Windows Programming of Stroh’s Finite Element for Two-Dimensional Anisotropic Elastic Solids

Abstract

Due to the anisotropy nature of composite materials, the mechanical behavior of composite structures is usually studied by using anisotropic elasticity. There are two main approaches dealing with the two-dimensional linear anisotropic elasticity, i.e., Lekhnitskii formulation and Stroh formalism. Both of these two formalisms are formulated by complex variable functions. Hence, it is not very popular in the engineering society. On the other hand, finite element method is an important and popular tool for mechanical analyses. They are usually formulated by variational principle. During the formulation, none of the results concluded by the complex variable formulation has been employed. The main purpose of this project is therefore to build a bridge connecting these two main formulations and see what advantages can be got from their marriage. The final form of the finite element formulation usually consists of a body integral and a surface integral. The former corresponds to the governing equation, while the latter is related to the boundary conditions. The body integral is responsible for the body-mesh that is the main source of large computing memory. The governing equation of anisotropic elasticity contains the constitutive laws, the kinematic equations and the equilibrium conditions. A general solution satisfying the governing equation has been well formulated by Stroh formalism. Therefore, by embedding this general solution into the finite element formulation, the body integral will vanish. The expected result is the saving of computer storage and memory. Moreover, the element becomes more accurate and efficient due to the satisfaction of governing equations. The above concept was formulated mathematically, then programmed by Fortran language. Finally, in order to have a friendly and easily-used program, the associated software for Windows was also designed through the use of Visual Basic.

Keywords: Anisotropic Elasticity, Stroh formalism, Finite Element Method, Composite Structures

1. Introduction

In the literature, there are two main approaches dealing with the two-dimensional linear anisotropic elasticity. One is Lekhnitskii formalism [1] which begins with the stresses, the other is Stroh formalism [2,3] which starts with the displacements. Both of these two formalisms are formulated by complex variable functions. Although they are well-known in the mechanics community, they are not very popular in the engineering society. On the other hand, finite element method is an important and popular tool for mechanical analyses. The numerous approximate procedures discussed in the literature generally fall into three categories: the direct method, the method of weighted residuals and the variational method [4]. Among them the displacement-based variational formulation is the most popular one. To the authors’ knowledge, none of the results concluded by the complex variable formulation has ever been employed in the displacement-based finite element formulation. In this project, we try to build a bridge connecting these two main formulations.

It is known that the equilibrium is usually not satisfied within elements and between elements for a displacement-based finite element. However, for a displacement based complex variable formulation - Stroh formalism, a general solution satisfying the strain-displacement relation, the
stress-strain laws and the equilibrium equations has been obtained explicitly [3]. Hence, it is expected to get some merits by embedding the general solution of Stroh formalism into the finite element formulation, which will be called "Stroh finite element" in this report.

In Stroh formalism, the general solution contains three arbitrary complex functions which are the basis of the whole field stresses and deformations. Like the shape functions used in the finite element method, the arbitrary functions can be chosen to be polynomials. Although they are complex variable functions, through the use of identities developed in the literature, for example [5], the entire formulation can be transformed into a real form expression like the usual finite element. Based upon this concept, the explicit expressions of the stiffness matrices for two basic elements: linear and quadratic elements, are derived in this project. A finite element computer windows program is then coded by using the FORTRAN and Visual Basic languages. Due to the use of the Stroh formalism, the compatibility and the equilibrium conditions within each element are all satisfied. The accuracy and versatility of the elements are then shown through several numerical examples.

2. Problem Statements and Solutions
2.1 Anisotropic Elasticity

In a fixed rectangular coordinate system \(x, t = 1,2,3\), let \(u, u, \varepsilon\) be, respectively, the displacements, stresses and strains. A general solution satisfying the strain-displacement equations, the stress-strain laws, and the equilibrium equations for two-dimensional linear anisotropic elasticity has been obtained as [3]

\[
\mathbf{u} = 2 \text{Re} \{ \mathbf{A} f(z) \}, \quad \mathbf{\phi} = 2 \text{Re} \{ \mathbf{B} f(z) \},
\]

where \(\mathbf{u} = (u_1, u_2, u_3)\) is the displacement vector and \(\mathbf{\phi} = (\phi_1, \phi_2, \phi_3)\) is the stress function vector which is related to the stresses by

\[
\sigma_{ij} = -\phi_{ij}, \quad \sigma_{ik} = \phi_{ij}.
\]

\(f(z) = [f_1(z), f_2(z), f_3(z)]\) is a function vector composed of three holomorphic complex function \(f_\alpha(z_\alpha)\), \(\alpha = 1, 2, 3\), which will be determined by satisfying the boundary conditions of the problems. The argument \(z_\alpha\) of each component function \(f_\alpha(z_\alpha)\) is written as \(z_\alpha = x + p_\alpha y\) in which \(p_\alpha\) is the material eigenvalue whose imaginary part has been arranged to be positive. \(\mathbf{A} = [a_1, a_2, a_3]\) and \(\mathbf{B} = [b_1, b_2, b_3]\) are \(3 \times 3\) complex matrices of which \((a_\alpha, b_\alpha)\), \(\alpha = 1, 2, 3\), are the material eigenvectors associated with \(p_\alpha\) and can be determined by the following eigen-relation:

\[
\mathbf{N} \xi = p \xi,
\]

where

\[
\mathbf{N} = \begin{bmatrix} N_1 & N_2 \\ N_3 & N_4 \end{bmatrix}, \quad \xi = \begin{bmatrix} a \\ b \end{bmatrix},
\]

and

\[
N_1 = -T^{-1}R^T, \quad N_2 = T^{-1} = N_7^T, \quad N_3 = RT^{-1}R^T - Q
\]

In (3c) \(C_{ijkl}\) is the elasticity constants, the superscript \(T\) stands for the transpose, and the superscript \(-1\) means inverse.

If \(t\) is the surface traction at a point on a curve boundary, then

\[
t = \partial \phi / \partial s (4)
\]

where \(s\) is the arc length measured along the curved boundary.

One of the special features of the Stroh formalism is the identities which transform the complex functions into real form expressions. With these identities, the mathematical manipulation becomes easier and the real form solution becomes possible. An identity which plays an important role in our later formulation is now listed below [6].

\[
\begin{array}{c}
\mathbf{A} \left< \begin{array}{c} z^d \\ \phi \end{array} \right> \mathbf{B}^T \quad \mathbf{A} \left< \begin{array}{c} z^d \\ \phi \end{array} \right> \mathbf{A}^T = \frac{1}{2} \mathbf{N}^T (I - i \mathbf{N}) \\
\mathbf{B} \left< \begin{array}{c} z^d \\ \phi \end{array} \right> \mathbf{B}^T \quad \mathbf{B} \left< \begin{array}{c} z^d \\ \phi \end{array} \right> \mathbf{A}^T = \frac{1}{2} \mathbf{N}^T (I - i \mathbf{N})
\end{array}
\]

where

\[
\mathbf{N} = \frac{1}{\pi} \begin{bmatrix} \mathbf{S} & \mathbf{H} \\ -\mathbf{L} & \mathbf{S}^T \end{bmatrix}
\]

and

\[
\mathbf{N} = x_1 \mathbf{I} + x_2 \mathbf{N}.
\]

In the above, the angular bracket \(< f_a >>\) denotes the diagonal matrix with the diagonal components varied according to the subscript \(a\). The real matrix function \(\mathbf{N}(\omega)\) is related to \(\mathbf{N}\) by \(\mathbf{N}(0) = \mathbf{N}\). The general definition of \(\mathbf{N}(\omega)\) is the same as that shown in (3) except that all the submatrices \(N_\alpha(\omega)\) is calculated based upon the real matrices \(\mathbf{Q}(\omega), \mathbf{R}(\omega)\) and \(\mathbf{T}(\omega)\) which are defined as

\[
Q_\alpha(\omega) = C_{\alpha\beta} n_\beta n_\alpha, \quad R_\alpha(\omega) = C_{\alpha\beta} n_\beta m_\alpha,
\]

\[
T_\alpha(\omega) = C_{\alpha\beta} m_\beta m_\alpha,
\]

where
2.2 Linear Element

By choosing the unknown function \( f(z) \) in each element to be a linear function as

\[
f(z) = c_0 + << z_0 >> c_1,
\]

and substituting (7) into the general solution (1), and expressing each of the complex coefficients \( c_k \), \( k = 1, 2 \) in terms of two real column vectors \( g_k \) and \( h_k \) as \( c_k = A^T g_k + B^T h_k \), the displacements and stress functions for each element may be expressed as

\[
\begin{bmatrix}
\mathbf{u} \\
\phi
\end{bmatrix} = 
\begin{bmatrix}
\mathbf{h}_0 \\
\mathbf{g}_0
\end{bmatrix} + N^2 \begin{bmatrix}
\mathbf{h}_1 \\
\mathbf{g}_1
\end{bmatrix}.
\tag{8}
\]

In order to satisfy the continuity between each element, the displacement and stress function are better represented in terms of the nodal displacements. To this end, the transformation between \( (\mathbf{h}_0, \mathbf{h}_1, \mathbf{g}_0) \) and the nodal displacements should be found first. For three unknown column vectors per element, we choose the triangular element as our linear element (Figure 1)

![Linear triangular element](image)

With the stress function expressed by the nodal displacement, the traction along the line connecting two vertices can also be expressed in terms of the nodal displacements. Since the traction is uniformly distributed along the straight boundary for the present linear element, the resultant forces along this line contributing to each node can be considered to be equal. Therefore, the vector of the nodal forces \( \mathbf{t}_n \) may finally be written in terms of the nodal displacements \( \mathbf{u}_n \) as

\[
\mathbf{t}_n = \mathbf{Ku}_n,
\tag{9}
\]

where \( \mathbf{K} \) is the element stiffness for the Stroh linear element and can be written explicitly as

\[
\mathbf{K} = \begin{bmatrix}
K_{11} & K_{12} & K_{13} \\
K_{21} & K_{22} & K_{23} \\
K_{31} & K_{32} & K_{33}
\end{bmatrix},
\tag{10a}
\]

where

\[
n(\omega) = (\cos \omega \sin \omega 0),
m(\omega) = (-\sin \omega \cos \omega 0)
\tag{6b}
\]

\[
K_d = \frac{1}{2E_0} (\Delta \mathbf{N}_d^T \mathbf{N}_d + E_d \Delta \mathbf{N}_d^T \mathbf{N}_d),
\tag{10b}
\]

and

\[
\Delta \mathbf{N}_d^{(i)} = (\Delta x_1)^{(i)} \mathbf{I} + (\Delta x_2)^{(i)} \mathbf{N}_1, \quad \Delta \mathbf{N}_d^{(i)} = (\Delta x_3)^{(i)} \mathbf{N}_3,
\]

\[
(\Delta x_1)^{(i)} = x^{(2)}_i - x^{(3)}_i, \quad (\Delta x_2)^{(i)} = x^{(3)}_i - x^{(1)}_i,
\]

\[
(\Delta x_3)^{(i)} = x^{(1)}_i - x^{(2)}_i, \quad i = 1, 2.
\tag{10c}
\]

The internal stresses at any point within the element can now be calculated by using the relation between the stress function and the stress. The explicit expression of the stresses in terms of the nodal displacements can be found in [7].

From the above derivation, we see that the displacements \( \mathbf{u} \) within the element, the traction \( \mathbf{t} \) along the boundary, and the internal stresses can all be expressed in terms of the nodal displacement. The element stiffness matrix which is the core matrix in the finite element formulation is also obtained explicitly in (10). It can be seen that all these formulae are written explicitly and do not contain any integrals. This feature may save us a lot of computational effort. Moreover, the present element is valid for any kind of anisotropic materials and considers not only in-plane deformations and stresses but also the anti-plane deformations and stresses.

2.3 Quadratic Element

We now consider the second order approximation of \( f(z) \), i.e.,

\[
f(z) = c_0 + << z_0 >> c_1 + << z_0^2 >> c_2.
\tag{11}
\]

By the way similar to the linear element, a quadratic element can be developed for the second order approximation of each element. The displacements and stress functions for each element may be expressed as

\[
\begin{bmatrix}
\mathbf{u} \\
\phi
\end{bmatrix} = 
\begin{bmatrix}
\mathbf{h}_0 \\
\mathbf{g}_0
\end{bmatrix} + N^2 \begin{bmatrix}
\mathbf{h}_1 \\
\mathbf{g}_1
\end{bmatrix} + N^3 \begin{bmatrix}
\mathbf{h}_2 \\
\mathbf{g}_2
\end{bmatrix}.
\tag{12}
\]

Similar to the linear element, the next step is trying to transform the expression in terms of the nodal displacements. Since we have five sets of unknown coefficients \( (\mathbf{h}_0, \mathbf{h}_1, \mathbf{g}_0, \mathbf{h}_1, \mathbf{g}_1) \), the most appropriate element should be quadrilateral (Figure 2). To avoid using any internal node, we choose

\[
\mathbf{h}_2 = \mathbf{g}_2 / E_0,
\tag{13}
\]

in which \( 1/E_0 \) is multiplied in order to avoid adding the terms with different units. With this selection, we now have only four sets of unknown coefficients corresponding to four nodes, and hence the transformation matrix between these four unknown coefficients and the nodal displacements of the quadrilateral element can be
obtained by a way similar to the linear element.

Figure 2: Quadratic quadrilateral element

The traction along the boundary can also be expressed in terms of the nodal displacement through the use of stress function given in (12), and the relation given in (4). Knowing that the traction is in a linear distribution along the boundaries of the element for a second order approximation, we may distribute the nodal forces by 1/3 and 2/3 of the total force along each boundary to the nodes connecting this straight boundary. Finally, the nodal forces can be expressed in terms of the nodal displacements like the expression shown in (9). The element stiffness can then be written explicitly, and the internal stresses can also be written in terms of the nodal displacements. Since all these expressions are rather complicated, to save the space of the report we did not write them down. One may refer to [7] for the detailed expressions. Although the final expression for the element stiffness looks complicate, it is written in an explicit real closed-form and no integrals are included. Therefore, in numerical calculation it runs quickly and can also be easily used in programming.


Based upon the usual finite element procedures and the formulations derived in subsections 2.2 and 2.3, a computer windows program was coded by using the FORTRAN and Visual Basic languages. To save the space of this report, one may refer to [7] for the detail programming techniques and numerous examples. In this section, only two simple examples are shown to illustrate the performance of these newly developed elements. The first check for our written computer program is the problem with uniform stress field whose solution should be exact no matter what element subdivision is made. The second one is the stress concentration problem of which the fine meshes near holes are needed to get a convergent solution. To show the versatility, in these two problems we include the cases of in-plane and anti-plane problems, and the anisotropic and isotropic materials. The main reason of including these special features is as follows: (1) the "two-dimensional" used in our paper title includes not only the in-plane but also the anti-plane problems and the problems where in-plane and anti-plane deformations couple each other; (2) the "anisotropic" which need not have any material symmetry restrictions also includes the degenerate cases of which the material eigenvalues are repeated such as the isotropic materials.

Example 1: Uniform stress field

Consider a rectangular composite laminate subjected to a uniform load along the boundary edges. The uniform load includes the in-plane tension $\sigma_{11}^0 =1000$ lb/in, $\sigma_{22}^0 =2000$ lb/in, in-plane shear $\sigma_{12}^0 =3000$lb/in and anti-plane shear $\sigma_{13}^0 =4000$lb/in, $\sigma_{23}^0 =5000$lb/in. The laminate has ply orientation $[0/30/-30]$ which will behave as an anisotropic material. Each lamina has a thickness of 0.04 in and is composed of Graphite/Epoxy whose material properties are: $E_1 =26.25 \times 10^6$ psi, $E_2 =1.49 \times 10^6$ psi, $G_{12} =1.04 \times 10^6$ psi, $\nu_{12} =0.28$. The length and width of the plate are $l =16$ in, $h=4$ in. To ensure the exactness of the solutions for the problems with uniform stress field, several different patches of elements have been tested and the Stroh finite element solutions (linear or quadratic elements discussed in this report) really coincide exactly with the exact uniform stress solution [7].

Example 2: Stress concentration

After the elements passed the simple test of uniform stress field, a more realistic test such as a circular hole in a rectangular plate subjected to uniform tension ($\sigma =5000$ lb/in) is considered. Two different kinds of plate are tested. One is an isotropic plate made of aluminum whose properties are: $E=10^7$ psi, $G=3.85 \times 10^6$ psi, $\nu =0.3$. The other is a fiber reinforced composite plate whose properties are: $E_1 =7.8 \times 10^6$ psi, $E_2 =2.6 \times 10^6$ psi, $G_{12} =1.25 \times 10^6$ psi, $\nu_{12} =0.25$. The length, width and thickness of the plate are, respectively, $l =48$ in, $h=40$ in and $t=1$in. Since the thickness of the plate is relatively thinner than the length and width, the plane stress condition will be assumed in our finite element modelling. Due to double symmetry, only a quadrant need be analyzed. A linear element mesh with 124 elements and 80 nodal points and a quadratic element mesh with 84 elements and 102 nodal
5 points are used in our examples. The roller boundary conditions are used to simulate the symmetry condition and to restrain the rigid body movement. The results for normal stresses on two edges are shown in Figures 3a and 3b in which the exact solutions for infinite plate [8, 9] are available for comparison. The agreement is satisfactory except for the points on the circular hole boundary, which can be improved by more fine meshes near holes. Figure 4 is a diagram showing the deformed configuration and the Von Mises stress contour, which is also agree with the plot shown by commercial finite element code NASTRAN.

4. Concluding Remarks

Through the numerical examples shown in the last section, we know that the Stroh finite element is accurate enough to perform the usual finite element task. Inheriting from the Stroh formalism, this newly developed finite element is valid for the generalized two-dimensional anisotropic elasticity problems which include in-plane and anti-plane problems with the most general anisotropic (includes monoclinic, orthotropic and isotropic, etc.) materials. Moreover, since the element already satisfies both of the compatibility and equilibrium conditions within each element, the efficiency and accuracy compared with the conventional finite element is also an interesting topic for the future study. It is hoped that through this work similar studies combining the analytical work with the numerical techniques can be raised to improve the existing technology.

Acknowledgements

The authors would like to thank the support by National Science Council, Republic of China, through Grant No. NSC 89-2212-E-006-022.

References


計畫成果自評:

本計畫已成功地結合了有限元素法及史磋複變解析方法並將數學列式以Fortran電腦語言設計程式，同時以Visual Basic設計一適用於Windows視窗的軟體。這在文獻史上應是第一次，我們寄望兩者的結合可以帶來數值方法的改善及解析方法在應用面上的延伸。