents

Notation

[B]	strain matrix
D	displacement error
D_{txi}, D_{tyi}	theoretical displacement errors for u and v at node i
D_{fxi}, D_{fyi}	FEA displacement errors for u and v at node i
D_{max}	maximum theoretical displacement over an element
E	Young's modulus
F	force
$\{F\}_e$	fictious forces acting on the element nodes
$\{F\}_{e\varepsilon_0}$	nodal force vector due to initial strains
$\{F\}_{e\sigma_0}$	nodal force vector due to the initial stresses
$\{F\}_g$	nodal forces due to any distribued external load on boundary elements
$\{F\}_p$	nodal forces due to distributed load $\{\underline{P}\}$
G	shear modulus
[J]	Jacobian matrix
$[K]_e$	stiffness matrix of the element
K_{tn}	stress concentration factor based on the net area
L	length of element
M	moment
P	interpolation function
$\{\underline{P}\}$	distributed load per unit volume
SE	total stress error
SE_x, SE_y	stress errors for σ_x and σ_y
SE_{xy}	stress error for τ_{xy}
a	radius of a hole
c	subscript for combinational effect
{g}	component of boundary pressure

i	subscript for the number of node
l	subscript for linear stress field
p	uniformly distributed load and subscript for parabolic stress field
s	subscript for size effect
t	thickness of plate
u	displacement in x direction
v	displacement in y direction
w	width of plate
x,y	coordinates
x	subscript for x direction
y	subscript for y direction
ξ,η	curvilinear coordinates
$\varepsilon_x,\varepsilon_y$	direct strains
γ_{xy}	shear strain
$\{\delta\}$	vector of nodal displacement
$\{\delta\}_e$	displacements associated with the element
$\{\varepsilon_\varepsilon\}$	matrix of total strain
θ	skew angle
φ	rotation
$\sigma_x,\sigma_y,\sigma_z$	direct stresses
τ_{xy}	shear stress
σ_{max}	maximum stress
σ_{nom}	norminal stress (based on net section)
$\sigma_{txi},\sigma_{tyi}$	theoretical stresses in the x and y direction at node i (i =1, 2, 3, 4)
σ_{fx},σ_{fy}	stresses from FEA solution in the x and y directions output from centriod of the element
ν	Poisson's ratio
ϕ	stress function

Chapter 1 Introduction

1.1 General review

The finite element method had its beginning in the area of structural analysis. It began as a numerical method of stress analysis and is still most widely used for this purpose. Also it has become useful in many other areas including heat conduction, seepage flow, fluid dynamics, and electric and magnetic fields. In the early 1940's, a mathematician named Courant (Courant, 1941) suggested an approach where the structure is divided into piecewise triangular subregions. However, the practicability of the method was limited due to the fact that there were no digital computers to do the calculation. By 1953 matrix methods (influence coefficient method) became popular in engineering circles, especially in the aerospace industry. This technique became feasible with the advent of the first digital computer which could solve large (over 100) simultaneous equations. By the mid 1960's the theory of the finite element method (FEM) was formalized in terms of mathematics and engineering mechanics. Large, general purpose FE programs emerged during the late 1960's-early 1970's period.

The finite element method is now firmly established as an engineering tool of wide applicability. No longer is it regarded as the sole province of the researcher or academic but it is now employed for design purposes in many branches of technology. One of the principal advantages of the FEM is the unifying approach it offers to the solution of diverse engineering problems.

During its early development for stress analysis problems the method relied heavily on a physical interpretation in which the structure was assumed to be composed of elements physically connected only at a number of discrete nodal points (e.g. skeletal frames). Later the application of the method to structural mechanics problems was developed through the use of the principle of virtual work and energy methods.

In engineering, physics and applied mathematics, three main areas of application of the FEM can be identified (Zienkiewicz, 1971; Desai & Abel, 1972; Gallagher, 1975; Norrie et al, 1973). These are:

Equilibrium problems in which the system does not vary with time. Examples of such problems include the stress analysis of linear elastic systems, electro-statics, magnetostatics, steady- state thermal conduction and fluid flow in porous media. The structure is first divided into distinct non-overlapping regions known as elements over which the main variables are interpolated. These elements are connected at a discrete number of points along their periphery known as nodal points.

Eigenvalue problems are extensions of equilibrium problems in which specific or critical values of certain parameters must be determined. The stability of structures and the determination of the natural frequencies of linear elastic systems are examples of such problems. In a finite element solution problem, each mode shape or eigenvector is associated with a particular frequency or eigenvalue.

Propagation problems include problems in which some time-dependent phenomena takes place. Hydrodynamics and dynamic transient analysis of elastic continua are two examples of such problems.

In each of the three areas of application, problems may contain some non-linear characteristics which complicates the analysis.

The success of the FEM as a practical design aid depends on the availability of an efficient means of solving the resulting system of linear or non-linear simultaneous equations. Clearly the existance of the computer is vital to the success of this.

In the last decades, the finite element method has become widely accepted by the engineering professions as an extremely valuable method of analysis. Both theoretical analysis and engineering design in structural analysis have made increasing use of the FEM. This application has enabled satisfactory solutions to be obtained for many problems which have hitherto been regarded as insoluble. With this powerful method, researchers and designers can simulate the real behaviour of a structure more closely. The FEM is the most powerful general analytical method now available in structural analysis since it enables continua with complex geometrical and material properties and loading conditions to be accurately analysed. The method involves extensive computations, but, because of the repetitive nature of these computations, it is ideally suited for computer programming. Early general purpose systems which appeared, included ASKA and NASTRAN.

The FEM involves discretization and it is important to note that the solution is an approximation to the 'real' behaviour. The elements (size and type) are chosen to approximate some structural behaviour which is also approximate (e.g. plane stress, thin plate bending, etc.). In any continuum the actual number of degrees of freedom is infinite and, unless a closed form solution is available, an exact analysis (within the assumptions made) is impossible. For any numerical approach an approximate solution is attempted by assuming that the behaviour of the continuum can be represented by a finite number of unknowns. The numerical processes used (e.g. integration rules, equation solving techniques) introduce approximations and numerical errors.

It might be expected that the better the simulation of the structure, the more accurate the final solution. Generally, a simpler simulation of the structure gives a cheaper solution, but it is necessary to establish practical limits of accuracy and suitability. In setting these limits, it is obviously best to compare predicated behaviour with quantities which have practical engineering significance. For general structural analysis, these include displacements, stresses, strains, energy, velocities, accelerations etc. If such a comparison shows that two different sets of approaches give a similar solution, then it would make sense to accept the cheaper and simpler method, even if the other has given a stronger physical basis.

If detailed parametric studies are to be made on certain classes of structures, then the method of analysis must be economic to use. The simplest approximations and devices must be found which give the required information as accurately as necessary. The complexity of the model should be determined by the accuracy required, and by the accuracy with which the input data is known.

In the past, the economic limitations imposed by computer costs have restricted the general use of such techniques. However, this barrier is being rapidly removed and the finite element solution is already economically acceptable for selected industrial applications. With the development of the computer and gradual decrease in the price of hardware, software based on the finite element method has become a powerful tool for design engineering and also validation. Several general purpose finite element packages available in the market encompass many analytical capabilities such as static, non-linear, dynamic, vibration, heat transfer, transient response, etc. Examples of such codes are: ABAQUS, ANSYS, PAFEC, ESDUFINE, GIFTS, MSC/NASTRAN, and COSMIC/NASTRAN. Those finite element analysis packages are widely used in mechanical, aeronautics, civil engineering and so on. Structures such as pressure vessels, bridges, offshore structures, turbines, buildings and reinforced concrete structures are regularly analysed by the method.

Currently, developments in computer technology have helped to increase the speed and in core memory size of the micro-computer. This, in turn, enables some of the analytical capabilities, which were available only on mainframes, to be introduced to micro-computers.

As computer hardware became more sophisticated (and less costly), finite element programs have been able to be run on smaller computers. Today, a finite element program is capable of solving a wide variety of problems and can be run on a micro-computer. This puts finite element analysis capabilities at the fingertips of engineers from many disciplines. A personal computer (PC) is a stand-alone smaller version of a mainframe or more traditional computer. It can be thought of as a one person work station dedicated to one function or process at a time. Since a PC was defined to be a one person work station, there is no special log-on procedure required. Once the PC is turned on the user has full control of the machine and it is consequently easier to use than a mainframe. Therefore, the micro-computer is becoming more and more attractive, particularly because of the number of software packages which have been developed to meet engineering requirements.

Analysis of a problem using the finite element method follows a standard pattern with most available codes. The first step is to clearly define the problem and plan its solution; the second step is to model the problem using a pre-processor; then compute the solution using an analysis processor; and, finally, examine the results using a post-processor. Some packages have interactive graphics which make the code user friendly. Increasing automation requires less data from the user in order to create the model, load, and boundary conditions, thus decreasing the pre-processing time. In the post-processor, the final output of the results, such as displacements, stresses, velocities, accelerations, temperatures, and failure criteria, take the form of both graphics output and data tables.

1.2 Scope and purpose

A main aim of this work is to determine the accuracy of FEA solutions in terms of element shape distortions, size effect and combinational effect. The ultimate purpose of this is to establish simple error measurement parameters and give guidelines on modelling. Another aim is to compare the merits and demerits of different finite element packages to help the user choose that most suitable one from those available (Mair, 1988; Thomson, 1988).

4-noded and 8-noded isoparametric elements were used for all the sample problems in this study. In addition, a 3-noded transition element was used in some instances.

A review of the finite element method and a description of basic steps in its use are presented in Chapter 2.

In Chapter 3 a detailed survey of available software on the IBM-PC, and some guidelines on choosing a suitable package for solving different types of problems are given. There are many packages available on the IBM-PC, each having its own features and functions (analysis type and element library). All of these are tabulated, and some typical problems are solved by several types of elements to compare the accuracy of solution and computing time.

The following chapter (Chapter 4) discusses the assessment of the accuracy of FEM solutions. Because the FEM is approximate, the results of FEA are not exactly the same as the real values. In other words, the FEA solution will produce errors on the stresses and displacements. Therefore, it is necessary to evaluate those errors. The ~~energy method was introduced briefly in this chapter, and~~ single element tests have been carried out with constant, linear and parabolic stress fields in order to assess the errors . In single element tests, only two types of element (4-noded & 9-noded) with different shapes were used. These elements are in regular use but they produced different errors in each stress field. Four factors affect the values of the errors, viz: size, location, shape and orientation. Of these, the effect of element orientation is the least significant.

(Insert one paragraph - see the following page)

Once the element shape sensitivity and size effect have been established, the mesh could be chosen to minimize the errors in the results. Compared with an 9-noded element, a 4-noded element is simple, takes less computation and is consequently less expensive. Thus, based on 4-noded elements a simple assessment of accuracy has been found to estimate the actual stress values and minimize the analysis cost. These analyses will be detailed in Chapter 5.

Finally, general conclusions and recommendations are presented in Chapter 6.

(after paragraph 1 on Page 7)

In this study, the reference element in the single element tests was placed at a particular location (10, 15) in the three stress fields. Due to restrictions in time only one of a range of stress functions was used in each of the constant, linear and parabolic stress fields but these were more general than those used by Robinson (1985) and Burrows (1988). However, they are representatives of a range of stress field, and indicate how those four factors influence the errors of FEA solutions.

Chapter 2 Finite Element Formulation

2.1 Introduction

The concept of the finite element method was originally introduced for structural analysis by Turner et al (1956) and Argyris and Kelsey in the mid-50's (Argyris & Kelsey, 1960). The name "finite element" was initially coined in a paper by Clough in 1960 (Clough, 1960), in which the technique was presented for plane stress analysis.

Since then general progress has been so rapid that the method is now one of the most powerful tools available in structural analysis. It has also been recognized as a general numerical method for approximately solving various systems of partial differential equations with known boundary conditions. Thus its applications cover a wider range of physical as well as structural problems. For instance, problems arising in such fields as fluid mechanics, magneto- and electro-dynamics, temperature fields, etc., can be solved.

Zienkiewicz (1977) covers the mainstream of the development and includes a wide bibliography of the publications reflecting these activities.

This method is based on a general discretization procedure for solving continuum problems defined by certain classes of mathematical statements. Firstly, a structure is subdivided (hypothetically) into finite regions termed "elements", which are small enough that the shape of the displacement or stress field can then be reasonably well approximated, leaving only the coefficients of the approximated function to be found. These elements connect with each other through common points existing on their boundaries. At these points continuity and compatibility of the field variables are enforced. These common points are termed "nodes" or grid points. In the real structure the number of the interconnecting nodes is infinite. However, due to the finite character of the connectivity, many engineering problems need only to be discretized into a limited number of elements connected by discrete number of nodes.

Having the structure broken down into these two or three dimensional elements, it is now possible to perform a finite element analysis in several steps:

1. Define the structure idealization (elements and nodes).
2. Form each individual element stiffness matrix.
3. Assemble the element stiffness to form the global stiffness using equilibrium and solve the resulting equations.

In structural mechanics problems, the unknown field variables can be displacements, stresses, or both. These give rise to the displacement (stiffness) method, the force (flexibility) method, or the hybrid method. This research uses the stiffness method applied to constant, linear and parabolic stress fields, and further details will be explained in the Chapter 4.

The FEM is unique in the way it can formulate the properties of individual elements, whose behaviour is readily understood, for any type of problem. One of its main attractions is the ease with which it can be applied to problems with geometrically complicated boundaries.

The price that must be paid for this flexibility is in the amount of numerical computation required. Usually a large number of simultaneous equations have to be solved; if more elements and nodes are included for increased accuracy, then more equations will result. However, modern methods of equation solving e.g. the frontal solution techniques, banded solution method (Iron & Ahmed, 1981; Hinton & Owen, 1977), etc., have been evolved to solve these equations as economically in storage as possible. It is possible with the power of modern computers to solve large sets of equations.

The routine solution of linear problems by the finite element method, has been well established. For instance programs for solving problems in the theory of elasticity, thin and thick plate theory, and in three dimensional solids etc., have all been well developed and have now reached a high degree of sophistication.

The analysis process may be divided into the following stages:

1. selecting a domain for analysis, using symmetry if possible to minimise the size of the problem.
2. determining the boundary conditions.
3. subdividing the structure into subregions.
4. generating a suitable grid mesh for each region.
5. performing the analysis using a finite element program.
6. displaying the output using computer graphics and print out.

2.2 Basic steps in FEM

A derivation of the displacement linear elastic finite element method will be given in conjunction with the formulation of isoparametric elements in section 2.3. First, however, the basic steps will be described in general terms in the following section.

2.2.1 Selection of element type and discretization of the continuum

The first step is to decide on the type of element to be used, and then to subdivide the continuum or solution region into a suitable number of elements with associated nodes. In general the following points are considered in element selection:

(A) Element type:

The selection of the element will be related to the type of problem to be solved. Generally these can be grouped into four classes:

1. Plane stress/plane strain/axisymmetric (i.e. mathematically a 2D problem).
2. Plate bending.
3. Shells.
4. Three dimensional (solid analysis).

These are based on classifications relating to the problem type. Within a given type it may be possible to select specific element types.

In each group different levels of accuracy can be obtained. This depends on the number of nodal points and corresponding degrees of freedom which are associated with the element type. Nodal points are usually placed on the boundaries of the elements, although internal nodes can also be included in certain elements in order to increase efficiency. Usually the higher the order of element (i.e. the more degrees of freedom), the solution is more accurate but more expensive.

It would be expected that a solution would be more accurate if more elements were used (i.e. if a finer mesh was used). However, certain basic requirements have to be satisfied when selecting an element type to ensure convergence to the correct solution as the mesh becomes finer. These can be listed as follows:

1. The displacement (or stress) field within an element must be continuous.

2. The displacement model must include the constant strain state of the element, i.e. the element should be able to reproduce a constant strain field, if the nodal displacements require it.

3. The element should be able to reproduce rigid-body motions, i.e. when nodal degrees of freedom correspond to rigid-body motion, the element must exhibit zero strain and zero nodal forces. This is a special case of the constant strain criteria.

4. Elements should be compatible, i.e. there should be no interelement gaps or overlaps. Elements that violate these requirements in a mesh are called "incompatible" or "nonconforming". However an incompatible element can be valid and convergence is obtainable, if the incompatibilities disappear with increasing mesh refinement and the element approaches a state of constant strain.

5. An element should have no preferred direction. In other words, an element should be geometrically invariant, and give the same results in whatever direction it is orientated.

Elements edges can be straight or curved. This usually depends on the number of nodes defining the element edges. For example, straight edged elements will result from 3-noded triangles or 4-noded rectangular elements; curved edged elements will result from 6-noded triangular or 8-noded quadrilateral isoparametric elements, because each edge is defined by three nodes. In this work curved edged plane stress/plane strain elements were not used.

(B) Element size

In general the finer the mesh, the better the accuracy; however at the same time a larger computational effort is required. The number of elements to be used will be decided by the type of structure to be analysed, but generally more elements are required in regions where stresses vary rapidly than in regions where they vary gradually. However, for complex elements coarser meshes will produce efficiencies as good as fine meshes for simpler elements i.e. less elements are needed. From a practical point, the number of unknowns (numerically) should be no greater than that justified by the accuracy of the input data available (loads, constraints, materials, geometry).

(C) Element aspect ratio

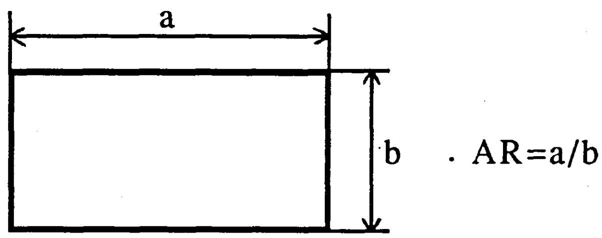


Fig. 2.1

The aspect ratio for two dimensional elements is defined as the ratio of the element sides (as shown in Fig. 2.1). The optimum aspect ratio at any location within the mesh depends largely upon the difference in the rate of change of displacements in different directions. For instance if the displacements vary at about the same rate in each direction, the closer the aspect ratio to unity the better the quality of the solution. Desai (Desai & Abel, 1972) carried out a study using different aspect ratios to analyse a beam bending problem. In the study four noded rectangular elements were used, and it was found that as long as the aspect ratios were near unity, accuracy was acceptable.

In practical structural analysis it is unusual to select elements which have an aspect ratio of unity. However, it is advisable to keep this ratio as near to unity as possible. Indeed large values, which imply long narrow elements, should be avoided because numerical problems may arise in the calculations of the stiffness, if extremely large or small terms are involved.

2.2.2 Shape functions

A shape function defines the variation of the field variable, and its derivatives, through an element in terms of its values at the nodes. Therefore shape functions are closely related to the number of nodes and hence the type of element.

Polynomials are usually selected as shape functions as they are relatively easy to manipulate mathematically, particularly with regard to integration and differentiation. However, the degree of polynomial chosen will clearly depend on the number of nodes and the degrees of freedom associated with the element.

2.2.3 Assembly of element properties

Element properties have to be assembled to determine the behaviour of the entire solution region or system. In other words, the element matrix equations have to be combined in some fashion.

In the structural displacement method, the assembly process is based on the laws of compatibility and equilibrium. Also, displacements of two concident points must have identical values for compatibility to be satisfied. The matrix equation for the system has the same form as the equations for an individual element except that it now contains terms associated with all nodes.

This equation is then modified to take into account the boundary condition of the problem. These are the physical constraints or supports that must exist so that the structure or continuum has a unique solution.

2.3 Isoparametric elements

2.3.1 Introduction

The family of isoparametric elements was first introduced by Taig and Irons (1961, 1966a & 1966b). The term isoparametric implies that the same interpolation function is used to define the displacement variation within the element and the element geometry.

The basic procedure is to express the element coordinates and displacements by functions expressed in terms of the natural coordinates of the element. A natural coordinate system is a local system defined by the element geometry and not by the element orientation in the global system. Moreover these systems are usually normalized such that the natural coordinate has unit magnitude at the primary external boundaries used.

Isoparametric elements are used in the present work. The particular elements chosen based on strain (displacement) assumptions, are the nine noded and four noded isoparametric elements as well as constant strain triangles. Figure 2.2 and Figure 2.3 show these elements and their natural coordinates systems. It is interesting to note that this family only contains membranes and solids, and there are no isoparametric stress membrane elements.

A different family of isoparametric elements, based on stress assumptions, has been introduced by Robinson (Robinson, 1973), termed "isoparametric stress elements". However, these elements are not widely used yet and thus in this work only the strain shape function elements are employed. Details will be explained in the following section but the reasons for using isoparametric elements will be explained first.

1. For a given number of degrees of freedom, complex isoparametric elements

are far more accurate and versatile than simple elements. Moreover a considerable saving of computer effort is obtained, even though a complex element requires more time to formulate. This is because an acceptable solution requires fewer elements compared with simpler elements.

2. Data preparation is considerably reduced with complex elements, although this can be minimised to a certain extent by automatic mesh generator schemes.

3. Numerical integration makes the evaluation of the characteristics of curved complex elements relatively straightforward.

4. Curved boundaries can be described easily using curved sided elements.

5. Curved element sides preclude the necessity for mesh refinements where the boundaries of a structure are curved. However sometimes the reduced number of complex elements may not be adequate to represent all the geometries of a particular problem.

6. In linear elasticity for the 8-noded isoparametric element, the displacement field is not significantly affected for different aspect ratios in the range between 0.5 and 1.0.

2.3.2 Isoparametric strain membrane elements

The membrane elements considered here will be restricted to general quadrilaterals. A four-noded and an eight-noded quadrilateral are shown in Figure 2.2 and 2.3 respectively.

1. Shape functions

The shape of a membrane element can be expressed in terms of interpolation functions and its modal coordinates, that is,

$$x = \sum_{i=1}^{N} P_i(\xi, \eta)\, x_i \qquad (2.1)$$

and

$$y = \sum_{i=1}^{N} P_i(\xi, \eta)\, y_i \qquad (2.2)$$

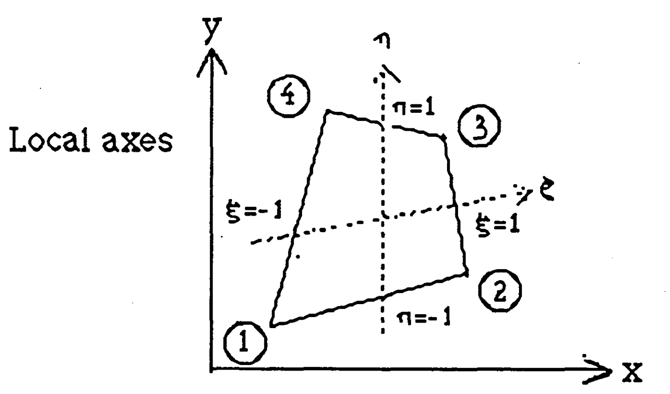


Fig. 2.2 Four-node quadrilateral (linear quadrilaterial)

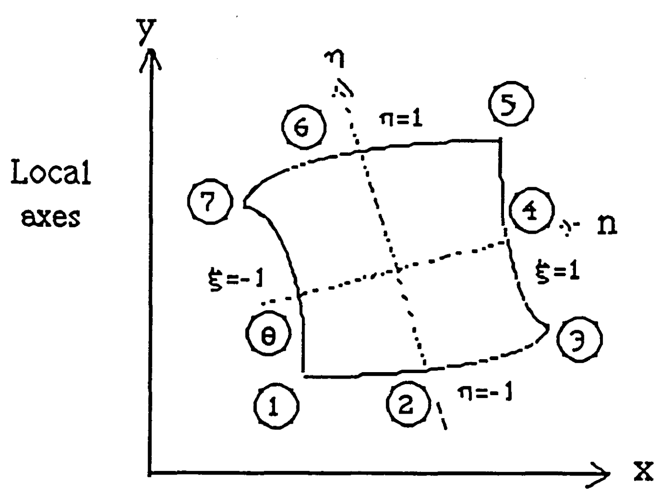


Fig. 2.3 Eight-node quadrilateral (quadratic quadrilateral)

where $P_i(\xi, \eta)$, i =1 to N are interpolation functions in the curvilinear coordinates ξ and η, x_i and y_i are the local x, y coordinates of node i, and N are the total number of nodes for the element. The interpolation functions for the linear quadrilateral (four nodes) are shown in Table 2.1 and for the quadratic quadrilateral (eight nodes) in Table 2.2.

Table. 2.1 Interpolation functions and derivatives for a linear quadrilateral

			Interpolation function	Derivatives	
Node i	ξ_i	η_i	$P_i(\xi, \eta)$	$\partial P_i/\partial\xi$	$\partial P_i/\partial\eta$
1	-1	-1	$(1-\xi)(1-\eta)/4$	$-(1-\eta)/4$	$-(1-\xi)/4$
2	1	-1	$(1+\xi)(1-\eta)/4$	$(1-\eta)/4$	$-(1+\xi)/4$
3	1	1	$(1+\xi)(1+\eta)/4$	$(1+\eta)/4$	$(1+\xi)/4$
4	-1	1	$(1-\xi)(1+\eta)/4$	$-(1+\eta)/4$	$(1-\xi)/4$

These interpolation functions can be expressed in general terms. For the linear quadrilateral

$$P_i = (1 + \xi\,\xi_i)\,(1 + \eta\,\eta_i)/4 \tag{2.3}$$

where ξ_i and η_i take their nodal values.

For the quadratic quadrilateral corner nodes,

$$P_i = (1+\xi\xi_i)(1+\eta\eta_i)/4 - (1-\xi^2)(1+\eta\eta_i)/4 - (1+\xi\xi_i)(1-\eta^2)/4 \tag{2.4}$$

Midside nodes, $\xi_i = 0$, $\eta_i = \pm 1$,

$$P_i = (1-\xi^2)\,(1+\eta\eta_i)/2 \tag{2.5}$$

Midside nodes, $\xi_i = \pm 1$, $\eta_i = 0$,

$$P_i = (1+\xi\xi_i)\,(1-\eta^2)/2 \tag{2.6}$$

It should be noted that midside nodes are midside in a curvilinear sense; for example, the midside node on the boundary $\eta = \pm 1$ is at $\xi = 0$. The position of all nodes is arbitrary in a Cartesian sense.

The interpolation functions have the special property that at node j

$$P_i = \begin{cases} 1 & (i \neq j) \\ 0 & (i = j) \end{cases} \tag{2.7}$$

2. Displacement functions

The displacement functions for a membrane element are u and v, where u is the displacement in the x-direction and v is the displacement in the y-direction. By definition, the displacement and shape interpolation functions are the same. Hence the displacement functions are given by:

$$u = \sum_{i=1}^{N} P_i(\xi, \eta)\, u_i \tag{2.8}$$

and

$$v = \sum_{i=1}^{N} P_i(\xi, \eta)\, v_i \tag{2.9}$$

where u_i and v_i are the nodal displacements in the x and y direction at node i.

3. Stress and strain fields:

The total strain field $\{\varepsilon_\varepsilon\}$ is given by:

$$\{\varepsilon_\varepsilon\} = \{\varepsilon_x\ \varepsilon_y\ \gamma_{xy}\} \tag{2.10}$$

where

$$\varepsilon_x = \partial u/\partial x$$
$$\varepsilon_y = \partial v/\partial y$$
$$\gamma_{xy} = \partial u/\partial y + \partial v/\partial x \tag{2.11}$$

Substituting Equations (2.8) and (2.9) into (2.10) leads to:

$$\{\varepsilon_\varepsilon\} = [B]\ \{\delta\} \tag{2.12}$$

where

$$\{\delta\} = \{u_1\ v_1\ u_2\ v_2 \cdots u_i\ v_i \cdots u_n\ v_n\} \tag{2.13}$$

= element deformation variables

$$[B] = [B_1(\xi,\eta)\ B_2(\xi,\eta)\ ...\ B_i(\xi,\eta)\ ...\ B_n(\xi,\eta)] \tag{2.14}$$

= the strain matrix

in which

$$\left[B_i(\xi,\eta)\right] = \begin{bmatrix} \frac{\partial P_i}{\partial x} & 0 \\ 0 & \frac{\partial P_i}{\partial y} \\ \frac{\partial P_i}{\partial y} & \frac{\partial P_i}{\partial x} \end{bmatrix} \tag{2.15}$$

Since the interpolation functions P_i are defined in terms of the curvilinear coordinates ξ and η, a transformation from local to global coordinates is required in equation (2.15). It is well known that the cartesian and the curvilinear derivatives are related by:

$$\begin{bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \end{bmatrix} = [J]^{-1} \begin{bmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \end{bmatrix} \tag{2.16}$$

where [J] is the Jacobian matrix,

$$[J] = \begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{bmatrix} \tag{2.17}$$

Differentiating Equations (2.1) and (2.2) in accordance with (2.17) gives the

Jacobian matrix as

$$[J]=\begin{bmatrix} \frac{\partial P_1}{\partial \xi} & \frac{\partial P_2}{\partial \xi} & \cdots & \frac{\partial P_i}{\partial \xi} & \cdots & \frac{\partial P_N}{\partial \xi} \\ \frac{\partial P_1}{\partial \eta} & \frac{\partial P_2}{\partial \eta} & \cdots & \frac{\partial P_i}{\partial \eta} & \cdots & \frac{\partial P_N}{\partial \eta} \end{bmatrix} \begin{bmatrix} x_1 & y_1 \\ x_2 & y_2 \\ \cdot & \cdot \\ \cdot & \cdot \\ x_i & y_i \\ \cdot & \cdot \\ \cdot & \cdot \\ x_N & y_N \end{bmatrix}$$

(2.18)

The derivatives of Equation (2.15) are now obtained using Equations (2.16) and (2.18),

$$\begin{bmatrix} \frac{\partial P_i}{\partial x} \\ \frac{\partial P_i}{\partial y} \end{bmatrix} = [J]^{-1} \begin{bmatrix} \frac{\partial P_i}{\partial \xi} \\ \frac{\partial P_i}{\partial \eta} \end{bmatrix}$$

(2.19)

The stress-strain relationship is given by following quation,

$$\{\sigma\} = [D]\,(\{\varepsilon_\varepsilon\} - \{\varepsilon_0\}) \tag{2.20}$$

where

$$\{\sigma\} = \{\sigma_x\ \sigma_y\ \tau_{xy}\} \tag{2.21}$$

and, for an isotropic linear elastic material, and plane stress conditions

$$[D] = \frac{E}{(1-\nu^2)} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{(1-\nu)}{2} \end{bmatrix} \tag{2.22}$$

The stress field is now obtained in terms of the deformation variables by substituting Equation (2.12) into (2.20),

$$\{\sigma\} = [D]\,([B]\,\{\delta\}-\{\varepsilon_0\}) + \{\sigma_0\} \tag{2.23}$$

4. Element stiffness and force evaluation:

Element stiffnesses are derived from the variational principal of minimum total potential energy. The total potential energy P of a structure is defined in terms of the field variable, and is then minimized with respect to this field variable, subject to specific boundary conditions. When the potential energy is at its minimum then equilibrium conditions are satisfied.

If the strain energy of an element is P_e, (which will be in terms of the nodal displacements), and the imposed potential energy due to external load is W, then the total potential energy can be defined as:

$$P = \sum P_e + W \tag{2.24}$$

The minimized condition with respect to displacements can then be written as:

$$\frac{\partial P}{\partial\{\delta\}} = \sum \frac{\partial P_e}{\partial\{\delta\}} + \frac{\partial W}{\partial\{\delta\}} = 0 \tag{2.25}$$

The element contribution to this energy is:

$$\frac{\partial P_e}{\partial\{\delta\}_e} = [K]_e \{\delta\}_e + \{F\}_e$$

(2.26)

where $\{\delta\}_e$ = the displacements associated with the element

$\{\delta\}$ = the global displacements

$[K]_e$ = the stiffness matrix of the element

$\{F\}_e$ = the fictious forces acting on the element nodes which can further defined by:

$$\{F\}_e = \{F\}_{e\varepsilon_0} + \{F\}_{e\sigma_0} \tag{2.27}$$

where $\{F\}_{e\varepsilon_0}$ = the nodal force vector due to initial strains

$\{F\}_{e\sigma_0}$ = the nodal force vector due to the initial stresses

The minimization of the imposed load is expressed as:

$$\partial W/\partial\{\delta\} = \{F\}_p \ \{F\}_g - \{F\} \tag{2.28}$$

where $\{F\}_p$ = the nodal forces due to distributed load $\{\underline{P}\}$

$\{F\}_g$ = the nodal forces due to any distribued external load on boundary elements

$\{F\}$ = any external load acting on nodes

Subtituting equations (2.26), (2.27), (2.28) into equation (2.25) gives the minimized condition as follows:

$$\frac{\partial P}{\partial\{\delta\}} = \sum_{1}^{\text{No. of ele.}} ([K]_e \{\delta\}_e + \{F\}_{e\varepsilon_0} + \{F\}_{e\sigma_0}) + \{F\}_p + \{F\}_g - \{F\} = 0$$

(2.29)

or $$[K]\{\delta\} = \{R\} \tag{2.30}$$

which represents the assembly of the final equilibrium equations together with prescribed boundary conditions.

It can be shown that:

$$[K]_e = \int_V [B]^T [D] [B] \, dv \tag{2.31}$$

$$\{F\}_{e\varepsilon_0} = -\int_V [B]^T [D] \{\varepsilon_0\} \, dv \tag{2.32}$$

$$\{F\}_{e\sigma_0} = \int_V [B]^T \{\sigma_0\} \, dv \tag{2.33}$$

$$\{F\}_p = \sum \{F\}_{ep} = \sum -\int_V [P_i(n,g)]^T \{\underline{P}\} \, dv \tag{2.34}$$

$$\{F\}_g = \sum \{F\}_{eg} = \sum -\int_A [P_i(n,g)]^T \{g\} \, dA \tag{2.35}$$

where $\{\underline{P}\}$ = distributed load per unit volume

$\{g\}$ = component of boundary pressure

For 2-dimensional problems the incremental volume dv is:

$$dv = t \, dx \, dy$$

where t = the thickness of the element.

The relation between the Cartesian and the curvilinear coordinates is:

$$dx \, dy = \det [J] \, d\xi \, d\eta$$

in which det[J] is the determinant of the Jacobian matrix.

2.4 Finite elements and finite differences

A comparison of the finite element and finite difference methods is of interest. Both are discretization techniques and in both methods a continuum is represented by a set of nodal generalized coordinates, requiring the solution of a set of simultaneous algebraic equations. Aside from these similarities the two methods may appear quite different, since the deformation of a finite element is determined entirely by nodal displacements within the element and on its boundaries, while in a finite difference mesh there are nodes outside each "element". Also, the finite element approach is usually viewed as minimization of a functional without reference to differential equations, while the finite difference approach has usually been presented as a method for approximating the governing differential equations without reference to functionals. However, recently it has been found profitable to derive finite difference models from functionals, using, for example, the same potential energy expression as used to generate finite element models. Thus, the two methods may be said to differ in that the finite difference method is differential and FEM is integral, these differing in the choice of generalized coordinates and location of nodes.

Available information(Bushnell; Forsberg) suggests that there are types of problems to which finite differences are better suited than the finite elements, and vice versa. There are accurate and user-oriented programs based on each method. It appears that neither method will wholly supplant the other. For a given number of d.o.f. both appear capable of about the same accuracy. Less computer time may be needed to generate structural equations by the finite difference method; however, comparisons inevitably depend on the type of problem, mesh regularity, and program organization as well as the basic analysis method. It might be noted that the finite difference method is not necessarily restricted to the regular meshes adopted in most explanations of the subject. However, the mathematical description of the structural geometry may be awkward. Consider, for example, the complexity of a motor vehicle consistary of odd-shaped stiffened panels.

Generally, the advantages of the finite element method arise from its physical appeal (Zienkiewicz, 1970) and the relative ease with which a complicated structure can be modelled and its boundary conditions treated. Finite elements are in a sense pieces of the actual structure; therefore the engineer may be aided by his structural intuition and a sense of physical reality when selecting what elements to use, joining elements of different types or different orientations in space, etc. Similarly, improvements in the properties of finite elements have often come from physical insight into element behaviour. Physical insight is perhaps less readily applied to finite difference methods, because there is usually no definite "element" that can be visualized.

Chapter 3 FEA Software on Microcomputers

3.1 Introduction

The microcomputers of today are marvels of technology but they would remain splendid toys without professional software to enable one to take full advantage of their possibilities as an efficient professional tool.

In the past several decades an ever broadening spectrum of new and cheap computers has been presented to the public. Various new types of desktop computers and to some extent so called personal computers (PC's) have become very popular for engineering applications in smaller companies, construction offices etc., where the local availabity of a relatively small and slow computer system is more effective for most applications than usage of a mainframe.

The development of desktop computers was pioneered by several companies about 20 years ago (Schrefler et al, 1984). Enabled by new technologies the central processing unit (CPU), memory boards and interface boards could be reduced in size to fit into a very small cabinet. A keyboard, small display and an external storage device (e.g. a floppydisk driver) could be combined in a small housing like a typewriter to form a desktop computer.

Desktop computers also differed significantly from other computers available at that time in their software. The idea was to make handling and programming a desktop computer as easy as possible.

The introduction of microcomputers which give an analyst independence has provided a new impetus to software development. It was soon realized that it was not sufficient simply to transfer traditional programs and methodology to the new microcomputers. For instance, alphanumeric and graphical interactive facilities are now taken for granted with the new desktop computers, whereas it was not long ago that punched cards were used. The hardware revolution, however, has resulted in a much greater number of computer users, so that today the typical user is no longer a computer specialist. Thus, a high degree of user comfort and, even for the occasional user, a good system overview are two of the main requirements for microcomputer software.

Numerical techniques such as the finite difference, boundary element, and finite element method have helped engineers, especially structural engineers, to solve complex problems otherwise unsolvable by analytical approaches. Among these three, the finite element method has gained more popularity principally due to its generality, and ease of modelling geometry, material behaviour, discontinuity in material behaviour, boundary condition, loading, etc., along with the ease of coding. The finite element method is a firmly established technique and almost perfect tool for today's structural engineers. In the past 15 years significant advances have been made in finite element applications to various engineering problems. There are numerous general purpose finite element packages commercially available. However, most of these packages require use of large computers. This is of particular concern to small and medium size engineering firms which usually do not have access to large computers. Also buying time on a main frame is expensive.

Recently, microcomputers and desktop computers have gained wide acceptance in engineering fields. Complete user control over the analysis is probably the major factor. In the PC environment the analyst is more inclined to experiment with the model to obtain a realistic solution. The inherent constraints on model size may have a beneficial influence since there is a tendency to construct unnecessarilly large models when the facilities exist to handle these.

To meet the needs of the engineering problems, a number of FEA software packages have emerged. Usually, these packages include six modules:

1) a preprocessor for the input of starting data and for mesh generation.

2) a program part for temperature field analysis.

3) a program part for linear deformation and stress analysis.

4) a program part for non-linear deformation and stress analysis.

5) a program part for dynamic model analysis.

6) a postprocessor for the graphical representation of results.

Most systems can perform static analysis, dynamic analysis, vibration analysis, gap analysis, heat transfer analysis, thermal stress analysis and transient response analysis. In this study, only static mechanics part of FEA software is looked at in details, especially on the abilities of dealing with linear deformation and stress analysis of linear elastic problems.

3.2 The features and functions for different programs

As in all products there are good and not-so-good features and the costs have to be balanced against the features and performance of the product. The features and functions of a program have to be known before choosing that most suitable for a specific application (Ramsay, 1988). Generally, most packages can cope with many types of problems, but some address specific engineering problems, that is, each package has its own strengths and weaknesses.

A number of surveys on FEA packages have been undertaken (Mackerle, 1986, 1988; Mair, 1988). Mair (1988) lists the types of analysis, element libraries and other features for a number of programs (Table 3.1, 3.2, 3.3a, 3.3b).

GIFTS (Graphical Interactive Finite Element Total System) was selected for the solution of the problems for this project, with some use of ANSYS-PC/LINEAR and MSC/PAL2. GIFTS is marketed by CASA/GIFTS Inc. GIFTS is a package of

Table 3.1 ANALYSIS TYPE

	LINEAR STATICS	LINEAR DYNAMICS	NON-LINEAR STATICS	NON-LINEAR DYNAMICS	HEAT TRANSFER	FLUID FLOW	FIELD PROBLEMS	DESIGN OPTIMISATION	MODAL ANALYSIS	DYNAMIC RESPONSE	RESPONSE SPECTRUM	STABILITY ANALYSIS
ANSYS-PC	*	*	*		*		*	*	*	*	*	
COSMOS-M	*	*	*	*	*				*	*	*	*
ESDUFINE	*				*		*					
FESDEC	*	*	*		*		*		*	*	*	
FLASH	*	*	*						*			*
GIFTS	*	*			*				*	*	*	
IMAGES-3D	*	*							*	*	*	
LUSAS	*								*	*	*	*
MICROFIELD	*		*		*	*	*					
MSC PAL2	*	*			*		*		*	*	*	
NISA-II	*	*			*	*	*	*	*	*	*	*
SAP80	*	*	*						*	*	*	
SAP86/mTAB	*	*			*				*	*	*	
SUPERSAP	*	*	*	*	*			*	*	*	*	

* Tables 3.1 to 3.3 are from the paper writen by Mair (1988)

Table 3.2 ELEMENT LIBRARY

	BEAMS	MEMBRANES	PLATES	SHELLS	AXISYMMETRICS	SOLIDS	SPRINGS	PIPES	FLUID ELEMENTS	HEAT TRANSFER ELEMENTS	SHEAR PANELS	STIFFENERS	USER DEFINED ELEMENTS
ANSYS-PC	*	*	*	*	*	*	*	*		*			
COSMOS-M	*	*	*	*	*	*	*	*		*	*		*
ESDUFINE		*	*		*	*							
FESDEC	*	*	*	*	*	*	*	*		*		*	
FLASH	*	*	*	*	*		*						
GIFTS	*	*	*	*	*	*	*	*		*		*	
IMAGES-3D	*	*	*		*	*	*						
LUSAS	*	*	*		*		*				*		
MICROFIELD	*	*	*	*	*	*	*						
MSC PAL2	*	*	*	*	*	*	*	*		*	*	*	*
NISA-II	*	*	*	*	*	*	*		*	*		*	
SAP80	*	*	*	*	*	*	*						
SAP86/mTAB	*	*	*	*	*	*	*	*		*			
SUPERSAP	*	*	*	*	*	*		*		*		*	

Table 3.3a OTHER FEATURES (1)

	ISOTROPY	ANISOTROPY	ELASTO-PLASTIC	TEMPERATURE DEPENDENT	MECHANICAL LOADS	THERMAL LOADS	MULTI-POINT CONSTRAINTS	BAND/FRONT OPTIMISER	RESTART	SUBSTRUCTURING	CODE CHECKS
ANSYS-PC	*			*	*	*	*	*	*		
COSMOS-M	*	*	*	*	*	*	*	*	*	*	*
ESDUFINE	*				*	*		*			
FESDEC	*	*	*		*	*	*	*	*		*
FLASH	*	*			*	*	*	*	*		
GIFTS	*	*			*	*	*	*	*	*	*
IMAGES-3D	*				*		*				*
LUSAS	*	*		*	*	*	*	*	*		*
MICROFIELD	*	*	*	*	*	*	*	*			*
MSC PAL2	*			*	*	*	*	*	*		
NISA-II	*	*		*	*	*	*	*	*		*
SAP80	*				*	*	*				
SAP86/mTAB	*	*		*	*	*	*	*	*	*	
SUPERSAP	*	*		*	*	*	*	*	*		

Table 3.3b OTHER FEATURES (2)

	MATERIAL FILE	CYCLIC SYMMETRY	FAILURE CRITERIA	CAD LINK	ANIMATION	COLOUR	NAFEMS BENCHMARKS	COMPOSITES	MENU DRIVEN	ON-LINE HELP
ANSYS-PC	*			*	*	*	*		*	*
COSMOS-M	*		*	*	*	*		*	*	*
ESDUFINE	*			*		*	*		*	*
FESDEC	*			*	*	*	*		*	*
FLASH			*	*		*				
GIFTS	*		*	*	*	*	*	*		*
IMAGES-3D	*			*	*	*			*	*
LUSAS						*	*			
MICROFIELD	*	*	*	*		*	*	*	*	*
MSC PAL2	*		*	*	*	*				
NISA-II	*		*	*	*	*	*	*	*	*
SAP80										
SAP86/mTAB	*		*	*	*	*	*			
SUPERSAP				*	*	*			*	

computer aided structural analysis and a graphical interactive finite element total system (GIFTS manuals, 1979). The ANSYS (Swanson Analysis Systems) computer program is a large-scale, general purpose computer program for the solution of several classes of engineering problems (ANSYS manual, 1986). MSC/PAL2 is from the Macneal-Schwendler Corporation. It is a collection of programs for stress and vibration analysis of mechanical systems, components, and structures (MSC/PAL2 manuals, 1987).

GIFTS is a powerful system with many capabilities. It can perform static analysis, dynamic analysis, vibration analysis, gap analysis, heat transfer analysis, thermal stress analysis and transient response analysis. It provides geometric modelling and automatic discretization of frames, plates, shells, and solids, handles standard mathematical surfaces, and allows control of mesh spacing. It is suitable for two- and three-dimensional models. It has substructuring and multilevel substructuring analysis which is very efficient for analysis of large structures and structures with geometric repetition. The element library includes:

QA4--First-order 4-sided axisymmetric solid.

QA9--Second-order 4-sided axisymmetric solid.

QB4--First-order 4-sided plate.

QM4--First-order 4-sided membrane.

QM9--Second-order 4-sided membrane.

ROD2--First--order rod.

ROD3--Second--order rod.

SLD8--First--order solid brick.

SPRINA--Axisymmetric spring element.

SPRING--Axial spring element.

TA3--First-order 3-sided axisymmetric solid.

TET4--First-order solid tetrahedron.

TM3--First-order 3-sided membrane.

TM6--Second-order 3-sided membrane.

TSPRING--Torsional spring.

GIFTS can generate a model automatically. The standard procedure for model generation is as follows:

1. Define material and thickness menus.
2. Define key points located in space to assist in defining a line.
3. Select line element properties (optional).
4. Define line boundaries (line element generation optional).
5. Select grid element properties (plates or shells).
6. Define grids (surface elements generated, except for solid models).
7. Select solid element properties (solid models).
8. Define solid chunks (solid models).

In comparison with GIFTS, ANSYS-PC/LINEAR and MSC/PAL2 have fewer element types and no 8 or 9 node elements. ANSYS is a powerful computer program used worldwide for structural, thermal, fluid, electrical, and static electromagnetic analysis. Since ANSYS-PC/LINEAR is a complete program including preprocessing and postprocessing, linear static and modal analysis can be prepared, solved, and evaluated. The ANSYS-PC/LINEAR element library contains 13 ANSYS elements:

STIF42,45--2- and 3-D solids
STIF11,63--2- and 3-D shells
STIF3,54,4,44--2- and 3-D beams
STIF1,8--2- and 3-D spars
STIF14--1-, 2- and 3-D spring
STIF21--1-, 2- and 3-D mass
STIF27--3-D stiffness mass matrix

MSC/PAL2 uses the FEM to solve for displacements, forces, and stresses of two- and three-dimensional systems. Static, normal modes, transient response, and frequency response analysis capabilities are provided. Its element library is as shown in Table 3.2.

3.3 How to choose a suitable program for solving different types of problems

Because of the large variety of finite element packages available, the choice of a suitable package is a difficult one for users. Suppliers have a similar problem in supplying satisfactory software for a wide range of technical requirements.

The following aspects would be considered in choosing a package:

1. Environment

1) machine: on which machines the program can run.
2) operating system: the program is supported by which version of operating system.
3) the programing language.
4) specification of hardware.
   a. size of memory: usually, for microcomputer , e.g. IBM-PC and compatibles, there are two sizes of memory 640K or 512K, and about 20 Mb hard discc storage.
   b. peripherals: printer or plotter
   c. support tools: e.g. security device

2. Specification of a package

I. User-friendliness

1) commands

The concept of user-friendliness can best be explained by remembering that, for efficient operation, a dialogue needs to be set up between the computer and the operator. Therefore, it would be best to use a readily understood command language and simple words.

It is hoped that all commands are easily used, i.e. the commands' function can be viewed from the command itself. For example, commands "LINE & SURFACE" are to generate the lines and surfaces of a structure. Illegal commands input should not cause the program to fail; if a wrong command is input, the computer should not simply stop running without giving the user any warnings.

2) documentation quality

A good package should include a complete set of manuals including a primer, a user's reference manual, a theoretical manual and a sample problems manual. They should be easily understood and convenient to use.

3) mesh generator

In order to conduct a finite element analysis the structure must first be idealised into some form of mesh (Carnet et al). The art of successfully applying the technique lies in the combined choice of element types and shapes. Before a finite element mesh can be specified, the problem to be analysed must first be identified. This requires the user to define four blocks of information:

a. The geometry.

b. The boundary conditions.

c. The loadings.

d. The required results.

The reason for carrying out the analysis should be clearly defined and well understood before any computing is undertaken. The requirements can significantly affect the choice of the mesh and will always affect the cost of the analysis. In order to conduct an FE analysis the structural continuum must be idealised as a series of discrete finite elements. When specifying these elements four different sets of information must be given:

a. The nodal point coordinates.

b. The element topology (element node points and their interconnections).

c. The element geometrical properties; typically plate thickness or beam second moments of area.

d. The element material properties; typically Young's Modulus, Poisson's Ratio, density and coefficient of thermal expansion.

Therefore, it is essential that an uniform or graded mesh can be generated easily (it is very useful for the problems of the stress concentration). Transition elements are necessary to couple elements of different type or to facilitate mesh grading.

4) creating a user file

There are several ways to create an input data file. One way is for the preprocessor of the program to automatically generate a data file once the user has given a name for his input data (e.g. GIFTS). Another method is for the computer to send all the input data given in the preprocessing stage to a default output file (e.g. ANSYS). In this case, if the data is needed later on, it is necessary to rename the data file, otherwise, when new sets of data are input, the old data will be over written. The third way of creating input data file is using DOS commands "EDIT" and "EDLIN" to edit all data.

5) others

For a new user, a 'HELP' facility is essential to ensure rapid familarity with the program commands, their format and syntax.

II. Size of the problem

It should be known what size of problem or how many degrees of freedom can be solved by the package. For a complicated problem, especially when some parts of the problem are more important than others, substructure analysis is the most convenient method to use. It is therefore important that in the package substructural analysis is available and more than one substructure can be defined.

III. Capability of software

An important feature of a good package is its ability to tackle many types of problems, such as 2D, 3D, linear, nonlinear problems, static & dynamic analysis, vibration analysis, gap analysis, heat transfer analysis, etc. Its element library should be comprehensive, including triangular, rectangular and quadrilateral elements, beam, solid, spring as well as different orders of elements.

It is not unusual for more than one material to be used. A good program should allow for this eventuality as well as multiple loading cases. It should be possible to define different coordinate systems, e.g. local, global, polar, cylindrical, spherical, etc.

It is very important that a user has a range of graphical facilities available for presenting the results of an analysis. For any real structure there is an enormous volume of data available in the results and a good deal of this has to be assimilated. There are two functions that the graphical output can satisfy. The obvious one is for a user to get a pictorial presentation of how the structure is modelled and connected. The second function of the graphical output is to allow the user to add a measure of quality assurance to the results by investigating the effect of load on the finite element model and to satisfy the user that the results are in fact acceptable. The features that are required of an output processor are: tabulating the results of principal and component stresses, strains, displacements and coordinates at the points where the stresses are output etc. and plotting the deformed shape and contours of any stress component of the structure analysed. To print the graphs, a hard copy facility is needed.

IV. Editing

In most applications, the input data may be subject to several revisions, hence a good editing facility is essential. A checkpoint/restart capability reduces computation costs and processing time.

V. Speed (CPU time)

Computational time affects the expenses of the solution. Therefore, before the solution phase, the problem should be optimized, such as solving the matrix equations and optimization of bandwidth, or front-width, and renumbering the nodes or elements to minimizing solution time.

For some packages, there are many processors with different functions to calculate the stiffness matrices, deflections, stresses and so on separately (e.g. GIFTS), and also it is possible to chain the commands for each phase of the solution.

VI. Others

In addition to the points mentioned above, the cost of the package must be considered. Error fixes, reliability and reputation of the supplier should also be taken into consideration.

3.4 FEA solution using GIFTS for some typical problems

The following three problems are solved by GIFTS to illustrate the features mentioned above. The first example shows a comparison of the accuracy and computational time by using different types of elements and the transition elements. This problem also shows the range of meshes for two dimensional continuum under tension available in GIFTS. The next sample illustrates how to deal with a bending problem when there are no higher order elements. The use of symmetry of a structure is very important for reducing the size of a problem. The problem of a plate with double holes finally illustrates the procedure of solving a more complicated problem as few times as possible to obtain an accurate solution. The effect of boundary conditions on the accuracy of the results is discussed.

3.4.1 A plate with a central hole

Because both the structure and the loads for this special problem are doubly symmetric, it is necessary to model only a quarter of the whole structure to predict the behaviour of the whole. The dimensions of a quarter of the plate are shown in Fig. 3.1. The distributed loading along side 34 is 0.1 kN/mm^2. The thickness of the plate equals 1 mm and Young's modulus is 1 kN/mm^2 and Poisson ratio 0.3, so, the total load F applied on the plate equals 16 kN. In terms of the shape of the plate and the applied loading case, it is obvious that the stress concentration must occur at the point 1 (Benham, 1973). For a finite-width plate with a hole under tension, the stress concentration factor K_{tn} varies with the value of the ratio a/w (shown in Fig. 3.2) and

$$K_{tn} = \sigma_{max} / \sigma_{nom} \quad \text{(based on net section)}$$

where

$$\sigma_{nom} = F/(w-a)t = 16/(160-80) = 0.2 \ kN/mm^2$$

From the book "Stress Concentration Factors" (Peterson, 1974), when a/w = 0.5, K_{tn} is about 2.16.

The details of the analysis by using different types of elements of GIFTS are as follows:

1) Using triangular elements

The plate is divided into three grids (shown in Fig. 3.3). Element type is TM3 (first-order triangular membrane). Along edge 12, there are four elements, viz: 1, 3, 5, 7.

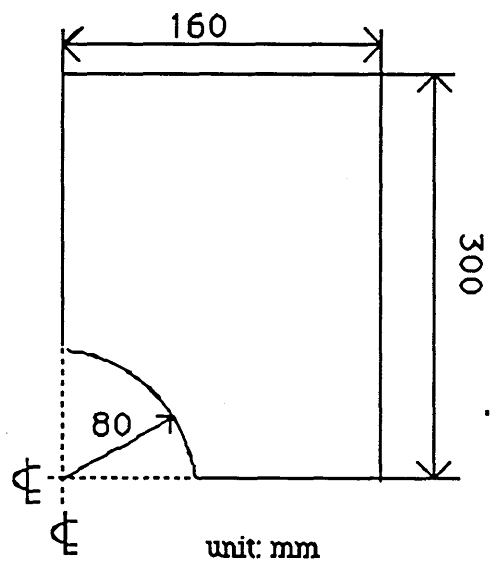


Fig. 3.1

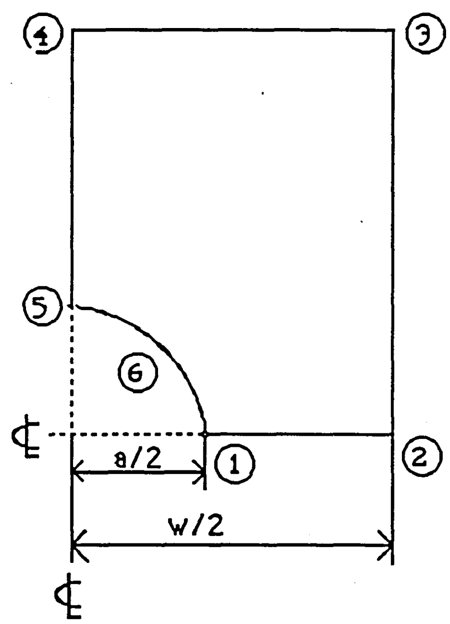


Fig. 3.2

It is found that the stress of element 1 which equals 0.191 is not the biggest stress. The biggest one occurs in element 17 (shown in Fig. 3.4), and its value is 0.254. This surprising result may be due to the discretization chosen. The smallest stress which is -0.070 occurs in element 23 which is near the point 5.

The relationship between the stresses and distances along edge 12 is indicated in Fig. 3.8 (K1). After curve fitting, the maximum stress is estimated to be 0.33, and the stress concentration factor is 1.65 .

2) Using first-order quadrilateral membrane elements

There are three grids in the plate (shown in Fig. 3.3).

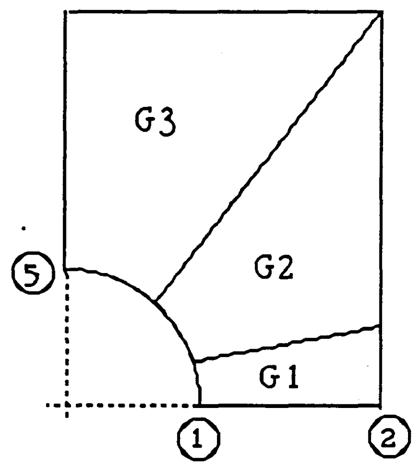


Fig. 3.3

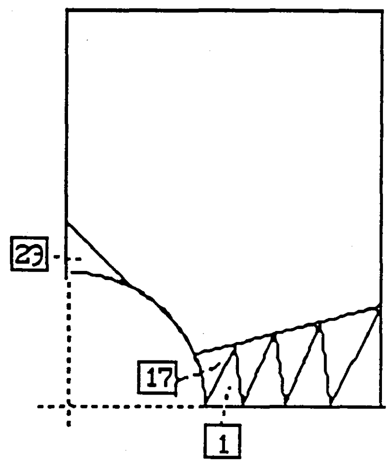


Fig. 3.4

The element type used is QM4: a first-order 4-sided membrane. Along the edge 12, there are five elements. The stress of element 1 in the y-direction is the maximum and equals 0.325, and the minimum equal -0.060 occured near the point 5. By extrapolation the maximum stress is about 0.36, and Ktn is 1.8 (shown in Fig. 3.8: K2).

3) Using second-order quadrilateral membrane elements.

Before establishing the model, the plate is divided into two grids so as to determine the key points (corner points) of the model (shown in Fig. 3.5). In this case, the element type is QM9 (a second-order 4-sided membrane). Care must be taken to have an odd number of points on each of the lines (because there must be three points on each side of the elements), otherwise, second-order quadrilateral membrane elements can not be formed but first-order elements. There are three elements along side 12: 1, 3, 5. The stresses are output at four special points, i.e.

Gauss points for each elements. Here, only the stress of the first point (shown in Fig. 3.6) for each element is chosen to find the maximum stress by plotting the stresses against the distances along edge 12. The maximum stress at point 1 is 0.34, and Ktn = 1.7 (shown in Fig. 3.8: K3). The smallest stress is -0.062.

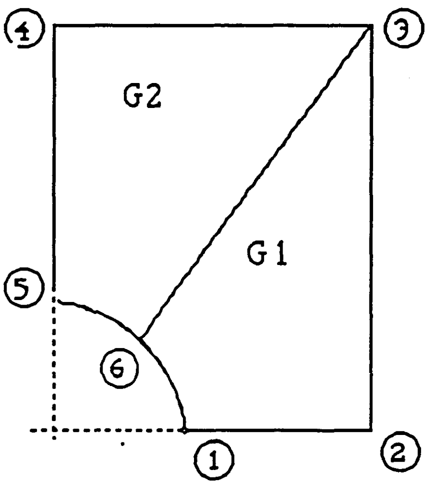


Fig. 3.5

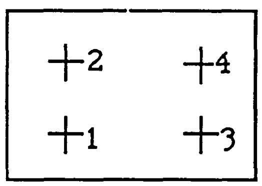


Fig. 3.6

Uniform meshes were used in each of the above models. In order to get more elements in the areas of high stress gradients with a coarser mesh elsewhere, graded meshes and transition elements are used below.

4) Using the elements of graded meshes

In this case, the first-order quadrilateral elements are used again, and the plate is divided into two parts: G1 and G2 (shown in Fig. 3.5). Besides selecting the number of points on each edge of a grid and determining where those edges are to lie, is biased the spacing on the individual lines at the same time. There are five elements along the edge 12. From the Fig. 3.8 (K4) the maximum stress is 0.41, and the stress concentration factor is 2.05. The minimum stress is -0.060.

5) Using the graded meshes and transition elements (Fig. 3.7).

To improve the accuracy of the results and save the computing time, transition elements are introduced. The sole function of transition elements is to make the

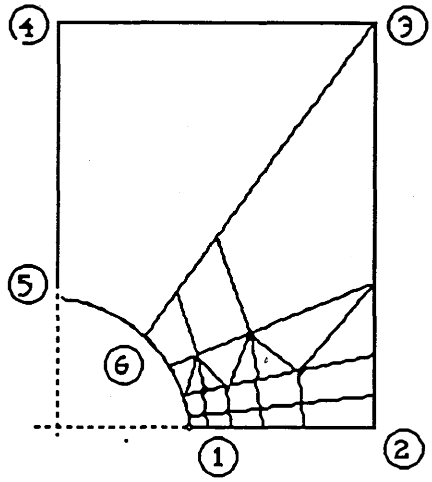


Fig. 3.7

transition from one number of points to the other, and two coincident lines with different numbers of points can be defined in order to merge points from the denser line into the coarser. The number of elements are the same as in the above 4 along side 12, and they are. The stress of element 99 is 0.377. After curve fitting, the maximum stress obtained is 0.43, so K_{tn} = 2.15 (shown in Fig. 3.8: K5). The minimum stress is -0.084 which is not accurate enough comparied with value of K_{tn}, because the attention was paid on the high stress area.

From the above analysis, it was found that the triangular element has the advantage of being simple to use. The quadrilateral elements (QM4) are slightly more accurate than the triangular elements by using the same or fewer number of elements (shown in Table 3.4), as they assume a linear distribution of strain over the element and are therefore better able to represent regions with a high stress gradient. Fewer quadrilateral elements would have to be used to give equal accuracy to triangular elements. However, the triangular element has the advantage that it can be used for bodies with irregular boundary shapes and it is also more amenable to the production of graded meshes. The most accurate prediction of the

The table below indicates main data of different types of elements.

Table 3.4 Comparison of the results by using different types of elements for the problem of a plate with a hole

	Type of elements	Number of elements	Number of nodes	S. C. F.	Half Bandwidth	CPU Time (seconds)
1	TM3	48	36	1.65	33	
2	QM4	45	63	1.8	44	
3	QM9	12	65	1.71	94	88.37
4	QM4 (Graded mesh)	25	54	2.05	38	
5	QM4 (Transition & graded mesh)	66	128	2.15	41	24.16

It is clear from the table that the solution obtained using the transition and graded mesh is in close agreement with the value given by the exact solution.

Stress concentration factor

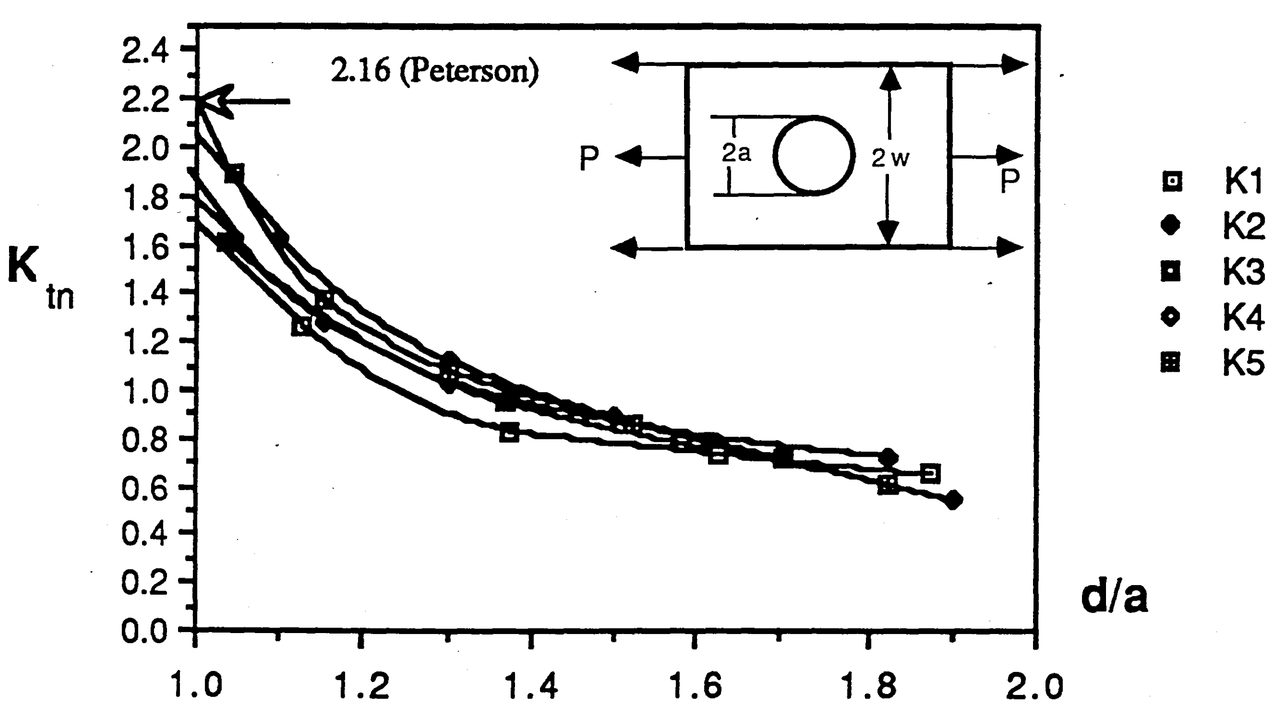


Fig. 3.8

* d: distance from 80-160 mm in the x direction

maximum stress and minimum stress is given by solution 5. The computational half bandwidthes are 33, 44, 94, 38 and 41 respectively, therefore, although the bandwidth of solution 1 is the smallest, the results are less accurate than others. In conclusion, the results of the methods using graded meshes to have more elements in the higher stress area are more accurate than other methods.

3.4.2 A folded-plate

Usually, there are four types of elements used in the finite element method, viz: membrane, plate, shell elements and solid bricks. For the problem of the bending folded-plate, membrane elements can not be used, and plate elements also are not good elements. The most suitable type of element in this case is a shell element (Rockey et al, 1975), and the shell elements are divided into two kinds, i.e. facet shell elements and semi-loof shell elements. In GIFTS, only facet shell elements are available.

1. Analysis using bending elements

According to the User's Reference Manual of GIFTS, there are three facet shell elements in GIFTS, viz: QB4 (four nodes quadrilateral elements), QB9 (nine nodes quadrilateral elements) and QB16 (16 nodes).

From "Selected FE Benchmarks in Structural and Thermal Analysis" (NAFEMS, 1987), it is known that the folded-plate is subjected to torsion as well as bending. It is built in (fully restrained) at one end and is loaded with differential shear forces acting along the top and bottom arms of the Z. A uniform mesh of eight elements along length and one element across the width of the flange or web is taken for this special problem. The dimensions of this folded-plate are shown in Fig. 3.9. In dealing with this kind of problem where bending and torsion exists the plate must be modelled using bending elements.

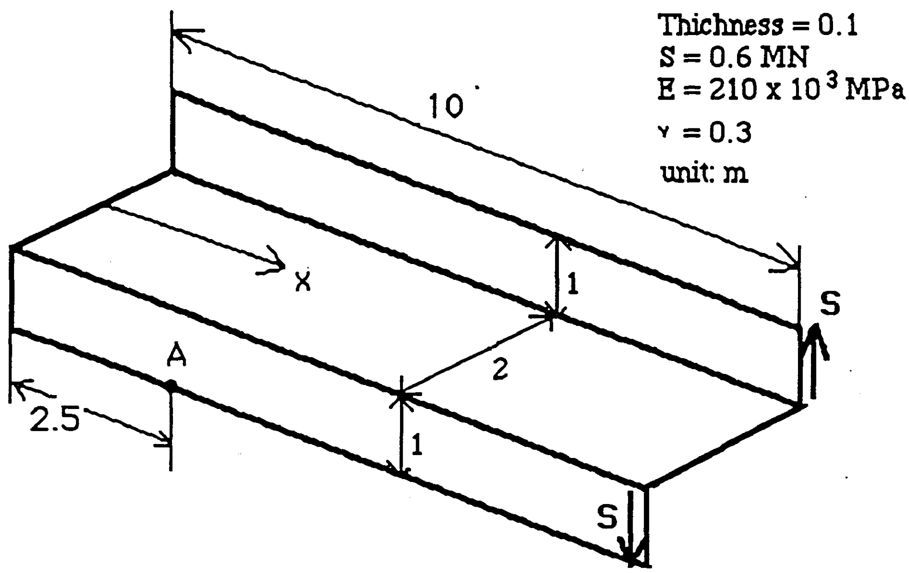


Fig. 3.9

Since the size of each element is very large using a uniform mesh of eight elements along the length, the stress in the element 2 which refers to the stress at the central point of the element is very different from the stress at the point A. For instance, by using QB4 elements with a uniform mesh of eight elements, the stress at the element centroid in x-direction is about -28.66 MPa. If the same mesh is used again with a different element type (TB3-three nodes triangular bending element), dividing the plate into many triangular elements gives the value of stress at point A of about -58 MPa. However, the stress at point A should be -108.8 MPa from the theoretical solution.

In previous calculations QB4 elements were used, and the results showed were very poor. To increase the accuracy of the solution, higher order elements or finer meshes should be used. But GIFTS on PC does not support 9-noded bending elements (QB9). Therefore, instead of QB9, a finer mesh of QB4 elements has to be used. Because the stress at point A is required, only one flange is divided into small elements in order to save time and storage space of the computer. Secondly, the number of elements of one flange, i.e. 32 elements, is four times greater than those in the first solution. The result of the stress at point A is -74.84 MPa which is better than the first instance, but not accurate enough. Finally, more elements should be used to gain a more accurate solution. A division of one flange into 48 elements leads to an excellent agreement between this result and target at point A which are -108.98 MPa and -108.8 MPa respectively. This result was obtained by using extrapolation and interpolation methods. Firstly, the stresses at the middle nodes of the elements along the width of flange are extrapolated and the stresses at the edge LK are obtained. Secondly the stresses at the points along the edge LK are interpolated and the stress at point A is obtained showing a very accurate result.

2. Further comments

Before concluding this discussion of the analysis of folded-plate, one possible source of complications must be mentioned. The plate was divided into 24, 48, 64, 80 elements and the result obtained from the finite element solutions are compared with the result from the analytic method in Table 3.5. Excellent agreement is obtained between the 80-elements solution and the target, and it is apparent that the accuracy of the solution increases as the mesh is refined when using the same type of elements. Now the critical computer resources for a finite element analysis are disk space and processing time. Disk space is directly proportional to both the number of unknowns and the bandwidth. Processing speed is proportional to the number of unknowns, and the square of the bandwidth. Thus, care must be taken in optimizing the bandwidth of stiffness matrix.

The plate is subjected to torsion, and so the stresses and the contours of the two flanges must be symmetric about the centre line of the section. If the number of elements on the two flanges are the same, then all results are the same. But if different numbers of elements are taken on two flanges, the stresses and the contours are also different. In order to be sure whether the results are correct or not, a following check should be undertaken: in the section through point A, the

Table 3.5 The results of the proble

of the folded-plate

Type of elements	Number of elements	Number of nodes	Unknowns	Half bandwidth	The stress at point A (MPa)
QB4	24	36	192	32	-28.7 *
TB3	48	36	192	32	-58 *
QB4	64	85	480	39	-74.84
QB4	80	102	576	45	-108.98
Target					-108.8

* Stress value constant over element and given at cent:

sum of all forces acted on every element along x-direction should be zero. Following this check, the result is as expected. Further, it should be mentioned here is that, owing to the different number of elements in zones G1 and G3, G3 is stiffer than G1. As a result of this, the displacements at node 5 and node 8 (shown in Fig. 3.10) are not the same and the displacement at node 5 is larger than the one at node 8 even though the displacements at these two nodes should theoretically be equal. Because the geometry and loading are symmetric about the central line, to reduce the size of the problem and the computational time, only half of it need have been taken into the analysis.

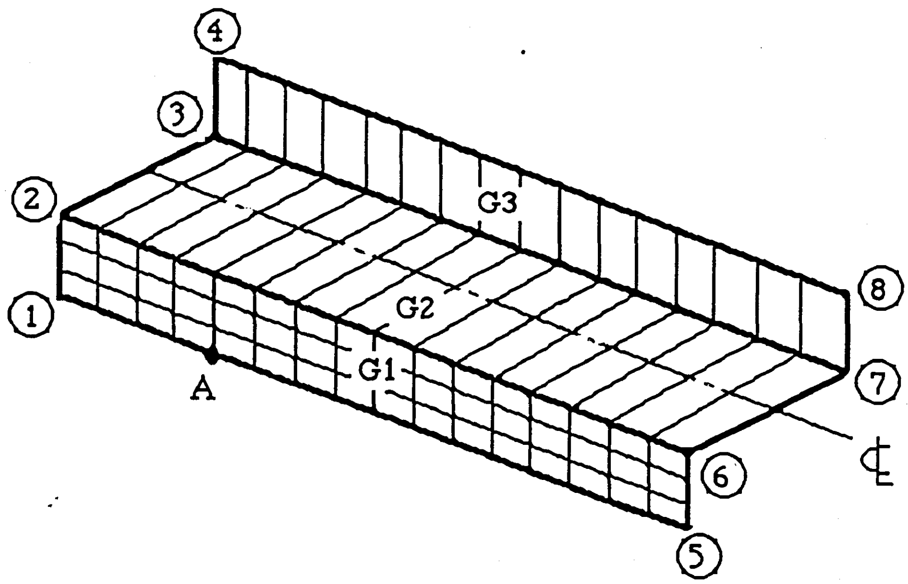
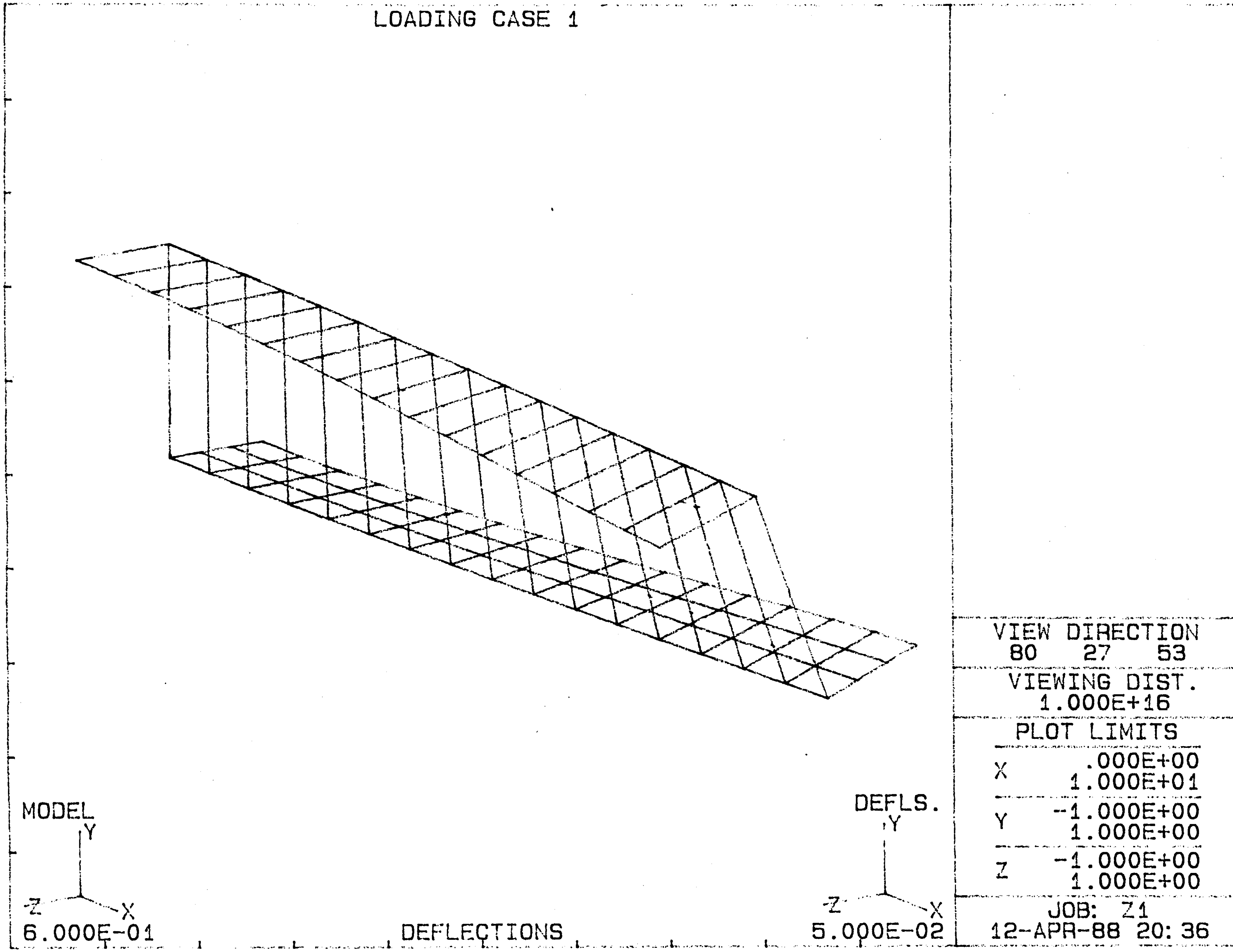


Fig. 3.10

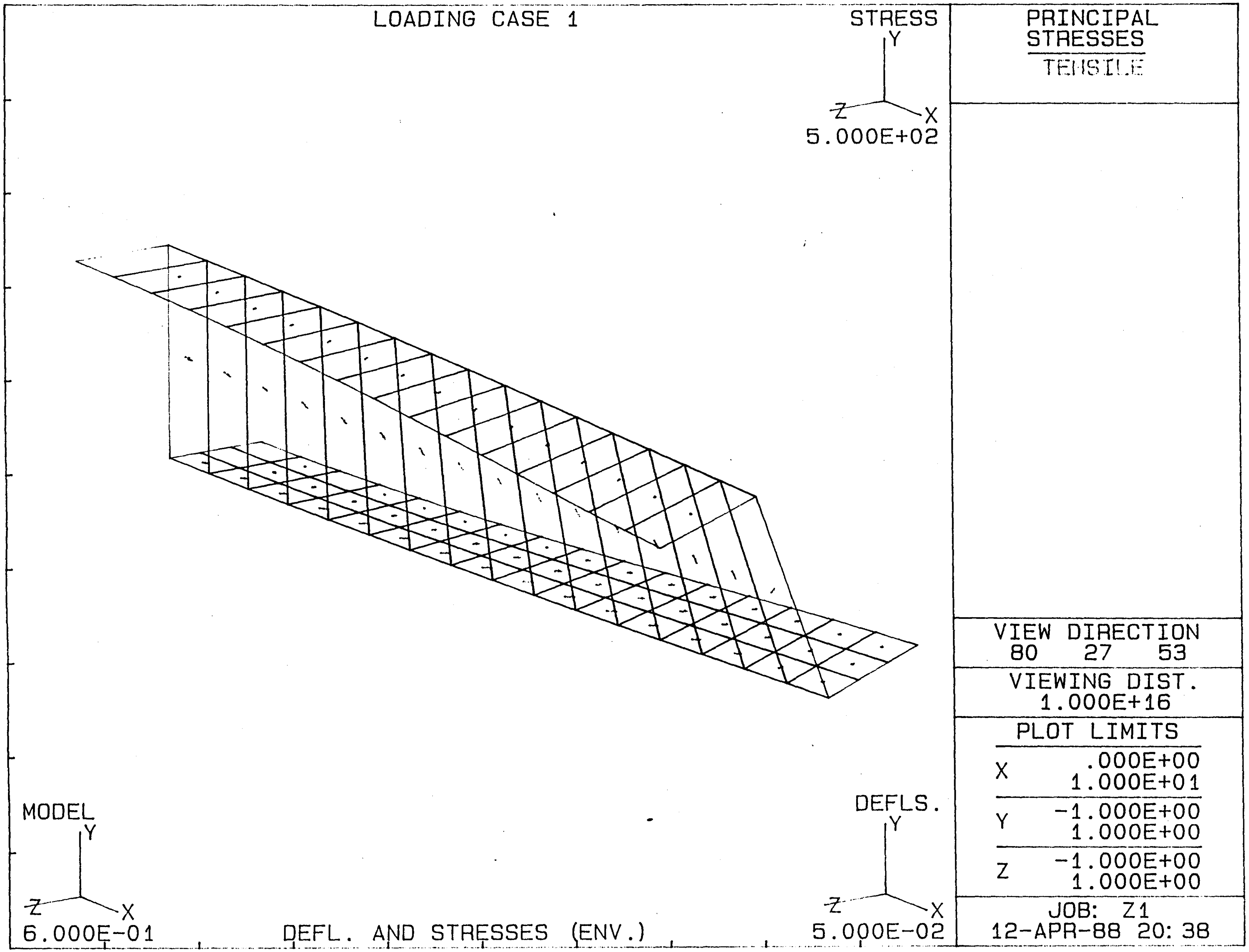
3.4.3 A plate with double holes

This example is similar to the first one, but it is more complicated. Usually, before analyzing a structure, it is important to divide the structure into zones correctly.

In deciding how to subdivide the structure, the following guidelines are used:



LOADING CASE 1
MODEL
Y
-Z
X
6.000E-01
DEFLECTIONS
DEFLS.
Y
-Z
X
5.000E-02
VIEW DIRECTION
80 27 53
VIEWING DIST.
1.000E+16
PLOT LIMITS
X .000E+00
1.000E+01
Y -1.000E+00
1.000E+00
Z -1.000E+00
1.000E+00
JOB: Z1
12-APR-88 20: 36


LOADING CASE 1
STRESS
Y
Z
X
5.000E+02
PRINCIPAL STRESSES
TENSILE
VIEW DIRECTION
80 27 53
VIEWING DIST.
1.000E+16
PLOT LIMITS
X .000E+00 1.000E+01
Y -1.000E+00 1.000E+00
Z -1.000E+00 1.000E+00
MODEL
Y
Z
X
6.000E-01
DEFL. AND STRESSES (ENV.)
DEFLS.
Y
Z
X
5.000E-02
JOB: Z1
12-APR-88 20: 38

1) Make the interfaces (which are the boundaries of subregions) as short as possible.

2) Take advantage of repetitive geometry.

3) When appropriate, divide the structure into the natural regions dictated by its topology.

4) Avoid interfaces passing through areas where high stress concentrations are expected.

5) Use substructuring to isolate areas requiring extensive local detail which would add too many degrees of freedom to a standard analysis.

Initially a classical problem (a plate with a hole in the centre) was analyzed. Now, a similar problem which is more complicated than the first one will be tackled using the finite element analysis package GIFTS.

This flat plate with two internal holes with different diameters is subjected to a tensile load. The tensile force is assumed to be a uniform load ($p = 1kN/m^2$). The dimensions of this plate are as shown in Fig. 3.11. The Young's modulus is $1kN/m^2$ and Poisson's ratio 0.3, and the thickness is 1unit. It problem is assumed that this is a plane strain problem.

From the location of two holes, it can be assumed that the stress concerntrates in the vicinities of the holes (such as areas: I, II, III) due to the reduced amount of material (shown in Fig.3.12). To obtain a reasonable description of the stress distribution in the vicinities of two holes and throughout the plate, a fairly detailed finite element model must be used. When subdividing the structure into many finite elements, more attention should be paid to these areas. In other words, more elements are required in these areas of high stress concentration.

On the other hand, the determination of the boundary conditions is also very important in dealing with the problem. In order to ensure that the deformation of this plate happens only under tension not including any action of bending and

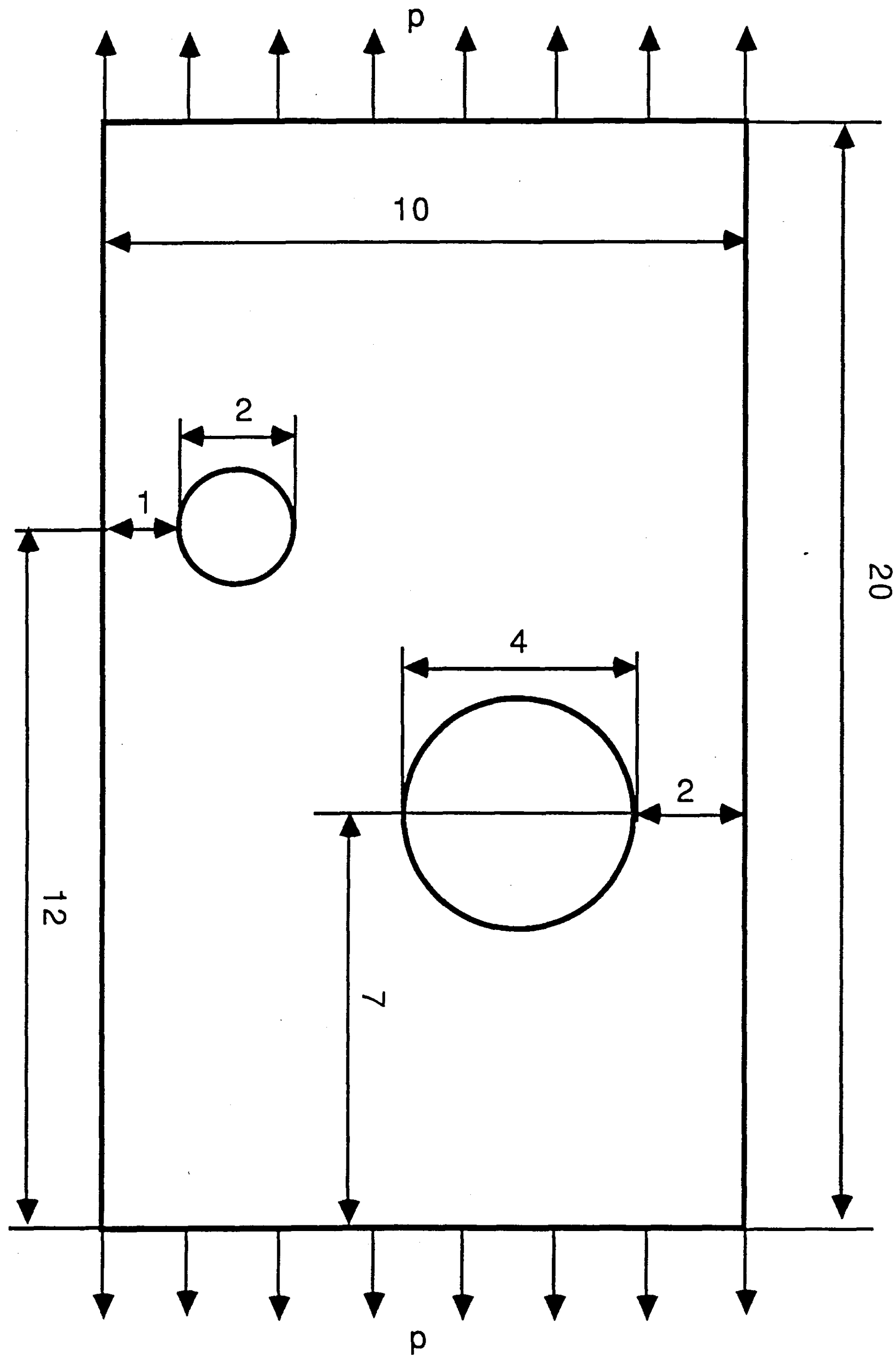


Fig. 3.11

p=1 t=1 E=1 ν=0.3

Units: kN, m

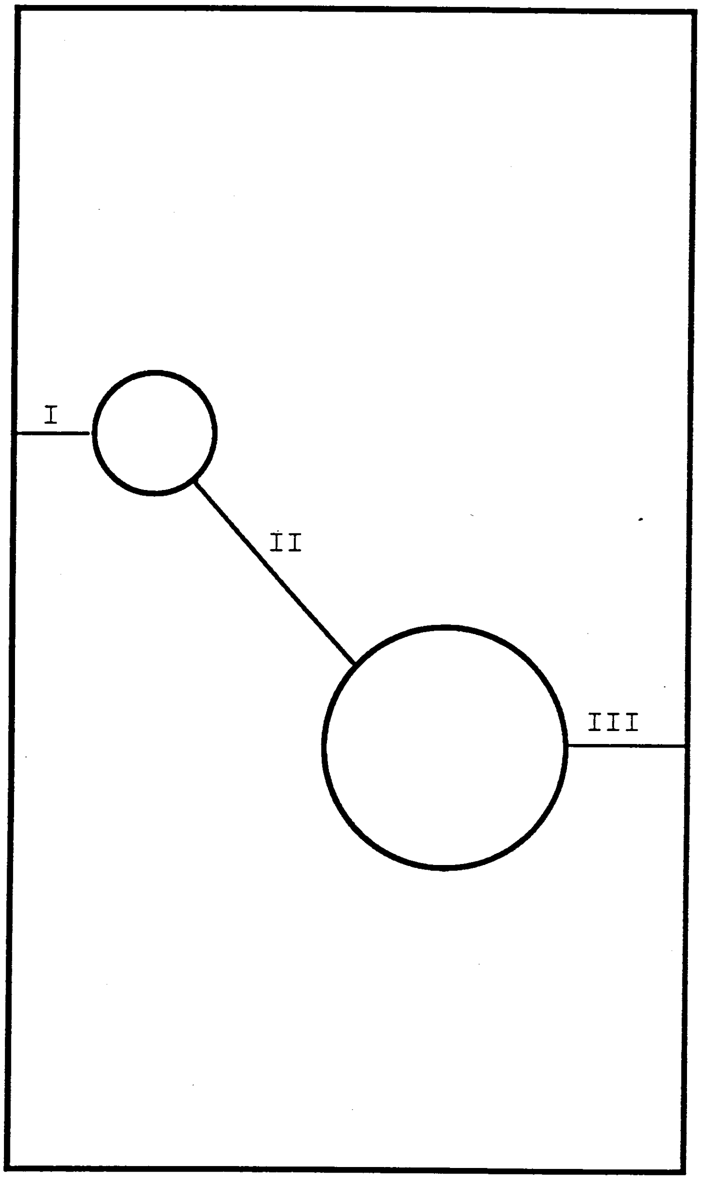


Fig.. 3.12

twisting, some possible boundary conditions were tried. To prevent the plate from bending and twisting, the plate should be constrained at three points. The location of these three points is a key problem in analyzing the structure correctly. Three boundary conditions are used (shown in Fig. 3.13) to determine which one is more suitable for this problem. In these three cases, the results are gained and listed in the Table 3.6.

From the comparison of three boundary conditions, it is seen that the stresses at the same points in these cases are exactly the same, but there are small differences in displacements between them. It is due to the way of putting the restraints to this plate. In fact, the solution is based on the original geometry and so the resulting stresses will be identical irrespective of the location of the minimum constraints. But the final position of the deformed geometry will have the same shape although with a different orientation. However, because the third restraints position (i.e. 95, 51, 88 respectively) is not in the same point in the three cases, the plate has a rotation around point 50. Since in the first and third cases, the third point is far away from point 50 and, apparently, the rotations are larger than the second one, their displacements at relevant points are smaller than the second one.

From the above analysis, it is apparent that the second boundary condition is the most suitable for this situation. It can also be seen that the highest stress concentration occurs at point A.

3. Further discussion

After determining the boundary conditions, more attention is paid to determine the stresses.

According to the guidelines mentioned above, the plate is divided into ten grids (shown in Fig. 3.14). Because 9 noded quadrilateral membrane elements (QM9) are used, all zones should have four sides corresponding to the shape of the

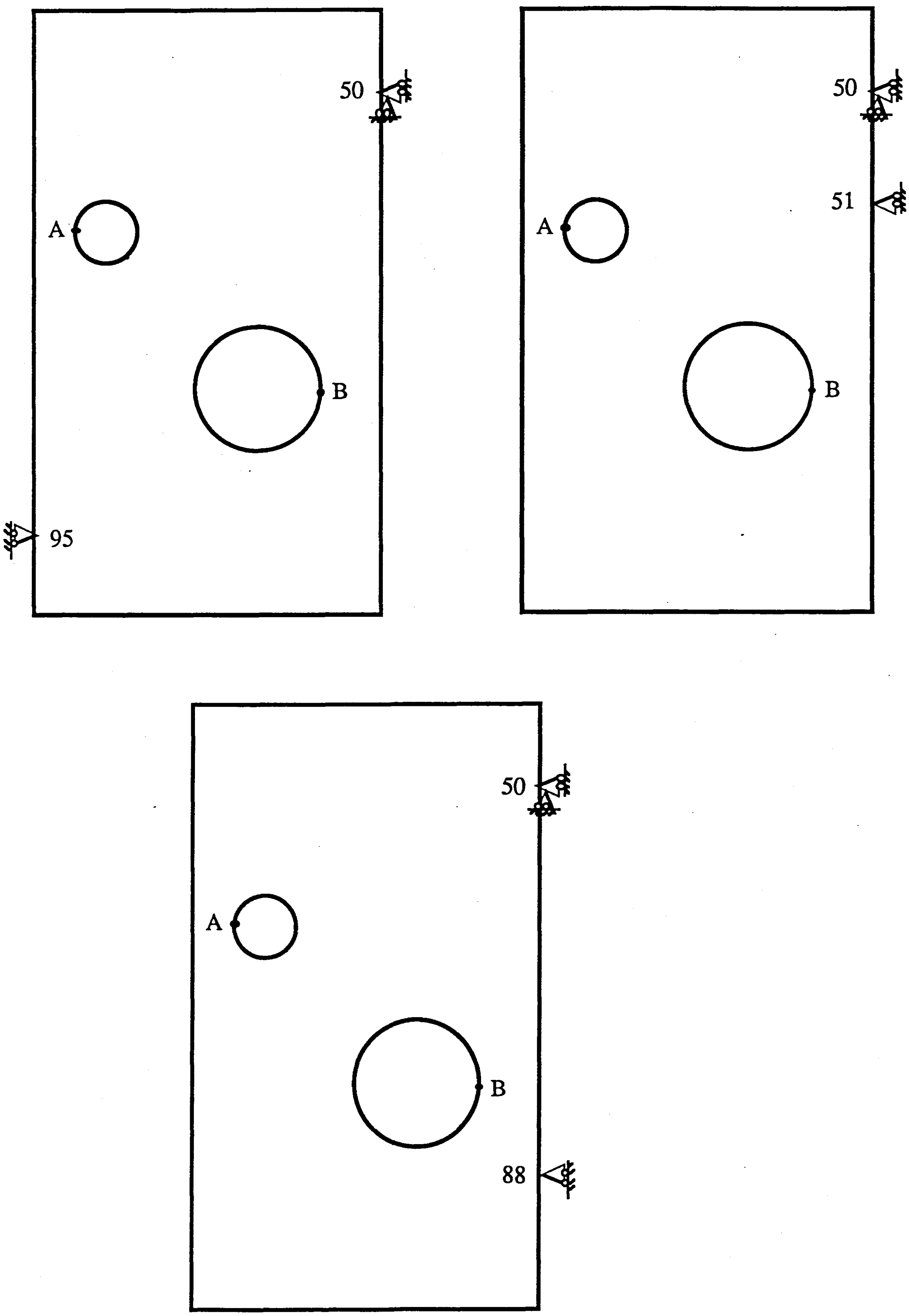
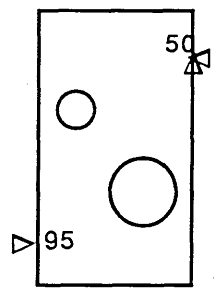
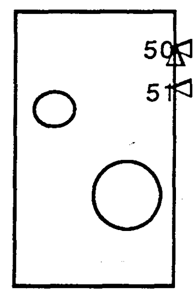
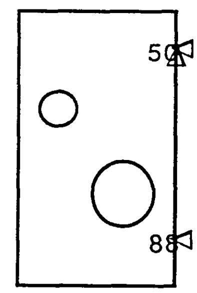


Fig.. **3.13**

Table 3.6 The comparison of solution by using different boundary conditions

			
	(No. Ele.)	(No. Ele.)	(No. Ele.)
Maximum Stress (σ_y)	4.117 (128)	4.117 (128)	4.117 (128)
2nd largest Stress (σ_y)	3.772 (159)	3.772 (159)	3.772 (159)
1st largest displacement (y direction)	-1.096E-04 (9)	-1.144E-04 (9)	-1.124E-04 (9)
2nd largest displacement (y direction)	-1.065E-04 (8)	-1.065E-04 (8)	-1.065E-04 (8)
Unknowns	1383	1383	1383
Half Bandwidth	225	225	225

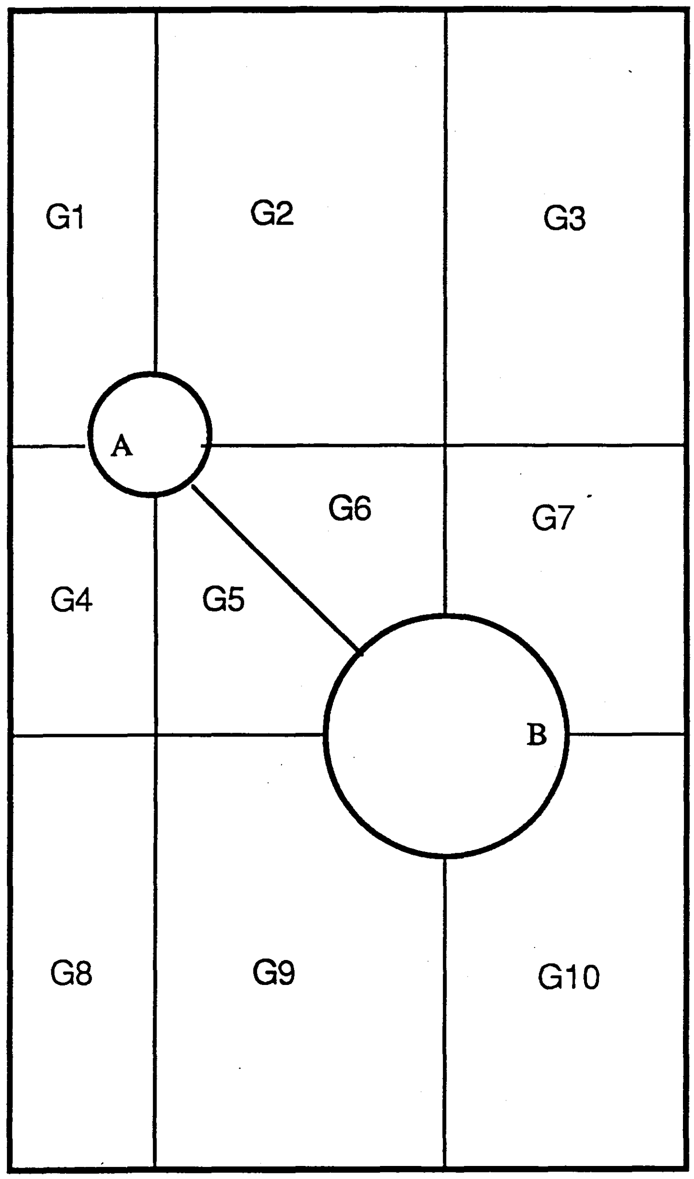


Fig. 3.14

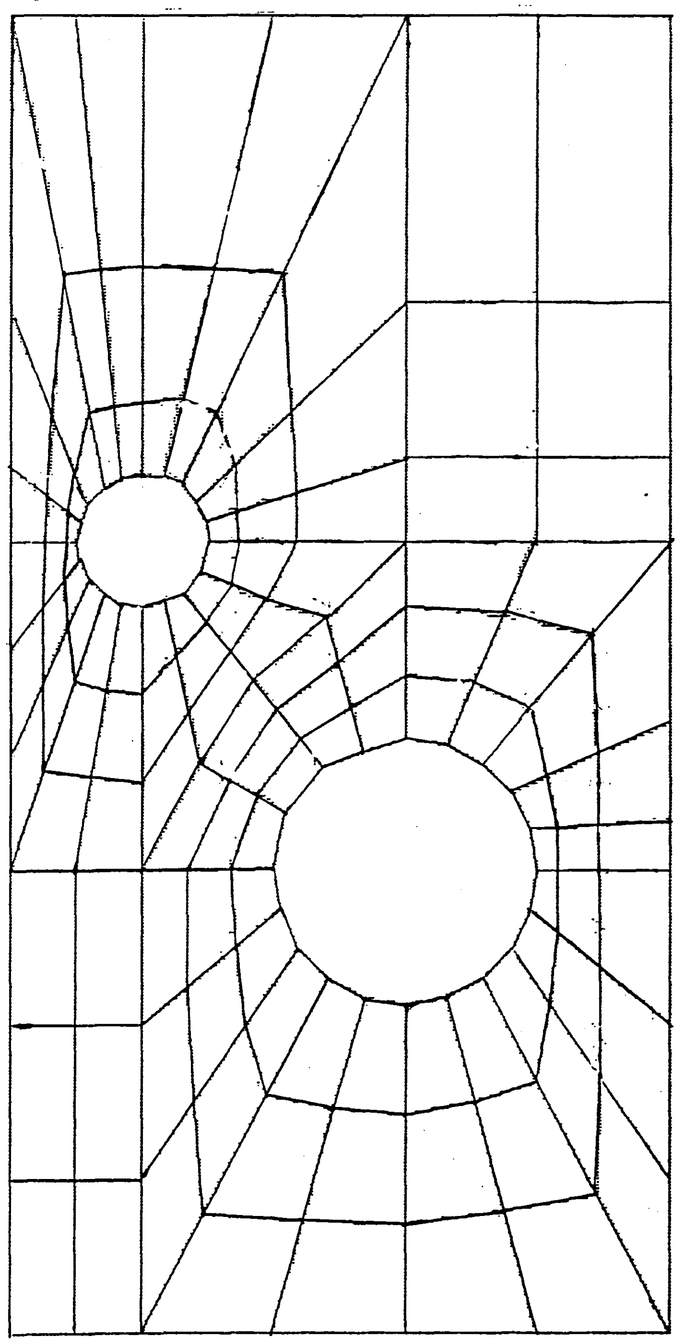


Fig. 3.15

elements. But, with the exception of zones G3 and G8, all zones consist of five sides, so, a composite line which is composed of two lines is defined in each zone. Non-uniform mesh is generated in this problem.

At the beginning, a coarser mesh and a simple element type (QM4) are used only to see where the highest stress concentration area is. It took a very short time to determine that the highest stress concentration occurs near point A, and the second biggest one is near point B. Therefore, using finer elements in these two areas and fewer elements elsewhere is a more efficient way of solving this problem. Secondly, the two areas near point A and B, especially the area near A, are remeshed (as shown in Fig. 3.15)to obtain a detailed description of the stress distribution, and QM9 elements are used for the whole plate. The results show that the biggest stress is 4.117 in the y direction at point A and the largest displacement is -1.144E-04 at node 9. Thirdly, the mesh in the area near point A is generated again to gain more accurate results. Actually, the output data shows that the stress at A is increased slightly to 4.269 (σ_y) but the largest displacement remains the same. The difference between these two stresses is less than 4 percent. Usually, calculation about this problem can be stopped here, but in order to see the change of results, the sizes of elements around point A are further reduced to make the elements as equiaxed as possible. Then, as in the previous two cases largest displacement is obtained, and the biggest stress has a slight difference (less than 0.1 percent) compared with above two, i.e. $\sigma_y = 4.272$. Thus, it can be seen that the biggest stress is about 4.270, and the largest displacement is -1.144E-04. If carrying on refining the elements, the results will almost maintain the same values. The comparison of these results show in Table 3.7.

In Table 3.7, for two later cases, most of the parameters are the same except the stress, because in the last case, only bias of graded mesh is changed to shrink the element size near point A.

As FEM is an approximate method, and the results will depend on the mesh

Table 3.7 Comparison of results of a plate with double holes

Type of elements	QM9	QM9	QM9
No. of elements	174	182	182
No. of nodes	767	807	807
No. of unknowns	1383	1443	1443
Half bandwidth	225	235	235
The stress at point A (σ_y)	4.117	4.269	4.272
The displacement at node 9 (v)	-1.144E-04	-1.144E-04	-1.144E-04
CPU time (Sec.)	449.46	485.16	485.37

chosen, how can it be known that the results of the stresses are in good agreement with the real problem? In this case, attention has to be devoted to the changes of the displacements. If the changes of the displacements for two sets of results are less than 3%-5% when number of elements are changed, it can be said that the results of stresses are very close to the real stresses, and further calculation is unnecessary.

So far, the accuracy of FEA solutions of three problems is considered as a whole, and it is obvious that the accuracy is closely related to the mesh generation of a structure - for instance, the size, shape and type of elements. Therefore, in the following chapter, the errors of FEA results are looked at in closer detail for a single element to illustrate how the size, type and shape parameters affect the accuracy of a solution.

Chapter 4 Assessment of accuracy of FEA solution

4.1 Introduction

Today, the finite element method is becoming more and more attractive to engineers. Structural analysts make particularly frequent use of FEA despite the approximations inherent in the method. The structural behaviour can be modelled as accurately as required by refining the mesh. However, this factor must be balanced with the expense of the analysis which is directly proportional to the increase in CPU time. The sophistication of the analysis should be commensurate with the accuracy required.

"Error" means a difference between computed results and exact results, and any other sources leading to this difference. Errors are divided into several categories. The following terminology will be used (Cook, 1974). A "mistake" is a slip or oversight, such as an input data error, or misunderstanding the purpose of an available program. A "discretization error" may arise from an error of judgment, as in selecting a poor mesh. It may also be the inherent approximate nature of the finite element method. This is the error concerned in this research work. A "computation error" is produced by the digital computer as it manipulates data. Some computation error arises even in a logically correct and properly used program but is aggravated by poor discretization on the part of the user.

The basic concept of FEM is that a finite number of elements (nodes) are employed to model the real behaviour of a structure which, in fact, consists of infinite points. Given the analogy, FEM will produce the errors on the stresses and displacements because of the discretization of a structure. "Discretization error" means inaccuracy arising from the fact that the discretized model, what is actually analyzed, is never an "exact" representation of the physical structure. The element mesh may not fit exactly the structure geometry. For example, the actual distribution of load and possibly variations of thickness and elastic properties may be approximated by simple interpolation functions; boundary conditions may also be approximated. But even if these factors are exactly represented, it is unlikely that the true displacement field can be exactly represented by the piecewise interpolation field permitted by a model having only a finite number of degrees of freedom.

There are many types of elements, such as triangles, rectangles and quadrilaterals with different nodes. 3-noded triangular and 4-noded quadrilateral elements are first-order elements, i.e. there is a constant stress and a linear displacement distribution across the element. In a constant stress field, they can perfectly match the physical situation, but produce the errors in higher order stress fields. Other elements in common use are 6-noded triangular and 8-noded (or 9-noded) quadrilateral elements which have linear stress and second-order displacement variation across the elements. Such second order elements (if square or equilative trianglular) are better than first-order elements, and produce no errors in both constant and linear stress fields. Even the same topology but different shape will produce different errors. An unfortunate choice of element shape or size when discretizing a structure aggravates subsequent numerical error in computation. These are termed shape sensitivity and size effect.

4.2 Survey of existing work on single element tests

To assess the accuracy of FEA results, the single element test plays an important role in understanding the distortion, size, and other effects of a finite element with different shapes and nodes applied to a range of membrane type elements. The application to other types will be self-evident. The single element test was originally devised by Robinson and Haggenmacher (1970).

The report presented by Robinson (Robinson, 1985) describes a general approach to element testing which can be utilised for a variety of element types with a varying number of nodes. It is known as the 'Continuum-Region-Element' Method and can be applied to membrane, plate bending or solid elements.

This method is based upon placing a single 2D finite element somewhere in a region with a known stress field. The stress fields used are:

(a) Constant Direct Stress

(b) Constant Moment

(c) Linear Moment (with parabolic shear)

Three shape parameters are considered:

(a) Aspect ratio (AR)

(b) Skew angle (SK)

(c) Taper (T)

By varying individual shape parameters and, for some tests, more than one parameter, it is possible to illustrate how this distortion to the element shape subsequently produces finite elements which vary from the known theoretical values for displacements and stresses.

The work done by Burrows (Burrows, 1986) is based on ideas proposed by Robinson. The tests considered the effect of varying single and combined shape parameters, implemented under an automatic procedure, the results being stored in a database. Burrows used the shape parameter definitions and some theoretical stress fields proposed by Robinson. As noted in Robinson's report, the effects of edge curvature were therefore ignored. In addition it was found that the Linear Moment Loading, which results in a parabolic direct and shear stress field, was dominated by the constant term. This tended to 'swamp out' errors due to shape caused by higher order stress terms.

The later part of this paper presents a discussion of the procedures involved in deriving an element stiffness matrix and calculating stresses from the displacement field. From this it is possible to identify, the underlying parameters which influence the error most significantly for certain simple stress fields.

In Robinson's other report (Robinson, 1987) he extends the definitions of shape parameters described in his earlier document to include the effects of edge curvature and midside node distortion. In addition to the four previously proposed measures (aspect ratio, skew and two tapers) he proposes eight offsets. These are the tangential and normal deviations of each midside node from its ideal position. He illustrates the use of the parameters with a range of eight examples. In his earlier document, the author demonstrated the link between his four shape parameters and the Jacobian. Unfortunately, with the addition of eight further parameters, the link cannot directly be expressed except in certain special cases.

The approach put forward by Barlow (Barlow, 1987) is quite different from those reviewed previously. The basis of this work relies on:

(i) The geometry of the finite element is described by a polynomial of the local coordinate parameters. The maximum order of the polynomial characterizes the range of elements in a polynomial. Of order 1 is a parallelogram, of order 2 in a tapered element and so on.

(ii) The strain field can also be characterized by a polynomial of a certain order.

The process of successively refining a mesh, if carried out properly, results in the reduction in the order of both these polynomials. For example, breaking a highly distorted quadrilateral into four by one division in each dimension will, if carried out optimally, result in four elements with less distortion. The process of refining a mesh to improve the approximation to the true strain field amounts to reducing the order of the polynomial approximated by each element. In the author's view, the errors can be determined within certain bands identified as 0(1) (for errors of the same magnitude as the strains), 0(E) for errors of E multiplied by the strains where E is the measure of the element distortion, 0(E^2) etc.

The errors are calculated analytically, and attributed to two sources:

(i) Calculating strains from displacements

(ii) Calculating the stiffness matrix by numerical integration

Burrows (Burrows, 1988) formulated a set of single element tests and defined a set of suitable shape parameters in order to be able to assess the likely performance of membrane finite elements prior to analysis.

The process of reviewing the literature and carrying out over a thousand tests on single elements revealed that Aspect Ratio and Skew do not affect the inherent element performance. Distortions due to Taper and Edge Curvature cause deterioration in the accuracy of the element stiffness matrix and the stress recovery process. A set of five single element benchmarks are tabulated highlighting the range of inaccuracies caused by shape distortions.

4.3 Description of the tests done in this study

All single element tests are defined in a region where there is a known stress distribution. The nodal forces and displacements can then be calculated and applied for each individually defined element.

Two types of elements were used in the single element tests:

(a) Four node quadrilateral

(b) Nine node quadrilateral

all with plane stress assumptions.

4.3.1 Shape parameter definitions

The shape parameters involved in the tests are:

(a) Aspect ratio (AR)

(b) Skew angle (θ)

(c) Height (h)

For rectangular elements, only AR is used to define the element shape. AR is equal to a/b (as shown in Fig. 4.1). There are two parameters in parallelograms, viz: θ and hight h (as shown in Fig. 4.2). For a trapezium, the shape parameters are AR and θ, but the definition of AR is different from that used in rectangular elements (as shown in Fig. 4.3).

4.3.2 The stress fields used in single element tests

The finite element method is a very powerful tool in structural analysis. There are different shapes of elements with varying numbers of nodes. Usually, different shapes of elements will produce different errors, and, if the polynomial order of the stress field is greater than that contained within the element formulation, the stresses will almost certainly be in error. For example, 4-noded elements are perfect in the constant stress field, i.e. for a linear displacement field the results are very close to theoretical values. However, if 4-noded elements are used in the linear stress field, larger errors will be obtained. Most earlier work (Robinson, 1976; Burrows, 1988) has dealt with single element tests, however in these works some special stress fields, such as constant direct stress, constant moment and linear moment applied to a cantilever. To find a general error assessment for different problems, two approaches exist:

1. An assumed displacement function (Cook, 1974)
2. An assumed stress function (Timoshenko, 1970)

Here, only the second method is used.
(Insert one paragraph - see the following page)

(after the last line on Page 55)

An assumed stress function is a more common expression of stress fields than a function used solely for specific stress field. To satisfy the conditions of compatibility and equilibrium, different combinations of coefficients in the stress function can be employed. However, in the present study, only one set of coefficients is used for each stress field. Although they have a particular set of coefficients, hopefully, they can represent a range of stress field, and also illustrate different effects of various factors.

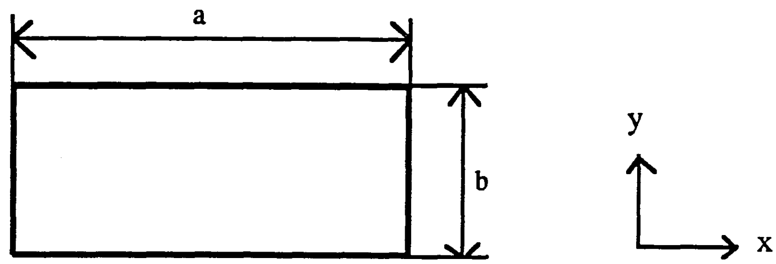


Fig. 4.1

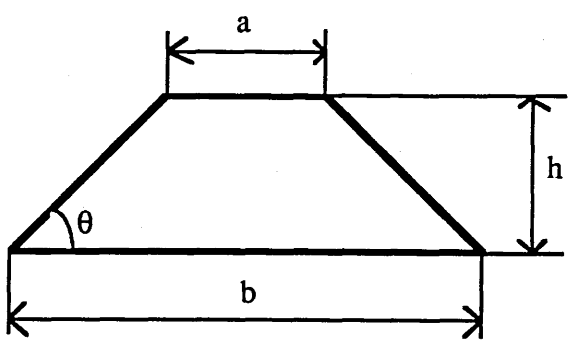


Fig. 4.2

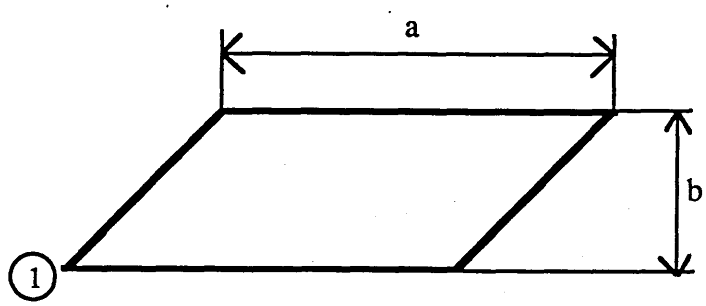


Fig. 4.3

4.3.3 Stress function

Primarily, for a continuum, with a stress function of $\phi(x,y)$, stresses σ_x, σ_y, τ_{xy} can be obtained by $\partial^2\phi/\partial^2 y$, $\partial^2\phi/\partial^2 x$ and $-(\partial^2\phi/\partial x\partial y)$ respectively. If we assume a plane strain/stress problem, the strains can be obtained from stresses, and displacements from strains. The following example shows the procedure.

Assuming a stress function:

$$\phi = x^3 + x^2y + xy^2 + y^3 \tag{4.1}$$

it should satisfy the following conditions:

a) Compatibility equation

$$\frac{\partial^4\phi}{\partial x^4} + 2\frac{\partial^4\phi}{\partial x^2\partial y^2} + \frac{\partial^4\phi}{\partial y^4} = 0 \tag{4.2}$$

b) Differential equations of equilibrium:

$$\begin{cases} \dfrac{\partial\sigma_x}{\partial x} + \dfrac{\partial\tau_{xy}}{\partial y} = 0 \\ \dfrac{\partial\tau_{xy}}{\partial x} + \dfrac{\partial\sigma_y}{\partial y} = 0 \end{cases} \tag{4.3}$$

for plane stress: ($\sigma_z = 0$):

$$\varepsilon_x = (\sigma_x - \nu\sigma_y)/E = \partial u/\partial x \tag{4.4}$$

$$\varepsilon_y = (\sigma_y - \nu\sigma_x)/E = \partial v/\partial y \tag{4.5}$$

$$\gamma_{xy} = \partial u/\partial y + \partial v/\partial x \tag{4.6}$$

there are:

$$\begin{cases} \sigma_x = \partial^2\phi/\partial^2 y = 2x + 6y \\ \sigma_y = \partial^2\phi/\partial^2 x = 6x + 2y \\ \tau_{xy} = -(\partial^2\phi/\partial x\partial y) = -2\,(x + y) \end{cases} \tag{4.7}$$

for plane stress problem,

$$\varepsilon_x = (\sigma_x - \nu\,\sigma_y)/E = [(2\text{-}6\nu)x + (6\text{-}2\nu)y]/E \tag{4.8}$$

$$\varepsilon_y = (\sigma_y - \nu\,\sigma_x)/E = [(6\text{-}2\nu)x + (2\text{-}6\nu)y]/E \tag{4.9}$$

$$\gamma_{xy} = \tau_{xy}/G = -2(x+y)/G \tag{4.10}$$

because,

$$\varepsilon_x = \partial u/\partial x, \quad \varepsilon_y = \partial v/\partial y, \quad \gamma_{xy} = \partial u/\partial y + \partial v/\partial x \tag{4.11}$$

These three strain components are expressed by two functions u and v, and the following equation:

$$\frac{\partial^2 \varepsilon_x}{\partial y^2} + \frac{\partial^2 \varepsilon_y}{\partial x^2} = \frac{\partial^2 \gamma_{xy}}{\partial x \partial y} \qquad (4.12)$$

This differential relation, called the condition of compatibility, must be satisfied by the strain components to secure the existence of functions u and v connected with the strain components.

By integration of strains ε_x and ε_y, we have

$$u = \int \varepsilon_x dx = [(1\text{-}3\nu)\, x^2 + (6\text{-}2\nu)\, xy] + f_1(y) + c_1]/E \qquad (4.13)$$

$$v = \int \varepsilon_y dy = [(6\text{-}2\nu)\, xy + (1\text{-}3\nu)\, y^2] + f_2(x) + c_2]/E \qquad (4.14)$$

Here, the constants c_1 and c_2 are the rigid body displacements, they can be defined as zero. Now, the problem is how to determine $f_1(y)$ and $f_2(x)$.

Because $\gamma_{xy} = \partial u/\partial y + \partial v/\partial x$ (4.15)

ie.

$$\frac{1}{E}(6 - 2\nu)\, x + \frac{df_1(y)}{dy} + \frac{1}{E}(6 - 2\nu)\, y + \frac{df_2(x)}{dx} = \frac{2}{G}(x+y)$$

(4.16)

$$\frac{1}{E}(6 - 2\nu)\, x + \frac{df_2(x)}{dx} = -\frac{2}{G}\, x$$

(4.17)

$$\frac{1}{E}(6 - 2\nu)\, y + \frac{df_1(y)}{dy} = \frac{2}{G}\, y$$

(4.18)

so, $f_2(x) = -(5 + \nu)x^2/E + c_3$ (4.19)

$f_1(y) = -(5 + \nu)y^2/E + c_4$ (4.20)

In these equations c_3 and c_4 are rigid body displacements, they can be considered to be equal to zero.

Therefore,

$$u = \{[(1-3\nu)\, x^2 + (6-2\nu)]\, xy - (5+\nu)\, y^2\}/E \qquad (4.21)$$

$$v = \{[(6-2\nu)\, xy + (1-3\nu)\, y^2] - (5+\nu)\, x^2\}/E \qquad (4.22)$$

4.3.4 Applied nodal forces and displacements

Once the stress and displacement equations are obtained, the theoretical solutions for stresses and displacements can be calculated and, single element tests can be carried out. The prescribed displacements specified for each single element test will be the horizontal (u) and the vertical (v) displacements at node 1 in each single element and the vertical displacement (v) at node 4 in the quadrilateral elements. For each loading case, the applied nodal forces corresponding to the known stress state will be calculated. These forces will be computed by finding the stress resultant distributions at the boundaries and transforming them into boundary loadings. Finally, these values are replaced by the equivalent nodal forces in a manner which is consistent with the element formulation.

For a 4-noded element irrespective of its shape, the equivalent nodal forces on one edge are (as shown in Fig. 4.4):

$$\begin{bmatrix} F1 \\ F2 \end{bmatrix} = \frac{1}{6}\begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix}\begin{bmatrix} p1 \\ p2 \end{bmatrix} \qquad (4.23)$$

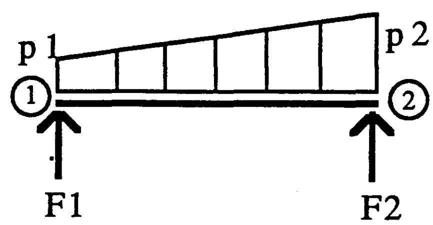


Fig. 4.4

For an 8-noded (or 9-noded) element, the equivalent nodal forces on one side are (as shown in Fig.4.5):

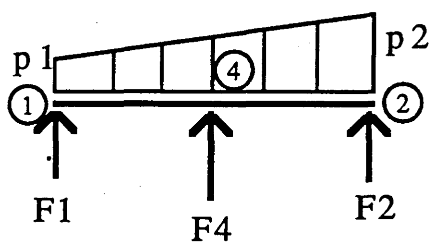


Fig. 4.5

$$\begin{bmatrix} F1 \\ F4 \\ F2 \end{bmatrix} = \frac{1}{6} \begin{bmatrix} 1 & 0 \\ 2 & 2 \\ 0 & 1 \end{bmatrix} \begin{bmatrix} p1 \\ p2 \end{bmatrix} \qquad (4.24)$$

After the nodal forces are calculated, single element tests can be started by changing the aspect ratio (AR), skew angle (θ) and height (h) to test the element shape sensitivity and using 4-noded and 9-noded elements in different stress fields while at the same time comparing the accuracy of the results.

All the single element tests presented in this work were solved by the GIFTS package or PC-ANSYS/LINEAR. Young's Modulus E and Poisson's Ratio ν are 1 and 0.3 respectively. The forces applied at corner points of the element are calculated from the known stress functions. The applied load and boundary conditions are shown in Fig. 4.6.

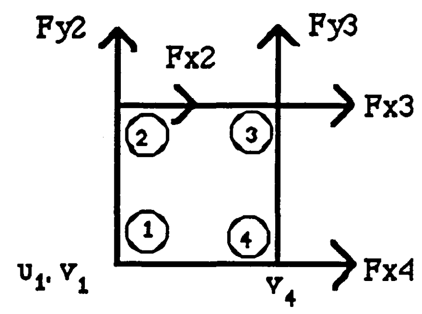


Fig. 4.6

4.3.5 Error measure

Since the FEM is an approximation, there must be some errors between the FEA and theoretical results, and different shape of elements will compound these error values. In determining the accuracy of the results, and deciding whether or how a structure should be remeshed, the method of measuring these errors has special importance.

The emphasis is how to measure the errors, which should be easy to assess the accuracy of the FEA resutls. Various methods for the calculation of the errors are proposed (Hoog, 1973). The method used in this work is discussed below.

1. Stress error (SE)

For four node elements, the three individual errors (SE_{xx}, SE_{yy}, SE_{xy}) are calculated from the finite element results and their respective theoretical values. Since the theoretical stresses are calculated at 4 nodes but the FEA results which have constant stress accross the element are output from the centroid, the FEA stresses are compared as follows:

$$SE_{xx} = (\sigma_{txi} - \sigma_{fx})/\sigma_{max} \quad (4.25)$$

where

σ_{txi}: theoretical stress in the x direction at point i (i =1, 2, 3, 4)

σ_{fx}: FEA stress in the x direction output from centriod of the element

σ_{max}: theoretical maximum equivalent stress

From the same calculation, SE_{yy} and SE_{xy} can be obtained:

$$SE_{yy} = (\sigma_{tyi} - \sigma_{fy})/\sigma_{max} \quad (4.26)$$

$$SE_{xy} = (\tau_{txyi} - \tau_{fxy})/\sigma_{max} \quad (4.27)$$

Finally, the total error is:

$$SE = (SE_{xx}^2 + SE_{yy}^2 + SE_{xy}^2)^{1/2} \quad (4.28)$$

at each of the corner nodes, and the largest value will be taken into consideration.

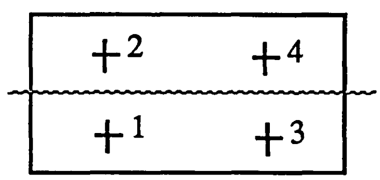


Fig. 4.7

For 9-noded elements, the stresses are output from 4 points (as shown in Fig. 4.7). The error measure is similar to that used for 4-noded elements (see above Equations (4.25), (4.26), (4.27) & (4.28). However, the theoretical stresses are calculated at the same points as the output stresses.

(2) Displacement error (D)

For both 4-noded and 9-noded elements, only the displacements at 4 corner points are considered. The two displacement errors (D_x, D_y) calculated in a similar way are given by the following:

$$D_x = (D_{txi} - D_{fxi})/D_{max} \quad (4.29)$$

$$D_y = (D_{tyi} - D_{fyi})/D_{max} \tag{4.30}$$

$$D = (D_x^2 + D_y^2)^{1/2} \tag{4.31}$$

where D_{txi}, D_{tyi}: theoretical displacement errors for u and v at node i

D_{fxi}, D_{fyi}: FEA displacement errors for u and v at node i

D_{max}: maximum theoretical displacement over an element

at each of the corner nodes.

The maximum of SE and D over the element is then obtained. Thus for any particular combination of shape parameters, there is just one value of SE and one value of D per element per loading case. It is of course possible that when varying the shape parameters the position of the maximum error can move from one node to another.

The error measure used in this research is exactly the same as that used by Burrows, to permit a direct comparison.

4.4 Constant stress field

In this field, all the stress values are constant, i.e. the stress function is a second order equation, viz:

$$\phi = x^2 + xy + y^2 \tag{4.32}$$

hence,

$$\sigma_x = 2, \quad \sigma_y = 2, \quad \tau_{xy} = -1 \tag{4.33}$$

As all the stresses are constant, 4-noded elements are used to model this problem. It is shown that the FEA results (stresses and displacements) are exactly the same as the theoretical results whenever AR is different (even up to AR=15) and also 8-noded elements are perfect in this stress field.

4.5 Linear stress field

4.5.1 Stress function

Another issue considered in this work is the solution of linear stress field. The stress function is given in 4.3.3;

$$\phi = x^3 + x^2y + xy^2 + y^3$$

and the stresses are:

$$\sigma_x = 2x+6y, \quad \sigma_y = 6x+2y, \quad \tau_{xy} = -2\,(x+y)$$

and the displacements are:

$$u = \{[(1\text{-}3\nu)\,x^2 + (6\text{-}2\nu)]\,xy - (5+\nu)\,y^2\}/E$$

$$v = \{[(6\text{-}2\nu)\,xy + (1\text{-}3\nu)\,y^2] - (5+\nu)\,x^2\}/E$$

4.5.2 Using different order of elements

Firstly, the results are presented using lower order elements (4-noded elements). Because the stresses are linear functions of x and y, and displacements (u,v) are quadratic functions of x & y, the elements situated in different locations will produce different errors. In particular larger errors will be produced as the corrdinates x and y decrease. Thus, if an element is taken from the area which is near the origin of coordinates (0,0), larger errors of stresses and displacements will be obtained even for square elements. The stresses (σ_x, σ_y, τ_{xy}) are output from the centroid of the element, which should equal the average of the stresses at 4 nodes. The displacements are output from 4 nodes.

For the 9-noded elements, used in the linear stress field, the errors are very close to zero. The stresses are printed out from 4 points.

Both 4-noded and 9-noded elements are used with three different shapes, viz: rectangle, parallelogram and tripezium. Therefore, some shape parameters are used in the single element tests:

(1) aspect ratio (AR): 1, 3, 5, 8, (15)

(2) skew angle (θ): 15, 30, 45

(3) height (h): 1

4.5.3 Distortion effect

1. Rectangular element

For this kind of element, there is only one shape parameter - AR.

From the single element tests it is known that if an element is moved to a different location, the errors will be different (this will be discussed in a later section). To isolate the effect of aspect ratio, a single element, at an identical location (centroid (10.5,15.5)) and with the same area 1, is selected for the AR test. The results are shown in Fig. 4.8 & 4.9 which are not affected by the location effect. As shown in Fig. 4.8 by using a 4-noded quad (QM4), when AR is increased (from 1 to 15), the errors of the stresses and displacements are increased as well. Obviously, the stress and displacement errors are smaller than before. All the plots come in pairs, the first being the stress error, the second the displacement error.

For 9-noded elements, the FEA results show a perfect match in the linear stress field. Even when AR = 8, very small errors are produced (stress error = 0.02%, displacement error = 0.163%) as expected.

2. Parallelogram

The stress and displacement errors of using QM4 are shown in Fig. 4.10 and Fig. 4.11. The errors of stress and displacement are quickly increased with the variation of AR from 1 to 8. Errors grow from 4.17% to nearly 30% in the stress. If AR is kept constant, and the skew angle is changed only, the stress error will decrease as the angle is increased (Fig. 4.12).

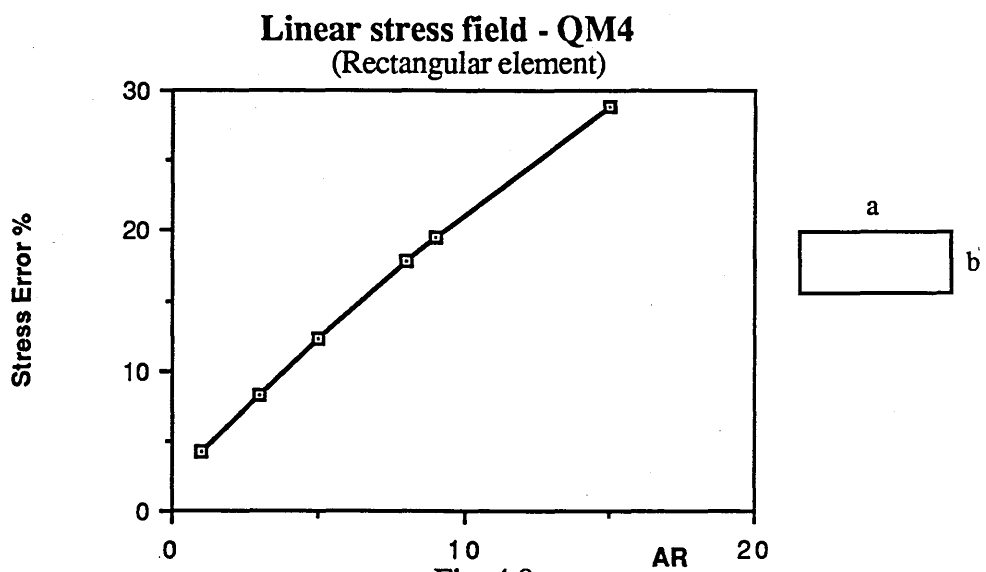


Fig. 4.8

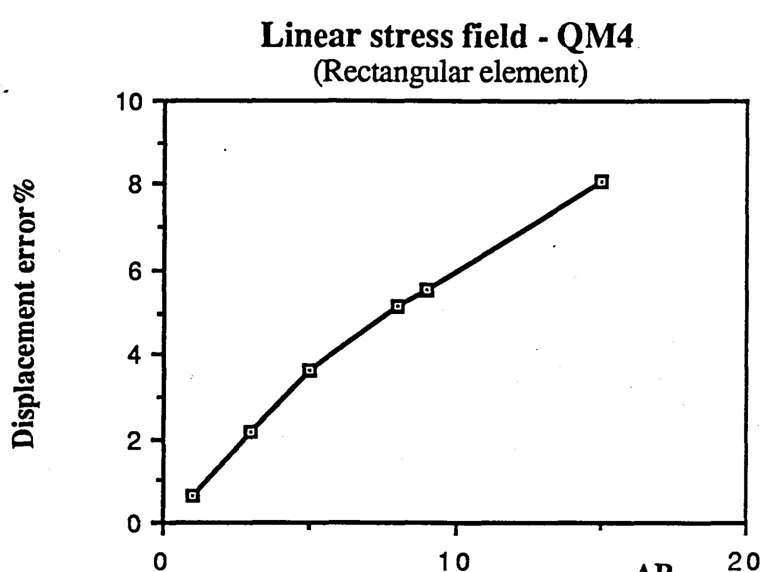


Fig. 4.9

3. Trapezium

From the Fig. 4.13 and Fig. 4.14 it can be known that the increase in the errors of stresses and displacements are proportional to the increase of AR.

It is clear from the above three shapes of elements that high skew leads to most

Parallelogram (Linear stress field) -QM4

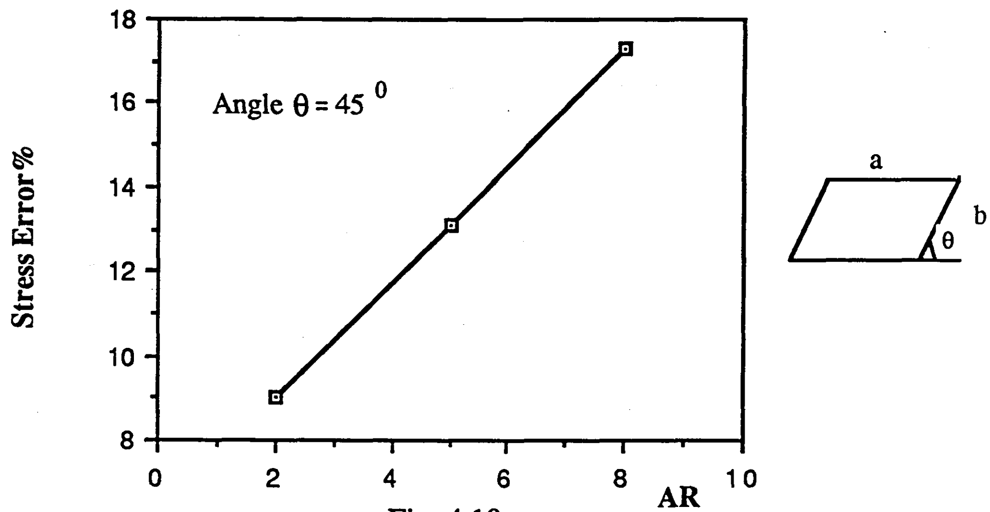


Fig. 4.10

Parallelogram (Linear stress field) - QM4

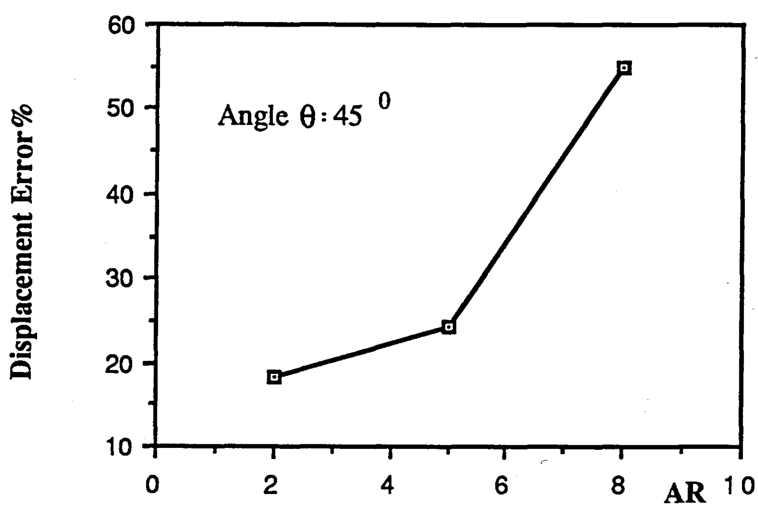


Fig. 4.11

significant errors, and taper is more sensitive than changing AR in rectangular elements. Therefore, when meshing a structure, the elements should be kept as rectangular (or square) as possible. If the skew angle is less than 10^{o}, a very large error will be produced.

4.5.4 Size effect

Apart from the shape parameters, the size effect is another important source of the errors.

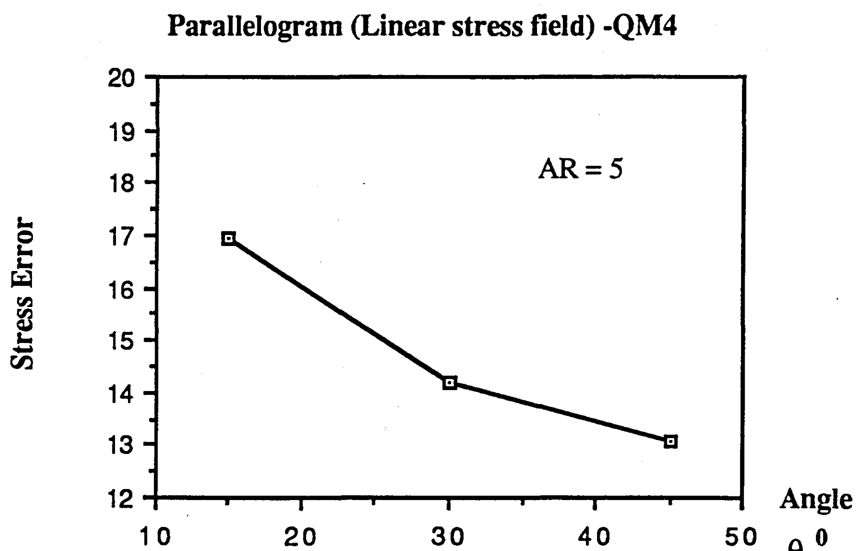


Fig. 4.12

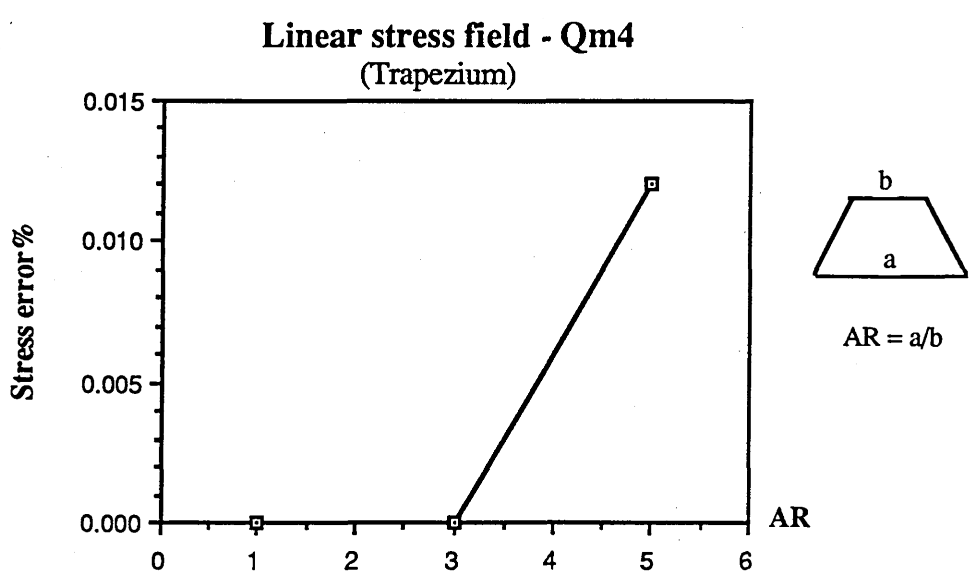


Fig. 4.13

From these single element tests in the linear stress field (which position is shown in Fig. 4.15), it is seen that the errors are significantly affected by the size of element: as increase of the size, the stress errors vary linearly with size. The relationship between the ratio of the element size and the stress error (SE) is (shown in Fig. 4.16):

$$SE = 0.7758 + 3.5\ L \tag{4.34}$$

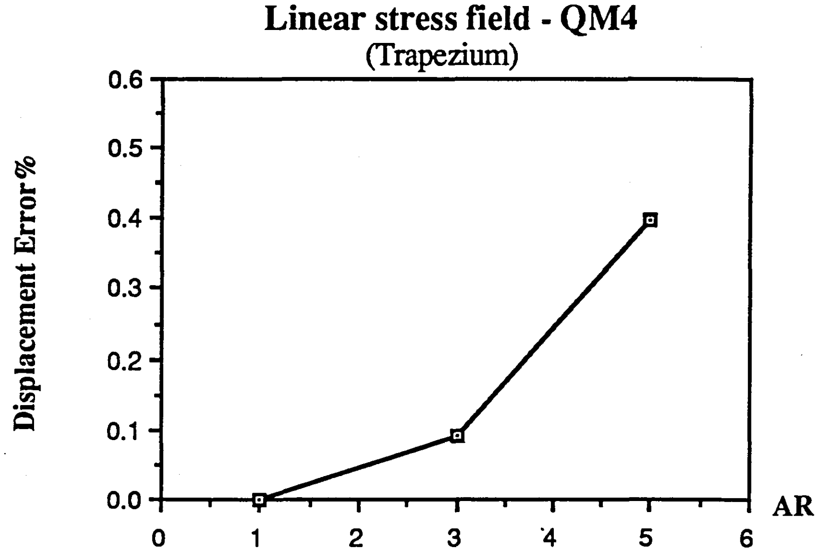


Fig. 4.14

where $L = L_l/L_s$: L_l is the length of the bigger element

L_s is the length of the smaller element

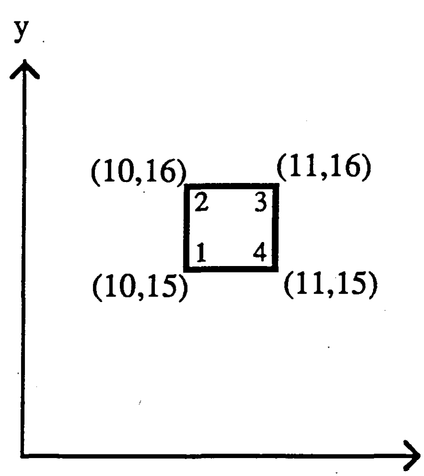


Fig. 4.15

4.5.5 Orientation effect

This is a less significant factor affecting the accuracy of the FEA results. All tests are based on the square element (1 by 1) by rotating the element through certain angles at the same location. The errors of stress and displacement by rotating the element in different angles are shown in Fig. 4.17.

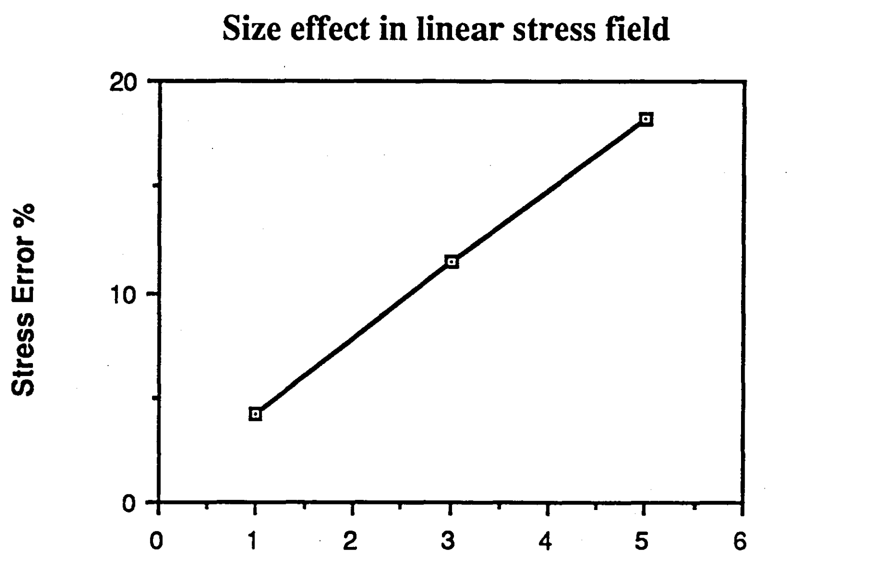


Fig.4.16

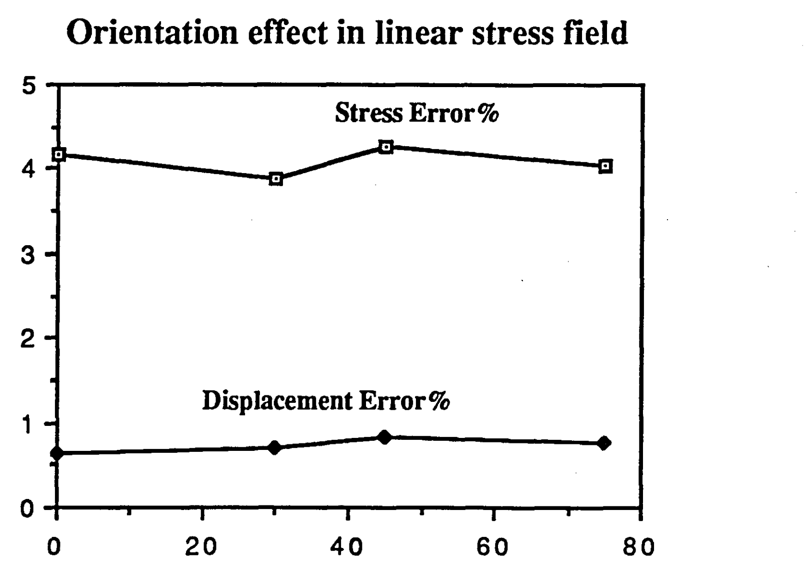


Fig. 4.17

4.6 Quadratic stress field

4.6.1 Stress function

To satisfy the conditions mentioned above, the stress function assumed is:

$$\phi = x^4 - 6x^2y^2 + x^3y + y^4 \qquad (4.35)$$

thus the stresses evaluated are:

$$\sigma_x = -12x^2 + 6xy + 12y^2 \qquad (4.36)$$

$$\sigma_y = 12x^2 + 6xy - 12y^2 \qquad (4.37)$$

$$\tau_{xy} = -3x^2 + 24xy - 3y^2 \qquad (4.38)$$

From the equations, the displacements can be expressed as:

$$u = [-4(1+\nu)x^3 + 3(1-\nu)x^2y + 12(1+\nu)xy^2 - (3+\nu)y^3]/E \qquad (4.39)$$

$$v = [12(1+\nu)x^2y + 3(1-\nu)xy^2 - 4(1+\nu)y^3 - (3+\nu)x^3]/E \qquad (4.40)$$

Once these stress and displacement equations have been obtained, the single element tests can be carried out. Because of limited time, the tests for different element shapes could not be done (they are similar to the linear stress field). Only square and rectangular elements are used to predict the errors produced by 4-noded and 9-noded elements in the parabolic stress field.

(Insert two paragraphs - see the following page)

4.6.2 The discussion of the solution

1. Aspect ratio (AR)

In this stress field, there are increases of errors in stress and displacement with the increase of AR (Fig. 4.18). When AR varies from 1, 3 to 5, the stress errors using 4-noded quads are 7.64%, 13.35% to 21.92% respectively, but the change of displacement errors using 9-noded elements is very rapid (0.036%, 9.62% and 47.76% respectively). Two possible reasons for this rapid increase are:

1) the nature of this stress field (Fig. 4.19)

(at the end of section 4.6.1 on Page 71)

From the single element tests it is known that there is a location effect (see Fig 4.20 to 4.23 below). As coordinates of the sample element decrease, the stress and displacement errors will increase, inevitably the results would be better if these were single element tests for a set of reference elements at different locations in the stress field where differing severities of stress variation exist. However, this work make a limited contribution to the global work on the assessment of FEA results. The reader should clearly note that a more representative location of the element with large linear variation in the stress field over its length would have been more useful. Similarly a more exhaustive look at the effect of other stress function fields would give more comprehensive results. What has been presented in this thesis is a philosophical approach, exexemplified throught specific stress functions and one reference element location.

2) the location effect

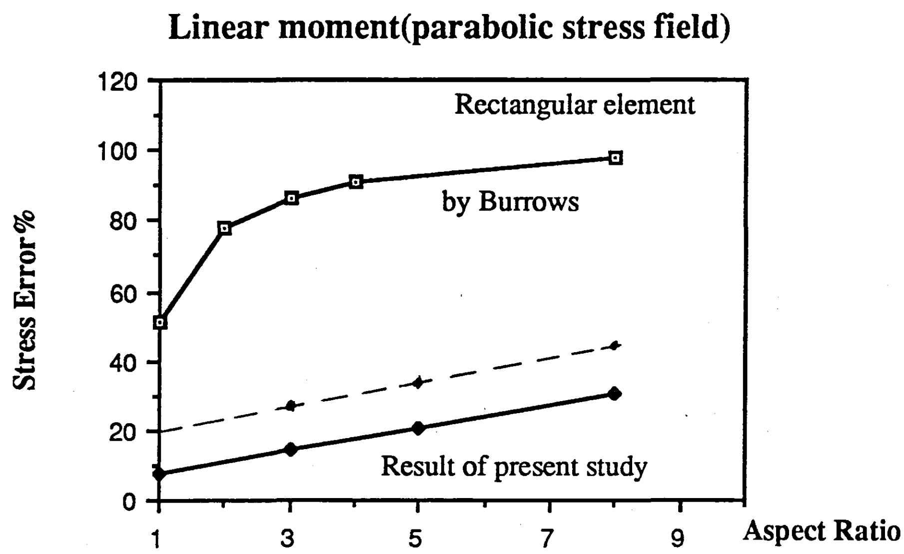


Fig.4.18

From Fig. 4.19 it can be seen that if an element is located in the area of stress changing rapidly, large errors will be produced, and if the element size is too big, it could not be compatible with the stress field.

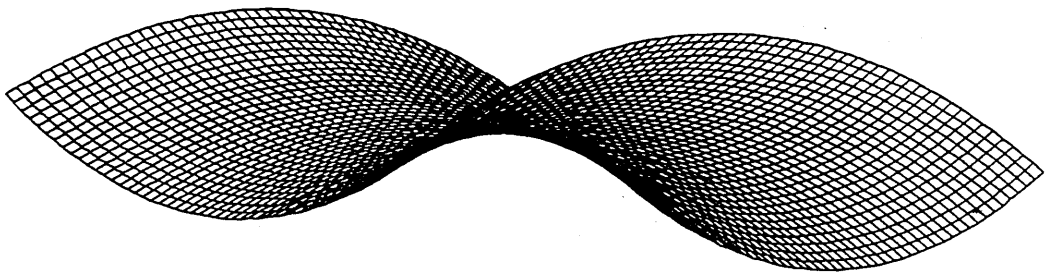
2. Location effect

This has not been studied by early researchers. This effect varies from one stress field to another, i.e. it is very much depending on the stress distribution of the structure (the distribution of this stress field is shown in Fig. 4.19). For testing the location effect, all single elements used are square and 4-noded. By keeping the y coordinates the same for each element, viz:

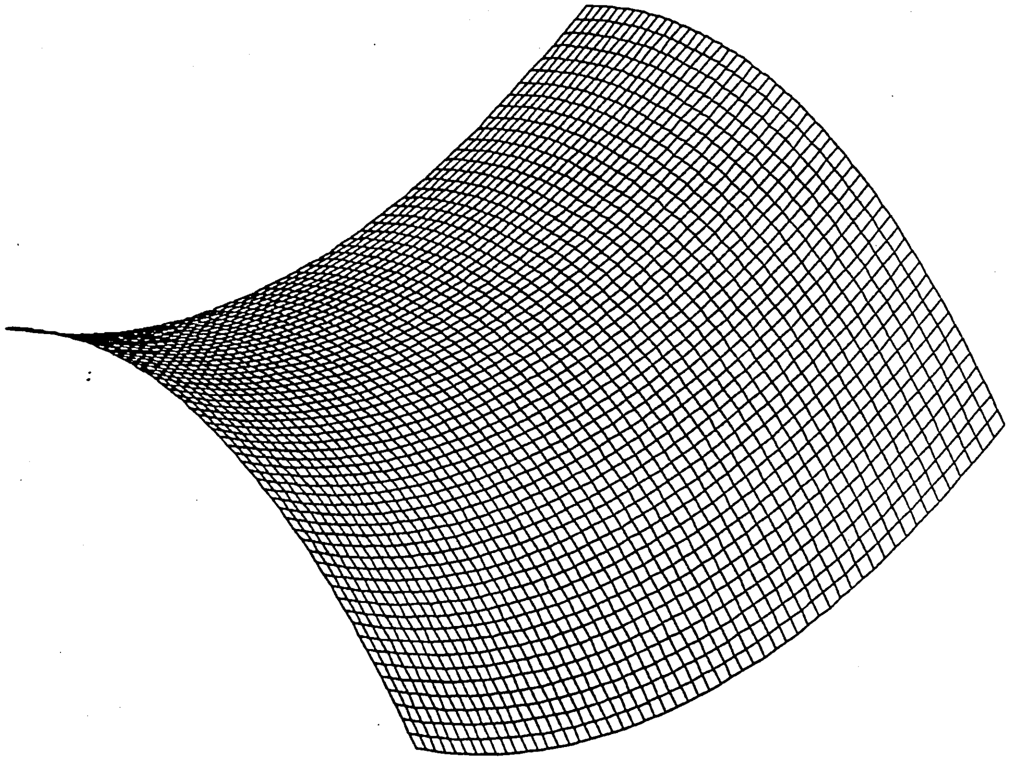
$$y = [15\ 16\ 16\ 15]^{-1} \qquad (4.41)$$

the variation of the stresss errors is shown in Fig. 4.20. The Figure 4.21 shows that when fixing the y cordinates the errors of displacement are different with the variation of the element loaction along x direction.

Similarly, if the x coordinates are kept as:

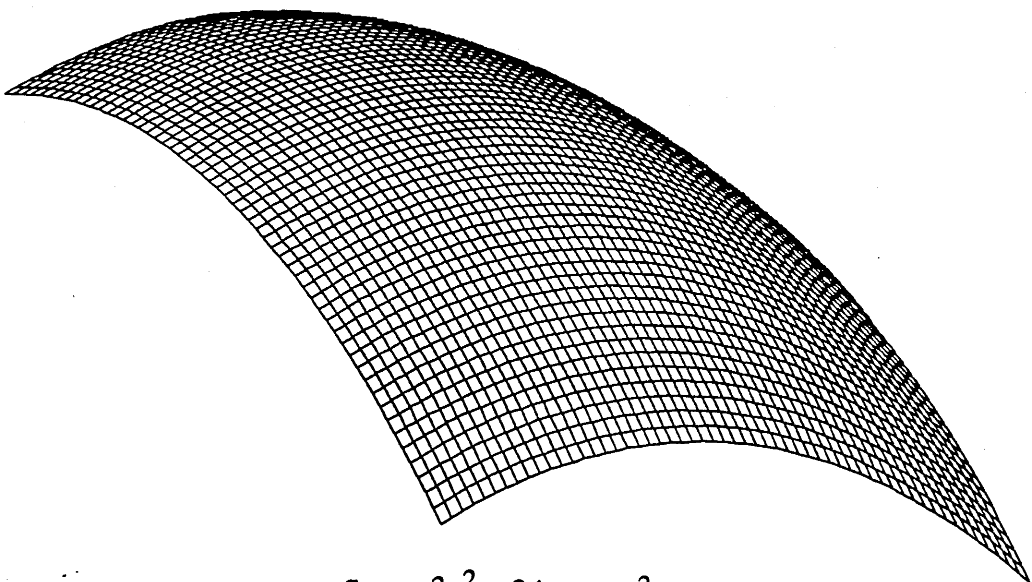


$\sigma_x = -12x^2 + 6xy + 12y^2$



$\sigma_y = 12x^2 + 6xy - 12y^2$

Fig. 4.19a



$\tau_{xy} = -3x^2 + 24xy - 3y^2$

Fig. 4.19b

x = [10 10 11 11] (4.42)

for each element, the stress and displacement errors will rise with the decrease of the y coordinates (Fig. 4.22, Fig. 4.23).

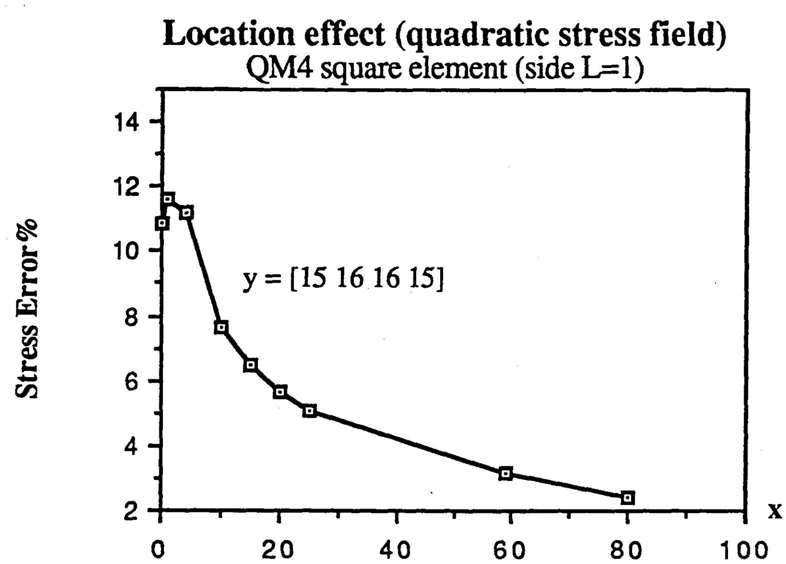


Fig. 4.20

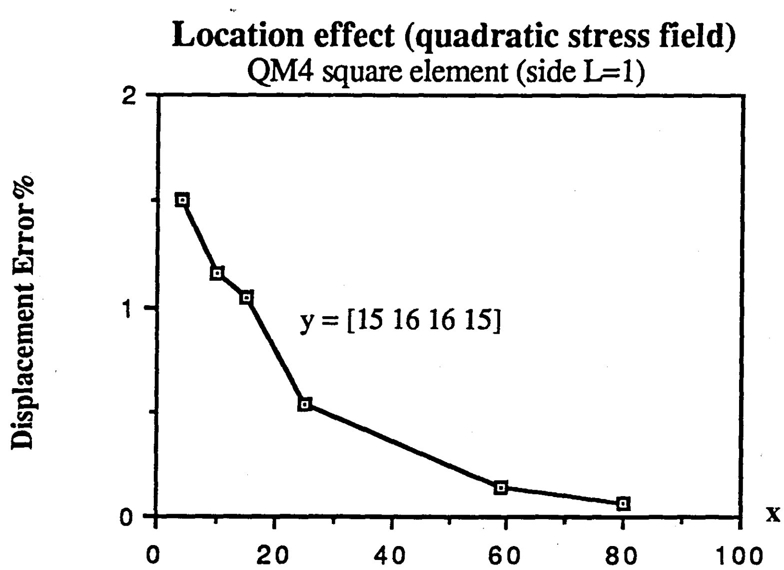


Fig. 4.21

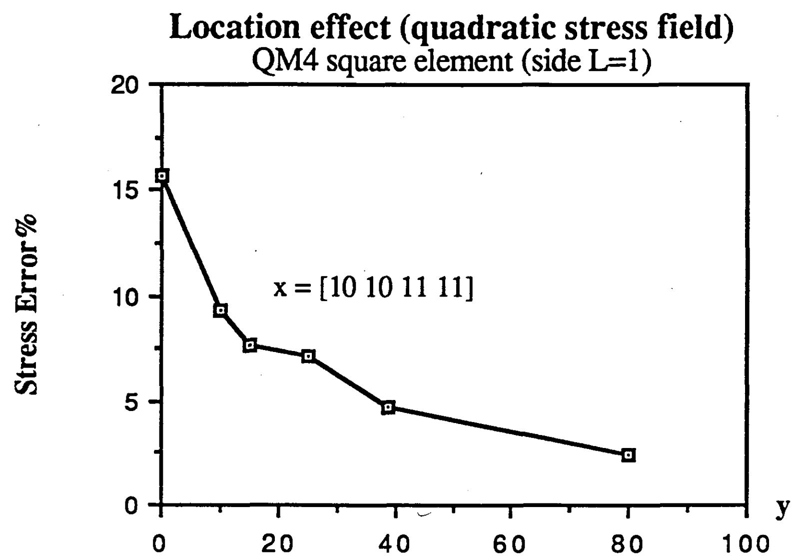


Fig. 4.22

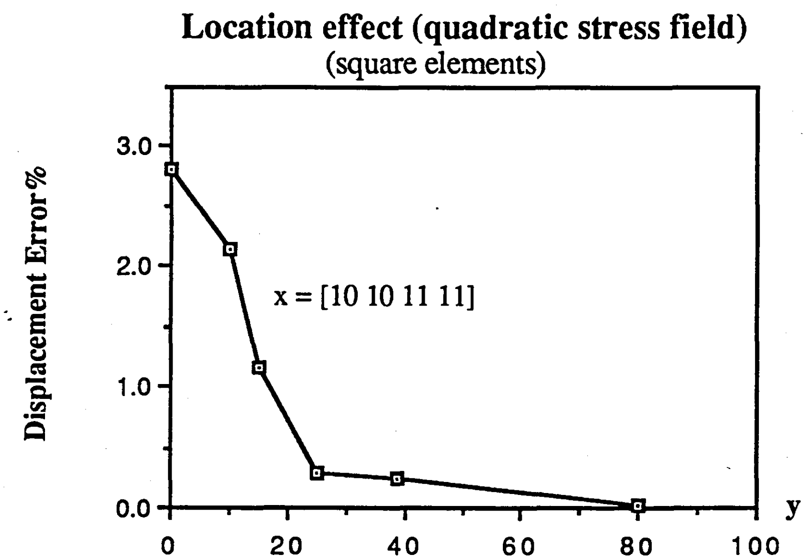


Fig. 4.23

3. Size effect

In the parabolic stress field, the size effect differs from that in the linear stress field. When the element is lengthened from 1, 3, 5 to 8, the stress errors become from 10.87%, 28.15%, 42.15% to 56.28% respectively. All the errors are much larger than those in the linear stress field when the size of element is unchanged

(Fig. 4.24), but the relation between the sizes of elements and stress errors is still nearly linear variation.

(Insert two paragraphs - see the following page)

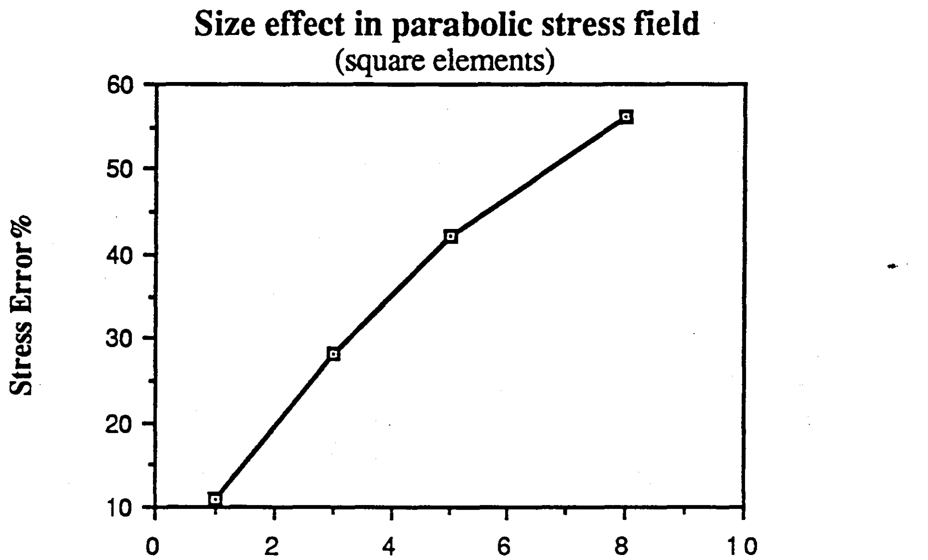


Fig. 4.24

4.7 Comparison with work of Burrows

From the Burrows' reports (Burrows 1988), all the single elements were placed at the centroid of the structure (the location effect was not studied), the element sizes being 2 by 2 (in this research, the element size is 1 by 1).

Since the major tests of this project are done by using 4-noded elements, the results from only 4-noded elements are compared with those from Burrows' in the same order of stress field. The stress fields where the single element tests were carried out by Burrows are:

1. Constant direct stress:

$$\sigma_x = \sigma_0$$

$$\sigma_y = \tau_{xy} = 0$$

2. Constant moment:

$$\sigma_x = \lambda y$$

(after paragraph 1, above Fig. 4.24 on Page 75)

These two points should be clearly noted :

1) although the element in these tests is at a specific location, the relation between the size of element and stress errors is more or less the same with the change of the sample element locations, ie. the same gradient (see Fig. 4.24a below).

2) The result from the location effect (Figs. 4.20-23) show that if the reference element is placed at point (10, 15), the stress error is about the average of those values in a range of x and y from 0 to 80. In this case, all results will be more general and not at two extreme ends (stress errors either too high or too low).

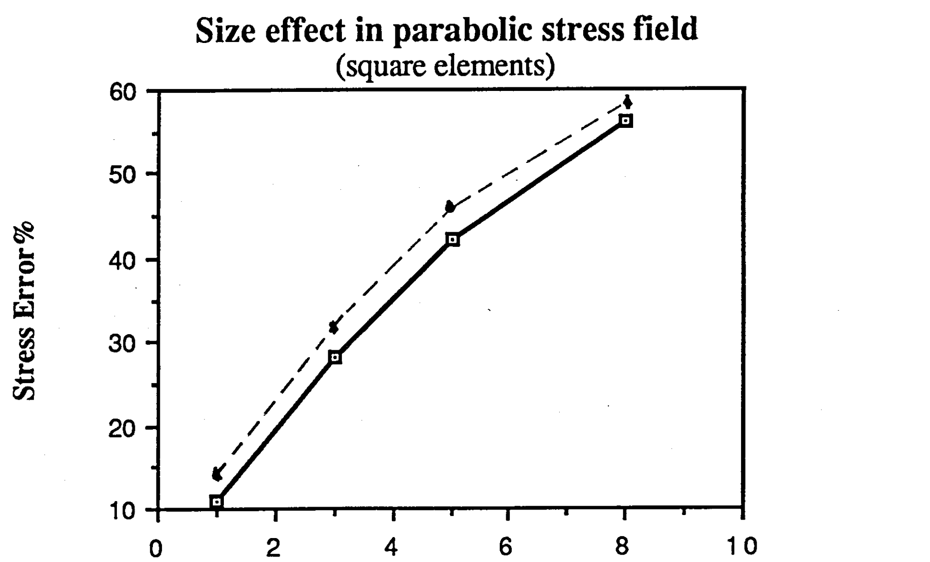


Fig. 4.24 a

$\sigma_y = \tau_{xy} = 0$

3. Linear moment:

$$\sigma_x = \lambda \ (x - l)\ y$$

$$\sigma_y = 0$$

$$\tau_{xy} = -\ 1/2\ \lambda\ y^2$$

where λ and l are constants

From above equations, it is obvious that stress equations in this study are more general. These stress fields reflect better the main characteristics of other stress fields. Because of the location effect, the centroid of all the single elements in this study was located at the same point (10, 15), except those elements used in the location effect tests.

The results of the single element tests show that the location effect varies with the change of the stress field. In some area of a stress field, it affects the errors significantly. Therefore, for any structure, if its stress distribution can be known roughly from a crude solution, different types of elements can be placed in different area to reduce this effect.

The results from Burrows' three stress fields are compared in Fig. 4.18, Fig. 4.25, Fig. 4.26 with the constant, linear and parabolic stress fields respectively.

The comparison shows that the errors from the present study are smaller than those from Burrows' tests. There are three possible resons which are:

1. Location effect: this is the most important source of errors. Because Burrows' solution lacks of consideration of the location effect, the place he sited the elements for the tests resulted in larger errors.

2. Size effect: the elements used by Burrows are 2 by 2, but in present study the size is 1 by 1. However, even when the same size of elements is used, the errors are still smaller than those from Burrows' (shown in Figs. 4.18, 4.25, 4.26).

3. The comparison of different stress fields: although the stress fields Burrows

used are the same order as those in present study, the actual stress functions are different. As we know, the location effect will differ from one stress field to another. Therefore, the difference might lead to various errors.

Constant moment (linear stress field)

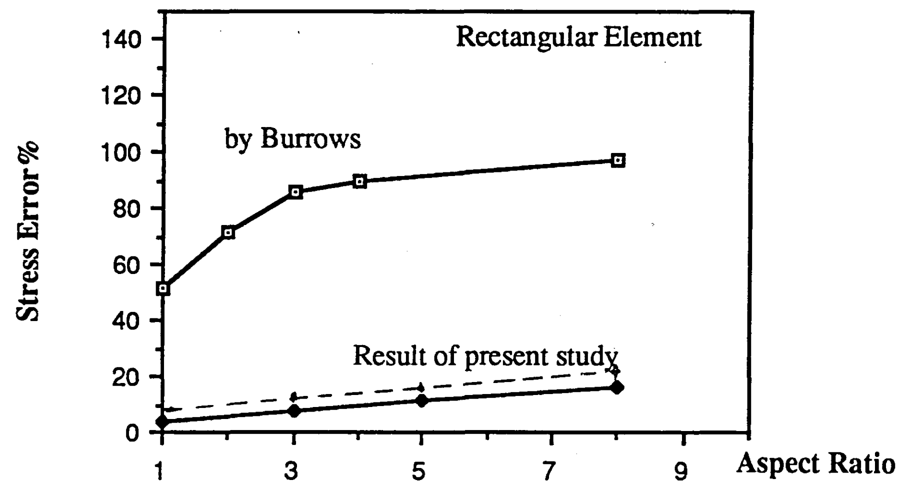


Fig. 4.25

Constant moment (linear stress field)

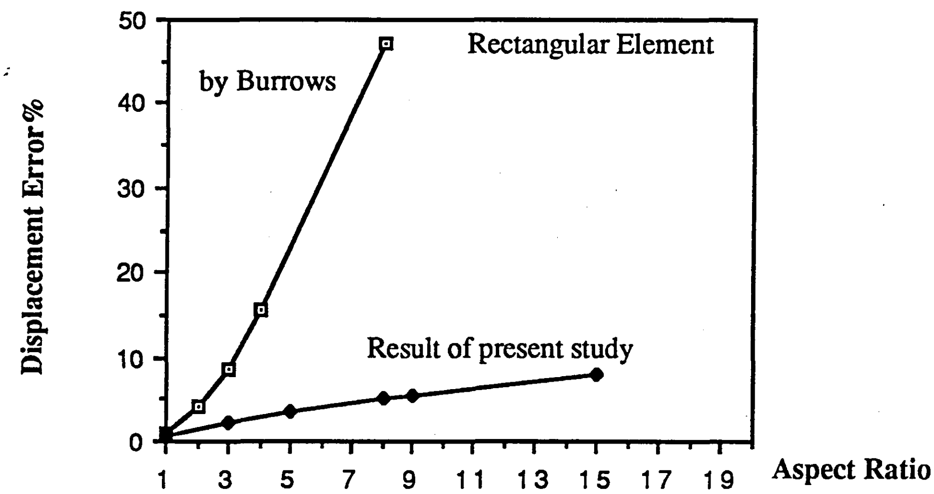


Fig. 4.26

However perhaps the importance lies not in the absolute values but in determining the variation of error with for example size.

Chapter 5 Application of the results of the single element tests

5.1 Introduction

Adaptive approaches and *a posteriori* estimates of the accuracy of computed solutions have recently attracted great interest (Ed. Babuska et al, 1986). An *a posteriori* analysis is sufficient to provide us with detailed knowledge of the errors in a particular finite element approximation. If the initial solution is rejected then an entirely new set of data, representing the new mesh, must be prepared by the analyst. This is a very costly and wasteful procedure without any guarantee that the new mesh is sufficiently accuracte. An *a posteriori* estimate of the accuracy of computed results enables a computer to be given an acceptable level of error and then automatically create a mesh which will achieve that level efficiently. In the present research, greater attention has been paid to find a simple and sufficient error measure for refinement of the structure based on the results of the single element tests.

Single element tests have been devised by Robinson, Burrows etc. (Robinson, 1976; Burrows, 1988; Cheng, 1988). The most important issue is how to apply the results of the tests in practice. As the method is an approximate method it is necessary to have a good idea of the expected solution, together with an understanding of the consequences of the assumptions made within the element types to be used. So, when a finite element analyst designs a mesh the process usually involves a mixture of experience, intuition, and guesswork. If the results of the finite element approximation appear reasonable then these are accepted; if not, then the mesh is redesigned. The drawbacks of this procedure are obvious. Without an *a posteriori* error estimate there is no reliable way of judging the acceptability of the solution.

All the tests of element evaluation are very necessary and helpful in gaining insight to the behaviour of various finite elements. It will take long time to establish a unique error assessment which could be considered as a full investigation. This work tends to concentrate exclusively on the simple and practical way to assess the accuracy of FEA results using membrane type elements in constant, linear and parabolic stress fields. Other high stress fields can be modelled by the combination of these stress fields for which shape parameter, size and location effects have been found from the single element tests.

Following this basic idea, although the stress field may be complex, it can be represented by a constant stress field, a linear stress field and a quadratic stress field once the first derivatives of the stresses are known. Because of the variation of the stresses within a stress field, different errors for stress and displacement will be produced even when the same type of elements are in different locations. Therefore, a procedure for solving the problem is suggested below to gain an accurate answer at a reasonable cost for even a moderately complicated structure;

1) Use a coarse mesh (with, at least 4 elements along any cross-section in the structure for third order or higher polynomial curve fitting) to obtain a crude solution.

2) Curve fit using a 3rd order or higher polynomial along several cross sections for stresses to obtain the stress distribution within the structure.

3) Since the approximate stress distribution within the structure is known, attention can be paid to the dominant stress, and its first derivative.

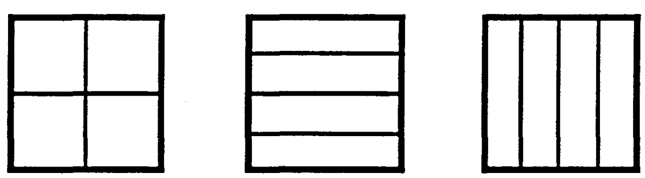
4) The difference of the values of the first derivatives obtained for some points determine what types of elements should be employed in different areas (10% proposal will be explained later). For a high stress area, not only the element type used has to be considered but also the element size (details will be discussed later).

In other words, this step involves a decision on whether to use the constant (4-noded) or the linear (8- or 9-noded) stress fields to model higher order stress fields.

5) Remesh the structure (increasing the element density in areas of high stress concentration, or shrinking the element towards the high stress points and maintaining the same number of elements) to obtain a second solution. The choice of the mesh density is dictated by the stress distribution obtained throughout the structure.

6) Having carried out the finite element analysis it is necessary to assess the results that have been obtained. From the two solutions, the evaluated "real" values of stresses at a high stress area can be calculated.

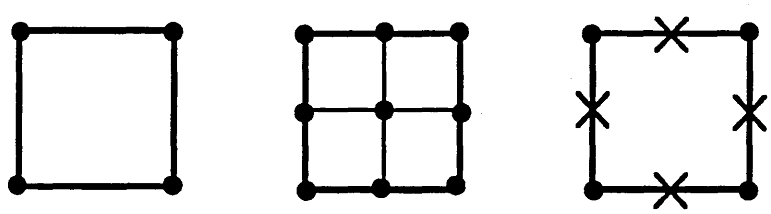
In some structures, it is found that in addition to all the effects (shape, size, location and so on) covered in single element tests, the errors are affected by the different combination of the adjacent elements (called the 'combinational effect' - K_c). In order to estimate the errors accurately, some tests to evaluate this feature are necessary. In this study, only three types of element combination are tested (shown in Fig. 5.1).



(a) (b) (c)

Fig. 5.1

After a crude solution mesh refinement is achieved by either keeping the elements of the same order and subdividing them, or retaining the same mesh and increasing the order of approximation in each element. The first technique is referred to as the h method and the second is referred to as the p method, and it is of course possible to combine both into an h-p procedure. From some tests, it is found that a 9-noded element will produce smaller errors of stresses and displacements than the errors produced by using four elements of one fourth the size of the original 4-noded element (shown in Fig. 5.2). In this research work, the emphasis is placed on the continuum (2D) problems treated by the h method (using the same order (4-node) elements and a constant number of elements, and increasing the element density in high stress areas). Therefore, attention is focused on the use of 4-noded elements in different stress fields and the corresponding errors.



(a) 4-noded (less accurate) (b) 4-noded (accurate) (c) 8-noded (more accurate)

Fig. 5.2

Based on the test results with particular attention to critical regions, an accurate assessment of the "true" stress values can be estimated. The practical application of the tests in structural analysis will be discussed in details later. All the results presented in this work relate mostly to the continuum problems.

5.2 Element combinational effect

To assess the errors correctly for a real problem, apart from the individual element characteristics, their combination is also very important. For the same boundary conditions and loading cases, different combination of elements will produce different errors. On the other hand, the same combination of elements if they are employed with different boundary conditions and loading cases will give rise to different errors.

Primarily, the combinational element test is done for two elements together in the quatratic stress field, and the coordinates of four corner points are (9,15), (9,16), (11,16) and (11,15) respectively (shown in Fig. 5.3). If considering a single element of aspect ratio (AR) 2 (Fig. 5.3 a), the maximum stress and displacement errors are 11.56% & 15.38% respectively. By dividing it into two elements (Fig. 5.3 b), the test of this combination gives maximum errors of stress and displacement of 8.76% & 8.16% as well as 6.58% & 6.57% for two elements respectively.

(a) (b)

Fig. 5.3

In order to understand how the combination of elements affects the errors, and the relationship between the errors from the single element tests and those from the combination of elements, more combined elements should be tested below.

Rectangular elements (no skew and taper) are employed to isolate the procedure of finding element combinational effect K_c. From three combinational tests (Fig. 5.1) in the linear stress field, the results illustrate that the combination of all square elements (the coordinates of four corner points are (9, 14), (9, 16), (11, 14) and (11, 16) respectively: Fig. 5.1 a) is the best one (as shown in Table 5.1). Thus, in the practice of the structural analysis this combination alone is used.

Table 5.1

Combination of element (No. of Figure)	Total stress error (%)
4 elements of AR=1 (Fig. 5.1 a)	4.57
4 elements of AR=4 (Fig. 5.1 b)	6.21
4 elements of AR=1/4 (Fig. 5.1 c)	6.49

5.3 A technique of error assessment in structural analysis

In structural analysis practice, for most structures engineers are more concerned about stresses than displacements. Generally, the stress fields of the structural problems are of lower order than displacement fields. Single element tests in a quadratic stress field show that different locations will produce variable errors. This is due to the distribution of stresses across the structure. If the stress distribution is known from a crude solution, and appeared to be for example, of third order, by calculating the first derivative for the dominant stress and knowing the stress distribution across some critical areas, the position and the size of the constant or linear strain isoparametric elements can be decided. In other words, this high order stress field can be approximated by 4-noded elements in the areas where the first derivertive of the dominant stress is about zero, and by 9-noded elements in the areas where its first derivative remains almost constant. Nevertheless, because the element libraries of a number of FEA programs on microcomputers lack 8 or 9 noded elements, only 4-noded elements are employed. Compared with the linear stress field, the errors in the parabolic stress field are much larger by using 4-noded elements. Usually, with the increase of the stress field order, the errors increase rapidly. Therefore, in a high stress area, the size of elements is reduced such that field representation is approximately linear in order to gain more accurate results.

This can guarantee that the new mesh is sufficiently accurate. The tests show if the difference of the first derivative between two adjacent points is controlled to less than 10%, the total stress error for an element will be less than 5% which is acceptable in engineering.

If the shape sensitivity of an element is ignored (using square elements only), the error magnitude is directly proportional to the element size effect K_s and combinational effect K_c. If the stresses are estimated properly, both effects K_s and K_c should be quantified.

From the single element tests, the size effect has been given in Chapter 4:

$(SE)_{sl} = 0.776 + 3.502L_s$	linear stress field	(5.1)
$(SE)_{sp} = 7.236 + 6.412L_s$	parabolic stress field	(5.2)

where L_s: the ratio of the lenghs of two single elements

$(SE)_s$: the stress error with the change of the element size in the linear and parabolic stress fields.

From the combinational effect tests (4 elements of AR = 1, shown in Fig. 5.1a) in the linear and quatratic stress fields, the relationship of the stress errors (SE) and the length of the elements (L_c) is:

$(SE)_{cl} = 3.77L_c + 0.8$	linear stress field	(5.3)
$(SE)_{cp} = 9.06L_c - 2.66$	parabolic stress field	(5.4)

where $L_c = L_l/L_s$: L_l is the length of the element in the first solution

L_s is the length of the element in the second solution (=1) (as shown in Fig. 5.4).

so, the "true" stress value can be estimated by the equation:

$$(\sigma - \sigma_2)/(\sigma - \sigma_1) = (SE)_2/(SE)_1 \quad (5.5)$$

where σ: the estimated actual stress value

σ_1, σ_2: the maximum stress values from the first and the second solutions

$(SE)_1$, $(SE)_2$: the stress errors from two solutions, they can be calculated by the equations (5.3) and (5.4):

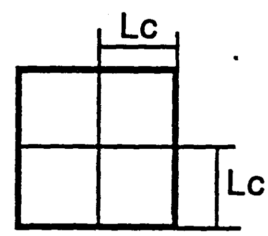


Fig. 5.4

The above two equations (5.3) and (5.4) also incorporate the size effect. They are particularly useful to obtain an accurate solution quickly and cheaply, and will be discussed in following section.

5.4 The use of the technique in practice

(Insert one paragraph - see the following page)

In order to demonstrate the technique, two structural problems with known solutions from elasticity theory are demonstrated. It is assumed that the actual stress values must be evaluated satisfactorily in only two stages. This is due to the consideration of the analysis cost on computational time.

5.4.1 A plate with a central hole

The problem of a plate with a hole is taken once more as an example. The dimensions and loading are exactly the same as those in Chapter 3 and the stress concentration factor and the maximum stress of analytical solution are 2.16 and 0.432 N/mm^2 respectively.

(after the title of section 5.4 on Page 85)

The work presented in this chapter tries to gather the results from the limited range of the single element tests sited in more general yet specific examples of stress function field and by generalising apply them into practical work, which has not been alone by Robinson (1985) and Barrows (1988). The test results from these particular stress fields have a limitation for their applications, so the two examples in the following sections are presented to give only an indication of the approach to be adopted and a basic idea of the aims of this work. As more and more tests are undertaken in more general stress field, the technique can be refined to achieve an accurate solution of FEA results. The attention in the following two examples is devoted to only the critical stress points in the structures, that is, the location effect can be ignored and only other factors are taken into consideration.

The approach adopted is to decrease the size of element such that it is used in a near linear stress field for which single element test data can be applied. If a wider range of single element tests at other higher variation stress locations had been available, a more flexible approach could have been adopted in resizing of the target element.

1) The first solution

Intuitively, there is a stress concentration around the hole; hence a coarse graded mesh is used with 16 elements and a higher mesh density near the hole (the size of the first element is 16 by 16, shown in Fig. 5.5 a). The results give the maximum stress in the y direction (σ_y) as 0.2892 N/mm^2. By third order polynomial curve fitting, the stress distribution along any cross section can be estimated (Fig. 5.6), the stress distribution along the minimum cross section is:

$$\sigma_y = 0.3492 - 0.0085x + 0.000138x^2 - 0.00000838x^3 \qquad (5.6)$$

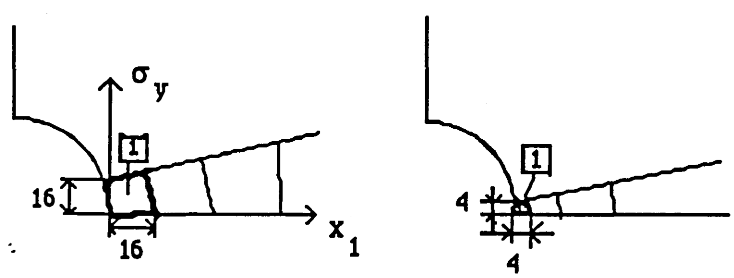
and the first derivative

$$\sigma_y' = -0.0085 + 0.000276x_1 - 0.000002514x_1^2 \qquad (5.7)$$

$x_1 = 0 \qquad \sigma_{y'_0} = -0.0085$

$x_1 = 2 \qquad \sigma_{y'_2} = -0.00796$

error% $= [\sigma_{y'_2} - \sigma_{y'_0}] / \sigma_{y'_2} = 6.78\%$



a) The first solution b) The second solution

Fig. 5.5

2) Analysis for remeshing the structure

The first solution shows that in the area nearest to the edge of the hole the stress σ_y varies sharply (stress concemtration point is at point 1), therefore more elements should be taken in this area. By calculating the differences of the first derivative between each two points, it is known that if the size of element 1 is smaller than 2mm (since the difference of stress gradient between x = 0 and x = 2 is less than 10%), as shown in Fig. 5.7, it can be considered as in a linear stress field, and the results will be more accuracte than the first solution.

Stress distribution from the first solution

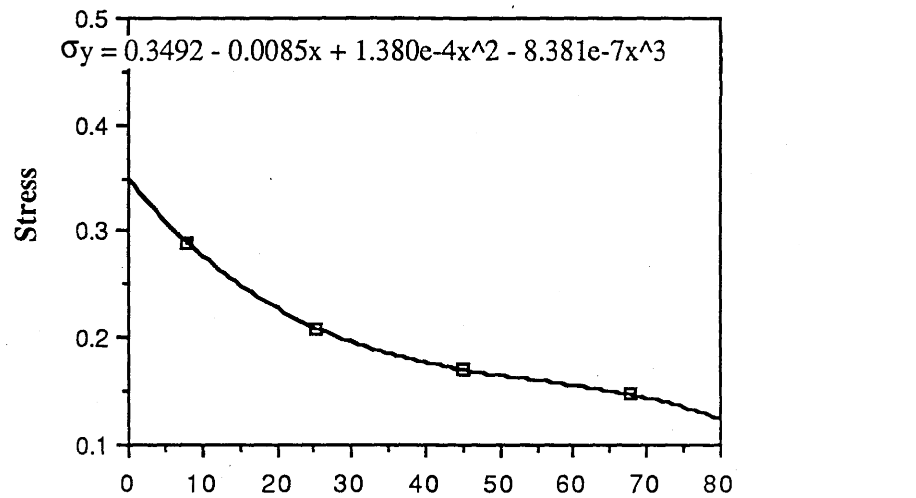


Fig. 5.5

Stress distribution from the first solution

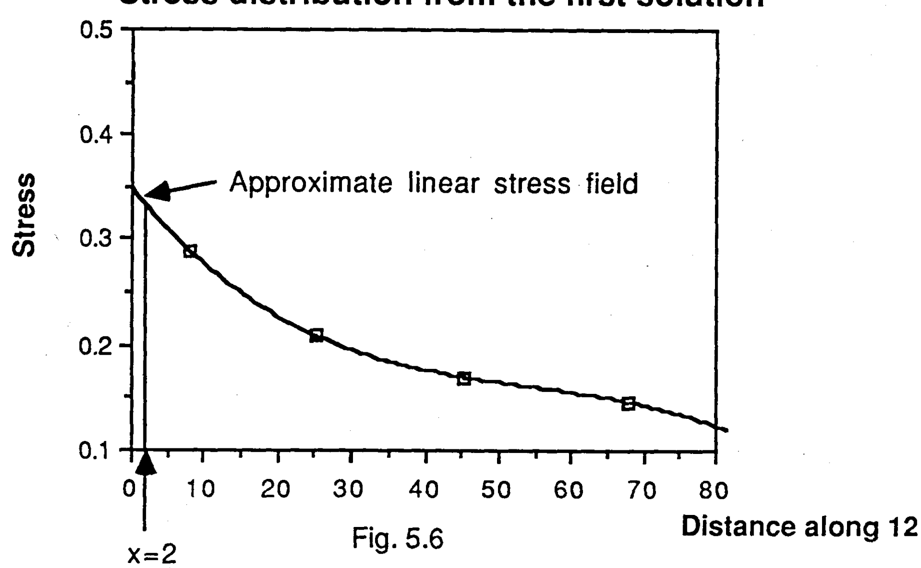


Fig. 5.6

3) The second solution

On the basis of the above analysis, there must be an increase of the element density around the hole. Maintaining the number of elements constant (16), all the elements are condensed around the hole and the size of element 1 is reduced to 1 by 1. In this case the stress output from element 1 is 0.4251 N/mm^2.

4) The estimation of the "true" stress value

Once the two solutions are obtained, the "true" stress value at point 1 can be estimated.

Because the size of element 1 in the second solution is 1 by 1 and in the first solution 16 by 16, from Fig. 5.7 it is clear that when the size of the first element is less than 2 by 2 (especially, the length of the side which is vertical to the dominant stress should be shorter thhan 2), the area crossed by this element can be assumed within the linear stress field and elsewhere in quadratic stress field. The error can be calculated by using the equations (5.3) and (5.4):

$L_c = 1$, $(SE)_{cl} = 4.57\%$

$L_c = 16$, $(SE)_{cp} = 142.3\%$

$\sigma = 0.4296$ (error is 0.56%)

Comparing this value with the target value 0.4296 N/mm^2, the error is 0.56% which in engineering is negligible.

If changing the size of element 1 to 4 by 4 in the first solution, the evaluated value could not vary too much. For instance,

when $L_c = 4$, $(SE)_{cp} = 33.58\%$

so, the element size in the second solution is still 1 by 1,

$[(\sigma-0.4251)/\sigma] / [(\sigma-0.3635)/\sigma] = 4.57/33.58$

hence, σ can be gained: $\sigma = 0.4348$ N/mm^2 (error is 0.648%)

when $L_c = 8$, $(SE)_{cp} = 69.82\%$

the size in 2nd solution remains the same, then

$[(\sigma-0.4251) / (\sigma-0.335)] = 4.57/69.82$

$\sigma = 0.4314$ N/mm^2 (error is 0.14%)

But if the size of element 1 is 2 by 2 in the second solution, the stress output from element 1 is 0.4015 N/mm^2. Thus the stress value equals 0.415 N/mm^2. The error obtained from the comparing this value with the target is 3.94% which is acceptable. However, it can be seen that the second solution is very sensitive to the final estimation. The more suitable the sizes of the elements in the second mesh, the more accurate stress value will be gained. This technique can be used to achieve as good results as required.

5.4.2 An infinite plate with many holes of the same diameter

This is also a stress concentration problem, which is taken from book "Stress concentration factors" (Peterson, 1974). With different ratio of dimensions a, b and c, the stress concentration factor K_{tg} which is equal to 9 is decided (as shown in Fig. 5.8). Young's modulus, Poisson's ratio and the thickness of the plate are assumed as 1 N/mm^2, 0.3 and 1 mm respectively. The prescribed displacement loading is applied on line 56 uniformly in the y direction, and equal to 1 mm.

1) The first solution

For this problem, there must be a stress concentration, but it might occur at point 2 or 3. In order to know where the high stress area is, a coarse mesh is used firstly (shown in Fig. 5.9). The highest stress is 0.00948 N/mm^2 at point 3. After curve fitting the stress distribution along the cross section which is through point 3 and parallel to x axis (shown in Fig. 5.9) is:

$\sigma_y = 0.009423-0.0002547x_1+0.000004533x_1^2-0.0000000373x_1^3$

and the first derivative

$\sigma_y' = -0.0002547 + 0.000009066x_1 - 0.0000001119x_1^2$

At $x_1 = 0$ $\sigma_{y'0} = -0.0002547$

At $x_1 = 2$ $\sigma_{y'_2} = -0.0002369$

At $x_1 = 4$ $\sigma_{y'_4} = -0.0002199$

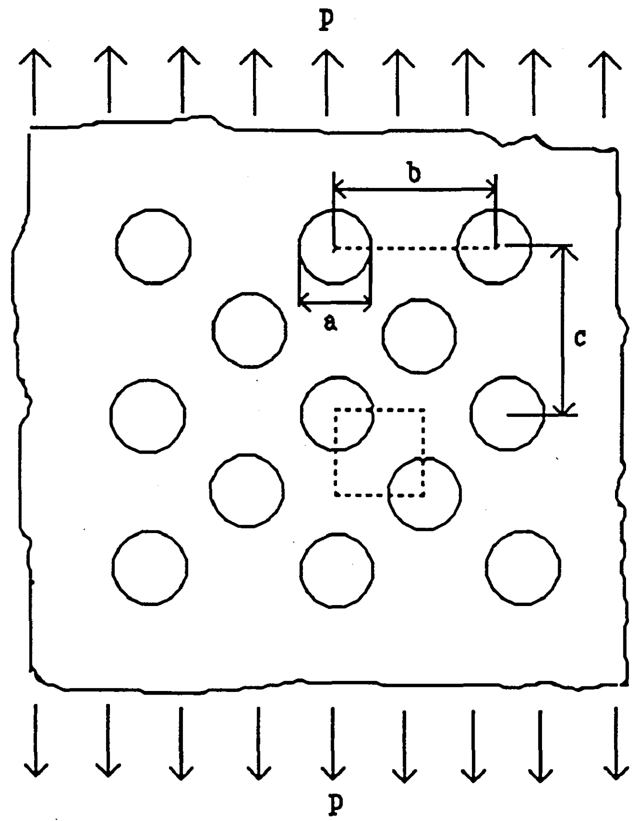


Fig. 5.8 a

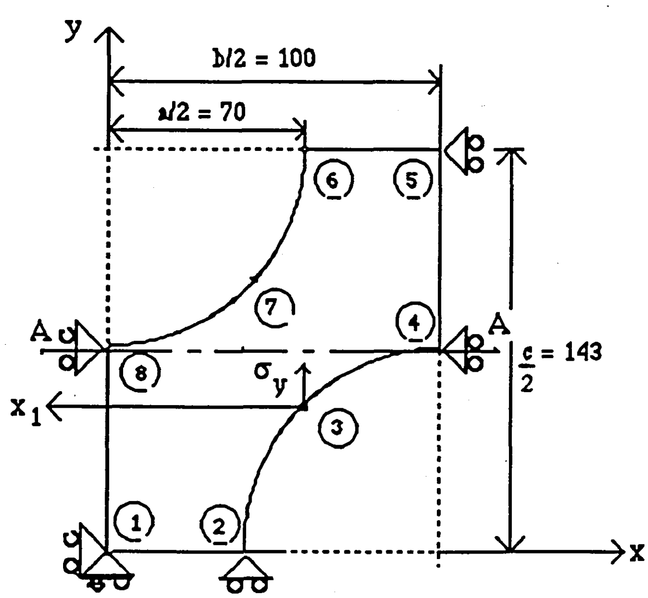


Fig. 5.8 b

The error% = $[\sigma_{y'2} - \sigma_{y'0}]/\sigma_{y'2} = 7.5\%$

The error% = $[\sigma_{y'4} - \sigma_{y'0}]/\sigma_{y'4} = 15.82\%$

Characteristiclly the elements in the first solution around node 3 have length of side 8.

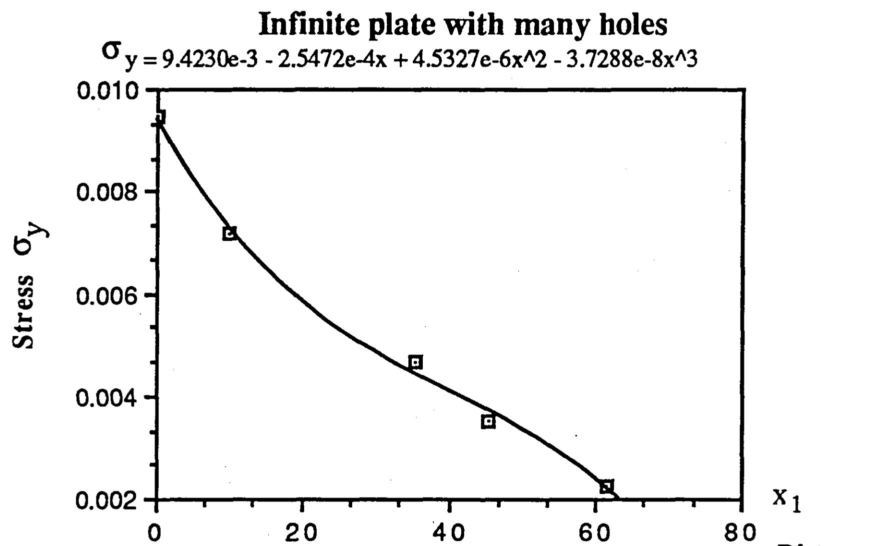


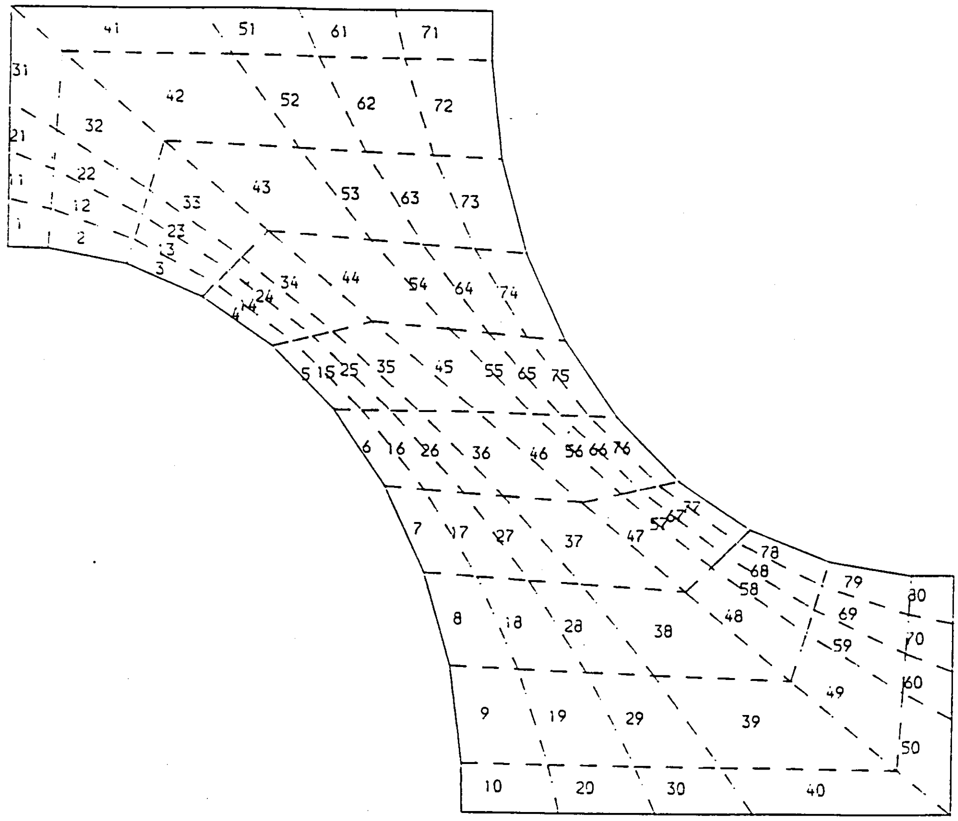
Fig. 5.9

2) Analysis for remeshing the structure

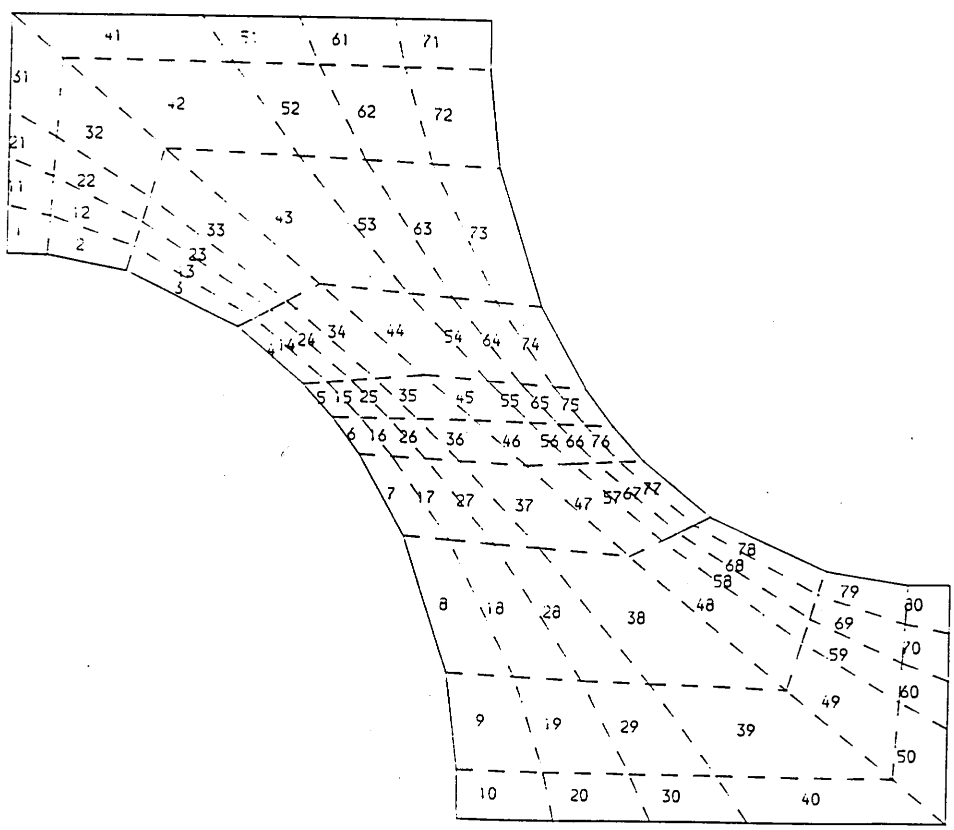
The stress results from the first solution indicate that the stress concentration point occurs at the middle point of the quater segment which is different from the first example. Therefore, the size of elements around this point (point 3) should be reduced. As analysed above, if the size of element which includes point 3 is less than 2, an accurate stress value can be obtained since this small area can be approximated by a linear stress field. Otherwise it will be taken to be a parabolic stress field.

3) The second solution

In order to find the maximum stress at point 3 accurately without increasing the number of elements, the element density near this point should be increased by shrinking all the elements on the hole towards this point. Because of the geometry of the structure, it is very difficult to obtain square elements near point 3, thus



The first solution



The second solution

producing less accurate results. The stress at point 3 is now computed to be 0.00974 N/mm^2.

4) The estimation of the actual stress value

Since two solutions have already been produced, the actual stress value can be evaluated as described in Section 5.3. The average stress on cross section A-A which is calculated from the reaction forces (0.1475 N) and the cross section area (100 mm^2) is 0.001475 N/mm^2. The theoretical maximum stress is about 0.0133 N/mm^2, since the stress concentration factor K_{tg} is 9.

The load is applied in the y direction, so the sides of elements approximately normal to this applied load is more important than the other sides. In the first solution, the length of former sides is about 8, and about 4 in the second solution which can be all considered as in a quadratic stress field. From Equations (5.4) and (5.5):

$L_c = 8$, $(SE)_{cp1} = 69.82\%$

$L_c = 4$, $(SE)_{cp2} = 33.58\%$

so, from the equation (5.5) the estimated stress is:

$\sigma_{y1} = 0.01$ (error% = 24.8%)

This value is very inaccurate comparied with the theoretical value 0.0133 N/mm^2 because of the effects of aspect ratio and skewed shape. However those two factors can be taken into account to determine the actual stress.

a) effect of AR

In a parabolic stress field, if AR = 3 - 3.5, then SE = 14 - 16% (see Fig. 4.18), so the stress σ_{y1} from above is improved to

$\sigma_{y2} = \sigma_{y1}/(1\text{-}SE) = 0.0116 - 0.0119$ N/mm^2

The error now is 12.78% - 10.53%.

b) shape effect (skew)

From the single element tests, it is known that in a parabolic stress field, when a rectangle is skewed 45^0, the stress error is 10% (see Fig. 4 10). And so the improved stress σ_{y2} from a) above is further adjusted to σ_{y3};

$\sigma_{y3} = \sigma_{y2}/ (1\text{-SE}) = 0.129 - 0.132$

The error% = 3 - 0.75%

After the consideration of the two effects, the difference between the estimated stress and the theoretical stress, as shown above, is acceptably small.

This example is resulted by using PAFEC program on the mainframe, because there are some limitation on PC-based packages. For instance, the applied diplacement load for the cell of the infinite plate is that the edge 56 moves 1mm in the y direction uniformly which, however, can not be done on PC-based package.

These two examples show roughly how the technique of accuracy assessment works. If more tests could have been done, different factors could be looked at in more detail for many types of meshes.

Chapter 6 Conclusions and Recommendations

6.1 General observations and conclusions

The main observations and conclusions obtained from this thesis can be summarised as follows:

So far, the single element and combinational element tests done are mainly based upon using 4-noded membrane elements, and the application of error assessment is similarly restricted.

The effects of finite element shape parameters, size, location and combination in linear static structural analysis are complex and significantly important in the prediction of the accuracy of the FEA solution. These factors must be defined clearly in any analysis so that a solution can be judged in relation to them, otherwise there might be an element of doubt in the adequacy of the results. Misuse of these parameters can lead to different solution accuracy. Variations can be obtained even if a single parameter is changed. In particular the following factors were found to affect the results of an analysis:

(a) Shape parameters: aspect ratio, skew angle, taper and orientation. Of these, skew angle is the most significant parameter affecting the accuracy of FEA results. But it should be noted that with the change of element type (i. e. 4- or 9-noded elements) those effects will be slightly different.

(b) Size effect is a very important factor for efficient discretization of a structure. In a variety of stress fields, the different errors will be obtained by varying the size of elements. From the single element tests the relation between the size of element and the stress error is an almost linear variation in the linear and quadratic stress fields.

(c) Location effect: Although the shape parameters, the size and type of elements are maintained constant, as the element location varies, the errors produced vary. This effect is directly related to the distribution of the stresses across the structure. In the areas where the stresses varv within a small range, the effect of the location is insignificant.* (Insert one paragraph - see the following page) Whereas, in areas of stress concentration care should be taken to choose a suitable type of elements in particular locations.

(d) Combinational effect: Apart from the effects established in the single element tests, an actual structural problem usually has many elements, and errors will be different of those from the single element tests due to so called "combinational effect". The magnitude of errors from a variety of element combinations depend on stress fields. The tests of combinational effect show that the combination of all square elements is most effective for any stress field. Other combinations often are limited in certain stress fields.

The assessment of the accuracy of the FEA solution can be carried out based on the above tests. Due to limited time, the established error measure only concerns the size effect and combinational effect for a particular area and an unchanged element shape. A simple method was proposed to evaluate the actual stress values. Certain approximations were required in order to treat the high order stress field linear or parabolic. Attempts have been made to formalise the information from the element tests. The following equation allows the estimation of the stress values:

$$(\sigma - \sigma_2)/(\sigma - \sigma_1) = (SE)_2/(SE)_1$$

where σ: the estimated actual stress value

σ_1, σ_2: the stress values from the first and the second solution

$(SE)_1$, $(SE)_2$: the stress errors from two solutions, they can be calculated by the following equations:

$$(SE)_{cl} = 3.77\, L_c + 0.8 \qquad \text{linear stress field}$$

(add in (c) Location effect, line 5, following 'insignificant' on Page 95)

All the single elements in the tests are located at the same point (10, 15) with exception of the location effect tests. The reason is that at this location, the stresses vary not so rapidly, ie. a small location effect, so other effects can be tested out with little interference of the location effect.

$$(SE)_{cp} = 9.06\ L_c - 2.66 \qquad \text{Parabolic stress field}$$

The first solution only gives a rough stress distribution of the structure. Based on this solution, the structure is remeshed to obtain a more accurate result by increasing the element density in the high stress area for the same number of elements, since the actual stress value estimated is associated closely with the accuracy of the second solution. An important issue considered at this stage is the size of elements in the higher stress area which can be decided by determining the first derivative of the dominant stress. The difference of the first derivatives across the first element should be maintained within 10%. The example showed that the estimated stress value compares well with the true value. The range of errors is acceptable.

6.2 Recommendations for future work

Further study is required to refine the technique for error assessment. In addition to the size effect and combinational effect, the shape parameters and location effect should be studied. This will facilitate the application of accuracy assessment techniques to more general problems.

(Insert one paragraph - see the following page)

Another set of single element tests should be devised for plate elements to enable the accuracy assessment for out-off plane bending problems. It would be useful to confirm the generalization of the assessment, and would consolidate the conclusions reached here. Accuracy assessment is essential in providing the user with confidence in the results of his FEA analysis.

More combinational effect tests should be done to predict the most suitable combination of elements in a range of stress fields. It was found from some structural analysis that if σ_x is much larger than σ_y, elements enlongated along the x direction will give better results than elements elongated along the y direction.

(add in section 6.2, after the first paragraph on Page 96)

In views of the above tests done in there specific stress fields and the reference element taken from a special point, more comprehensive tests should be done in over the full range of stress fields and at a set of reference elements placed in different locations giving a representative set of results of element response to differing severity of stress variations.

Further study is required on this feature.

References

ANSYS "User's manual", Swanson Analysis Systems, Inc., 1986

Argyuis, J. H. & Kelsey, G. "Energy theorems and structural analysis", Butterworth, 1960

Babuska, I.; Zienkiewicz, O. C.; Gago, J.; Oliveira, E. R. de A. (Editors) "Accuracy estimates and adaptive refinements in finite element computation", John Wiley & Sons, Chichester, 1986

Barlow, J. "Distortion Effects in Isoparametric Elements - An Analytical Approach", NAFEMS AGM/Technical Session, U. K., 1987

Benham, P. P. & Warnock, F. V. "Mechanics of solids and structures", Pitman Publishing, London, 1973

Botkin, M. E. "Shape optimization of plate and shell structures", AIAAJ., 20, No. 2, 268-273, 1982

Brown, D. K. "An introduction to the finite element method using BASIC programs", Chapman and Hall, New York, 1984

Brown, S. A. & Kamat, M. P. "Structural analysis of large scale problems using a microcomputer", Eng. Comput., vol. 2, 155-158, 1985

Burrows, D. J. "A finite element shape sensitivity study" Proceedings of "Reliability of methods for engineering analysis", Pineridge Press, July 1986

Burrows, D. J. "Shape Measuring Criteria and the Establishment of Benchmark Tests for Single Membrane Elements", NAFEMS, 1988

Bushnell, D. "Finite difference energy models versus finite element models: Two variational approaches in one computer program", ONR Sympos., 291-336

Carnet, J.; Ladereze, P. & Leguillon, D. "An optimal mesh procedure in finite element method"

Cheng, J. H. "Automatic adaptive remeshing for finite element simulation of forming processes", Int. J. numer. methods eng., vol. 26, 1-18, 1988

Clough, R. W. "The finite element method in plane stress analysis", Proc. 2nd

Conf. on Electronic Computation, ASCE, New York, 345-377, 1960

Cook, R. D. "Concepts and applications of finite element analysis", John Wiley & Sons, London, 1974

Courant, R. "What is mathematics? an elementary approach to ideas and methods", Oxford U.P., London, c1941

Desai, C. S. & Abel, J. F. "Introduction to the finite element method", Van Nostrand, 1972

Forsberg, K. J. "General summary of the conference", CompSh Conf., 1250-1284

Gallagher, R. H. "Finite element analysis fundamentals", Prentice Hall, Mew Jersey, 1975

GIFTS manuls, CASC/GIFTS Inc., 1979

Hinton, E. & Owen, D. R. J. (Ed.) "Finite element programing", Acdamic Press, New York, 1977

Hinton, E. & Owen, D. R. J. "Finite element software for plate and shells", Pineridge Press, Swansea, 1984

Hoog, F. R. de "Error, approximation and accuracy", 1973

Irons, B. M. " Numerical integration applied to finite element methods", Conf. use of digital computers in Struct. Eng., Newcastle University, 1966a

Irons, B. M. "Engineering application of numerical integration in stiffness method", J. A. I. A. A., Vol. 14, 1966b

Iron, B. M. & Ahmed, S. "Technique of finite elements", John Wiley & Sons, 1981

Mackerle, J. "Finite element codes for microcomputers - a review", Computers and structures 24, 657-682, 1986

Mackerle, J. "Finite element codes for microcomputers, an addendum", Computers and structures, April 1988

Mair,W. M. "The microcomputers and finite element analysis", U. K., NAFEMS AGM/TEchMCAL Session,1988

Mohr, G,A. & Milner, H. R. "A microcomputer introduction to the finite element method", 1986

MSC/PAL2 manuals, Macneal - Schwendler Corporation, 1987

NAFEMS "Selected FE Benchmarks in structural and thermal analysis", NAFEMS, U. K. 1987

NAFEMS "A finite element primer", NAFEMS, U. K., 1986

Norrie, D. H. & de Vries, G. "The finite element method - Fundamentals and applications", Academic Press, London, 1973

Peterson, R. E. "Stress concentration factors", John Wiley & Sons, New York, 1974

Ramsay, A. C. "Managing advanced computational engineering", 42-44, CME April 1988

Robinson, J. and Haggenmacher, G. W. "Evaluation of Calac semi-monocoque membrane element", Lockheed-California Company Report 24055, 1970

Robinson, J. "Integrated theory of finite element method", John Wiley & Sons, U. K., 1973

Robinson, J. "A single element test", Comp. Methods Appl. Mech. Eng. 7, 191-200, 1976

Robinson, J. "Basic shape sensitivity tests for membrane and plate bending finite elements", Robinson and Associates, January 1985

Robinson, J. "Distortion measures for quadrilaterals with curved boundaries", Robinson and Associates, May 1987

Rockey, K. C.; Evans, H. R.; Griffiths, D. W. & Nethercot, D. A. "The finite element method a basic introduction", Crosby Lockwood Staples, London, 1975

Schrefler, B. A.; Lewis, R. W. & Odorizzi, S. A. (Editors) "Engineering software for microcomputers", Pineridge Press, Swansea, 1984

Taig, I. C. "Structural analysis by the matrix displacement method", Engl. Electric Aviation report N. So17, 1961

Thomson, R. D. "Choosing a finite element package", NAFEMS, 1988

Timoshenko, S. P. & Woinowsky-Krceger, S. "Theory of plates and shells", Magraw-Hill, New York, 1959

Timoshenko, S. P. & Goodier, J. N. "Theory of Elasticity" (Third edition), Kogakusha, Tokyo, 1970

Turner, M. J., Clough, R. W., Martin, H. C. & Topp, L. J. "Stiffness and deflection analysis of complex structures", Journal of the aeronautical sciences, vol. 25, 805-823, 1956

Zienkiewicz, O. C. "The finite element method: from intuition to generality", APRev, vol. 23, No. 3, 249-256, 1970

Zienkiewicz, O. C. "The finite element method in engineering science", McGraw - Hill, London, 1971

Zienkiewicz, O. C. "Finite Element Method" (3rd Ed.), New York, 1977

Zienkiewicz, O. C. and Zhu, J. Z. "A simple error estimator and adaptive procedure for practical engineering analysis", Int. J. numer. methods eng., vol. 24, 337-357, 1987