

21718 256 UTIS

SAND77-1900

Unlimited Release

M

CHILES 2 - A Finite Element Computer Program that Calculates the Intensities of Linear Elastic Singularities in Isotropic and Orthotropic Materials

MASTER

Steven E. Benzley, Zelma E. Benzinger



Sandia Laboratories

5010 (7-73)

DISTRIBUTION OF THIS DOCUMENT IS UNLIMITED

Issued by Sandia Laboratories, operated for the United States Department of Energy by Sandia Corporation.

NOTICE

This report was prepared as an account of work sponsored by the United States Government. Neither the United States nor the United States Department of Energy, nor any of their employees, nor any of their contractors, subcontractors, or their employees, makes any warranty, express or implied, or assumes any legal liability or responsibility for the accuracy, completeness or usefulness of any information, apparatus, product or process disclosed, or represents that its use would not infringe privately owned rights.

Printed in the United States of America

Available from:
National Technical Information Service
U. S. Department of Commerce
5285 Port Royal Road
Springfield, VA 22161
Price: Printed Copy \$4.50; Microfiche \$3.00

SAND77-1900
Unlimited Release
Printed February 1978

CHILES 2 - A FINITE ELEMENT COMPUTER PROGRAM THAT
CALCULATES THE INTENSITIES OF LINEAR ELASTIC
SINGULARITIES IN ISOTROPIC AND ORTHOTROPIC MATERIALS

Steven E. Benzley
Applied Mechanics Division II

Zelma E. Beisinger
Applied Mechanics Division I

Sandia Laboratories
Albuquerque, NM 87115

NOTICE

This report was prepared as an account of work sponsored by the United States Government. Neither the United States nor the United States Department of Energy, nor any of their employees, nor any of their contractors, subcontractors, or their employees, makes any warranty, expressed or implied, or assumes any legal liability or responsibility for the accuracy, completeness or usefulness of any information, apparatus, product or process disclosed, or represents that its use would not infringe privately owned rights.

ABSTRACT

CHILES 2 is a finite element computer program that calculates the strength of singularities in linear elastic bodies. A generalized quadrilateral finite element that includes a singular point at a corner node is incorporated in the code. The displacement formulation is used and interelement compatibility is maintained so that monotone convergence is preserved. Plane stress, plane strain and axisymmetric conditions are treated. Isotropic and orthotropic crack tip singularity problems are solved by this version of the code but any type of singularity may be properly modeled by modifying selected sub-routines in the program.

TABLE OF CONTENTS

1.	Introduction	6-7
2.	Mathematical Modeling of Singularities	9
3.	Numerical Modeling	14
	3.1 Element Formulation	14
	3.2 Global Equations	19
4.	Operating Instructions	19
	4.1 Input Cards	20
	4.2 Program Capabilities and Limitations	29
	4.3 Precautions Considering Fracture Mechanics and CHILES 2	30
5.	Programming Information	30
	5.1 Subroutine Operations	30
	5.2 Definition of Variables	32
	5.3 Read/Write Statements for Pre- and Post-Processors	34
6.	Sample Problems	36
	6.1 Tensile Strip with Side Crack	36
	6.2 Circumferentially Cracked Round Bar Subject to Tension	36
	6.3 Compact Tension Specimen	42
	6.4 45° Crack in Finite Width Strip	42
	6.5 Central Crack in an Orthotropic Plate	47
	References	49

1. Introduction

Mathematical modeling of linear elastic structures often gives solutions that have stress and strain singularities at particular points in the body. Specifically these points are crack tips, re-entrant corners, or other non-continuous loading and boundary configurations. These features are of major concern to the stress analyst because they can lead to catastrophic structural failure. Critical components in nuclear reactors, aircraft and space vehicles, nuclear waste storage vessels, and turbine generators, to mention a few, must be designed against any such failure.

Several different analysis techniques have been used to gain solutions to problems that have singularities. An excellent compilation of closed form solutions to elastic crack tip problems is given by Sneddon^{(1)*}. Although these solutions may be considered "exact," they are severely limited to the most simple loading and geometry conditions. Sih⁽²⁾ gives several techniques, both analytical and numerical, for solving crack tip problems, and Tada⁽³⁾ has compiled a handbook cataloging most of the crack tip geometries that have been solved either exactly or approximately. Again, most of these methods are restricted to very simple geometrical and loading conditions.

The finite element technique is a very versatile method for mathematically analyzing arbitrarily shaped structures with complicated boundary conditions. A reader unfamiliar with finite elements should consult Zienkiewicz⁽⁴⁾ for a complete treatment of the subject. A survey of this technique as applied to the crack tip singularity problem has been presented by Gallagher.⁽⁵⁾ Essentially, the finite element method may be used in the following ways to characterize singularities.

1. Direct Method - Conventional elements are used by defining the region near the singularity with extremely small elements. This is obviously a very costly and inefficient procedure.

* Superscripts refer to references listed at the end of this report.

2. Energy Release Method - Conventional elements are used by defining the singular region with relatively small elements. The energy of the system is then monitored as the crack length grows (i.e., several static solutions with increasing crack lengths are run). This method proves to be more accurate than 1. and requires significantly fewer elements but remains awkward and time-consuming by requiring the solution to several similar problems. A modification of this method for three-dimensional problems given by Parks⁽⁶⁾ appears encouraging. One drawback of this method is its inability to distinguish between Mode I and Mode II* intensities.
3. Superposition Methods - Conventional elements with coarse grids are used in conjunction with a classical solution. The method is attractive but somewhat cumbersome.
4. Singularity Function Formulations - A special element is formulated that contains the proper singularity. This procedure requires the formulation of a complicated element stiffness matrix but can be coupled easily to conventional elements. Coarse meshing gives very accurate results.

The CHILES 2 computer program is a two-dimensional solid finite element code that uses a singularity function formulation to model both isotropic and orthotropic elastic bodies with singular points. This version of CHILES 2 contains subroutines that calculate the state of strain at the tip of a crack in either plane stress, plane strain or axisymmetric geometries. These subroutines can be modified by the user to calculate the state of any singularity of the form given in the following section of this report (i.e., Equation 11.) Linear isotropic stress-strain material properties are used and small strain theory is assumed. Only axisymmetrical mechanical loadings are accepted by the code.

* See Figure 2 for the definitions of Modes I and II.

2. Mathematical Modeling of Singularities

2.1 Isotropic Materials

Figure 1 shows the local coordinate system used to define the neighborhood of a crack tip. The singular elastic stress field in the vicinity of the tip has the form⁽¹⁰⁾

$$\begin{pmatrix} \sigma_{r'} \\ \sigma_{z'} \\ \sigma_{rz'} \end{pmatrix} = \frac{K_I}{(2\pi\rho)^{1/2}} \begin{bmatrix} \cos \frac{\theta}{2} \left(1 - \sin \frac{\theta}{2} \sin \frac{3\theta}{2} \right) \\ \cos \frac{\theta}{2} \left(1 + \sin \frac{\theta}{2} \sin \frac{3\theta}{2} \right) \\ \cos \frac{\theta}{2} \sin \frac{\theta}{2} \cos \frac{3\theta}{2} \end{bmatrix} + \frac{K_{II}}{(2\pi\rho)^{1/2}} \begin{bmatrix} -\sin \frac{\theta}{2} \left(2 + \cos \frac{\theta}{2} \cos \frac{3\theta}{2} \right) \\ \sin \frac{\theta}{2} \cos \frac{\theta}{2} \cos \frac{3\theta}{2} \\ \cos \frac{\theta}{2} \left(1 - \sin \frac{\theta}{2} \sin \frac{3\theta}{2} \right) \end{bmatrix} \quad (1)$$

the corresponding displacement field is

$$\begin{pmatrix} u_{r'} \\ u_{z'} \end{pmatrix} = \frac{K_I(2\rho)^{1/2}}{8G(\pi)^{1/2}} \begin{bmatrix} (2\kappa - 1) \cos \frac{\theta}{2} - \cos \frac{3\theta}{2} \\ (2\kappa + 1) \sin \frac{\theta}{2} - \sin \frac{3\theta}{2} \end{bmatrix} + \frac{K_{II}(2\rho)^{1/2}}{8G(\pi)^{1/2}} \begin{bmatrix} (2\kappa + 3) \sin \frac{\theta}{2} + \sin \frac{3\theta}{2} \\ -(2\kappa - 3) \cos \frac{\theta}{2} - \cos \frac{3\theta}{2} \end{bmatrix} \quad (2)$$

where

- K_I, K_{II} = stress intensity factors
- σ_{ij} = stresses $i = 1, 2$ (i.e., $\Rightarrow r', z'$)
- u_i = displacements
- G = Elastic shear modulus

$\kappa = (3 - 4\nu)$ for plane strain or axisymmetric

$= (3 - \nu)/(1 + \nu)$ for plane stress

$\nu =$ Poisson's Ratio

$\rho, \theta =$ polar coordinates as shown in Figure 1

2.2 Orthotropic Materials

The singular elastic stress field in the vicinity of a tip of a crack in an orthotropic material has the form (13)

$$\begin{pmatrix} \sigma_r \\ \sigma_z \\ \sigma_{rz} \end{pmatrix} = \frac{K_I}{(2\pi\rho)^{1/2}} \begin{pmatrix} \text{Re} \left\{ \frac{S_1 S_2}{S_1 - S_2} \left(\frac{S_2}{(\cos \theta + S_2 \sin \theta)^{1/2}} - \frac{S_1}{(\cos \theta + S_1 \sin \theta)^{1/2}} \right) \right\} \\ \text{Re} \left\{ \frac{1}{S_1 - S_2} \left(\frac{S_1}{(\cos \theta + S_2 \sin \theta)^{1/2}} - \frac{S_2}{(\cos \theta + S_1 \sin \theta)^{1/2}} \right) \right\} \\ \text{Re} \left\{ \frac{S_1 S_2}{S_1 - S_2} \left(\frac{1}{(\cos \theta + S_1 \sin \theta)^{1/2}} - \frac{1}{(\cos \theta + S_2 \sin \theta)^{1/2}} \right) \right\} \end{pmatrix} \\ + \frac{K_{II}}{(2\pi\rho)^{1/2}} \begin{pmatrix} \text{Re} \left\{ \frac{1}{S_1 - S_2} \left(\frac{S_2^2}{(\cos \theta + S_2 \sin \theta)^{1/2}} - \frac{S_1^2}{(\cos \theta + S_1 \sin \theta)^{1/2}} \right) \right\} \\ \text{Re} \left\{ \frac{1}{S_1 - S_2} \left(\frac{1}{(\cos \theta + S_2 \sin \theta)^{1/2}} - \frac{1}{(\cos \theta + S_1 \sin \theta)^{1/2}} \right) \right\} \\ \text{Re} \left\{ \frac{1}{S_1 - S_2} \left(\frac{S_1}{(\cos \theta + S_1 \sin \theta)^{1/2}} - \frac{S_2}{(\cos \theta + S_2 \sin \theta)^{1/2}} \right) \right\} \end{pmatrix} \quad (3)$$

The corresponding displacement field is

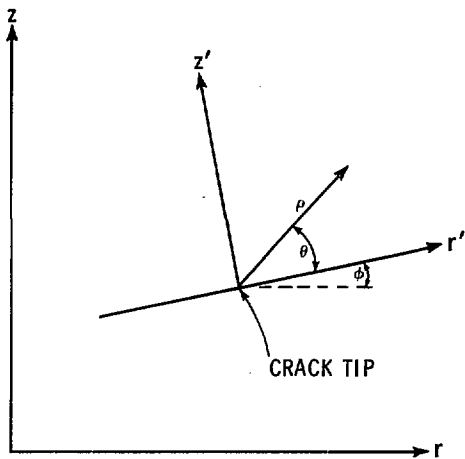


FIGURE 1. LOCAL COORDINATE SYSTEM FOR A CRACK TIP

$$\begin{pmatrix} u_r \\ u_z \end{pmatrix} = K_I \frac{(2\rho)^{\frac{1}{2}}}{(\pi)^{\frac{1}{2}}} \begin{bmatrix} R_e \left\{ \frac{1}{S_1 - S_2} [S_1 p_2 (\cos\theta + S_2 \sin\theta)^{\frac{1}{2}} - S_2 p_1 (\cos\theta + S_1 \sin\theta)^{\frac{1}{2}}] \right\} \\ R_e \left\{ \frac{1}{S_1 - S_2} [S_1 q_2 (\cos\theta + S_2 \sin\theta)^{\frac{1}{2}} - S_2 q_1 (\cos\theta + S_1 \sin\theta)^{\frac{1}{2}}] \right\} \end{bmatrix} \\ + K_{II} \frac{(2\rho)^{\frac{1}{2}}}{(\pi)^{\frac{1}{2}}} \begin{bmatrix} R_e \left\{ \frac{1}{S_1 - S_2} [p_2 (\cos\theta + S_2 \sin\theta)^{\frac{1}{2}} - p_1 (\cos\theta + S_1 \sin\theta)^{\frac{1}{2}}] \right\} \\ R_e \left\{ \frac{1}{S_1 - S_2} [q_2 (\cos\theta + S_2 \sin\theta)^{\frac{1}{2}} - q_1 (\cos\theta + S_1 \sin\theta)^{\frac{1}{2}}] \right\} \end{bmatrix} \quad (4)$$

where

K_I, K_{II} = stress intensity factors

σ_{ij} = stresses

u_i = displacements

ρ, θ = polar coordinates

S_1, S_2 = roots of

$$a_{11}S^4 - 2a_{16}S^3 + (2a_{12} + a_{66})S^2 - 2a_{26}S + a_{22} = 0 \quad (5)$$

The parameters p_1, p_2, q_1 and q_2 are defined as

$$p_1 = a_{11}S_1^2 + a_{12} - a_{16}S_1 \quad (6)$$

$$q_1 = (a_{12}S_1^2 + a_{22} - a_{26}S_1)/S_1 \quad (7)$$

$$p_2 = a_{11}S_2^2 + a_{12} - a_{16}S_2 \quad (8)$$

$$q_2 = (a_{12}S_2^2 + a_{22} - a_{26}S_2)/S_2 \quad (9)$$

The constants a_{11} , a_{12} , etc., are obtained from the generalized form of Hooke's law for an anisotropic material in a plane state of stress or strain:

$$\begin{Bmatrix} \epsilon_{r'} \\ \epsilon_{z'} \\ \gamma_{r'z'} \end{Bmatrix} = \begin{bmatrix} a_{11} & a_{12} & a_{16} \\ a_{21} & a_{22} & a_{26} \\ a_{61} & a_{62} & a_{66} \end{bmatrix} \begin{Bmatrix} \sigma_{r'} \\ \sigma_{z'} \\ \tau_{r'z'} \end{Bmatrix} \quad (10)$$

2.3 Generalized Formulation

The near field displacement Equations 2 and 4 may be generalized as

$$u_i = K_I Q_{Ii}(\rho, \theta, \dots) + K_{II} Q_{IIi}(\rho, \theta, \dots) \quad (11)$$

where the Q_{ki} 's are any generalized functions that define the correct displacement field in the region of the singularity. In the specific case of crack tip problems, K_I is the strength of the Mode I singularity and K_{II} is the strength of the Mode II singularity. These modes are depicted in Figure 2.

The displacements given by Equation 2 are "near field" terms that characterize the singularity only over a region close to the singular point. Consequently, an elastic body may be properly modeled with finite elements by defining the region near the singularity with elements enriched with the displacement field of Equation 11 and defining the remainder of the body with conventional elements. A rigorous treatment of this actual procedure is given by Hilton.⁽²⁾ Several special finite elements that incorporate "embedded singularities" have been developed.⁽⁵⁾ The one used in CHILES 2 was reported by Benzley⁽⁷⁾ and is described in the following section.

3.0 Numerical Modeling

3.1 Element Formulation

An arbitrary quadrilateral element with a singular point at node i is shown in Figure 3. The effects of the singularity are included in the element by "enriching" a bilinear element displacement assumption with terms that give the proper singularity at node i , i.e.,

$$u_i = \alpha_{i1} + \alpha_{i2}a + \alpha_{i4}ab + K_I Q_{1i}(\rho, \theta) + K_{II} Q_{2i}(\rho, \theta) \quad (12)$$

where

- $i = 1, 2$
- α_{ij} = unknown coefficients
- a, b = local, nondimensional oblique coordinates
- Q_{2i} = specific singular assumptions, see Equation 2.

Solving Equation 12 for the unknown coefficients α_{ij} 's in terms of the nodal displacements, \bar{u}_{ik} where $k = 1, 2, 3, 4$, the displacement assumption may be written as

$$u_i = \sum_{k=1}^4 f_k \bar{u}_{ik} + K_I \left(Q_{1i} - \sum_{k=1}^4 f_k \bar{Q}_{1ik} \right) + K_{II} \left(Q_{2i} - \sum_{k=1}^4 f_k \bar{Q}_{2ik} \right) \quad (13)$$

where

\bar{Q}_{2ik} = the value of Q_{2i} evaluated at node k

$$f_k = \frac{1}{4}(1 - aa_k)(1 - bb_k)$$

a_k, b_k = the coordinates of node k ($i \neq k$)

The terms enclosed in the parentheses in Equation 13 account for the singularity. It is desirable to let this singularity "live" only over a portion of the elastic body. The reasons for this are:

1. Typically, the Q_{2i} 's are "near field" terms.

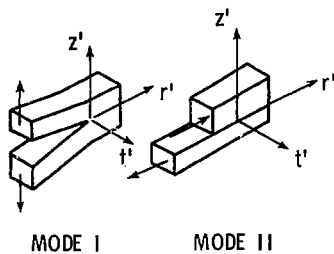


FIGURE 2. CRACK OPENING MODE I AND SLIDING MODE II

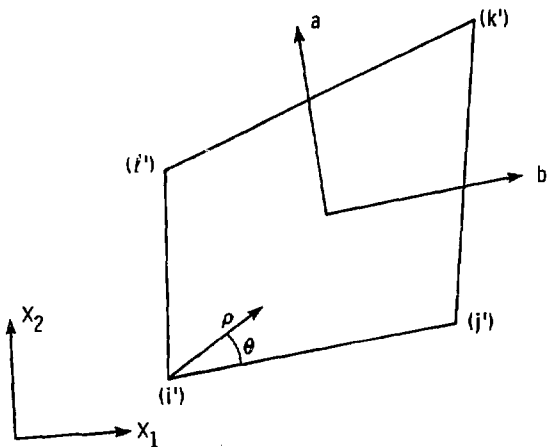


FIGURE 3. A QUADRILATERAL FINITE ELEMENT WITH A SINGULAR POINT AT NODE (i).

2. Away from the singular point the bilinear displacement assumptions are adequate to describe weaker strain gradients.
3. Some calculational savings are obtained by carrying the singularity only over a few elements in the neighborhood of the singularity.

It is obvious that an element with the displacement assumption of Equation 13 connected to an element with a bilinear displacement assumption, would produce a slight incompatibility between the adjoining elements. This incompatibility can be removed by altering the displacement assumption of Equation 13 to

$$u_i = \sum_{k=1}^4 f_k \bar{u}_{ik} + R(a,b) \left\{ K_I \left(Q_{1i} - \sum_{k=1}^4 f_k \bar{Q}_{1ik} \right) + K_{II} \left(Q_{2i} - \sum_{k=1}^4 f_k \bar{Q}_{2ik} \right) \right\} \quad (14)$$

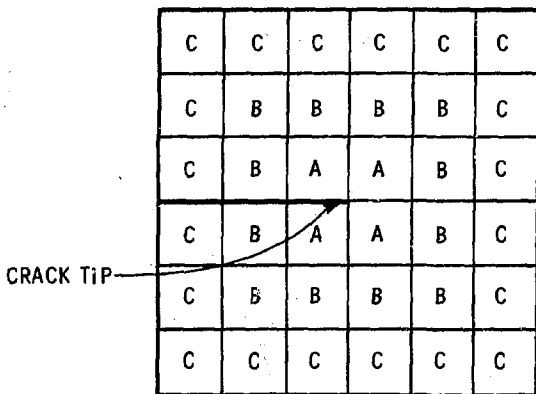
where $R(a,b)$ is chosen such that it equals 1 on boundaries adjacent to "enriched" elements and equals 0 on boundaries adjacent to "bilinear" elements. An example of an element configuration using the above procedure is shown in Figure 4. Using the assumption given by Equations 13 or 14 in a standard finite element formulation, the left hand side of the equilibrium equations for an element become:

$$\begin{bmatrix} k^{11} & | & k^{12} \\ \dots & & \dots \\ k^{21} & | & k^{22} \end{bmatrix} \begin{Bmatrix} \bar{u} \\ \dots \\ K_I \\ K_{II} \end{Bmatrix}$$

where

- \bar{u} = element nodal point displacements
- k^{11} = "regular" stiffness matrix
- k^{22} = stiffness matrix from the "enrichment" terms
- k^{12} = coupled stiffness matrix from regular and enrichment terms
- k^{21} = transpose (k^{12})

Integration of k^{12} and k^{22} must be done very accurately because they contain singular terms. For this work a high order (7x7) Gaussian



- A - Elements with displacement assumptions given by Eq. 5
- B - Elements with displacement assumptions given by Eq. 6
- C - Elements with bilinear displacement assumptions only

FIGURE 4. COMPATIBLE FINITE ELEMENT CONFIGURATION WITH ELEMENT TYPES A, B, AND C.

quadrature was used for elements of type A and B (see Figure 4).

3.2 Global Equations

The global set of equations is obtained by summing the contributions of each element in the total set of equations in the standard fashion, thus

$$\begin{bmatrix} K^{11} & | & K^{12} \\ \dots & & \dots \\ K^{21} & | & K^{22} \end{bmatrix} \cdot \begin{Bmatrix} \bar{u} \\ \dots \\ K_{I} \\ K_{II} \end{Bmatrix} = \begin{Bmatrix} F \\ \dots \\ \bar{F} \end{Bmatrix} \quad (15)$$

where

K^{11} = Global banded stiffness matrix as would exist if no singular terms were present

K^{12} = Global stiffness matrix from coupled regular and "enrichment" terms (a sparse matrix)

K^{22} = Global stiffness matrix from "enrichment" terms

K^{21} = Transpose (K^{12})

F = Load vector

\bar{F} = "Singular" load vector. This is a null vector if no A or B element is on a loaded boundary.

Note that Equation 15 has a banded portion (i.e., K^{11}) plus extra active columns (i.e., K^{12} and K^{22}). This set of equations is easily solved by including the "extra" columns in a Gaussian elimination procedure. Only a slight increase in the computer storage requirements comes with carrying K^{12} .

4. Operating Instructions

CHILES 2 has the option of either reading a tape that contains geometry information* or internally creating this information by a semi-automatic procedure. An output tape is written to provide

*QMESH⁽⁸⁾ is the pre-processor mesh generator currently in use with CHILES 2.

information for a plotting post-processor. The specific formats for writing the input tape by the pre-processor and reading the output by the post-processor are given in Section 5.

4.1 Input Cards

This section lists the formats and the variables of the data cards read by CHILES 2. Five blocks of cards are used. These blocks are denoted by Roman numerals in the list that follows.

I PROBLEM IDENTIFICATION CARDS

Card 1 (8A10) - Problem title

Col 1-80 Any information to be printed as the title of the problem

(Note: Do not use the word END starting in Column 1.

Card 2 (715) - Problem Parameter Card

Col 1-5 Number of Singular Points (NSP)(3 Max.)

6-10 Geometry Selector (NPAR)

1 for axisymmetric geometry

2 for plane stress geometry

3 for plane strain geometry

10-15 Number of Materials (NMAT)(10 Max.)

16-20 Not Used

21-25 Print Option (IPTCW)

0 outputs displacements stresses and strains

1 outputs R-Z coordinates, element connectivity table, displacements, stresses and strains

26-30 Stress Intensity Boundary Switch (KBSW)

0 - No boundary conditions on intensities

1 - Intensity boundary condition card read.

31-35 Geometry Input Switch (KGEOSW)

0 - Geometry information read from tape

1 - Block III read for geometry information

36-40 Boundary Condition Table Cards (NUMTB) - See Block IV

41-45 Orthotropic Material Switch (KORTSW)

0 - Isotropic Material

1 - Orthotropic Material

NOTE: If KORTSW = 1, the Sandia MATHLIB routine must be used.

-
11. MATERIAL IDENTIFICATION CARDS (one card for each material, 10 material maximum input in sequential order. Card 1a is used for isotropic materials, i.e., KORTSW = 0 and Card 1b is used for orthotropic materials, i.e., KORTSW = 1.)
-

Card 1a (2E10.3)

- Col 1-10 Young's modulus
11-20 Poisson's ratio
-

Card 1b (7E10.3)

- Col 1-10 E_{11} , Young's modulus in the first principle direction of material orthotropy.
11-20 ν_{12} , Poisson's ratio in 1-2 plane.
NOTE: $\nu_{12}/E_{11} = \nu_{21}/E_{22}$
21-30 E_{22} , Young's modulus in the second principle direction of material orthotropy.
31-40 E_{33} , Young's modulus normal to the plane of analysis.
41-50 ν_{31} , Poisson's ratio in 1-3 plane.
51-60 ν_{32} , Poisson's ratio in 2-3 plane.
61-70 θ , Angle principle axes of orthotropy make with respect to r-z coordinates.

III GEOMETRY DEFINITION CARDS (If KGEOSW = 0, these cards are omitted. KGEOSW is read on card 2 of Set I.)

Card 1 (4I5) Geometry Parameters

Col 1-5 Number of Elements (NEL)
6-10 Number of Nodes (NODES)
11-15 Number of Load Cards (NUMPC)

Card 2 (6I5) (N,IX-Array)

Col 1-5 Element number (N)
6-10 Ith nodal point (IX-Array)
11-15 Jth nodal point (IX-Array)
16-20 Kth nodal point (IX-Array)
21-25 Lth nodal point (IX-Array)
26-30 Material number (MAT, IX-Array)

Maximum difference between nodal point I.D. must be equal to or less than 26.

In general, every element must be defined; but with the semi-automatic mesh generation feature, a minimum of one element per row need be input. For example, if element 10 is read with values I=12, J=13, K=24, L=23, and MAT=1, and the next element read is element 15 with values I=23, J=24, K=35, L=34 and MAT=1, then element 11 would be assigned values 13, 14, 25, 24, and 1.

Card 3 (15, F5.0, 4F10.0)

Col 1-5 Nodal point number (N)
6-10 Boundary condition code (CODE)
11-20 Radial coordinate (R)
21-30 Axial coordinate (Z)
31-40 Radial force or displacement (XR)
41-50 Axial force or displacement (XZ)

In general, every nodal point boundary condition must be defined, but since the program has a semi-automatic mesh generation feature, a minimum of two nodal points per row need be input and the intervening points will be assigned coordinates based on a linear interpolation procedure. For example, if nodal point 1 is the first point in a row with coordinates (2.5, 5.4), and nodal point 11 is the next point defined with coordinates (12.5, 10.4), then nodal point 2 will be located at (3.5, 5.9), etc. The boundary condition CODE will be set 0 unless points 1 and 11 have the same CODE, in which case all intervening points will be assigned the same CODE as the two end points. The radial and axial forces or displacements will be set 0 in all cases.

The boundary condition code is interpreted in the following manner as given in Table 1.

<u>CODE</u>	<u>Radial (r)</u>	<u>Axial (z)</u>
0.	Force	Force
1.0	Displ	Force
2.0	Force	Displ
3.0	Displ	Displ

Table 1 - BOUNDARY CONDITION CODE DEFINITIONS

Card 4 (2I5,2F10.0) Load Cards

Col 1-5 Ith nodal point (IP)

6-10 Jth nodal point (JP)

11-20 Normal traction (PR)

21-30 Shear traction in R-Z plane (PT)

This subsection is required only if NUMPC > 0. Nodes I and J must be such that the body is on the left of the line going from I to J (Figure 5). This requires that I and J be in the same sequence on the traction cards as on the element cards. Positive tractions are indicated on the diagram in Figure 5, and the tractions are assumed constant over the length of the element boundary.

BOUNDARY CONDITION FLAG TABLE (NUMTB cards are read; these cards define a boundary condition code or traction option to be used with a boundary flag number as set in QMESH). Boundary condition codes are defined in Table I)

Card 1 (I10,3E10.0)

- Col. 1-10 IBC - Boundary flag number as set in QMESH
- 11-20 BCODE - Boundary code*
- 21-30 XR or PN value of displacement, force or normal traction
- 31-40 XZ or SH - Value of displacement, force or shear traction

*NOTE: If $BCODE \geq 0$, the boundary codes are defined in Table I and the 3rd and 4th fields of the card are normal forces or displacements. If $BCODE < 0$ the 3rd and 4th fields of this card are the normal and shear tractions applied on an element face. In this case, the sign convention is as shown in Figure 5.

V. SINGULAR POINTS IDENTIFICATION CARDS

Card 1 (3(I5,E10.3)) Node and Angle Data (If NSP = 0, omit this card. ϕ is defined on Figure 1.)

Col 1-5 Node number of singular point 1
6-15 Angle ϕ (degrees) for singular point 2
 $-180. \leq \phi \leq 180.$
16-20 Node number of singular point 2
21-30 Angle ϕ (degrees) for singular point 2
 $-180. \leq \phi \leq 180.$
31-45 Node number of singular point 3
36-45 Angle (degrees) for singular point 3
 $-180. \leq \phi \leq 180.$

Card 2 (3I5) Boundary code for singular points (If KBSK = 0, omit this card.)

Col. 1-5 KODE for point 1
6-10 KODE for point 2
11-15 KODE for point 3
KODE = 1 K_I set equal to 0.0
KODE = 2 K_{II} set equal to 0.0

This set of data cards must be followed with a card that has END beginning in column 1.

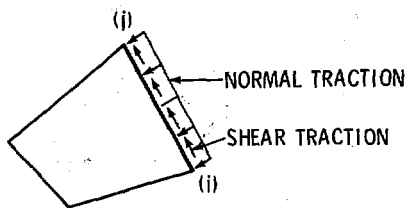


FIGURE 5. PRESSURE BOUNDARY CONDITION

4.2 Program Capabilities and Limitations

This section of the report lists the capabilities and limitations of CHILES that have resulted because of theoretical restrictions, economic consideration, computer storage capacity, etc.

1. CHILES performs a linear elastic stress analyses of any two-dimensional body in a plane stress, plane strain or axisymmetric state. Singular points are treated with enriched finite elements.
2. Up to three singular nodes may be defined in the body.
3. 1000 nodal points may be used.
4. 1000 elements may be used.
5. Bandwidth is limited to 54 (i.e., difference between node numbers in any one element must be < 27).
6. Only mechanical loads are accepted.
7. A pre-created mesh and boundary condition scheme may be read from tape.
8. Displacements, stresses and strains are output on tape for plotting.
9. CHILES automatically surrounds a singularity with type A and type B elements as shown in Figure 4.
10. Small strains are assumed, a condition that is violated at the crack tip.
11. Up to 10 different materials can be defined.
12. Special elements are compatible with conventional elements.
13. Users may replace subroutines (CALQ and CALQI) to model singularities other than crack tips.

4.3 Precautions Considering Fracture Mechanics and CHILES 2

Certain precautions must be taken when applying fracture mechanics considerations to stress analysis. In linear elastic fracture mechanics (LEFM) a crack in a brittle material becomes unstable as K_I reaches K_{IC} where K_{IC} is the critical stress intensity factor. The same holds true for K_{II} and K_{IIc} . The critical stress intensity factor, a material property, can be measured in the laboratory for "brittle" materials with accuracy of about 5%. CHILES calculations are well within this accuracy. Care must be exercised, however, when using critical stress intensity factors for materials that are not "brittle." A plastic zone always exists at the tip of the crack and as long as this remains "small" LEFM applies. As this zone increases in size, LEFM breaks down and "critical stress intensity factors" are not useful. Also, mixed mode fracture criteria are not well established. (2) The stress analyst is cautioned to consult with a fracture mechanician before applying CHILES to problems where the material may not be brittle and/or mixed mode conditions are present.

5. Programming Information

The overall flow diagram for CHILES 2 is shown in Figure 6.

5.1 Subroutine Operations

The following is a brief description of the operation of the main program and each subroutine.

1. CHILES is the main program that controls the logic for the code. It performs the initial data reads, calls subroutines that form global matrices and solves the equations, and runs the output logic. A flow chart is shown in Figure 7.
2. ADSING is the subroutine that adds the "singular" rows and columns to the element stiffness matrix (i.e., k^{12} , k^{21} , k^{22} .)
3. CALB11 calculates the "B" matrix used in forming k^{11} (i.e., $k^{11} = \int B^{11T} D B^{11} dv$) where B^{11} is the matrix relating element strain to nodal point displacements and D is the matrix relating strain to stresses. (4)

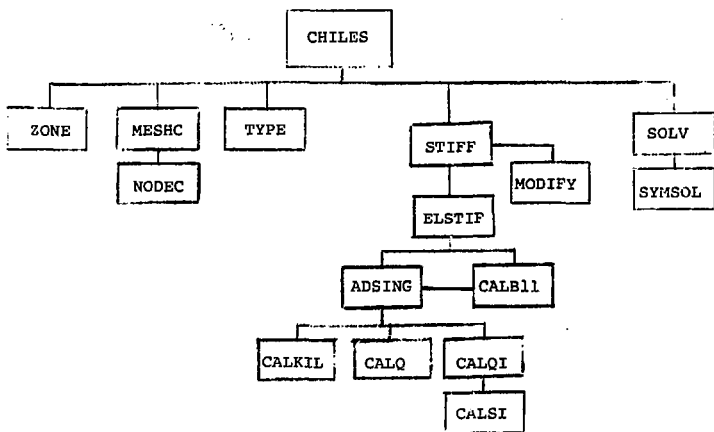


FIGURE 6. FLOW DIAGRAM FOR CHILES

4. CALKIL calculates the kill function, R, used in eliminating the incompatibility in element type B.
5. CALQ calculates the "Q" terms used in formulating the singular part of the element stiffness matrix.
6. CALQI calculates the \bar{Q} terms used in formulating the singular part of the element stiffness matrix.
7. CALSI is a routine that calculates the sign of an angle between two vectors.
8. ELSTIF controls the calculation of the element stiffness matrix, computes element stress and strain matrices, and stores them on tape.
9. MODIFY sets the boundary conditions in the global stiffness matrix.
10. CALG computes the coefficients defined in Equations 5 and 6.
11. ROOTS computes the complex roots of the orthotropic material's characteristic equation.
12. SOLVE reduces the global set of equations block by block, performs the back substitution, and writes the displacement solution.
13. STIFF is the subroutine that controls the forming of the global stiffness matrix and loading in blocks. It also modifies the global matrices by incorporating the boundary conditions.
14. SYMSOL solves a small set of banded equations.
15. TYPE is a subroutine that identifies the element types and catalogs the different singular points.
16. ZONE reads meshing cards internally and creates a mesh. Table 2 lists many of the main variables of the code and identifies their function.

5.2 Definition of Variables

<u>FORTRAN NAME</u>	<u>DESCRIPTION</u>
BETA (1000)	Identifies the element type (i.e., 1, 2, 3, 4 = singular element type A with singularity at node 1, 2, 3, 4; 5 = element type "B"; 6 = conventional element type C.)
CODE (1000)	Boundary condition code

D(4,4,10)	Material properties array
ES(4)	Stress array
EST(4)	Strain array
HED(8)	Problem title array
ICODE(50)	Boundary array indicator
INODE(50)	Boundary point array
IP(200)	INODE for pressure load
IPTSW	Printed switch
IS(200)	INODE for shear load
ISMAT(6)	Singular point material type
ISP(1000)	Singular region identifier for element
IX(5,1000)	Element connectivity and material
JNODE(50)	Boundary array indicator
JP(200)	J node for pressure load
JS(200)	J node for shear load
KBSW	Stress intensity boundary switch
KGEOSW	MESH generation switch
KODE(3)	Intensity boundary type
MBAND	Bandwidth
NODF	2 * NEL
NEL	Number of elements
NFORCE	Number of pressure load cards
NMAT	Number of materials
NMESH	Number of mesh change cards
NODES	Number of nodal points
NPAR	Geometry indicator
NRN(10)	Row number array
NSINN(3)	Nodal point of singularity 1, 2, and 3
NSP	Number of singular points
NST	Number of singular terms
NUMSC	Number of shear cards
PHI(3)	Angle ϕ for singular point 1, 2, and 3
PR(200)	Pressure at node
PRI(50)	Loading arrays
PRJ(50)	Loading arrays

PZI(50)	Loading arrays
PXJ(50)	Loading arrays
R(1000)	R coordinate array
RC(3)	R coordinate of singular points
SH(200)	Shear loading array
SIG(4,1000)	Stress array output on tape
ST(4,10)	Element strain matrix
U(10)	Element displacement array
XBM(108,54)	One block of banded stiffness matrix
XBT(6)	Intensity "loads"
XC(108,6)	Coupled conventional - singular global stiffnesses (in block form), K^{12}
XCT(6,6)	Singular part of global stiffness matrix, K^{22}
XF(108)	Blocked loading vector
XK(10,10)	Element stiffness matrix
XR(1000)	Nodal load or displacement R component
XZ(1000)	Nodal load or displacement Z component
Z(1000)	Z coordinate array
ZC(3)	Z coordinate of singular point

Table 2. Names of Variables used in CHILES 2

5.3 Read/Write Statements for Pre- and Post-Processors

CHILES reads TAPE 9 for a pre-created meshing scheme, loading and boundary conditions. The read statements for this information as they exist in CHILES are compatible with QMESH and RENUM.⁽⁸⁾ The variable definitions are given in Section 5.2.

TAPF 10 is written to be used in a post-processor for plotting. This tape is written as follows:

```
WRITE(10)      HED, NEL, NODES
WRITE(10)      TDUM
WRITE(10)      (XF(I),I=1,NDOF), (XF(I),I=1,NDOF), (XF(I),NDOF)
WRITE(10)      ((SIG(I,J),J=1,NEL),I=1,4)
WRITE(10)      ((STRN(I,J),J=1,NEL),I=1,4)
```

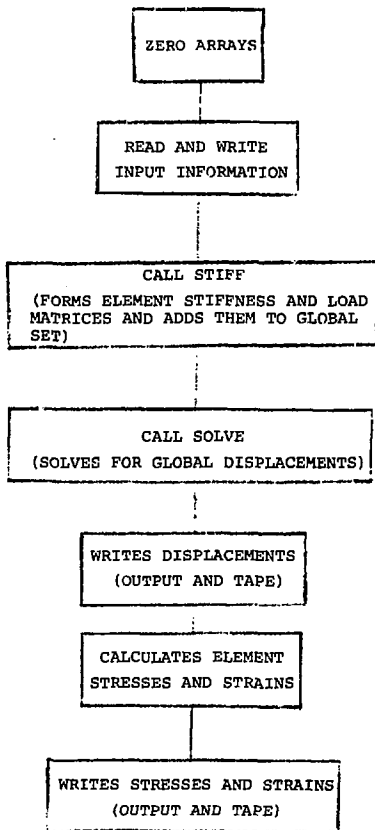



FIGURE 7. FLOW CHART FOR CHILES

The first record contains the problem title, number of elements and number of nodes, respectively. The second record is a dummy time output. The third record lists the displacements and two dummy lists. Records four and five have the element stresses and strains, respectively.

6. Sample Problems

Four example problems are given. The first two problems compare the results obtained from CHILES with other solutions. The third is an example of a very practical problem, the compact tension specimen commonly used in fracture mechanics tests. The final problem presented is to be used as a test problem in that it exercises most of the features of the code. In all examples, elements of type A exist at the crack tip, elements of type B are their next neighbors and elements of type C define the remainder of the problem. This "typing" is done automatically in CHILES 2.

6.1 Tensile Strip with Side Crack

A closed form solution to this problem was done by Keer and Freedman.⁽⁹⁾ The finite element definition of the problem is shown in Figure 8. Figure 9 gives a comparison between the finite element solution and the closed form solution for the stress intensity factor K_I as a function of crack length. Figure 10 compares the opening displacement of the crack tip of the two solutions and Figure 11 shows a typical crack opening shape.

6.2 Circumferentially Cracked Round Bar Subjected to Tension

This problem demonstrates the solution of an axisymmetric solid with CHILES. The mesh used was similar to that shown in Figure 8 with 10 elements in the radial direction but 13 in the axial direction. The radius of the bar is 1.0 in. and the half length is 3.0 in. A plot comparing the CHILES solution with two other approximate solutions⁽¹⁰⁾ is given in Figure 12.

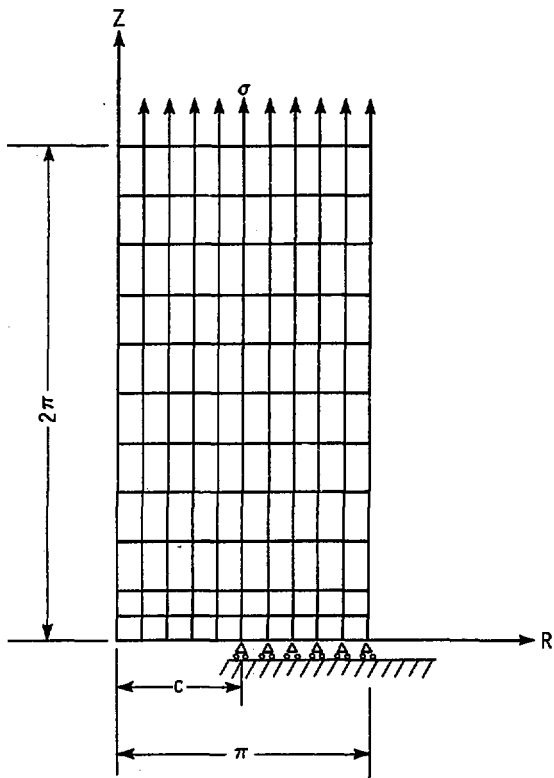


FIGURE 8. FINITE ELEMENT DEFINITION OF SIDE CRACKED PANEL.

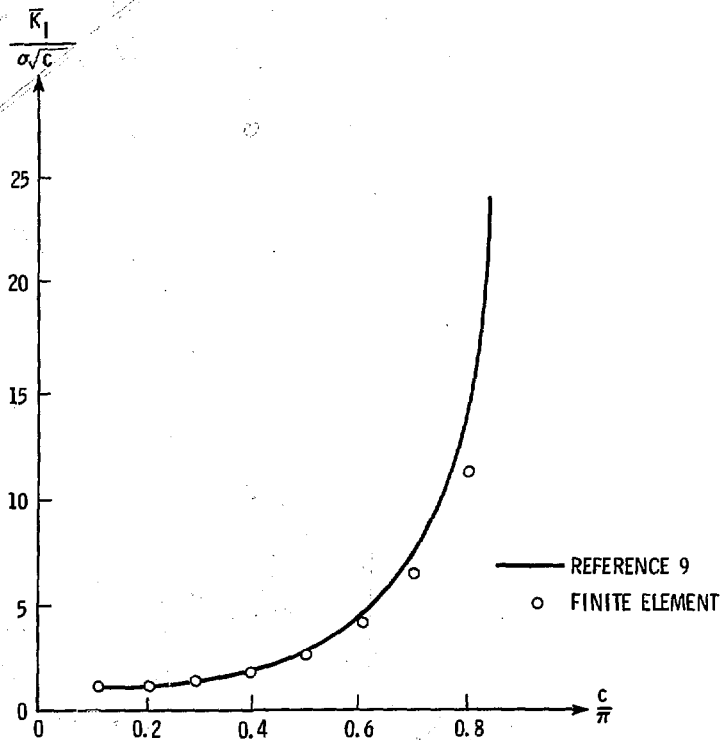


FIGURE 9. STRESS INTENSITY FACTOR AS A FUNCTION OF CRACK LENGTH

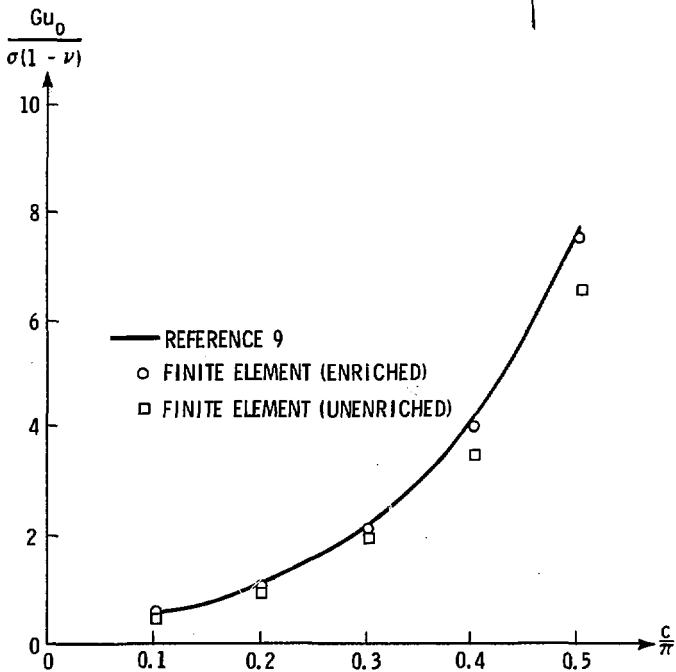


FIGURE 10. CRACK MOUTH OPENING AS A FUNCTION OF CRACK LENGTH.

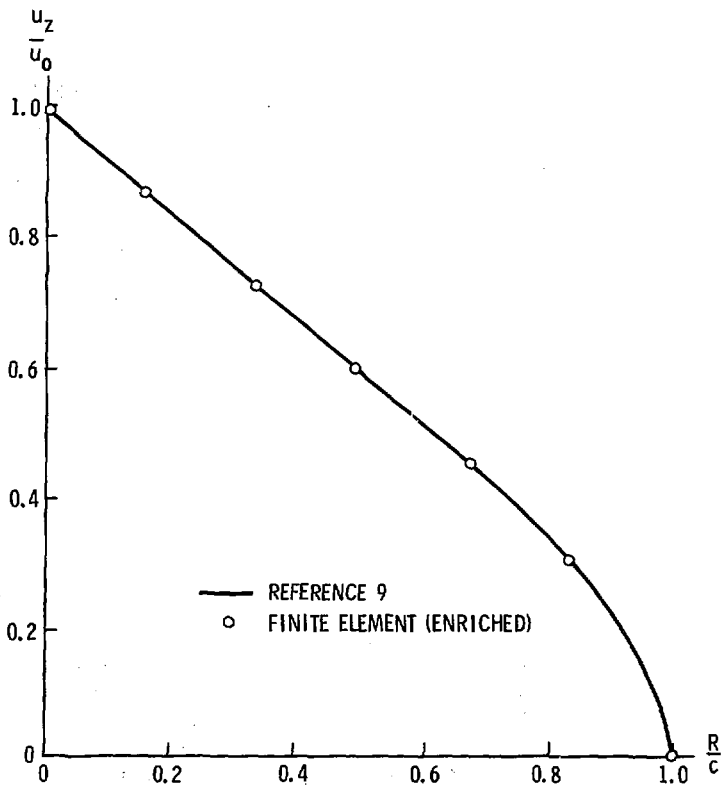


FIGURE 11 NON-DIMENSIONALIZED CRACK OPENING SHAPE FOR $c/n = .6$

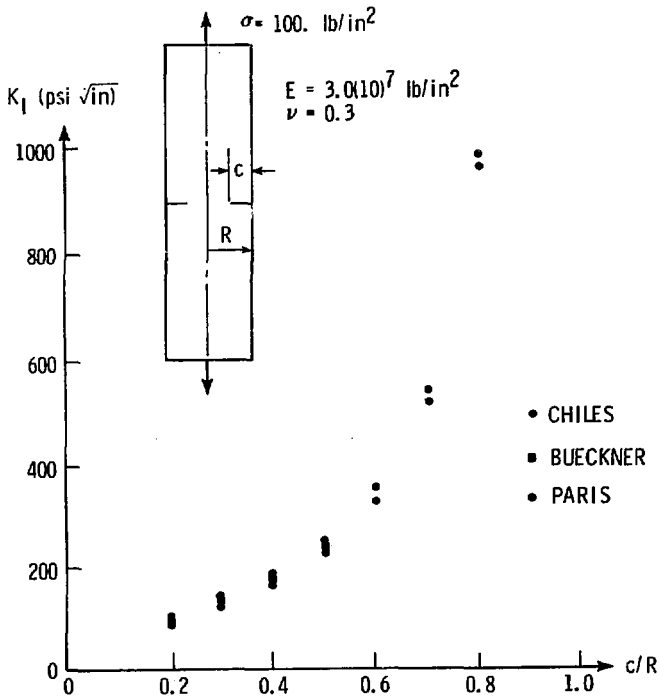


FIGURE 12. COMPARISON OF STRESS INTENSITY FACTOR, K_I , VS CRACK LENGTH FOR CIRCUMFERENTIALLY CRACKED ROUND BAR.

6.3 Compact Tension Specimen

The mesh used for a compact tension specimen is shown in Figure 13. The geometry condition was assumed to be plane stress and a concentrated load of 8000 lbs. was applied at the top of the cutout hole as shown. The results compared to a boundary collocation analysis are given in Table 2. An example of a stress contour plot is given in Figure 14.

	Stress Intensity K_I (psi $\sqrt{\text{in}}$)	Crack Opening at Load Line (in)
CHILES 2	54,216	.00948
Roberts (11)	54,306.	.00992

Table 3. Comparison of Finite Element and Boundary Collocation Results for Compact Tension Specimen

6.4 45° Crack in Finite Width Strip

This problem shows the versatility of CHILES 2 in that

1. Multiple singular points are defined.
2. Elements are skewed at the crack tip.
4. Mode I and Mode II intensities are present.

The meshing scheme and material properties are shown in Figure 15. The listing of the CHILES input cards is given in Figure 16. The results obtained from the code are compared to the solution of an infinite plate with a 45° crack⁽¹⁰⁾ in Table 3.

Singular Point	K_I	K_{II}	Infinite Plate Solution ($K_I = K_{II}$)
1	75.7	77.5	73.0
2	75.5	76.9	73.0

Table 4. Comparison of Stress Intensity Factors of Plate with 45° Crack.

$P = 8000 \text{ lb.}$
 $E = 3.0 \times 10^7 \text{ lb/in}^2$
 $\nu = 0.3$

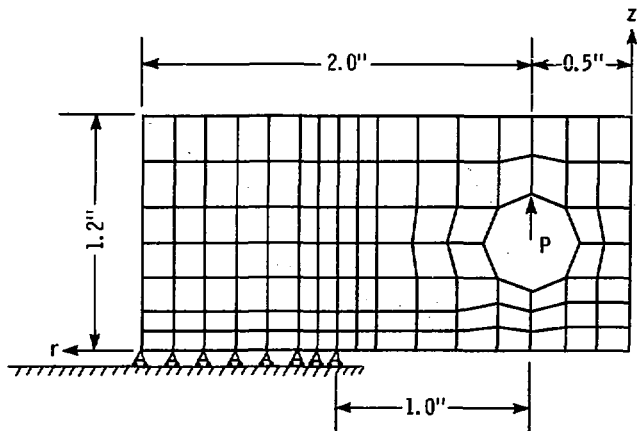


FIGURE 13. PROBLEM DESCRIPTION FOR COMPACT TENSION SPECIMEN

COMPACT TENSION SPECIMEN PLANE STRESS

EPS MAX

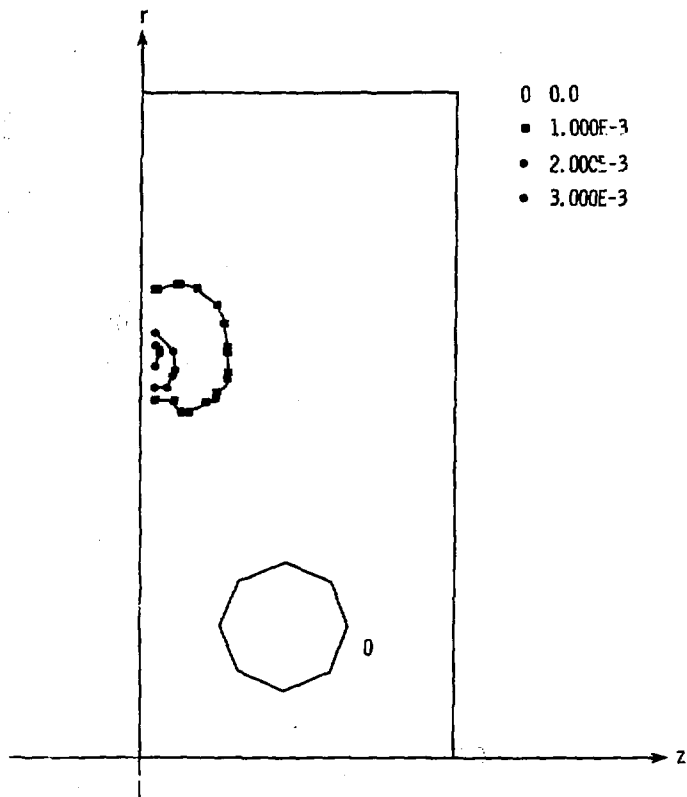
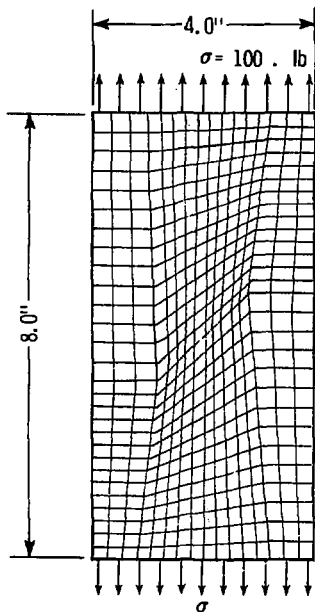
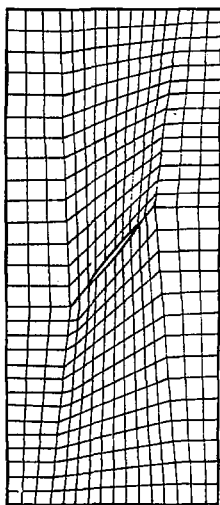


FIGURE 14. MAXIMUM STRAIN CONTOUR PLOT FOR COMPACT TENSION SPECIMEN



UNDEFORMED SHAPE



DEFORMED SHAPE

CRACK LENGTH ($2a$) = 1.358"

$E = 3.0(10)^7 \text{ lb/in}^2$

$\nu = 0.3$

FIGURE 15. GEOMETRY FOR 45° CRACKED PLATE IN TENSION

45 DEGREE CRACKED PLATE IN TENSION

2	3	1	
3.0E7			
244	-135.	250	45.0
100	2		
200	-1	100	
300	1		
400	1		

END OF DATA

FIGURE 16. LISTING OF INPUT CARDS FOR CHILES
FOR 45° CRACKED PLANE IN TENSION

6.5 Central Crack in an Orthotropic Plate

The plane problem of a central crack in a rectangular sheet of orthotropic material has been analyzed by Bowie and Freese⁽¹³⁾ by a "modified mapping collocation" technique. The problem solved by Bowie is defined in Figure 17. A comparison between some of the results obtained by Bowie and Freese are compared to CHILES 2 calculations in Table 5. The input cards required for both QMESH and CHILES 2 are shown in Figure 18. The finite element grid was a regular 10 x 10 quadrilateral mesh.

β_1^2	β_2^2	E_1	E_2	ν_{12}	u_{12}	K_{BOWIE}	$K_{\text{CHILES 2}}$
1	.1	1	10	0	.909090	231.86	232.37
1	.3	1	3.33	0	.769231	196.77	202.17
1	.5	1	2.0	0	.6666666	182.98	187.67
1	.7	1	1.4286	0	.588235	174.21	179.17
1	.9	1	1.1111	0	.52632	169.20	173.61
1	1.1	1	.90909	0	.47619	165.44	169.72
1	1.5	1	.6667	0	.4	160.42	164.65
1	2.5	1	.4	0	.28571	155.41	158.66
1	3.5	1	.2857	0	.22222	152.90	156.00
1	4.5	1	.2222	0	.1818181	150.40	154.49

Table 5. Comparison of Stress Intensity Factors Between CHILES 2 and Reference 13 for $L/b = .5$, $C/b = 1$. $\beta_1\beta_2 = (E_1/E_2)^{1/2}$ and $\beta_1 + \beta_2 = \sqrt{2} \{ (E_1/E_2)^{1/2} + E_1/2u_{12} - \nu_{12} \}^{1/2}$.

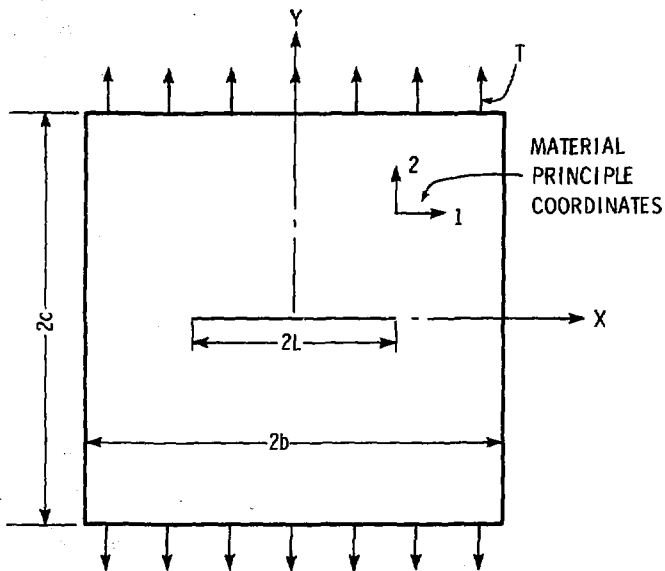


FIGURE 17. GEOMETRY OF CENTRAL CRACK IN AN ORTHOTROPIC PLATE

REFERENCES

1. Sneddon, I. N. and Lowengrum, M., Crack Problems in the Classical Theory of Elasticity, SIAM ser in Appl. Math., Wiley, 1969.
2. Sih, G. C., ed., Methods of Analysis and Solutions of Crack Problems, Hech. of Fract., Vol. 1, Noordoff, 1973.
3. Tada, H., The Stress Analysis of Cracks Handbook, Del Research Corp., Hellertown, Pa., 1973
4. Zienkiewicz, O. C., The Finite Element Method in Engineering Science, McGraw-Hill, London, 1971.
5. Gallagher, R. H., "Survey and Evaluation of the Finite Element Method in Fracture Mechanics Analysis," 1st Int. Conf. in Reactor Tech., Berlin, Sept. 1971.
6. Parks, D. M., "A Stiffness Derivative Finite Element Technique for Determination of Elastic Crack Tip Stress Intensity Factors," Tech Rept. NASA NGL 40-002-000/13, May 1973, (Taken from M.S. Thesis, Brown Univ., 1973).
7. Benzley, S. E., "Representation of Singularities with Isoparametric Finite Elements," Int. J. Num. Meth. Eng., Vol. 8, No. 3, 1974.
8. Jones, R. E., "Users Manual for QMESH, A Self-Organizing MESH Generation Program," SLA-74-0239, Sandia Laboratories, Albuquerque, NM, July 1974.
9. Keer, L. M., and Freedman, J. M., "Tensile Strip with Edge Cracks," Int. J. of Eng. Sci. (to be published).
10. Paris, P. C. and Sih, G. C., "Stress Analysis of Crack," Fracture Toughness Testing and Its Application, ASTM STP #381 (1965) pp. 30-83.
11. Roberts, E., "Elastic Crack-Edge Displacements for the Compact Tension Specimen," Matrl. Res. and Standards, 9, #2 (1969) p. 27.
12. Jones, R. M. and Crose, J. G., SAASII, Finite Element Stress Analysis of Axisymmetric Solids with Orthotropic, Temperature-Dependent Material Properties, Aerospace Report No. TR-0200(54980)-1, Aerospace Corporation, San Bernardino, CA (Sept. 1968).
13. Bowie, O. L., and Freese, C. E., "Central Crack in Plane Orthotropic Rectangular Sheet," Int. J. of Fr. Mech., Vol. 8, No. 1, March, 1972.