View metadata, citation and similar papers at core.ac.uk
brought to you by CORE
provided by NASA Technical Reports Server

N75-24875

Grumman Research Department Report RE-480

File with N74-30339

APPLICATION OF A SUBSTRUCTURING TECHNIQUE TO THE PROBLEM OF CRACK EXTENSION AND CLOSURE

Prepared under Contract NAS 1-12198

for

Langley Research Center
National Aeronautics and Space Administration
Hampton, Virginia 23365

by

H. Armen, Jr.

Research Department
Grumman Aerospace Corporation
Bethpage, New York 11714

July 1974

Approved by: Charles E. Mack, Jr.
Charles E. Mack, Jr.
Director of Research

Page Intentionally Left Blank

FOREWORD

This study was conducted by the Research Department of Grumman Aerospace Corporation, Bethpage, New York, under partial support of Contract NAS 1-12198.

The author wishes to acknowledge the valuable contribution of Drs. A. Pifko and H. Levine for discussions concerning the substructuring technique. Thanks also go to Ms. E. Yander for data preparation and programming and to Ms. C. O'Regan for the technical illustrations.

Page Intentionally Left Blank

TABLE OF CONTENTS

LIST OF ILLUSTRATIONS

LIST OF SYMBOLS

e	total strain
e^e	elastic strain
e^p	plastic strain
F	combined load vector
K	assembled stiffness influence coefficient matrix
k	element stiffness influence coefficient matrix
$\bar{k}$	element initial strain influence coefficient matrix
P	applied load vector
Q	effective plastic load vector
R	equilibrium imbalance load vector
T	forward solution used in substructing technique
u	generalized nodal displacements
u^*	modified nodal displacements

Notation

i	used as a subscript denotes unmodified generalized nodal displacements
e	used as a subscript denotes modified generalized nodal displacements
{ }	column vector
[ ]	square or rectangular matrix
$[\ ]^T$	transpose of matrix
$[\ ]^{-1}$	inverse of matrix

SUMMARY

A substructuring technique, originally developed for the efficient reanalysis of structures, is incorporated into the methodology associated with the plastic analysis of structures. An existing finite-element computer program that accounts for elastic-plastic material behavior under cyclic loading was modified to account for changing kinematic constraint conditions - crack growth and intermittent contact of crack surfaces in two dimensional regions. Application of the analysis is presented for a problem of a center-crack panel to demonstrate the efficiency and accuracy of the technique.

Page Intentionally Left Blank

1. INTRODUCTION

Fatigue crack propagation, until recently, was generally assumed to be directly related to the linear elastic stress-intensity factor range during cyclic loading. Implicit in this concept were the assumptions that only the tensile portion of the load cycle was effective in growing the crack and that crack closure occurs only at zero load. Elber (Refs. 1 and 2) has shown experimentally that fatigue cracks close at positive stresses during constant amplitude stress cycling. He has also indicated that fatigue-crack closure may be a significant factor in causing the stress-interaction effects (crack growth delay or acceleration) on crack growth under general cyclic loading. The crack closure phenomenon, associated with an extending crack, is believed to be caused by residual plastic deformations remaining in the wake of an advancing crack tip. A reasonable analytic model for crack closure and for the extending crack problem must possess the capability of accounting for changing boundary conditions (crack growth and intermittent contact of the crack surfaces) during a specified load history. These changing boundary conditions must be incorporated into the equations that govern the nonlinear load-deformation behavior.

The present report is concerned with the modification of an existing nonlinear finite element program (Ref. 3) to account for crack extension and crack closure and the application of this program to the study of fatigue-crack closure. The modification of the existing finite element program consists of incorporating a method for the efficient reanalysis of structures (Refs. 4 and 5) that undergo changes in material properties or restraint conditions. The procedure has the advantage of not requiring the reformulation of the stiffness influence coefficients of the original, unmodified structure.

The technique to treat plasticity in the FAST (Fracture Analysis of STructures) program is based on the initial strain concept where an effective plastic load vector is introduced, in addition to the applied mechanical load, to account for the development of plastic deformation. Thus, the procedure used for the reanalysis of structures with variable restraint conditions is extended to include the effects of plasticity and has been incorporated into the final program.

Previous Studies

The problem of an extending crack has been previously treated within the framework of a nonlinear finite element analysis. The procedure as described in Refs. 6 and 7 involves establishing an elastic-plastic state including the nodal forces (reactive forces) that hold together an element node directly ahead of a crack tip. An equal and opposite nodal force is then applied in increments to the crack tip node until this restraining force is completely removed. The node is then assumed to be free and displaces an amount representing the crack opening displacement. In Ref. 6, the crack tip is assumed to advance to a new position, corresponding to the adjacent node of the previous location of the crack tip. After a "step-of-growth," as described in Ref. 7, the finite element mesh is shifted to a new position so that the new crack tip is in the same position as the previous one. Details associated with this shifting procedure, which simulates an infinite process, are not given. However, the following statement that appears in Ref. 6 should be noted: "In practice, the cumulative numerical errors involved in these incremental loadings become intolerable after a few small increments of crack extension." For reasons discussed in Section 2, the procedure used in Refs. 6 and 7 appears to be incomplete and should, consequently, lead to numerical difficulties.

Another approach (Ref. 8), involves the use of a layer of "variable-material" elements beneath an axis of symmetry along which a crack can extend. These special elements take on soft, "jell-type" stiffness properties along the open (and extended) crack length and become relatively stiff, to simulate a rigid constraint, during crack closure. This model, though physically satisfying, requires, at best a partial reformulation of the assembled stiffness matrix, and at worst (if the program is so constructed) a complete reformulation and reanalysis.

Another approach involves an iterative solution to the governing equations that are determined for an *a priori* set of kinematic boundary conditions. The appeal of this approach is that the stiffness influence coefficient matrix remains unchanged. Thus, a solution technique such as the Cholesky triangularization of a positive definite, symmetric matrix need be performed only once, leading to a potentially rapid iterative process. However, application of this technique to simple yet representative crack extension problems reveals this procedure to be nonconvergent. It is our opinion that the process may be made convergent by the introduction of a relaxation factor in the iterative process. Since the form of this factor is arbitrary and will, in general, vary with each problem, and within a problem for various combinations of separation and contact, the iterative approach was abandoned.

2. METHOD OF ANALYSIS

The approach selected for the plastic analysis of structures, as available in the FAST program, incorporates the initial strain concept within the framework of the finite element method. In this technique the load-deflection relations are written to include the effects of initial (or thermal) strains. These initial strains are then interpreted as plastic strains and the problem is solved by using the load-deflection-initial strain relations with subsidiary constitutive relations for an elastic-plastic engineering material. An extensive presentation of the governing matrix equations and of the nonlinear constitutive relations is presented in Refs. 3, 9, and 10 and is only briefly reviewed here in the context of the modifications required to treat intermittent contact and separation.

Governing Equations

The matrix equation governing the response of a structure to some arbitrary history of loading can be written (as in Ref. 10);

$$[K]\left\{u\right\} = \left\{P\right\} + \left\{Q\right\} + \left\{R\right\} \tag{1}$$

where

- $[K]$ = the conventional elastic stiffness influence coefficients
- $\{u\}$ = generalized nodal displacements
- $\{P\}$ = generalized nodal forces
- $\{Q\}$ = "effective" plastic load that accounts for the presence and development of plastic strains
- $\{R\}$ = an equilibrium imbalance that may exist as a result of the nature of the solution procedure (Ref. 10)

If we decompose the total strains, $\{e\}$, into elastic $\{e^e\}$ and plastic components $\{e^p\}$ as

$$\{e\} = \{e^e\} + \{e^p\} \tag{2}$$

then for small deformations (no geometric nonlinearities), the stiffness matrix $[K]$ in Eq. (1) is the assemblage of the element stiffness matrices (see Ref. 10), and the effective plastic load vector $\{Q\}$ is

$$\{Q\} = \sum_{i=1}^{N} [\bar{k}_i] \{e^p\} \tag{3}$$

where N is the number of finite elements in the plastic range, and $[\bar{k}]$ is the initial strain matrix of the individual elements. Note that in the FAST program the plastic strains are assumed to be constant quantities (at centroids of elements). If a distribution of plastic strain within an element is desired, its assumed functional form must be considered in determining $[\bar{k}]$.

Solution Procedure for Unmodified Structure

The algorithm for the incremental procedure used to solve for stresses, strains, and displacements for a typical load history follows:

1) Determine the generalized displacements, $\{u\}$ from the solution of Eq. (1)

2) Use the solution from step 1 and appropriate strain-displacements relations to determine the total strains, $\{e\}$, and the increment of total strain $\{\Delta e\}$ for the increment of load, $\{\Delta p\}$

3) Compute the increments of stress $\{\Delta\sigma\}$ and the increment of plastic strain $\{\Delta e^p\}$ from the total strain increment $\{\Delta e\}$

4) Determine an updated plastic load vector $\{Q\}$ from Eq. (3) and mechanical load $\{p\}^k = \{p\}^{k-1} + \{\Delta p\}$

5) Repeat steps 1 through 4

From the above it is apparent that at load step k the plastic strains and hence the effective plastic load vector cannot be determined without an *a priori* knowledge of the displacement. In the method chosen for use, the effective plastic load is taken to be equal to that computed in the preceding load increment and is thus taken as a known quantity in the equation.

The final form of Eq. (1) is

$$[K]\{u\}^k = \{P\}^k + \{Q\}^{k-1} + \{R\}^k \tag{4}$$

where k and k-1 are the current and preceding load steps, respectively. The magnitude of the vector of equilibrium imbalance loads, $\{R\}$ is a result of the incremental linearization of the nonlinear problem and the error associated with using an estimated value of $\{Q\}$. The effect of the linearization error is reduced by using sufficiently small load steps, whereas the latter error is accounted for by setting

$$\{R\}^k = \sum_{i=1}^{N} [\bar{k}_i]\{\Delta e^P\}^{k-1}$$

where $\{\Delta e^P\}$ is the increment of plastic strain determined in the preceding step.

<u>Solution Procedure for Variable Restraint Conditions</u>

It is desired to determine a set of modified displacements $\{u^*\} = \{u + \delta u\}$ due to a change in stiffness from $[K]$ to $[K + \delta K]$. This modification may result from the separation (crack extension) or coalescing of two nodes to simulate contact. Equation (4) now becomes

$$[K + \delta K]\{u^*\} = \{F\} \tag{5}$$

where

$$\{F\} = \{P\} + \{Q\} + \{R\} \tag{6}$$

The modified displacement vector $\{u^*\}$ can, of course, be found by a complete reanalysis (including reassembly) using the modified stiffness influence coefficients. For localized changes (under consideration here) this would be most uneconomical.

The procedure chosen for implementation originally appeared in Refs. 4 and 5 and resembles the substructuring approach presented in Ref. 11. This technique, modified to treat the introduction of additional degrees of freedom, is incorporated into the FAST program, and is briefly described as follows:

Consider Eq. (5) to be partitioned into

$$\left[\begin{array}{c|c} K_{ii} & K_{ie} \\ \hline K_{ei} & K_{ee} \end{array}\right] \left\{\begin{array}{c} u_i \\ \hline u_e \end{array}\right\} = \left\{\begin{array}{c} F_i \\ \hline F_e \end{array}\right\} \tag{7}$$

where using the notation of Ref. 5, the subscripts i refer to the unmodified degrees of freedom and e to those that are to be modified; i.e., originally restrained against motion and now allowed to displace. Equation (7) may be expanded into the two equations

$$[K_{ii}]\left\{u_i^*\right\} + [K_{ie}]\left\{u_e^*\right\} = \left\{F_i\right\} \tag{8a}$$

$$[K_{ei}]\left\{u_i^*\right\} + [K_{ee}]\left\{u_e^*\right\} = \left\{F_e\right\} \tag{8b}$$

If we initially consider $\{u_e^*\} = 0$, then we can, from Eq. (8a), solve for a quantity $\{\bar{u}_i\}$, as

$$\left\{\bar{u}_i\right\} = [K_{ii}]^{-1}\left\{F_i\right\} \tag{9}$$

By means of the Cholesky triangularization (see Appendix C) of a positive definite, symmetric matrix the stiffness matrix $[K_{ii}]$ can be written as the product

$$[K_{ii}] = [L][L^T] \qquad (10a)$$

where L is a lower triangular array. Therefore, we can write

$$[K_{ii}]^{-1} = [L]^{-T}[L]^{-1} \qquad (10b)$$

From Eqs. (8a) and (9), we can express the displacement in the modified structure at the i^{th} degree of freedom as

$$\left\{u_i^*\right\} = \left\{\bar{u}_i\right\} + \left\{\delta u_i^*\right\} \qquad (11)$$

where the change in displacements $\{\delta u_i^*\}$ is written as

$$\left\{\delta u_i^*\right\} = -[L]^{-T}[T_e]\left\{u_e^*\right\} \qquad (12)$$

and $[T_e] = [L]^{-1}[K_{ie}]$. The displacement at the e^{th} degrees of freedom are obtained by solving Eq. (8b) and using substitutions from Eqs. (11) and (12), i.e.,

$$[K_{ee} - T_e^T T_e]\left\{u_e^*\right\} = \left\{F_e\right\} - [K_{ie}]^T\left\{\bar{u}_i\right\} \qquad (13)$$

Solving for $\{u_e^*\}$ from Eq. (13), we can determine the changes in the remainder of the structure by using Eq. (12).

The above procedure has the advantage of not having to reformulate the stiffness influence coefficient matrix [K]. It does involve determining the value of $[T_e]$, from Eq. (12) (a forward solution of the triangularized array); a solution for $\{u_e\}$ (solving a system of n simultaneous linear equations, where n is the number of modified degrees of freedom); and determining $\{u_i^*\}$ from Eq. (12) (a backward solution of the triangularized system). The number of computational operations required for this technique, as determined in Ref. 5, is

$$M = n\left(\frac{bn_r}{2} + \frac{n_r^2}{4}\right) + \frac{n_r^3}{6} \tag{14}$$

where M = the operation count, n = the total number of degrees of freedom of the modified structure, n_r = degrees of freedom to be modified, and b = semibandwidth of stiffness matrix. The break-even point of this technique versus a complete reformulation occurs when $n_r = 0.75b$.

In Refs. 4 and 5 the condition that $\{F_e\} = 0$ is imposed as the requirement that the "e" degrees of freedom are free to displace. This condition, although necessary, is not sufficient to determine the actual displacements associated with the modified degrees of freedom and their influence on the remainder of the structure.

The algorithm for the incremental procedure to treat the intermittent contact and separation remains the same as that previously presented for the unmodified structure with the exception that the flow of operations is interrupted between steps 1 and 2. Nodal displacements at nodes, chosen a priori, are monitored to determine whether they are to be released (representing crack extension), remain open (crack opening), or remain fixed (crack closure). The displacements associated with these decisions are then computed and their effect on the displacement vector $\{u\}$, computed in step 1, is finally determined.

3. NUMERICAL RESULTS

A uniformly loaded rectangular center-crack panel, shown in Fig. 1, is chosen to demonstrate the previously described method of substructuring to treat crack extension and intermittent contact of crack surfaces. The finite-element idealization of a quadrant of the panel, shown in Fig. 2, consists of 396 elements having 249 nodes resulting in 474 degrees of freedom when symmetry boundary conditions are applied.

The crack displacement profile, assuming elastic behavior, is illustrated in Fig. 3 for two crack lengths. The solid curve corresponds to the displacement profile for the initial crack length $2a$; the dashed curve is associated with crack length $2a'$, where $a'/a = 1.1$. These results were obtained without utilizing the substructuring technique. The circles correspond to the displacement profile for crack length $2a'$, determined by means of the substructuring technique — i.e., nodes A B C and D were "broken," extending the crack length from $2a$ to $2a'$. As can be seen from the figure the results from this technique are identical to those obtained by assuming an initial crack length of $2a'$.

The crack displacement profile for elastic-plastic behavior is shown in Fig. 4. The results are for an elastic-ideally plastic material and for a loading (S_{max}) corresponding to 31 percent of the yield load (S_{yield}) of the uncracked specimen. At S_{max} the crack is extended from a_o to a' (node B to C in Fig. 2) where $a'/a_o = 1.024$. The results indicate a chisel-shaped profile for the displacements in the vicinity of the extended crack-tip. A similar discontinuity for the profile of an extended crack is presented in Ref. 11. Details of the displacement profile in the near-tip region is presented in Fig. 5. The results for the extended crack are compared to corresponding results for the crack of initial length $2a'$. The results from the latter case do not, of course, indicate the discontinuous chisel-shape profile corresponding to the extended crack.

4. CONCLUDING REMARKS

Application of a substructuring technique to the problem of crack extension and closure has been outlined and implemented into an existing nonlinear finite element analysis program for two dimensional membrane stressed structures. The method, readily implemented without a significant degree of disruption of the flow of the original program, appears to be particularly well suited for adaptation within the framework of the initial strain approach for the treatment of nonlinear material behavior. The advantage is associated with the fact that the original stiffness influence coefficient matrix for the unmodified structure need not be altered at any point in the analysis.

Results, demonstrating the technique to the problem of elastic and elastic-plastic crack extension, are presented for a uniformly loaded center-crack panel. A more comprehensive application of the technique to crack extension and closure is presented in Ref. 12.

APPENDIX A

DESCRIPTION OF FAST

FAST is an acronym for Fracture Analysis of STructures. The analytic foundation of the program is the displacement method of finite element techniques for structural analysis. The program represents a spin-off of a previous program labeled PLANE (Ref. 3) developed for the nonlinear analysis of structures subjected to plane stress conditions. The additional capability of FAST, and one that distinguishes it from PLANE, is the ability to treat variable restraint conditions so that consideration can be given to the problem of an extending crack or to determining the effects of crack closure.

The program is capable of treating the elastic and the elastic-ideally plastic response of orthotropic materials. In addition, consideration is given to isotropic materials exhibiting elastic-ideally plastic, linear strain hardening, or nonlinear strain hardening behavior. Further, the kinematic hardening theory of plasticity is used (Refs. 13 and 14) so that provision for cyclic loading conditions involving reversed plastic deformation is included.

The finite element library and a description of each element follows.

Constant Strain Triangle

Constant Strain Triangle (CST), a well-known plane stress membrane element, was used successfully for the idealization of structures presented in Ref. 9. Its derivation is based upon the assumption of a linear distribution for the in-plane displacements u and v, and consequently, leads to a constant strain state within the element (Fig. A-1). Each vertex is allowed two degrees of freedom (the in-plane displacements u and v) for a total of six degrees of freedom for the element. Consistent with the total strain distribution, the initial strains (plastic strains) are assumed to be constant within each element. Stiffness and initial strain matrices have been developed and successfully used in Refs. 9 and 10.

Linear Strain Triangle

In regions of high strain gradient, the CST triangle is not sufficiently accurate to be used in a plasticity analysis unless a very fine grid is employed. The linear strain triangle (LST) remedies this shortcoming of the CST element. The assumption of a quadratic distribution for the in-plane displacements allows for a linear strain variation within the triangle. Two degrees of freedom at each node (u,v) for each of the six nodes (three vertex and three midside nodes) give this element a total of 12 degrees of freedom. The initial strains are assumed to be constant within each element and are evaluated at the centroid. Both stiffness and initial strain matrices have been developed and successfully used in Ref. 10.

Hybrid Triangles

In transition regions — those regions in which stresses and strains change from rapidly varying to slowly varying — it becomes convenient and efficient to switch from linear strain triangles to constant strain triangles (Fig. A-3). This is accomplished by using four and five node triangles to maintain compatibility with both the CST and LST elements. These elements together with the CST and LST elements were originally used in Ref. 15, and are referred to as the TRIM 3 through TRIM 6 family. For these mixed formulation hybrid elements, the displacements along edges may vary quadratically or linearly, depending upon whether an LST or CST triangle is contiguous to the respective sides. Again, the plastic strain distribution is assumed constant within each element.

Stringers

For many aircraft structures — fuselages, wings, etc. — local stiffening is required to provide adequate stability in compression (Fig. A-4). Special one dimensional finite-elements are required to represent the stringers used for this purpose. Uniform cross section stringer elements have been developed using both constant and linearly varying strain assumptions, so that compatibility with both the CST and LST elements can be maintained. For the constant strain stringer, a linear axial displacement and a constant initial (plastic) strain distribution within the element are assumed. The linear strain stringer stiffness matrix is based upon a quadratic displacement assumption and the initial (plastic) strains are constant within the element.

APPENDIX B

USERS MANUAL

The input to the program can be categorized in sections. A description of the sections follows.

1. Problem Title (20A4)

Any 80-character title describing the problem.

2. NPNTC NPRNT (2I5)

$0 \leq$ NPNTC ≤ 63:

NPNTC is the sum of the following codes according to the options desired. A zero value of NPNTC indicates that none of the intermediate output will be printed.

1 Print the load vector.

2 Print element stiffness matrix without stringer contribution for elements with stringers.

4 Print element stiffness matrix before condensation to 8 x 8 or 10 x 10 for 4- and 5-node elements, respectively.

8 Print each element stiffness matrix in its final form.

16 Print each element stiffness matrix entry to be stacked with its stacking index.

32 Print the master assembled stiffness matrix for the structure to be analyzed.

NPRNT:

If ≤ 0, perform elastic analysis only.

If > 0, perform plastic analysis, printing output every NPRNT increment of load.

3. Nodes Within Partitions (16I5)

A partition is defined as a set of degrees of freedom of a subregion of the structure that can interact only with its neighboring subregions. The resulting stiffness influence coefficient matrix is blocked by partitions into a tridiagonal form as shown in Eq. (C-1) of Appendix C, where those partitions not in the tridiagonal region are null arrays.

Specify the nodes of each partition by listing them in consecutive fields. Node numbers must be positive and not exceed 32,768. One blank field must separate the listings of each pair of adjacent partitions. Two consecutive blank fields (with one or both on a final blank card if and only if necessary) denote the end of the section. Two shorthands are allowed. Specifying m and -n consecutively is the equivalent of m, m+1, m+2, ..., n, and specifying m, -n, and -p consecutively is the equivalent of m, m+n, m+2n, ..., m+kn, where m+kn is the highest integer of this form less than or equal to p.

The maximum number of partitions has, arbitrarily, been set to 20 and a minimum of 2 partitions is necessary for a successful solution. Consecutive numbering of nodes within any partition is not required. It should be noted that the nodes of any element must be in either one partition, or at most, two contiguous partitions. The total maximum number of nodes for any structure is set at 900. The total number of degrees of freedom ($\approx$ 2 x number of nodes) within any partition is determined by the value of NCORP defined in a data statement in subroutine MAIN. A typical value of NCORP = 10,000 words results in a maximum of $\approx$ 80 degrees of freedom in any of two contiguous partitions. A more detailed discussion of the storage allocation is presented in Appendix C.

4. Modified Nodes and Degrees of Freedom for Elastic Analysis Only: (3(2I5, 15 x)) MNOD, MODOF

The first I5 field is associated with the node to be released (MNOD) from a fixed boundary condition; the second represents the degree of freedom, u (in the global x-direction and/or V (in the global y-direction). If

MDOF = 10, u is released

= 1, V is released

= 11, u and V are released

Two separate elastic solutions for displacement strains and stresses will be printed. The first solution is associated with the case where the modified nodes are assumed to be fixed. The second solution represents the corresponding results when the modified nodes are released. A blank first I5 field ends this section. A blank card is required when there are no modified nodes.

5. Nodes and Stringers for Members 2(7I5,2 x,3I1)

The first I5 field gives the member number, which must be positive and no greater than 32,768. The next six I5 fields give the nodes, beginning with a major node and continuing around the perimeter of the member alternately major and minor. The absence of a minor node must be indicated by a zero or blank field in the proper position. The 3I1 digits indicate, respectively, the presence or absence of a stringer connected to sides 4, 5, and 6, where side 4 is the side opposite the first major node given in the six I5 fields, side 5 is opposite the second, and side 6 is opposite the third (see Fig. B-1). It is suggested that the digits 4, 5, and 6 thus be used in the appropriate positions to denote the presence of a stringer on the corresponding side, although any nonzero digit will suffice. A zero in an I1 field denotes the absence of a stringer from the corresponding side. Two members may be specified per card (columns 1-40). A zero or blank first card field ends the section. The program will accept a maximum number of 600 elements. It should be noted that the program will accept the case of specifying a minor node in only one of two adjacent elements sharing a common side. The displacement field along this side of the elements will be compatible only at the two major end nodes and not along the length of the side.

6. X-Coordinates of Nodes (E15.7,13I5)

The X-coordinates of the nodes appearing in the I5 fields are set to the value in the E15.7 field. Any number of continuation cards may be used; their first fifteen columns are ignored. A zero or blank I5 field terminates the node list for a given X-coordinate. A zero or blank first I5 field (columns 16-20) where a new X-coordinate can be specified end the section. Both shorthand representations of Section 3 are allowed. Coordinates for major nodes only are errors, but will be ignored.

7. <u>Y-Coordinates of the Nodes</u> (Same as Section 6)

8. <u>Stringer Information (5E15.7,/,(16I5))</u>

To be provided only if there are stringers. The first card gives, in order,

E	Young's modulus
A	Cross-sectional area
FN	If SIGZS≠0, FN=n, the shape parameter used in Ramberg-Osgood representation of stress-strain behavior; If SIGZS=0, FN=α, the slope of the linear strain-hardening stress-strain representation, i.e., $\alpha = E_T/E$ where E_T is the tangent modulus.
SIGO	Yield stress
SIGZS	If ≠/0; SIGZS = Ramberg-Osgood parameter $\sigma_{0.7}$
Note:	If FN=0 and SIGZS=0 the material for the stringer element(s) is assumed to be elastic-ideally plastic.

The following cards give the end points of stringers to which the above values are to be assigned. Every pair of nonzero integers appearing in this input is treated as a stringer end point pair. Blank fields are ignored, except that a fully blank card ends the end point pair input. A zero E ends the section. If an odd number of end points are specified, problem execution is terminated.

9. Boundary Conditions (I4,I1,5x,14I5)

The first two fields give a boundary condition specification, in the order u_x, u_y.

Zero denotes a fixed degree of freedom.

One denotes a free degree of freedom.

Two will result in the application of a unit displacement.

The 14I5 fields give applicable nodes, with both shorthands of Section 3 permitted. Any number of continuation cards may be used for a given specification. A zero or blank I5 field ends each node listing. A zero or blank first I5 field (columns 11-15), where a new specification is expected, ends the section. If a node's boundary conditions are not specified in this section, both degrees of freedom are assumed to be free.

10. Material Properties (5E15.7,/,4E15.7,/,5E15.7,/,(16I5))

The first three cards specify material properties, as follows.

Card 1: EONE = Young's modulus in principal property axis (1)
ETWO = Young's modulus in principal property axis (2)
BETA = Angle measured from global x-axis and principal property axis (1)
GONTO = Shear modulus in (1)-(2) principal property axes.
VONTO = Poisson's ratio, ν_{12}

Note (1): $\nu_{12} = \frac{E_2}{E_1} \nu_{21}$

Card 2: SIGOX = Yield stress in global x-axis
SIGOY = Yield stress in global y-axis
SIGOZ = Yield stress in global z-axis
SIGXY = Shear yield stress in global x-y plane

Card 3: RMOSN = If SIGZS$\neq$0; RMOSN=n, the shape parameter used in Ramberg-Osgood representation of stress-strain behavior.

If SIGZS=0; RMOSN=α the slope of the linear strain-hardening stress-strain representation, i.e., $\alpha = E_T/E$ where E_T is the tangent modulus.

SIGZS = If $\neq 0$; SIGZS = Ramberg-Osgood parameters $\sigma_{0.7}$

Note (2): If RMOSN=0 and SIGZS=0 the material for the element(s) is assumed to be elastic-ideally plastic.

RMOSE = Ramberg-Osgood parameter E (Young's modulus)
YLDST = Yield stress in tension
YLDSC = Yield stress in compression

Note (3): Only initially isotropic materials can be treated when considering linear or nonlinear strain hardening.

Succeeding cards give applicable members; both shorthands of Section 3 are permitted. Any number of continuation cards may be used for a given specification. A zero or blank I5 field ends each member listing. A zero or blank EONE ends the section.

11. Member Thicknesses (E15.7,/,(16I5))

The first card specifies the member thickness, the 16I5 cards give applicable members, as in 10. A zero or blank thickness ends the section.

12. Applied Load Components (3I5,4E15.7)

Each card gives the load components applied at a member side, as follows:

First I5 field:	number of node (m)
Second I5 field:	number of minor node on the side, or zero or blank if there is no minor node
Third I5 field:	number of other end point node (n)
First E15.7 field:	x load component at node m
Second E15.7 field:	y load component at node m
Third E15.7 field:	x load component at node n
Fourth E15.7 field:	y load component at node n

As many load components as desired may be specified. A zero or blank first I5 field ends the section.

13. Members to Be Printed - Elastic Solution (16I5)

Specify the members whose strains and stresses are to be printed in the elastic analysis in the order in which they are desired. Blank fields are ignored, provided one nonblank field appears on a card. Both shorthands of Section 3 are allowed. Should three consecutive negative integers appear, a "1" is inserted before the third. A maximum of 600 members may be specified. Members in excess of 600 and undefined member numbers are ignored. A blank card or card with only zero entries end the section.

14. Nodes to Be Printed - Elastic Solution (16I5)

Up to 900 nodes whose displacements are to be printed for the elastic analysis, as per Section 13.

15. Members to Be Printed - Plastic Solution (As per Section 13)

16. Nodes to Be Printed - Plastic Solution (As per Section 13)

17. Modified Nodes and Degrees of Freedom for Plastic Analysis Only: (3(2I5,15x)) MNOD, MDOF (As per Section 4)

A blank card is required when there are no modified nodes.

18. Plastic Parameters (3E15.7)

In order,

PPCT	Load increment as a percentage of yield load.
PMAX	Maximum load to be applied.
YEPS	$\leq$ 1.0; the amount by which YEPS is less than 1.0 represents the tolerance requirements in determining whether a particular stress state satisfies the yield condition.

19. Load Range for Problems Involving Modified Nodes Only: (A4,1x,E15.7) TEMP(1), PMAX

PMAX represents the maximum or minimum value of the applied loading for any half-cycle of load. Any number of half-cycle loadings may be applied. The field for TEMP(1) must be blank for any specified value of PMAX and must read

"ENDb," where b denotes a blank, for the lost input card. If there are no modified nodes this section is ignored (no blank card is required)

20. <u>For Succeeding Load Cycles, One Card (I5,2E15.7) Giving New NPRNT, PMAX, and PPCT</u>

Zero NPRNT signifies no new load cycle. To change properties of any group of members, as given in Section 9, there may follow any number of cards (I5,5E15.7) giving I any member of the group whose properties are to be changed.

$$\left.\begin{array}{l}\text{RMOSN}\\ \text{RMOSS}\\ \text{RMOSE}\\ \text{YLDST}\\ \text{YLDSC}\end{array}\right] \text{— New Ramberg-Osgood parameters}$$

I = 0 indicates that the changes are complete.

This section is ignored if the problem involves modified nodes.

21. <u>Each Problem's Input Must Be Ended with a Card Reading "ENDb," where b Denotes a Blank, in Columns 1-4</u>

An additional "ENDb" is not required for those problems involving modified nodes.

APPENDIX C

SOLUTION ALGORITHM

The algorithm chosen for the solution of symmetric, positive definite matrix equations in block tridiagonal form is the Cholesky method (referred to as Danilevskii method in Ref. 16). This algorithm factors the total stiffness matrix into the product of a lower triangular array and its transpose and then solves a pair of triangular sets of equations. This factorization is possible only for positive definite matrices.

Problem

Solve $AX = Y$, where

$$A = \begin{bmatrix} A_1 & B_1 & & \\ B_1 & A_2 & B_2^T & \\ & B_2 & A_3 & B_3^T \\ & & B_3 & A_4 \end{bmatrix} \tag{C-1}$$

is the block tridiagonal, positive definite, symmetric stiffness matrix and X and Y (representing the generalized nodal displacements and loads, respectively) are partitioned correspondingly as

$$X = \begin{Bmatrix} X_1 \\ X_2 \\ X_3 \\ X_4 \end{Bmatrix}, \quad Y = \begin{Bmatrix} Y_1 \\ Y_2 \\ Y_3 \\ Y_4 \end{Bmatrix} \tag{C-2}$$

Triangularization

Assume $A = LL^T$ with

$$L = \begin{bmatrix} L_1 & & & \\ M_1 & L_2 & & \\ & & & \\ & M_2 & L_3 & \\ & & M_3 & L_4 \end{bmatrix} \tag{C-3}$$

and A partitioned as already indicated. Then

$$A_1 = L_1 L_1^T$$

$$B_1 = M_1 L_1^T \qquad \text{so that} \qquad M_1 = B_1 L_1^{-T}$$

$$A_2 = M_1 M_1^T + L_2 L_2^T \qquad \text{so that} \qquad L_2 L_2^T = A_2 - M_1 M_1^T$$

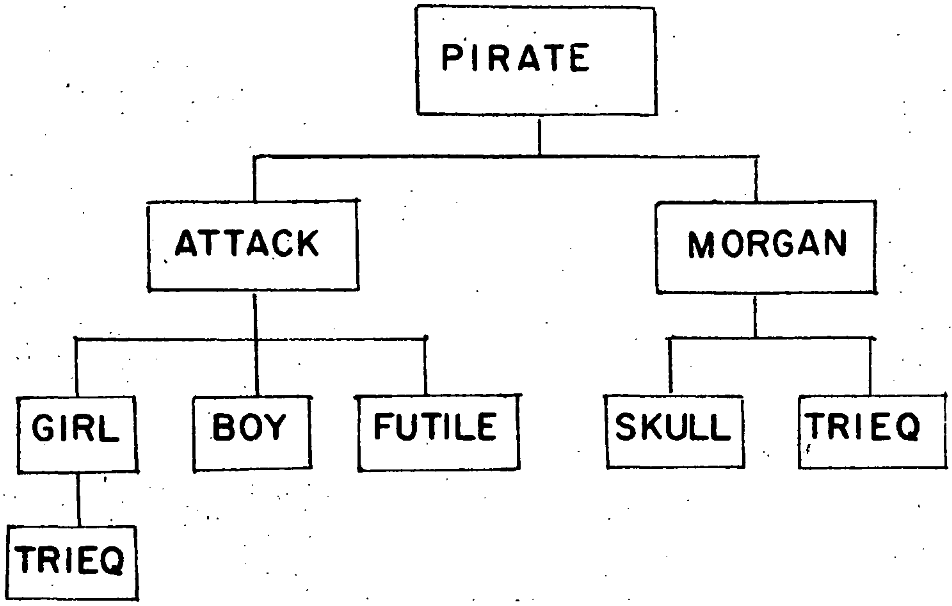
$$B_2 = M_2 L_2^T \qquad \text{so that} \qquad M_2 = B_2 L_2^{-T}$$

$$A_3 = M_2 M_2^T + L_3 L_3^T \qquad \text{so that} \qquad L_3 L_3^T = A_3 - M_2 M_2^T$$

etc.

These equations are used in turn to determine L_1, M_1, L_2, M_2, etc., obtaining each diagonal block by the Cholesky algorithm and each off-diagonal block by solving triangular equations.

Flow of Subroutines



PIRATE is the user interface and supervisory routine. It uses the Cholesky algorithm, which factors the total stiffness matrix into LL^T (where L is a lower triangular matrix) and then solves a pair of triangular sets of equations. It is well known that this factorization is possible if A is positive definite.

The factorization is carried out by subroutine ATTACK as follows:

Subroutine FUTILE performs Cholesky factorizations to construct the diagonal (triangular) blocks L_1, L_2, ...

Subroutine BOY forms the products $M_k M_k^T$

Subroutine GIRL solves equations $L_k M_k^T = B_k^T$; GIRL calls TRIEQ for each column of B_k^T to get the corresponding column of M_k^T

PIRATE uses a large storage array; at a typical moment, the array holds one diagonal block of A or L and one off-diagonal block B or M. Diagonal blocks are placed in the beginning of the large array, with off-diagonal blocks positioned at the end. This arrangement prevents storage conflicts, provided that the array is large enough to hold any diagonal block along with either (not both) of the adjoining off-diagonal blocks. The symmetry of A makes it possible to store only about half of the diagonal blocks.

Solution

After factorization, the problem can be posed in the form $LL^TX = Y$; PIRATE uses subroutine MORGAN to solve $LZ = Y$ for Z, and then $L^TX = Z$ for X; X is then the required solution because $L^TX = Z$ implies $L(L^TX) = LZ$, i.e., $AX = Y$.

The "forward" solution $LZ = Y$ can be expressed as

$$\begin{bmatrix} L_1 & & & \\ M_1 & L_2 & & \\ & M_2 & L_3 & \\ & & M_3 & L_4 \end{bmatrix} \begin{Bmatrix} Z_1 \\ Z_2 \\ Z_3 \\ Z_4 \end{Bmatrix} = \begin{Bmatrix} Y_1 \\ Y_2 \\ Y_3 \\ Y_4 \end{Bmatrix}, \tag{C-4}$$

which lead to

$$\begin{aligned} L_1Z_1 &= Y_1 \\ L_2Z_2 &= Y_2 - M_1Z_1 \\ L_3Z_3 &= Y_3 - M_2Z_2 \\ &\text{etc.} \end{aligned} \tag{C-5}$$

MORGAN calls SKULL to form the products M_kZ_k and TRIEQ to solve the triangular equations for the Z_k.

The "back" solution is so called because it obtains the solution elements in reverse order; in partitioned form the process is

$$L^T X = \begin{bmatrix} L_1^T & M_1^T & & \\ & L_2^T & M_2^T & \\ & & L_3^T & M_3^T \\ & & & L_4^T \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \\ X_3 \\ X_4 \end{Bmatrix} = \begin{Bmatrix} Z_1 \\ Z_2 \\ Z_3 \\ Z_4 \end{Bmatrix} \qquad \text{(C-6)}$$

i.e.,

$$\begin{aligned} L_4^T X_4 &= Z_4 \\ L_3^T X_3 &= Z_3 - M_3^T X_4 \qquad \text{(C-7)} \\ L_2^T X_2 &= Z_2 - M_2^T X_3 \\ &\text{etc.} \end{aligned}$$

Once again, SKULL generates the products $M_k^T X_{k+1}$ and TRIEQ provides the solutions X_k. Since the X_k are obtained in reverse order, it is necessary for MORGAN to read them backwards in order to produce the solution in normal order in core.

Data Set Usage

PIRATE uses 5 data sets, referred to here as $T_1, \ldots, T_5$. T_1 must contain the A matrix and T_2 the Y matrix when PIRATE is entered. The L matrix is written on T_3 and Z on T_4, with the solution X returned on T_5 in reverse order. It is possible to overlap the usage somewhat, but the following restrictions must be complied with:

T_1 must be different from T_2 and T_3

T_2 must be different from T_3 and T_4

T_3 must be different from T_4 and T_5

T_4 and T_5 must be different.

REFERENCES

1. Elber, W., "The Significance of Fatigue Crack Closure," ASTM STP-486, 1971.

2. Elber, W., "Fatigue Crack Closure under Cyclic Tension," Engineering Fracture Mechanics, Vol. 2, No. 1, 1970.

3. Armen, H., Levine, H., Pifko, A., and Levy, A., "Nonlinear Analysis of Structures," NASA CR-2351, March 1974.

4. Kavlie, D. and Powell, G. H., "Efficient Reanalysis of Modified Structures," Journal of the Structures Division, Proceedings of the ASCE, Vol. 97, No. ST1, January 1971.

5. Argyris, J. H. and Roy, J. R., "General Treatment of Structural Modifications," Journal of the Structures Division, Proceedings of the ASCE, Vol. 98, No. ST2, February 1972.

6. Kobayoshi, A. S., Chiu, S. T., and Beevwkes, R., "Elastic-Plastic State in Plate with an Extending Crack," Proceedings of the Army Symposium on Solid Mechanics, 1970, Light Weight Structures, AMMRC MS 70-5, December 1970.

7. Anderson, Hans, "The Steadily Growing Elastic-Plastic Crack Tip in a Finite Element Treatment," Int. Journ. of Fracture, Vol. 9, 1973.

8. Newman, J., Jr., "Finite Element Models of Fatigue Crack Closure," Ph.D. Dissertation, Virginia Polytechnic Institute and State University, 1974.

9. Isakson, G., Armen, H., Jr., and Pifko, A., "Discrete-Element Methods for the Plastic Analysis of Structures," NASA CR-803, October 1967.

10. Armen, H., Jr., Pifko, A., and Levine, H. S., "Finite Element Analysis of Structures in the Plastic Range," NASA CR-1649, February 1971.

11. Wells, A. A., "Fracture Control: Past, Present and Future," Experimental Mechanics, Vol. 13, No. 10, October 1973.

12. Newman, J., Jr. and Armen, H., Jr., "Elastic-Plastic Analysis of a Propagating Crack under Cyclic Loading," presented at the AIAA/ASME/SAE 15th Structures, Structural Dynamics and Materials Conference, Las Vegas, Nevada, April 1974.

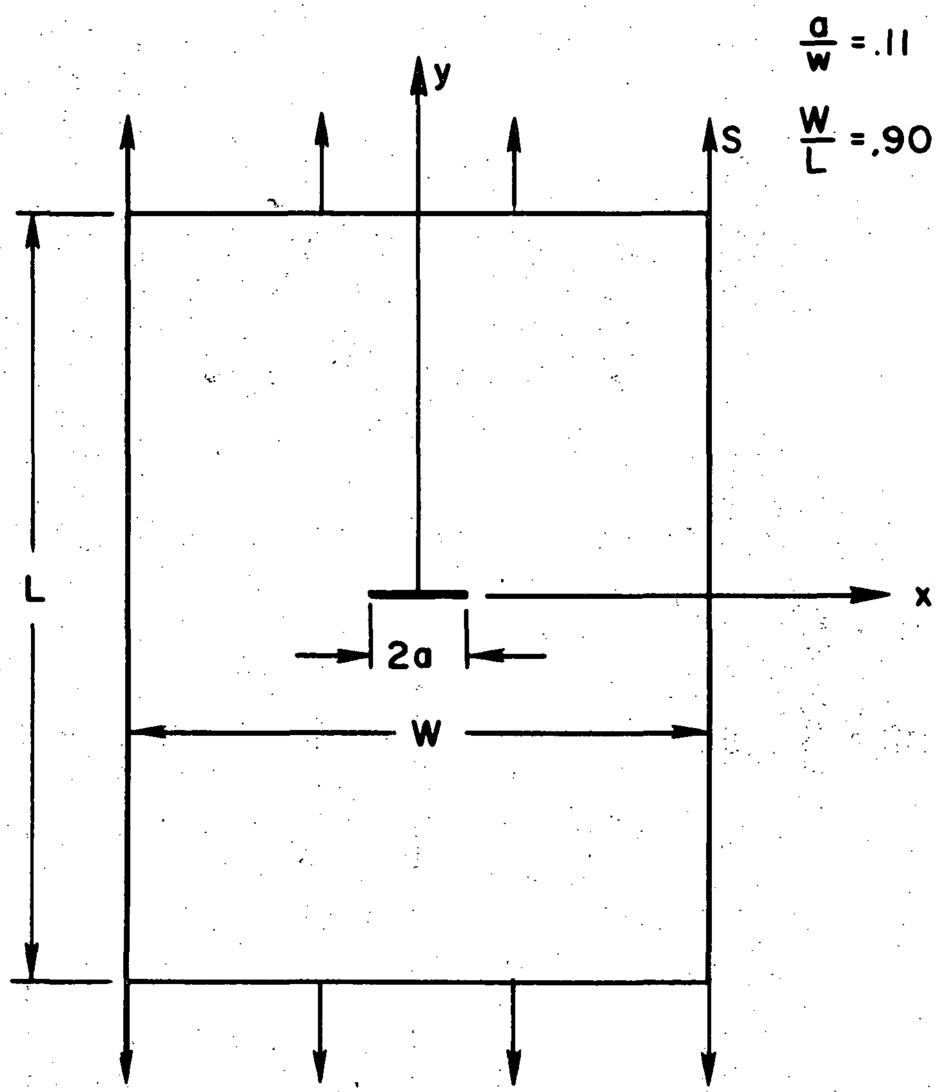


Fig. 1 UNIFORMLY LOADED CENTER CRACK PANEL.

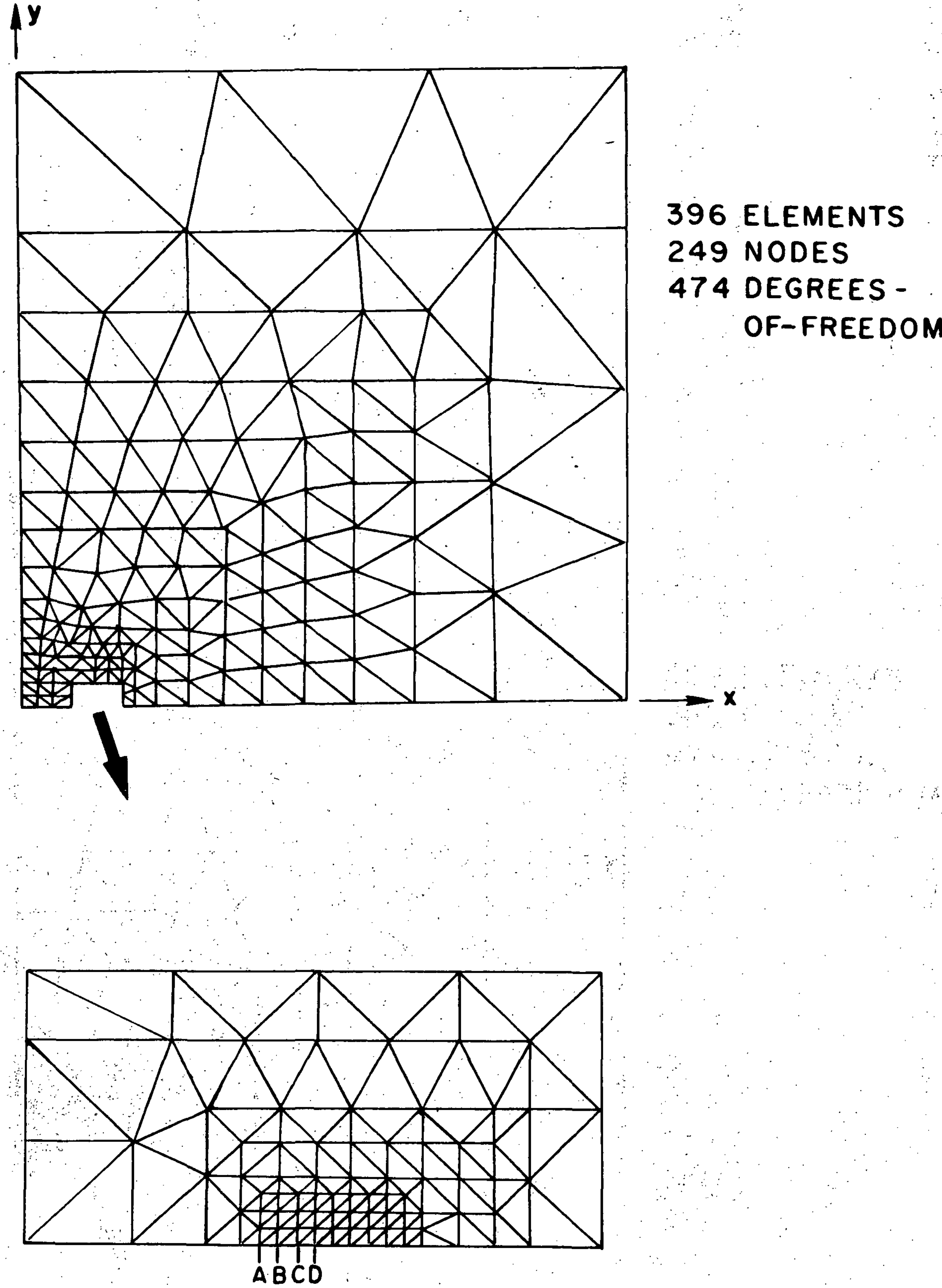


Fig. 2 IDEALIZATION OF UPPER RIGHT QUADRANT OF PANEL

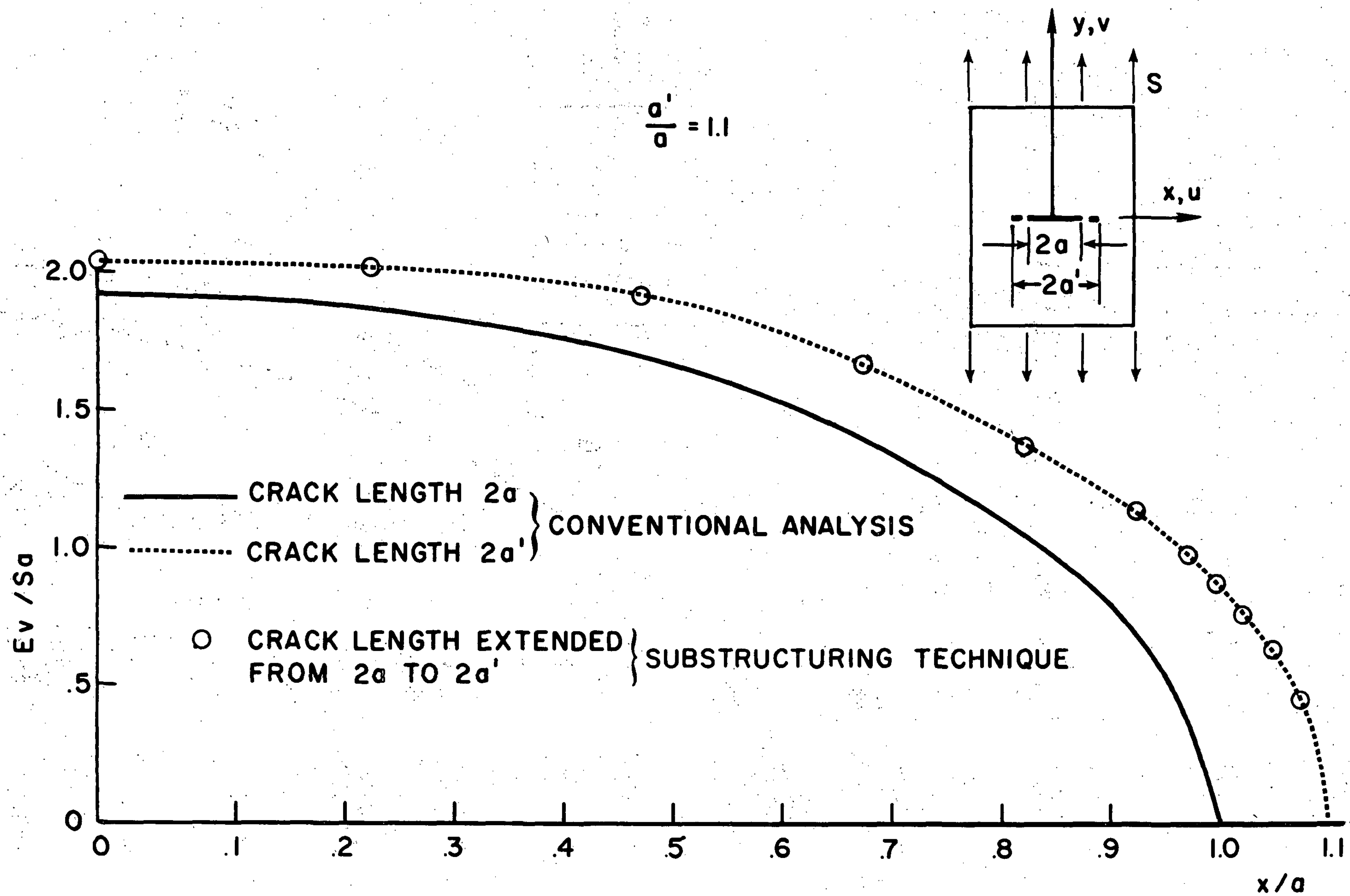


Fig. 3. CRACK OPENING DISPLACEMENT PROFILE–ELASTIC BEHAVIOR

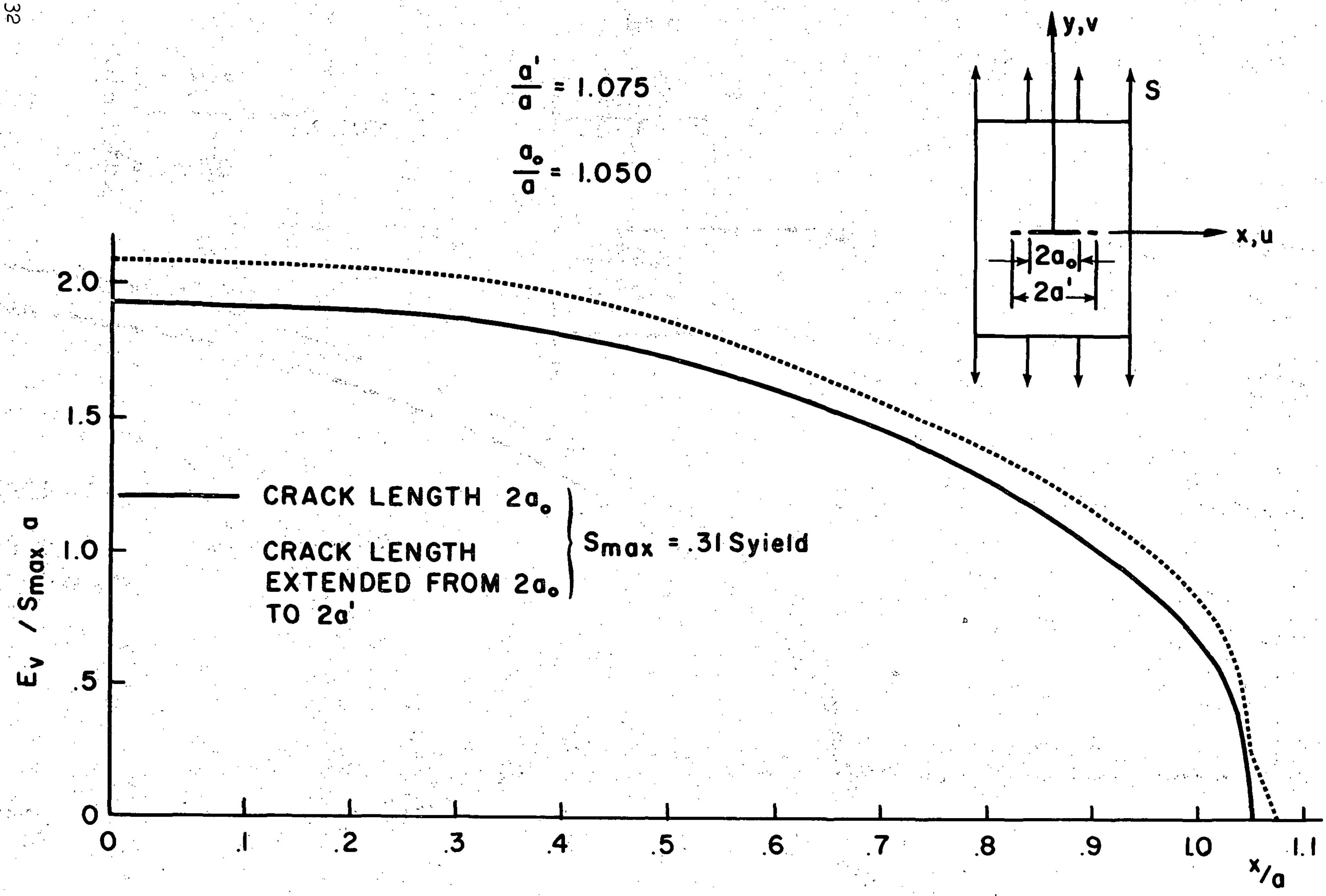


Fig. 4 CRACK OPENING DISPLACEMENT PROFILE-ELASTIC PLASTIC BEHAVIOR

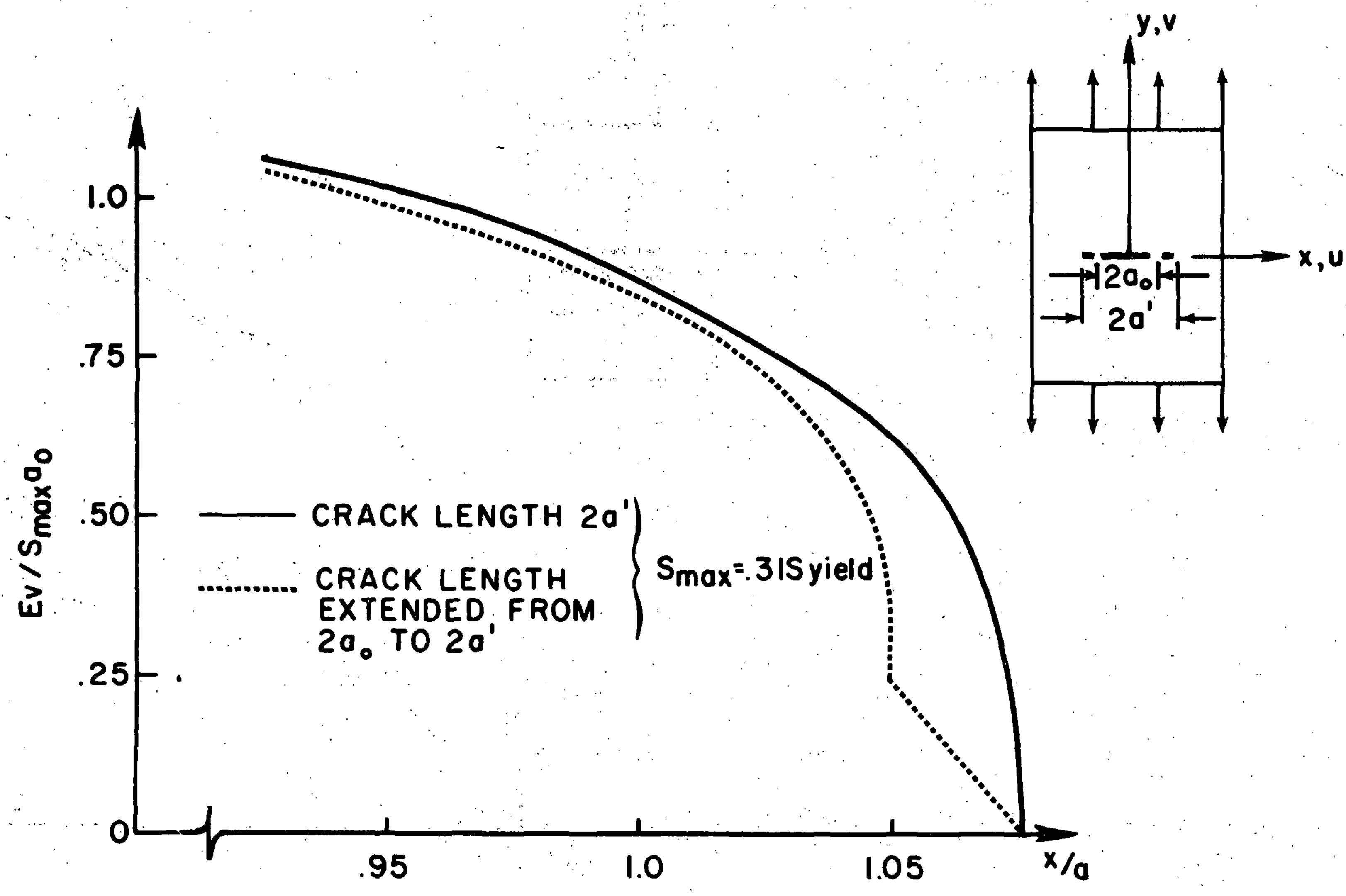


Fig.5 COMPARISON OF CRACK OPENING DISPLACEMENT PROFILE ELASTIC PLASTIC BEHAVIOR

Page Intentionally Left Blank

Page Intentionally Left Blank

GRUMMAN AEROSPACE CORPORATION
BETHPAGE, NEW YORK 11714