TWO AND THREE-DIMENSIONAL FINITE ELEMENT ANALYSIS OF
PLASTICITY-INDUCED FATIGUE CRACK CLOSURE
–A COMPREHENSIVE PARAMETRIC STUDY

By
Kiran Nainmal Solanki

A Thesis
Submitted to the Faculty of
Mississippi State University
in Partial Fulfillment of the Requirements
for the Degree of Master of Science
in Mechanical Engineering
in the Department of Mechanical Engineering

Mississippi State, Mississippi
December 2002
TWO AND THREE-DIMENSIONAL FINITE ELEMENT ANALYSIS OF
PLASTICITY-INDUCED FATIGUE CRACK CLOSURE
– A COMPREHENSIVE PARAMETRIC STUDY

By
Kiran Nainmal Solanki

Approved:

Steven R. Daniewicz
Associate Professor of Mechanical Engineering (Director of Thesis)

James C. Newman, Jr.
Professor of Aerospace Engineering (Committee Member)

E. William Jones
Professor of Mechanical Engineering (Committee Member)

James C. Newman, III
Associate Professor of Aerospace Engineering and Engineering Mechanics (Committee Member)

Rogelio Luck
Graduate Coordinator of the Department of Mechanical Engineering

A. Wayne Bennett
Dean of the College of Engineering
Finite element analyses are frequently used to model growing fatigue cracks and the associated plasticity-induced crack closure. Two-dimensional, elastic-perfectly plastic finite element analyses of middle-crack tension (M(T)), bend (SEB), and compact tension (C(T)) geometries were conducted to study fatigue crack closure and to calculate the crack opening values under plane-strain and plane-stress conditions. The loading was selected to give the same maximum stress intensity factor in both geometries, and thus similar initial forward plastic zone sizes. Mesh refinement studies were performed on all geometries with various element types. For the C(T) geometry, negligible crack opening loads under plane-strain conditions were observed. In contrast, for the M(T) specimen, the plane-strain crack opening stresses were found to be significantly larger. This difference was shown to be a consequence of in-plane constraint. Under plane-stress conditions, it was found that the in-plane constraint has negligible effect, such that the
opening values are approximately the same for the C(T), SEB, and M(T) specimens. Next, the crack opening values of the C(T), SEB and M(T) specimens were compared under various stress levels and load ratios. The effect of a highly refined mesh on crack opening values was noted and significantly lower crack opening values than those reported in literature were found. A new methodology is presented to calculate crack opening values in planar geometries using the crack surface nodal force distribution under minimum loading as determined from finite element analyses. The calculated crack opening values are compared with values obtained using finite element analysis and more conventional crack opening assessment methodologies. It is shown that the new method is independent of loading increment, integration method (normal and reduced integration), and crack opening assessment location. The compared opening values were in good agreement with strip-yield models.
ACKNOWLEDGEMENTS

I would like to express sincere gratitude to the many people without whose selfless assistance this thesis could not have materialized. First, I would like to thank Dr. Steven R. Daniewicz, my major advisor, for his tireless efforts in invaluable guidance and assistance throughout this research and for opening the door to many professional opportunities along the way. Next, expressed appreciation is also due to Dr. James C. Newman, Jr. of Aerospace Engineering Department, for his magnanimity in expending time and effort in guiding me towards a qualitative methodology. I would also like to acknowledge my other committee members Dr. E. William Jones and Dr. James C. Newman, III for their support and suggestions throughout my tenure. Finally, I would like to thank Dr. Robert H. Dodds, Jr. of the University of Illinois at Urbana-Champaign for making available the computer program for the generation of the surface crack meshes used in this study.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>ACKNOWLEDGEMENTS</th>
<th>ii</th>
</tr>
</thead>
<tbody>
<tr>
<td>LIST OF FIGURES</td>
<td>vi</td>
</tr>
<tr>
<td>CHAPTER</td>
<td></td>
</tr>
<tr>
<td>I.  INTRODUCTION</td>
<td></td>
</tr>
<tr>
<td>1-1 Background</td>
<td>1</td>
</tr>
<tr>
<td>1-2 Fatigue Crack Propagation</td>
<td>2</td>
</tr>
<tr>
<td>1-3 Fatigue Crack Closure</td>
<td>4</td>
</tr>
<tr>
<td>1-4 Crack Tip Nomenclature</td>
<td>6</td>
</tr>
<tr>
<td>II. LITERATURE REVIEW</td>
<td>8</td>
</tr>
<tr>
<td>2-1 Two-dimensional Finite Element Modeling Issues</td>
<td>8</td>
</tr>
<tr>
<td>2-1-1 Crack Surface Contact</td>
<td>8</td>
</tr>
<tr>
<td>2-1-2 Mesh Refinement</td>
<td>10</td>
</tr>
<tr>
<td>2-1-3 Stabilization of Crack Opening Load</td>
<td>12</td>
</tr>
<tr>
<td>2-1-4 Crack Advance Scheme</td>
<td>13</td>
</tr>
<tr>
<td>2-1-5 Crack Opening Assessment Location</td>
<td>15</td>
</tr>
<tr>
<td>2-1-6 Variable Amplitude Loading</td>
<td>16</td>
</tr>
<tr>
<td>2-1-7 Plane-stress and Plane-strain Condition</td>
<td>16</td>
</tr>
<tr>
<td>2-1-8 Geometry Effects</td>
<td>18</td>
</tr>
<tr>
<td>2-1-9 $R$ Ratio Effects</td>
<td>21</td>
</tr>
<tr>
<td>2-2 Three-dimensional Modeling Aspects</td>
<td>23</td>
</tr>
<tr>
<td>2-2-1 Mesh Refinement</td>
<td>25</td>
</tr>
<tr>
<td>2-2-2 Crack Advance Scheme</td>
<td>26</td>
</tr>
<tr>
<td>2-2-3 Aspect Ratio Evolution</td>
<td>27</td>
</tr>
<tr>
<td>2-2-4 Influence of Loading History</td>
<td>27</td>
</tr>
<tr>
<td>2-3 Element Types and Configurations</td>
<td>28</td>
</tr>
<tr>
<td>2-4 Material Model Effects</td>
<td>30</td>
</tr>
<tr>
<td>2-5 Overview of Thesis</td>
<td>32</td>
</tr>
<tr>
<td>III. FINITE ELEMENT ANALYSIS</td>
<td>34</td>
</tr>
<tr>
<td>3-1 Two-dimensional Finite Element Modeling Issues</td>
<td>35</td>
</tr>
<tr>
<td>3-2 Three-dimensional Finite Element Analysis</td>
<td>43</td>
</tr>
<tr>
<td>CHAPTER</td>
<td>Page</td>
</tr>
<tr>
<td>---------</td>
<td>------</td>
</tr>
<tr>
<td>IV. FINITE ELEMENT RESULT</td>
<td>46</td>
</tr>
<tr>
<td>4-1 Two-dimensional Finite Element Analysis</td>
<td>46</td>
</tr>
<tr>
<td>4-1-1 Geometry Effects on Closure under Plane-strain Condition</td>
<td>46</td>
</tr>
<tr>
<td>4-1-2 Effect of Load Ratios and Stress Levels under Plane-stress</td>
<td>61</td>
</tr>
<tr>
<td>4-2 Three-dimensional Finite Element Analysis</td>
<td>67</td>
</tr>
<tr>
<td>IV. CONTACT STRESS METHOD FORMULATION</td>
<td>72</td>
</tr>
<tr>
<td>5-1 Formulation</td>
<td>73</td>
</tr>
<tr>
<td>5-2 Result</td>
<td>80</td>
</tr>
<tr>
<td>VI. CONCLUSION</td>
<td>86</td>
</tr>
<tr>
<td>REFERENCES</td>
<td>89</td>
</tr>
<tr>
<td>APPENDIX</td>
<td></td>
</tr>
<tr>
<td>A.1 ANSYS INPUT FILE APPBCS.MAC</td>
<td>97</td>
</tr>
<tr>
<td>A.2 ANSYS INPUT FILE StrtCyc.MAC</td>
<td>102</td>
</tr>
<tr>
<td>A.3 ANSYS INPUT FILE FIRSTLOAD.MAC</td>
<td>104</td>
</tr>
<tr>
<td>A.4 ANSYS INPUT FILE ADVANCECRACK.MAC</td>
<td>106</td>
</tr>
<tr>
<td>A.5 ANSYS INPUT FILE UNLOADCRACK.MAC</td>
<td>109</td>
</tr>
<tr>
<td>A.6 ANSYS INPUT FILE ContactStress.MAC</td>
<td>112</td>
</tr>
<tr>
<td>A.7 ANSYS INPUT FILE LOADCRAK.MAC</td>
<td>117</td>
</tr>
<tr>
<td>A.8 ANSYS INPUT FILE SELECTNODES.MAC</td>
<td>121</td>
</tr>
<tr>
<td>A.9 ANSYS INPUT FILE CLEARST.MAC</td>
<td>123</td>
</tr>
<tr>
<td>A.10 ANSYS INPUT FILE APPLLOAD.MAC</td>
<td>125</td>
</tr>
<tr>
<td>A.11 ANSYS INPUT FILE APPLLOAD.MAC, T-STRESS</td>
<td>127</td>
</tr>
<tr>
<td>A.12 ANSYS INPUT FILE LOADCRAK.DAT, T-STRESS</td>
<td>129</td>
</tr>
<tr>
<td>A.13 ANSYS INPUT FILE UNLOADCRCAK.DAT, T-STRESS</td>
<td>132</td>
</tr>
<tr>
<td>A.14 ANSYS INPUT FILE StrtCyc.MAC</td>
<td>136</td>
</tr>
<tr>
<td>B.1 ANSYS INPUT FILE SEB.DAT</td>
<td>138</td>
</tr>
<tr>
<td>B.2 ANSYS INPUT FILE MT.DAT</td>
<td>141</td>
</tr>
<tr>
<td>B.3 ANSYS INPUT FILE MT.DAT, T-STRESS</td>
<td>144</td>
</tr>
<tr>
<td>C.1 ANSYS INPUT FILE APPBCS.MAC, SPIKE OVERLOAD</td>
<td>147</td>
</tr>
<tr>
<td>C.2 ANSYS INPUT FILE StrtCyc.MAC, SPIKE OVERLOAD</td>
<td>150</td>
</tr>
</tbody>
</table>
### APPENDIX

| C.3 | ANSYS INPUT FILE RAJU.MAC, SPIKE OVERLOAD | 154 |
| C.4 | ANSYS INPUT FILE SELCTNODES.MAC, SPIKE OVERLOAD | 156 |
| C.5 | ANSYS INPUT FILE LOADCRACK.MAC, SPIKE OVERLOAD | 159 |
| C.6 | ANSYS INPUT FILE UNLOADCRACK.MAC, SPIKE OVERLOAD | 169 |
| C.7 | ANSYS INPUT FILE STRTCYC.MAC, CSE | 173 |
| C.8 | ANSYS INPUT FILE SELCTNODES.MAC, CSE | 176 |
| C.9 | ANSYS INPUT FILE LOADCRACK.MAC, CSE | 179 |
| C.10 | ANSYS INPUT FILE UNLOADCRACK.MAC, CSE | 186 |
| C.11 | ANSYS INPUT FILE SURFACECRACK.MAC, CSE | 190 |
| C.12 | ANSYS INPUT FILE SURFACECRACKSPIKE.MAC, SPIKE OVERLOAD | 193 |
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>FIGURE</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1-1</td>
<td>Typical Fatigue Crack Growth Behavior in Metals</td>
<td>3</td>
</tr>
<tr>
<td>1-2</td>
<td>Typical Fatigue Crack Closure in Metals</td>
<td>4</td>
</tr>
<tr>
<td>1-3</td>
<td>Definition of Effective Stress Intensity Range</td>
<td>5</td>
</tr>
<tr>
<td>1-4</td>
<td>Plastic Deformation Around a Growing Crack</td>
<td>6</td>
</tr>
<tr>
<td>2-1</td>
<td>Mesh Refinement Studies</td>
<td>10</td>
</tr>
<tr>
<td>2-2</td>
<td>Stabilization of Crack Opening Values under Plane-stress and Plane-strain</td>
<td>12</td>
</tr>
<tr>
<td>2-3</td>
<td>Comparison of Crack Opening Values Based on Crack Advance Scheme</td>
<td>14</td>
</tr>
<tr>
<td>2-4</td>
<td>Effect of $T$-stress on the Crack Opening Value Stabilization</td>
<td>20</td>
</tr>
<tr>
<td>2-5</td>
<td>Effect of $T$-stress under Plane-stress</td>
<td>21</td>
</tr>
<tr>
<td>2-6</td>
<td>Effect of Stress Ratio on Crack Opening Values under Plane-stress</td>
<td>22</td>
</tr>
<tr>
<td>2-7</td>
<td>Surface Crack Mesh Refinement Studies</td>
<td>26</td>
</tr>
<tr>
<td>2-8</td>
<td>Typical Elements and Configuration</td>
<td>29</td>
</tr>
<tr>
<td>2-9</td>
<td>Effect of Strain Hardening</td>
<td>31</td>
</tr>
<tr>
<td>3-1</td>
<td>Distribution of Mesh Refinement Levels under Plane-strain</td>
<td>38</td>
</tr>
<tr>
<td>3-2</td>
<td>Distribution of Mesh Refinement Levels under Plane-stress</td>
<td>39</td>
</tr>
<tr>
<td>3-3</td>
<td>(a) Typical Middle-crack Tension Model, (b) Typical Compact Tension Model, (c) Typical Side Edge Bend Model</td>
<td>40</td>
</tr>
<tr>
<td>3-4</td>
<td>Typical Surface Crack Mesh</td>
<td>44</td>
</tr>
<tr>
<td>FIGURE</td>
<td>Description</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
<td>------</td>
</tr>
<tr>
<td>4-1</td>
<td>Crack Tip Plastic Deformation for Growing Crack</td>
<td>47</td>
</tr>
<tr>
<td>4-2</td>
<td>Variation in the Plastic Zone Sizes with Mesh Discretization</td>
<td>48</td>
</tr>
<tr>
<td>4-3</td>
<td>(a) Comparison of Calculated Crack Opening Values under Plane-strain (M(T) Specimen), (b) Comparison of Calculated Crack Opening Values under Plane-strain (C(T) Specimen)</td>
<td>49</td>
</tr>
<tr>
<td>4-4</td>
<td>Typical Crack Opening Load Transient Behavior</td>
<td>51</td>
</tr>
<tr>
<td>4-5</td>
<td>(a) Effect of Load and Large-scale Deformation, (b) Effect of Node-released Schemes</td>
<td>53</td>
</tr>
<tr>
<td>4-6</td>
<td>Comparison of Calculated Crack Opening Values under Plane-stress</td>
<td>55</td>
</tr>
<tr>
<td>4-7</td>
<td>Crack Opening Process</td>
<td>57</td>
</tr>
<tr>
<td>4-8</td>
<td>Effect of Assessment Location</td>
<td>58</td>
</tr>
<tr>
<td>4-9</td>
<td>Effect of T-stress</td>
<td>60</td>
</tr>
<tr>
<td>4-10</td>
<td>Effect of T-stress on the Crack Opening Process</td>
<td>61</td>
</tr>
<tr>
<td>4-11</td>
<td>Variation in the Plastic Zone Sizes with Mesh Discretization under Plane-stress</td>
<td>62</td>
</tr>
<tr>
<td>4-12</td>
<td>(a) Comparison of the Forward Plastic Zone Size – M(T) Specimen, (b) Comparison of the Forward Plastic Zone Size – SEB Specimen</td>
<td>63</td>
</tr>
<tr>
<td>4-13</td>
<td>Comparison of Calculated Crack Opening Values under Plane-stress</td>
<td>65</td>
</tr>
<tr>
<td>4-14</td>
<td>Typical Crack Opening Value Transient Behavior under Plane-stress</td>
<td>65</td>
</tr>
<tr>
<td>4-15</td>
<td>Effect of Stress Ratio on Crack Opening Values under Plane-stress</td>
<td>66</td>
</tr>
<tr>
<td>4-16</td>
<td>Surface Crack Mesh Refinement Studies</td>
<td>68</td>
</tr>
<tr>
<td>4-17</td>
<td>Effect of Crack Advance Scheme</td>
<td>69</td>
</tr>
<tr>
<td>4-18</td>
<td>Surface Flaw Crack Opening Stress Transient</td>
<td>70</td>
</tr>
<tr>
<td>4-19</td>
<td>Predicted Crack Shape Evolution under Constant Amplitude Loading</td>
<td>71</td>
</tr>
<tr>
<td>5-1</td>
<td>Infinite Plane with a Semi-infinite Crack Subjected to a Segment of Linearly Varying Stress</td>
<td>74</td>
</tr>
<tr>
<td>FIGURE</td>
<td>Description</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>-----------------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>5-2</td>
<td>Infinite Plane with a Finite Crack Subjected to a Segment of Linearly Varying Stress</td>
<td>76</td>
</tr>
<tr>
<td>5-3</td>
<td>Semi-infinite Plane with a Edge Crack Subjected to a Segment of Linearly Varying Stress</td>
<td>78</td>
</tr>
<tr>
<td>5-4</td>
<td>Comparison of Predicted Crack Opening Values under Plane-strain Conditions</td>
<td>81</td>
</tr>
<tr>
<td>5-5</td>
<td>Comparison of predicted crack opening values using contact stress method and conventional method under plane-stress conditions</td>
<td>82</td>
</tr>
<tr>
<td>5-6</td>
<td>Nodal Force Distribution along the Crack Surface under Plane-strain Conditions</td>
<td>83</td>
</tr>
<tr>
<td>5-7</td>
<td>Nodal Force Distribution along the Crack Surface under Plane-stress Conditions</td>
<td>84</td>
</tr>
<tr>
<td>5-8</td>
<td>Effect of Stress Ratio on Crack Opening Values under Plane-stress</td>
<td>85</td>
</tr>
</tbody>
</table>
CHAPTER I
INTRODUCTION

1-1 Background

Fatigue has been defined as “the progressive localized permanent structural change that occurs in a material subjected to repeated or fluctuating strains at stresses having a maximum value less that the tensile strength of the material” [1].

Many different mechanical failure modes exist. These failures can occur in simple, complex, inexpensive, or expensive components or structures. It has been estimated that between 50 and 90 % of these failures are due to fatigue [2]. Failures due to fatigue culminate in cracks (or) fracture after a sufficient number of fluctuations of load.

Fracture of a structural member due to repeated cycles of load is commonly referred to as a \textit{fatigue failure} or \textit{fatigue fracture}. The corresponding number of load cycles or the time during which the member is subjected to these loads before fracture occurs is referred to as the \textit{fatigue life} of the member. The fatigue life of a member is affected by many factors [1]. For example, it is affected by (1) the type of load (uniaxial, bending, torsion), (2) the nature of the load-displacement curve (linear, nonlinear), (3) the frequency of load repetitions or cycling, (4) the load history (cyclic loading with constant or variable amplitude), (5) the size of the member, (6) the material flaws, (7) the manufacturing method (surface roughness, notches), (8) the operating temperature (high temperature that results in creep, low temperature that results in brittleness), (9) the
environmental operating conditions (corrosion) [1]. In practice, accurate estimates of fatigue life are difficult to obtain, because for many materials, small changes in these conditions may strongly affect fatigue life. The designer may therefore be forced to rely on testing of full-scale members under in-service conditions. However, testing of full-scale members is time-consuming and costly. Therefore data from laboratory tests are often used to establish fatigue failure criteria.

1-2 Fatigue Crack Propagation

The total period of fatigue life may be considered to consist of three phases: (1) initial fatigue damage that produces crack initiation, (2) propagation of a crack or cracks that results in partial separation of a cross section of a member, until the remaining uncracked cross section unable to support the applied load, and (3) final fracture of the member. The typical log-log plot of $da/dN$ versus $\Delta K$ is shown schematically in Figure 1-1. The sigmoidal shape can be divided into three major regions. Region I is the near threshold region and exhibits a threshold value, $\Delta K_{th}$, below which there is no observable crack growth. Below $\Delta K_{th}$, fatigue cracks are characterized as nonpropagating cracks. Microstructure, mean stress, frequency, and environment primarily control region I crack growth. Region II shows essentially a linear relationship between $\log da/dN$ and $\log \Delta K$, which corresponds to the formula

$$\frac{da}{dN} = C(\Delta K)^m$$

(1)
first suggested by Paris et al.[3]. Here $m$ and $C$ are material constants. Region II (Paris region) fatigue crack growth corresponds to stable macroscopic crack growth that is typically controlled by environment. Microstructure and mean stress have less influence on fatigue crack growth behavior in region II than region I. In the region III the fatigue crack growth rates are very high as they approach instability, and little fatigue crack growth life is involved. This region is controlled primarily by fracture toughness $K_c$, which in turn depends on the microstructure and environment.

![Figure 1-1 Typical Fatigue Crack Growth Behavior in Metals](image-url)
1-3 Fatigue Crack Closure

The phenomenon of plasticity-induced crack closure was first proposed and investigated by Elber [4], and led to new concepts in fatigue crack growth. Since then, several additional closure mechanisms have been identified [2-6], but the primary mechanism under many conditions is plasticity. During loading, large tensile plastic strains are developed near the crack tip, which are not fully reversed upon unloading as the crack extends. This leads to the formation of a plastic wake with plastic deformation in a direction normal to the advancing crack.

Roughness and oxide induced fatigue crack closure are predominate in the near threshold crack growth regime. These two mechanisms are similar to plasticity-induced fatigue crack closure in that the material in the wake region contacts while under tensile loading. Roughness-induced fatigue crack closure occurs when the crack growth is not planer and the mixed-mode loading at the kinked crack tip causes a mismatch of the wake region material. For oxide-induced fatigue crack closure, an oxide film forms on the surface in the wake region and makes contact while under tensile loads.

![Figure 1-2 Typical Fatigue Crack Closure in Metals](image-url)
Elber postulated that crack closure decreases the fatigue crack growth rate by reducing the effective stress intensity range. Figure 1-3 illustrates the closure concept. When a specimen is cyclically loaded between $K_{\text{max}}$ and $K_{\text{min}}$, the crack faces are in contact below $K_{\text{open}}$, the stress intensity at which the crack fully opens. Elber assumed that the portion of the cycle that is below $K_{\text{open}}$ does not contribute to fatigue crack growth. He defined an effective stress intensity range as follows:

$$\Delta K_{\text{eff}} = K_{\text{max}} - K_{\text{open}}$$  \hspace{1cm} (2)

Elber then proposed a modified Paris equation:

$$\frac{da}{dN} = C (\Delta K_{\text{eff}})^m$$  \hspace{1cm} (3)

![Figure 1-3  Definition of Effective Stress Intensity Range](image-url)
1-4 Crack Tip Nomenclature

As a fatigue crack propagates, two different types of crack tip plastic zones are generated as shown in Figure 1-4. The forward plastic zone is defined as the material near the crack tip undergoing plastic deformation at the maximum load. The second zone of interest is the reversed plastic zone, which is defined as the material near the crack tip undergoing compressive yielding at the minimum load [6]. These crack tip plastic zones will be used to characterize the degree of finite element mesh refinement later.

![Figure 1-4 Plastic Deformation Around a Growing Crack](image)

The nature of plastic deformation near the crack tip is strongly influenced by the 2-D idealization assumed. The permanent elongation of material in the direction normal
to the crack requires the transfer of material from somewhere in the cracked body due to incompressibility requirements during plastic deformation [7]. Under plane-stress, a potential mechanism of material transfer is obvious. Since out-of-plane deformation is not constrained, material can be transferred from the thickness direction to the axial direction [7]. However, the mechanism of material transfer postulated for plane-stress is not admissible for plane-strain. By definition, no net out-of-plane contraction can occur, and therefore it has been suggested that there can be no net axial stretch of material in the plastic wake behind the crack tip as discussed by Fleck [8], which implies no plasticity-induced crack closure. The existence of plasticity-induced crack closure under plane-strain conditions has been a topic of intense debate [9-27].

Many researchers have performed finite element analyses simulating plasticity-induced fatigue crack closure, considering different two-dimensional configurations under plane-strain or plane-stress conditions [7-52]. Far fewer efforts have been directed toward the three-dimensional problem [53-63]. Newman [5] and McClung [64] have presented general reviews in their respective papers.

In this research work, a detailed and comprehensive finite element analysis of plasticity-induced fatigue crack closure for both planer geometries and three-dimensional geometries is performed. Emphasis is focused on the difficulties in modeling with respect to mesh refinement level, crack advancement schemes, crack opening assessment location, crack shape evolution, overload effects, and opening value assessment techniques.
CHAPTER II
LITERATURE REVIEW

Finite element analysis of plasticity-induced fatigue crack growth is conceptually simple. A mesh is created with an initial crack, and the mesh is loaded by remotely applied tractions. For constant amplitude loading, the loading is cycled between a maximum applied stress $S_{\text{max}}$ and the minimum applied stress $S_{\text{min}}$. During the cyclic loading the crack is advanced in some fashion, leading to the formation of a plastic wake behind the crack tip. This modeling concept is simple; however, there are several issues which must be addressed during the fatigue crack growth simulation. These issues have been summarized and categorized into different sections in this chapter.

2-1 Two-dimensional Finite Element Modeling Issues

2-1-1 Crack Surface Contact

A changing boundary condition characterizes a crack under cyclic loading. In order to prevent the crack surfaces from penetrating as the minimum load is approached, some mechanism must be implemented into the finite element simulation. This can be achieved by changing the stiffness of spring elements attached to the crack surface, by removing or imposing crack surface nodal constraints, by using truss elements on the crack surface, or by using contact elements. The implementation of contact
elements along the crack surface, however, can lead to convergence problems and long execution times [60].

Newman [48] was the first to implement spring (truss) elements to simulate the changing boundary condition. The element was connected to each boundary node on the crack surface. For open nodes, the spring stiffness was set equal to zero, and for closed nodes, the stiffness was assigned a large value. McClung et al. [7,14,30,31,42-44,49,50] followed Newman’s approach in their earlier studies. However, the large imposed stiffness values for constrained crack surface nodes were found to be a source of numerical difficulties, and they investigated an alternate approach to simulate the cyclic crack surface contact. During loading and unloading, stresses and displacements were monitored along the crack surface. A negative nodal displacement indicated that the crack was closed at this point, and the displacement was set to zero. A tensile nodal stress indicated that the crack was open at this point, and the nodal restraint was removed. This more direct approach has also been used by Blom et al. [11].

Wu et al. [46] have used a truss element with a varying stiffness together with pairs of contact elements and the element death option. The element death option was incorporated to deactivate truss elements or cut the truss elements. They have shown that with this approach a node can be released any time during a load cycle irrespective of the magnitude of the deformation caused by the release of the node. Consequently, fewer problems with convergence were encountered and also several nodes can be released simultaneously.
2-1-2 Mesh Refinement

As a fatigue crack propagates, two different types of crack tip plastic zones are generated. The forward plastic zone is defined as the material near the crack tip undergoing plastic deformation at the maximum load. The second zone of interest is the reversed plastic zone, which is defined as the material near the crack tip undergoing compressive yielding at the minimum load [6]. These crack tip plastic zones have been used to characterize the degree of finite element mesh refinement required when modeling plasticity-induced closure [13,14,28,29].

Newman [48] was the first to study the effects of finite element mesh refinement on opening load computations under plane-stress conditions. He modeled a middle-crack tension (M(T)) specimen of width $2W$ with constant-strain triangle (CST) elements and found that the crack opening loads converged with increasing levels of mesh refinement at high applied stress levels as shown in Figure 2-1.

![Figure 2-1](image)

Figure 2-1  Mesh Refinement Studies

In the figure, $d$ is the element size ahead of the crack tip. For small applied stresses, convergence was not observed. Convergence may be a consequence of the number of
elements present in the reversed crack tip plastic zone. Newman considered the
discretization of the forward plastic zone only, and did not consider the reversed plastic
zone. Thus, the reversed zone may have been discretized with an insufficient number of
elements.

McClung et al. [14,30,31] performed mesh refinement studies on a crack
emanating from a circular hole, the M(T) specimen, and an edge-crack specimen. They
found that mesh refinement should be based on the number of elements present in the
forward plastic zone in the crack plane. They also suggested that adequate refinement to
capture the reversed plastic zone may be important. Dougherty et al. [13] performed
mesh refinement and element shape studies on C(T) and M(T) geometries under plane-
strain, and found that an aspect ratio less than or equal to 2 should be used for elements
ahead of the crack. They also found that the mesh density ahead of the crack should
satisfy \( \Delta a/r_f \leq 0.1 \), where \( 2r_f \) is an approximation of the forward plastic zone given by:

\[
r_f = \frac{1}{2\alpha\pi} \left( \frac{K_{\text{max}}}{\sigma_0} \right)^2
\]  

(2-1)

where \( \alpha \) is equal to 1 and 3 for plane-stress and plane-strain respectively, \( \sigma_0 \) is the flow
stress, and \( K_{\text{max}} \) is the maximum stress intensity factor. Park et al. [41] suggested that
mesh refinement levels for the M(T) specimen should be chosen to produce opening
stress values that compare well with experimental results. In the opinion of the authors,
this approach is flawed given the difficulties associated with measuring opening load
values [54,65-68].
Most of the work reported in the literature has incorporated large applied stresses, which allows for the use of coarse meshes while still satisfying mesh refinement requirements.

2-1-3 Stabilization of Crack Opening Load

Under constant amplitude loading, the crack opening load will typically increase monotonically with increasing crack growth until a stabilized value is reached as illustrated in Figures 2-2a and 2-2b.

McClung [14] has shown that under constant amplitude loading conditions, the crack must be advanced completely through the initial forward plastic zone to form a stabilized plastic wake. This is necessary to obtain non-varying crack opening values. However, Fleck et al. [9] and Wu et al. [46] have shown a variation in crack opening values even after the crack has progressed through initial forward plastic zone. Fleck’s results are shown in Figure 2-2b. Using a strip-yield model, Daniewicz et al. [69] have...
shown that a large amount of crack growth will produce a decreasing opening value if
the remaining ligament becomes small enough. Achieving the large amounts of crack
growth required to observe an initial stabilization followed by a subsequent decay in the
crack opening value is difficult when using finite element analysis, because of the
computationally intensive nature of the simulation. However, such an effort has been
reported by McClung [49].

2-1-4 Crack Advance Scheme

To produce a plastic wake behind the crack tip, the crack must be incrementally
advanced in some fashion under the applied cyclic loading. The most common means of
crack advance is to release the crack tip node, thus advancing the crack by an amount
equal to the crack tip element size. It is important to realize that modeling an incremental
crack advance with a node release involves no consideration of the physics of fatigue
crack growth, since the crack extension is independent of stress level and the strain in the
vicinity of the crack tip. Consequently, the finite element analysis is used to predict the
crack opening value, but not the fatigue crack propagation life. Recently, some
researchers have suggested the use of a cohesive element to advance the crack in a
physics-based manner [70]. Newman used a critical strain to advance the crack [46].
Nakagaki and Atulri [26] proposed a stress criterion for advancing the crack tip.

When performing analyses using the conventional node release technique, the
preferred node release scheme for simulating an incremental crack advance is unclear.
The crack tip node may be advanced at the minimum load level [7,10,14,28,33-35,37-
40,46], at the maximum load level [8,10,11,15,16,42,47-49], after the maximum load
[14,30,31,23,24], during the loading/unloading cycle [26,37,32,51] or during the second cycle [7]. Ogura et al. [10] have implemented a simple criteria for crack advance. The crack was advanced by one element when the crack tip reaction force became zero during the loading cycle. Alternately, Palazotto et al. [32] in their study proposed a criterion of growing the crack at 98% of the maximum load.

Advancing the crack at the maximum load level may create convergence problems; conversely, there is no such problem with advancing the crack using the minimum load level scheme. The convergence problem related to the maximum load released can be eased by incrementally releasing the crack front nodes [11,28].

Some research [7,28] has concluded that in terms of the resulting crack opening value, there is no difference when using the either maximum or minimum load node release schemes. However, other research has shown significant differences [14,30,31,46].

Figure 2-3  Comparison of Crack Opening Values Based on Crack Advance Scheme
Figure 2-3a shows results from a crack advance scheme comparison performed by McClung et al. [30,31]. From the figure, there is a significant difference in opening value when using a different crack advance scheme. Later, McClung et al. [7] showed that this difference was a consequence of using truss elements for crack surface node fixity, and that changing the boundary conditions on the crack surface nodes directly yields approximately the same results for the different crack advance schemes. Wu et al. [46] in their independent study found a variation in crack opening values when using the minimum and maximum loading node release schemes, and their results are shown in Figure 2-3b. This difference may be a consequence of computing the crack opening values based upon zero crack tip nodal reaction force, which is likely influenced by the size of the elements near the crack tip. This variation may also be due to a insufficient discretization of the reversed plastic zone.

2-1-5 Crack Opening Assessment Location

Under constant amplitude loading, the crack tip is the last point to open along the crack surface under an increasing load. Most researchers have used the first node behind the crack tip to assess the crack opening values [7-12,15-51]. Wu et al. [46] have used the crack tip itself to assess the crack opening values. They have proposed that when the compressive stress borne by the crack tip node changes to a tensile one, the crack is fully open. Others have used the second node behind the crack tip [29,55]. McClung et al. [7] and Fleck et al. [9] have shown that the results obtained when using first node behind the crack tip can be mesh dependent.
2-1-6 Variable Amplitude Loading

It may be argued that crack closure analyses are primarily of interest when considering variable amplitude loading. However, the majority of research has considered only constant amplitude loading. Some effort has been directed toward simple load histories such as low-high, or high-low, or a single overload [10,13,16,22,26,27, 36,39,41,43,48]. This research has been used to explain crack growth acceleration and retardation. Due to the computationally intensive nature of closure modeling with finite element analysis, complex load histories are generally not suitable for study, since a large amount of crack growth and a subsequently large number of load cycles are required.

2-1-7 Plane-stress and Plane-strain Condition

The nature of plastic deformation near the crack tip is strongly influenced by the two-dimensional idealization assumed. The permanent elongation of material in the direction normal to the crack requires the transfer of material from somewhere in the cracked body due to incompressibility requirements during plastic deformation. Under plane-stress, a potential mechanism of material transfer is obvious. Since out-of-plane deformation is not constrained, material can be transferred from the thickness direction to the axial direction. However, the mechanism of material transfer postulated for plane-stress is not admissible for plane-strain. By definition, no net out-of-plane contraction can occur, and therefore it has been suggested that there can be no net axial stretch of material in the plastic wake behind the crack tip, which implies no plasticity-induced crack closure [7-9,13,28]. The existence of plasticity-induced crack closure under plane-strain conditions has been a topic of intense debate [9-27].
A study of crack tip plastic zone sizes and crack opening behavior for the M(T) specimen under plane-strain and plane-stress conditions has been performed by McClung et al. [30,31]. Crack closure was found to occur in plane-strain, with lower opening values than those observed under plane-stress. Ogura et al. [10] have also simulated fatigue crack growth under plane-strain conditions using the finite element method. They also found that closure does exist under plane-strain conditions, and that the opening values reached a constant steady-state value after a sufficient amount of crack growth. Their results, however, are suspect as the ratio of the element length $\Delta a$ to the forward plastic zone size $r_f$ was a relatively coarse $\Delta a/r_f = 0.66$. A combined numerical and experimental study of crack closure in AA2024-T3 was conducted by Blom and Holm [11]. A plane-stress and plane-strain model of the C(T) specimen was constructed with constant strain triangular (CST) elements. Under plane-strain conditions closure was observed, and the plane-strain closure levels were smaller than those for plane-stress. Their results are also questionable due to a relatively coarse mesh and the use of element type which is susceptible to plane-strain locking [71]. Under a stress ratio $R = -1$, Lalor and Sehitoglu found that the plane-strain closure levels were lower than those for plane-stress. However, when the applied stress was increased to $S_{max}/\sigma_0 = 0.8$, the opening values were larger [24].

Dougherty et al. [13] performed two-dimensional analyses of C(T) and M(T) geometries under plane-strain, and demonstrated a good comparison between predicted closure levels and experimental results. Their finite element meshes were composed of four-noded and eight-noded quadrilateral elements. Ashbaugh et al. [12] performed a study similar to that conducted by Blom and Holm [11], focusing on finite element
analysis of plasticity-induced crack closure in the C(T) specimen under plane-strain conditions. In their analyses four-noded quadrilateral elements were used, and their results indicated that closure does occur in plane-strain. Again, their results are also suspect due to a lack of mesh refinement and potential plane-strain locking. Conversely, Fleck and Newman [9] have shown that closure does not occur for a bend specimen under plane-strain conditions, while closure does occur for the M(T) geometry under plane-strain. This may be due to the fact that the M(T) geometry has a compressive $T$-stress, while the bend specimen has a tensile $T$-stress. They found crack closure occurring for a single element on the crack surface of the bend specimen, and suggested that this closure was an artifact of the finite element analysis.

2-1-8 Geometry Effects

Fleck [18], Fleck and Newman [9], Larsson and Carlsson [72], and Rice [73] have shown that an influence of specimen geometry upon crack tip plastic deformation, beyond that associated with the stress intensity factor, may be accounted for in terms of the $T$-stress. This stress is the nonsingular constant second term in the near crack tip series expansion, and represents a normal stress parallel to the crack. The $T$-stress is directly proportional to the applied load and also depends on geometry. A two-dimensional asymptotic expansion for the stresses $\sigma_{ij}$ near the crack tip for mode I loading is given by [74]:

$$\sigma_{ij} = \frac{K_I}{\sqrt{2\pi r}} f_{ij}(\theta) + T \delta_{ii} + O(\sqrt{r}) + \ldots$$ (2-2)
where \( r \) and \( \theta \) are polar coordinates located at the crack tip, \( \delta_{ij} \) is the kronecker delta, \( K_I \) is the stress intensity factor, and the \( f_{ij} \) are dimensionless functions.

The C(T) and M(T) geometries differ in that each exhibits a distinctly different in-plane constraint, as quantified using the elastic \( T \)-stress, where the \( T \)-stress is defined as [74]:

\[
T = \frac{K_I \beta}{\sqrt{\pi a}} \tag{2-3}
\]

where \( a \) is the crack length, \( K_I \) is the stress intensity factor, and \( \beta \) is the biaxiality ratio. This ratio is equal to \(-1\) and \(0.425\) for the M(T) and C(T) geometries respectively [75].

Fleck [8] has shown a decrease in the closure level as the \( T \)-stress was varied from compressive to tensile using bend and M(T) specimens. The effect of \( T \)-stress on the crack opening value stabilization and crack opening process for different geometries is shown in Figure 2-4.
Fleck [8] studied the effect of \( T \)-stress variation on the crack opening value stabilization for different geometries by changing the applied maximum stress intensity factor, and results are shown in Figure 2-4. It should be noted from Figure 2-4 that as the \( T \)-stress becomes more tensile in nature, the opening values are approximately zero except for the single element immediately behind the crack tip.

Under a plane-stress condition, the effect of \( T \)-stress on closure is negligible. Fleck [8] performed finite element analysis under plane-stress on two-different geometries with the same applied maximum stress intensity factor and found no difference in the crack opening process as shown in Figure 2-5a.
However, McClung [50] has shown a significant difference in the crack opening stress at higher applied stresses, and negligible difference at lower applied stresses as shown in Figure 2-5b.

2-1-9 $R$ Ratio Effects

Of the many finite element analyses of plasticity-induced crack closure, the majority have considered a small positive stress ratio $R$. A larger positive $R$ results in a smaller reversed plastic zone, and hence increases the mesh refinement requirements.
The effect of $R$ on the crack opening stress for an M(T) specimen has been investigated by McClung et al. [31] and Newman [48] under plane-stress, and their results are shown in Figure 2-6.

![Figure 2-6 Effect of Stress Ratio on Crack Opening Values under Plane-stress](image)

Both of these researchers had approximately 10 elements in the forward plastic zone, which may have resulted in an insufficient number of elements in the reversed plastic zone. This suggests that mesh refinement study is essential to predict crack closure levels and further study is required with higher and lower $R$ ratio, both for positive and negative values.
2-2 Three-dimensional Modeling Aspects

A three-dimensional description of fatigue crack closure would increase the ability to predict crack growth behavior, which is inherently a three-dimensional problem. Even the simplest geometries and loading conditions, such as constant amplitude loading of middle-crack tension (M(T)) specimens, exhibit three-dimensional crack shapes in the form of crack tunneling. More complex geometries, such as surface cracks, or loading conditions, such as spectrum loading, will exhibit or result in even more dramatic shape changes. These shape changes are due to the three-dimensional variation of both the opening stress and the stress intensity factor along the crack front, and cannot be predicted by two-dimensional models.

A relatively small number of investigators have modeled plasticity-induced crack closure in three-dimensional geometries using finite element methods [53-63]. In a three-dimensional geometry, the crack opening value will vary along the crack front. This variation will result in different portions of the crack front growing with different rates under the cyclic loading. Consequently, the crack front shape will naturally evolve. This shape evolution makes modeling of three-dimensional geometries much more complex. For simplicity this shape evolution is generally neglected and the crack front is extended uniformly during the finite element analysis. The three-dimensional component is also more difficult to model because the required finite element meshes are large, inducing a severe computational burden.

The majority of the three-dimensional modeling efforts have considered the M(T) geometry [54,56,57,61-63]. In addition, limited modeling of the plasticity-induced crack
closure in the part-through surface flawed geometry [53,58-60,63] has also been performed.

Chermahini et al. [56-58] were the first to simulate crack growth and closure in three-dimensional geometries, and they considered both the M(T) specimen and the semi-elliptical surface crack. They found an initial rise in the crack opening stresses followed by a subsequent decay. Their results are suspect due to a lack of mesh refinement and an inadequate amount of crack growth, such that the crack opening stresses were not stabilized. However, Riddell et al. [62], using a more refined mesh for the M(T) specimen, have shown similar results.

Recently, Roychowdhury et al. [55] performed finite element analysis of plasticity-induced crack closure under small scale yielding conditions with a zero $T$-stress and a stress ratio $R = 0$. They computed the opening stresses for a through-crack with thickness $B$ using the second node behind the crack tip. Their results indicated that for a give $R$ value, the normalized opening stress $S_o / S_{max}$ scales with $\lambda = K_{max} / (\sigma_o \sqrt{B})$ such that a constant value of $\lambda$ will always result in the same crack opening stress. They have also shown that for $\lambda = 1$, in the mid-thickness region (plane-strain zone) of the model, little or no closure was noted. On the other hand, at the free surfaces (plane-stress zone), a significant amount of closure was observed. As $\lambda \to 2$, a sharp change in closure was observed at the mid-thickness region and little change was noted at the free surface.

Seshadri [63] compared predicted opening and closing stresses with experimental (fractographic) results for the compact-tension, single edge notch, and part-through crack geometries. Significant differences were observed between the experimental and
numerical results. Recently, Skinner et al. [60] compared results from a three-dimensional finite element analysis of the part-through crack with experimental results reported by Putra et al. [77], and showed significant differences between measured and computed crack opening stresses. Blandford et al. [54] and other researchers [65-68] have discussed the difficulties associated with measuring opening load values using experimental methods [54,67-70]. Dawicke et al. [67] have shown that displacement and strain gage crack closure measurements techniques may fail to give a complete description of plasticity-induced crack closure behavior in a thick specimen. They have also shown that while these techniques may be suitable for simple loading conditions, when the loading conditions become more complicated the three-dimensional effects become more pronounced. To accurately model these effects, a better understanding of the three-dimensional aspects of fatigue crack closure is needed.

2-2-1 Mesh Refinement

Mesh refinement issues are more complicated for three-dimensional models. Along the crack front near the free surface a near plane-stress condition exists while a plane-strain condition exists in the interior. Since a plane-stress forward plastic zone is approximately three times larger than a plane-strain forward plastic zone, the numbers of elements in the forward plastic zone in the interior must be used to determine an appropriate mesh size, and regions near the free surface may be over meshed. Zhang et al. [59] was the first to perform a mesh refinement study on a semicircular surface crack, and suggested that the crack opening and closing stresses are influenced by the degree of mesh refinement. Skinner et al. [60] have attempted to determine the required level of
mesh refinement for surface flaws. They proposed that mesh refinement can be quantified in terms of the number of elements present in the forward plastic zone. Their research efforts suggested that five elements in the forward plastic zone are sufficient to obtain stabilized crack opening values as shown in Figure 2-7.

Figure 2-7    Surface Crack Mesh Refinement Studies

2-2-2 Crack Advance Scheme

When modeling crack advance using a node-release scheme, two-dimensional finite element analyses of plasticity-induced crack closure have shown a negligible influence of the specific crack release scheme used. Zhang et al. [59] have performed node releases at three different values of load, 10% of $S_{\text{max}}$, 50% of $S_{\text{max}}$, and at $S_{\text{max}}$, and
found no difference in the crack opening displacement for these crack release schemes, while small differences were noted for the crack opening and closing stresses.

2-2-3 Aspect Ratio Evolution

There is ample experimental evidence to show that the shape of a three-dimensional fatigue crack front changes as the crack grows under cyclic loading [65, 77-79]. The aspect ratio \( a / c \) of a part-through crack changes under cyclic loading [79]. While the crack shape for a through-crack evolves from a straight line to a curved line, with faster growth in the interior. Many researchers have modeled the through-crack and part-through surface crack with a uniform crack extension such that the initial crack shape remains unchanged. However, this is inconsistent with the concept of crack closure where crack growth rate is governed by:

\[
\frac{d\eta}{dN} = C \left( \Delta K_{\text{eff}} \right)^m
\]  \hspace{1cm} (2-4)

where \( \frac{d\eta}{dN} \) is the growth rate normal to the crack front at a point on the crack front, \( C \) and \( m \) are material constants and \( \Delta K_{\text{eff}} \) is the effective stress intensity factor.

2-2-4 Influence of Loading History

A number of researchers have modeled plasticity-induced crack closure using two-dimensional finite element analysis with simplified load histories and shown crack retardation and acceleration. Daniewicz et al. [61] was first to attempt a three-dimensional elastic-plastic finite element analysis of a M(T) specimen under variable
amplitude loading. A continual load reduction to simulate the load history associated with fatigue crack growth threshold measurement was employed. Their results indicated the crack opening process is three-dimensional in nature, with regions in the interior opening prior to regions near the free surface.

2-3 Element Types and Configuration

The selection of element types for finite element analysis of plasticity-induced closure has become well established, however there are some important issues that need to be considered when implementing plane-strain conditions for two-dimensional finite element analysis. Most researchers have utilized constant strain triangle or 4-noded quadrilateral elements [8-52], while some of have used higher order quadrilateral elements [13,21] as shown in Figure 2-8.
During plane-strain analysis these elements generally do not meet the incompressibility requirement associated with plastic strains as shown by Nagtegaal et al. [71], and are thus susceptible to plane-strain locking. It has been shown that an arrangement of constant strain triangular elements in a “union-jack” configuration will enable the incompressibility requirement to be nearly satisfied [71]. Using a reduced integration scheme for quadrilateral or constant strain elements is also helpful for
avoiding plane-strain locking. When locking occurs, the stresses oscillate from one element to the next.

For three-dimensional analysis, generally 8-noded elements have been used [53-63], although some researchers have used a 6-noded element [59] as shown in Figure 2-8. In regions along the crack front under plane-strain, these elements are susceptible to plane-strain locking [71]. Again, using a reduced integration scheme for 8-noded and 6-noded elements is helpful for avoiding plane-strain locking [71]. Another similar method to avoid plane-strain locking is the $B_\text{element}$ formulation developed by Hughes [80]. This method replaces the volumetric strain at the Gauss integration points with the average volumetric strain of the element.

**2-4 Material Model Effects**

The elastic-perfectly plastic material model has been used extensively for two-dimensional and three-dimensional finite element analysis of plasticity-induced crack closure. However, the effects of material hardening have been considered as well, assuming both kinematic hardening and isotropic hardening. Using kinematic hardening will approximate the Bauschinger effect, and the use of isotropic hardening neglects this effect. Hardening will also affect plastic zone sizes, and hence mesh refinement requirements. To date, no finite element modeling effort have employed constitutive equations invoking concepts from cyclic plasticity theory. This, cyclic hardening and softening effects have not been considered, the crack opening values are assumed unaffected by potential shakedown and ratcheting.
Ashbaugh et al. [12] and McClung et al. [31] have shown that power law relationships between effective stress and effective strain generally give higher opening values than linear relationships. McClung et al. [31] have also show that variations in the linear hardening slope also impact the crack opening values. Seshadri [63] was the first to investigate linear and power law material model effects on three-dimensional closure analyses of the compact-tension specimen, the single edge notch specimen, and the part-through crack, and he showed significant differences in the crack opening and closing stress. This may be consequence of inadequate refinement. Similarly, Skinner et al. [60] have used a bi-linear material with kinematic hardening when modeling the surface flaw. A significant change in the opening values was observed as the hardening slope was changed as shown in Figure 2-9.

![Figure 2-9 Effect of Strain Hardening](image-url)
Recently, Roychowdhury et al. [55] performed small scale yielding analyses of the three-dimensional through-crack with pure kinematic hardening and they have shown that normalized opening load values remain unchanged for materials with a varying $\sigma_o/E$ ratio, provided the strain-hardening modulus $E_T$ retains a fixed ratio with $E$, where $E$ is the Young’s modulus and $\sigma_o$ is the material flow stress.

When the crack of interest is small with respect to the grain size, then the plastic deformation is no longer isotropic and constitutive relationships from crystal plasticity theory are needed. A study of plasticity-induced fatigue crack closure using crystal plasticity theory was first conducted by Gall et al. [81,82]. They studied the growth of a fatigue crack in a single crystal, and showed the effect of crystallographic parameters on the crack opening value. Recently, Potirniche et al. [83] have implemented crystal plasticity theory to study fatigue crack closure for growing fatigue cracks propagating through a grain boundary. As the crack approaches the grain boundary, acceleration and retardation of the crack were noted based upon the crystallographic orientation of the adjacent grain.

2-5 Overview of Thesis

Finite element analysis is a promising tool for simulating and predicting plasticity-induced fatigue crack closure based parameters. In this thesis several modeling issues were considered. First, a two-dimensional mesh refinement study was performed considering different geometries with different element types and configurations under both plane-stress and plane-strain. Different means of mesh refinement were also
considered, in order to reduce the number of elements and nodes, thereby reducing computational time. Secondly, in-plane constraint was varied to predict the effect on computed crack closure level under plane-strain. The effect of the crack node release schemes on the crack opening values were also studied. Under plane-stress, the effect of varying the load ratio $R$ with different stress levels on different geometries was studied. Next, the different crack opening value assessment locations were compared and a unique new methodology was developed to compute crack opening values.

For three-dimensional analyses, mesh refinement criteria were set for the part-through crack. The effect of crack node release schemes and a spike overload on the crack opening values were studied. A new methodology was developed to model crack shape evolution.
CHAPTER III

FINITE ELEMENT ANALYSIS

The plasticity-induced fatigue crack closure model concepts were incorporated into the finite element package ANSYS using ANSYS Parametric Design Language (APDL) by Skinner [53], and were modified to accommodate many other aspects of modeling plasticity-induced crack closure in this research. A command line listing for all the routines involved for two-dimensional finite element analysis is included in Appendix A. Sample input files for different planer geometries are included in Appendix B. For three-dimensional analysis, command line routines involved are included in Appendix C.

The basic finite element algorithm of modeling plasticity-induced fatigue crack closure is conceptually simple. A mesh is created with an initial crack, and the mesh is loaded by remotely applied tractions. For constant amplitude loading, the loading is cycled between a maximum applied stress $S_{\text{max}}$ and a minimum applied stress $S_{\text{min}}$. During the cyclic loading the crack is advanced in some fashion, leading to the formation of a plastic wake behind the crack tip. This modeling concept is simple; however, there are several issues which must be addressed during the fatigue crack growth simulation.
3-1 Two-dimensional Finite Element Analysis

Fatigue crack closure analyses were performed using ANSYS 6.1 [84]. Two-dimensional finite element analyses of compact-tension (C(T)), middle-crack tension (M(T)) and bend (SEB) geometries were conducted using 4-noded quadrilateral elements and 3-noded triangular elements under plane-stress and plane-strain conditions. The material was assumed to be elastic-perfectly plastic with modulus of elasticity $E = 200$ GPa and flow stress $\sigma_o = 230$ MPa. A load ratio $R = 0$ and $–1.0$ was selected. The von-Mises yield criterion and associated flow rule were used. Small deformation theory was employed, except where noted.

Most of the previous finite element analyses reported in the literature for plane-strain analysis utilized CST and 4-noded quadrilateral elements. These elements generally do not meet the incompressibility requirement associated with plastic strains as shown by Nagtegaal et al. [71], and are thus susceptible to plane-strain locking. They have shown that an arrangement of constant strain triangular elements in a “union-jack” configuration will enable the incompressibility requirement to be nearly satisfied. They also found that a reduced integration method for quadrilateral or CST elements is helpful for avoiding plane-strain locking. When locking occurs, the stresses oscillate from one element to the next. Another similar method to avoid plane-strain locking is the $\overline{B}$ element formulation developed by Hughes [80]. This method replaces the volumetric strain at the Gauss integration points with the average volumetric strain of the element. The present study includes an evaluation of all the above element types and configurations, as well as the reduced integration method.
Fatigue crack growth was modeled by repeatedly loading the geometry, advancing the crack, then unloading. A large amount of crack growth may be required before stabilized crack opening values are generated. The model was incrementally loaded to the maximum load (loading increments of 0.0125 $S_{max}$), at which time the crack tip node was released, allowing the crack front to advance one elemental length $\Delta a$ per load cycle. The applied load was then incrementally lowered until the minimum load was attained (unloading increments of 0.0125 $S_{max}$). Crack surface closure was modeled by changing the boundary conditions on the crack surface nodes. During unloading the crack surface nodal displacements were monitored and if the nodal displacement became negative the node was closed and node fixity was applied to prevent crack surface penetration. Similarly, during loading the reaction forces on the closed nodes were monitored and when the reaction force became positive the nodal fixity was removed. Herein, the remote load at which the last fixity is removed is defined as the crack opening load and corresponds to the closest node to the crack tip. The opening loads can be found only to the resolution of the loading increment. To obtain a better estimate of the load when the crack surface actually opens, linear interpolation was used. For the load step before the crack surface node opens, the nodal reaction force is negative. Upon opening, the reaction force becomes positive. Linear interpolation is used to determine the remote load at which the reaction force became zero. The cyclic loadings were repeated as necessary to produce a prescribed amount of crack growth. Meshes with a higher degree of refinement and smaller element size $\Delta a$ required more load cycles to produce a prescribed amount of crack growth. Each load cycle consisted of two complete elastic-plastic monotonic analyses.
In order to determine the initial finite element mesh, an iterative technique was employed. An initial mesh was designed to give perhaps 2 to 3 elements ahead of crack tip in the initial forward plastic zone using the following approximate equation [74]

\[ 2r_f = \frac{1}{\alpha \pi} \left( \frac{K_{\text{max}}}{\sigma_0} \right)^2 \]  

(3-1)

where \( \alpha \) is equal to 1 and 3 for plane-stress and plane-strain respectively.

Following a monotonic analysis, if the actual initial forward plastic zone extended out of the refined region and into the transition region (as shown in Figure 3-3) then the refined region was enlarged so that the entire plastic zone was captured. This procedure was used to obtain the initial mesh and subsequent initial \( \Delta a \) value to perform crack growth analyses for both geometries. To study the effects of mesh refinement, crack growth analyses were next performed after reducing the element size consecutively by a factor of 1/2 or 1/3. Each time a more refined mesh was used, the same amount of total crack growth was modeled. This naturally led to an extremely refined mesh and the use of many load cycles. For perspective, the element sizes reported in the literature normalized with equation 3-1 are shown in Figures 3-1a and 3-2. In Figure 3-1b the element sizes are normalized with the specimen width \( W \). Clearly, the meshes used in the current study are significantly more refined than many of those discussed in the literature.
Figure 3-1  Distribution of Mesh Refinement Levels under Plane-strain
The C(T), SEB and M(T) geometries are shown in Figure 3-3. The C(T) geometry had an initial crack length of 25 mm with \( a/W = 0.33 \), SEB geometry has initial crack length of 2 mm with \( a/W = 0.1 \), and the M(T) geometry had an initial crack length of 4 mm with \( a/W = 0.1 \). The maximum loading was selected to give the same initial maximum normalized stress intensity factor \( K_{\text{max}} / \sigma_o = 1.07 \sqrt{mm} \) in all the geometries, and thus approximately similar initial forward plastic zone sizes. The effect of different applied stress levels and load ratios under plane-stress condition were also study with the M(T) and the SEB specimens.

![Figure 3-2](image-url)  
**Figure 3-2** Distribution of Mesh Refinement Levels under Plane-stress
$a = 4 \text{ mm}$

$H = 40 \text{ mm}$

$W = 40 \text{ mm}$

$S = 69 \text{ N/mm}^2$

Number of elements = 7759

Figure 3-3a  Typical Middle-crack Tension Model
$a = 25 \text{ mm}$
$W = 76 \text{ mm}$
$P = 355 \text{ N}$
Number of elements = 11759

Figure 3-3b  Typical Compact Tension Model
Figure 3-3c  Typical Side Edge Bend Model

\[ a = 2 \, mm \]
\[ W = 20 \, mm \]
\[ H = 40 \, mm \]
\[ M = 314.385 \, N \, mm \]

Number of elements = 30444
3-2 Three-dimensional Finite Element Analysis

A three-dimensional description of fatigue crack closure would increase the ability to predict crack growth behavior, which is inherently a three-dimensional problem. Even the simplest geometries and loading conditions, such as constant amplitude loading of middle crack tension (M(T)) specimens, exhibit three-dimensional crack shapes in the form of crack tunneling. More complex geometries, such as surface cracks, or loading conditions, such as spectrum loading, will exhibit or result in even more dramatic crack shape changes. These shape changes are due to the three-dimensional variation of both the opening stress and the stress intensity factor along the crack front, and cannot be predicted by two-dimensional models.

Fatigue crack closure analyses were performed using ANSYS 6.1 [84]. Three-dimensional finite element analyses of part-through surface crack were conducted using 8-noded brick elements. The material was assumed to be elastic-perfectly plastic with modulus of elasticity $E = 1060$ ksi and flow stress $\sigma_o = 75$ ksi. A load ratio $R = 0.1$ was selected. The von-Mises yield criterion and associated flow rule were used. Small deformation theory was employed, except where noted. The part-through surface flaw geometry is shown in Figure 3-4. The surface crack geometry had an initial aspect ratio $a/c = 1.0$ with $a/w = 0.1$ and $t/w = 1.0$. 
Mesh refinement issues become complicated for three-dimensional models. For a semi-circular flaw, at the free surface a plane-stress condition exists while a plane-strain condition is present at the deepest point of penetration. Since a plane stress plastic zone is approximately three times larger than a plane strain plastic zone, the number of elements in the plastic zone at the deepest point of penetration should be used to determine an appropriate mesh size. Unfortunately, this forces the mesh to have more than adequate refinement at the free surface, and necessitates nearly three times as many load cycles for crack opening level stabilization in this region. Similar approach was employed to determine initial mesh size as used for two-dimensional.
There is ample experimental evidence to show that the shape of a three-dimensional fatigue crack front changes as the crack grows under cyclic loading [65,77-79]. The aspect ratio \((a / c)\) of a part-through crack changes under cyclic loading [79], while the crack shape for a through-crack evolves from a straight line to a curved line, with faster growth in the interior. To date, most researchers have modeled the through-crack and the part-through surface crack with a uniform crack extension such that the initial crack shape remains unchanged. However, this is inconsistent with the concept of crack closure where crack growth rate is governed by:

\[
\frac{d\eta}{dN} = C (\Delta K_{eff})^m
\]  

(3-2)

where \(d\eta/dN\) is the growth rate normal to the crack front at a point on the crack front, \(C\) and \(m\) are the material constants, and \(\Delta K_{eff}\) is the effective stress intensity factor at the point of interest.

The above methodology will be employed and equation 3-2 will be used at each point along a semi-circular crack front to model both plasticity-induced crack closure and the subsequent aspect ratio evolution. Since the fatigue crack growth behavior of materials can be anisotropic, unique material constants \(C\) and \(m\) were determined for the deepest point of penetration and the free surface using experimental data [78,79]. A linear interpolation was used to find the material constants for each node along the crack front. The Newman-Raju stress intensity factor equation [85] was employed to compute \(\Delta K_{eff}\), using the opening stress values from the finite element analysis. Lastly, closure behavior will be studied under single spike overload factor of 1.5.
CHAPTER IV

FINITE ELEMENT RESULT

4-1 Two-dimensional Finite Element Analysis

4-1-1 Geometry Effect on Closure under Plane-strain Condition

To determine the sizes of the forward and reversed plastic zones in all the geometries, normalized values of the von-Mises stress $\sigma / \sigma_o$ were plotted ahead of crack tip. A typical result is shown in Figure 4-1 for the M(T) geometry under plane-strain. Similar results were also found for the C(T) geometry. A crack growing under cyclic loading with $R = 0$ showed a reversed plastic zone of about 1/10 the forward plastic zone. This is in contrast to the stationary crack, which theoretically exhibits a reversed plastic zone of 1/4 the forward plastic zone [86]. This difference is a consequence of the plastic wake which forms behind the growing crack. A criterion of $0.95 \leq \sigma / \sigma_o \leq 1$ was assumed to define the number of elements in the reversed and forward plastic zones.
Figure 4-1 Crack Tip Plastic Deformation for Growing Crack

The mesh refinement studies were next performed. Each mesh was refined until a converged opening load was determined. Figure 4-2 illustrates the variation in the number of elements in the plastic zones for the C(T) and the M(T) specimens as the mesh refinement was carried out. It is clear from the figure that the number of elements along the crack plane in the reversed plastic zone are significantly lower than in the forward plastic zone. Thus, a large refinement level is required to accurately capture the reversed plastic zone.
Several mesh refinement issues were studied in an effort to reduce the number of nodes and elements. If the total size of the highly refined region is reduced such that the initial plastic zone extends outside this region, then a significant reduction in the total number of elements can be achieved. However, when the initial forward plastic zone was allowed to extend into the transition region, the opening value found gave poor agreement with the value found when the initial forward plastic zone was fully captured by the finely meshed region. The influence of the proximity of the crack tip to the transition elements was studied. It was found that if the crack tip was too close to the transition region, significant variation in the opening values resulted. Thus, a refined mesh was required behind, ahead, and above the crack tip. Lastly, the size of adjacent
elements within the transition region was studied and it was found that a gradual transition with a size ratio less than or about 3 is needed.

Figure 4-3a Comparison of Calculated Crack Opening Values under Plane-strain (M(T) Specimen)
The degree of mesh refinement was continued until convergence of the opening values was observed as shown in Figure 4-3. The opening loads were also compared with results from two strip-yield models [76,87] assuming plane-strain conditions. It is clear from Figure 4-3a that for the M(T) geometry the opening stresses converged as the mesh refinement was carried out. For the C(T) geometry, the opening loads did not converge as seen in Figure 4-3b. The opening values reported were steady state after growing approximately twice the initial forward plastic zone, and typical results are illustrated in Figure 4-4. It should be noted from Figures 4-2 and 4-3a that for the M(T) specimen, approximately 3 to 4 elements are required in the reversed plastic zone to obtain an accurate opening stress. Considering only the coarse meshes in Figure 4-3b, some semblance of convergence is apparent, which explains why previous studies found in the

Figure 4-3b Comparison of Calculated Crack Opening Values under Plane-strain (C(T) Specimen)
literature have reported the existence of crack closure under plane-strain. Further refinement results in a continuing decrease in the opening values, which suggests that little or no closure exists under plane-strain for the C(T) specimen. The author would discourage the notion of extrapolating the results in Figure 4-3b to smaller $\Delta a$ values since a converged crack opening value would approach a horizontal asymptote. Other potential reasons for this lack of convergence include plane-strain locking and excessive plastic deformation.

![Diagram showing crack opening load transient behavior](image)

**Figure 4-4** Typical Crack Opening Load Transient Behavior

Analyses were also performed using CST elements. A similar variation in opening values was noted as the mesh refinement was carried out. The results are also shown in Figure 4-3b. The opening values found utilizing the CST elements were higher than those obtained using the quadrilateral elements. Plane-strain locking behavior can potentially
influence the predicted opening values and the arrangement of CST elements in a “union-jack” configuration can help to minimize this effect [71]. Additional analyses were performed utilizing CST elements arranged in a “union-jack” configuration and the results are again shown in Figure 4-3b. As seen in the figure, no improvement was observed with regard to convergence. Another technique to minimize the effects of plane-strain locking is to employ a reduced integration method [71]. Analyses were performed on the C(T) and the M(T) specimens using quadrilateral elements with reduced integration. It can be seen from Figure 4-3b that using reduced integration also resulted in a lack of convergence for the C(T) specimen.

Additional analyses were performed to determine if excessive plasticity was the reason for the lack of C(T) convergence, with the applied maximum load reduced by a factor of 2. The total amount of crack growth simulated was the same. The results are shown in Figure 4-5a. It is seen that the C(T) geometry with the lower load also did not converge. Lastly, the large-scale deformation option within ANSYS was used. From Figure 4-5a, enabling large-scale deformations also did not improve convergence.
It is also possible that the observed lack of C(T) convergence is an artifact of the crack growth algorithm. McClung et al. [7,30,31] have shown little or no difference in opening behavior when the crack is advanced at maximum or minimum load. To verify this, analyses were performed to observe the effects of the node-released scheme. The C(T) specimen was modeled to allow the node to release and the crack to advance at minimum load. A similar opening behavior trend was observed and is shown in Figure 4-5b. The results presented in this figure indicate that the node-released schemes are not responsible for the lack of opening value convergence.
The consequence of assuming plane-strain for the C(T) geometry was next investigated. Plane-stress analyses were performed for the M(T) and the C(T) specimens with the same original crack length and maximum stress intensity factor. The results are shown in Figures 4-4 and 4-6, and it appears that under plane-stress conditions convergence is readily achieved and in-plane constraint has negligible effects on closure. It should be noted from the Figures 4-2 and 4-6 that approximately 3 to 4 elements are required in the reversed plastic zone to obtain an accurate opening value.

Figure 4-5b Effect of Node-Release Schemes
Figure 4-6  Comparison of Calculated Crack Opening Values under Plane-stress

To further address the existence of closure for the C(T) specimen under plane-strain, the crack opening profile for the final cycle of loading was evaluated and is shown in Figure 4-7. If closure does occur, then the opening process should be smooth with the load required to open the crack monotonically increasing as the distance from the original crack tip increases. From Figure 4-7a, a coarse mesh under plane-strain exhibits some semblance of a smooth opening process, but as the refinement is carried out the entire crack is predicted to open instantaneously with the exception of the node just behind the crack tip. This would suggest that there is negligible closure in plane-strain for the C(T)
specimen. To justify this statement further, the opening behavior of the M(T) and C(T) specimens were compared. An approximately monotonically increasing opening of the crack was noted for the M(T) specimen during loading as shown in Figure 4-7b, which implies that closure does occur for the M(T) specimen under plane-strain conditions. For further evaluation, the crack opening behavior was compared under plane-strain and plane-stress conditions for the C(T) specimen. Monotonically increasing opening behavior under plane-stress was observed, similar to the M(T) specimen under plane-strain, as shown in Figure 4-7c.
Distance From the Initial Crack Tip, $X$ (mm)

Figure 4-7 Crack Opening Process
Crack closure was observed immediately behind the crack tip for the C(T) specimen under plane-strain as seen for the refined mesh in Figure 4-7a. This behavior is believed to be an artifact of the finite element approximation. Computing opening values at the second node behind the crack tip may help to reduce the approximation error. Figure 4-8 shows the opening behavior convergence for the C(T) specimen under plane-strain when using the second node behind the crack tip to obtain the opening load. Trends similar to those observed when using the node immediately behind the crack tip were seen for constant strain triangle, but in case of quadrilateral, quadrilateral with reduced integration, and union-jack, as the mesh refinement is carried out a zero opening value was observed which again suggests that closure is negligible for the C(T) specimen under plane-strain.

![Figure 4-8  Effect of Assessment Location](image-url)
Significant closure levels were observed under plane-strain conditions for the M(T) specimen, but not for the C(T) specimen. Consequently, a geometry effect is clearly evidenced. One way these two geometries differ is that each exhibits a distinctly different in-plane constraint, as quantified using the elastic $T$-stress. To evaluate the influence of the $T$-stress, the M(T) specimen was modeled with an externally induced $T$-stress to observe the subsequent change in closure levels. A $T$-stress was induced by applying tractions parallel to the crack in addition to the conventional tractions perpendicular to the crack. When no tractions parallel to the crack are applied, the M(T) specimen exhibits an inherent compressive $T$-stress, where the $T$-stress is defined as [74]:

$$T = \frac{K_I \beta}{\sqrt{\pi a}}$$  \hspace{1cm} (6)

where $a$ is the crack length, $K_I$ is the stress intensity factor, and $\beta$ is the biaxiality ratio. This ratio is equal to –1 and 0.425 for the M(T) and C(T) geometries respectively [75]. The M(T) specimen has $T = 0$ when it is loaded biaxially [75].
To explore the influence of the $T$-stress, the M(T) specimen was modeled with an externally induced $T$-stress $-0.75 \leq T/S_{\text{max}} \leq 1$. The subsequent opening behavior is shown in Figure 4-9. This figure shows the opening behavior convergence of the M(T) specimen under different in-plane constraint values. As the mesh refinement was carried out non-convergence was noted for $-0.75 \leq T/S_{\text{max}} \leq 1$, including the value $\beta = 0.425$ associated with the C(T) specimen. The crack opening process is shown in Figure 4-10. Both of these figures indicate that crack closure is negligible or is not occurring for $-0.75 < T/S_{\text{max}} \leq 1$. As the $T$-stress become more tensile in value (including the $T$-
stress related to C(T) specimen), the crack front was fully open except for the element just behind the crack tip, which would indicate negligible closure.

![Figure 4-10 Effect of T-stress on the Crack Opening Process](image)

4-1-2 Effect of Load Ratios and Stress Levels under Plane-stress

The effect of $R$ on the crack opening stress for an M(T) specimen has been investigated by McClung et al. [31] and Newman [48] under plane-stress. Similar finite element analyses were performed with M(T), C(T) and SEB geometries to study the effect of highly refined mesh.
To determine the sizes of the forward and reversed plastic zones in all the geometries, normalized values of the von-Mises stress $\sigma_e / \sigma_0$ were plotted ahead of crack tip as discussed previously. The mesh refinement studies were next performed with a stress level of 0.3 and a load ratio of 0.0. Each mesh was refined until a converged opening load was determined. Figure 4-11 illustrates the variation in the number of elements in the plastic zones for the C(T), the SEB and the M(T) specimens as the mesh refinement was carried out. It is clear from the figure that the number of elements along the crack plane in the reversed plastic zone are significantly lower than in the forward plastic zone. Thus, a large refinement level is required to accurately capture the reversed plastic zone.

![Figure 4-11 Variation in the Plastic Zone Sizes with Mesh Discretization under Plane-stress](image)

Figure 4-11 Variation in the Plastic Zone Sizes with Mesh Discretization under Plane-stress
For perspective forward plastic zone sizes were compared with stationary FEA, cyclic FEA, and Irwin’s approximation of the forward plastic zone and as shown in Figure 4-12. Significant changes in the forward plastic zone sizes were noted between the cyclic and the stationary analysis. These differences were a consequence of the plastic wake formation. However, negligible difference was noted between Irwin’s approximation and stationary FEA.

Figure 4-12a  Comparison of the Forward Plastic Zone Size – M(T) Specimen
The degree of mesh refinement was increased until convergence of the opening values was observed as shown in Figure 4-13. It is clear from Figure 4-13 that the convergence of opening values was noted for the geometries as the mesh refinement was carried out. The opening values reported were steady state after growing approximately twice the initial forward plastic zone, and typical results are illustrated in Figure 4-14. Negligible difference in the crack opening values was noted with mesh refinement as shown in Figure 4-13, which suggests that under plane-stress condition, the effect of T-stress on closure is not significant.
Figure 4-13  Comparison of Calculated Crack Opening Values under Plane-stress

Figure 4-14  Typical Crack Opening Value Transient Behavior under Plane-stress
The mesh refinement studies were performed for M(T), C(T) and SEB geometries with a load ratio of 0.0 and a stress level of 0.3. For further analysis with different stress levels, the cyclic forward plastic size and total crack growth were fixed as those for the converged opening value for a stress level of 0.3. Approximately 35 elements were present in the cyclic forward plastic zone, which results in 3 to 4 elements in the reversed plastic zone. Next, the effect of stress levels on the crack opening value were studied under $R = 0.0$ and $-1.0$ with the M(T) and the SEB geometries and results are shown in Figure 4-15.

![Figure 4-15 Effect of Stress Ratio on Crack Opening Values under Plane-stress](image-url)

Figure 4-15 Effect of Stress Ratio on Crack Opening Values under Plane-stress
From the above figure it should be noted that the crack opening values from highly refined converged mesh were lower than those reported in literature, and were in better agreement with strip-yield model results generated using FASTRAN [76]. The meshes employed in the present study were more refined than those used by McClung et al. [30,31] and Newman [48]. Mesh refinement is the likely reason for the discrepancy, since to accurately capture the reversed plastic zone as shown in Figure 4-11, a high degree of refinement is required.

4-2 Three-dimensional Finite Element Analysis

Fatigue crack closure analyses were performed using ANSYS 6.1 [84]. Three-dimensional finite element analyses of part-through surface crack with initial aspect ratio of 1.0 were conducted using 8-noded brick elements. The methodology described in chapter 3 was used to model crack shape evolution.

To determine the sizes of the forward and reversed plastic zones at the deep point of penetration and at the free surface in the three-dimensional geometry, normalized values of the von-Mises stress $\sigma_c/\sigma_o$ were plotted ahead of crack tip as discussed previously for two-dimensional analysis. Mesh refinement issues become complicated for three-dimensional models. For a semi-circular flaw, at the free surface a plane-stress condition exists while a plane-strain condition is present at the deepest point of penetration. Since a plane stress plastic zone is approximately three times larger than a plane strain plastic zone, the number of elements in the plastic zone at the deepest point of penetration should be used to determine an appropriate mesh size. Unfortunately, this forces the mesh to have more than adequate refinement at the free surface, and
necessitates nearly three times as many load cycles for crack opening level stabilization in this region. Due to hardware limitations, the mesh refinement studies were performed with respect to the deep point of penetration. To determine the initial mesh size, similar approached is used as described earlier for two-dimensional. The degree of mesh refinement was continued until convergence of the opening values was observed as shown in Figure 4-16. A maximum of 20 elements were present in the forward plastic zone, but it is found that five elements in the forward plastic zone is sufficient to obtain converged crack opening values.

Figure 4-16  Surface Crack Mesh Refinement Studies
Next, the fatigue crack growth analysis was performed using different node release schemes. Negligible differences in the crack opening stresses when the crack front nodes are released at the minimum and maximum load are noted and shown in Figure 4-17. For these results, at the deepest point of penetration, approximately 2 elements yielded in compression under the minimum loading.

![Figure 4-17 Effect of Crack Advance Scheme](image)

The effects of crack shape evolution and spike overload on crack opening stress and consequently the fatigue crack closure behavior was next studied. A spike overload factor of 1.5 was used. Differences in the predicted opening values were noted when the aspect ratio was allowed to vary and also when spike overload is applied as shown in Figure 4-18. Crack retardation was also noted at the free surface as shown in Figure 4-19.
These results suggest that crack shape evolution can be successfully model and predicted using finite element analysis. Further study is required with significant crack growth at the free surface.

Figure 4-18  Surface Flaw Crack Opening Stress Transient
Figure 4-19  Predicted Crack Shape Evolution under Constant Amplitude Loading
CHAPTER V

CONTACT STRESS METHOD FORMULATION

The discussion from the earlier chapters suggests some discrepancy in the calculation of crack opening values from finite element analysis. Significant closure level was observed for the M(T) specimen under plane-strain, however, elemental or negligible closure was observed for the C(T) specimen under plane-strain. Results reported in the literature by some researchers suggest that the computed crack opening values from the node immediately behind the crack tip node should not be used because this node is too close to the poorly modeled crack tip, and suggest that second node behind the crack tip should be used to overcome this artifact of the FEA. On the other hand, results discussed in the earlier chapter indicate that the use of the second node behind the crack tip yields a zero opening value for the C(T) specimen under plane-strain, which implies no closure. It may be wise to compute the opening load considering whole crack front, not just one node. In the following chapter a new methodology will be discussed and developed to compute the crack opening value.
5-1 Formulation

Dill and Saff [88] were the first to introduce a contact stress method to compute crack opening loads, and employed the methodology in a strip-yield model. In this method, the stress intensity factor required to open the crack $K_o$ is computed using the contact stresses along the closed crack surface under the minimum loading. The stress intensity factor $K_c$ associated with this loading must be overcome to open the crack giving $K_c = K_o$. In the research described herein, this methodology was revisited and applied to compute crack opening values from finite element analysis results.

To compute $K_o$ for the C(T) specimen from the crack surface nodal stress distribution under the minimum loading, first consider the stress intensity factor for an infinite plane with a semi-infinite crack subjected to point load $p$ on the crack surface [89]

$$K = \frac{2}{\sqrt{\pi \varepsilon}} p \quad (5-1)$$

where $K$ is the stress intensity factor and $\varepsilon$ is the distance from the origin.
Figure 5-1 Infinite Plane with a Semi-infinite Crack Subjected to a Segment of Linearly Varying Stress.

If the crack surface is subjected to a segment of linearly distributed stress as shown in Figure 5-1, then equation 5-1 may be used to write the incremental stress intensity factor

\[ dK = \sqrt{\frac{2}{\pi x}} s \, dx \]  

where \( s \) is the linear stress distribution at any distance \( x \) from the origin

\[ s = (c_1 + xc_2) \]  

and \( c_1 \) and \( c_2 \) are constants. As shown in Figure 5-1, when \( x = b, s = s_1 \) and when \( x = c, s = s_2 \). Consequently

\[ c_1 = \frac{bs_2 - cs_1}{b-c} \quad \text{and} \quad c_2 = \frac{s_1 - s_2}{b-c} \]
The nodal stresses \( s_1 \) and \( s_2 \) are computed from the nodal forces \( p_1 \) and \( p_2 \) as follows

\[
s_1 = \frac{p_1}{\Delta a} \quad \text{and} \quad s_2 = \frac{p_2}{\Delta a}
\]  

\( (5-5) \)

where \( \Delta a \) is the element size. Substituting equation 5-3 into 5-2 and integrating over \( c \leq x \leq b \) yields

\[
K = \frac{2}{3} \left[ \frac{2}{\pi} \sqrt{ \frac{3}{b-c} \left( \frac{bs_2 - cs_1}{b-c} + c \left( s_1 - s_2 \right) \right) - \sqrt{b} \left( \frac{bs_2 - cs_1}{b-c} + b \left( s_1 - s_2 \right) \right) } \right] 
\]  

\( (5-6) \)

The \( K_o \) value associated with the contact stress for each finite element on the crack surface can be calculated using equation 5-6. Superposition may then be applied to find the \( K_o \) value for the entire crack surface loading.

\[
\frac{K_o}{K_{max}} = \frac{2}{3} \left[ \sum_{i=1}^{n} \frac{\sqrt{C_i} \left( \frac{b(s_i) - c_i(s_i)}{b_i - c_i} + c_i \left( s_i - s_i \right) \right)}{\sqrt{b_i} \left( \frac{b(s_i) - c_i(s_i)}{b_i - c_i} + b_i \left( s_i - s_i \right) \right)} \right]
\]  

\( (5-7) \)

where \( K_{max} \) is the maximum stress intensity factor, \( \{s_i\} \) and \( \{s_i\} \) are the \( i^{th} \) nodal stress values at distances \( b_i \) and \( c_i \) from the crack tip respectively, and \( n \) is the total number of nodal stress values.

To compute \( K_o \) for the M(T) specimen, next consider the stress intensity factor for an infinite plane with a finite crack subjected to point load \( p \) on the crack surface [89]

\[
K = \frac{p}{\sqrt{\pi a} \sqrt{a^2 - e^2}}
\]  

\( (5-8) \)
where $K$ is the stress intensity factor, $2a$ is the crack length and $\varepsilon$ is the distance from the origin.

![Figure 5-2 Infinite Plane with a Finite Crack Subjected to a Segment of Linearly Varying Stress](image)

If the crack surface is subjected to a segment of linearly distributed stress as shown in Figure 5-2, then equation 5-8 may be used to write the incremental stress intensity factor

$$dK = \sqrt{\frac{2}{\pi a}} \frac{s}{\sqrt{a^2 - x^2}} dx$$

(5-9)

where $s$ is the linear stress distribution at any distance $x$ from the origin

$$s = (c_1 + xc_2)$$

(5-10)
and \( c_1 \) and \( c_2 \) are constants. As shown in Figure 5-2, when \( x = b \), \( s = s_1 \) and when \( x = c \), \( s = s_2 \). Consequently

\[
c_1 = \frac{bs_2 - cs_1}{b - c} \quad \text{and} \quad c_2 = \frac{s_1 - s_2}{b - c}
\]  

(5-11)

Substituting equation 5-10 into 5-9 and integrating over \( b \leq x \leq c \) yields

\[
K = \frac{2}{\sqrt{a\pi}} \left[ \left( \sqrt{a^2 - b^2} - \sqrt{a^2 - c^2} \right) \frac{s_1 - s_2}{b - c} + \left( \sin^{-1} \left( \frac{c}{a} \right) - \sin^{-1} \left( \frac{b}{a} \right) \right) \frac{bs_2 - cs_1}{b - c} \right]
\]  

(5-12)

For the entire crack surface

\[
\frac{K}{K_{\text{max}}} = \frac{2}{\sqrt{a\pi}} \sum_{i=1}^{n} \left[ \left( \sqrt{a^2 - b_i^2} - \sqrt{a^2 - c_i^2} \right) \frac{(s_h)_i - (s_l)_i}{b_i - c_i} + \sin^{-1} \left( \frac{c_i}{a} \right) \frac{b_i(s_h)_i - c_i(s_l)_i}{b_i - c_i} - \sin^{-1} \left( \frac{b_i}{a} \right) \frac{b_i(s_h)_i - c_i(s_l)_i}{b_i - c_i} \right]
\]  

(5-13)

To compute \( K_o \) for the SEB specimen, next consider the stress intensity factor for an semi-infinite plane with an edge crack subjected to point load \( p \) on the crack surface

\[
K = \frac{2.6}{\sqrt{\pi a}} \frac{p}{\sqrt{1 - (\varepsilon/a)^2}}
\]  

(5-14)

where \( K \) is the stress intensity factor, \( a \) is the crack length and \( \varepsilon \) is the distance from the origin.
If the crack surface is subjected to a segment of linearly distributed stress as shown in Figure 5-3, then equation 5-14 may be used to write the incremental stress intensity factor

\[ dK = \frac{2.6}{\sqrt{\pi a}} \frac{s}{\sqrt{1 - (x/a)^2}} \]  \hspace{1cm} (5-15)

where \( s \) is the linear stress distribution at any distance \( x \) from the origin

\[ s = (c_1 + xc_2) \]  \hspace{1cm} (5-16)

and \( c_1 \) and \( c_2 \) are constants. As shown in Figure 5-3, when \( x = b \), \( s = s_1 \) and when \( x = c \), \( s = s_2 \). Consequently

\[ c_1 = \frac{bs_2 - cs_1}{b - c} \text{ and } c_2 = \frac{s_1 - s_2}{b - c} \]  \hspace{1cm} (5-17)

Substituting equation 5-16 into 5-15 and integrating over \( b \leq x \leq c \) yields
The contact stress method described above overcomes the limitation of focusing attention on a single node considering instead the global behavior of the entire crack surface. In addition, when using the contact stress method, the opening values are computed at the minimum load after the unloading cycle is complete. This eliminates any consideration of loading increments as required in the more conventional method. Thus the contact stress method is also independent of the unloading increment size.

When using the conventional methodology, the crack opening value is defined based on the last node to open. Consequently, the element immediately behind the crack
tip is not considered. This limitation is eliminated when using the contact stress method. The nodal forces were used to compute the required contact stress under the minimum loading, because forces are more accurate than stresses in any finite element analysis.

5-2 Result

The opening values computed for the C(T), the SEB and the M(T) specimens under plane-strain and plane-stress are shown in Figures 5-4 and 5-5.
Figure 5-4 Comparison of Predicted Crack Opening Values under Plane-strain Conditions

It may be noted from Figure 5-4 that assessment of crack opening loads at the second node behind the crack tip yields significantly lower plane-strain crack opening values. For all these geometries under plane-strain, the opening values from the contact stress method were higher than those found using the first node behind the crack tip, however small difference was noted under plane-stress. This was expected, since the element immediately behind the crack tip is also considered when calculating the opening load using the contact stress method. The opening values were approximately the same when using the contact stress method with reduced and full integration, which suggests that the contact stress method is resistant to the effects of plane-strain locking \[71\]. It may also be noted that under plane-strain conditions, the C(T) specimen opening values
did not converge as the mesh refinement was carried out. This would suggest that plasticity-induced closure does not exist or is negligibly small.

Figure 5-5 Comparison of Predicted Crack Opening Values Using Contact Stress Method and Conventional Method under Plane-stress Conditions

Next, the nodal force distribution used to compute $K_o$ was plotted and is shown in Figures 5-6 and 5-7. From these figures, it is clear that the element just behind the crack tip contributes significantly to the crack opening values. The contribution of the element immediately behind the crack tip when computing plane-strain crack opening values using the contact stress was determined. For the C(T) specimen, approximately 85% of the total opening value was associated with this element when using either full or reduced integration. In contrast, this value was approximately 50% for the M(T) specimen. For the C(T) specimen under plane-stress, a 30% value was observed. Consequently, the element immediately behind the crack tip plays a major role in the crack opening value
computation. In cases where this role is exceedingly large, such as the C(T) specimen under plane-strain, the validity of the computed opening value become questionable. This fact, in conjugation with the observation that for the plane-strain C(T) specimen the crack surface loading is predominantly zero, suggests that plasticity-induced closure is negligible for this configuration. Lastly the effect of load ratios and stress level was studied using the contact stress method under plane-stress. Small difference was noted as shown in Figure 5-8 when compared to first node immediately behind the crack tip.

Figure 5-6 Nodal Force Distribution along the Crack Surface under Plane-strain Conditions
Figure 5-7 Nodal Force Distribution along the Crack Surface under Plane-stress Conditions
Figure 5-8 Effect of Stress Ratio on Crack Opening Values under Plane-stress
CHAPTER VI
CONCLUSION

Finite element analysis is a promising tool to simulate plasticity-induced fatigue crack closure. The modeler must consider different aspects of modeling to get better finite element crack opening values and other fracture parameters as described in this research. Modeling aspects of two-dimensional finite element analysis under plane-strain and plane-stress conditions have been well established and it is suggested that more work is needed with three-dimensional models.

1. The middle-crack tension geometry crack opening stress exhibits convergence as mesh refinement is carried out under plane-strain conditions.

2. The compact tension geometry crack opening load does not exhibit convergence as mesh refinement is carried out under plane-strain conditions. This indicates that plasticity-induced closure is negligible or does not exist under plane-strain for the C(T) specimen.

3. Opening values for the compact tension, the side edge bend and middle-crack tension geometries crack exhibit convergence as mesh refinement is carried out under plane-stress conditions. This suggests that closure does exist under plane-stress.
4. Opening values vary when the initial crack tip plastic zone extends into the mesh refinement transition region and also when the crack tip is close to transition elements.

5. The middle-crack tension geometry crack opening stress does not exhibit convergence as the mesh refinement is carried out when different $T$-stress values with $-0.75 \leq T/S_{\text{max}} \leq 1$ are applied externally. This indicates that the level of in-plane constraint dictates the level of plasticity-induced crack closure under plane-strain.

6. Approximately 3 to 4 elements are required in the reversed plastic zone for the middle-crack tension specimen under plane-strain to obtain accurate opening values.

7. Approximately 3 to 4 elements are required in the reversed plastic zone for the compact tension, the side edge bend and middle-crack tension specimens under plane-stress to obtain accurate opening values.

8. The in-plane constraint effect on crack closure is negligible under plane-stress conditions.

9. The contact stress method of computing crack opening values was revisited and applied for the first time to finite element analyses. This global method has the advantage of not being associated with a single node and is independent of loading and unloading increment size.
10. Crack opening values computed using the contact stress method were unchanged when implementing reduced integration schemes. This suggests that the method is resistant to the effects of plane-strain locking.

11. Depending on the geometry and stress state, the element immediately behind the crack tip may play a major role in the crack opening value computation. This element is not considered in more conventional crack opening assessments.

12. Under plane-stress condition for different stress levels and load ratios, significantly lower crack opening values were found compare to those reported in literature.

13. For the three-dimensional geometry crack shape evolution can be modeled and subsequent crack growth retardation and acceleration can be predicted using FEA.

Lastly, it should be noted that while crack closure was shown either not to exist or to be negligible under plane-strain for certain levels of in-plane constraint, plane-strain is a two-dimensional idealization that cannot occur in practice. In the opinion of the author, the plane-stress condition existing at and near the free surface of a three-dimensional body will have a strong influence on the plane-strain interior, regardless of thickness of geometry.
REFERENCE


[34] Nakamura, H., Kobayashi, S., Yanase, S., and Nakazawa, H., “Finite Element Analysis of Fatigue Crack Closure in Compact Specimen,” Mechanical Behavior of


[84] ANSYS 6.1 ANSYS, Inc.


APPENDIX A1

ANSYS INPUT FILE APPBCS.MAC, APPLICATION OF
BOUNDARY CONDITIONS, ANY LOAD RATIO
/prep7

! Element Shape Checking Off
SHPP,OFF

! Define Material Properties for Solid Elements
MP,EX,1,E
TB,BKin,1,1,1,,
TBMODIF,2,1,YS
!TBMODIF,3,1,HTAN

*IF,MTYPE,EQ,'SC',THEN
  ET,1,SOLID45,,,,
*ELSE
  !Normal Plane Element
  !ET,1,PLANE42,,,2,,
  !Reduced Integration Element
  ET,1,PLANE182,0,,0,,
  ! KEYOPT(3) = 0 Plane Stress
  ! KEYOPT(3) = 2 Plane Strain
  ! KEYOPT(3) = 3 Plane Stress w/ thk
*ENDIF

! Begin Building Model: Read Solid Elements from File
MAT,1
TYPE,1
REAL,1
nread,%JN%,crd
eread,%JN%,elm

*IF,MTYPE,EQ,'CT',THEN
  ! Create Linear Elastic Material Properties for Elastic "plug"
  MP,EX,2,E
  LOCAL,12,1,w,height,0,0,0,0
  NSEL,S,LOC,X,0,r
  ESLN,S,1
  EMODIF,ALL,MAT,2
  NSEL,ALL
  ESEL,ALL
  CSYS,0
*ENDIF

*IF,MTYPE,EQ,'2DCT',THEN
  ! Create Linear Elastic Material Properties for Elastic "plug"
  MP,EX,2,E
  LOCAL,12,1,w,height,0,0,0,0
  NSEL,S,LOC,X,0,r
  ESLN,S,1
  EMODIF,ALL,MAT,2
  NSEL,ALL
  ESEL,ALL
  CSYS,0
*ENDIF
*IF, MTYPE, EQ, 'SEB', THEN
  ! Create Linear Elastic Material Properties for Loading Elements
  MP, EX, 2, E

  ! NSEL, S, LOC, Y, height
  ! NSEL, R, LOC, X, 0-10E-5, 0+10E-5
  ! ESLN, 1

  ! NSEL, S, LOC, Y, height
  ! *GET, SOUT, NODE,, COUNT
  ! DELT=W/(SOUT-1)
  ! NSEL, S, LOC, Y, height-delt-0.00001, height-delt-0.00001
  ! NSEL, R, LOC, X, DELT-0.00001, DELT+0.00001

  Nsel, s, loc, y, height-2, height
  nsel, r, loc, 0, 2
  ESLN, 1

  EMODIF, ALL, MAT, 2

  ! NSEL, S, LOC, Y, height-delt-0.00001, height-delt-0.00001
  ! NSEL, R, LOC, X, W-DELT-0.00001, W-DELT+0.00001
  Nsel, s, loc, y, height-2, height
  nsel, r, loc, w-2, w
  ESLN, 1

  EMODIF, ALL, MAT, 2
  NSEL, ALL
  ESEL, ALL

*ENDIF

! Constrain Nodes on Bottom Surface of Plate in the Vertical-direction
!(Constraints will be removed during crack growth)
!
! Also, create component containing crack surface nodes (used
! when negative loads are applied). Assumes elliptical geometries
! are notched, all others are not.

*IF, MTYPE, EQ, 'SC', THEN
  NOTCH=NY(NODE(0,0,0))
  LOCAL, 11, 1, 0, 0, 0, 0, 90, 0, a/c
  NSEL, S, LOC, X, c, 100000
  NSEL, R, LOC, Z, 0, 0
  NSEL, U, LOC, X, 0, c-0.05*da
  D, ALL, UY, 0
  ! Block Added for negative R,
  NSEL, S, LOC, Z, 0
  NSEL, R, LOC, X, 0, c
  NSEL, U, LOC, X, c-0.3*da, c+0.3*da
  CSYS, 0
  NSEL, A, LOC, Y, NOTCH
  CM, CSNODES, NODE
! End of Block
*ELSE
!Block Added for negative R,
NSEL, S, LOC, Y, 0
NSEL, R, LOC, X, 0, c
NSEL, U, LOC, X, c-0.3*da, c+0.3*da
CM, CSNODES, NODE
!End of Block
*ENDIF

!Apply Appropriate Conditions in X-direction:

*IF, MTYPE, EQ, 'CT', THEN
NSEL, S, LOC, Y, 0
NSEL, R, LOC, Z, 0
NSEL, R, LOC, X, 3.75
D, ALL, UX, 0
NSEL, ALL
*ENDIF

*IF, MTYPE, EQ, 'MT', THEN
NSEL, S, LOC, X, 0
D, ALL, UX, 0
NSEL, ALL
*ENDIF

*IF, MTYPE, EQ, 'SC', THEN
NSEL, S, LOC, X, 0
D, ALL, UX, 0
NSEL, ALL
*ENDIF

*IF, MTYPE, EQ, '2DCT', THEN
NDLOC=0
NSEL, S, LOC, Y, 0
*GET, MINnode, NODE,, NUM, MIN
NODEno=MINnode
NDLOC=nx(NODEno)
*GET, NSNODES, NODE,, COUNT
*DO, L, 1, NSNODES-1
NODEno=NDNEXT(NODEno)
NDLOC1=nx(NODEno)
*IF, NDLOC1, GT, NDLOC, THEN
NDLOC=NDLOC1
MAXnode=NODEno
*ENDIF
*ENDDO
NSEL, R,,, MAXnode
D, ALL, UX, 0
NSEL, ALL
*ENDIF

*IF, MTYPE, EQ, 'SEB', THEN
NDLOC=0
NSEL, S, LOC, Y, 0
*GET, MINnode, NODE,, NUM, MIN
NODEno=MINnode

NDLOC=nx(NODEno)
*GET,NSNODES,NODE,,COUNT
*DO,L,1,NSNODES-1
  NODEno=NDNEXT(NODEno)
  NDLOC1=NX(NODEno)
  *IF,NDLOC1,GT,NDLOC,THEN
    NDLOC=NDLOC1
    MAXnode=NODEno
  *ENDIF
*ENDDO
NSEL,R,,MAXnode
D,ALL,UX,0
NSEL,ALL
*ENDIF

!Apply Symetry BC's at Z=0 plane
*IF,MTYPE,EQ,'SC',THEN
  NSEL,S,LOC,Y,0
  NSEL,R,LOC,X,0
  NSEL,R,LOC,Z,t
  D,ALL,UZ,0
  NSEL,ALL
*ENDIF

*IF,MTYPE,EQ,'2DCT',THEN
  NSEL,S,LOC,Z,0
  D,ALL,UZ,0
  NSEL,ALL
*ENDIF

!Select Crack Mouth Node...Create Component CMNODES
NSEL,S,LOC,X,0
NSEL,R,LOC,Y,0
CM,CMNODES,NODE
CMSEL,ALL
NSEL,ALL
ESEL,ALL
WSORT,ALL,0 !Sort Elements to minimize wavefront
SAVE
FINISH
APPENDIX A2

ANSYS INPUT FILE *StrtCyc.MAC*, CONTROL OF CYCLIC LOADING, ANY LOAD RATIO
FirstLoad
ClearRST,BDrive,BDir,MaxDir

*DO, I, 1, NLC
  AdvanceCrack, I
  ClearRST,BDrive,BDir,''
  UnloadCrack, I
  ClearRST,BDrive,BDir,MinDir
  LoadCrack, I
  ClearRST,BDrive,BDir,MaxDir
*ENDDO
APPENDIX A3

ANSYS INPUT FILE FIRSTLOAD.MAC, APPLICATION OF FIRST LOAD, ANY LOAD RATIO
! Apply Maximum Load on First Cycle:

/SOLU
Appload, height, StrsMax
AUTOTS, ON
NSUBST, 5, 10000, 5, ON
TIME, 0.45
SOLVE
SAVE
APPENDIX A4

ANSYS INPUT FILE ADVANCECRACK.MAC, INCREMENTALLY

ADVANCE THE CRACK TIP, ANY LOAD RATIO
! AdvanceCrack.mac
! Macro File to advance crack uniformly one element
!
! Should be executed as:
!
! AdvanceCrack,LoadCycleNumber
!

AUTOTS,OFF
NSUBST,1,1,1

/PREP7
SelCTNodes,arg1
*GET,NNODES,NODE,,COUNT
NODNO=0
*DO,JJ,1,NNODES
    NODNO=NDNEXT(NODNO)
    *GET,NODERF,NODE,NODNO,RF,FY
    DDELETE,NODNO,UY
    F,NODNO,FY,NODERF
*ENDDO
NSEL,ALL
ESEL,ALL
/SOLU
ANTYPE,,REST

TimeVar=0.45+0.05/(NCGECut+1)+(arg1-1)
Time,TimeVar
SOLVE
SAVE

*DO,J,1,NCGECut-1
/PREP7
SelCTNodes,arg1
NODNO=0
*DO,JJ,1,NNODES
    NODNO=NDNEXT(NODNO)
    *GET,NODERF,NODE,NODNO,F,FY
    F,NODNO,FY,NODERF/NODERF
*ENDDO
NSEL,ALL
ESEL,ALL
/SOLU
ANTYPE,,REST
TimeVar=0.45+(J+1)*0.05/(NCGECut+1)+(arg1-1)
Time,TimeVar
SOLVE
SAVE
*ENDDO

/PREP7
SelCTNodes,arg1
NODNO=0
*DO,JJ,1,NNODES
    NODNO=NDNEXT(NODNO)
    *GET,NODERF,NODE,NODNO,F,FY
    FDELETE,NODNO,FY
*ENDDO
NSEL,ALL
ESEL, ALL
/SOLU
ANTYPE,,REST
TimeVar=arg1-0.5
Time, TimeVar
SOLVE
SAVE
APPENDIX A5

ANSYS INPUT FILE *UNLOADCRACK.MAC*, UNLOAD MODEL,

ANY LOAD RATIO
AUTOTS, ON
NSUBST, 1, 100, 1, OFF
*CFOOPEN, &JN%_unload_%arg1%, dat
!*VWRITE
! (*NODE# Node r NodeAng S/Smax UY OStat Remote Stress*)
CurrLInc=UIDCC
StrsLvl=StrsMax/StrsMax
RStrs=StrsMax
*DO, JJ, 1, arg1
SelICTNodes, JJ
NSEL, U, D, UY, 0
*GET, NSNODES, NODE,, COUNT
*IF, NSNODES, GT, 0, THEN
  NODNO=0
  *DO, JJJ, 1, NSNODES
    NODNO=NDNEXT (NODNO)
    NODYSTRS=UY (NODNO)
    *IF, MTYPE, EQ, 'SC', THEN
      CSYS, 11
      NDANG=NY (NODNO)
      NXLOC=NX (NODNO)
      CSYS, 0
    *ELSE
      NDANG=NZ (NODNO)
      NXLOC=NX (NODNO)
    *ENDIF
    NodeStat=0
    *VWRITE, NODNO, NXLOC, NDANG, StrsLvl, NODYSTRS, NodeStat, RSTRS
      (F6.0, 2X, E12.6, 2X, E10.4, 2X, F8.6, 2X, E12.6, 2X, F4.0, 2X, E12.6)
  *ENDDO
*ENDIF
*ENDDO

*DO, J, 1, 1/UIDCC
  TimeVar=TimeVar+CurrLInc*0.5
  RStrs=StrsMax-(StrsMax-StrsMin)*(TimeVar+0.5-arg1)/0.5
  *IF, TimeVar, GE, arg1, Then
    RStrs=StrsMin
    TimeVar=arg1
    Time, TimeVar
    AppLoad, height, RStrs
    SOLVE
    SAVE
    *EXIT
  *ENDIF
  Time, TimeVar
  StrsLvl=RStrs/StrsMax
  AppLoad, height, RStrs
  SOLVE
  SAVE
  ClearRST, BDrive, BDir, ''
  OPENSTAT=0
  OpnRwCnt=0
*DO, JJ, 1, arg1
  SelICTNodes, JJ
  NSEL, U, D, UY, 0
  *GET, NSNODES, NODE,, COUNT
  *IF, NSNODES, GT, 0, THEN
    NODNO=0
  *ENDIF
*ENDDO
*DO, JJJ, 1, NSNODES
  NodeStat=0
  NODNO=NDNEXT (NODNO)
  NODYSTRS=UY (NODNO)
  *IF, MTYPE, EQ, 'SC', THEN
    CSYS, 11
    NDANG=NY (NODNO)
    NXLOC=NX (NODNO)
    CSYS, 0
  *ELSE
    NDANG=NZ (NODNO)
    NXLOC=NX (NODNO)
  *ENDIF
  *IF, NodYStrs*1e10, LT, 0, THEN
    CurrLInc=UIDCC
    OPENSTAT=1
    NODESTATE=1
    D, NODNO, UY, 0
  *ENDIF
  *VWRITE, NODNO, NXLOC, NDANG, StrsLv1, NODYSTRS, NodeStat, RStrs
     (F6.0, 2X, E12.6, 2X, E10.4, 2X, F8.6, 2X, E12.6, 2X, F4.0, 2X, E12.6)
  *ENDDO
  *ELSEIF, NSNODES, EQ, 0, THEN
    OpnRwCnt=OpnRwCnt+1
  *ENDIF
  *ENDDO
NSEL, ALL

! Close Crack surface nodes if negative load applied
! Added for negative R -1
*IF, RStrs, LE, 0, THEN
  CMSEL, S, CSNODES
  NSEL, U, D, UY, 0
  *GET, NSNODES, NODE, , COUNT
  *IF, NSNODES, GT, 0, THEN
    NODNO=0
  *DO, JJJ, 1, NSNODES
    NODNO=NDNEXT (NODNO)
    NODYSTRS=UY (NODNO)
  *IF, NodYStrs*1e10, LT, 0, THEN
    OPENSTAT=1
    D, NODNO, UY, 0
  *ENDIF
  *ENDIF
  *ENDIF
NSEL, ALL
*ENDIF
  ! End of Block
*IF, OPENSTAT, EQ, 1, THEN
  Time, Timevar+CurrLInc*0.01
  SOLVE
  SAVE
  ClearRST, BDrive, BDir, ''
  *ENDIF
  *IF, OpnRwCnt, EQ, arg1, THEN
    CurrLInc=UIACC
  *ENDIF
  *ENDIF
*ENDDO
*CFCLOSE
ContactStress, arg1
APPENDIX A6

ANSYS INPUT FILE ContactStress.MAC, CONTACT STRESS METHOD, ANY LOAD RATIO
! contact stress method

NSEL,ALL
ESEL,ALL
NODY1=0
NODY2=0

K=0
*CFOpen,K1_%JN%_%arg1%,txt

PI=acos(-1)

SELECTNODES,(arg1+1)

*GET,FON,NODE,,NUM,MAX
AA=NX(FON)
NSEL,ALL

*IF,MTYPE,EQ,'2D',THEN
  NSEL,S,LOC,Y,0
  NSEL,R,LOC,X,0-(da*0.00001),AA+(da*0.00001)
  *GET,TNODE,NODE,,COUNT
  NSEL,S,LOC,Y,0
  NSEL,R,LOC,X,0-10e-5,0+10e-5
  *GET,FON1,NODE,,NUM,MAX
  A1=NX(FON1)
  NSEL,S,LOC,Y,0
  NSEL,R,LOC,X,0-(da*0.00001),AA+(da*0.00001)
  *DO,II,2,TNODE
    FON2=NNEAR(FON1)
    A2=NX(FON2)
    *GET,NODY1,NODE,FON1,RF,FY
    *GET,NODY2,NODE,FON2,RF,FY
    !CONVERSION OF NODAL FORCE IN TO ELEMENT STRESS
    NODY1=NODY1/da
    NODY2=NODY2/da
  !FIND KMAX FOR MT SPECIMEN
  F=(1-(0.025*((aa/w)**2))+(0.06*((aa/w)**4)))*sqrt(1/cos((PI*aa)/(2*w)))
  KMAX=StrsMax*sqrt(PI*aa)*F
  !FIND KOPENING FOR MT SPECIMEN MODEL
  !AS INFINITE PLATE WITH A FINITE CRACK
  !WITH LINEAR ELEMENT STRESS DISTRIBUTION
  !MORE REFERENCE ORIGINAL PAPER
  !BY KIRAN SOLANKI, S.R.DANIEWICZ, J.C. NEWMAN JR.
  BB=A1
  CC=A2
  C1=(NODY2*BB-NODY1*CC)/(BB-CC)
  C2=(NODY1-NODY2)/(BB-CC)
  tem=asin(cc/aa)
  J1=(tem*C1)-(asin(BB/AA)*C1)+(C2*((sqrt(AA**2-bb**2))-(sqrt(AA**2-
    cc**2)))))
  K=((2*(sqrt(1/(AA*PI))))*(J1))/KMAX+K
  NSEL,U,LOC,X,0-(da*0.00001),a1+(da*0.00001)
  FON1=FON2
  A1=A2
  NODY1=NODY1*da
NODY2=NODY2*da
!OUTPUT FOR THE FILE READ AS
!NODE NO, NODE LOCATION, NODAL FORCE 1,
!KOPEN/KMAX, DISTANCE FROM THE CRACK TIP, NODALFORCE 2
*VWRITE,FON1,NXLOC,NODY1,K,BB,NODY2
   (F6.0,2X,E12.6,2X,E12.6,2X,E12.6,2X,F8.6,2x,E12.6)
NSEL,U,LOC,X,0-(da*0.00001),a1+(da*0.00001)
FON1=FON2
A1=A2
NODY1=0
NODY2=0
*ENDDO
*ENDIF

*IF,MTYPE,EQ,'2DCT',THEN
   !FIND KMAX FOR CT/SEB SPECIMENS
   AW=AA/w  !A/W RATIO
   T=1  !THICKNESS
   F=((2+AW)/((1-AW)**1.5))*(0.886+4.64*AW-13.32*(AW**2)+14.72*(AW**3)-
      5.6*(AW**4))
   KMAX=F*StrsMax/(sqrt(W)*B)
   !FIND KOPENING FOR CT SPECIMEN MODEL
   !AS INFINITE PLATE WITH A SEMI-INFINITE CRACK
   !WITH LINEAR ELEMENT STRESS DISTRIBUTION
   !MORE REFERENCE ORIGINAL PAPER
   !BY KIRAN SOLANKI, S.R.DANIEWICZ, J.C. NEWMAN JR.
   NSEL,S,LOC,Y,0
   NSEL,R,LOC,X,0-(da*0.00001),AA+(da*0.00001)
   *GET,TNODE,NODE,,COUNT
   NODN=0
   NODN=NDNEXT(NODN)
   NX1=NX(NODN)
   FON1=NODN
   *DO,Z,2,TNODE
      NODN=NDNEXT(NODN)
      NX2=NX(NODN)
      *IF,NX2,LT,NX1,THEN
         NX1=NX2
         FON1=NODN
      *ENDIF
   *ENDDO
   NSEL,S,LOC,Y,0
   A1=AA-NX1
   NSEL,S,LOC,Y,0
   NSEL,R,LOC,X,0-(da*0.00001),AA+(da*0.00001)
   *DO,II,2,TNODE
      FON2=NNEAR(FON1)
      A2=NX(FON2)
      A2=AA-A2
      *GET,NODY1,NODE,FON1,RF,FY
      *GET,NODY2,NODE,FON2,RF,FY
      !CONVERSION OF NODAL FORCE IN TO ELEMENT STRESS
      NODY1=NODY1/da
      NODY2=NODY2/da
   *ENDIF
   *ENDDO
   NSEL,S,LOC,Y,0
   A1=AA-NX1
   NSEL,S,LOC,Y,0
   NSEL,R,LOC,X,0-(da*0.00001),AA+(da*0.00001)
   *DO,II,2,TNODE
      FON2=NNEAR(FON1)
      A2=NX(FON2)
      A2=AA-A2
      *GET,NODY1,NODE,FON1,RF,FY
      *GET,NODY2,NODE,FON2,RF,FY
      !CONVERSION OF NODAL FORCE IN TO ELEMENT STRESS
      NODY1=NODY1/da
      NODY2=NODY2/da
   *ENDIF
   *ENDDO

   !Find Kopening FOR CT/SEB Models
   !As Infinite Plate With A Semi-infinite Crack
   !With Linear Element Stress Distribution
   !More Reference Original Paper
   !By Kiran Solanki, S.R.Daniewicz, J.C. Newman JR.
BB = A1
CC = A2
C1 = (NODY2*BB - NODY1*CC) / (BB - CC)
C2 = (NODY1 - NODY2) / (BB - CC)
K = ((2/3)*sqrt(2/PI)*((sqrt(CC)*(3*C1+CC*C2)) - (sqrt(BB)*(3*C1+BB*C2)))) / KMAX + K

NODY1 = NODY1*da
NODY2 = NODY2*da
! OUTPUT FOR THE FILE READ AS
! NODE NO, NODE LOCATION, NODAL FORCE 1, KOPEN/KMAX, DISTANCE FROM THE CRACK TIP, NODAL FORCE 2
!*WRITE, FON1, NXLOC, NODY1, K, BB, NODY2
!(F6.0, 2X, E12.6, 2X, E12.6, 2X, E12.6, 2X, F8.6, 2X, E12.6)
NSEL, U, LOC, X, 0 - (da*0.00001), a1 + (da*0.00001)
FON1 = FON2
A1 = A2
NODY1 = 0
NODY2 = 0
*ENDDO
*ENDIF

*IF, MTYPE, EQ, 'SEB', THEN
!*FIND KMAX FOR CT/SEB SPECIMENS
!AW = AA / w ! A/W RATIO
B = 1 ! THICKNESS
F = ((2 + AW) / ((1 - AW)**1.5)) * (0.886 + (4.64*AW) - (13.32*(AW**2)) + (14.72*(AW**3)) - (5.6*(AW**4)))
KMAX = F * (StrsMax / (sqrt(W)*B))
!*FIND KOPENING FOR CT SPECIMEN MODEL
!*AS INFINITE PLATE WITH A SEMI-INFINITE CRACK
!*WITH LINEAR ELEMENT STRESS DISTRIBUTION
!*MORE REFERENCE ORIGINAL PAPER
!*BY KIRAN SOLANKI, S.R. DANIIEWICZ, J.C. NEWMAN JR.
NSEL, S, LOC, Y, 0
NSEL, R, LOC, X, 0 - (da*0.00001), aA + (da*0.00001)
*GET, TNODE, NODE, COUNT
NODN = 0
NODN = NDNEXT (NODN)
NX1 = NX (NODN)
FON1 = NODN
*DO, Z, 2, TNODE
NODN = NDNEXT (NODN)
NX2 = NX (NODN)
*IF, NX2, LT, NX1, THEN
NX1 = NX2
FON1 = NODN
*ENDIF
*ENDDO
a4 = nx1
A1 = AA - NX1
NSEL, S, LOC, Y, 0
NSEL, R, LOC, X, 0 - (da*0.00001), aA + (da*0.00001)
*DO, II, 2, TNODE
FON2 = NNEAR (FON1)
A3 = NX (FON2)
A2 = AA - A3
*GET, NODY1, NODE, FON1, RF, FY
*GET, NODY2, NODE, FON2, RF, FY
!CONVERSION OF NODAL FORCE IN TO ELEMENT STRESS
NODY1=NODY1/da
NODY2=NODY2/da
!Find Kopening FOR CT/SEB Models
!As Infinite Plate With A Semi-infinite Crack
!With Linear Element Stress Distribution
!More Reference Original Paper
!By Kiran Solanki, S.R.Daniewicz, J.C. Newman JR.

BB=A1
CC=A2
C1=((NODY2*BB)-(NODY1*CC))/(BB-CC)
C2=(NODY1-NODY2)/(BB-CC)
t1=3*C1*(sqrt(bb)-sqrt(cc))
t2=C2*((sqrt(BB)*BB)-(sqrt(CC)*CC))
tem=sqrt(2/PI)*(2/3)*(t1+t2)
K=(tem/kmax)+K

NODY1=NODY1*da
NODY2=NODY2*da

!OUTPUT FOR THE FILE READ AS
!NODE NO, NODE LOCATION, NODAL FORCE 1,
!KOPEN/KMAX, DISTANCE FROM THE CRACK TIP, NODALFORCE 2
*WRITE,FON1,NXLOC,NODY1,K,BB,NODY2
(F6.0,2X,E12.6,2X,E12.6,2X,F8.6,2x,E12.6)
NSEL,U,LOC,X,0-(da*0.00001),a4+(da*0.00001)
FON1=FON2
A1=A2
NODY1=0
NODY2=0
a4=a3

*ENDDO
*ENDIF
APPENDIX A7

ANSYS INPUT FILE LOADCRACK.MAC, LOAD MODEL, ANY LOAD RATIO
AUTOTS, ON
NSUBST, 1, 100, 1, OFF
*CFOpen, %JN%_load_%arg1%_dat

!*Wwrite
! (*"Node# Node r NodeAng S/Smax SY OStat Remote Stress")

CurrLInc=LIBCO
StrsLvl=StrsMin/StrsMax
RStrs=StrsMin
*DO, JJ, 1, arg1
SelCTNodes, JJ
SEL,R,D,UY, 0
*GET, NSNODES, NODE, , COUNT
*IF, NSNODES, GT, 0, THEN
NODNO=0
*DO, JJJ, 1, NSNODES
NODNO=NDNEXT(NODNO)
*GET, NODYSTRS, NODE, NODNO, S, Y
*IF, MTYPE, EQ, 'SC', THEN
CSYS, 11
NDANG=NY(NODNO)
NXLOC=NX(NODNO)
CSYS, 0
*ELSE
NDANG=NZ(NODNO)
NXLOC=NX(NODNO)
*ENDIF
NodeStat=0
*VWRITE, NODNO, NXLOC, NDANG, StrsLvl, NODYSTRS, NODEStat, RStrs
(F6.0, 2X, E12.6, 2X, E10.4, 2X, E12.6, 2X, E12.6, 2X, E12.6, 2X, E12.6)
!*ENDDO
*ENDIF
*ENDDO
*DO, J, 1, /LIDCO
TimeVar=TimeVar+CurrLInc*0.45
RStrs=(StrsMax-StrsMin)*(TimeVar-arg1)/0.45+StrsMin
*IF, TimeVar, GE, arg1+0.45, Then
RStrs=StrsMax
TimeVar=arg1+0.45
Time, TimeVar
AppLoad, height, RStrs
SOLVE
SAVE
*EXIT
*ENDIF
Time, TimeVar
StrsLvl=RStrs/StrsMax
AppLoad, height, RStrs
SOLVE
SAVE
ClearRST, BDrive, BDir, ''
OPENSTAT=0
OpnRwCnt=0
*DO,JJ,1,arg1
  SeICTNodes,JJ
NSEL,R,D,UY,0
*GET,NSNODES,NODE,,COUNT
*IF,NSNODES,GT,0,THEN
  NODNO=0
  *DO,JJJ,1,NSNODES
    NodeStat=0
    NODNO=NDNEXT(NODNO)
    *GET,NodYStrs,NODE,NODNO,S,Y
    *IF,MTYPE,EQ,'SC',THEN
      CSYS,11
      NDANG=NY(NODNO)
      NXLOC=NX(NODNO)
      CSYS,0
    *ELSE
      NDANG=NZ(NODNO)
      NXLOC=NX(NODNO)
    *ENDIF
    *IF,NodYStrs*1e10,GT,0,THEN
      CcurrLinc=LIDCO
      OPENSTAT=1
      NODESTAT=1
      DDELE,NODNO,UY
    *ENDIF
  *ENDIF
  *VWRITE,NODNO,NXLOC,NDANG,StrsLvl,NODYSTRS,NodeStat,RStrs
    (F6.0,2X,E12.6,2X,E10.4,2X,E12.6,2X,E12.6,2X,E12.6,2X,F4.0,2X,E12.6)
  *ENDDO
*ELSEIF,NSNODES,EQ,0,THEN
  OpnRwCnt=OpnRwCnt+1
  *ENDIF
*ENDDO
NSEL,ALL

! Block Added for negative R,

CMSEL,S,CSNODES
NSEL,R,D,UY,0
*GET,NSNODES,NODE,,COUNT
*IF,NSNODES,GT,0,THEN
  NODNO=0
  *DO,JJ,1,NSNODES
    NODNO=NDNEXT(NODNO)
    *GET,NodYStrs,NODE,NODNO,S,Y
    *IF,NodYStrs*1e10,GT,0,THEN
      OPENSTAT=1
      DDELE,NODNO,UY
    *ENDIF
  *ENDIF
  *ENDIF
*ENDDO
NSEL,ALL

! End of New Block

*IF,OPENSTAT,EQ,1,THEN
  Time,Timevar+CurrLinc*0.01
  SOLVE
  SAVE
  ClearRST,BDrive,BDir,''
*ENDIF
*IF,OpnRwCnt,EQ,arg1,THEN
  CurrLInc=LIACO
*ENDIF
*ENDDO
*CFCLOSE
APPENDIX A8

ANSYS INPUT FILE SELECTNODES.MAC, SELECT CRACK
TIP NODE, ANY LOAD RATIO
! SelCTNodes.mac
! Macro to Select the Crack tip nodes for load cycle "N"
!
! Executed as follows:
! SelCTNodes,N
!
!Set Coordinate System to Elliptical

*IF,MTYPE,EQ,'SC',THEN
! The following are for a surface crack: (Using Faleskog Numbering Scheme)
LOCAL,11,1,0,0,0,0,90,0,a/c
NSEL,S,LOC,X,c-0.3*da,c+0.3*da
NSEL,R,LOC,Z,0
*GET,FNODNO,NODE,,NUM,MIN
*GET,LNODNO,NODE,,NUM,MAX
DELTANN=NDNEXT(FNODNO)-FNODNO
*GET,NODCNT,NODE,,COUNT
NSEL,S,NODE,,FNODNO+(arg1-1),LNODNO+(arg1-1),DELTANN
CSYS,0
*ELSE
NSEL,S,LOC,Y,0
NSEL,R,LOC,X,c+da*(arg1-1.25),c+da*(arg1-0.75)
*ENDIF

!LOCAL,11,1,0,0,0,0,90,0,(a+da*(arg1-1))/(c+da*(arg1-1))
!NSEL,S,LOC,X,c+(arg1-1.45)*da,c+(arg1-0.55)*da
!NSEL,R,LOC,Z,0
!CSYS,0
APPENDIX A9

ANSYS INPUT FILE CLEARRST.MAC, DELETE UNNECESSARY OUTPUT FILES, ANY LOAD RATIO
This Macro Saves Disk Space by deleting 'jobname'.rst
It also provides a tool for stopped jobs by saving the
db and rst from the last completed loadstep to a backup
directory
Execution of this macro should be done with the following command:
ClearRST,BDrive,Bdir1,Bdir2
If Bdir2 is unnecessary, '' should be input
Modified 1/28/2000

SAVE
FINISH
!pltpzone
!/COPY,,RST,,,,%arg1%arg2%arg3%
!/COPY,,EMAT,,,,%arg1%arg2%arg3%
!/COPY,,OSAV,,,,%arg1%arg2%arg3%
!/COPY,,ESAV,,,,%arg1%arg2%arg3%
!/COPY,,MNTR,,,,%arg1%arg2%arg3%
!/COPY,,DB,,,,%arg1%arg2%arg3%
!/COPY,,DB,,%jn%_%arg4%,,%arg1%arg2%arg3%
!DELETE,,RST
!SOLU
!ATYPE,,REST
APPENDIX A10

ANSYS INPUT FILE APPLOAD.MAC, APPLICATION OF LOAD, ANY LOAD RATIO
This macro is used to apply Surface Pressure along the top of the hole in the CT model!
The center of the hole should be at coordinates\[ x = w, \ y = h \]!
Use should be as follows:
\[ \text{SCappLoad, height, Pressure} \]

*IF, MTYPE, EQ, 'CT', THEN
NSEL, S, LOC, X, \[ w \]
NSEL, R, LOC, Y, \[ \text{arg1} \]
NSEL, R, LOC, Z, \[ 0 \]
F, ALL, FY, \[ \text{arg2} \]
NSEL, ALL
*ENDIF

*IF, MTYPE, EQ, '2DCT', THEN
NSEL, S, LOC, X, \[ w \]
NSEL, R, LOC, Y, \[ \text{arg1} \]
NSEL, R, LOC, Z, \[ 0 \]
F, ALL, FY, \[ \text{arg2} \]
NSEL, ALL
*ENDIF

*IF, MTYPE, EQ, 'SEB', THEN
NSEL, S, LOC, Y, \[ \text{height} \]
NSEL, R, LOC, X, \[ 0-10E-5, 0+10E-5 \]
F, ALL, FY, \[ \text{arg2} \]
NSEL, S, LOC, Y, \[ \text{height} \]
NSEL, R, LOC, X, \[ \text{W} \]
F, ALL, FY, \[ -\text{arg2} \]
NSEL, all
*ENDIF

*IF, MTYPE, EQ, 'SC', THEN
NSEL, S, LOC, Y, \[ \text{arg1} \]
!D, ALL, UY, \[ \text{arg2} \]
SF, ALL, PRES, \[ -\text{arg2} \]
NSEL, ALL
*ENDIF

*IF, MTYPE, EQ, 'MT', THEN
NSEL, S, LOC, Y, \[ \text{arg1} \]
!D, ALL, UY, \[ \text{arg2} \]
SF, ALL, PRES, \[ -\text{arg2} \]
NSEL, ALL
*ENDIF
APPENDIX A11

ANSYS INPUT FILE *APPLOAD.MAC*, APPLICATION

OF LOAD, T-STRESS
This macro is used to apply Surface Pressure along the top of the hole in the CT model. The center of the hole should be at coordinates $x = w$, $y = h$. Use should be as follows: SCappLoad, height, Pressure

*IF, MTYPE, EQ, 'CT', THEN
  NSEL, S, LOC, X, w
  NSEL, R, LOC, Y, arg1
  NSEL, R, LOC, Z, 0
  F, ALL, FY, arg2
  NSEL, ALL
*ELSEIF, MTYPE, EQ, '2DCT'
  NSEL, S, LOC, X, w
  NSEL, R, LOC, Y, arg1
  NSEL, R, LOC, Z, 0
  ! nsel, s,,,,bon
  F, ALL, FY, arg2
  NSEL, ALL
*ELSEIF, MTYPE, EQ, '2D'
  NSEL, S, LOC, Y, arg1
  ! D, ALL, UY, arg2
  SF, ALL, PRES, -arg2
  NSEL, ALL
! T-STRESS
  NSEL, S, LOC, X, w
  SF, ALL, PRES, -arg2
  NSEL, ALL
*ELSE
  NSEL, S, LOC, Y, arg1
  ! D, ALL, UY, arg2
  SF, ALL, PRES, -arg2
  NSEL, ALL
*ENDIF
APPENDIX A12

ANSYS INPUT FILE *LOADCRACK.DAT*, LOAD MODEL, T-STRESS
AUTOTS, ON
NSUBST, 1, 100, 1, OFF
*CFCOPEN, &JN%_load_%arg1%, dat

!*VWRITE
! (*"NODE# Node r NodeAng S/Smax SY OStat Remote Stress")

CurrLInc=LIBCO
StrsLv1=StrsMin/StrsMax
RStrs=StrsMin
*DO, JJ, 1, arg1
   SelICTNodes, JJ
   NSEL, R, D, UY, 0
   *GET, NSNODES, NODE, , COUNT
   *IF, NSNODES, GT, 0, THEN
      NODNO=0
      *DO, JJJ, 1, NSNODES
         NODNO=NDNEXT(NODNO)
         *GET, NODYSTRS, NODE, NODNO, S, Y
         *IF, MTYPE, EQ, 'SC', THEN
            CSYS, 11
            NDANG=NY(NODNO)
            NXLOC=NX(NODNO)
            CSYS, 0
         *ELSE
            NDANG=NZ(NODNO)
            NXLOC=NX(NODNO)
         *ENDIF
      NodeStat=0
      *VWRITE, NODNO, NXLOC, NDANG, StrsLv1, NODYSTRS, NODESTATE, RSTRS
         (F6.0, 2X, E12.6, 2X, E10.4, 2X, F8.6, 2X, E12.6, 2X, F4.0, 2X, E12.6)
      *ENDDO
   *ENDIF
   *ENDDO
   *DO, J, 1, 1/LIDCO
      TimeVar=TimeVar+CurrLInc*0.45
      RStrs=(StrsMax-StrsMin)*(TimeVar-arg1)/0.45+StrsMin
      *IF, TimeVar, GE, arg1+0.45, Then
         RStrs=StrsMax
         TimeVar=arg1+0.45
         Time, TimeVar
         AppLoad, height, RStrs
         SOLVE
         SAVE
         *EXIT
      *ENDIF
      Time, TimeVar
      StrsLv1=RStrs/StrsMax
      AppLoad, height, RStrs
      SOLVE
      SAVE
      ClearRST, BDrive, BDir,''
      OPENSTAT=0
      OpnRwCnt=0
      *DO, JJ, 1, arg1
         SelICTNodes, JJ
         NSEL, R, D, UY, 0
         *GET, NSNODES, NODE, , COUNT
         *IF, NSNODES, GT, 0, THEN
         NODNO=0
      *ENDDO
      *ENDDO
*DO,JJJ,1,NSNODES
  NodeStat=0
  NODNO=NDNEXT(NODNO)
  *GET,NodYStrs,NODE,NODNO,S,Y
  *IF,MTYPE,EQ,'SC',THEN
    CSYS,11
    NDANG=NY(NODNO)
    NXLOC=NX(NODNO)
    CSYS,0
  *ELSE
    NDANG=NZ(NODNO)
    NXLOC=NX(NODNO)
  *ENDIF
  *IF,NodYStrs*1e10,GT,0,THEN
    CurrLInc=LIDCO
    OPENSTAT=1
    NODESTATE=1
    DDELETE,NODNO,UY
  *ENDIF
  *VWRITE,NODNO,NXLOC,NDANG,StrsLvl,NODYSTRS,NodeStat,RStrs
     (F6.0,2X,E12.6,2X,E10.4,2X,F8.6,2X,E12.6,2X,F4.0,2X,E12.6)
  *ENDIF
  *ELSEIF,NSNODES,EQ,0,THEN
    OpnRwCnt=OpnRwCnt+1
  *ENDIF
  *ENDDO
*ELSEIF,NSNODES,EQ,0,THEN
  OpnRwCnt=OpnRwCnt+1
*ENDIF
*ENDDO
NSEL,ALL
*IF,OPENSTAT,EQ,1,THEN
  Time,Timevar+CurrLInc*0.01
  SOLVE
  SAVE
  ClearRST,BDrive,BDir,''
  *ENDIF
*IF,OpnRwCnt,EQ,arg1,THEN
  CurrLInc=LIACO
*ENDIF
*ENDDO
*CFCLOSE
APPENDIX A13

ANSYS INPUT FILE *UNLOADCRCAK.DAT*, UNLOAD MODEL, T-STRESS
AUTOTS,ON
NSUBST,1,100,1,OFF
*CFOpen,"%JN%_unload_%arg1%.dat"
!*VWrite
! ("NODE# Node r NodeAng S/Smax UY OStat Remote Stress")
CurrLInc=UIBCC
StrsLv1=StrsMax/StrsMax
RStrs=StrsMax
*DO,JJ,1,arg1
SelCTNodes,JJ
NSEL,U,D,UY,0
*GET,NSNODES,NODE,,COUNT
*IF,NSNODES,GT,0,THEN
  NODNO=0
  *DO,JJJ,1,NSNODES
    NODNO=NDNEXT(NODNO)
    NODYSTRS=UY(NODNO)
    *IF,MTYPE,EQ,'SC',THEN
      CSYS,11
      NDANG=NY(NODNO)
      NXLOC=NX(NODNO)
      CSYS,0
    ELSE
      NDANG=NZ(NODNO)
      NXLOC=NX(NODNO)
    *ENDIF
  NodeStat=0
  *VWrite,NODNO,NXLOC,NDANG,StrsLv1,NODYSTRS,NODESTAT,RSTRS
    (F6.0,2X,E12.6,2X,E10.4,2X,F8.6,2X,E12.6,2X,F4.0,2X,E12.6)
  *ENDDO
*ENDIF
*ENDDO

*DO,J,1,1/UIDCC
TimeVar=TimeVar+CurrLInc*0.5
RStrs=StrsMax-(StrsMax-StrsMin)*(TimeVar+0.5-arg1)/0.5
*IF,TimeVar,GE,arg1,Then
  RStrs=StrsMin
  Time=TimeVar=arg1
  AppLoad,height,RStrs
  SOLVE
  SAVE
  !modified
*DO,JJ,1,arg1
SelCTNodes,JJ
NSEL,U,D,UY,0
*GET,NSNODES,NODE,,COUNT
*IF,NSNODES,GT,0,THEN
  NODNO=0
  *DO,JJJ,1,NSNODES
    NodeStat=0
    NODNO=NDNEXT(NODNO)
    NODYSTRS=UY(NODNO)
    *IF,MTYPE,EQ,'SC',THEN
      CSYS,11
      NDANG=NY(NODNO)
      NXLOC=NX(NODNO)
    ELSE
      NDANG=NZ(NODNO)
      NXLOC=NX(NODNO)
    *ENDIF
  *ENDDO
*ENDIF
*ENDDO


*ELSE
    NDANG=NZ (NODNO)
    NXLOC=NX (NODNO)
*ENDIF
*IF, NodYStrs*le10, LT, 0, THEN
    CurrLInc=UIDCC
    OPENSTAT=1
    NODESTAT=1
    D, NODNO, UY, 0
*ENDIF
*VWRITE, NODNO, NXLOC, NDANG, StrsLv1, NODEYSTRS, NodeStat, RStrs
        (F6.0, 2X, E12.6, 2X, E10.4, 2X, F8.6, 2X, E12.6, 2X, F4.0, 2X, E12.6)
*ENDDO
*endif
*enddo
!end of modification
*EXIT
*ENDIF
Time, TimeVar
StrsLv1=Strs/RStrs/StrsMax
AppLoad, height, RStrs
SOLVE
SAVE
ClearRST, BDrive, BDir, ''
OPENSTAT=0
OpnRwCnt=0
*DO, JJ, 1, arg1
    SelectNodes, JJ
    NSEL, U, D, UY, 0
*GET, NSNODES, NODE, /, COUNT
*IF, NSNODES, GT, 0, THEN
    NODNO=0
*DO, JJJ, 1, NSNODES
    NodeStat=0
    NODNO=NDNEXT (NODNO)
    NODEYSTRS=UY (NODNO)
*IF, MTYPE, EQ, 'SC', THEN
    CSYS, 11
    NDANG=NY (NODNO)
    NXLOC=NX (NODNO)
    CSYS, 0
*ELSE
    NDANG=NZ (NODNO)
    NXLOC=NX (NODNO)
*ENDIF
*IF, NodYStrs*le10, LT, 0, THEN
    CurrLInc=UIDCC
    OPENSTAT=1
    NODESTAT=1
    D, NODNO, UY, 0
*ENDIF
*VWRITE, NODNO, NXLOC, NDANG, StrsLv1, NODEYSTRS, NodeStat, RStrs
        (F6.0, 2X, E12.6, 2X, E10.4, 2X, F8.6, 2X, E12.6, 2X, F4.0, 2X, E12.6)
*ENDDO
*ELSEIF, NSNODES, EQ, 0, THEN
    OpnRwCnt=OpnRwCnt+1
*ENDIF
*ENDDO
NSEL, ALL
*IF, OPENSTAT, EQ, 1, THEN
Time,Timevar+CurrLInc*0.01
SOLVE
SAVE
ClearRST,BDrive,BDir,''
*ENDIF
*IF,OpnRwCnt,EQ,arg1,THEN
  CurrLInc=UIACC
*ENDIF
*ENDDO
save
*CFCLOSE
APPENDIX A14

ANSYS INPUT FILE StrtCyc.MAC, CONTROL OF CYCLIC LOADING, MIN LOAD RELEASE SHCEME
FirstLoad
ClearRST,BDrive,BDir,MaxDir

*DO,I,1,NLC
   UnloadCrack,I
   ClearRST,BDrive,BDir,MinDir
   LoadCrack,I
   ClearRST,BDrive,BDir,MaxDir
*ENDDO
APPENDIX B1

ANSYS INPUT FILE SEB.DAT, INPUT FILE FOR SEB SPECIMEN
! This runs the script "SCmodel.dat" to import the Solid
! Elements and Nodes from the files "jobname.crd" & "jobname.elm",
! and applies necessary boundary conditions for Crack Growth
! It then calls "SCLoadfile.dat" to run growth analysis.
!
!Note all lengths are in in, and pressures in Psi

/CONFIG,NPROC,1

!Loading information:

StrsMax=314.385 ! Maximum Applied Stress
StrsMin=0 ! Minimum Applied Stress
!NLC=1
NLC=40 ! Total Number of Loading Cycles to execute

!Geometry Information:

MTYPE='SEB'
t=0 ! Thickness of plate
w=20 ! Plate Half-Width
height=40 ! Model Height
c=2 ! Initial Crack half-length initial craktip node number 362
a=0 ! Initial Depth of Surface Crack
da=0.0185185 ! Crack Growth Increment

!Material Properties:

E=200e3 ! Young's Modulus
YS=230 ! Yield Stress

!Matrix Element Properties:

KCGELE=10**12 ! Crack Growth Element Stiffness
NCGECut=10 ! Number of bisections to matrix stiffness before
CGERF=2 ! Crack Growth Element Stiffness Reduction Factor

*get,JN,ACTIVE,,JOBNAM
/TITLE, Plasticity Induced Closure of model %JN%

AppBCs

BDrive='d:' ! Drive for file backups
BDir='\backup' ! Directory for file backups
MaxDir='\maxload' ! Directory for backup at Max Load
MinDir='\minload' ! Directory for Backup at Min Load

!Solution Information:

/SOLU ! Enter Solution Processor

LIBCO=0.0125 ! Loading Increment before crack opening
LIDCO=0.0125 ! Loading Increment during crack opening
LIACO=0.01250 ! Loading Increment after crack opening
UICCO=0.0125 ! Un-load Increment before crack closing
UIDCC=0.0125  ! Un-load Increment during crack closing
UIDACC=0.0125 ! Un-load Increment after crack closing

NSUB,1
NEQIT,8 ! Number of Equilibrium Iterations before bisection
NROPT,full,,on ! Full Newton Rapson Option, Adaptive Descent ON
EQSLV,PCG,1.0e-8 ! Use the Pre-Conditioned Conjugate Solver (In Core)

RESCONTROL,DEFINE,NONE,NONE,0
StrtCyc
APPENDIX B2

ANSYS INPUT FILE *MT.DAT*, INPUT FILE FOR M(T) SPECIMEN
/BATCH

! This runs the script "SCmodel.dat" to import the Solid
! Elements and Nodes from the files "jobname.crd" & "jobname.elm",
! and applies necessary boundary conditions for Crack Growth
! It then calls "SCLoadfile.dat" to run growth analysis.
!
!Note all lengths are in in, and pressures in Psi

/CONFIG,NPROC,1

!Loading information:

StrsMax=0.2*230 ! Maximum Applied Stress
StrsMin=-0.2*230 ! Minimum Applied Stress
NLC=45 ! Total Number of Loading Cycles to execute

!Geometry Information:

MTYPE='MT'
t=0 ! Thickness of plate
w=40 ! Plate Half-Width
height=40 ! Model Height
c=4 ! Initial Crack half-length initial craktip node number 362
a=0 ! Initial Depth of Surface Crack
da=0.005486968 ! Crack Growth Increment

!Material Properties:

E=200e3 ! Young's Modulus
YS=230 ! Yield Stress

!Matrix Element Properties:

KCGELE=10**12 ! Crack Growth Element Stiffness
NCGECut=5 ! Number of bisections to matrix stiffness before
CGERF=2 ! Crack Growth Element Stiffness Reduction Factor

*get,JN,ACTIVE,,JOBNAM
/TITLE, Plasticity Induced Closure of model %JN%

AppBCs

BDrive='c:' ! Drive for file backups
BDir='\backup' ! Directory for file backups
MaxDir='\maxload' ! Directory for backup at Max Load
MinDir='\minload' ! Directory for Backup at Min Load

!Solution Information:

/SOLU ! Enter Solution Processor

LIBCO=0.0125 ! Loading Increment before crack opening
LIDCO=0.0125 ! Loading Increment during crack opening
LIACO=0.01250 ! Loading Increment after crack opening
UIBCC=0.0125 ! Un-load Increment before crack closing
UIDCC=0.0125 ! Un-load Increment during crack closing
UIACC=0.01250 ! Un-load Increment after crack closing

NSUB, 1
NEQIT, 8 ! Number of Equilibrium Iterations before bisection
NROPT, FULL,, ON ! Full Newton Rapson Option, Adaptive Descent ON
EQSLV, PCG, 1.0e-8 ! Use the Pre-Conditioned Conjugate Solver (In Core)

RESCONTROL, DEFINE, NONE, NONE, 0
StrtCyc
APPENDIX B3

ANSYS INPUT FILE $MT.DAT$, INPUT FILE FOR M(T) SPECIMEN, T-STRESS
This is the input file for schiv_15.dat
This runs the script "SCmodel.dat" to import the Solid Elements and Nodes from the files "jobname.crd" & "jobname.elm",
and applies necessary boundary conditions for Crack Growth. It then calls "SCLoadfile.dat" to run growth analysis.

Note all lengths are in in, and pressures in Psi

/CONFIG,NPROC,1

!Loading information:

StrsMax=69 ! Maximum Applied Stress
StrsMin=0 ! Minimum Applied Stress

It stress for MT t=0.435

NLC=36 ! Total Number of Loading Cycles to execute

!Geometry Information:

MTYPE='MT'

t=0 ! Thickness of plate
w=40 ! Plate Half-Width

height=40 ! Model Height
c=4 ! Initial Crack half-length initial cracktip node number 362

a=0 ! Initial Depth of Surface Crack

da=0.0030864 ! Crack Growth Increment

!Material Properties:

E=200e3 ! Young's Modulus
YS=230 ! Yield Stress

!Matrix Element Properties:

KCGELE=10**12 ! Crack Growth Element Stiffness
NCGECut=10 ! Number of bisections to matrix stiffness before
CGERF=2 ! Crack Growth Element Stiffness Reduction Factor

*get,JN,ACTIVE,,JOBNAM
/TITLE, Plasticity Induced Closure of model %JN%

AppBCs

BDir='E:' ! Drive for file backups
BDir='\backup' ! Directory for file backups
MaxDir='\maxload' ! Directory for backup at Max Load
MinDir='\minload' ! Directory for Backup at Min Load

!Solution Information:

/SOLU ! Enter Solution Processor

LIBCO=0.05 ! Loading Increment before crack opening
LIDCO=0.025 ! Loading Increment during crack opening
LIACO=0.10 ! Loading Increment after crack opening
UIBCC=0.05 ! Un-load Increment before crack closing
UIDCC=0.025 ! Un-load Increment during crack closing
UIACC=0.10 ! Un-load Increment after crack closing
SOLCONTROL, ON
NSUB, 1
NEQIT, 8 ! Number of Equilibrium Iterations before bisection
NROPT, FULL, ,ON ! Full Newton Rapson Option, Adaptive Descent ON
EQSLV, PCG, 1.0e-8 ! Use the Pre-Conditioned Conjugate Solver (In Core)
RESCONTROL, DEFINE, NONE, NONE, 0

StrtCyc
APPENDIX C1

ANSYS INPUT FILE *APPBCS.MAC*, APPLICATION OF BOUNDARY CONDITIONS, SPIKE OVERLOAD
/prep7

! Element Shape Checking Off
SHPP,OFF

! Define Material Properties for Solid Elements
MP,EX,1,E
TB,BKIN,1,1,1,,
TBMODIF,2,1,YS
!TBMODIF,3,1,HTAN

*IF,MTYPE,EQ,'2D',THEN
  ET,1,PLANE42,,,2,,
  ! KEYOPT(3) = 0 Plane Stress
  ! KEYOPT(3) = 2 Plane Strain
  ! KEYOPT(3) = 3 Plane Stress w/ thk
*ELSE
  ET,1,SOLID45,,,,,
*ENDIF

! Begin Building Model: Read Solid Elements from File
MAT,1
TYPE,1
REAL,1
nread,%JN%,crd
eread,%JN%,elm

*IF,MTYPE,EQ,'CT',THEN
  ! Create Linear Elastic Material Properties for Elastic "plug"
  MP,EX,2,E
  LOCAL,12,1,w,height,0,0,0,0
  NSEL,S,LOC,X,0,r
  ESLN,S,1
  EMODIF,ALL,MAT,2
  NSEL,ALL
  ESEL,ALL
  CSYS,0
*ENDIF

!Constrain Nodes on Bottom Surface of Plate in the Vertical-direction
!(Constraints will be removed during crack growth)
!
! Also, create component containing crack surface nodes (used
! when negative loads are applied). Assumes elliptical geometries
! are notched, all others are not.

*IF,MTYPE,EQ,'SC',THEN
  LOCAL,11,1,0,0,0,0,90,0,a/c
  NSEL,S,LOC,X,c,100000
  NSEL,R,LOC,Z,0,0
  NSEL,U,LOC,X,0,c-0.05*da
  D,ALL,UY,0
  CSYS,0
*ELSE
  NSEL,S,LOC,Y,0
  NSEL,R,LOC,X,c,100000
*ENDIF
NSEL,U,LOC,X,0,c-da*0.25
D,ALL,UY,0

*ENDIF

!Apply Appropriate Conditions in X-direction:
*IF,MTYPE,EQ,'CT',THEN
  NSEL,S,LOC,Y,0
  NSEL,R,LOC,Z,0
  NSEL,R,LOC,X,3.75
  D,ALL,UX,0
  NSEL,ALL
*ELSE
  NSEL,S,LOC,X,0
  D,ALL,UX,0
  NSEL,ALL
*ENDIF

!Apply Symmetry BC's at Z=0 plane
*IF,MTYPE,EQ,'SC',THEN
  NSEL,S,LOC,Y,0
  NSEL,R,LOC,X,0
  NSEL,R,LOC,Z,t
  D,ALL,UZ,0
  NSEL,ALL
*ELSE
  NSEL,S,LOC,Z,0
  D,ALL,UZ,0
  NSEL,ALL
*ENDIF

!Select Crack Mouth Node...Create Component CMNODES

NSEL,S,LOC,X,0
NSEL,R,LOC,Y,0
CM,CMNODES,NODE

CMSEL,ALL
NSEL,ALL
ESEL,ALL

WSORT,ALL,0  !Sort Elements to minimize wavefront

SAVE
FINISH
APPENDIX C2

ANSYS INPUT FILE *StrtCyc.MAC*, CONTROL OF CYCLIC
LOADING, SPIKE OVERLOAD
TimeVar=0.45
FirstLoad
ClearRST,BDrive,BDir,MaxDir

PE=0
SPAC=0
POP=0

*DO,I,1,NLC

*IF,PE,NE,0,THEN
  I=PE+1
*ENDIF
*IF,I,EQ,1,THEN
  UnloadCrack,I
  ClearRST,BDrive,BDir,MinDir
*DO,O,1,I
  JK_%O% = 1
*ENDDO
  LoadCrack,I
  ClearRST,BDrive,BDir,MaxDir
  SU=I
  SU_%I%=I
  POP=POP+1
  POP_%POP%=I
  PE=I

*ELSE

*IF,spike,EQ,I,THEN
!sort the array in ascending order
  *MOPER,ORDE(1),JUNK_%SU%(1,1),sort,JUNK_%SU%(1,1)
  VES=JUNK_%SU%(nodeNO,1)
  FUNI=SU+VES
  VES_%FUNI%=VES
!rearranging the array
  *MOPER,ORDE(1),JUNK_%SU%(1,1),sort,ORDE(1,1)
  JUMP=1
  JUMP_%I%=1

*DO,TEE,1,VES

*IF,TEE,GT,1,THEN
  I=I+1
  JUMP_%I%=JUMP+1
  JUMP=JUMP+1
*IF,TEE,EQ,VES,THEN
  JUMP_%I%=0
*ENDIF
*ENDIF

*IF,TEE,EQ,VES,THEN
  JUMP_%I%=0
  POP_%I-TEE+1%=I
  POP=I-TEE+1
*ENDIF
StrsMax=spmax
UnloadCrack,I
ClearRST,BDrive,BDir,MinDir
*DO,O,1,I
  JK_%O%=1
*ENDDO
LoadCrack,I
ClearRST,BDrive,BDir,MaxDir
*ENDDO
TEE=VES
NLC=NLC+TEE-1
SU=I
POP=POP+1
POP_%I-TEE+1%=I
JUMP=1
SU_%I%=I
PE=I
SPAC=1
save
*ELSE
StrsMax=smax
*MOPER,ORDE(1),JUNK_%SU%(1,1),sort,JUNK_%SU%(1,1)
VES=JUNK_%SU%(nodeNO,1)
FUNI=SU+VES
VES_%FUNI%=VES
!rearranging the array
*MOPER,ORDE(1),JUNK_%SU%(1,1),sort,ORDE(1,1)
JUMP=1
JUMP_%I%=1
*DO,TEE,1,VES
*IF,TEE,EQ,1,THEN
  *IF,SPAC,EQ,0,THEN
    spike=spike+VES-1
  *ENDIF
*ENDIF
*IF,TEE,GT,1,THEN
  I=I+1
  JUMP_%I%=JUMP+1
  JUMP=JUMP+1
*IF,TEE,EQ,VES,THEN
  JUMP_%I%=0
*ENDIF
*ENDIF
*IF,TEE,EQ,VES,THEN
  POP=POP+1
  POP_%POP%=I
  JUMP_%I%=0
*ENDIF
UnloadCrack,I
ClearRST,BDrive,BDir,MinDir
*DO,0,1,I
  JK_%I%=1
*ENDDO
LoadCrack,I
ClearRST,BDrive,BDir,MaxDir
*ENDDO
TEE=VES
NLC=NLC+TEE-1
SU=I
JUMP=1
SU_%I%=I
PE=I
*ENDIF
*ENDIF
SAVE
*IF,PE,EQ,nic,exit
*ENDDO
APPENDIX C3

ANSYS INPUT FILE RAJU.MAC, NEWMAN-RAJU

CONSTANT, SPIKE OVERLOAD
!CONSTANTS FOR NEWMAN-RAJU EQUATION

PI=(1989/22+7)**0.25
M1=1.13-(0.09*(a/c))
M2=-0.54+(0.89/(0.2+(a/c)))
M3=0.5-(1.0/(0.65+(a/c)))+14*((1-(a/c))**24)
QW=1+(1.464*((a/c)**1.65))
FW=sqrt(cos(((pi*c)/(2*w))*sqrt(a/t)))

StrtCyc
APPENDIX C4

ANSYS INPUT FILE SELECTNODES.MAC, SELECT

CRACK FRONT, SPIKE OVERLOAD
SelCTNodes.mac
Macro to Select the Crack tip nodes for load cycle "N"

Executed as follows:
SelCTNodes,N

Set Coordinate System to Elliptical

*IF,argl,EQ,1,THEN
  !for first cycle
  LOCAL,11,1,0,0,0,0,90,0,a/c
  NSEL,S,LOC,X,c-0.3*da,c+0.3*da
  NSEL,R,LOC,Z,0
  *GET,FNODNO,NODE,,NUM,MIN
  *GET,LNODNO,NODE,,NUM,MAX
  DELTANN=NDNEXT(FNODNO)-FNODNO
  *GET,NODCNT,NODE,,COUNT
  nodeNO=((LNODNO-FNODNO)/DELTANN)+1
  NSEL,S,NODE,,FNODNO+(arg1-1),LNODNO+(arg1-1),DELTANN
  CSYS,0
*ENDIF

*IF,JUMP_%arg1%,GE,1,THEN
  *IF,arg1,NE,1,THEN
  !selection of node
  LOCAL,11,1,0,0,0,0,90,0,a/c
  NSEL,S,LOC,X,c-0.3*da,c+0.3*da
  NSEL,R,LOC,Z,0
  NSEL,S,NODE,,JUNK_%arg1%(1,3)+JUMP_%arg1%,JUNK_%arg1%(1,3)+JUMP_%arg1%,1
  *DO,QP,2,DD_%arg1%
    nsel,a,NODE,,JUNK_%arg1%(QP,3)+JUMP_%arg1%,JUNK_%arg1%(QP,3)+JUMP_%arg1%,1
  *ENDDO
  CSYS,0
  *ENDIF
*ENDIF

*IF,JUMP_%arg1%,EQ,0,THEN
*IF,arg1,NE,1,THEN
  !selection of node
  LOCAL,11,1,0,0,0,0,90,0,a/c
  NSEL,S,LOC,X,c-0.3*da,c+0.3*da
  NSEL,R,LOC,Z,0
  KII=VES_%arg1%
  nsel,s,,,JUNK_%arg1-KII%(1,3)+JUNK_%arg1-KII%(1,1),JUNK_%arg1-KII%(1,3)+JUNK_%arg1-KII%(1,1),1
  *DO,K,2,nodeNO
    nsel,a,,,JUNK_%arg1-KII%(K,3)+JUNK_%arg1-KII%(K,1),JUNK_%arg1-KII%(K,3)+JUNK_%arg1-KII%(K,1),1
  CSYS,0
  *ENDDO
KIR_%arg1%=arg1
KIR=arg1
*ENDIF
*ENDIF
APPENDIX C5

ANSYS INPUT FILE *LOADCRACK.MAC*, LOAD MODEL, SPIKE OVERLOAD
AUTOTS,ON
NSUBST,1,100,1,OFF
*CFOPEN,%JN%_load_%arg1%,dat

!*WRTE
! (*NODE# Node r NodeAng S/Smax SY OStat Remote Stress*)

*DO,JK,1,arg1-1
*DO,KJ,1,K_%JK%
    NEXT_%JK%(KJ,2)=0
*ENDDO
*ENDDO

SelCTNodes,arg1
NSEL,R,D,UY,0
*GET,NSNODES,NODE,,COUNT
*DIM,NEXT_%arg1%,array,NSNODES,4
nsel,all
CurrlInc=LIDCO
StrsLvl=StrsMin/StrsMax
RStrs=StrsMin

*DO,JJ,1,arg1
    SelCTNodes,JJ
    NSEL,R,D,UY,0
    *GET,NSNODES,NODE,,COUNT
    *DIM,PRE_%JJ%,array,NSNODES,3
    KA_%JJ%=NSNODES
    !storing the no. of node for NEXT cycle
    *IF,JJ,EQ,arg1,THEN
        *GET,FNO,NODE,,NUM,MIN
        *GET,LNO,NODE,,NUM,MAX
        K_%JJ%=NSNODES
    *ENDIF
    *IF,NSNODES,GT,0,THEN
        NODNO=0
        *DO,JJJ,1,NSNODES
            NODNO=NDNEXT(NODNO)
            *GET,NODYSTRS,NODE,NODNO,S,Y
            *IF,MTYPE,EQ,'SC',THEN
                CSYS,11
                NDANG=NY(NODNO)
                NXLOC=NX(NODNO)
                CSYS,0
            *ELSE
                NDANG=NZ(NODNO)
                NXLOC=NX(NODNO)
            *ENDIF
        NodeStat=0
        *IF,NodYStrs*1e10,GT,0,THEN
            CurrlInc=LIDCO
            OPENSTAT=1
            NODESTAT=1
            DDELE,NODNO,UY
        *ENDIF
    *ENDIF

*WRITE,NODNO,NXLOC,NDANG,StrsLvl1,NODYSTRS,NODESTAT,RSTRS
    (F6.0,2X,E12.6,2X,E10.4,2X,F8.6,2X,E12.6,2X,F4.0,2X,E12.6)
*IF, NodeStat, EQ, 1, THEN
  *IF, JJ, LT, arg1, THEN
  *DO, RS, 1, K_%JJ%
    *IF, nodno, EQ, NEXT_%JJ%(RS, 3), THEN
    NEXT_%JJ%(RS, 2) = StrsMin/StrsMax
    *ENDIF
  *ENDIF
*ELSE
  NEXT_%JJ%(RS, 2) = StrsMin/StrsMax
  *ENDIF
*ENDIF

*ELSE
  NEXT_%JJ%(JK_%JJ%, 4) = NDANG
  NEXT_%JJ%(JK_%JJ%, 3) = NODNO
  NEXT_%JJ%(JK_%JJ%, 2) = StrsMin/StrsMax
  JK_%JJ% = JK_%JJ% + 1
  *ENDIF
*ELSE
  PRE_%JJ%(JJJ, 3) = NODNO
  PRE_%JJ%(JJJ, 2) = StrsLvl
  PRE_%JJ%(JJJ, 1) = NODYSTRS
  *ENDIF
*ENDIF
*ENDDO
*ENDIF
*ENDDO

*DO, J, 1, 1/LIDCO
  TimeVar = TimeVar + CurrLInc * 0.45
  RStrs = (StrsMax - StrsMin) * (TimeVar - arg1) / 0.45 + StrsMin
  *IF, TimeVar, GE, arg1 + 0.45, THEN
    RStrs = StrsMax
    TimeVar = arg1 + 0.45
    Time, TimeVar
    AppLoad, height, RStrs
    SOLVE
    SAVE
  *EXIT
  *ENDIF
*ENDIF
Time, TimeVar
StrsLvl = RStrs/StrsMax
AppLoad, height, RStrs
SOLVE
SAVE
ClearRST, BDrive, BDir, ''
OPENSTAT = 0
OptronCnt = 0
*DO, JJ, 1, arg1
  SelectNodes, JJ
  NSEL, R, D, UY, 0
  *GET, NSNODES, NODE,, COUNT
  *IF, NSNODES, GT, 0, THEN
    NODNO = 0
    *DO, JJJ, 1, NSNODES
      NodeStat = 0
      NODNO = NDNEXT (NODNO)
    *GET, NodYStrs, NODE, NODNO, S, Y
    *IF, MTYPE, EQ, 'SC', THEN
      CSYS, 11
      NDANG = NY (NODNO)
      NXLOC = NX (NODNO)
      CSYS, 0
    *ENDIF
  *ENDIF
*ELSE
   NDANG=NZ (NODNO)
   NXLOC=NX (NODNO)
*ENDIF
*IF,NodYStrs*1e10,GT,0,THEN
   CurrLInc=LIDCO
   OPENSTAT=1
   NODESTAT=1
   DDELE,NODNO,UY
*ENDIF
*VWRITE,NODNO,NXLOC,NDANG,StrsLvl,NODYSTRS,NodeStat,RStrs
   (F6.0,2X,E12.6,2X,E10.4,2X,F8.6,2X,E12.6,2X,F4.0,2X,E12.6)
*IF,NodeStat,EQ,1,THEN
   *IF,JJ,LT,arg1,THEN
   *DO,L,1,KA_%JJ%
      *IF,nodno,EQ,PRE_%JJ% (1,3),THEN
      JUNK= (NODYSTRS-PRE_%JJ% (L,1))
      Q1=PRE_%JJ% (L,2)
      *DO,RS,1,K_%JJ%
         *IF,nodno,EQ,NEXT_%JJ% (RS,3),THEN
         NEXT_%JJ% (RS,2)=((NODYSTRS*Q1)-(PRE_%JJ% (L,1)*StrsLvl))/JUNK
         *ENDIF
         *ENDIF
      *ENDDO
   *ENDIF
   *ENDDO
   *ELSE
   NEXT_%JJ% (JK_%JJ%,4)=NDANG
   NEXT_%JJ% (JK_%JJ%,3)=NODNO
   *DO,L,1,KA_%JJ%
      *IF,NODNO,eq,PRE_%JJ% (L,3),THEN
      JUNK= (NODYSTRS-PRE_%JJ% (L,1))
      Q1=PRE_%JJ% (L,2)
      NEXT_%JJ% (JK_%JJ%,2)=((NODYSTRS*Q1)-(PRE_%JJ% (L,1)*StrsLvl))/JUNK
   *ENDIF
   *ENDDO
   JK_%JJ%=JK_%JJ%+1
   *ENDIF
*ELSE
   *DO,L,1,KA_%JJ%
      *IF,PRE_%JJ% (L,3),EQ,NODNO,THEN
      PRE_%JJ% (L,3)=NODNO
      PRE_%JJ% (L,2)=StrsLvl
      PRE_%JJ% (L,1)=NODYSTRS
   *ENDIF
   *ENDDO
   *ENDIF
*ENDIF
NSEL,ALL
*IF,OPENSTAT,EQ,1,THEN
   Time,Timevar+CurrLInc*0.01
   SOLVE
   SAVE
   ClearRST,BDrive,BDir,''
*ENDIF
*IF,OpnRwCnt,EQ,arg1,THEN
   CurrLInc=LIA CO
EndIf
EndDo
CfClose

*If,arg1,Eq,1,Then
!Newman Raju Equation to find SIF
*Dim,Tempi,Array,NodeNo
*Dim,Fa,Array,NodeNo
*Dim,G,Array,NodeNo
*Dim,F,Array,NodeNo
*Dim,Kmax,Array,NodeNo
*Dim,Kopen,Array,NodeNo
*Dim,Cons,Array,NodeNo

*Do,Jjj,1,NodeNo
Tempi(Jjj)=Next_%Arg1%(Jjj,4)*(pi/180)
Aa=Nz(Fno)
Cc=Nx(Lno)
Cons(Jjj)=(8.1932965e-9)*Tempi(Jjj)+1.503e-8
Fa(Jjj)=(((Aa/Cc)**2)*(cos(Tempi(Jjj)**2))+(sin(Tempi(Jjj)**2))**0.25
G(Jjj)=1+((0.1+0.35*((Aa/t)**2))*((1-sin(Tempi(Jjj))**2))
f(Jjj)=((ml+((m2*(Aa/t)**2))+((m3*(Aa/t)**4)))*Fa(Jjj)*G(Jjj)*Fw
Kmax(Jjj)=Strsmax*sqrt(PI*(Aa/Qw))*F(Jjj)
Kopen(Jjj)=Next_%Arg1%(Jjj,2)*Strsmax*sqrt(PI*(Aa/Qw))*F(Jjj)
Next_%Arg1%(Jjj,1)=((Kmax(Jjj)-Kopen(Jjj))**2.881)*Cons(Jjj)
*EndDo

!sort the array in Ascending order
*Dim,Orde,,NodeNo
*Moper,Orde(1),Next_%Arg1%(1,1),Sort,Next_%Arg1%(1,1)

!Delta k effective
*Do,Jjj,2,NodeNo
Next_%Arg1%(Jjj,1)=Nint(Next_%Arg1%(Jjj,1)/Next_%Arg1%(1,1))
*EndDo

!Min. delta K effective
Next_%Arg1%(1,1)=1
Ppq=Next_%Arg1%(NodeNo,1)
!rearranging the array
*Moper,Orde(1),Next_%Arg1%(1,1),Sort,Orde(1,1)

!Exchange of the matrix values to store permanently
*Dim, Junk_%Arg1%, Array, NodeNo, 4
*Do,Jjj,1,NodeNo
Junk_%Arg1%(Jjj,1)=Next_%Arg1%(Jjj,1)
Junk_%Arg1%(Jjj,2)=Next_%Arg1%(Jjj,2)
Junk_%Arg1%(Jjj,3)=Next_%Arg1%(Jjj,3)
Junk_%Arg1%(Jjj,4)=Next_%Arg1%(Jjj,4)
*EndDo

*If,Ppq,Gt,1,Then
*Do,PPQ,2,Ppq
Dd_%FPQ%=0
*DO,QA,1,nodeNO
*IF,JUNK_%arg1%(QA,1),GE,PPQ,THEN
  DD_%PPQ%=DD_%PPQ%+1
*ENDIF
*ENDDO
*ENDDO

*IF,PQP,GT,1,THEN
*DO,PPQ,2,PQP
  QQQ=1
  *DIM,JUNK_%PPQ%,array,DD_%PPQ%,4
  *DO,QA,1,nodeNO
    *IF,JUNK_%arg1%(QA,1),GE,PPQ,THEN
      JUNK_%PPQ%(QQQ,1)=JUNK_%arg1%(QA,1)
      JUNK_%PPQ%(QQQ,2)=JUNK_%arg1%(QA,2)
      JUNK_%PPQ%(QQQ,3)=JUNK_%arg1%(QA,3)
      JUNK_%PPQ%(QQQ,4)=JUNK_%arg1%(QA,4)
      QQQ=1+QQQ
    *ENDIF
  *ENDDO
*ENDDO
*ENDIF

!write array in to file
*CFCOPEN,%JN%_mat_%arg1%,dat
  *vwrite,JUNK_%arg1%(1,1),JUNK_%arg1%(1,2),JUNK_%arg1%(1,3),JUNK_%arg1%(1,4)
  (F8.6,2X,E12.6,2X,F6.0,2X,E12.6,2X)
*CFCLOSE

*ELSE

!swap the required matrix
*IF,JUMP,EQ,YES,THEN
  *DIM,JUNK_%arg1%,array,nodeNO,4
  *DO,KO,1,nodeNO
    JUNK_%arg1%(KO,1)=NEXT_%arg1%(KO,1)
    JUNK_%arg1%(KO,2)=NEXT_%arg1%(KO,2)
    JUNK_%arg1%(KO,3)=NEXT_%arg1%(KO,3)
    JUNK_%arg1%(KO,4)=NEXT_%arg1%(KO,4)
  *ENDDO

!SPIke overload modIFication- for cycles after SPIke

*IF,spac,EQ,1,THEN
  *DIM, SPIK_%arg1%, array, nodeno, 6
  !up to second cycle
  DE=POP_2
  *DO,SP,1,K,DE%
  *DO,SPS,1,K,1
    *IF,NEXT_1(SPS,3)+NEXT_1(SPS,1),EQ,NEXT_%DE%(SP,3),THEN
      *IF,NEXT_1(SPS,2),LT,NEXT_%DE%(SP,2),THEN
        SPIK_%arg1%(SP,1)=NEXT_%DE%(SP,1)
        SPIK_%arg1%(SP,2)=NEXT_%DE%(SP,2)
        SPIK_%arg1%(SP,3)=NEXT_%DE%(SP,3)
        SPIK_%arg1%(SP,4)=NEXT_%DE%(SP,4)
        SPIK_%arg1%(SP,5)=NEXT_%DE%(SP,1)
        SPIK_%arg1%(SP,6)=NEXT_%DE%(SP,3)
      ELSE
        SPIK_%arg1%(SP,1)=NEXT_1(SPS,1)
        SPIK_%arg1%(SP,2)=NEXT_1(SPS,2)

      *ENDIF
    *ENDIF
  *ENDDO
*ENDIF

ELSE
SPIK_%arg1%(SP,3)=NEXT_1(SPS,3)
SPIK_%arg1%(SP,4)=NEXT_1(SPS,4)
SPIK_%arg1%(SP,5)=NEXT_%DE%(SP,1)
SPIK_%arg1%(SP,6)=NEXT_%DE%(SP,3)
*ENDIF
*ENDIF
*ENDDO
*ENDDO

*DO,SP,3,POP-1
SS=POP_%SP%
*DO,SPSS,1,K_%SS%
*IF,NEXT_%SS%(SPSS,3),EQ,SPIK_%arg1%(SPP,5)+SPIK_%arg1%(SPP,6),THEN
*IF,NEXT_%SS%(SPSS,2),LT,SPIK_%arg1%(SPP,2),THEN
SPIK_%arg1%(SPP,5)=NEXT_%SS%(SPSS,1)
SPIK_%arg1%(SPP,6)=NEXT_%SS%(SPSS,3)
*ELSE
SPIK_%arg1%(SPP,1)=NEXT_%SS%(SPSS,1)
SPIK_%arg1%(SPP,2)=NEXT_%SS%(SPSS,2)
SPIK_%arg1%(SPP,3)=NEXT_%SS%(SPSS,3)
SPIK_%arg1%(SPP,4)=NEXT_%SS%(SPSS,4)
SPIK_%arg1%(SPP,5)=NEXT_%SS%(SPSS,1)
SPIK_%arg1%(SPP,6)=NEXT_%SS%(SPSS,3)
*ENDIF
*ENDIF
*ENDDO
*ENDDO
*ENDDO

! for current cycle

*DO,SP,1,K_1
*DO,SPS,1,K_1
*IF,SPIK_%arg1%(SPS,5)+SPIK_%arg1%(SPS,6),EQ,JUNK_%arg1%(SP,3),THEN
*IF,SPIK_%arg1%(SPS,2),LT,JUNK_%arg1%(SP,2),THEN
SPIK_%arg1%(SPS,2)=JUNK_%arg1%(SP,2)
SPIK_%arg1%(SPS,3)=JUNK_%arg1%(SP,3)
SPIK_%arg1%(SPS,4)=JUNK_%arg1%(SP,4)
*ENDIF
*ENDIF
*ENDDO
*ENDDO

!END of modification

!Newman Raju Equation to find SIF

*DO,JJJ,1,nodeNO
AA=N2(fno)
CC=NX(lno)
TEMPI(JJJ)=SPIK_%arg1%(JJJ,4)*(PI/180)
CONS(JJJ)=(8.1932965e-9)*TEMPI(JJJ)+1.503e-8
FA(JJJ)=((((AA/CC)**2)*(cos(TEMPI(JJJ))**2)+(sin(TEMPI(JJJ))**2))**0.25
G(JJJ)=1+((0.1+0.35*((AA/t)**2))*((1-sin(TEMPI(JJJ)))**2))
F(JJJ)=((M1+(M2*((AA/t)**2))+(M3*((AA/t)**4)))*FA(JJJ)*G(JJJ)*FW
KMAX(JJJ)=strsMax*sqrt(PI*(AA/QW))*F(JJJ)
KOPEN(JJJ)=SPIK_%arg1%(JJJ,2)*strsMax*sqrt(pi*(AA/QW))*F(JJJ)
SPIK_%arg1%(JJJ,1)=((KMAX(JJJ)-KOPEN(JJJ))*2.881)*CONS(JJJ)
*ENDDO

! swap the delta k value to the current JUNK mat

*DO,B,1,nodeNO
*DO,BB,1,nodeNO
  *IF,SPIK_%arg1%(BB,5)+SPIK_%arg1%(BB,6),EQ,JUNK_%arg1%(B,3),THEN
    JUNK_%arg1%(B,1)=SPIK_%arg1%(BB,1)
  *ENDIF
*ENDDO
*ENDDO

! sort the array in ascending order
*DIM,ORDE,,nodeNO
*MOPER,ORDE(1),JUNK_%arg1%(1,1),sort,JUNK_%arg1%(1,1)

! Delta K effective

*DO,JJJ,2,nodeNO
  JUNK_%arg1%(JJJ,1)=NINT(JUNK_%arg1%(JJJ,1)/JUNK_%arg1%(1,1))
*ENDDO

! min. delta K effective factor

JUNK_%arg1%(1,1)=1
PQP=JUNK_%arg1%(nodeNO,1)

! rearranging the array

*MOPER,ORDE(1),JUNK_%arg1%(1,1),sort,ORDE(1,1)

*IF,PQP,GT,1,THEN
  *DO,PPQ,2,PQP
    QLQ=arg1+PPQ-1
    DD_%QLQ%=0
  *DO,QA,1,nodeNO
    *IF,JUNK_%arg1%(QA,1),GE,PPQ,THEN
      DD_%QLQ%=DD_%QLQ%+1
    *ENDIF
  *ENDIF
  *ENDDO
*ENDIF

*IF,PQP,GT,1,THEN
  *DO,PPQ,2,PQP
    QQQ=1
    QLQ=arg1+PPQ-1
    *DIM,JUNK_%QLQ%,array,DD_%QLQ%,4
*DO,QA,1,nodeNO
*IF,JUNK_%arg1%(QA,1),GE,PPQ,THEN
  JUNK_%QLQ%(QQQ,1)=JUNK_%arg1%(QA,1)
  JUNK_%QLQ%(QQQ,2)=JUNK_%arg1%(QA,2)
  JUNK_%QLQ%(QQQ,3)=JUNK_%arg1%(QA,3)
  JUNK_%QLQ%(QQQ,4)=JUNK_%arg1%(QA,4)
  QQQ=1+QQQ
*ENDIF
*ENDDO
*ENDDO
*ENDIF

!write array in to file

*CFOpen,%JN%_mat_%arg1%,dat
*VWrite,JUNK_%arg1%(1,1),JUNK_%arg1%(1,2),JUNK_%arg1%(1,3),JUNK_%arg1%(1,4)
  (F8.6,2X,E12.6,2X,F6.0,2X,E12.6,2X)
*CFClose

*ELSE

!for normal cycles, up to SPike overload

!Newman Raju Equation to find SIF

*DO,JJJ,1,nodeNO
  AA=N2(FNO)
  CC=NX(LNO)
  TEMPI(JJJ)=JUNK_%arg1%(JJJ,4)*(PI/180)
  CONS(JJJ)=(8.1932965e-9)*TEMPI(JJJ)+1.503e-8
  FA(JJJ)=(((AA/CC)**2)*((cos(TEMPI(JJJ))**2))+(sin(TEMPI(JJJ))**2))*0.25
  G(JJJ)=1+((0.1+0.35*(((AA/t)**2))*((1-sin(TEMPI(JJJ))**2)))
  F(JJJ)=((M1+(M2*(((AA/t)**2))+M3*(((AA/t)**4)))*FA(JJJ))*G(JJJ)*FW
  KMAX(JJJ)=strsMax*sqrt(PI*(AA/QW))*F(JJJ)
  KOPEN(JJJ)=JUNK_%arg1%(JJJ,2)*strsMax*sqrt(PI*(AA/QW))*F(JJJ)
  JUNK_%arg1%(JJJ,1)=((KMAX(JJJ)-KOPEN(JJJ))**2.881)*CONS(JJJ)
*ENDDO

!sort the array in ascending ORDER

*DIM,ORDE,,nodeNO
*MOPER,ORDE(1),JUNK_%arg1%(1,1),sort,JUNK_%arg1%(1,1)

!Delta K effective

*DO,JJJ,2,nodeNO
  JUNK_%arg1%(JJJ,1)=NINT(JUNK_%arg1%(JJJ,1)/JUNK_%arg1%(1,1))
*ENDDO

!min. delta K effective Factor

  JUNK_%arg1%(1,1)=1
  PQP=JUNK_%arg1%(nodeNO,1)

!rearranging the array

*MOPER,ORDE(1),JUNK_%arg1%(1,1),sort,ORDE(1,1)
*DO, SP, 1, nodeNo
*DO, SPS, 1, nodeNo
*IF, JUNK_%arg1%(SP, 3), EQ, NEXT_%arg1%(SPS, 3), THEN
  NEXT_%arg1%(SPS, 1) = JUNK_%arg1%(SP, 1)
*ENDIF
*ENDDO
*ENDDO

*IF, PPQ, GT, 1, THEN
*DO, PPQ, 2, PPQ
  QLO = arg1 + PPQ - 1
  DD_%QLQ% = 0
*DO, QA, 1, nodeNO
  *IF, JUNK_%arg1%(QA, 1), GE, PPQ, THEN
    DD_%QLQ% = DD_%QLQ% + 1
  *ENDIF
*ENDDO
*ENDDO
*ENDIF

*IF, PQP, GT, 1, THEN
*DO, PPQ, 2, PPQ
  QQQ = 1
  QLO = arg1 + PPQ - 1
  *DIM, JUNK_%QLQ%, array, DD_%QLQ%, 4
  *DO, QA, 1, nodeNO
    *IF, JUNK_%arg1%(QA, 1), GE, PPQ, THEN
      JUNK_%QLQ%(QQQ, 1) = JUNK_%arg1%(QA, 1)
      JUNK_%QLQ%(QQQ, 2) = JUNK_%arg1%(QA, 2)
      JUNK_%QLQ%(QQQ, 3) = JUNK_%arg1%(QA, 3)
      JUNK_%QLQ%(QQQ, 4) = JUNK_%arg1%(QA, 4)
      QQQ = 1 + QQQ
    *ENDIF
  *ENDDO
  *ENDDO
  *ENDIF

!write array in to file
*CFOPEN, %JN%_mat_%arg1%, dat
*vwrite, JUNK_%arg1%(1, 1), JUNK_%arg1%(1, 2), JUNK_%arg1%(1, 3), JUNK_%arg1%(1, 4)
  (F8.6, 2X, E12.6, 2X, F6.0, 2X, E12.6, 2X)
*CFCLOSE
*ENDIF
*ENDIF
*ENDIF
APPENDIX C6

ANSYS INPUT FILE UNLOADCRACK.MAC, UNLOAD

MODEL, SPIKE OVERLOAD
AUTOTS,ON
NSUBST,1,100,1,OFF
*CFOPEN,%JN%_unload_%arg1%,dat

!*VWRITE
! (*NODE# Node r NodeAng S/Smax UY OStat Remote Stress*)
TimeVar=TimeVar+0.05
CurrLInc=UIBCC
StrsLvl=StrsMax/StrsMax
RStrs=StrsMax
*DO,JJ,1,arg1
  SelCTNodes,JJ
  NSEL,U,D,UY,0
  *GET,NSNODES,NODE,,COUNT
  *IF,NSNODES,GT,0,THEN
    NODNO=0
  *ENDIF
  *DO,JJJ,1,NSNODES
    NODNO=NDNEXT(NODNO)
    NODYSTRS=UY(NODNO)
    *IF,MTYPE,EQ,'SC',THEN
      CSYS,11
      NDANG=NY(NODNO)
      NXLOC=NX(NODNO)
      CSYS,0
    *ELSE
      NDANG=NZ(NODNO)
      NXLOC=NX(NODNO)
    *ENDIF
    NodeStat=0
    *VWRITE,NODNO,NXLOC,NDANG,StrsLvl,NODYSTRS,NODESTAT,RSTRS
      (F6.0,2X,E12.6,2X,E10.4,2X,F8.6,2X,F4.0,2X,E12.6)
  *ENDDO
  *ENDIF
*ENDDO

*DO,J,1,1/UIDCC
  TimeVar=TimeVar+CurrLInc*0.5
  RStrs=StrsMax-(StrsMax-StrsMin)*(TimeVar+0.5-arg1)/0.5
  *IF,TimeVar,GE,arg1,THEN
    RStrs=StrsMin
    TimeVar=arg1
    Time,TimeVar
    AppLoad,height,RStrs
    SOLVE
    SAVE
  *ENDIF
*ENDDO

*modified
*DO,JJ,1,arg1
  SelCTNodes,JJ
  NSEL,U,D,UY,0
  *GET,NSNODES,NODE,,COUNT
  *IF,NSNODES,GT,0,THEN
    NODNO=0
  *ENDIF
  *DO,JJJ,1,NSNODES
    NodeStat=0
    NODNO=NDNEXT(NODNO)
    NODYSTRS=UY(NODNO)
    *IF,MTYPE,EQ,'SC',THEN
      CSYS,11
NDANG=NY (NODNO)
NXLOC=NX (NODNO)
CSYS, 0
*ELSE
NDANG=NZ (NODNO)
NXLOC=NX (NODNO)
*ENDIF
*IF,NodYStrs*1e10,LT,0,THEN
CurrlInc=UIDCC
OPENSTAT=1
NODESTAT=1
D,NODNO,UY, 0
*ENDIF
*VWRITE,NODNO,NXLOC,NDANG,StrsLvl,NODYSTRS,NodeStat,RStrs
(F6.0, 2X,E12.6, 2X,E10.4, 2X,F8.6, 2X,F12.6, 2X,E12.6, 2X,E12.6)
*ENDDO
*endif
*enddo
!end of modification
*EXIT
*ENDIF
Time,TimeVar
StrsLvl=RStrs/StrsMax
AppLoad, height, RStrs
SOLVE
SAVE
ClearRST,BDrive,BDir,''
OPENSTAT=0
OpnRwCnt=0
*DO, JJ,1,arg1
  SelCTNodes,JJ
  NSEL,U,D,UY,0
  *GET, NSNODES,NODE,, COUNT
  *IF, NSNODES,GT,0,THEN
  NODNO=0
  *DO, JJ,J1,NSNODES
    NodeStat=0
    NODNO=NDNEXT (NODNO)
    NODYSTRS=UY (NODNO)
    *IF, MTYPE, EQ, 'SC', THEN
    CSYS, 11
    NDANG=NY (NODNO)
    NXLOC=NX (NODNO)
    CSYS, 0
  ELSE
    NDANG=NZ (NODNO)
    NXLOC=NX (NODNO)
  *ENDIF
  *IF,NodYStrs*1e10,LT,0,THEN
  CurrlInc=UIDCC
  OPENSTAT=1
  NODESTAT=1
  D,NODNO,UY,0
  *ENDIF
  *VWRITE,NODNO,NXLOC,NDANG,StrsLvl,NODYSTRS,NodeStat,RStrs
(F6.0,2X,E12.6,2X,E10.4,2X,F8.6,2X,F12.6,2X,F4.0,2X,E12.6)
  *ENDDO
*ELSEIF, NSNODES, EQ, 0, THEN
  OpnRwCnt=OpnRwCnt+1
*ENDIF
*ENDDO
NSEL,ALL
*IF,OPENSTAT,EQ,1,THEN
  Time,Timevar+CurrLIinc*0.01
  SOLVE
  SAVE
  ClearRST,BDrive,BDir,''
*ENDIF
*IF,OpnRwCnt,EQ,arg1,THEN
  CurrLIinc=UIACC
*ENDIF
*ENDDO
*CFCLOSE
APPENDIX C7

ANSYS INPUT FILE STRTCYC.MAC, CONTROL OF CYCLIC LOADING, CRACK SHAPE EVOLUTION
TimeVar=0.45
FirstLoad
ClearRST,BDrive,BDir,MaxDir
PE=0

*DO,I,1,NLC
  *IF,PE,NE,0,THEN
    I=PE+1
  *ENDIF
  *IF,I,EQ,1,THEN
    UnloadCrack,I
    ClearRST,BDrive,BDir,MinDir
    *DO,O,1,I
      JK_%O%=-1
    *ENDDO
    LoadCrack,I
    ClearRST,BDrive,BDir,MaxDir
    SU=I
    SU_%I%=I
    PE=I
  *else
    !sort the array in ascending order
    *MOPER,ORDE(1),JUNK_%SU%(1,1),sort,JUNK_%SU%(1,1)
    VES=JUNK_%SU%(NODENO,1)
    FUNI=SU+VES
    VES_%FUNI%=VES
    !rearranging the array
    *MOPER,ORDE(1),JUNK_%SU%(1,1),sort,ORDE(1,1)
    JUMP=1
    JUMP_%I%=1
    *DO,TEE,1,VES
      *IF,TEE,GT,1,THEN
        I=I+1
        JUMP_%I%=JUMP+1
        JUMP=JUMP+1
      *IF,TEE,EQ,VES,THEN
        JUMP_%I%=0
      *ENDIF
    *ENDIF
  *ENDIF
  *IF,TEE,EQ,VES,THEN
    JUMP_%I%=0
  *ENDIF
  UnloadCrack,I
  ClearRST,BDrive,BDir,MinDir
  *DO,O,1,I
    JK_%O%=-1
  *ENDDO
  LoadCrack,I
  ClearRST,BDrive,BDir,MaxDir
*ENDDO
TEE=VES
NLC=NLC+TEE-1
SU=I
JUMP=1
SU_%I%=I
PE=I
*ENDIF
SAVE
*IF, PE, EQ, NLC, EXIT
*ENDDO
APPENDIX C8

ANSYS INPUT FILE SELECTNODES.MAC, SELECT CRACK FRONT, CRACK SHAPE EVOLUTION
! SelCTNodes.mac
! Macro to Select the Crack tip nodes for load cycle "N" for surface crack
without spike overload
!
! Executed as follows:
! SelCTNodes,N
!
! written by Kiran Solanki
! Set Coordinate System to Elliptical

*IF,arg1,eq,1,THEN
! for first cycle
LOCAL,11,1,0,0,0,0,90,0,a/c
NSEL,S,LOC,X,c-0.3*da,c+0.3*da
NSEL,R,LOC,2,0
*GET,FNODNO,NODE,,NUM,MIN
*GET,LNODNO,NODE,,NUM,MAX
DELTANN=NDNEXT(FNODNO)-FNODNO
*GET,NODCNT,NODE,,COUNT
NODENO=((LNODNO-FNODNO)/DELTANN)+1
NSEL,S,NODE,,FNODNO+(arg1-1),LNODNO+(arg1-1),DELTANN
CSYS,0
*ENDIF

*IF,JUMP_%arg1%,GE,1,THEN
*IF,arg1,NE,1,THEN
! selection of node
LOCAL,11,1,0,0,0,0,90,0,a/c
NSEL,S,LOC,X,c-0.3*da,c+0.3*da
NSEL,R,LOC,2,0
NSEL,S,NODE,,JUNK_%arg1%(1,3)+JUMP_%arg1%,JUNK_%arg1%(1,3)+JUMP_%arg1%,1
*DO,QP,2,DD_%arg1%
  nsel,a,NODE,,JUNK_%arg1%(QP,3)+JUMP_%arg1%,JUNK_%arg1%(QP,3)+JUMP_%arg1%,1
*ENDDO
CSYS,0
*ENDIF
*ENDIF

*IF,JUMP_%arg1%,EQ,0,THEN
*IF,arg1,NE,1,THEN
! selection of node
LOCAL,11,1,0,0,0,0,90,0,a/c
NSEL,S,LOC,X,c-0.3*da,c+0.3*da
NSEL,R,LOC,2,0
KII=VES_%arg1%
nsel,s,,,JUNK_%arg1-KII%(1,3)+JUNK_%arg1-KII%(1,1),JUNK_%arg1-
KII%(1,3)+JUNK_%arg1-KII%(1,1),1
*DO,k,2,NODENO
  nsel,a,,,JUNK_%arg1-KII%(K,3)+JUNK_%arg1-KII%(K,1),JUNK_%arg1-
  KII%(K,1)+JUNK_%arg1-KII%(K,3),1
CSYS, 0
*ENDDO
KIR_%argl%=arg1
KIR=arg1
*ENDIF
*ENDIF
APPENDIX C9

ANSYS INPUT FILE \textit{LOADCRACK.MAC}, LOAD MODEL,

CRACK SHAPE EVOLUTION
AUTOTS,ON
NSUBST,1,100,1,OFF
*CFOpen,%JN%\_load\_arg1%,dat
!*VWRITE
! (*NODE# Node r NodeAng S/Smax SY OStat Remote Stress*)

SelCTNodes, arg1
NSEL,R,D,UY,0
*GET, NSNODES, NODE,, COUNT
*DIM, NEXT\_arg1\%, ARRAY, NSNODES, 4
NSEL, ALL
CurrLInc=LIBCO
StrsLvl=StrsMin/StrsMax
RStrs=StrsMin

*DO, JJ, 1, arg1
SelCTNodes, JJ
NSEL, R, D, UY, 0
*GET, NSNODES, NODE,, COUNT
*DIM, PRE\_JJ\%, ARRAY, NSNODES, 3
! storing the no. of node for next cycle
*IF, JJ, EQ, arg1, THEN
*GET, FNO, NODE,, NUM, MIN
*GET, LNO, NODE,, NUM, MAX
K\_JJ%=NSNODES
*ENDIF
*IF, NSNODES, GT, 0, THEN
NODNO=0
*DO, JJJ, 1, NSNODES
  NODNO=NDNEXT (NODNO)
  *GET, NODYSTRS, NODE, NODNO, S, Y
  *IF, MTYPE, EQ, 'SC', THEN
    CSYS, 11
    NDANG=NY (NODNO)
    NXLOC=NX (NODNO)
    CSYS, 0
  ELSE
    NDANG=NZ (NODNO)
    NXLOC=NX (NODNO)
  *ENDIF
NodeStat=0
*IF, NodYStrs*1e10, GT, 0, THEN
  CurrLInc=LIDCO
  OPENSTAT=1
  NODESTAT=1
  DDELE, NODNO, UY
*ENDIF
*VWRITE, NODNO, NXLOC, NDANG, StrsLvl, NODYSTRS, NODESTAT, RSTRS
(F6.0, 2X, E12.6, 2X, E10.4, 2X, F8.6, 2X, E12.6, 2X, F4.0, 2X, E12.6)

*IF, NodeStat, EQ, 1, THEN
  *IF, JJ, EQ, arg1, THEN
    NEXT\_JJ\% (JK\_JJ\%, 4)=NDANG
    NEXT\_JJ\% (JK\_JJ\%, 3)=NODNO
    NEXT\_JJ\% (JK\_JJ\%, 2)=StrsMin/StrsMax
    JK\_JJ%=JK\_JJ%+1
  *ENDIF
*ELSE
  *IF, JJ, EQ, arg1, THEN
    PRE\_JJ\% (JJJ, 3)=NODNO
PRE_%JJ%(J JJ, 2) = StrsLvl
PRE_%JJ%(J JJ, 1) = NODYSTRS
*ENDIF
*ENDIF
*ENDDO
*ENDIF
*ENDDO

*DO, J, 1, 1/LIDCO
  TimeVar = TimeVar + CurrLInc * 0.45
  RStrs = (StrsMax - StrsMin) * (TimeVar - arg1) / 0.45 + StrsMin
  *IF, TimeVar, GE, arg1 + 0.45, Then
    RStrs = StrsMax
    TimeVar = arg1 + 0.45
  *ENDIF
  Time, TimeVar
  AppLoad, height, RStrs
  SOLVE
  SAVE
  *EXIT
  *ENDIF
  Time, TimeVar
  StrsLvl = RStrs / StrsMax
  AppLoad, height, RStrs
  SOLVE
  SAVE
  ClearRST, BDrive, BDir, ''
  OPENSTAT = 0
  OpnRwCnt = 0
  *DO, JJ, 1, arg1
    SelICTNodes, JJ
    NSel, R, D, UY, 0
    *GET, NSNODES, NODE, / COUNT
    *IF, NSNODES, GT, 0, THEN
      NODNO = 0
      *DO, JJJ, 1, NSNODES
        NodeStat = 0
        NODNO = NDNEXT (NODNO)
        *GET, NodYStrs, NODE, NODNO, S, Y
        *IF, MTYPE, EQ, 'SC', THEN
          CSYS, 11
          NDANG = NY (NODNO)
          NXLOC = NX (NODNO)
          CSYS, 0
        *ELSE
          NDANG = NZ (NODNO)
          NXLOC = NX (NODNO)
        *ENDIF
        *IF, NodYStrs * 1e10, GT, 0, THEN
          CurrLInc = LIDCO
          OPENSTAT = 1
          NODESTAT = 1
          DDELE, NODNO, UY
        *ENDIF
        *WRITE, NODNO, NXLOC, NDANG, StrsLvl, NODYSTRS, NodeStat, RStrs
          (F6.0, 2X, E12.6, 2X, E10.4, 2X, E8.6, 2X, F12.6, 2X, F4.0, 2X, E12.6)
        *IF, NodeStat, EQ, 1, THEN
          *IF, JJ, EQ, arg1, THEN
            NEXT_%JJ%(J K_%JJ%, 4) = NDANG
            NEXT_%JJ%(J K_%JJ%, 3) = NODNO
          *DO, L, 1, K_%JJ%
*IF, NODNO, EQ, PRE_%JJ%(L, 3), THEN
  JUNK = (NODYSTRS - PRE_%JJ%(L, 1))
  Q1 = PRE_%JJ%(L, 2)
  NEXT_%JJ%(JK_%JJ%, 2) = ((NODYSTRS * Q1) - (PRE_%JJ%(L, 1) * StrsLvl)) / JUNK
*ENDIF
*ENDDO
JK_%JJ% = JK_%JJ% + 1
*ENDIF
*ELSE
  *IF, JJ, EQ, arg1, THEN
  *DO, L, 1, K_%JJ%
    *IF, PRE_%JJ%(L, 3), EQ, NODNO, THEN
      PRE_%JJ%(L, 3) = NODNO
      PRE_%JJ%(L, 2) = StrsLvl
      PRE_%JJ%(L, 1) = NODYSTRS
    *ENDIF
  *ENDDO
*ENDIF
*ENDIF
*ENDIF
*ENDIF
*ELSEIF, NSNODES, EQ, 0, THEN
  OpnRwCnt = OpnRwCnt + 1
*ENDIF
*ENDDO
NSEL, ALL
*IF, OPENSTAT, EQ, 1, THEN
  Time, Timevar + CurrLInc * 0.01
  SOLVE
  SAVE
  ClearRST, BDrive, BDir, ''
*ENDIF
*IF, OpnRwCnt, EQ, arg1, THEN
  CurrLInc = LIACO
*ENDIF
*ENDIF
*ENDIF
*CFCLOSE

*IF, arg1, EQ, 1, THEN
  ! Newman Raju Equation to find SIF
  *DIM, TEMPI, array, NODENO
  *DIM, FA, array, NODENO
  *DIM, G, array, NODENO
  *DIM, F, array, NODENO
  *DIM, KMAX, array, NODENO
  *DIM, KOPEN, array, NODENO
  *DIM, CONS, array, NODENO
  *DO, JJJ, 1, NODENO
    TEMPI(JJJ) = NEXT_%arg1%(JJJ, 4) * (PI / 180)
    AA = NZ(FNO)
    CC = NX(LNO)
    CONS(JJJ) = (8.1932965e-9) * TEMPI(JJJ) + 1.503e-8
    FA(JJJ) = (((AA / CC) ** 2) * (cos(TEMPI(JJJ)) ** 2) + (sin(TEMPI(JJJ)) ** 2)) ** 0.25
    G(JJJ) = 1 - ((0.1 + 0.35 * ((AA / T) ** 2)) * ((1 - sin(TEMPI(JJJ)) ** 2))
    F(JJJ) = (M1 + (M2 * ((AA / T) ** 2)) + (M3 * ((AA / T) ** 4))) * FA(JJJ) * G(JJJ) * FW
    KMAXx(JJJ) = strsMax * sqrt(PI * (AA / QW)) * F(JJJ)
    KOPEN(JJJ) = NEXT_%arg1%(JJJ, 2) * strsMax * sqrt(PI * (AA / QW)) * F(JJJ)
    NEXT_%arg1%(JJJ, 1) = (KMAX(JJJ) - KOPEN(JJJ)) ** 2.881 * CONS(JJJ)
*ENDDO
!sort the array in ascending order
*MOPER,ORDE(1),NEXT_%arg1%(1,1),sort,NEXT_%arg1%(1,1)

!Delta k effective
*DO,JJJ,2,NODENO
    NEXT_%arg1%(JJJ,1)=NEXT_%arg1%(JJJ,1)/NEXT_%arg1%(1,1)
*ENDDO

!min. delta K effective
NEXT_%arg1%(1,1)=1
PQP=NINT(NEXT_%arg1%(1,NODENO))

!rearranging the array
*MOPER,ORDE(1),NEXT_%arg1%(1,1),sort,ORDE(1,1)

!Exchange of the matrix values to store permanently
*DIM,JUNK_%arg1%,array,NODENO,4
*DO,JJJ,1,NODENO
    JUNK_%arg1%(JJJ,1)=NINT(NEXT_%arg1%(JJJ,1))
    JUNK_%arg1%(JJJ,2)=NEXT_%arg1%(JJJ,2)
    JUNK_%arg1%(JJJ,3)=NEXT_%arg1%(JJJ,3)
    JUNK_%arg1%(JJJ,4)=NEXT_%arg1%(JJJ,4)
*ENDDO

!JUNK stuff
*IF,PQP,GT,1,THEN
    *DO,PPQ,2,PQP
        DD_%PPQ%=0
    *DO,QA,1,NODENO
        *IF,JUNK_%arg1%(QA,1),GE,PPQ,THEN
            DD_%PPQ%=DD_%PPQ%+1
        *ENDIF
    *ENDIF
*ENDDO
*ENDIF

!JUNK stuff
*IF,PQP,GT,1,THEN
    *DO,PPQ,2,PQP
        QQQ=1
        *DIM,JUNK_%PPQ%,array,DD_%PPQ%,4
    *DO,QA,1,NODENO
        *IF,JUNK_%arg1%(QA,1),GE,PPQ,THEN
            JUNK_%ppq%(QQQ,1)=JUNK_%arg1%(QA,1)
            JUNK_%ppq%(QQQ,2)=JUNK_%arg1%(QA,2)
            JUNK_%ppq%(QQQ,3)=JUNK_%arg1%(QA,3)
            JUNK_%ppq%(QQQ,4)=JUNK_%arg1%(QA,4)
            QQQ=1+QQQ
        *ENDIF
    *ENDIF
*ENDDO
*ENDIF

!write array in to file
*CFOPEN,%JN%_mat_%arg1%,dat
*vwrite,JUNK_%arg1%(1,1),JUNK_%arg1%(1,2),JUNK_%arg1%(1,3),JUNK_%arg1%(1,4)
    (F8.6,2X,E12.6,2X,F6.0,2X,E12.6,2X)
*CFCLOSE

*ELSE

!swap the required matrix
*IF,JUMP,EQ,YES,THEN
  *DIM,JUNK_%arg1%,array,NODENO,4
  *DO,KO,1,NODENO
      JUNK_%arg1%(KO,1)=NEXT_%arg1%(KO,1)
      JUNK_%arg1%(KO,2)=NEXT_%arg1%(KO,2)
      JUNK_%arg1%(KO,3)=NEXT_%arg1%(KO,3)
JUNK_%arg1%(KO,4)=NEXT_%arg1%(KO,4)
*ENDDO
*DO,J,1,NODENO
  *IF,JUNK_%arg1%(J,3),EQ,0,THEN
    JUNK_%arg1%(J,3)=JUNK_%arg1%-1%(J,3)+JUNK_%arg1%-1%(J,1)
    JUNK_%arg1%(J,2)=StrsMin/StrsMax
    DUM=JUNK_%arg1%-1%(J,4)
    JUNK_%arg1%(J,4)=JUNK_%arg1%-1%(J,4)
  *ENDIF
*ENDDO

!Newman Raju Equation to find SIF
*DO,JJJ,1,NODENO
  AA=NZ(FNO)
  CC=NX(LNO)
  TEMPI(JJJ)=JUNK_%arg1%(JJJ,4)*(PI/180)
  CONS(JJJJ)=(8.1932965e-9)*TEMPI(JJJJ)+1.503e-8
  G(JJJ)=1.0-(0.10.35*(AA/T)**2)*((1-sin(TEMPI(JJJJ))**2))**0.25
  F(JJJ)=(M1*(AA/CC)**2)+(M2*(AA/T)**2))+(M3*(AA/T)**4)*)*FA(JJJJ)*G(JJJJ)*FW
  KMAX(JJJ)=strsMax*sqrt(PI*(AA/QW))*/F(JJJJ)
  KOPEN(JJJ)=JUNK_%arg1%(JJJ,2)*strsMax*sqrt(PI*(AA/QW))*/F(JJJJ)
  JUNK_%arg1%(JJJ,1)=(KMAX(JJJ)-KOPEN(JJJ))**2.881)*CONS(JJJJ)
*ENDDO

!sort the array in ascending order
*DIM,ORDE,,NODENO
*MOPER,ORDE(1),JUNK_%arg1%(1,1),sort,JUNK_%arg1%(1,1)
!Delta K effective
*DO,JJJ,2,NODENO
  JUNK_%arg1%(JJJ,1)=NINT(JUNK_%arg1%(JJJ,1)/JUNK_%arg1%(1,1))
*ENDDO

!min. delta K effective factor
JUNK_%arg1%(1,1)=1
PQP=JUNK_%arg1%(NODENO,1)
!rearranging the array
*MOPER,ORDE(1),JUNK_%arg1%(1,1),sort,ORDE(1,1)
!JUNK stuff
*IF,PQP,GT,1,THEN
  *DO,PPQ,2,PQP
    QLQ=arg1+PPQ-1
    DD_%QLQ%=0
    *DO,QA,1,NODENO
      *IF,JUNK_%arg1%(QA,1),QE,PPQ,THEN
        DD_%QLQ%=DD_%QLQ%+1
      *ENDIF
    *ENDDO
  *ENDDO
*ENDDO
*ENDIF
!JUNK stuff
*IF,PQP,GT,1,THEN
  *DO,PPQ,2,PQP
    QLQ=arg1+PPQ-1
    DD_%QLQ%=0
    *DO,QA,1,NODENO
      *IF,JUNK_%arg1%(QA,1),QE,PPQ,THEN
        DD_%QLQ%+DD_%QLQ%+1
      *ENDIF
    *ENDDO
*ENDDO
*ENDIF
!write array in to file
*CFOPEN,%JN%_mat_%arg1%,dat
*vwrite,JUNK_%arg1%(1,1),JUNK_%arg1%(1,2),JUNK_%arg1%(1,3),JUNK_%arg1%(1,4)
   (F8.6,2X,E12.6,2X,F6.0,2X,E12.6,2X)
*CFCLOSE
*ENDIF
*ENDIF
*ENDIF
APPENDIX C10

ANSYS INPUT FILE *UNLOADCRACK.MAC*, UNLOAD MODEL,

CRACK SHAPE EVOLUTION
AUTOTS,ON
NSUBST,1,100,1,OFF
*COPEN, &JN%_unload_%arg1%,dat
!*VWRITE
! ("NODE# Node r NodeAng S/Smax UY OStat Remote Stress")
TimeVar=TimeVar+0.05
CurrLInc=UIDCC
StrsLvl=StrsMax/StrsMax
RStrs=StrsMax
*DO,JJ,1, arg1
SetCTNodes,JJ
NSEL,U,D,UY,0
*GET,NSNODES,NODE,,COUNT
*IF,NSNODES,GT,0,THEN
  NODNO=0
  *DO,JJJ,1,NSNODES
    NODNO=NDNEXT(NODNO)
    NODYSTRS=UY(NODNO)
    *IF,MTYPE,EQ,'SC',THEN
      CSYS,11
      NDANG=NY(NODNO)
      NXLOC=NX(NODNO)
      CSYS,0
    *ELSE
      NDANG=NZ(NODNO)
      NXLOC=NX(NODNO)
    *ENDIF
  NodeStat=0
  *VWRITE,NODNO,NXLOC,NDANG,StrsLvl,NODYSTRS,NODESTAT,RSTRS
    (F6.0,2X,E12.6,2X,E10.4,2X,F8.6,2X,F12.6,2X,F4.0,2X,E12.6)
  *ENDDO
*ENDIF
*ENDDO
*DO,J,1,1/UIDCC
  TimeVar=TimeVar+CurrLInc*0.5
  RStrs=StrsMax-(StrsMax-StrsMin)*(TimeVar+0.5-arg1)/0.5
  *IF,TimeVar,GE,arg1,Then
    RStrs=StrsMin
    Time,TimeVar
    AppLoad,height,RStrs
    SOLVE
    SAVE
  !modified
  *DO,JJ,1, arg1
  SetCTNodes,JJ
  NSEL,U,D,UY,0
  *GET,NSNODES,NODE,,COUNT
  *IF,NSNODES,GT,0,THEN
    NODNO=0
    *DO,JJJ,1,NSNODES
      NodeStat=0
      NODNO=NDNEXT(NODNO)
      NODYSTRS=UY(NODNO)
      *IF,MTYPE,EQ,'SC',THEN
        CSYS,11
        NDANG=NY(NODNO)
        NXLOC=NX(NODNO)
        CSYS,0
  }
*ELSE
  NDANG=NZ (NODNO)
  NXLOC=NX (NODNO)
*ENDIF
*IF,NodYStrs*1e10,LT,0,THEN
  CurrLInc=UIDCC
  OPENSTAT=1
  NODESTAT=1
  D,NODNO,UY,0
*ENDIF
*VWRITE,NODNO,NXLOC,NDANG,StrsLvl,NODYSTRS,NodeStat,RStrs
  (F6.0,2X,E12.6,2X,E10.4,2X,F8.6,2X,E12.6,2X,F4.0,2X,E12.6)
*ENDDO
*endif
*enddo
!end of modification
*EXIT
*ENDIF
Time,TimeVar
StrsLvl=RStrs/StrsMax
AppLoad,height,RStrs
SOLVE
SAVE
ClearRST,BDrive,BDir,'
OPENSTAT=0
OpnRwCnt=0
*DO,JJ,1,arg1
  SelCTNodes,JJ
  NSEL,U,D,UY,0
  *GET,NSNODES,NODE,,COUNT
  *IF,NSNODES,GT,0,THEN
    NODNO=0
    *DO,JJJ,1,NSNODES
      NodeStat=0
      NODNO=NDNEXT (NODNO)
      NODYSTRS=UY (NODNO)
      *IF,MTYPE,EQ,'SC',THEN
        CSYS,11
        NDANG=NY (NODNO)
        NXLOC=NX (NODNO)
        CSYS,0
      ELSE
        NDANG=NZ (NODNO)
        NXLOC=NX (NODNO)
      *ENDIF
      *IF,NodYStrs*1e10,LT,0,THEN
        CurrLInc=UIDCC
        OPENSTAT=1
        NODESTAT=1
        D,NODNO,UY,0
      *ENDIF
      *VWRITE,NODNO,NXLOC,NDANG,StrsLvl,NODYSTRS,NodeStat,RStrs
        (F6.0,2X,E12.6,2X,E10.4,2X,F8.6,2X,E12.6,2X,F4.0,2X,E12.6)
    *ENDDO
  ELSEIF,NSNODES,EQ,0,THEN
    OpnRwCnt=OpnRwCnt+1
  *ENDIF
*ENDDO
NSEL,ALL
*IF,OPENSTAT,EQ,1,THEN
Time, Timevar+CurrLInc*0.01
SOLVE
SAVE
ClearRST, BDrive, BDir, ''
*ENDIF
*IF, OpnRwCnt, EQ, arg1, THEN
  CurrLInc = UIACC
*ENDIF
*ENDDO
*CFCLOSE
APPENDIX C11

ANSYS INPUT FILE SURFACECRACK.MAC, SAMPLE INPUT

FILE, CRACK SHAPE EVOLUTION
! This is the input file for pca1548
! This runs the script "appbcs.mac" to import the Solid
! Elements and Nodes from the files "jobname.crd" & "jobname(elm",
! and applies necessary boundary conditions for Crack Growth
! It then calls "strtcyc.mac" to run growth analysis.
!
! This script has been modified to work with ANSYS 5.6 and ANSYS 5.7
! with the addition of the RESCONTROL Command (not valid for ANSYS 5.5 and below)
!
! Note all lengths are in mm, and pressures in MPa

!CONFIG,NPROC,1

! Loading information:
StrsMax=22e3   ! Maximum Applied Stress
StrsMin=2.2e3  ! Minimum Applied Stress
NLC=6          ! Total Number of Loading Cycles to execute

! Geometry Information:

MTYPE='SC'
t=0.5         ! Thickness of plate
w=0.5         ! Plate Half-Width
height=1.1875 ! Model Height
c=0.0501      ! Initial Crack half-length
a=0.05        ! Initial Depth of Surface Crack
da=0.00012500/2 ! Crack Growth Increment

! Material Properties:
E=10.6E6     ! Young's Modulus
YS=75e3      ! Yield Stress

! Crack Growth Options:
NCGECut=5    ! Number of bisections to matrix stiffness before death
CGERF=2      ! Crack Growth Element Stiffness Reduction Factor

*get,JN,ACTIVE,,JOBNAME
/TITLE, Plasticity Induced Closure of model %JN%

AppBCs       ! Import Solid model and apply BC's

BDrive='e:'   ! Drive for file backups
BDir='\backup' ! Directory for file backups
MaxDir='\maxload' ! Directory for backup at Max Load
MinDir='\minload' ! Directory for backup at Min Load

! Solution Information:

/SOLU       ! Enter Solution Processor

LIBCO=0.05   ! Loading Increment before crack opening
LIDCO=0.025  ! Loading Increment during crack opening
LIACO=0.10 ! Loading Increment after crack opening
UIBCC=0.05 ! Un-load Increment before crack closing
UIDCC=0.025 ! Un-load Increment during crack closing
UIACC=0.10 ! Un-load Increment after crack closing
SOLCONTROL, ON
NSUB,1
NEQIT, 8 ! Number of Equilibrium Iterations before bisection
NROPT,FULL, ,ON ! Full Newton Rapson Option, Adaptive Descent ON
EQSLV,PCG,1.0e-8 ! Use the Pre-Conditioned Conjugate Solver (In Core,
default tolerance)
RESCONTROL,DEFINE,NONE, NONE, 0 ! Set Resume Controls to act like ANSYS 5.5.3
and below
! (Single Frame Restart)
raju
APPENDIX C12

ANSYS INPUT FILE SURFACECRACKSPIKE.MAC

SAMPLE INPUT FILE, SPIKE OVERLOAD
/BATCH
! This is the input file for pca1548
! This runs the script "appbcS.mac" to import the Solid
! Elements and Nodes from the files "jobname.crd" & "jobname.elm",
! and applies necessary boundary conditions for Crack Growth
! It then calls "strtcyc.mac" to run growth analysis.
!
! This script has been modified to work w/ ANSYS 5.6 and ANSYS 5.7
! with the addition of the RESCONTROL Command (not valid for ANSYS 5.5 and
! below)

/CONFIG,NPROC,1

! Loading information:

StrsMax=22e3 ! Maximum Applied Stress
StrsMin=2.2e3 ! Minimum Applied Stress
spmax=44e3 ! spike overload
smax=22e3 ! Maximum applied stress
NLC=1 ! Total Number of Loading Cycles to execute

! Geometry Information:

MTYPE='SC'
t=0.5 ! Thickness of plate
w=0.5 ! Plate Half-Width
height=1.1875 ! Model Height
c=0.0501 ! Initial Crack half-length
a=0.05 ! Initial Depth of Surface Crack
da=0.0002500/2 ! Crack Growth Increment

! Material Properties:

E=10.6E6 ! Young's Modulus
YS=75e3 ! Yield Stress

! Crack Growth Options:

NCGECut=5 ! Number of bisections to matrix stiffness before death
CGERF=2 ! Crack Growth Element Stiffness Reduction Factor

*get,JN,ACTIVE,,JOBNAME
/TITLE, Plasticity Induced Closure of model %JN%

AppBCs ! Import Solid model and apply BC's

BDrive='e:' ! Drive for file backups
BDir='\backup' ! Directory for file backups
MaxDir='\maxload' ! Directory for backup at Max Load
MinDir='\minload' ! Directory for Backup at Min Load

! Solution Information:

/SOLU ! Enter Solution Processor

LIBCO=0.05 ! Loading Increment before crack opening
LIDCO=0.025 ! Loading Increment during crack opening
LIACO=0.10 ! Loading Increment after crack opening
UIBCC=0.05 ! Un-load Increment before crack closing
UIDCC=0.025 ! Un-load Increment during crack closing
UIACC=0.10 ! Un-load Increment after crack closing
SOLCONTROL,ON
NSUB,1
NEQIT,8 ! Number of Equilibrium Iterations before bisection
NROPT,FULL,,ON ! Full Newton Rapson Option, Adaptive Descent ON
EQSLV,PCG,1.0e-8 ! Use the Pre-Conditioned Conjugate Solver (In Core, default tolerance)
RESCONTROL,DEFINE,NONE,NONE,0 ! Set Resume Controls to act like ANSYS 5.5.3 and below
                                   ! (Single Frame Restart)
raju