PARAMETRIC STUDY OF A COMPACT SHEAR FRACTURE SPECIMEN BY FINITE ELEMENT TECHNIQUES

Daniel Hoyniak

Technical Memorandum
File No. TM 76-196
June 14, 1976
Contract No. N00017-73-C-1418

Copy No. 5

The Pennsylvania State University
Institute for Science and Engineering
APPLIED RESEARCH LABORATORY
Post Office Box 30
State College, PA 16801

APPROVED FOR PUBLIC RELEASE
DISTRIBUTION UNLIMITED

NAVY DEPARTMENT
NAVAL SEA SYSTEMS COMMAND
**Title:** Parametric Study of a Compact Shear Fracture Specimen by Finite Element Techniques

**Author(s):** Daniel Hohniak

**Performing Organization Name and Address:** The Pennsylvania State University, Applied Research Laboratory, P. O. Box 30, State College, PA 16801 Unit Park

**Date:** June 1976

**Number of Pages:** 95 pages & figures

**Distribution Statement (of this Report):** Approved for public release, distribution unlimited, per NSSC (Naval Sea Systems Command), 7/15/76

**Abstract:** A compact shear fracture specimen has been proposed for the study of the edge sliding mode, Mode II, of crack propagation. The purpose of this investigation was to numerically determine the effect that crack length, specimen geometry, and applied load, has on the Mode II stress intensity factors for this specimen. Numerical results were compared to those obtained independently by another investigator using boundary collocation and photoelastic techniques. In addition, the initial angle of propagation was numerically determined for...
20. ABSTRACT (continued)

selected specimen configurations and crack lengths. The Mode II stress intensity factors were generated by using displacement and strain energy release rate methods. The initial angle of propagation was obtained by the strain energy density technique.

The optimum specimen configuration established during this investigation has a tang width $H = 1.0$ inch. This model configuration exhibited a relatively constant nondimensionalized Mode II stress intensity factor for varying crack lengths. For a tang width of 0.5 inch, the nondimensionalized Mode II stress intensity factors were found to be load dependent for short crack lengths. For a tang width $H = 1.5$ inch, the nondimensionalized Mode II stress intensity factors were influenced by bending of the specimen over the entire range of crack lengths studied. Boundary effects were also noticed, for all specimen configurations studied, as the crack tip approached the upper and lower boundaries of the specimen. The numerically determined initial angle of propagation was found to be $77^\circ$ for the specimen configurations and crack lengths selected to be investigated. This study also showed the usefulness of the finite element method in determining the Mode II stress intensity factors for different loading conditions and model geometries.
ACKNOWLEDGEMENTS

The author expresses sincere gratitude to Dr. Joseph C. Conway, Associate Professor of Engineering Mechanics, for his encouragement and assistance in the research and preparation of this thesis. Appreciation is also extended to Mr. Hiroshi Honda for his assistance in the programming of the strain energy subroutine, and to the Applied Research Laboratory of The Pennsylvania State University for use of their facilities and support for this research under contract with the Naval Sea Systems Command.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACKNOWLEDGEMENTS</td>
<td>i1</td>
</tr>
<tr>
<td>LIST OF TABLES</td>
<td>v</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>vi</td>
</tr>
</tbody>
</table>

## I. INTRODUCTION TO PROBLEM

1.1 General Introduction                                               | 1    |
1.2 Purpose of Investigation                                           | 3    |
1.3 Scope of Investigation                                             | 3    |
1.4 Approach Used in the Investigation                                 | 5    |

## II. FINITE ELEMENT TECHNIQUES APPLIED TO LINEAR ELASTIC FRACTURE MECHANICS

2.1 Introduction                                                       | 6    |
2.2 Displacement Method                                                | 6    |
2.3 Strain Energy Release Rate                                         | 9    |
2.4 Strain Energy Density                                              | 12   |

## III. MODEL USED IN THE INVESTIGATION AND VERIFICATION OF NUMERICAL PROCEDURE

3.1 Introduction                                                       | 14   |
3.2 Grid Pattern                                                       | 14   |
3.3 Model Geometry                                                     | 16   |
3.4 Load and Boundary Conditions                                       | 18   |
3.5 Verification of Gridding Program and Numerical Techniques          | 18   |

## IV. ANALYSIS OF RESULTS

4.1 Introduction                                                       | 24   |
4.2 Stress Intensity Factors                                           | 24   |
4.3 Stress Boundary Conditions                                         | 30   |
4.4 Initial Angle of Propagation                                       | 41   |

## V. CONCLUSIONS AND RECOMMENDATIONS

5.1 Conclusions                                                        | 44   |
5.2 Recommendations                                                    | 46   |
<table>
<thead>
<tr>
<th>BIBLIOGRAPHY</th>
<th>47</th>
</tr>
</thead>
<tbody>
<tr>
<td>APPENDIX A: Finite Element Program Used in the Investigation</td>
<td>48</td>
</tr>
<tr>
<td>APPENDIX B: Closed Form Solution</td>
<td>54</td>
</tr>
<tr>
<td>APPENDIX C: Grid Generation</td>
<td>57</td>
</tr>
</tbody>
</table>
## LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1. Model Parameters Used in the Investigation</td>
<td>17</td>
</tr>
<tr>
<td>2. Numerically Generated Mode II Stress Intensity Factors</td>
<td>25</td>
</tr>
<tr>
<td>3. Numerically Predicted Initial Angle of Propagation for Selected a/W Ratios, with a Tang Width H = 1.0 Inch</td>
<td>42</td>
</tr>
</tbody>
</table>
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Modes of Crack Deformation</td>
<td>2</td>
</tr>
<tr>
<td>2</td>
<td>Compact Shear Fracture Specimen</td>
<td>4</td>
</tr>
<tr>
<td>3</td>
<td>Crack Tip Coordinate System</td>
<td>8</td>
</tr>
<tr>
<td>4</td>
<td>Mode II Stress Intensity Factor Versus r/a for a/W = 0.5, H = 1.0 Inch and a Point Load of P = 1.0 Pound</td>
<td>10</td>
</tr>
<tr>
<td>5</td>
<td>Typical Grid Pattern, a/W = 0.5</td>
<td>15</td>
</tr>
<tr>
<td>6</td>
<td>Boundary Conditions for (a) Point Load with Center Point of Tangs Fixed (b) Uniform Load with Upper Tang Boundary Fixed</td>
<td>19</td>
</tr>
<tr>
<td>7</td>
<td>Compact Tension Specimen</td>
<td>20</td>
</tr>
<tr>
<td>8</td>
<td>Mode I Stress Intensity Factor Versus r/a for Compact Tension Specimen</td>
<td>23</td>
</tr>
<tr>
<td>9</td>
<td>Nondimensionalized Mode II Stress Intensity Factor Versus a/W Ratio for Selected Tang Widths H</td>
<td>26</td>
</tr>
<tr>
<td>10</td>
<td>Nondimensionalized Mode II Stress Intensity Factor Versus a/W Ratio for a Tang Width H = 0.5 Inch</td>
<td>28</td>
</tr>
<tr>
<td>11</td>
<td>Nondimensionalized Mode II Stress Intensity Factor Versus a/W Ratio for a Tang Width H = 1.0 Inch</td>
<td>29</td>
</tr>
<tr>
<td>12</td>
<td>Nondimensionalized Mode II Stress Intensity Factor Versus a/W Ratio for a Tang Width H = 1.5 Inches</td>
<td>31</td>
</tr>
<tr>
<td>13</td>
<td>Stress Boundary Conditions Assumed in Boundary Collocation Study</td>
<td>33</td>
</tr>
<tr>
<td>14</td>
<td>Numerically Generated Stress Distribution Along Upper Tang Boundary for a/W = 0.5, H = 1.0 Inch, Point Load, Center Point of Tangs Fixed, and Plane Strain Conditions</td>
<td>34</td>
</tr>
<tr>
<td>15</td>
<td>Numerically Generated Stress Distribution Along Upper Tang Boundary for a/W = 0.5, H = 1.0 Inch, Uniform Loading, Center Point of Tangs Fixed, and Plane Strain Conditions</td>
<td>35</td>
</tr>
<tr>
<td>Figure</td>
<td>Page</td>
<td></td>
</tr>
<tr>
<td>--------</td>
<td>------</td>
<td></td>
</tr>
<tr>
<td>16. Numerically Generated Stress Distribution Along Vertical Axis of Symmetry for $a/W = 0.5$, $H = 1.0$ Inch, Point and Uniform Loading, Center Point of Tangs Fixed, and Plane Strain Conditions</td>
<td>36</td>
<td></td>
</tr>
<tr>
<td>17. Numerically Generated Stress Distribution Along Upper Tang Boundary for $a/W = 0.5$, $H = 1.0$ Inch, Point Load, Upper Tang Boundary Fixed, and Plane Strain Conditions</td>
<td>37</td>
<td></td>
</tr>
<tr>
<td>18. Numerically Generated Stress Distribution Along Upper Tang Boundary for $a/W = 0.5$, $H = 1.0$ Inch, Uniform Loading, Upper Tang Boundary Fixed, and Plane Strain Conditions</td>
<td>38</td>
<td></td>
</tr>
<tr>
<td>19. Numerically Generated Stress Distribution Along Vertical Axis of Symmetry for $a/W = 0.5$, $H = 1.0$ Inch, Point and Uniform Loading, Upper Tang Boundary Fixed, and Plane Strain Conditions</td>
<td>39</td>
<td></td>
</tr>
<tr>
<td>20. Global Coordinate System Used in SSAP</td>
<td>50</td>
<td></td>
</tr>
<tr>
<td>21. Local Coordinate System Used in SSAP</td>
<td>50</td>
<td></td>
</tr>
<tr>
<td>22. Center Cracked Infinite Strip Used in Closed Form Solution</td>
<td>55</td>
<td></td>
</tr>
<tr>
<td>23. Notched Plate Used in Sample Problem</td>
<td>58</td>
<td></td>
</tr>
<tr>
<td>24. Grid Pattern for Notched Plate Without Triangular Transition Elements</td>
<td>59</td>
<td></td>
</tr>
<tr>
<td>25. Completed Grid Pattern for Notched Plate</td>
<td>61</td>
<td></td>
</tr>
<tr>
<td>26. Listing of Input to Grid Generation Program for Notched Plate</td>
<td>62</td>
<td></td>
</tr>
</tbody>
</table>
CHAPTER I

INTRODUCTION TO PROBLEM

1.1 General Introduction

The use of high-strength materials in the design of engineering structures has lead to the development of theories which can be used to predict the reduced strength of these structures caused by induced or inherent flaws in the material. Engineering fracture mechanics utilizes the concepts of stress intensity factor, \( K \), and critical stress intensity factor, \( K_c \), to predict this reduced strength. The stress intensity factor, \( K \), is a function of applied load and geometry, while the critical stress intensity factor, \( K_c \), is an experimentally determined constant for a given material and mode of deformation.

There are three possible modes of deformation associated with a crack as shown in Figure 1: opening mode, Mode I; edge sliding mode, Mode II; and tearing mode, Mode III. Until recently, Mode I has been considered as the most significant mode of failure. As a consequence, preceding investigations in fracture mechanics have dealt primarily with this mode, and data on critical stress intensity factors is restricted to Mode I loading. Recent investigations, however, indicate that Mode II may be a significant mode of failure in certain cases. Jones and Chisholm [1] established a compact shear fracture specimen to study the phenomenon of Mode II fracture and determine accurate critical stress intensity data.
Figure 1. Modes of Crack Deformation
The object of this investigation was to check the performance of the compact shear fracture specimen by numerically generating stress intensity factors using finite element techniques. In addition, initial crack propagation angles were numerically predicted.

1.2  **Purpose of Investigation**

This study was conducted to investigate, by finite element techniques, the behavior of the compact shear fracture specimen developed by Jones and Chisholm. In particular, Mode II stress intensity factors and stress boundary conditions were numerically generated for comparison with those obtained by Jones using boundary collocation and photoelastic methods. In addition, the initial angle of propagation which could not be obtained by Jones was determined for selected specimen geometry and crack lengths.

1.3  **Scope of Investigation**

This study utilized the compact shear fracture specimen established by Jones and Chisholm to numerically determine the effect of changing crack length, applied load, and specimen geometry on the Mode II stress intensity factor, $K_{II}$. The specimen, shown in Figure 2, has a specimen height $W$, thickness $B$, crack length $a$, and tang width $H$. The model parameters varied in this investigation were crack length $a$, and tang width $H$.

Point and uniform loads were utilized in this investigation to simulate conditions that could be achieved in a rigid loading frame, and also to approximate as closely as possible the loading conditions adopted by Jones. The stress distribution along the upper boundary and
Figure 2. Compact Shear Fracture Specimen
vertical axis of symmetry obtained for selected load conditions and model configurations were compared to those assumed in the boundary collocation study.

The initial angle of propagation of the crack, for a given crack length, applied load, and model geometry, was also determined.

1.4 Approach Used in the Investigation

A numerical solution, the finite element method, was employed to generate displacements along the crack flanks, the normal and tangential stress distribution along the upper tang boundary, and the normal stress distribution along the vertical axis of symmetry. A plane strain condition was utilized for the numerical analysis throughout this investigation.

The stress intensity factors for the various configurations studied were determined by the displacement [2] and strain energy release rate methods [2]. Numerical results were compared to those obtained in a closed form solution for a model with similar configuration and loading conditions, and to the boundary collocation results. A description of the closed form solution is given in Appendix B.

The numerically generated normal and tangential stresses along the upper boundary and vertical axis of symmetry represented the stress boundary conditions actually occurring in the model for a given load and model configuration. These were compared to the stress boundary conditions assumed by Jones and Chisholm.

A strain energy density technique [6] was used to numerically obtain the initial angle of propagation of the crack for selected crack lengths and specimen geometry.
CHAPTER II

FINITE ELEMENT TECHNIQUES APPLIED TO LINEAR ELASTIC FRACTURE MECHANICS

2.1 Introduction

The finite element method was utilized to generate displacements along the free flanks of the crack, and the normal and tangential stress distributions along the vertical axis of symmetry and upper boundary of the compact shear fracture specimen under consideration. The methods used to calculate Mode II stress intensity factors for given specimen configuration, crack length and applied loading were the displacement method and the method of strain energy release rate. A brief description of the finite element program used is given in Appendix A. The initial angle of propagation for selected cases was numerically determined by the strain energy density method.

2.2 Displacement Method

The displacement method [2] utilized the displacements of nodal points along the crack flanks and equations describing the displacement field near the crack tip. The equations describing the displacement field are those derived by Westergaard [3], and shown below for Mode I and Mode II plane strain deformations. For Mode I loading,

\[ u_I = \frac{K_I}{\mu} \left( \frac{r}{2\pi} \right)^{1/2} \cos \frac{\theta}{2} \left[ 1 - 2v + \sin^2 \left( \frac{\theta}{2} \right) \right] \]  (2.1)
and

\[ v_I = \frac{K_I}{u} \left[ \frac{r}{2\pi} \right]^{1/2} \sin \frac{\theta}{2} \left[ 2 - 2\nu - \cos^2 \left( \frac{\theta}{2} \right) \right], \quad (2.2) \]

while for Mode II loading,

\[ u_{II} = \frac{K_{II}}{u} \left[ \frac{r}{2\pi} \right]^{1/2} \sin \frac{\theta}{2} \left[ 2 - 2\nu + \cos^2 \left( \frac{\theta}{2} \right) \right] \quad (2.3) \]

and

\[ v_{II} = \frac{K_{II}}{u} \left[ \frac{r}{2\pi} \right]^{1/2} \cos \frac{\theta}{2} \left[ -1 + 2\nu + \sin^2 \left( \frac{\theta}{2} \right) \right], \quad (2.4) \]

where \( \mu \) is the shear modulus, \( \nu \) is Poisson's ratio, \( r \), \( \theta \), \( u \) and \( v \) are defined in Figure 3. \( K_I \) and \( K_{II} \) are the Mode I and Mode II stress intensity factors.

By using nodal displacements, \( u^* \) and \( v^* \), along the crack flank as determined by the finite element method, a stress intensity factor \( K_I^* \) or \( K_{II}^* \) can be found at each nodal point by use of Equations (2.1) through (2.4). Chan [2] found that the most accurate values of \( K_I^* \) and \( K_{II}^* \) are attained by using the equations for \( v_I \) and \( u_{II} \) with \( \theta = 180^\circ \). Thus,

\[ K_I^* = \left[ \frac{(2\pi)^{1/2} E}{4(1 - \nu^2)} \right] \frac{v_I^*}{r^{1/2}} \quad (2.5) \]

and

\[ K_{II}^* = \left[ \frac{(2\pi)^{1/2} E}{4(1 - \nu^2)} \right] \frac{u_{II}^*}{r^{1/2}} \quad (2.6) \]
Figure 3. Crack Tip Coordinate System
If the exact displacements were used as \( r \) approaches zero, the exact stress intensity will be determined. Since the finite element method gives accurate solutions along the flanks, and very inaccurate solutions at the crack tip, a tangent extrapolation must be used to determine stress intensity factors.

This is accomplished by plotting a graph of the stress intensity factor, \( K^* \), versus the nondimensionalized distance from the crack tip, \( r/a \), Figure 4, using Equations (2.5) and (2.6). A tangent to the straight line region of this curve is then extrapolated back to the point where it intersects the stress intensity axis. This intercept is taken as the value of the stress intensity factor \( K \). A least squares fit to the data in the straight line region was used to determine the intercept in this study. As can be seen in Figure 4, the straight line region does not extend the length of the crack flank. The observed nonlinearity is due to the boundary condition imposed at the load points, and the fact that the equations for displacement, Equations (2.1) through (2.4), are strictly valid near the crack tip. The procedure used in this investigation was to perform a least squares fit in the region of \( r/a \) ratios ranging from 0.1 to 0.2.

2.3 Strain Energy Release Rate

The concept of strain energy release rate [2] states that whenever the strain energy released by the structure is greater than the energy needed to create new crack surface area, the crack will propagate unstably. Mathematically, strain energy released rate, \( G \), can be written as:
where $U$ is the strain energy stored in the structure and $A$ is the new crack surface area formed as the crack extends. Strain energy release rate is related to Mode I and Mode II stress intensity factors by the relation [4]

$$K_I^2 + K_{II}^2 = \frac{EG}{1 - \nu^2}.$$  \hfill (2.8)

This equation is valid for plane strain conditions.

The finite element method can be used to solve for the strain energy release rate, expressed by Equation (2.7), by the following means. The total strain energy of the structure, $U_T$, is determined by numerically summing the strain energy of each element of the structure. The crack is then extended by a small incremental length, $\Delta a$, and the total strain energy, $U'_T$, is again determined. For plane strain analysis, with the thickness of the model set equal to unity, Equation (2.7) can now be rewritten as:

$$G = \frac{U'_T - U_T}{\Delta a}.$$  \hfill (2.9)

Utilizing Equation (2.9) and the fact that for the fracture specimen being studied, Mode I stress intensity factors were negative and therefore could be neglected, $K_{II}$ can be determined from the relation:

$$K_I^2 + K_{II}^2 = \frac{E}{1 - \nu^2} \left( \frac{U'_T - U_T}{\Delta a} \right).$$  \hfill (2.10)
2.4 Strain Energy Density

Because cracks which are not oriented perpendicular to the applied load tend to propagate in a direction other than along the axis of the crack, Sih [5] proposed the concept of strain energy density in order to analytically determine the direction of crack growth. Sih determined that the magnitude of the energy field in the vicinity of the crack tip can be written as:

\[ S = a_{11} K_{I}^2 + a_{12} K_{I} K_{II} + a_{22} K_{II}^2 \]  

(2.11)

where

\[ a_{11} = \frac{1}{16\mu} \left[ (1 + \cos \theta)(\kappa - \cos \theta) \right] \]

\[ a_{12} = \frac{1}{16\mu} \sin \theta [2 \cos \theta - (\kappa - 1)] \]

and

\[ a_{22} = \frac{1}{16\mu} \left[ (\kappa + 1)(1 - \cos \theta) + (1 + \cos \theta)(3 \cos \theta - 1) \right] \]

(2.12)

In these equations, \( \kappa = 3 - 4\nu \) for plane strain and \( (3 - \nu)(1 + \nu) \) for plane stress problems, \( \theta \) is defined in Figure 3, \( \mu \) is the shear modulus and \( K_I \) and \( K_{II} \) are the Mode I and Mode II stress intensity factors. The concept of strain energy density states that a crack will propagate in the direction for which the strain energy density \( S \) possesses a stationary minimum value, or
\[
\frac{\partial S}{\partial \theta} = 0. 
\] (2.13)

The initial direction of propagation can now be determined by first using finite element techniques to solve for the stress intensity factors, \( K_I \) and \( K_{II} \). Equations (2.11) and (2.12) are then used to numerically identify the value of \( \theta \) which will result in a minimum value of \( S \). In this study, the solution of this problem produced two values of \( \theta \) which yielded a minimum value of \( S \) with the correct value being determined by a physical argument.
CHAPTER III

MODEL USED IN THE INVESTIGATION AND VERIFICATION OF NUMERICAL PROCEDURE

3.1 Introduction

As stated earlier, the model considered in this study is a fracture specimen proposed by Jones and Chisholm and previously shown in Figure 2. This model was gridded for generation of displacements and stresses by the finite element method. The boundary and load conditions applied to the model are discussed in the following paragraphs and verification of the gridding program and finite element technique is established.

3.2 Grid Pattern

A typical grid pattern utilized in this investigation is shown in Figure 5. The pattern consists of a fine region at the crack tip, a transition region and a course region. In the fine region, a typical ratio of element area to the square of crack length, $A/a^2$, is $2.6 \times 10^{-3}$ for $a/W = 0.5$. This ratio for the course region is $1.04 \times 10^{-2}$. Ratios were chosen in each region to assure optimal convergence to the true solution [2]. The material above the loading pins was not included in the finite element model because it did not contribute any significant stiffness. The fine grid region was moved with the crack tip as crack length was varied. This required that the
Figure 5. Typical Grid Pattern, $a/W = 0.5$
mesh be regenerated each time a new crack length was considered. In order to decrease the amount of time spent in regridding the model, a grid generation code obtained from another finite element program (SAAS—Stress Analysis of Axisymmetric Solids)[6] was modified to grid the specimen. The modification and required input into this program is discussed in detail in Appendix C. Briefly, this program interpolates the position of internal nodal points from those specified on the external boundaries. The program then eliminates nodal points as described in Appendix C in order to achieve the desired reduction of the grid pattern. Finally, the element information is obtained by using generated nodal point data.

3.3 Model Geometry

A list of the parameters considered in this investigation is given in Table I. The ratio of crack length to specimen height, $a/W$, was varied from 0.1 to 0.8. This range was selected to determine the effect that the model boundaries have on the Mode II stress intensity factors for short and long cracks. The dependence of the Mode II stress intensity factor on distance from the load point to crack flank was studied by changing the tang width, $H$, of the specimen. Three widths were considered; $H = 1.5$ inches, $H = 1.0$ inch, and $H = 0.5$ inch. For $H = 1.5$ inches, the same grid pattern utilized for $H = 1.0$ inch was adopted with the outer boundary elements expanded to give the desired tang width. The grid pattern for $H = 0.5$ inch had to be regenerated. This was easily accomplished by a modification of the gridding program which allowed it to accept data for $H = 1.0$ inch, and grid the model
TABLE I

MODEL PARAMETERS USED IN THE INVESTIGATION

<table>
<thead>
<tr>
<th>$H$ (INCHES)</th>
<th>$a/W$</th>
<th>$W$ (INCHES)</th>
<th>$B$ (INCHES)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>0.2</td>
<td>3.115</td>
<td>1.00</td>
</tr>
<tr>
<td>0.5</td>
<td>0.3</td>
<td>3.115</td>
<td>1.00</td>
</tr>
<tr>
<td>0.5</td>
<td>0.5</td>
<td>3.115</td>
<td>1.00</td>
</tr>
<tr>
<td>0.5</td>
<td>0.7</td>
<td>3.115</td>
<td>1.00</td>
</tr>
<tr>
<td>1.00</td>
<td>0.1</td>
<td>3.115</td>
<td>1.00</td>
</tr>
<tr>
<td>1.00</td>
<td>0.2</td>
<td>3.115</td>
<td>1.00</td>
</tr>
<tr>
<td>1.00</td>
<td>0.3</td>
<td>3.115</td>
<td>1.00</td>
</tr>
<tr>
<td>1.00</td>
<td>0.4</td>
<td>3.115</td>
<td>1.00</td>
</tr>
<tr>
<td>1.00</td>
<td>0.5</td>
<td>3.115</td>
<td>1.00</td>
</tr>
<tr>
<td>1.00</td>
<td>0.6</td>
<td>3.115</td>
<td>1.00</td>
</tr>
<tr>
<td>1.00</td>
<td>0.7</td>
<td>3.115</td>
<td>1.00</td>
</tr>
<tr>
<td>1.00</td>
<td>0.8</td>
<td>3.115</td>
<td>1.00</td>
</tr>
<tr>
<td>1.50</td>
<td>0.2</td>
<td>3.115</td>
<td>1.00</td>
</tr>
<tr>
<td>1.50</td>
<td>0.4</td>
<td>3.115</td>
<td>1.00</td>
</tr>
<tr>
<td>1.50</td>
<td>0.5</td>
<td>3.115</td>
<td>1.00</td>
</tr>
<tr>
<td>1.50</td>
<td>0.6</td>
<td>3.115</td>
<td>1.00</td>
</tr>
<tr>
<td>1.50</td>
<td>0.7</td>
<td>3.115</td>
<td>1.00</td>
</tr>
<tr>
<td>1.50</td>
<td>0.8</td>
<td>3.115</td>
<td>1.00</td>
</tr>
</tbody>
</table>
for \( H = 0.5 \) inch. The modification restricted the range over which
the program will search for nodal points, thus giving the desired tang
width.

3.4 Load and Boundary Conditions

The loading conditions used in this investigation consisted of
point and uniform loads applied to the specimen as shown in Figures 6a
and 6b, respectively. In order to simulate the loading conditions used
in the boundary collocation analysis [1], it was necessary to pin the
load points in the y-direction as shown in Figure 6a. The left boundary
of the model was pinned in the y-direction because it is a line of
symmetry. The uniform loading condition, with the nodal points on the
upper boundary pinned in the y-direction as shown in Figure 6b, was
selected to simulate a fixed grip loading frame. The crack tip was
pinned in the z-direction to eliminate any rigid-body movement of the
crack flanks in the z-direction.

3.5 Verification of Gridding Program and Numerical Techniques

Verification of the gridding program and numerical techniques
for obtaining stress intensity factors was accomplished by generating
a representative grid pattern and loading the compact shear specimen as
a compact tension specimen. The transformation from compact shear to
compact tension specimen was easily made by simply applying the loads
at the side and vertical axis of symmetry along a line perpendicular to
the crack axis as shown in Figure 7. The Mode I stress intensity factor
was then numerically obtained using both the displacement and strain
energy release rate techniques and compared to an available closed form
Figure 6. Boundary Conditions for (a) Point Load with Center Point of Tangent Fixed (b) Uniform Load with Upper Tang Boundary Fixed
Figure 7. Compact Tension Specimen
solution. A closed form solution for the compact tension specimen, shown in Figure 7, is given by Sih [10] as:

\[ K_I = \frac{P(2W + a)}{\sqrt{\pi} h(W - a)^{3/2}} F \left( \frac{a}{W}, \frac{b}{W}, \frac{c}{b} \right) \]  \hspace{1cm} (3.1)

where \( F \left( \frac{a}{W}, \frac{b}{W}, \frac{c}{b} \right) \) is given for various parameters by Sih [10] and will be used here only for the geometry considered. The boundary conditions necessary for numerical generation of the Mode I stress intensity factor are shown in Figure 7. With these boundary conditions and the model configuration for a compact shear fracture specimen with \( a/W = 0.5 \) and \( H = 1.0 \) inch, the physical parameters of Equation (3.1) become

\[ a = 0.775 \text{ inch} \]
\[ b = 1.00 \text{ inch} \]
\[ W = 2.340 \text{ inches} \]
\[ c/b = 0.3 \]
\[ P = 100 \text{ pounds} \]
\[ h = 1.0 \text{ inch} \]

A \( c/b \) ratio of 0.3 was chosen because this was the closest value listed by Sih which fit the loading conditions applied to the compact tension specimen. It also should be noted that the value of stress intensity factor given by Equation (3.1) is not the same as that determined by the displacement technique. The relation between these two values is given by

\[ K_I = \sqrt{\pi} k_I \] \hspace{1cm} (3.2)
where $k_I$ is determined by Equation (3.1) and $K_I$ is determined by the displacement method.

A graph of stress intensity factor versus nondimensionalized distance from the crack tip for the compact tension specimen is given in Figure 8. A least squares fit was performed on the points in the region $r/a$ from 0.1 to 0.2 and $K_I$, the intercept, was found to be $K_I = 500 \text{ psi} \sqrt{\text{in}}$. The value determined from Equations (3.1) and (3.2) with $F \left( \frac{a}{W}, \frac{b}{W}, \frac{c}{b} \right) = 2.010$ is $K_I = 493 \text{ psi} \sqrt{\text{in}}$. These values are in good agreement with each other and therefore verify the displacement technique and grid generation program.
Figure 8. Mode I Stress Intensity Factor Versus r/a for Compact Tension Specimen
CHAPTER IV

ANALYSIS OF RESULTS

4.1 Introduction

Having verified the numerical methods and grid generation program used in this investigation, the behavior of the compact shear fracture specimen as a function of crack length, loading condition and specimen geometry is discussed. Mode II stress intensity factors generated by finite element techniques are compared to those determined by boundary collocation analysis, and to a closed form solution for geometry and loading conditions that closely approximate those of the compact shear fracture specimen. The numerically generated stress distribution on the upper model boundary and vertical axis of symmetry are compared with those assumed by Jones and Chisholm [1] in a boundary collocation study. The initial angle of propagation is also given for selected crack lengths and specimen tang widths.

4.2 Stress Intensity Factors

Numerically generated Mode II stress intensity factors for selected nondimensionalized crack lengths and tang widths are listed in Table II. This data is graphically portrayed in Figure 9 where the Mode II stress intensity factors have been nondimensionalized by dividing $K_{II}$ by the normal stress in the tang, $\sigma = P/BH$, and the square root of crack length, $\sqrt{a}$. The curves shown in Figure 9
TABLE II

NUMERICALLY GENERATED MODE II STRESS INTENSITY FACTORS

<table>
<thead>
<tr>
<th>a/W</th>
<th>H = 0.5 Inch</th>
<th>H = 1.00 Inch</th>
<th>H = 1.50 Inches</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Displacement</td>
<td>Energy</td>
<td>Displacement</td>
</tr>
<tr>
<td>0.1</td>
<td>-</td>
<td>-</td>
<td>63.29</td>
</tr>
<tr>
<td>0.2</td>
<td>148.09</td>
<td>137.30</td>
<td>78.59</td>
</tr>
<tr>
<td>0.3</td>
<td>152.49</td>
<td>145.10</td>
<td>92.80</td>
</tr>
<tr>
<td>0.4</td>
<td>-</td>
<td>-</td>
<td>115.83</td>
</tr>
<tr>
<td>0.5</td>
<td>137.68</td>
<td>136.70</td>
<td>122.06</td>
</tr>
<tr>
<td>0.6</td>
<td>-</td>
<td>-</td>
<td>145.85</td>
</tr>
<tr>
<td>0.7</td>
<td>158.44</td>
<td>166.44</td>
<td>173.09</td>
</tr>
<tr>
<td>0.8</td>
<td>-</td>
<td>-</td>
<td>209.27</td>
</tr>
</tbody>
</table>
Figure 9. Nondimensionalized Mode II Stress Intensity Factor Versus $a/W$ Ratio for Selected Tang Widths $H$
indicate that the specimen having a tang width \( H = 1.0 \) inch is the most stable, as evidenced by a relatively constant nondimensionalized Mode II stress intensity factor. The curves for \( H = 0.5 \) inch and \( H = 1.5 \) inches exhibit rather wide variations in the nondimensionalized Mode II stress intensity factors with varying crack length. These cases will, therefore, be examined in more detail.

Figure 10 shows the variation of the nondimensionalized Mode II stress intensity factor with crack length for a tang width \( H = 0.5 \) inch. Excellent correlation is shown between results obtained by finite element techniques and those determined from a closed form solution [7]. However, the boundary collocation data does not agree with the numerical and closed form results. The curves for the finite element and closed form solutions show a region of constant nondimensionalized Mode II stress intensity factors from \( a/W = 0.4 \) to \( a/W = 0.7 \). For \( a/W \) ratios greater than 0.7, the lower specimen boundary influences the Mode II stress intensity factors. At \( a/W \) ratios lower than 0.4, the curves show a load dependence as the load is now being applied close to the crack tip. The boundary collocation curve exhibits constant nondimensionalized Mode II stress intensity factors over the entire range of \( a/W \) ratios because of incorrect stress boundary conditions assumed in the boundary collocation analysis. This error in assumed stress boundary conditions will be discussed in detail in a later section.

Figure 11 shows the variation of nondimensionalized Mode II stress intensity factors with crack length for a tang width \( H = 1.0 \) inch. All curves are in good agreement with each other and have a region of constant nondimensionalized Mode II stress intensity factors...
Figure 10. Nondimensionalized Mode II Stress Intensity Factor Versus $a/W$ Ratio for a Tang Width $H = 0.5$ Inches
Figure 11. Nondimensionalized Mode II Stress Intensity Factor
Versus a/W Ratio for a Tang Width H = 1.0 Inch
from \( a/W = 0.2 \) to \( a/W = 0.6 \). At \( a/W \) ratios above 0.6 and below 0.2, the specimen boundaries are being sensed by the crack tip, causing the curves to turn upward.

Figure 12 gives the variation of nondimensionalized Mode II stress intensity factors with crack length for a tang width \( H = 1.5 \) inches. The finite element and closed form results do not exhibit a region over which the nondimensionalized Mode II stress intensity factors are constant because the specimen is now behaving like a beam in three point bending. The influence of bending is evident at all \( a/W \) ratios particularly those less than \( a/W = 0.3 \), for in this region the curves were expected to turn upward as the crack tip sensed the upper boundary of the specimen. The boundary collocation data exhibits a different behavior from that of the finite element and closed form results because of incorrect stress boundary conditions assumed in the boundary collocation study.

4.3 Stress Boundary Conditions

Stress boundary conditions were determined along the upper boundaries of each tang and along the vertical axis of symmetry in order to check those assumed in the boundary collocation analysis and to provide useful data for future investigations utilizing boundary value techniques. Of particular interest in this investigation was the stress distribution along the upper tang boundary and vertical axis of symmetry for application of a point load with the load points fixed in a direction perpendicular to the direction of loading, as shown in Figure 6a. This method of loading was of interest because it simulates that used by
Figure 12. Non-dimensionalized Mode II Stress Intensity Factors Versus $a/W$ Ratio for a Tang Width $W = 1.5$ Inches
Jones and Chisholm so that resultant numerically generated stress distributions could be used to check the stress boundary conditions assumed by Jones and Chisholm. A uniform load with each nodal point on the upper tang boundary fixed in a direction perpendicular to the direction of loading, as shown in Figure 6b, was also applied in order to determine the stress boundary conditions needed in order to perform boundary value analyses for this common type of loading. The two loading conditions just discussed are obtainable through a rigid loading frame. The stress distribution for two other loading conditions are also discussed in this section. These loading conditions are a point load with all nodal points on the upper tang boundary fixed in a direction perpendicular to the direction of loading, and a uniform load with the center point of each tang fixed in a direction perpendicular to the direction of loading. Although these conditions are easily simulated in a numerical study, they are very difficult to obtain experimentally and were considered in this investigation simply to determine the sensitivity of stress intensity factors to changes in boundary conditions.

The stress boundary conditions assumed in the boundary collocation analysis are shown in Figure 13. The stress distribution along the upper tang boundary consists of a uniform tensile stress on the outside tang, a uniform compressive stress along the center tang and a cosine distribution of tangential stress along the top of both tangs. A bilinear bending stress was assumed along the vertical axis of symmetry.

The numerically generated stress distributions along the upper tang boundary and vertical axis of symmetry for $a/W = 0.5$, $H = 1.0$ inch and plane strain conditions are shown in Figures 14 through 19 for
Figure 13. Stress Boundary Conditions Assumed in Boundary Collocation Study
Figure 14. Numerically Generated Stress Distribution Along Upper Tang Boundary for $a/W = 0.5$, $H = 1.0$ Inch, Point Load, Center Point of Tang Fixed, and Plane Strain Conditions
Figure 15. Numerically Generated Stress Distribution Along Upper Tangential Boundary for $a/W = 0.5$, $H = 1.0$ Inch, Uniform Loading, Center Point of Tangs Fixed, and Plane Strain Conditions.
Figure 16. Numerically Generated Stress Distribution Along Vertical Axis of Symmetry for $a/W = 0.5$, Center Point of Tongs Fixed, and Plane Strain Conditions
Figure 17. Numerically Generated Stress Distribution Along Upper Tang Boundary for $a/W = 0.5$, $H = 1.0$ Inch, Point Load, Upper Tang Boundary Fixed, and Plane Strain Conditions
Figure 18. Numerically Generated Stress Distribution Along Upper Tang Boundary for $a/W = 0.5$, $H = 1.0$ Inch, Uniform Loading, Upper Tang Boundary Fixed, and Plane Strain Conditions
Figure 19. Numerically Generated Stress Distribution Along Vertical Axis of Symmetry for $a/W = 0.5$, $H = 1.0$ Inch, Point and Uniform Loading, Upper Tang Boundary Fixed, and Plane Strain Conditions
selected load and displacement boundary conditions. Figures 14 through 16 show the stress distributions for a point and uniform load with the center point of each tang fixed in a direction perpendicular to the direction of loading. Figures 17 through 19 show the stress distributions for a point and uniform load with each nodal point along the upper tang boundary fixed in a direction perpendicular to the direction of applied load.

As can be seen by referring to Figures 14 and 17, the normal and tangential stress distribution along the upper tang boundaries caused by the application of a point load does not change as the displacement boundary conditions are varied. Figures 15 and 18 show that the tangential stress distribution along the upper tang boundaries caused by the application of a uniform load is in fact the only distribution affected by variation in displacement boundary conditions. Figures 16 and 19 show practically no change in the normal stress distribution along the vertical axis of symmetry for variation in displacement boundary conditions.

Comparison of Figures 13, 14 and 17 indicates that the numerically generated stress distributions for point loading are not in good agreement with those assumed in the boundary collocation analysis. The major differences occur in the normal stress distribution along the upper tang boundary. This difference in stress distribution is responsible for the discrepancies observed in the nondimensionalized Mode II stress intensity factors determined by numerical techniques and boundary collocation analysis.
The stress distributions caused by the application of a uniform load with all the nodal points along the upper tang boundary fixed, as shown in Figure 18, were somewhat similar to those assumed in the boundary collocation analysis, shown in Figure 13. The major difference in the two distributions is that the numerically generated tangential stress distribution on the center tang is not a cosine distribution as assumed in the boundary collocation analysis. Figures 16 and 19 show that the bending stress along the vertical axis of symmetry for both point and uniform loading is not a simple bilinear distribution as shown in Figure 13.

Numerically generated stress distributions were also obtained for other a/W ratios and tang widths resulting in distributions having the same form but different magnitudes than those shown in this section. Plane stress conditions were also tried for selected load conditions and model geometries resulting in no difference in stress distributions when compared to those determined using plane strain conditions. The Mode II stress intensity factors were unaffected by changes in loading and displacement boundary conditions discussed in this section.

4.4 Initial Angle of Crack Propagation

The angle of propagation \( \theta \) is defined in Figure 3. By use of the strain energy density technique proposed by Sih [5] and discussed in Chapter II, the angle of propagation was determined for \( a/W = 0.5 \) through 0.8 with \( H = 1.0 \) inch. These values are listed in Table III. These particular values of crack length and specimen tang width were chosen because the experimental work performed by Jones considered an
### TABLE III

Numerically Predicted Initial Angle of Propagation for Selected $a/W$ Ratios With a Tang Width $H = 1.00$ Inch

<table>
<thead>
<tr>
<th>$a/W$</th>
<th>Theta $\theta$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>77°</td>
</tr>
<tr>
<td>0.6</td>
<td>77°</td>
</tr>
<tr>
<td>0.7</td>
<td>77°</td>
</tr>
<tr>
<td>0.8</td>
<td>75°</td>
</tr>
</tbody>
</table>
a/W ratio of 0.8 with specimen tang width, H = 1.0 inch. With these parameters, and notching the model along the plane of the crack, Jones and Chisholm were able to obtain a straight fracture of the specimen, \( \theta = 0^\circ \). As can be seen from Table III, the strain energy density technique predicts that the initial angle of propagation is 75° for \( a/W = 0.8 \) and 77° for \( a/W = 0.5 \) to 0.7. The path of the crack was not investigated after the initial angle of propagation because of the amount of computer time necessary to accomplish this.
CHAPTER V

CONCLUSIONS AND RECOMMENDATIONS

5.1 Conclusions

The effect of crack length and specimen geometry on the Mode II stress intensity factors for the compact shear fracture specimen shown in Figure 2 was investigated. Stress intensity factors were generated by the displacement and strain energy release rate methods and compared to those determined by boundary collocation. Both numerical and boundary collocation results obtained for a tang width $H = 1.0$ inch show that nondimensionalized Mode II stress intensity factors are constant for this specimen over a wide range of crack lengths. This configuration of the compact shear fracture specimen will therefore be most suitable for determining critical Mode II stress intensity factors, $K_{Ic}$.

Numerical results for $H = 0.5$ inch and $H = 1.5$ inches show that nondimensionalized Mode II stress intensity factors are not as stable with increasing crack length as the boundary collocation results suggest. The stress intensity factors obtained for a tang width $H = 1.5$ inches are affected by bending of the specimen, a fact that was not brought out by the boundary collocation results. Results for tang width $H = 0.5$ inch indicated that the nondimensionalized Mode II stress intensity factors were influenced by applied load for $a/W$ ratios less than 0.4. Because of these nonlinearities, the configurations
having tang width \( H = 0.5 \) inch and \( H = 1.5 \) inches are not suitable for determining the critical Mode II stress intensity factor, \( K_{IIc} \).

For the crack lengths and specimen geometries investigated, the non-dimensionized Mode II stress intensity factors were found to be independent of the applied load, and uninfluenced by the lower free boundary through the intermediate ranges of crack lengths.

Numerical results indicated that the tangential stress along the upper boundary of the center tang did not follow a cosine distribution as assumed in the boundary collocation analysis. In addition the numerically determined bending stress distribution along the vertical axis of symmetry was not found to be the simple bilinear distribution which was used in the boundary collocation investigation. Therefore, incorrect stress boundary conditions assumed in the boundary collocation analysis were responsible for differences observed in the behavior of the non-dimensionized Mode II stress intensity factors when numerical and collocation results are compared.

The initial angle of propagation was found numerically to be 77° for the geometry and crack lengths considered. This differs from the experimental work performed by Jones and Chisholm [1], which showed an angle of 0° of crack propagation. The angle of 0° might have been the result of notching the fracture specimen along the plane of the crack. The difference between the numerically predicted angle of propagation and experimental results might not have been so great if it were possible to numerically determine the complete trajectory of the crack. This was not done because of the amount of computer time involved.
This study also showed the usefulness of the finite element method in determining Mode II stress intensity factors, for different loading conditions and specimen parameters. The accuracy of the finite element techniques discussed in this investigation was brought out by the good agreement with a closed form solution.

5.2 Recommendations

It is recommended that:

1) An experimental verification of the stress boundary conditions and Mode II stress intensity factors predicted by the finite element techniques used in this investigation be performed.

2) A verification of the crack trajectory by experimental or numerical techniques, with a notch introduced into the model for comparison with the experimental work performed by Jones and Chisholm be undertaken.
BIBLIOGRAPHY


APPENDIX A

FINITE ELEMENT PROGRAM USED IN THE INVESTIGATION

A.1 Finite Element Program

The finite element program utilized in this investigation was SSAP-2 (Static Analysis Program For Solid Structures). A complete listing of the program as well as a description of the elements and input to the program is given in the program manual [8]. A brief description of the plate element used in the study will be given here.

A.2 Isoparametric Elements

An isoparametric element [9] is defined as one where both the displacement and geometry of the element are described by the same parameter. This means that the relation between the local and global coordinate systems as well as the displacement approximation for the element are given in terms of the same interpolation function. An interpolation function is one that has a unit value at one nodal point and is zero at all others in the element. Two advantages of using interpolation functions are:

1) If continuity of geometry and displacement both within and between adjacent elements are satisfied, compatibility is satisfied in global coordinates.

2) If the interpolation function is able to give rigid body displacements in the local coordinate system, rigid body displacement
and constant strain are satisfied in the global system. This is necessary for convergence of the method.

The interpolation functions used in SSAP will now be discussed. For the quadrilateral element shown in Figures 20 and 21 the local and global coordinates are related by

\[ x = \sum_{i=1}^{4} h_i x_i \]

and

\[ y = \sum_{i=1}^{4} h_i y_i , \]

where the interpolation functions, \( h_i \), are given as:

\[ h_1 = \frac{1}{4} (1 - s)(1 - t) , \]
\[ h_2 = \frac{1}{4} (1 + s)(1 - t) , \]
\[ h_3 = \frac{1}{4} (1 + s)(1 + t) \]
\[ h_4 = \frac{1}{4} (1 - s)(1 + t) , \]

where \( s \) and \( t \) are defined in Figure 21.

The displacements of the element are written in terms of the same interpolation functions as follows:

\[ u_x(s,t) = \sum_{i=1}^{4} h_i u_{xi} \]
Figure 20. Global Coordinate System Used in SSAP

Figure 21. Local Coordinate System Used in SSAP
and

\[ u_y(s,t) = \sum_{i=1}^{4} h_{1,i} u_{yi} . \]  

(A.3)

The element strains can now be written as

\[ \varepsilon_{xx} = \frac{\partial u_x}{\partial x} = \sum_{i=1}^{4} h_{1,i} u_{xi} , \]

\[ \varepsilon_{yy} = \frac{\partial u_y}{\partial y} = \sum_{i=1}^{4} h_{1,i} u_{yi} \]  

(A.4)

and

\[ \varepsilon_{xy} = \frac{\partial u_y}{\partial x} + \frac{\partial u_x}{\partial y} = \sum_{i=1}^{4} h_{1,i} u_{xi} + \sum_{i=1}^{4} h_{1,i} u_{yi} . \]

These equations can be written in matrix form as:

\[ \varepsilon = a(s,t)u = \begin{bmatrix} H_{,x} & 0 \\ 0 & H_{,y} \\ H_{,y} & H_{,x} \end{bmatrix} \begin{bmatrix} u_x \\ u_y \end{bmatrix} , \]  

(A.5)

where \( u_x \) and \( u_y \) are the displacements in the \( x \) and \( y \) directions, respectively and

\[ H_{,x} = [h_{1,x} h_{2,x} h_{3,x} h_{4,x}] \]

and

\[ H_{,y} = [h_{1,y} h_{2,y} h_{3,y} h_{4,y}] , \]  

(A.6)

where

\[ h_{1,x} = h_{i,s} S_{,x} + t_{i,t} t_{,x} \]

and

\[ h_{1,y} = h_{i,s} S_{,y} + h_{i,t} t_{,y} . \]
Since the $h_i$ are given in terms of the natural coordinates, $s$ and $t$, the chain rule must be applied in order to compute the derivatives in terms of the $x$ and $y$ coordinate system. The result of the application of the chain rule is shown in matrix form below as:

$$
\begin{bmatrix}
s, x, t, x \\
s, y, t, y
\end{bmatrix} = \frac{1}{J} \begin{bmatrix} y, t, -y, s \\ -x, t, x, s \end{bmatrix}, \quad \text{(A.7)}
$$

where $J$ is the Jacobian which is defined by

$$
J = \sum_{i=1}^{4} h_i, s x_i + \sum_{i=1}^{4} h_i, t x_i,
$$

$$
x, s = \sum_{i=1}^{4} h_i, s x_i,
$$

$$
x, t = \sum_{i=1}^{4} h_i, t x_i, \quad \text{(A.8)}
$$

$$
y, s = \sum_{i=1}^{4} h_i, s y_i
$$

$$
y, t = \sum_{i=1}^{4} h_i, t y_i.
$$

Now Equations (A.7) and (A.8) can be used with Equations (A.6) to determine the strain-displacement matrix of Equation (A.5).

A.3 Element Stiffness

The element stiffness for unit thickness is given as

$$
K = \int_a^T T \epsilon \sigma \ dA. \quad \text{(A.9)}
$$
\( c \) is the stress-strain matrix and the integration is performed over the area of the element. This equation can be written in terms of the natural coordinate system as

\[
K = \int_{-1}^{1} \int_{-1}^{1} \mathbf{a}^T c_{aa} \mathbf{J} ds dt .
\]  

(A.10)

Standard numerical integration is used to determine \( K \) for a given element.

A.4 Total Stiffness Matrix

Once the element stiffnesses have been determined, the total stiffness matrix is obtained by summing element stiffnesses in the conventional manner,

\[
K_{TOTAL} = \sum K_{ELEMENT} .
\]  

(A.11)

A complete description of the finite element method is given by Des1 [9].
APPENDIX B

CLOSED FORM SOLUTION

B.1 Geometry and Equations

The verification of the finite element results was accomplished by comparison with a closed form solution for a model with geometry and loading conditions similar to those of the compact shear fracture specimen being studied. Since a geometry and loading condition that exactly matched this model was not available, a configuration was considered that would give a lower bound to the finite element results. The loading, geometry and closed form solution were chosen from a handbook by Tada [7] and the model is shown in Figure 22. This configuration was chosen because the loading applied is similar to the loading on the compact shear fracture specimen. The configuration will give a lower bound to the problem because it is an infinite strip and therefore stiffer than the actual plate. The equations for the Mode II stress intensity factor, \( K_{II} \), are given below:

\[
K_{II} = \frac{Q}{\sqrt{2b}} \left\{ P_{II} \left( \frac{a}{b}, \frac{s}{b} \right) \right\},
\]

where

\[
P_{II} \left( \frac{a}{b}, \frac{s}{b} \right) = f \left( \frac{a}{b}, \frac{s}{b} \right) \left[ 1 - \frac{a \frac{\pi s}{2b} \tanh \frac{\pi s}{2b}}{\cosh \frac{\pi a}{2b} - 1} \right] P_{III} \left( \frac{a}{b}, \frac{s}{b} \right), \quad (B.2)
\]
Figure 22. Center Cracked Infinite Strip Used in Closed Form Solution
\[ F_{III}(\alpha, \beta) = \left\{ \frac{1}{\tan \frac{\pi a}{2b}} \right\} \frac{1}{1 - \frac{\cos \frac{\pi a}{2b}}{\cosh \frac{\pi a}{2b}}} \] (B.3)

\[ f(\alpha, \beta) = 1 + (0.297 + 0.115(-\text{sech} \frac{\pi a}{2b}) \sin \frac{\pi a}{b}) (1 - \cos \frac{\pi a}{2b}) \] (B.4)

and

\[ \alpha = \begin{cases} \frac{1 + \nu}{2} & \text{plane stress} \\ \frac{1}{2(1 - \nu)} & \text{plane strain} \end{cases} \] (B.5)
APPENDIX C

GRID GENERATION

C.1 Introduction

The mesh generation program utilized in this investigation is a modified version of one used in SAAS [6]. The modification is in the element generation subroutine and nodal point interpolation. Element generation was changed in order to generate triangular elements in regions of transition from coarse to fine gridding and nodal point interpolation was modified to make grid reduction possible.

C.2 Input Required

Input into the gridding program will be described by considering the gridding of a plate with a notch as shown in Figure 23. The requirements for gridding a crack problem, without the use of special crack tip elements, are that a fine grid be placed in the crack tip region to accommodate the high stress gradient there. Also, in order to save computer time and storage, a relatively coarse grid should be applied at the boundaries. These two requirements can be accomplished by a transition region of triangular elements. The procedure for obtaining input data for this program is as follows. Determine what type of grid pattern is needed at the crack tip, using only quadrilateral elements. This will give a very fine element distribution as shown in Figure 24. Each horizontal line with the same z-coordinate is
Figure 23. Notched Plate Used in Sample Problem
Figure 24. Grid Pattern for Notched Plate Without Triangular Transition Elements
assigned a J index to identify it, starting with the $z = 0$ line. The same is done with every vertical line with the same $R$ coordinate, except these are assigned I indices; therefore, every intersection, nodal point, has an I,J index to identify it. The transition can now be accomplished by eliminating sections of J rows and I columns in order to achieve the desired grid pattern. The only restriction is that a section of eliminated J row cannot intersect a portion of I column that will be eliminated because the program will not be able to generate a valid element. Figure 25 shows a completed grid pattern.

The sequencing of the input data will now be described. A listing of input data for this problem is given in Figure 26. The first three cards are control cards and are described in the manual for SAAS and the control data for the finite element program has been eliminated. The cards and formatting are described below.

**Title Card**
Describe grid pattern
Format (8A10)
Columns 1-60 Title

**Job Control Card**
Format (I2, I3, I5, I2, I3, 5I5, I3, I2, 2I5, I5.0, 3I5)
Columns 3-5 Start parameter
   = 1 Fresh set of data
Column 6-10 Stop parameter
   = 1 Stop after mesh generation
   = 33 Punch mesh data on cards
Figure 25. Completed Grid Pattern for Notched Plate
SAMPLE PROBLEM: NOTCHED PLATE

1 1 1 1
400 1
1 1 0.0 0.0 1
2 1 1.000 0.0 1
4 1 2.000 0.0 1
8 1 3.000 0.0 1
10 1 4.000 0.0 1
11 1 5.000 0.0 1
1 1 0.0 0.0 -1
1 3 0.0 2.000 1
1 11 0.0 4.000 1
1 12 0.0 5.000 1
-2
11 12 5 6 6 7
1 3 7 11 12

2.5000 3.0000
6
4 4 8
5 3 9
6 4 8
8 4 8
9 3 9
10 4 8
4
3 3 11
5 3 12
7 3 12
9 3 11
11 5 3
1 1 11 1 12

Figure 26. Listing of Input to Grid Generation Program for Notched Plate
Columns 21-25 Finite element mesh generation parameter

  ≠ 0 Generate mesh using line segment data
  = 0 Mesh data read from cards

Columns 26-30 Temperature field

  = 0 Must be set equal to zero

Columns 31-34 Number of materials (6 maximum)

These are the only parameters needed to generate a mesh, the rest can be set equal to zero.

Mesh Generation Card

Format (4I5, 2F10.0, 2I5)

Columns 1-5 Number of line segment cards. The actual number of cards may be less than this, as long as a line segment terminator card follows them.

Columns 11-15 Number of material assignment cards

Line Segment Cards

They describe the boundary of the model.

Format (2I3, 2F8.3, I5)

Columns 1-3 I2 (or I1 if this is the first card)

  I - index of point

Columns 4-6 J2 (or J1 if first card)

  J - index of point

Columns 7-14 R2 R coordinate of point

Columns 15-22 Z2 Z coordinate of point
Columns 45-49 IPTION Integer value that determines type of line segment

<table>
<thead>
<tr>
<th>IPTION</th>
<th>Description of Line Segment</th>
</tr>
</thead>
<tbody>
<tr>
<td>-1</td>
<td>Jump to this point from last, no mesh data will be generated and no line will connect the points.</td>
</tr>
<tr>
<td>0</td>
<td>Same as -1 except a line will be drawn between points.</td>
</tr>
<tr>
<td>1</td>
<td>Connect last point specified on preceding card to this one and interpolate into equal parts as specified by I, J data.</td>
</tr>
</tbody>
</table>

**Line Segment Terminator Card**

Format (I3)

Columns 2-3 = -1 Signals end of line segment cards. Must be included if number of line segment cards is less than number specified on mesh generation card.

= -2 The program was modified to fix the nodal points of each vertical line in the same position, same Z coordinate, as specified on the external boundary line.

**Internal Nodal Point Generation**

If -2 is specified on the line segment terminator card, the program has been modified to fix the Z coordinate of the internal nodal points to those specified on the external boundary. If this option is
specified, it is only necessary to specify the positions of nodal points on two external boundaries. For example, the boundaries with the Z and R coordinates of zero.

First Card Format (16I5)

Columns 1-5 IBMAX—the I index of the last column on which nodal points will be fixed.

Columns 6-10 JBMAX—the J index of the last row on which nodal points will be fixed.

Columns 11-15 NUMJ—the number of J indices to be read in on the following card.

Columns 16-20 ICT—I index of the column passing through the center of the crack.

Columns 21-25 JCT—JCT is set equal to ICT.

Columns 26-30 JCMAX—J index of crack tip.

Second Card Format (16I5) – This card contains the J indices, in sequential order, specified on the boundary with Z coordinate of zero.

A Blank Card Must Appear Here

Shift Axis Card

If coordinate axis are to be moved from the position specified on line segment card, the move can be made with this card. If change is not necessary, leave blank. Format (2F10.0).
Columns 1-10  R shift-shift in R coordinate.
Columns 11-20  Z shift-shift in Z coordinate.

The following data is needed to instruct the program how to perform the transition from course to fine grid.

**J Omit Card**

Specified the number of J rows which will have segments omitted.

Format (I5)  JOMAX

**J Segment Cards**

Specifies segment of J row that is to be included. Cards must be in sequential order starting from lowest J-index.

Format (3I5)

Columns 1-5  JOMIT  J index of row which segments are to be omitted.

Columns 6-10  MSTART  I index of first nodal point, with J index JOMIT that is to be included.

Columns 11-15  MSTOP  I index of last nodal point, with J index JOMIT that is to be included.

**I OMIT Cards**

Specifies the number of I columns which will have segments omitted.

Format (I5)  IOMAX
I Segment Cards

Specifies segment of the I column which is to be included in the grid pattern. Cards must be in sequential order starting with lowest I index.

Format (315)

Columns 1-5    IOMIT    I index of column which segments are to be omitted.

Columns 6-10   LSTART   J index of first nodal point, with I index of IOMIT that is to be included.

Columns 11-15  LSTOP    J index of last nodal point, with I index of IOMIT that is to be included.

Expansion Parameter Card

The information on this card signifies the beginning and the end of the fine gridding in the crack tip region, in terms of the J index.

Format (315)

Columns 1-5    IEXPAN   The J index of the row following the last JOMIT index specified in the J segment data.

Columns 6-10   IIEXP    I index of a node point which tells the program that after this point is passed it is to begin generating triangular elements when the nodal points are in the correct formation. This point is directly below the
crack tip and in the row with a J-index one less than the first JOMIT index specified in the J segment data.

Columns 11-15 JJEXP J index of the nodal point described in IXEXP.

Material Block Assignment

Each card assigns a material number to a block of elements defined by I-J data. The number of cards must agree with the number specified in columns 11-15 of mesh control data card.

Format (515)
Columns 1-5 Material definition number
Columns 6-10 Minimum I
Columns 11-15 Maximum I
Columns 16-20 Minimum J
Columns 21-25 Maximum J
C MAIN PROGRAM

C******************************************************************************
C SAAS III: FINITE ELEMENT STRESS ANALYSIS OF AXISYMMETRIC AND PLANE
C SOLIDS WITH DIFFERENT ANISOTROPIC, TEMPERATURE-DEPENDENT MATERIAL
C PROPERTIES IN TENSION AND COMPRESSION INCLUDING THE EFFECTS OF
C INTERNAL PORE FLUID PRESSURES AND THERMAL STRESSES.
C******************************************************************************

COMMON/BASIC/NUMNP,NUMEL,NUMPC,NUMSC,ACELZ,ANGVEL,TREF,VOF,IFREQ
COMMON/NPDATA/H(1000),CODE(1000),AX(1000),AZ(1000),T(1000)
DOUBLE PRECISION CRZ,XI,RR,ZZ,ZR,RRR,ZZZ
COMMON/ELDATA/Ix(1000),y,EMPR(1000),ALPHA(1000),PST(1000)
COMMON/SOLVE/NCODE(30),100,NUMTC
COMMON/TO/IMIN(100),MAX(100),MIN(30),MAX(30),MAXI,MAXJ,NMTL,NBC
1 *MINI,MINJ
COMMON /OM6/ ISTOP
COMMON/ICDZ/ K1,RP(250),ZP(250)
COMMON/CMC,NMC,IMIV(30),MAX(30),JMAX(30),JMAX(30)
COMMON/OM/ JSTOP
INTEGER TITLE(20)

COMMON/OMT/ISTIF
COMMON/OM2/ KP,RP(250),ZP(250)
COMMON/CMC,NMC,IMIV(30),MAX(30),JMAX(30),JMAX(30)
COMMON/OM/ JSTOP

INTEGER TITLE(20)

C CONTINUE
C CALL DATA
ZFORCE = 0.0
ELMASS = 0.0
VOLUM = 0.0

C******************************************************************************
C READ AND WRITE CONTROL INFORMATION
C******************************************************************************

READ(4,1000) TITLE,ISTIF,ISTOP,IDEF,IPLOT,NNLA,IMESH,NUMTC,
1 NPOPR,NNUMAT,NUMPC,NUMSC,TREF,NTCA,IFREQ
2 *IPLT,NUMP,NUMEL
WRITE(6,2000) TITLE (1),IV=1,15
1 ISTART,ISTOP,IDEF,IPLOT,NNLA,IMESH,
1 NUMTC,NPOPR,NNUMAT,NUMPC,NUMSC,TREF
2 *IPLT

C******************************************************************************
C GENERATE FINITE ELEMENT MESH
C******************************************************************************

215 CALL MESH

C

C INITIALIZE

C******************************************************************************
C READ AND WRITE NODAL POINT AND ELEMENT DATA
C******************************************************************************

225 CALL POINTS

C******************************************************************************
C OUTPUT ELEMENT DATA
C******************************************************************************

PRINT 3100
MPRINT=0
DO 350 N=1,NUMEL
IF(MPRINT=0)GO TO 300
WRITE(6,2008)
MPRINT=50
300 MPRINT=MPRINT-1
NSTRSS=20
THICK=0.281
350 WRITE(6,2009) N,(IX(N,I),I=1,5),ALPHA(N),T(N),PST(N),NSTRSS,THICK
IF (ISTOP.NE.33) GO TO 362
DO 360 NPUN=1,NUMP
PUNCH 3002,NPUN,(IX(NPUN,I),I=1,5),NSTRSS,THICK
360 CONTINUE
DO 361 NP=1,NUMP
PUNCH 3003,NP,R(NP),Z(NP)
361 CONTINUE
362 CONTINUE
370 IF (ISTOP.EQ.1.OR.ISTOP.EQ.11) GO TO 910
910 IF (NPP.NE.0) GO TO 200
IF (ISTOP.EQ.1) GO TO 200
IF (IFREQ.EQ.0) GO TO 200
C SUM THE ELEMENTS OF THE Z-DIRECTION BOUNDARY FORCE MATRIX
DO 905 I FOR = 1, NUMP
905 ZFORCE = ZFORCE + XZ(IFOR)
ZFORCE = ZFORCE * 2. * 3.1415927
ELMASS = ELMASS/2.0
WEIGHT = ELMASS/12.0 * 32.17
GFORCE = ELMASS * ACELZ
ELMASS = ELMASS/12.0
WRITE (6,3000) ELMASS
WRITE (6,3001) ZFORCE
WRITE(6,3003) WEIGHT
WRITE(6,3004) GFORCE
GO TO 200
C
1000 FORMAT(20A4/)
1 12*13*15*12*13*13*215*5*0*615)
1001 FORMAT (4F10.0)
2000 FORMAT(1H1,15A4/)
1 33M0 START PARAMETER-------------------14/
2 33M0 STOP PARAMETER--------------------14/
3 33M0 IF 1, PLOT DEFLECTIONS 14/
4 32M0 IF 1, SMALL PLOT. IF 2, LARGE 14/
5 33M0 PLOT. OTHERWISE NO PLOT.----------14/
6 33M0 NUMBER OF APPROXIMATIONS-------14/
7 33M0 IF 1, GENERATE MESH--------------14/
8 33M0 NUMBER OF TEMPERATURE CARDS------14/
9 33M0 NUMBER OF MATERIALS-------------14/
10 33M0 NUMBER OF EXTERNAL PRESSURES----14/
11 33M0 NUMBER OF SHEAR CARDS----------14/
12 33M0 REFERENCE TEMPERATURE----------12.4/
7 33M0 BOUNDARY PLOT OPTION----------14/
2001 FORMAT (7H A FUNDAMENTAL FREQUENCY WILL BE COMPUTED. A LONGER RUN
1 TIME WILL BE OBSERVED/60H DUE TO THE NEED TO RECOMPUTE EACH ELEME
2 NT STIFFNESS MATRIX IN SUBROUTINE STRESS)
2002 FORMAT (24H THE ANGULAR VELOCITY IS E12.4/31H AND THE AXIAL ACCEL
1 ERATION IS E12.4)
1 IN IS E12.4)
2004 FORMAT ('/42p THE PLANE STRAIN OPTION HAS BEEN SELECTED')
2005 FORMAT ('/42p THE PLANE STRESS OPTION HAS BEEN SELECTED')
2006 FORMAT ('90H1 EL I J K L MATERIAL ANGLE TEMPERATURE'
   '1 PRESSURE PRINT THICKNESS')
2009 FORMAT ('15,414,18,F11.1,2F13.3,5X,15,5X,F10.4')
2013 FORMAT ('30H1 PRESSURE BOUNDARY CONDITIONS')
2015 FORMAT ('27H1 SHEAR BOUNDARY CONDITIONS')
2020 FORMAT ('/29H THE PROCEDURE CONVERGED IN *12*34H TENSION - COMPRESSION'
   '1 SIGN ITERATIONS')
2021 FORMAT ('/36H THE PROCEDURE DID NOT CONVERGE IN *I2*33M TENSION - COMPRESSION'
   '1 ITERATIONS')
2022 FORMAT ('/29H THE PROCEDURE CONVERGED IN *I2*30M NONLINEAR ELASTIC'
   '1 ELASTIC ITERATIONS')
2023 FORMAT ('/36H THE PROCEDURE DID NOT CONVERGE IN *I2*30M NONLINEAR'
   '1 ELASTIC ITERATIONS')
2030 FORMAT ('/51H NUMBER OF TENSION-COMPRESSION APPROXIMATIONS---
   '1 14//')
3000 FORMAT ('1H***10X, 4HMAS20(2H..)*E20.14,3X,5HSLUGS')
3001 FORMAT ('1H***10X,35MAXIAL FORCE DUE TO NORMAL STRESSES***10X'
   '1 40HSHARE STRESSES, AND CONCENTRATED RADIAL, ***
   '21H***10X, 16HAND AXIAL FORCES ***14(2H..)*E20.14,3X,3HLBF')
3003 FORMAT ('1H***10X,6HWEIGHT*19(2H..)*E20.14,3X,3HLBF')
3004 FORMAT ('1H***10X,20MAXIAL INERTIAL FORCE ***12(2H..)*E20.14,3X'
   '1 3HLBF')
3002 FORMAT ('615,2CH*15,5X,F10.4')
3100 FORMAT ('1H ', '12ELEMENT DATA')
3200 FORMAT ('15,40X*2F10.4')
END
C MESH -- THIS ROUTINE GENERATES THE MESH GIVEN LINE SEGMENT INPUT
SUBROUTINE MESH

C*************************************************************************
C NCODE(I,J) = 1 FOR BOUNDARY DEFINITION POINTS
C = 2 FOR INTERPOLATED BOUNDARY POINTS
C = 3 FOR INTERPOLATED INTERIOR POINTS
C = 4 FOR EXTERIOR POINTS
C = 5 FOR VOID POINTS
C*************************************************************************
C IPTION = 0 FOR SINGLE POINTS
C = 1 FOR STRAIGHT LINES
C = 2 FOR INTERNAL DIAGONAL
C = 3 FOR 3-POINT ARC
C = 4 FOR 2 POINT + CENTER ARC
C = 5 FOR 2 POINT + RADIUS ARC (INITIALIZATION ONLY)
C = 6 FOR 2 POINT + HADIUS ARC
C*************************************************************************
C COMMON/TD/IMIN(100),IMAX(100),JMIN(30),JMAX(30),MAXI,MAXJ,NMTL,NBC
C MINI,MINT
C COMMON/NPODAT/R(1000),CODE(1000),XR(1000),ZR(1000),XZ(1000),T(1000)
C COMMON/ELDAT/A(1000),EPR(1000),ALPHA(1000),PST(1000)
C COMMON/LC2/KP,RP(250),SP(250)
C COMMON/BASIC/NUMNP,NUMEL,NUMPC,NUMSC,ANCELZ,ANGVEL,TREF,VL,IFREQ
C COMMON/OMG4/ISTOP
C COMMON/JEN/NMC,IMIV(30),IMAV(30),JMV(30),JMAV(30)
C COMMON/SOLVE/NCODE(30,100),NUMTC
C DIMENSION AH(30,100),AZ(30,100)
C DIMENSION JPB(20)
C EQUIVALENCE(K(1),AR(1,1)),(Z(1),AZ(1,1))
C*************************************************************************
C MESH CONTROL INFORMATION
C*************************************************************************
C READ(9,1000)NSEG,NBC,NMTL,NLIM,CONJ,CONJ,ISET,JSET
C WRITE(6,2000)NSEG,NBC,NMTL,NLIM,CONJ,CONJ,ISET,JSET
C*************************************************************************
C INITIALIZE
C*************************************************************************
KP=1
ISEG=0
PI=3.1415927
II=1
IC=0
DO 100 J=1,100
DO 100 I=1,30
NCODE(I,J)=4
AR(I,J)=0.
100 AZ(I,J)=0.
IMIN(J)=30
110 IMAX(J)=0
DO 120 I=1,30
JMAX(I)=0
120 JMIN(I)=100
C*************************************************************************
C LINE SEGMENT CARDS
C*************************************************************************
PRINT 2001
READ(9,1001) I2,J2,R2,Z2
OPTION=1
130 PRINT 2005
   PRINT 2010+I2+J2+R2+Z2
200 ISEG=ISEG+1
   I1=I2
   J1=J2
   R1=R2
   Z1=Z2
   AR(I1,J1)=R1
   AZ(I1,J1)=Z1
   RP(KP)=R1
   IF (OPTION.LT.0) RP(KP)=-R1
   ZP(KP)=Z1
   KP=KP+1
   NODE(I1,J1)=1
   CALL MINMAX(I1,J1)
   IF (ISEG.EQ.NSEG) GO TO 500
   IF (IC.EQ.0) GO TO 249
250 READ(9,1001) I2,J2,R2,Z2,I3,J3,R3,Z3,OPTION
249 IF (I2.NE.-2.AND.IC.EQ.0) GO TO 251
   IF (I2.NE.-2) GO TO 252
   READ(9,5000) IBMAX,JBMAX,NUMJ,IC,JCT+JCMAX
5000 FORMAT(16I5)
   READ(9,5000) (JJB(J),J=1,NM),NUMJ)
   II=II+1
   JJ=J
   I2=II
   J2=JJB(J)
   R2=AR(I2,J2)
   Z2=AZ(I2,J2)
   IF (OPTION.LT.0) RP(KP)=-R1
   IF (IC.EQ.0) GO TO 251
   GOTO 255
252 IF (I2.GT.IBMAX ) GO TO 255
   IF (I2.EQ.JBMAX) GO TO 253
   JJ=JJ+1
   I2=II
   J2=JJB(JJ)
   IF (I2.GE.ICT.AND.I2.LE.JCT.AND.I2.GT.JCMAX) GO TO 253
   OPTION=1
   GOTO 254
253 II=II+1
   JJ=J
   I2=II
   IF (I2.GT. IBMAX ) GO TO 255
   JJ=JJB(J)
   IF (I2.GE.ICT.AND.I2.LE.JCT.AND.I2.GT.JCMAX) GO TO 253
   OPTION=1
   GOTO 254
254 R2=AR(I2,J2)
   Z2=AZ(I2,J2)
   IC=1
   GOTO 251
255 IC=0
   IC=1
   GO TO 251
251 IF(I2 .EQ. -1) GO TO 500
    IF(IPTION.GT.0) GO TO 130
    WRITE(6*2010) I2,J2,R2,Z2,J3,R3,Z3,ITION
    IPTION=ITION+1
    GO TO (200,300,300,346,346,346,346) * IPTION
C*************************************************************************
C   GENERATE STRAIGHT LINES ON BOUNDARY
C*************************************************************************
  300 DI=ABS(FLOAT(I2-I1))
     DJ=ABS(FLOAT(J2-J1))
     ISTART=I1
     ISTOP=I2
     JSTART=J1
     JSTOP=J2
     DIFF=AMAX1(DI,DJ)
     ITEH=DIFF+1
     INC=0
     JINC=0
     IF(I2.NE.I1) IINC=(I2-I1)/IABS(I2-I1)
     IF(J2.NE.J1) JINC=(J2-J1)/IABS(J2-J1)
     KAPPA=1
     IF(I2.NE.I1 .AND. J2.NE.J1 .AND. IPTION.GT.3) KAPPA=2
     IF(KAPPA.GT.2) KAPPA=2
     NINC=(R2-R1)/DIFF
     ZINC=(Z2-Z1)/DIFF
     WRITE(6*2002) DI,DJ,NINC,DIFF,ZINC,KAPPA,ITION
C*************************************************************************
C   CHECK FOR INPUT ERROR
C*************************************************************************
   IF(IPTION.EQ.3 .AND. DI.NE.DJ) GO TO 310
   IF(KAPPA.GT.2 .OR. DI.EQ.DJ) GO TO 320
310 WRITE(6*2003)
   GO TO 200
C*************************************************************************
C   INTERPOLATE
C*************************************************************************
320 I=I1
   J=J1
   WRITE(6*2004)
   WRITE(6*2004)
   DO 340 N=1,ITER
   IF(ITEM.EQ.0 .AND. IPTION.EQ.2) GO TO 345
   IF(KAPPA.EQ.2) GO TO 330
   JOLD=I
   I=I+IINC
   JOLD=J
   J=J+JINC
   AR(I+J)=AR(I+J)+RINC
   AZ(I+J)=AZ(I+J)+ZINC
   WRITE(6*2005) I+J,AR(I+J),AZ(I+J)
   CALL MNIMX(I+J)
   IF(NCOUE(I+J).EQ.6) WRITE(6*2100) I,J
   NCODE(I+J)=2
GO TO 340
330 IOLD=I
   I=I+IINC
   AR(I+J)=AR(I+IOLD)+RINC
   AZ(I+J)=AZ(I+IOLD)+ZINC
   WRITE(*,2050) I,J,AR(I+J),AZ(I+J)
   IF(NCOUE(I,J).NE.4) WRITE(*,2100) I,J
   NODE(I,J)=2
   CALL MNIXM(I+J)
   JOLD=J
   J=J+JINC
   AR(I+J)=AR(I+JOLD)+RINC
   AZ(I+J)=AZ(I+JOLD)+ZINC
   IF(NCOUE(I+J).NE.4) WRITE(*,2100) I,J
   NODE(I+J)=2
   WRITE(*,2050) I+J,AR(I+J),AZ(I+J)
   CALL MNIXM(I+J)
340 CONTINUE
345 IF(KAPPA*EG.1) GO TO 200
   IOLD=I
   I=I+IINC
   AR(I,J)=AR(I+IOLD)+RINC
   AZ(I,J)=AZ(I+IOLD)+ZINC
   IF(NCOUE(I,J).NE.4) WRITE(*,2100) I,J
   NODE(I,J)=2
   WRITE(*,2050) I,J,AR(I,J),AZ(I,J)
   CALL MNIXM(I,J)
   GO TO 200
346 WRITE(*,4001)
   4001 FORMAT(' INPUT ERROR ----VALUE OF IPTION GREATER THEN 1
       ------EXECUTION TERMINATED')
   STOP
C********************************************************************
500 MAXI=0
   MAXJ=0
   MINI=30
   MINJ=100
   DO 503 I=1,30
      IF(JMAX(I).GT.MAXJ) MAXJ=JMAX(I)
      IF(JMIN(I).LT.MINJ) MINJ=JMIN(I)
   CONTINUE
503 DO 507 J=1,100
      IF(JMAX(J).GT.MAXI) MAXI=JMAX(J)
      IF(JMIN(J).LT.MINI) MINI=JMIN(J)
   CONTINUE
507 READ(*,1000) NOHC
   IF(NOHC.EQ.0) GO TO 511
   WRITE(*,8500)
   DO 508 NO=1,NOHC
      READ(*,8000) IMI,IMA,JMI,JMA
      IMIV(NO)=IMI
      IMAV(NO)=IMA
      JMIV(NO)=JMI
      JMNAV(NO)=JMA
   CONTINUE
508 CONTINUE

DC 508 I=IMI*IMA
508 NCODE(I,J)=5
800 FORMAT(415)
8500 FORMAT(1H0*20HVOID SPECIFICATIONS*/
1 21H IMI IMA JMI JMA )
IF(NUHC.LE.30)GO TO 511
WRITE(6*9000)
9000 FORMAT(33H0*ERRORRCC**TOO MANY VOID CARDS)
STCP
C******************************************************************************
C CALCULATE COORDINATES OF INTERIOR POINTS
C******************************************************************************
511 IF ( MAXI=1 .LE. 2 ) GO TO 530
511 I1=MINI+1
511 I2=MAXI-1
IF(NLIM.LT.1)NLIM=100
DO 520 N=1,NLIM
520 RESID=0.
DO 510 I=II,J2
510 J2=MAXA(I)-1
DO 510 0=1,J1-2
510 KODE=NCODE(I,J)
GO TO(100*510*509*506*510)*KODE
506 NCODE(I,J)=3
509 DH=AR(I+1,J)*AR(I-1,J)+AR(I,J+1)*AR(I,J-1))/4.,=AR(I,J)
1 *CONJ * (AR(I,J) + AR(I,J))/FLOAT(8*8(I*ISET))
2 * CONJ * (AR(I+1,J) + AR(I,J))/FLOAT(8*8(I*ISET))
DZ=AZ(I,1,J)*AZ(I,J)+AZ(I,J-1)*AZ(I,J))/4.,=AZ(I,J)
1 * CONJ * (AZ(I,J-1) - AZ(I,J))/FLOAT(8*8(I*ISET))
2 * CONJ * (AZ(I+1,J) - AZ(I,J))/FLOAT(8*8(I*ISET))
RESID=RESID+ABS(DR)+ABS(DZ)
AR(I,J) = AR(I,J)+1.8*DH
AZ(I,J) = AZ(I,J)+1.8*DZ
510 CONTINUE
IF(N.EQ.1) RES1=RES1
IF(N.EQ.1)ANC.RES1.EQ.0) GO TO 530
IF(NRES1/RES1.LT.1.E-4) GO TO 530
520 CONTINUE
530 WRITE(6*929) N
KP=KP-1
600 CONTINUE
999 WRITE(6*9000)
4000 FORMAT(1H0*9MEND MESH )
C******************************************************************************
C******************************************************************************
1000 FORMAT(415, 2F10.0*315)
1001 FORMAT(2(2I5,2F8.3),15)
2000 FORMAT(30H1 MESH GENERATION INFORMATION/
7 41H0 POLAR COORDINATE PARAMETER I--------------E12.4/>
8 41H0 POLAR COORDINATE PARAMETER J--------------E12.4/>
9 41H0 I CURVATURE MODIFICATION----------------I3,/
1 41H0 J CURVATURE MODIFICATION----------------I3,///)
2001 FORMAT(1M0, 81H INPUT I J R Z I2 J2 R2
1 Z IPTION )
2002 FORMAT(1M, 5H DI=F4.0, 5H DJ=F4.0, 7H DIFF=F4.0,
1 7H RINC=F8.3, 7H ZINC=F8.3, 7H ITER=1I3,
2 7H IINC=1I3, 7H JINC=1I3, 8H KAPPA=1I2)
2003 FORMAT (%X,38H**BAD INPUT--T HIS LINE IS NOT DIAGONAL)
2004 FORMAT (30H I J AR AZ)
2005 FORMAT (215,2F10.3)
2006 FORMAT (51M ** BAD INPUT - THESE POINTS DO NOT DEFINE A CIRCLE**
13X6F12.4+10X2E20+8)
2007 FORMAT(1M, 21H CENTER COORDINATE (F8.3,1H*F8.3,1H))
2008 FORMAT(1M , 7M ANG1=F9.6,7M ANG2=F9.6,7M DIFF=F3.0,
1 9H DELPHI=9F9.6)
2009 FORMAT(1M0, 30H COORDINATES CALCULATED AFTER 11I3,
1 11I ITERATIONS )
2010 FORMAT(7X,2(F14,2F8.3),16)
2100 FORMAT(54H*****WARNING***** NODAL POINT WITH (I+J) COORDINATES (,
1 I2+1M2+I2+1I M HAS BEEN RE-DEFINED)
3000 FORMAT(1D15)
3100 FORMAT(2E15.8,10)
3200 FORMAT(BF10.5)
RETURN
END
C

POINTS-- THIS ROUTINE ASSIGNS MATERIALS, TEMPERATURES, ETC.

SUBROUTINE PCINTS

COMMON/BASIC/NUMNP,NUMEL,NUMPC,NUMSC,ACELZ,ANGVEL,TRF,TREF,VOL,IFREQ
COMMON/NDATA/R(1000),CODE(1000),X(1000),Z(1000),XZ(1000),T(1000)
COMMON/ELDATA/IX(1000),EPR(1000),ALPHA(1000),PST(1000)
DOUBLE PRECISION X,Y,T,E
COMMON/SOLVE/NCODE(30),NUMTC
COMMON/TD/IMIN(100),IMAX(100),JMIN(30),JMAX(30),MAXJ,MINJ
COMMON/PLANE,NPP
COMMON/PHASE,IMIV(30),IMAV(30),JMIV(30),JMIV(30)
COMMON/OMG4/ISTOP
DIMENSION AR(30,100),AZ(30,100),MATRIL(20,5),BLKANG(20)
DIMENSION IORDER(30,100)
DIMENSION JOMIT(40),JSTART(40),JSTOP(40)
DIMENSION IOMIT(40),ISTART(40),ISTOP(40)
DATA IORDER/-2000*0/

EQUIVALENCE (R(1),AR(I,J)),(Z(1),AZ(I,J))

READ(9)*999) RSHIF#ZSHIF
4999 FORMAT(2F10.0)
READ(9*5000) JOMAX
5000 FORMAT(I5)
DO 7 J=1,JOMAX
READ(9*5001) JOMIT(J),JSTART(J),JSTOP(J)
5001 FORMAT(3I5)
7 CONTINUE
READ(9*5004) IOMAX
5004 FORMAT(I5)
DO 6 IU=1,10
READ(9*5001) IOMIT(I),LSTART(I),LSTOP(I)
READ(9*5001) IE1IEXP*IIEXP*JEXP
NP=0
J0=1
DO 120 J=MINU,JMAX
IF( J EQ J0) GO TO 90
NSTART=MIN(J)
NSTOP=IMAX(J)
GO TO 91
90 NSTART=MSTART(J)
NSTOP=MSTOP(I)
J0=J0+1
91 I0=1
DO 120 I=1,NSTART,NSTOP
120 J0=1
IF(NCUE(I,J0,ID),EQ,4) GO TO 120
IF(I-NE.IOMIT(I0)) GO TO 96
IF(J-LE.ISTART(I0)) GO TO 92
GO TO 93
92 IF(J-GE.LSTART(I0)) GO TO 95
93 I0=I0+1
GO TO 120
95 I0=I0+1
96 NP=NP+1
R(NP)=AR(I,J)
Z(NP)+AZ(I,J)
IORDER(I,J)=NP
120 CONTINUE
NUMNP=NP
C******************************************************************************************
C READ AND ASSIGN BOUNDARY CONDITIONS
C******************************************************************************************
C INITIALIZE
C
DO 150 I=1,1000
T(I)=0.0
150 PST(I)=0.0
DO 200 I=1,M,L,MNP
CODE(I)=0.
IF(M(I).EQ.0.0 AND NP.EQ.0.0) CODE(I) = 1.
AR(I)=0.
AZ(I)=0.
200 CONTINUE
C
IF(NBC.EQ.0.0) GO TO 220
DO 210 I=1,NBC
READ(9,1002) I1,I2,J1,J2,CUN,RCON,ZCON
DO 210 I=I1+12
DO 210 J=J1+2
NP=NORDER(I,J)
CODE(NP)=CUN
AR(NP)=RCON
AZ(NP)=ZCON
210
MPHINT=0
PRINT 1300,NLMNP
NP=0
J0=1
DO 240 J=MIN(J,MAXJ)
IF( J.EQ. JOMIT(J0) ) GO TO 225
NSTART=MIN(J0)
NSTOP=MAX(J)
GO TO 226
225 NSTART=MSTART(J0)
NSTOP=MSTOP(4..)
J0=J0+1
226 I0=1
DO 240 I=MSTART,NSTOP
IF( I.EQ. IOMIT(I0) ) GO TO 240
IF(I.NE.10M1(I0) ) GO TO 255
IF(I.LE.LSTOP(I0) ) GO TO 254
GO TO 257
254 IF(I.LE.LSTOP(I0) ) GO TO 255
257 I0=I0+1
GO TO 240
255 I0=I0+1
256 NP=NP+1
IF(MPRINT.NE.0) GO TO 230
WRITE(6*2000)
MPRINT=50
230 MPRINT=MPRINT+1
R(NP)=KH(NP)=RSHIF
Z(NP) = Z(NP) + Z(NP)
WRITE(6,0011) 1 + J + I, NP, CODE(NP), R(NP), Z(NP), XR(NP), XZ(NP), NCODE(I,J)
240 CONTINUE

C**********************************************************************************************
C ASSIGN MATERIALS IN BLOCKS
C**********************************************************************************************
DO 300 M = 1, 1000
300 I: (M+1) = 0
DO 310 IMLT = 1, NMTL
READ (9,1080) MTL, (MATERIAL(1MLT, IM) * IM = 2, 5), BLKANG(I MLT)
310 MATERIAL(I MLT, 1) = MTL

C**********************************************************************************************
C ESTABLISH ELEMENT INFORMATION
C**********************************************************************************************
JUMAX = MAX(J - 1)
ICODE = 1
JCODE = 1
JCODE2 = 1
NEL = 0
II = 0
II = 0
GO TO 312
311 CONTINUE
312 JJ = JJ + 1
IF (JJ .LT. JUMAX) GO TO 400
JCODE = 1
II = 0
NSTANT = MIN(JII)
NSTOP = MAX(JII)
II = NSTART
GO TO 314
313 II = II + 1
IF (II .GE. NSTCP) GO TO 311
314 IF (IUMUXER(II+JJ) .eq. 0) GO TO 313
II = II + 1
JJP = JJ + 1
IF (NCODE(II+JJ) .eq. 4) GO TO 317
IF (NCOUE(IIP+JJ) .eq. 4) GO TO 317
IF (NCOUE(IIP+JJ) .eq. 4) GO TO 317
IF (JJP .GE. JJEXP) AND II .GE. IIEXP) GO TO 316
IF (NCOUE(IIP+JJ) .eq. 5) GO TO 318
IF (NCOUE(IIP+JJ) .eq. 5) GO TO 318
IF (NCOUE(IIP+JJ) .eq. 5) GO TO 318
DO 315 NNO = 1, NNOH
IF (II .GE. IMIV(NO) AND II .LE. IMAV(NO) AND
1 JJ .GE. JIMIV(NO) AND JJ .LE. JMIV(NO) AND
2 JJP .GE. JIMIV(NO) AND JJP .LE. JMIV(NO) AND
3 JJP .GE. JMIV(NO) AND JJP .LE. JMIV(NO)) GO TO 313
315 CONTINUE
316 JJCODE = 2
GO TO 318
317 JJCODE = 2
GO TO 313
318 CONTINUE
C CHECK FOR DIRECTION OF EXPANSION
IF(JJCODE.EQ.2) GO TO 320
319 CONTINUE
IF(IORDER(II*JJ)*GT.0.AND.IORDER(IIP*JJP)*GT.0) GO TO 333
IF(IORDER(II*JJ)*GT.0.AND.IORDER(IIP*JJP)*GT.0) GO TO 321
WRITE(*,6000) II,JJ,IIP,JJP
6000 FORMAT('***ERROR*** POINTS* NOT* DEFINE* A* VALID
* EXPANSION EXECUTION TERMINATED')
C TOP
320 CONTINUE
IF( JJ .GE. IEXPAN ) GO TO 319
IF(IORDER(IIP*JJ)*EQ.0 ) GO TO 313
IF(IORDER(IIP*JJP)*EQ.0) GO TO 321
GO TO 319
321 CONTINUE
C EXPANSION IN J-DIRECTION
IF(IORDER(IIP*JJ)*EQ.0 .AND.IORDER(IIP*JJP)*EQ.0) GO TO 331
IF(IORDER(IIP*JJP)*GT.0.AND.IORDER(IIP*JJP)*GT.0) GO TO 329
IF(IORDER(IIP*JJP)*EQ.0) GO TO 322
IITEMP=II
JJTEMP=JJP
JCODE=3
GO TO 331
322 IITEMP=IIP
JJTEMP=JJP
JCODE=2
GO TO 331
323 CONTINUE
C TRIANGULAR ELEMENTS
IA=IORDER(IIP*JJ)
IB=IORDER(IIP*JJ)
IC=IORDER(IITEMP*JJTEMP)
ID=IC
GO TO 332
324 IA=IORDER(IIP*JJ)
IB=IORDER(IITEMP*JJTEMP)
IC=IORDER(IIP*JJP)
ID=IC
JCODE=4
GO TO 332
325 IA=IORDER(IIP*JJ)
IB=IORDER(IITEMP*JJTEMP)
IC=IORDER(IIP*JJP)
ID=IC
JCODE=1
GO TO 332
326 IA=IORDER(IIP*JJ)
IB=IORDER(IIP*JJ)
IC=IORDER(IITEMP*JJTEMP)
ID=1C
GO TO 332
327 IA=IORDER(IIP*JJ)
IB=IORDER(IIP*JJP)
IC=IORDER(IITEMP*JJTEMP)
ID=IC
JCODE=5
GO TO 332
328 IA=IORUER(IIF+JJP)
    IB=IORUER(IIF+JJP)
    IC=IORUER(IITEMP+JU+TEMP)
    ID=IC
    JCODE=1
    GO TO 332
329 GO TO (330,323,326) +JCODE
330 IA=IORUER(IJ+JJ)
    IB=IORUER(IIF+JJ)
    IC=IORUER(IIF+JJP)
    ID=IORUER(IJ+JJ)
    GO TO 332
331 JJP=JJP+1
    IF(JJP.GT.MAXJ) CALL ERROR(1,JJP,MAXJ,II)
    GO TO 321
332 NEL=NEL+1
    IF(NEL.GT.1000) CALL ERROR(2,NEL,1000,0)
    IX(NEL+1)=IA
    IX(NEL+2)=IB
    IX(NEL+3)=IC
    IX(NEL+4)=ID
    IX(NEL+5)=MTL
    ALPHANEL=BLKANG(WTL)
    GO TO (345,342,341,327,325,328) +JCODE
333 CONTINUE
334 IITEMP=IIP
    JJTEMP=JJP
    JCODE=2
    GO TO 337
335 GO TO (336,342,339) +ICODE
336 IA=IORUER(IJ+JJ)
    IB=IORUER(IIF+JJ)
    IC=IORUER(IIF+JJP)
    ID=IORUER(IJ+JJ)
    I=IIP+1
    GO TO 338
337 IIP=IIP+1
    IF(IIP.GT.NSTOP) CALL ERROR(3,IIP,NSTOP,JJ)
    GO TO 333
338 NEL=NEL+1
    IF(NEL.GT.1000) CALL ERROR(2,NEL,1000,0)
    IX(NEL+1)=IA
    IX(NEL+2)=IB
    IX(NEL+3)=IC
    IX(NEL+4)=ID
    IX(NEL+5)=MTL
    ALPHANEL=BLKANG(WTL)
GO TO ( 345+343+340+341+344 ), ICODE

339 CONTINUE

C TRIANGULAR ELEMENTS
IA=IORDER(I$+JJ)
IB=IORDER(IITEMP$+JUTEMP)
IC=IORDER($+JJP)
ID=IC
GO TO 338

340 IA=IORDER(I$+JJ)
IB=IORDER(I$+JJP)
IC=IORDER('ITEMP$+JUTEMP)
ID=IC
ICODE=4
GO TO 338

341 IA=IORDER(I$+JJ)
IB=IORDER(I$+JJP)
IC=IORDER(IITEMP$+JUTEMP)
ID=IC
II=IIP-1
ICODE=1
GO TO 338

342 IA=IORDER(I$+JJ)
IB=IORDER(I$+JJP)
IC=IORDER(IITEMP$+JUTEMP)
ID=IC
GO TO 338

343 IA=IORDER(I$+JJ)
IB=IORDER(I$+JJP)
IC=IORDER(IITEMP$+JUTEMP)
ID=IC
ICODE=5
GO TO 338

344 IA=IORDER(I$+JJ)
IB=IORDER(I$+JJP)
IC=IORDER(IITEMP$+JUTEMP)
ID=IC
II=IIP-1
ICODE=1
GO TO 338

345 GO 360 INTL=1, NMTL
IF (II$LT.MATFL(IMTL+2)) GO TO 360
IF (II$GE.MATFL(IMTL+3)) GO TO 360
IF (JJ$LT.MATFL(IMTL+4)) GO TO 360
IF (JJ$GE.MATFL(IMTL+5)) GO TO 360
NMTL = MATFL(IMTL+1)
GO TO 338

360 CONTINUE
PRINT '100 *NEL$II$JJ
NMTL=1
380 IX(NELS)$MNTL
ALPHA(NELS)=BLKANG%MNTL
GO TO 313
400 CONTINUE
NUMEL=NEL
480 IF (NUMEL$GT.1000) WRITE(6,2002)

C-------------------------------------------------------------------------
SET NOUAL POINT TEMPERATURE TO REFERENCE TEMPERATURE

IF(NUMTC.A.E.0)GO TO 550
DO 500 N=1,NLMNP
500 T(N)=TREF
550 WRITE(6*4000)
4000 FORMAT(1H * 10MEND POINTS )
RETURN

C
1000 FORMAT(515*F10.0)
1002 FORMAT(415*3F10.0)
1300 FORMAT(1MO, 40MNODAL POINT DATA---NO. OF NODAL POINTS= *IS)
2000 FORMAT(104H1 I J NP TYPE R-ORDINATE Z-ORDINATE
1TE H LOAD OR DISPLACEMENT Z LOAD OR DISPLACEMENT)
2002 FORMAT(35H BAD INPUT - TOO MANY NODAL POINTS)
2100 FORMAT(1MO, 8HELEMENT *I4,3X,23MH WITH (I*J) COORDINATES(*I2+1H*,*I3*
1 31M) HAS BEEN ASSIGNED MATERIAL 1 )
END
C     DATA--- THIS ROUTINE READS CARDS AND PUTS THEM ON TAPE NO. 9

SUBROUTINE DATA
COMMON/OMG5/, JSTOP
DIMENSION CARD(20)
REWIND 9
N=0
IF ( JSTOP .EQ. 1 ) GO TO 400
READ(5,1000) CARD
10 WRITE(6,1002)
    GO TO 200
100 READ(5,1000,END=300) CARD
200 WRITE(6,1003) CARD
WRITE(6,1000) CARD
   N=N+1
    GO TO 100
300 ENDFILE 9
REWIND 9
WRITE(6,1004)
JSTOP=1
RETURN

400 WRITE(6,1005)
STOP
1000 FORMAT(20A4)
1002 FORMAT(1H1)
1003 FORMAT(5X,20A4)
1004 FORMAT(1H , 11MEND OF DATA)
1005 FORMAT(1H , 10MEND OF JOB)
END
C     THIS IS A UTILITY ROUTINE
SUBROUTINE MNINX(I,J)
C
COMMON/TO/IMIN(100),IMAX(100),JMIN(30),JMAX(30),MAXI,MAXJ,NMTL,NBC
   *MINI,MINJ
C
   IF(J.LT.JMIN(I)) JMIN(I)=J
   IF(J.GT.JMAX(I)) JMAX(I)=J
   IF(I.LT.IMIN(J)) IMIN(J)=I
   IF(I.GT.IMAX(J)) IMAX(J)=I
C
RETURN
END
C NODE -- A SMALL UTILITY ROUTINE WHICH MAY BE NEEDED BY MESH OR POINTS

FUNCTION NODE(I,J)

COMMON/TO/IMIN(100),IMAX(100),JMIN(30),JMAX(30),MAXI,MAXJ,NMTL,NBC
1  MINJ

C

NODE=0
DO 100 JJ=1,JMAX
NSTART=IMIN(JJ)
NSTOP=IMAX(JJ)
DO 100 II=NSTART,NSTOP
    NODE=NODE+1
    IF(JJ.EQ.J.AND.II.EQ.I) RETURN
100 CONTINUE
C

RETURN
END
SUBROUTINE ERROR(NM*L*K)
  GO TO (1*2*3)*N
1  WRITE(b+10) *,L*K
10  FORMAT(* ,**JPM = **I5,**EXCEEDS JM = **I5,II = **I5)
    GO TO 50
2  WRITE(b+11) *,L
11  FORMAT(* ,**JEL = **I5,**EXCEEDS MANEL = **I5)
    GO TO 50
3  WRITE(b+12) *,L*K
12  FORMAT(* ,**II = **I5,**EXCEEDS NSTOP = **I5,**JJ = **I5)
50  WRITE(b+13)
13  FORMAT(* ,**EXECUTION TERMINATED*)
STOP
END
DISTRIBUTION

Commander (NSEA 09G32)
Naval Sea Systems Command
Department of the Navy
Washington, DC  20362

Copies 1 and 2

Commander (NSEA 0342)
Naval Sea Systems Command
Department of the Navy
Washington, DC  20362

Copies 3 and 4

Defense Documentation Center
5010 Duke Street
Cameron Station
Alexandria, VA 22314

Copies 5 through 16

Via: Commander (NSEA 09G32)
Naval Sea Systems Command
Department of the Navy
Washington, DC  20362