5-1-2011

Analysis of fatigue crack growth and residual stress

Shakhrukh Ismonov

Follow this and additional works at: https://scholarsjunction.msstate.edu/td

Recommended Citation
https://scholarsjunction.msstate.edu/td/640

This Dissertation is brought to you for free and open access by the Theses and Dissertations at Scholars Junction. It has been accepted for inclusion in Theses and Dissertations by an authorized administrator of Scholars Junction. For more information, please contact scholcomm@msstate.libanswers.com.
ANALYSIS OF FATIGUE CRACK GROWTH AND RESIDUAL STRESS

By

Shakhrukh Ismonov

A Dissertation
Submitted to the Faculty of
Mississippi State University
in Partial Fulfillment of the Requirements
for the Degree of Doctor of Philosophy
in Mechanical Engineering
in the Department of Mechanical Engineering

Mississippi State, Mississippi

April 2011
ANALYSIS OF FATIGUE CRACK GROWTH AND RESIDUAL STRESS

By

Shakhrukh Ismonov

Approved:

Steve R. Daniewicz
Professor and Interim Head
Mechanical Engineering
(Major Professor)

Professor of Aerospace Engineering
(Committee Member)

Mohamad S. Qatu
Professor of Mechanical Engineering
(Committee Member)

Oliver J. Myers
Professor of Mechanical Engineering
(Committee Member)

David L. Marcum
Professor of Mechanical Engineering
(Graduate Coordinator)

Sarah A. Rajala
Dean of Bagley College of Engineering
The first part of this dissertation employs a three-dimensional elastic-plastic finite element model of straight-through crack growth to correlate four well-known methodologies characterizing fatigue crack closure. The compliance offset and the adjusted compliance ratio (ACR) are experimental methods, whereas the node displacement and the contact stress methods are numerical approaches. Evolutions of crack closure from all four methodologies are compared for a numerical model of a single edge-cracked tension specimen subjected to different levels of constant amplitude cyclic loading.

In the second part, a detailed two dimensional stress analysis is conducted for a single pin-joint under plane stress conditions. This study investigates the influence of material nonlinearity, friction, and pre-existing residual stresses from cold-working process on the local radial and hoop stress levels around the pin-loaded hole.

Next, the beneficial influence of cold working process is quantified by computing the Mode I stress intensity factors $K_I$ for a single radial crack emanating from a side of a loaded hole. Two different loading configurations are considered: (a) an open hole in
tension, (b) a pin-loaded hole. The stress intensity factors are computed using the $J$ integral solutions and the weight functions specific to the crack configuration. The reductions in $K_I$ values due to different levels of cold-working process are presented for a range of crack lengths.

The final part of the research involves a numerical investigation of an on-line crack compliance technique that is used for experimental measurements of residual stress fields along the crack growth path. A finite rectangular sheet is considered with a single crack emanating from a side of a central hole. The residual stress field is introduced around the hole by cold-working simulation. As part of validation, the normalized residual stress intensity factors computed using the on-line crack compliance technique are compared with those from the $J$-integral approach for the case of elastic crack growth. The influence of crack tip plasticity on the performance of the on-line crack compliance technique is studied by comparing the solutions of the elastic and elastic-plastic crack growth models.
ACKNOWLEDGEMENTS

I owe my gratitude to several individuals who helped me make this dissertation possible.

First, I would like to sincerely thank my advisor Dr. Steve Daniewicz for his invaluable guidance, understanding and patience throughout my graduate studies at Mississippi State University. He gave me an enormous opportunity to explore various areas of computational modeling, and he allowed me to teach a number of courses to gain teaching experience.

My next special gratitude goes to Dr. James Newman Jr. for letting me work for his FAA project, which gave me a great deal of research experience and financially supported the major part of my studies.

I am also grateful to Dr. Mohammad Qatu and Dr. Oliver Myers for serving on my Ph.D. committee, showing their enthusiasm in my work and offering their valuable suggestions in order to make this dissertation better.

Last but not least, I would like to thank my wife and parents for their never-ending support and encouragement, and my lovely kids Ibrahim and Sarah for constantly being a source of joy and energy.
# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACKNOWLEDGEMENTS</td>
<td>ii</td>
</tr>
<tr>
<td>LIST OF TABLES</td>
<td>v</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td>vi</td>
</tr>
<tr>
<td>CHAPTER</td>
<td></td>
</tr>
<tr>
<td>I.  INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>1.1 Objective of this Research</td>
<td>1</td>
</tr>
<tr>
<td>1.2 Organization of Dissertation</td>
<td>2</td>
</tr>
<tr>
<td>II. SIMULATION AND COMPARISON OF CRACK CLOSURE ASSESSMENT METHODOLOGIES</td>
<td>4</td>
</tr>
<tr>
<td>2.1 Introduction</td>
<td>4</td>
</tr>
<tr>
<td>2.2 Finite Element Model</td>
<td>8</td>
</tr>
<tr>
<td>2.3 Material Model</td>
<td>10</td>
</tr>
<tr>
<td>2.4 Workflow</td>
<td>10</td>
</tr>
<tr>
<td>2.5 Crack Growth Modeling</td>
<td>11</td>
</tr>
<tr>
<td>2.6 Crack Closure Characterizing Techniques</td>
<td>12</td>
</tr>
<tr>
<td>2.6.1 Node Displacement Method</td>
<td>13</td>
</tr>
<tr>
<td>2.6.2 Contact Stress Method</td>
<td>13</td>
</tr>
<tr>
<td>2.6.3 Compliance Offset Method</td>
<td>16</td>
</tr>
<tr>
<td>2.6.4 ACR Method</td>
<td>18</td>
</tr>
<tr>
<td>2.7 Model Validation</td>
<td>19</td>
</tr>
<tr>
<td>2.8 Results and Discussion</td>
<td>20</td>
</tr>
<tr>
<td>2.9 Conclusions</td>
<td>25</td>
</tr>
<tr>
<td>2.10 References</td>
<td>27</td>
</tr>
<tr>
<td>III. STRESS ANALYSIS FOR COLD WORKED PIN-LOADED HOLES</td>
<td>29</td>
</tr>
<tr>
<td>3.1 Introduction</td>
<td>29</td>
</tr>
<tr>
<td>3.2 Configuration and Material Models</td>
<td>32</td>
</tr>
<tr>
<td>3.3 Finite Element Model and Boundary Conditions</td>
<td>34</td>
</tr>
<tr>
<td>3.4 Model Validation</td>
<td>36</td>
</tr>
</tbody>
</table>
LIST OF TABLES

<table>
<thead>
<tr>
<th>TABLE</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>20</td>
</tr>
</tbody>
</table>

Element counts in forward and reverse zone at peak load
# LIST OF FIGURES

<table>
<thead>
<tr>
<th>FIGURE</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>A load cycle with crack closure .................................................................5</td>
</tr>
<tr>
<td>2.2</td>
<td>Single edge-cracked tension specimen for fatigue crack growth, the sheet thickness is scaled for clarity. .........................................................8</td>
</tr>
<tr>
<td>2.3</td>
<td>Typical finite element mesh .........................................................................9</td>
</tr>
<tr>
<td>2.4</td>
<td>Workflow of the analysis .............................................................................10</td>
</tr>
<tr>
<td>2.5</td>
<td>Typical load cycle .....................................................................................12</td>
</tr>
<tr>
<td>2.6</td>
<td>Node displacement method .........................................................................13</td>
</tr>
<tr>
<td>2.7</td>
<td>Plastic wake zone .......................................................................................14</td>
</tr>
<tr>
<td>2.8</td>
<td>A single edge crack in a finite width plate .................................................14</td>
</tr>
<tr>
<td>2.9</td>
<td>Compliance offset method ..........................................................................17</td>
</tr>
<tr>
<td>2.10</td>
<td>A typical compliance offset curve ..............................................................18</td>
</tr>
<tr>
<td>2.11</td>
<td>Load–displacement curve for a cracked body ............................................19</td>
</tr>
<tr>
<td>2.12</td>
<td>Evolution of $U$ parameter at each though-thickness location: present study compared with Carlyle and Dodds [14] ......................................20</td>
</tr>
<tr>
<td>2.13</td>
<td>Evolutions of $U$ parameter from the node displacement method with the second node behind the crack tip, compliance offset method, and the ACR technique .................................................................21</td>
</tr>
<tr>
<td>2.14</td>
<td>Evolutions of $U$ parameter from the node displacement method with the first node behind the crack tip, compliance offset method, and the ACR technique .................................................................23</td>
</tr>
<tr>
<td>2.15</td>
<td>Evolution of $U$ parameter from the node displacement method with the first node behind the crack tip, and the contact stress method .........................24</td>
</tr>
<tr>
<td>3.1</td>
<td>A sketch of an infinite sheet loaded by a resultant pin load $P$ ........................30</td>
</tr>
</tbody>
</table>
4.7 Geometry factors for a single crack from a pin-loaded hole in a finite sheet....................................................................................................................62
5.1 Schematic illustration of crack extension..................................................70
5.2 Load-displacement curves under linear elastic conditions: (a) no residual stress field, (b) with compressive residual stress field .........................71
5.3 Sheet geometry with dimensions: \( h = 130 \text{ mm}, \ 2w = 44.45 \text{ mm}, \ r = 3.535 \text{ mm}, \ a = 0.345 \text{ mm} \) .............................................................................74
5.4 Stress-strain curve for AA7075-T6 Aluminum Alloy [20] .......................74
5.5 Finite element mesh ...................................................................................75
5.6 Hoop residual stress field around the hole for different radial interferences ...........................................................................................................77
5.7 Comparison of \( K_{irs}/K_{i,max} \) solutions from crack compliance technique and \( J \)-Integral method for elastic crack growth........................................78
5.8 Convergence study for elastic-plastic crack growth results ....................79
5.9 Comparison of \( K_{irs}/K_{i,max} \) solutions from elastic and elastic-plastic crack growth simulations with 0.8% cold work and \( S_{max}/\sigma_{ys} = 0.3 \) ...............80
5.10 Comparison of \( K_{irs}/K_{i,max} \) solutions from elastic and elastic-plastic crack growth simulations with 1.2% cold work and \( S_{max}/\sigma_{ys} = 0.4 \) ...............81
5.11 Comparison of \( K_{irs}/K_{i,max} \) solutions from elastic and elastic-plastic crack growth simulations with 1.6% cold work and \( S_{max}/\sigma_{ys} = 0.4 \) ...............83
A.1 Cracked plate subjected to fixed load \( P \) .................................................94
A.2 Cracked plate subjected to a fixed load \( P \) and virtual pair load \( F \) ..........96
CHAPTER I
INTRODUCTION

1.1 Objective of this Research

This research work has four main objectives. The first objective is to study and correlate four different methodologies characterizing fatigue crack closure using three dimensional finite element simulations of plasticity induced crack closure. The first two methods considered are numerical, and they are based on near crack tip behavior. The last two methods, namely the adjusted compliance ratio (ACR) and the standardized compliance offset methods, are experimental methods and they use remote load-displacement measurements to assess fatigue crack closure. ACR in particular does not have a strong theoretical foundation that it is built upon. Hence, the present study is intended to provide some mechanics based understanding of this technique.

The second objective is to determine stress fields around a single cold-worked pin-loaded hole using two dimensional elastic-plastic finite element analyses under plane stress conditions. This, in part, will help identify susceptible locations for crack initiation around the hole. The third objective is to quantify the beneficial influence of the residual stress field from the cold working process by computing stress intensity factors (SIF) for a single crack emanating from the cold worked pin-loaded and open holes loaded in tension.

Finally, the fourth objective is to perform numerical investigations of an on-line crack compliance technique that is used to measure residual stress fields in the crack
growth region. This relatively new experimental technique has a theoretical foundation that is based on linear elastic fracture mechanics (LEFM) principles. Because of simplicity and ease of use, this method is currently gaining some attention by the research community and experimentalists. The current study intends to evaluate the influence of more realistic conditions with plastic deformations near the crack tip on the performance of the on-line crack compliance technique.

1.2 Organization of Dissertation

Chapter II discusses plasticity induced fatigue crack closure simulations performed using a three dimensional finite element model of a single edge-cracked tension specimen under constant amplitude loading and load ratio $R = 0$. WARP3D finite element research code is employed to perform the crack growth simulations. ANSYS is used to create the finite element mesh and to prescribe the boundary conditions. Material plasticity is included using a bilinear kinematic hardening constitutive model with Von-Mises yield locus. Incremental loads are applied in a cyclic fashion and the through-thickness crack is set to grow by one element length in each cycle. The evolutions of crack driving force are compared in terms of $U = \Delta K_{\text{eff}} / \Delta K$, where $\Delta K_{\text{eff}}$ and $\Delta K$ are the effective and applied stress intensity factor ranges, respectively.

In chapter III, a detailed two-dimensional, elastic-plastic finite element study of a pin-loaded hole is presented. A thin rectangular aluminum alloy sheet (7075-T6) with a circular hole is considered under plane stress conditions. The hole is loaded purely by a rigid pin to different load magnitudes. Appropriate contact elements are used at the pin-hole interface to transfer the traction loads from one surface to another. Material nonlinearities for the sheet and friction are included in the analyses. Radial and hoop
stress solutions along the pin-hole interface are compared in elastically and plastically loaded holes. The influence of friction on the stress results is studied. The locations and magnitudes of the peak hoop stresses are determined. Lastly, an initial residual field is introduced around the hole by a cold expansion simulation before a subsequent pin loading analysis. Because the cold expansion process involves some reverse yielding, both isotropic and kinematic material hardening models are considered.

In chapter IV, mode I stress intensity factors are computed for a single crack emanating from a cold expanded hole in a finite sheet. Two different loading scenarios are considered, which are: (a) open hole loaded in tension, and (b) pin-loaded hole. The stress intensity factors are computed for different cold working levels and crack sizes using $J$ integral and weight function methods.

In chapter V, the on-line crack compliance technique for residual stress measurement is studied by using two dimensional (2D) plane stress finite element simulations of crack growth from a cold worked hole in a rectangular sheet. This experimental technique uses incremental crack face displacements measured during fatigue crack growth testing to generate information on the existing residual stresses along the crack line. In this part of the study, the stress intensity factors due to the residual stress field normalized by the maximum applied stress intensity factors, $K_{Iel}/K_{Imax}$, are obtained from the on-line crack compliance technique and the $J$-integral approach. Two different cases are considered with regard to material behavior: (a) purely elastic and (b) elastic-plastic. The obtained solutions are presented for different cold work levels and applied loadings.
CHAPTER II
SIMULATION AND COMPARISON OF CRACK CLOSURE ASSESSMENT METHODOLOGIES

2.1 Introduction

In 1970, Elber [1] discovered that a permanent plastic wake behind the crack tip causes a growing fatigue crack to prematurely close before the minimum load is reached. This phenomenon, known as plasticity induced crack closure (PICC), gave rise to a major advancement in understanding crack growth behavior in metallic materials, particularly under variable amplitude loading. In addition to PICC, other closure mechanisms have since been identified including oxide induced closure, roughness induced closure, and the closure induced by viscous fluids [2]. With all these mechanisms, the effective stress intensity factor range ($\Delta K_{\text{eff}} = K_{\text{max}}/K_o$, where $K_{\text{max}}, K_o$ are the maximum applied and crack opening stress intensity factors, respectively, as shown in Figure 2.1) is recognized to be the crack driving force.
Several analytical, numerical, and experimental methodologies have been developed to assess PICC. Two-dimensional analytical models were proposed by Budiansky and Hutchison in 1978 [3], and by Newman in 1981 [4]. Daniewicz et al. [5] later generalized Newman’s strip-yield model to consider arbitrary two-dimensional geometries for which a weight function is known. In this model, the plastic wake and crack tip plastic zone in an elastic body are discretized using elements under a uniaxial stress state exhibiting a rigid perfectly-plastic behavior. The crack opening stress intensity factor $K_o$ is determined from the crack surface traction that is exerted by the plastic wake under the minimum loading. This particular technique to compute $K_o$ is known as a contact stress method, since it involves the contact stress distribution along the crack surface under minimum loading.

More effort has been spent on numerical investigations of PICC using the finite element (FE) method. Researchers have suggested several ways to characterize the crack opening levels. The opening load has been defined most commonly as the applied load at which the first node behind the crack tip is detached from the crack plane [6–11]. This is motivated from the fact that a crack under plane stress conditions growing through a
residual stress free zone opens in an unzipping fashion such that the first node immediately behind the crack tip opens last. However, other investigators have used the second node behind the crack tip, because the first node produced an unrealistically large opening load level [12–14]. The authors in [15] and [11] proposed a different criterion, which is based on the assumption that a crack tip driving force is active only when the stress state at the crack tip is tensile. Consequently, they measured the opening load when the crack tip nodal stress changed sign from compressive to tensile. To eliminate the dependence on any specific node, Solanki et al. [16] implemented a contact stress method to compute opening load levels. DeMatos and Novell in their recent paper [17] assessed all of these techniques using an analysis of the middle crack in tension (MT) specimen. They concluded the contact stress method is optimal, providing good accuracy with less need for computing power. For a comprehensive review of literature on finite element analysis (FEA) of PICC, the reader may further refer to [17,18].

Experimental techniques that have been developed to measure $K_o$ include ultrasonics, potential drop, eddy current, acoustic emission, high magnification photography, and compliance measurement. The compliance measurement has become the conventional approach because of its experimental simplicity [19]. Conceptually this method employs Elber’s first closure observation [1], in which the compliance from the applied load and the subsequent displacement attains a constant value when the crack fully opens. In practice, however, subtle changes in compliance and the existence of noise in measured data requires the definition of compliance offset value to characterize opening loads in a repeatable manner [19].

Recently, new means for measuring the crack driving force experimentally, namely compliance ratio (CR) and adjusted compliance ratio (ACR) were proposed by
Donald et al. [20–22]. Both of these methods use the unloading load–displacement (or strain) records in a fatigue cycle. The benefit of such measurements is that they do not rely on any specified compliance offset value. They also conceptually account for additional crack tip strain occurring below the opening load levels. These additional strains may be of significance for a growing crack under low levels of applied loading, particularly near the threshold regime, where the opening loads are high [22].

The analytical and numerical methodologies cited above are all based on the near-tip crack behavior, whereas experimental methods such as compliance offset and ACR use the remote load–displacement measurements to assess crack closure. McClung and Davidson [23] were pioneers to employ a 2D elastic–plastic FE model under plane stress conditions to analytically investigate experimental methodologies with the remote location measurements. They reported almost no closure from the ACR method for a zero load ratio (i.e. \( R = \frac{K_{\text{min}}}{K_{\text{max}}} = 0 \)). For the same load ratio, however, ASTM compliance offset produced non-zero crack tip closure that was slightly greater than one-half of the applied stress intensity factor range (\( \Delta K \)). For the load ratio of \( R = -1 \), two methodologies resulted in non-zero levels of crack tip shielding, although their magnitudes were not the same. In a recent paper [24], a strip-yield model was employed to compare three different methodologies used for characterizing fatigue crack closure: (a) ACR, (b) crack wake influence (CWI), and (c) the conventional Elber approach (compliance offset). Similar observations were made as in [23] regarding the ACR and compliance offset method results. In a continuation of these two studies, present work considers three-dimensional finite element model to perform fatigue crack growth simulations. The crack driving forces were computed using the compliance offset and ACR methods. Results obtained are compared to crack tip driving forces computed using two local measurement
methods: (a) the displacements of the first and second nodes behind the crack tip and (b) the contact stress method.

2.2 Finite Element Model

The three-dimensional model of a single edge-cracked tension specimen used by Carlyle and Dodds [14] was chosen to carry out the numerical analysis. This model was chosen because the opening load levels from the nodal displacement method using the second node behind the crack tip are reported in [14] for load ratios $R = \frac{K_{\text{min}}}{K_{\text{max}}} = -1$ and 0, thereby providing a foundation to build upon. The full model geometry and the applied loadings are depicted in Figure 2.2. The in-plane dimension of the sheet is 608 mm ($h$) by 152 mm ($w$). The sheet thickness is $B = 1$ mm. For simplicity, the influence of T-stress is eliminated by setting the initial crack length to $a_i/w = 0.596$. The T-stress magnitude approaches zero at this crack length. As the fatigue crack grows to final size, this stress becomes slightly positive with a negligible influence on resulting opening loads [14].

![Figure 2.2](image)  
Figure 2.2  Single edge-cracked tension specimen for fatigue crack growth, the sheet thickness is scaled for clarity.
Figure 2.3 shows a typical mesh, which consists of nearly 36,500 nodes and 29,500 eight-node, isoparametric brick elements. One-quarter of the specimen is modeled using two symmetry conditions: one on the crack plane and the other at the mid-thickness of the sheet. A rigid, frictionless contact surface is defined as part of the symmetry condition along the crack plane. Roychowdhury and Dodds [12] demonstrated that a FE mesh having five non-uniform sized element layers over the half-thickness gives sufficiently accurate opening load levels at each through-thickness position. The present model contains five element layers with thicknesses $0.25B$, $0.15B$, $0.05B$, $0.03B$, and $0.02B$, respectively, starting from the mid-plane and moving toward the free surface of the sheet. Highly refined elements of equal in-plane size occupy the crack growth region as shown in Figure 2.3.

![Typical finite element mesh](image)
2.3 Material Model

A bilinear kinematic hardening constitutive model was used in this analysis. A Mises yield locus with a constant radius rigidly translates in the direction normal to the surface approximating Bauschinger’s effect when yielding occurs [25]. The modulus of elasticity and the Poisson’s ratio were $E = 250\sigma_o$ and $\nu = 0.3$, respectively, where $\sigma_o$ is the flow stress of the material. The tangent modulus used was $E_T = 0.05E$. These properties approximately represent 2024-T3 aluminum alloy sheet. According to Roychowdhury and Dodds [12], using a finite strain formulation does not alter the computed opening stress significantly, and a small strain formulation was chosen in the present study for computational efficiency.

![Figure 2.4 Workflow of the analysis](image)

2.4 Workflow

Figure 2.4 presents a general workflow of the analysis. The WARP3D fracture mechanics research code [24] was employed to perform the crack growth simulations. This FORTRAN based finite element code has specific features oriented toward solving
three dimensional elastic-plastic fracture mechanics problems. It does not have pre-processing program, but the software package includes PATWARP translator that can be used to translate PATRAN generated FE mesh with applied loading and boundary conditions into WARP3D input file. In the current study, ANSYS was used to generate the FE mesh and to prescribe necessary boundary conditions. Next, an ANSYS Parametric Design Language (APDL) script AN2PAT was used to export the FE model including the nodal coordinates, element incidences, nodal constraints and pressure loadings into the neutral file. It was then translated into WARP3D input file via the PATWARP translator. The input file was further edited to define material properties, loading increments, rigid contact surface along the crack plane and WARP3D specific instructions to execute straight-through crack growth by node release. Throughout the solution process the WARP3D continuously stored nodal force and displacement data in the binary output file. Finally, four different FORTRAN codes were developed to read through the output file and to extract the crack closure parameters $U$. Each post-processing code corresponds to one of the methodologies that are described in section 2.6.

2.5 Crack Growth Modeling

The sheet was subjected to a constant amplitude cyclic loading with a load ratio $R = 0$ as shown in Figure 2.4. The total amount of crack growth modeled was $\Delta a/B = 2.3$. The maximum loads applied during the simulations produced plastic zone sizes $r_p$ that were much less than in-plane dimensions of the sheet. The maximum applied stress intensity factors used were $\bar{K} = K/\sigma_0 \sqrt{B} = 1.5, 2.0, \text{ and } 2.5$. The through-thickness crack was forced to grow uniformly (through the thickness $B$) by node release at the peak
load of each cycle. For more details on three-dimensional crack growth modeling, the reader may refer to [14].

2.6 Crack Closure Characterizing Techniques

Attention is confined herein to the crack closure parameter $\Delta K$, where $\Delta K$ represents the stress intensity factor range from the applied loading. Using the conventional Elber crack closure approach, we may write

\[
U = \frac{\Delta K_{\text{eff}}}{\Delta K} = \frac{K_{\max} - K_o}{K_{\max} - K_{\min}} = \frac{1 - K_o / K_{\max}}{1 - R} \tag{2.1}
\]

where $K_{\max}$, $K_{\min}$ are the maximum and minimum applied stress intensity factors. Because the $R$ considered in this study is zero, the equation 2.1 becomes

\[
U = 1 - K_o / K_{\max} \tag{2.2}
\]

The parameter $U$ was determined from four different methodologies: (a) node displacement method, (b) contact stress method, (c) compliance offset, and (d) ACR.
2.6.1 Node Displacement Method

Figure 2.5 depicts the locations of the first and second nodes behind the crack tip. These nodes at each through-thickness position were monitored during the incremental loading process. The crack opening load step in which the vertical displacements of these nodes attained positive values was identified for each cycle and the opening SIFs $K_o$ were computed from the corresponding load levels. The $U$ values were computed using equation 2.2.

![Node displacement diagram]

Figure 2.6 Node displacement method

2.6.2 Contact Stress Method

The plastic wake shown in Figure 2.7 produces compressive residual stresses along the crack face under the minimum loading. Since the entire crack surface needs to be free of traction when it is fully open, existing tractions must be overcome by applying a stress distribution on the crack face that is equal in magnitude but opposite in sign to the residual contact stresses.
Although, the finite element model is three dimensional, for simplicity the through-thickness nodal stress results are averaged to transform the contact stress to a two-dimensional distribution. The opening stress intensity factor $K_o$ is determined using a weight function for a single edge crack in a finite width sheet. From [26], this weight function can be expressed as

$$m(\alpha, \xi) = \frac{1}{\sqrt{2\pi a}} \sum_{i=1}^{5} \beta_i(a) \left[ 1 - \frac{\xi}{\alpha} \right]^{-\frac{3}{2}}$$  \hspace{1cm} (2.3)

where $\xi$ and $\alpha$ are non-dimensional quantities $\alpha = a/w$ and $\xi = x/w$ (see Figure 2.8). The functions $\beta_i$ are given by

Figure 2.7  Plastic wake zone

Figure 2.8  A single edge crack in a finite width plate
\[ \beta_1(\alpha) = 2.0 \]
\[ \beta_2(\alpha) = \left[ 4af'_r(\alpha) + 2f_r(\alpha) + \frac{3}{2} F_2(\alpha) \right] / f_r(\alpha) \]
\[ \beta_3(\alpha) = \left\{ \frac{1}{2} \left[ 5F_3(\alpha) - F_2(\alpha) \right] \right\} / f_r(\alpha) \]
\[ \beta_4(\alpha) = \left\{ \frac{1}{2} \left[ 7F_4(\alpha) - 3F_3(\alpha) \right] \right\} / f_r(\alpha) \]
\[ \beta_5(\alpha) = \left\{ \frac{5}{2} F_4(\alpha) \right\} / f_r(\alpha) \]  \hspace{1cm} (2.4)

where

\[ F_1(\alpha) = 4f_r(\alpha) \]
\[ F_2(\alpha) = \frac{1}{12\sqrt{2}} \left[ 315\pi\phi(\alpha) - 105V_r(\alpha) - 208\sqrt{2} f_r(\alpha) \right] \]
\[ F_3(\alpha) = \frac{1}{30\sqrt{2}} \left[ -1260\pi\phi(\alpha) + 525V_r(\alpha) + 616\sqrt{2} f_r(\alpha) \right] \]
\[ F_4(\alpha) = \sqrt{2} V_r(\alpha) - [F_1(\alpha) + F_2(\alpha) + F_3(\alpha)] \]  \hspace{1cm} (2.5)

The related functions \( \phi(\alpha), E_1(\alpha) \) and \( f_r(\alpha) \) in equation 2.5 are expressed as

\[ \phi(\alpha) = \frac{1}{\alpha^2} \int_0^\alpha s \cdot [f_r(\alpha)]^2 ds \]
\[ V_r(\alpha) = \sum_{n=0}^3 [\gamma_n \alpha^n / (1 - \alpha^2)] \]
\[ f_r(\alpha) = \sum_{i=0}^7 [\lambda_i \alpha^i / (1 - \alpha)^{3/2}] \]  \hspace{1cm} (2.6)

and the coefficients \( \gamma_n \) and \( \lambda_i \) are

\( \gamma_n \): 1.1214, -1.6349, 7.3168, -18.7746, 31.8028, -33.2295, 19.1286, -4.6091.

The crack opening stress intensity factor \( K_o \) was computed using the weight

function integral

\[ K_o = \int_0^\alpha \sigma(\xi)m(\alpha, \xi) d\xi \]  \hspace{1cm} (2.7)
where $\sigma(\xi)$ is the contact stress distribution along the crack line under the minimum applied loading, and $m(\alpha, \xi)$ is the weight function given in equation 2.3.

Finally, the $U$ parameter is computed using equation 2.2. It must be noted that, because of the complex nature of the weight function, the expression in equation 2.7 was numerically integrated using Gauss-Chebyshev quadrature. This quadrature conveniently circumvents the crack tip singularity associated with the weight functions enhancing the efficiency of numerical integration [27].

2.6.3 Compliance Offset Method

To determine the crack opening loads using the compliance offset method, the procedure described in the ASTM standards [19] was followed. This method requires measurements of applied loads and corresponding displacements. Instead of experimentally acquiring these data, FEA solutions were used for comparison, with the displacement computed at the crack mouth. First, the open crack compliance $C_o$ was measured from the top 25% segment of the unloading portion of the load cycle. Then, starting from the maximum applied force on the loading curve, the compliance values $C$ were computed for each 10% segment of the load range, which are labeled as $S1$, $S2$, $S3$ and etc in Figure 2.8. These segments overlap each other by 5% of the load range. The compliance offset was computed for each segment as follows

$$\text{CompOff} = \frac{C_o - C}{C_o} \times 100\%$$

(2.8)
Figure 2.9   Compliance offset method

The opening levels $P_o$ corresponding to 0.5%, 1.0%, and 2.0% compliance offsets were recorded. The parameter $U$ was computed using the relationship in equation 2.2 with $P_o$ and $P_{max}$. A typical plot of the load versus compliance offset is given in Figure 2.9. Negative offset values observed near the maximum applied load are due to the plastic deformation in the vicinity of the crack tip. For the loads below the peak load, the offset curve approaches zero until the initiation of a positive deviation, which represents the start of the closure event.
2.6.4 ACR Method

Implementing the ACR method [20-22] to characterize the crack tip shielding, $U$ may be written as

$$ U = \frac{C_s - C_i}{C_o - C_i} $$

(2.9)

where $C_o$ is an open crack compliance, $C_i$ is the compliance of the initial notch, and $C_s$ is the secant compliance shown in Figure 2.11. These compliance values were computed at the crack mouth from the unloading portion of the same load-displacement results used in the compliance offset method.
2.7 Model Validation

For three-dimensional straight-through crack growth simulations, at least ten elements in the forward plastic zone and two elements in the reverse plastic zone are required to adequately discretize the crack front [12]. The element counts in the forward and reverse zones at each through-thickness location for the applied loadings are summarized in Table 2.1. The convergence criteria are satisfied for each case except when $K = 1.5$, for which the layer nearest the mid-thickness ($z/B = 0$) location has only one element in the reverse plastic zone. However, the $U$ solutions from the node displacement method compare well with the results of Carlyle et. al. [14] for $K = 1.5$ as shown in Figure 2.12, giving confidence for future simulations conducted in this study. The second node behind the crack tip was used as this was used by Carlyle et. al.
Table 2.1  Element counts in forward and reverse zone at peak load

<table>
<thead>
<tr>
<th>Layer Location (z/B)</th>
<th>( \bar{K} = 1.5 )</th>
<th>( \bar{K} = 2.0 )</th>
<th>( \bar{K} = 2.5 )</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Forward</td>
<td>Reverse</td>
<td>Forward</td>
</tr>
<tr>
<td>0.125</td>
<td>23</td>
<td>1</td>
<td>38</td>
</tr>
<tr>
<td>0.325</td>
<td>20</td>
<td>2</td>
<td>37</td>
</tr>
<tr>
<td>0.425</td>
<td>17</td>
<td>3</td>
<td>37</td>
</tr>
<tr>
<td>0.465</td>
<td>17</td>
<td>2</td>
<td>36</td>
</tr>
<tr>
<td>0.490</td>
<td>17</td>
<td>2</td>
<td>37</td>
</tr>
</tbody>
</table>

Figure 2.12  Evolution of \( U \) parameter at each though-thickness location: present study compared with Carlyle and Dodds [14]

2.8  Results and Discussion

Figure 2.13 presents the progression of \( U \) parameters from the node displacement, compliance offset, and ACR methodologies, which are plotted as a function of the normalized crack growth \( \Delta a / B \). The maximum applied stress intensity factors considered are \( \bar{K} = K / \sigma_y \sqrt{B} = 1.5, 2.0, \) and \( 2.5 \). Node displacement solutions, given in symbols, pertain to the second node behind the crack tip at three different through-thickness positions \( z/B = 0, 0.25, \) and \( 0.5 \). Compliance offset solutions are given in numerical
symbols for 0.5%, 1.0% and 2.0% offsets. It must be noted that 2.0% offset is the standard criterion used to experimentally measure the opening loads as defined by ASTM [19]. ACR values are plotted with solid lines in the figure.

![Figure 2.13](image)

Figure 2.13: Evolutions of $U$ parameter from the node displacement method with the second node behind the crack tip, compliance offset method, and the ACR technique

When the applied load is $K = 1.5$, a significant through-thickness variation is observed from the node displacement method with the free surface providing the lowest $U$ value and the mid-thickness location giving the highest (see Figure 2.13(a)). All compliance offset curves readily fall within this range. Also, note that the smaller offset criterion returned lower $U$ values when using the compliance offset method as expected.
When the applied load is increased to $\bar{K} = 2.0$, the through-thickness variation is diminished as shown in Figure 2.13b. When $\bar{K} = 2.5$, $U$ values at $z/B = 0$ and 0.25 converge to a single level, which is lower than that at outer surface (i.e. at $z/B = 0.5$, see Figure 2.13(c)). This indicates the free surface is being propped open due to the large amount of plasticity. For the applied loads $\bar{K} = 2$ and 2.5, the curves corresponding to 1.0% and 2.0% offsets remain at high levels, and the 0.5% curves approach the upper limits of the node displacement method solutions (Figure 2.13(b,c)). The ACR methodology consistently produced the largest $U$ values, with values slightly above 0.9 for $\bar{K} = 1.5$ and slightly below 0.9 for the higher load levels. Conceptually, this method should yield higher $U$ levels because of the additional cyclic strains that it considers below the conventional opening loads. However, the results are significantly greater than the next largest predicted levels that were obtained from the conventional 2% compliance offset as shown in Figure 2.13. Similar observations were reported previously in [23, 24] regarding the ACR methodology for load ratios $R = 0$ and 0.25 when a strip-yield model was used to model the crack growth.

In Figure 2.14, the same compliance offset and ACR measurements from the FE model are compared with the node displacement method solutions acquired by monitoring the first node behind the crack tip. The node displacement method solutions are given in symbols, compliance offset results are plotted in numeric symbols, and the ACR measurements are presented in solid lines. It is observed that the first node behind the crack tip returns $U$ values that are significantly lower than those obtained from the other two methodologies. Moreover, as shown in Figure 2.14(a,b), the through thickness variations for the applied loads $\bar{K} = 1.5$ and 2 are not as large when compared with the solutions pertaining to the second node behind the crack tip given in Figure 2.13. In fact,
all $U$ parameters across the sheet thickness merge to a nearly single level when $K = 2.0$.

For the applied load $K = 2.5$, the through thickness locations $z/B = 0$ and 0.25 provide the same $U$ value, which is lower than the value at $z/B = 0.5$ (Figure 2.14(c)).

![Figure 2.14](image)

**Figure 2.14** Evolutions of $U$ parameter from the node displacement method with the first node behind the crack tip, compliance offset method, and the ACR technique.

Finally, Figure 2.15 compares the evolution of $U$ parameters obtained from the contact stress method and the node displacement method using the first node behind the crack tip. The contact stress method results are given in solid lines, and the symbols represent the results from the nodal displacements. These two methodologies compare well. Particularly, the solutions obtained from the first node behind the crack tip at
through thickness location $z/B = 0.25$ are in good agreement with the contact stress method results computed at minimum load.

In general, it is observed that the results from the nodal displacement, compliance offset, and contact stress methods follow a similar trend; their curves decline monotonically to achieve a stabilized level throughout the crack growth. The ACR curves for $K = 2.0$ and 2.5 follow a different trend with rapid decrease to a minimum level that is followed by a gradual rise.

Figure 2.15 Evolution of $U$ parameter from the node displacement method with the first node behind the crack tip, and the contact stress method
2.9 Conclusions

Three dimensional finite element plasticity induced crack closure simulations under constant amplitude cyclic loading with $R = 0$ were performed. A single edge-cracked tension specimen was chosen to carry out the analysis. Crack tip driving forces in terms of $U$ were computed using four different techniques: (a) node displacement method, (b) contact stress method, (c) compliance offset and (d) ACR. Using the node displacement method, the first and second nodes behind the crack tip through the sheet thickness were monitored to assess the opening load levels. In the contact stress method, the average through-thickness residual stress distributions along the crack surface at minimum loading and the two dimensional weight function specific to the configuration were used to determine $K_o$. The opening loads corresponding to 0.5%, 1.0%, and 2.0% compliance offsets were also computed from the load versus crack mouth displacement results employing the procedure described in ASTM standards [19]. The unloading portion of the same load-displacement data was considered to obtain the ACR method $U$ value.

Amongst these four techniques, the node displacement method with the first node behind the crack tip and the contact stress method produced the lowest $U$ values. These two methods agreed well with one another for the considered applied loadings. Using the second node behind the crack tip produced higher $U$ values and exhibited larger through-thickness variation especially at lower load levels. The compliance offset method gave the next higher level. The 0.5% offset curve provided a promising agreement with the second node behind the crack tip results. The lowest compliance offset considered herein was 0.5%, whereas the opening loads must be ideally computed as soon as the instantaneous compliance starts to diverge from the open crack compliance. Reducing
this offset value may further improve the agreement with the second node displacement method. The highest $U$ values were obtained from the ACR for all considered applied loadings. Theoretically, this method is expected to produce larger $U$ levels, since it accounts for additional damaging cyclic strains below the conventional opening loads. However, the computed ACR $U$ values were close to unity, indicating that the ACR solutions for $R = 0$ are not significantly different from the LEFM conditions. Also, for the higher load levels, the evolution of the ACR values with crack growth exhibited a distinct shape with a rapid decrease to a minimum level that was followed by a gradual rise. This behavior was not observed when using any of the other methods.
2.10 References


CHAPTER III
STRESS ANALYSIS FOR COLD WORKED PIN-LOADED HOLES

3.1 Introduction

Pin loading is a classical problem, which drew attentions of many researchers in the past. Numerous independent techniques have emerged to evaluate the stress state around a pin loaded hole. Experimental methods include photoelasticity [1-3], moire interferometry [4-5], electronic speckle pattern interferometry [6], and strain gages [2]. Analytical solutions have been developed using linear elasticity with complex variable methods [7-10] and stress functions [11-14]. Noble and Hussain [7], for example, considered the problem of an infinite plate with a circular hole loaded by a pin under frictionless conditions. They formulated the problem in terms of dual series, which are then converted to an equivalent Cauchy-type or airfoil equation assuming the elastic constants of the pin and the plate satisfy the relationship \( (1-2\nu_1)/G_1 = (1-2\nu_2)/G_2 \) where \( G \) and \( \nu \) denote the shear modulus and Poisson’s ratio, respectively. Rao [11] employed an inverse method along with a collocation technique to solve for the unknown constants in an Airy stress function satisfying the prescribed boundary conditions. He considered both frictionless and nonzero friction, bonded interfaces, as well as clearance and interference fit pins. Mangalgiri et. al. [12] extended the work of Rao for pin joints under combined pin and plate loads with frictionless contact. In ref. [8], a complex Fourier series, collocation procedure, and iteration technique were used to solve for stresses around pin loaded orthotropic plates for different levels of friction as well as the hole clearance. For
the case of a rigid pin and zero friction, Muzushima and Hamada [13] also adopted a numerical approach to solve for the stress and displacement distributions around the loaded hole. Ho and Chau’s work [14] regarding an infinite plate loaded by a pin is one of the recent closed form solutions that include friction, arbitrary stiffness for the pin material, as well as uniform and non-uniform shear loads distributed over the pin section. They solve the problem by partitioning it into two auxiliary sub-problems. The first sub-problem solves for the stress distribution in the plate loaded by a pin of a different material that is perfectly bonded to the plate. The second auxiliary problem seeks a solution so that the normal $\sigma_r$ and shear $\sigma_{r\theta}$ stress components cancel out at the top half of the circular boundary ($0^\circ \leq \theta \leq 180^\circ$, $r = R$), whereas the normal and shear contact stress distributions proportional to $\sin(\theta)$ and $\sin(2\theta)$ are produced at the bottom half of the hole ($-180^\circ \leq \theta \leq 0^\circ$, $r = R$, see Figure 3.1). The sinusoidal radial pin load distribution assumption is also used in [10] in development of an analytical solution for pin loaded elastic orthotropic plates via complex stress functions for frictionless conditions ($\mu = 0$).

![Figure 3.1](image-url)  
**Figure 3.1** A sketch of an infinite sheet loaded by a resultant pin load $P$
The analytical solutions listed above are obtained for a linear elastic plate material. However, the problem becomes much more complicated if one considers the material non-linearity combined with friction and finite plate dimensions. Moreover, pinned connections rarely occur in isolation since most of the structures have multiple pinned connections with dissimilar configurations. This makes the finite element method (FEM) a more desirable approach, because it can eliminate all of these concerns as long as the model size is kept at a reasonable level. Two dimensional (2D) FEA of the pin loaded plates with an elastic plate, rigid pin, and a frictionless contact are previously reported in [15-16]. Local stress distributions from 2D and 3D finite element (FE) models with elastic pins and frictionless contact were presented in [18]. Iyer [18] and Lanza di Scalea [6] consider both pin elasticity as well as friction in their 2D plane strain [18] and plane stress [6] models. Iyer demonstrated that the stress solutions at the interface are largely independent of the material pair provided that the pin and the plate are metallic and friction is small. Yavari et. al. [19] have performed a parametric study with respect to some design factors such as the plate width, edge distance, clearance, and friction at the pin hole interface. Kumar [20] included material plasticity for the case of perfectly smooth pin-hole interface (i.e. \( \mu = 0 \)), and obtained the stress distributions around the hole under constant amplitude cyclic loading.

The objective of the present study is to continue the 2D FE modeling efforts of the pin loaded hole problem with further sophistication, namely considering the material nonlinearity and friction as well as pre-existing residual stresses around the hole from the cold-working process. First, convergence studies were performed to validate the elastic and elastic-plastic model results. Furthermore, the FE model with linear elastic material properties was validated using: (a) a closed form solution by Ho and Chau [14], (b) a FE
solution by Yavari et. al. [19]. Next, the linear elastic and elasto-plastic solutions were compared, and the influence of friction was studied. Finally, a compressive residual stress field was introduced around the hole by cold expansion simulation. Cold expansion of a hole, initially developed by the Boeing Company in 1960, is a life enhancement process used to mitigate the effect of the stress concentrations by creating a compressive circumferential residual stress field around the hole [21]. Because this process involves some reverse yielding, the effect of both isotropic and kinematic material hardening models is considered. The sheet was then pin loaded and the subsequent changes in the stress state around the hole are presented.

3.2 Configuration and Material Models

Consider a rectangular finite sheet with a circular hole as shown in Figure 3.2. The size of the thin rectangular sheet is 65.0 mm ($h$) by 44.45 mm ($2w$) with a 7.07 mm diameter hole ($2r_o$) located at 22.23 mm distance ($d$) from the bottom edge. The sheet thickness is $t = 2.03$ mm. The hole is loaded purely by a neat-fit pin with a magnitude $P$. The sheet is constrained in the $y$ direction at the top edge (Figure 3.2). The diameter of the pin is same as the hole diameter since it is a neat-fit configuration. Out of plane bending effects for the sheet are ignored, which can be a reasonable approximation for the dual lap joints. This simplification may not be appropriate for single lap pin joints.
The sheet material is 7075-T6 aluminum alloy with a yield strength of $\sigma_{ys} = 483$ MPa. The Young’s modulus and Poisson’s ratio are $E = 72.5$ GPa and $\nu = 0.3$, respectively. Figure 3.3 shows the stress-strain curve used [22]. For the elastic case, only the elastic domain of the curve is used with the $E$ and $\nu$ to describe the material behavior. For the elastic-plastic analysis, the stress-strain data in both elastic and plastic domains are used with the Von-Mises yield criterion. Two different work hardening material models were considered during the cold expansion simulation: multilinear isotropic hardening and multilinear kinematic hardening. As will be shown, these two hardening models result in different residual stress fields from the cold expansion simulation due to Bauschinger’s effect, which is incorporated in the kinematic hardening material model. For simplicity, the pin is assumed to be a rigid body.
3.3 Finite Element Model and Boundary Conditions

ANSYS 11.0 [23] was used to carry out the numerical simulations. 2D plane stress conditions were assumed because of the small sheet thickness. Due to symmetry, one half of the specimen was modeled with appropriate symmetry boundary conditions, which are shown as vertical rollers in Figure 3.4. These rollers are the constraints in the $x$ direction, whereas the horizontal rollers applied at the sheet top edge indicate the constraints in the $y$ direction.

The FE model was meshed using 2D 4-node linear structural elements with a highly refined mesh around the hole edge (Figure 3.5). Rigid-to-deformable surface-to-surface contact elements with an Augmented Lagrangian algorithm were used at the pin-hole interface so that the normal and shear traction loads can be transferred from the pin to the hole surface. Two different cases were studied regarding the friction. In the first case, the pin-hole interface was assumed to be perfectly smooth with zero friction. In the second case, a Coulomb friction model was used with an arbitrary friction coefficient $\mu = 0.2$. 

Figure 3.3 Elasto-plastic stress-strain curve of AA7075-T6 sheet
Assuming that the pin-loading is a sufficiently slow process, it was simulated in a quasi-static manner without considering any dynamic or time dependent response of the material. The magnitude of the point load $P$, which is applied downward at the center of
the pin (Figure 3.4), was gradually increased in several load-steps up to the maximum level. Note that the point load is, in general not realistic, since the shear loads are distributed over the pin section in some fashion. However this will not have any significance in the present work because the pin was assumed to be rigid.

In the last stage of the study, a residual stress field was created around the hole by cold expansion simulation prior to subsequent pin-loading analyses. Cold expansion is a three dimensional process, in which a tapered mandrel is pulled through the hole of the structure, and the resulting residual stress field varies through the sheet thickness [24]. Although, it is not possible to realistically simulate the actual process in 2D, it can be performed in a rather simplistic way by following the two steps given below:

1) Hole expansion. Uniform radial displacements, equal to the 2% interference amount, are applied on the nodes of the hole edge. By this means the hole of the sheet is plastically expanded.

2) Hole recovery. Pre-applied displacements are removed, which causes the partial but not full elastic recovery. This step creates the desired compressive residual stresses around the hole edge.

3.4 Model Validation

2D elastic and elastic-plastic FE models were first validated by performing convergence studies with respect to mesh density and loading increments (i.e. total number of load steps). Example convergence studies for the elastic-plastic model with friction coefficients $\mu = 0$ and 0.2 are given in Figure 3.6. In each plot of the figure, normalized stress values ($\sigma_i/\sigma_{ys}$) for progressively refined element sizes $L$ are plotted.
versus the angular location $\theta$ along the hole circumference. The applied normalized pin-load magnitude is $S_b/\sigma_{ys} = 1.0$, where $S_b$ is the average bearing stress

$$S_b = \frac{P}{2r_ot}$$

(3.1)

here, $P$ is the pin-load magnitude, $r_o$ hole radius, and $t$ thickness of the sheet.

Figure 3.6  Convergence studies for radial, and hoop stress distributions at $r = r_o$, $L =$ element size at the hole edge (normalized by $\pi r_o$)

Note that $L$ is the length of the elements at the hole edge normalized by $\pi R$. By default in ANSYS, $L$ was reduced by a factor of three during each mesh refinement. As shown in Figure 3.6(a,b), both radial and hoop stress distributions are easily converged in
the frictionless case when \( L \) is reduced to \( L = 0.01 \times 3^{-2} \). Including a friction coefficient \( \mu = 0.2 \) resulted in a slow local convergence in the vicinity of the angular location \( \theta = -90^\circ \). Although the radial stress magnitude \( (\sigma_r/\sigma_{ys}) \) near \( \theta = -90^\circ \) looks as if it may be converged with the smallest element size \( L = 0.01 \times 3^{-5} \), the hoop stress values \( (\sigma_\theta/\sigma_{ys}) \) require further mesh refinement (Figure 3.6(c,d)). Results away from this region are readily converged with the same level of mesh refinement as in the frictionless case (i.e. \( L = 0.01 \times 3^{-2} \)). Similar behavior was also observed when linear elastic material properties were used for the sheet, suggesting that the friction as well as the rigid pin assumption may be the source of the convergence difficulties. Because the smallest element size considered in this study \( (L = 0.01 \times 3^{-2}) \) lead to an impractically large model size with approximately 450K degrees of freedom, the second level of mesh refinement with \( L = 0.01 \times 3^{-2} \) was chosen to be the final and optimum mesh density for all cases. In the results that follow for \( \mu = 0.2 \), attention is confined to the interval \(-85^\circ \leq \theta \leq 90^\circ\), where the solutions are converged.

Further validations were done by solving two previously studied independent problems found in the literature: (a) a closed form solution by Ho and Chau [14] for an infinite sheet, (b) a numerical solution by Yavari et. al. [19]. From the Ho and Chau solution, if the pin is assumed to be rigid \( (E_\text{pin} \to \infty) \), the contact stresses are

\[
\sigma_r = \begin{cases} 
\frac{PM_1 \sin(\theta)}{\pi r_0} & \text{for } -\pi \leq \theta \leq 0 \\
0 & \text{for } 0 \leq \theta \leq \pi 
\end{cases}
\]

\[
\sigma_\theta = \begin{cases} 
-\frac{PM_2 \sin(2\theta)}{\pi r_0} & \text{for } -\pi \leq \theta \leq 0 \\
0 & \text{for } 0 \leq \theta \leq \pi 
\end{cases}
\]

(3.2)

The parameters \( M_1 \) and \( M_2 \) are related to the friction coefficient \( \mu \) by
Note that the shapes of the normal and shear stress distributions at the pin-hole interface given in equation 3.2 are presumed during the solution process. The stress components $\sigma_{rr}$, $\sigma_{r\theta}$, and $\sigma_{\theta\theta}$ are then determined in the sheet and in the pin. For the rigid pin, the hoop stress component $\sigma_{\theta\theta}$ along the hole circumference of the sheet under plane stress conditions is

$$\sigma_{\theta\theta} = \frac{P}{\pi r_0} \left[ \frac{2}{\pi} \left( \frac{2}{\pi} \sin(\theta) \right) M_1 + \left( \cos(2\theta) - \frac{4}{3\pi} \sin(\theta) + \frac{2}{\pi} \sin(2\theta) \tanh^{-1}(\cos(\theta)) \right) M_2 - \left( \frac{\nu + 1}{2} \right) \sin(\theta) \right]$$

for $\pi / 2 \leq \theta < 0$

$$\sigma_{\theta\theta} = \frac{P}{\pi r_0} \left[ \frac{2}{\pi} M_1 + \left( \cos(2\theta) - \frac{4}{3\pi} \sin(\theta) + \frac{2}{\pi} \sin(2\theta) \tanh^{-1}(\cos(\theta)) \right) M_2 - \left( \frac{\nu + 1}{2} \right) \sin(\theta) \right]$$

for $0 < \theta \leq \pi / 2$

$$\sigma_{\theta\theta} = \frac{P(2M_1 + \pi M_2)}{\pi^2 r_0}$$

for $\theta = 0$

where $\nu$ is the Poisson’s ratio of the sheet material. For further details regarding this solution the reader may refer to [14].

To model an infinite sheet in ANSYS, a large finite square sheet with $w/r_0 = 100$ was created. A concentrated pin load $P$ per unit thickness was applied downward at the center of the rigid, neat-fit pin, while the top edge of the sheet was constrained in the pin load direction. A modulus of elasticity of $E = 72\text{GPa}$, and a Poisson’s ratio of $\nu = 0.3$ were used for the sheet. Figures 3.7(a,b) gives the normal and shear stress distributions along the hole circumference for $\mu = 0$ and $\mu = 0.2$. In addition, a $[\sin(\theta)]^{0.6}$ function, as suggested in [17], is included in the plots. All stress components are normalized as
\( \sigma_i(\pi r_o/P) \). As shown in Figures 3.7(a,b), differences are evident for both \( \mu = 0 \) and \( \mu = 0.2 \). It is observed from Figure 3.7(a) that the \([\sin(\theta)]^{0.6}\) better approximates the shape of the normal contact stresses rather than the function used in reference [14], which is proportional to \( \sin(\theta) \) (see \( \sigma_{rr} \) in equation 3.2). The numerical solution indicates that the edge of the downward slope corresponding to the boundary between the closed and open contact does not start at exactly \( \theta = 0^\circ \). The shear stress values are zero everywhere on the hole surface as expected for \( \mu = 0 \). Greater variations were observed in both shapes and magnitudes of the \( \sigma_{rr}, \sigma_{r\theta} \) stress curves when a friction coefficient \( \mu = 0.2 \) is considered. As shown in Figure 3.7(b), normal contact stresses found from the Ho and Chau solution are lower in magnitude for \(-50^\circ \leq \theta \leq 0^\circ \) and they are greater in the range \(-85^\circ \leq \theta \leq 50^\circ \) with a maximum percentage difference of 27\% at \( \theta = -85^\circ \). Also, the \([\sin(\theta)]^{0.6}\) function fails to capture the accurate shape of the normal stress curve when friction is included. Shear stress values are significantly smaller than the FE solutions in the entire range, with a maximum percentage difference of 175\% at \( \theta = -84^\circ \). Moreover, it is observed that the distribution of the shear stress is not exactly proportional to \( \sin(2\theta) \), which is used in [14].

Comparisons between the hoop stress solutions of reference [14] with the FE results along the hole circumference are given in Figures 3.7(c,d) for \( \mu = 0 \) and \( \mu = 0.2 \). The variations in the stress magnitudes shown in the figure can be partially explained by the different contact stress distributions used as boundary conditions by Ho and Chau when compared to that predicted by the FE model. Percentage differences between the peak stress values are 11.5\% and 18.9\% for the cases with \( \mu = 0 \) and \( \mu = 0.2 \) respectively. However, it is observed that the analytical and numerical hoop stress curves follow a
similar trend. In particular, increasing the friction coefficient raises the peak hoop stress value and lowers the stress magnitudes at the lower values of $\theta$.

Figure 3.7  Comparison of the normal, shear and tangential stress distributions along the hole circumference ($r = r_o$) with [14]

For further validation, Yavari et. al.’s recent 2D numerical analysis [19] regarding the stress distribution around the elastically pin loaded hole in a finite sheet were replicated. The geometry of the model is similar to the one considered in the present study (see Figure 3.2). Dimensions of the sheet are: $d = 15.3$ mm, $2r_o = 6.12$ mm, $h = 168.3$ mm, and $2w = 30.6$ mm. The pin was modeled to be rigid, whereas $E = 70$ GPa and
\( \nu = 0.31 \) were used for the aluminum alloy sheet. A pin load \( P \) was applied downward at the pin center keeping the top edge of the sheet constrained in the \( y \) direction.

Figure 3.8 compares the normalized radial and hoop stress solutions along the hole circumference with the results given in [19] for \( \mu = 0 \) and 0.2. Stress components are normalized as \( \sigma_j(\pi r_o/P) \). As shown in Figure 3.8(a), the radial stress distributions compare well for both friction levels. Hoop stress curves were also in good agreement, although slight variations are observed for \( \theta < 0^\circ \) (Figure 3.8(b)).

![Figure 3.8](image)

**Figure 3.8** Comparison of the normal and tangential stress distributions along the hole circumference \((r = r_o)\) with [19]

### 3.5 Results

Figure 3.9 presents the comparisons between the stress solutions from the linear elastic and elasto-plastic models with \( \mu = 0 \) and 0.2. In Figure 3.9(a,b), normalized radial stress distributions \( \sigma_r/\sigma_{ys} \) are plotted along the hole circumference for the normalized pin load magnitudes \( S_b/\sigma_{ys} = 0.2, 0.6, 1.0 \). Results, in general, do not vary significantly, although a noticeable difference in the curve shapes are observed for the applied pin load \( S_b/\sigma_{ys} = 1 \) and a friction coefficient \( \mu = 0.2 \) (Figure 9b). Figures 9(c,d) show a significant influence of the material nonlinearity on the hoop stress distributions when the applied
load approaches its maximum level. For example, the peak hoop stress values for $S_b/\sigma_{ys} = 1$ are reduced by as much as 19% and 25% in the cases with $\mu = 0$ and 0.2, respectively. Further variations are observed in the region $\theta < 0^\circ$, particularly in the frictionless case with $S_b/\sigma_{ys} = 1$, where the magnitudes of the hoop stresses from the elasto-plastic models are reduced by a significant amount. The stress solutions for the smaller applied loadings are in good agreement as expected since there is little plastic deformation.

Figure 3.9  Hoop stress distributions along the hole circumference from elastic and elasto-plastic models

The influence of friction can be studied by comparing the radial stress plots (a) with (b) and the hoop stress plots (c) with (d) in Figure 3.9. The radial stress values are
lower in the case with friction, which is more obvious for higher applied loadings (see Figures 3.9(a,b)). For example, when the pin load is $S_b/\sigma_{ys} = 1$, the maximum percentage difference between $\mu = 0$ and 0.2 is about 22.5% at $\theta = -85^\circ$. It is observed that the friction raises the peak hoop stresses for the elastic material models particularly when the applied loading is high (compare the open symbols in Figures 3.9(c,d)). The percentage difference between the elastic peak hoop stress magnitudes with $\mu = 0$ and 0.2 is approximately 13% when $S_b/\sigma_{ys} = 1$. Furthermore, the elastic hoop stress solutions at the lower values of $\theta$ are larger with a maximum percentage difference of nearly 120% at $\theta = -85^\circ$ when $S_b/\sigma_{ys} = 1$. Note that similar observations were reported regarding friction in previous FE studies with the linear elastic material models in [18, 19]. However, the influence of friction is not the same when the material nonlinearity is included. For the high applied loadings, the elastic-plastic hoop stress values reach the same highest levels which are approximately unity for both cases of friction (compare the closed symbols in Figures 3.9(c,d)). This, of course, is a consequence of local material yielding. An additional difference with the elastic model solutions is that friction increased the hoop stress results at the lower values of $\theta$ when $S_b/\sigma_{ys} = 1$.

Of concern is the location of the maximum hoop stress, which is a susceptible region for crack initiation. For the elastic case, angles corresponding to the peak hoop stresses at the hole surface are approximately $\theta_{\text{max}} = -4.6^\circ$ and $-1.5^\circ$ for $\mu = 0$ and 0.2, respectively. These values are approximately $\theta_{\text{max}} = -3.6^\circ$ and $-1.0^\circ$ for the elasto-plastic frictionless case and the case with $\mu = 0.2$. However, when $S_b/\sigma_{ys} = 1$, the hoop stress values in the vicinity of these regions are approximately the same when material nonlinearity is included, indicating that the cracks may initiate anywhere around $\theta = 0^\circ$ (Figures 3.9(c,d)).
Graphical results of the pin-loaded hole with an initial compressive residual stress field created by the cold expansion simulation are presented in Figures 3.10-3.12. In Figure 3.10, the normalized radial stress distributions $\sigma_{rr}/\sigma_{ys}$ are plotted along the circumference of the cold expanded and pin-loaded hole with friction coefficients $\mu = 0$ and 0.2. The radial stress solutions from kinematic and isotropic hardening material models were nearly identical. The stress and the influence of friction are very similar to that obtained from the elastic pin loading analyses (see Figures 3.9(a,b)). For example, including friction reduces the radial stress magnitude by as much as 29% around $\theta = 85^\circ$, when the applied loading is $S_b/\sigma_{ys} = 1.0$.

Hoop stress distributions of the cold expanded, pin loaded hole for the isotropic and kinematic hardening material models are given in Figure 3.11 for $\mu = 0$ and 0.2. Results indicate that the hoop stress values are higher throughout the pin-loading process when the kinematic hardening material model is used. That is because the compressive
hoop stress magnitude produced from cold expansion simulation is lower. Lower residual stress values in kinematic hardening model result from Bauschinger’s effect, in which the material more readily yields under reverse loading. It is observed that including friction raises the maximum hoop stress values in both hardening models and for all applied load levels up to $S_b/\sigma_{ys} = 1$. The locations of the maximum hoop stresses at the hole surface are approximately $\theta_{max} = -3.7^\circ$ and $-1.3^\circ$ for $\mu = 0$ and 0.2 respectively. These values are the same for both hardening models as shown in Figures 3.11(a,b).

Figure 3.11  Hoop stress distributions along the circumference of the cold expanded and pin loaded holes
Figure 3.12  Hoop stress distributions along the line $y = 0$ around the cold expanded and pin loaded holes

Figure 3.13  Hoop stress contour plots after cold-expansion and pin loading for $\mu = 0$

The hoop stress distributions along the $x$-axis, as a function of the normalized distance $x/r_0$, are given in Figure 3.12. This figure demonstrates the variation of the stress...
field near the hole of the sheet material as the pin load is applied. At the highest load level (i.e. when $S_b/\sigma_{ys} = 1.0$), the hoop stress values at the hole edge (i.e. at $x/r_o = 1$) are $\sigma_{\theta\theta}/\sigma_{ys} = 0.23$ and $0.37$ for $\mu = 0$ and $0.2$ for the kinematic hardening model. However, it is observed that the location of the maximum hoop stress is not at the hole surface, but it is in the interior region of the sheet material. For example, the stress values near the region $x/r_o = 1.8$ are $\sigma_{\theta\theta}/\sigma_{ys} = 0.45$ and $0.48$ for $\mu = 0$ and $0.2$, which are significantly higher than those given at the hole edge (see Figure 3.12). This observation can also be made by inspecting the hoop stress contour plots around the cold-worked and subsequently pin loaded hole to different levels as shown in Figure 3.13. When the normalized applied load is $S_b/\sigma_{ys} \geq 0.8$, the formation of the maximum hoop stress region away from the hole edge can be easily detected. This indicates that the cracks near the cold worked hole may in fact initiate at the locations away from the hole surface. It must also be noted that after the applied pin-loading of $S_b/\sigma_{ys} = 1.0$ is removed, the residual stress state returns to its original level with a negligible residual stress relaxation during the load cycle.

Finally, in Figure 3.14, the stress distributions along the line $y = 0$ for an isotropic hardening material (given in Figure 3.12) are compared with the solutions obtained by superposing the elastic stress distribution due to the pin loading with the pre-determined residual stress curve. Results from both approaches compare well. Thus, once the residual stress field around the hole is known, the subsequent stress state for applied pin loadings $S_b/Y \leq 1.0$ on line $y = 0$ can readily be computed by superposition.
3.6 Conclusion

A detailed 2D FEA of a pin-loaded hole in a rectangular aluminum alloy 7075-T6 sheet was performed. Plane stress conditions were assumed. The material of the sheet was modeled to be elastic and elastic-plastic with multilinear isotropic work hardening. A kinematic hardening material model was also considered during the cold hole expansion simulation to study Bauschinger’s effect on the final stress results. The pin was assumed as rigid for simplicity. Convergence studies and comparison with other work were done to validate the FE model.

In the first phase of this work, the residual stress free hole was pin-loaded, and the influence of the material nonlinearity and friction on the stress distributions along the hole circumference was studied. Major variations in the hoop stress solutions from elastic and elastic-plastic models were observed in the region $\theta < 0^\circ$. Also, the peak hoop stress values were reduced by a significant amount when material non-linearity was considered. Friction increased the peak hoop stress values for the elastically loaded holes and for small applied loadings with the elastic-plastic material model. However, as the applied
load was increased to the maximum level in elastic-plastic model, the magnitudes of $\sigma_{\text{max}}$ approached unity for both friction cases.

The angles corresponding to the elastic peak hoop stresses were approximately $\theta_{\text{max}} = 4.6^\circ$ and $1.5^\circ$ for $\mu = 0$ and 0.2, respectively. These values were nearly $\theta_{\text{max}} = 3.6^\circ$ and $1.0^\circ$ in the elasto-plastic frictionless case and for the case with $\mu = 0.2$. The hoop stress values in the vicinity of these regions were approximately the same in the plastically loaded hole, indicating that the cracks may initiate anywhere around $\theta = 0^\circ$.

In the next stage, a compressive residual stress field was introduced around the hole prior to the subsequent pin-loading analysis. The residual stress field was produced by a simplified cold expansion simulation, where the hole of the sheet was uniformly expanded beyond the elastic limit of the material. Radial stress distributions and the influence of friction were very similar to the ones obtained from previous elastic pin loading analysis. Due to Bauschinger’s effect, kinematic hardening produced compressive residual stresses with lower magnitudes around the hole. This resulted in higher hoop stress values along the hole circumference when using kinematic hardening throughout the subsequent pin-loading simulations. The locations of the peak hoop stresses along the hole circumference were same for both hardening material models with $\theta_{\text{max}} = -3.7^\circ$ and $-1.3^\circ$ for $\mu = 0$ and 0.2 respectively. However, the locations of the absolute maximum hoop stresses for the cold-worked, pin-loaded holes were observed to be in the regions away from the hole surface. As the applied pin load was removed, the hole returned to its original residual stress state indicating the occurrence of negligible residual stress relaxation during this load cycle.
3.7 References


4.1 Introduction

Fastener joints are widely used in aircraft and rotorcraft industries. These joints are also known to be susceptible locations to fatigue failure due to the localized stress concentrations caused by the presence of the holes. To overcome this problem, a split-sleeve cold expansion process [1] has emerged as a life enhancement method to mitigate the effect of the stress concentrations by creating a compressive circumferential residual stress field around the hole. In this process, the hole is radially expanded using a tapered mandrel and a split sleeve. The diameter of the mandrel with the sleeve is greater than that of the hole. When the mandrel, pre-fitted in an internally lubricated sleeve, is pulled through the hole, the hole is plastically expanded. As the mandrel is removed, the hole undergoes a partial but not full elastic recovery, creating the desired compressive circumferential residual stresses.

Many analytical and numerical studies were previously conducted and published regarding the prediction of the residual stresses created by cold expansion. Rich and Impellizzeri [2] and Ball [3] developed two-dimensional (2D) elastic-plastic closed form solutions for the uniform radial expansion of fastener holes assuming small displacement, plane strain [2], and plane stress [3] conditions. In Refs. [4–8], 2D, 2D-axisymmetric, and 3D numerical solutions were obtained using a simplified approach, in which the hole was
uniformly expanded to simulate the interference between the mandrel and the hole. The assumption of a uniform expansion inherently limits the accuracy of the predicted residual stress.

Other studies have considered the mandrel insertion to model the process in a more physically realistic way. This is because the hole surface is expanded sequentially (rather than uniformly) in the actual process, starting from the mandrel entrance side toward the exit side of the hole. For sufficiently thick plates this may result in a significant variation in the residual stress magnitudes along the plate thickness. Garcia-Grenada et al. [9] conducted a 2D-axisymmetric analysis, whereas Papinkos and Meguid [10] and Chakherlou and Vogwell [11, 12] performed 3D simulations of cold hole expansion with the mandrel. More recent researchers [13, 14] have also included the elastic sleeve in their 3D models and simulated the process by pulling the tapered rigid mandrel through the sleeve pre-fitted in the hole.

The beneficial influence of cold-working process is associated with reduction in the applied mode I stress intensity factors $K_I$ due to the compressive residual stress field created around the hole. The reduced $K_I$ retards fatigue crack growth improving the fatigue resistance of the structural component. Moreira et. al [15] used the $J$-integral and the weight function methods to compute the stress intensity factors due to different levels of cold work with no applied loading. They considered a case of a 2D infinite plate having a hole with two symmetrical cracks. Pinho et. al [16] also studied a case of two symmetrical cracks in 2D finite rectangular sheets that are loaded either remotely in tension, or by a rigid pin. Both of the studies above used elastic perfectly plastic material models.
The objective of current research is to build upon these past two efforts to quantify the beneficial influence of cold working process using 2D FE model, with multilinear isotropic hardening material behavior. A single crack emanating from cold worked pin loaded and open holes loaded in tension is considered. The simulation of cold working process is performed using a simplified approach in 2D by uniform hole expansion with different interference levels. Next, the cold-worked hole is subsequently loaded and the stress intensity factors are computed using (a) $J$-integral, and (b) weight function methods. Obtained results for several cold-working levels are then compared with those of the non-cold-worked hole.

4.2 Finite Element Model and Boundary Conditions

A crack configuration considered for the stress intensity factor computation is depicted in Figure 4.1. It is a 2D rectangular sheet with a single horizontal crack emanating from the cold-worked hole. The size of the thin rectangular sheet is 65.0 mm ($h$) by 44.45 mm ($2w$) with a 7.07 mm diameter hole ($2r$) located at 22.23 mm distance ($d$) from the bottom edge. Two different loading cases were considered: (a) open hole in tension, (b) pin-loaded hole. In case (a) the sheet is loaded in tension with a normal traction applied on the bottom edge, while the sheet top edge is constrained in the vertical direction. For the case (b) the hole is loaded purely by a neat-fit pin downward with a magnitude $P$. The sheet is constrained in the $y$ direction at the top edge (Figure 4.2). The material properties given in section 3.2 were used for AA7076-T6 aluminum alloy sheet. The pin is assumed to be a rigid body, and no friction is considered between the pin and the hole.
4.3 Methodology

Two different approaches were used to obtain mode I stress intensity factor solutions ($K_I$) for the loaded hole configurations, which are: (a) $J$ integral approach, (b) weight function technique. The range of the normalized crack lengths is from $(a+r)/w = 0.175$ to $0.75$.

![Sheet geometry: $h = 65.0$ mm, $2w = 44.45$ mm, $2r = 7.07$ mm, $d = 22.23$ mm](image)

**Figure 4.1** Sheet geometry: $h = 65.0$ mm, $2w = 44.45$ mm, $2r = 7.07$ mm, $d = 22.23$ mm

![Case (b) loading](image)

**Figure 4.2** Case (b) loading
First, a FE mesh was created with a multilinear elastic plastic material definition for AA7075-T6 with an isotropic hardening behavior. The model contained highly refined 8 node quadrilateral plane stress elements near the crack tip regions as shown in Figure 4.3. A residual stress field was obtained by uniformly expanding the hole of the sheet beyond the elastic limit of the material as explained in section 3.3. Radial interferences considered during cold expansion simulation are

$$CW = \frac{\Delta r}{r} \% = 2\%,\ 3\%,\ and\ 4\%$$

(4.1)

where, $\Delta r$ is the radial displacements applied at the hole edge.

Figure 4.3 Sample FE mesh for $(a+r)/w = 0.25$
Next, the same meshed model was duplicated with linear elastic material properties. The residual stress field created using the elastic-plastic model was transferred to elastic model via an ANSYS command INISTATE. The sheet was then loaded in tension, and $J$ integral values were computed for ten different contours around the crack tip. The $K_I$ solutions were obtained from the average $J$ integral, $J_{ave}$, using the relationship for linear elastic materials

$$K_I = \sqrt{J_{ave} \cdot E}$$

(4.2)

The stress intensity factors for the given crack configuration were also computed using the weight function along with the superposition principle. From [17], the weight function for the configuration of a single crack from a hole in an infinite sheet can be expressed as

$$m(\alpha, \xi) = \frac{1}{\sqrt{2\pi a}} \sum_{i=1}^{3} \beta_i(a) \left(1 - \frac{\xi}{\alpha}\right)^{\frac{3}{2}}$$

(4.3)

provided that the crack length satisfies the condition $a/r \leq 2$. In equation 4.3, $\xi$ and $\alpha$ are non-dimensional quantities $\alpha = a/r$ and $\xi = x/r$ (see Figure 4.4). Functions $\beta_i$ are given by

![Figure 4.4](image)

**Figure 4.4** Transformed quantities for a single crack from a hole in an infinite plate
\[
\beta_1(\alpha) = 2.0
\]
\[
\beta_2(\alpha) = \left[4\alpha f'(\alpha) + 2f_r(\alpha) + \frac{3}{2}F(\alpha)\right]/f_r(\alpha) \tag{4.4}
\]
\[
\beta_3(\alpha) = \left[\alpha F''(\alpha) - \frac{1}{2}F(\alpha)\right]/f_r(\alpha)
\]

where

\[
F(\alpha) = \left[\sqrt{2}\pi\phi(\alpha) - 4E_1(\alpha)f_r(\alpha)\right]/E_2(\alpha) \tag{4.5}
\]

The related functions \(\phi(\alpha), E_1(\alpha)\) and \(f_r(\alpha)\) in equations 4.4 and 4.5 are given by

\[
\phi(\alpha) = \frac{1}{\alpha^2} \int_0^\infty s\left[f(\alpha)\right]^2 ds
\]

\[
E_1(\alpha) = \sum_{m=0}^{\infty} \left\{ \frac{2^{m+1}m!S_m\alpha^m}{\prod_{k=0}^{m}(1+2j+2k)} \right\}
\]

\[
f_r(\alpha) = 2.2421 - 2.8069\alpha + 4.1784\alpha^2 - 4.3940\alpha^3 + 2.9623\alpha^4 -
-1.1899\alpha^5 + 0.2565\alpha^6 - 0.0227\alpha^7
\]

and the coefficients \(S_m\) are

\(S_m\): 1.9991, -1.9543, 2.5773, -2.3769, 1.4367, -0.5393, 0.1066, -0.0089.

To obtain the \(K_I\) solutions via the weight function technique, the nodal stress distributions along the crack plane were first extracted from an uncracked loaded hole model in ANSYS. Then, the portion of the stress intensity factor due to the linear stress distribution over one element was computed as follows

\[
(\Delta K_I)_i = \int_{\xi_{i-1}}^{\xi_i} \sigma_i(\xi) \cdot m(\alpha, \xi) d\xi \tag{4.7}
\]

where, \(\sigma_i(\xi)\) is the linear stress distribution along the \(i\)th element (see Figure 4.5), and \(m(\alpha, \xi)\) is the weight function given in equation 4.3.

The summation of the \(\Delta K_I\) for each element on the crack face gives the crack tip stress intensity factor \(K_I\) due to the corresponding applied pin load
A correction factor can be incorporated into the \([K_I(a)]_{\text{inf}}\) solution to account the effect of the finite width of the sheet as follows [18-19]

\[
K_I(a) = f(a, w)[K_I(a)]_{\text{inf}}
\]

\[
f(a, w) = \sqrt{\frac{1}{\cos \left( \frac{\pi(2r+a)}{2(2w-a)} \right)} \cdot \sin \left( \frac{\pi(2r+a)}{2(2w-a)} \cdot \frac{a}{w} \right)}
\]

The expression in equation 4.7 was integrated numerically using the Gauss-Chebyshev quadrature, which accounts for the crack tip singularity associated with the weight functions.

Using the given weight function the stress intensity factors due to the residual stress field alone \(K_{\text{res}}\) were computed from the residual stress distribution along the crack edge. Next, the same weight function was employed to compute the applied \(K_{\text{app}}\) from the stress distribution along the crack line due to applied remote loading. Finally, these two solutions were superposed to give resulting \(K_I\) values for the cold worked open and pin-loaded holes.
Finally, the geometry factor $F$ for the different crack lengths were computed by normalizing the $K_I$

$$F = \frac{K_I(a)}{S\sqrt{\pi a}}$$

(4.11)

where, $S$ is the remotely applied stress, which is $S = 300\text{MPa}$ in this study.

Figure 4.6  Geometry factors for a single crack from an open hole in a finite sheet loaded in tension

4.4  Results

Figure 4.7 presents the geometry factor solutions computed using the two different approaches described above for the cracks in an open hole with lengths from $(a+r)/w = 0.175$ to 0.75. Symbols represent the $J$-integral method results, whereas the lines are the weight function solutions. $F$ values for the residual stress free hole are also included in the figure for comparison. Note that the results from the weight function
technique are given only for the range $0.175 \leq \frac{(a+r)}{w} \leq 0.45$, because of the maximum crack length limitation $a/r \leq 2$ for the weight function used in this analysis. Solutions from both the $J$-integral method and the weight function technique compare well. It is observed that cold working reduces geometry factors by a significant amount when the crack length is small. The higher cold working amount produced greater influence because of the larger compressive residual stress field generated. This significant impact however diminishes as the normalized crack length $(a+r)/w$ approaches 0.45, after which no influence of cold working is observed.

Figure 4.7  Geometry factors for a single crack from a pin-loaded hole in a finite sheet

Figure 4.7 presents the geometry factors $F$ computed using the two different approaches for the cracks in a pin-loaded hole with lengths from $(a+r)/w = 0.175$ to 0.75. The symbols represent the FE model results and the lines are the weight function
solutions. $F$ values for the pin-loaded hole with no residual stress are also included in Figure 4.7 for comparison. The solutions from both approaches have a good correlation. The beneficial influence of the cold working process for the pin-loaded hole is evident for the range of normalized crack length $(a+r)/w < 0.40$. The higher interference amount results in greater reduction in $F$. Similar to the open hole case, this influence diminishes as the normalized crack length $(a+r)/w$ approaches 0.40, after which no influence of cold working is detected.
4.5 References


CHAPTER V

STUDY OF AN ON-LINE CRACK COMPLIANCE TECHNIQUE FOR RESIDUAL STRESS MEASUREMENT USING 2D FINITE ELEMENT SIMULATIONS OF FATIGUE CRACK GROWTH

5.1 Introduction

Residual stresses are those which remain in a body without any external load. They may be introduced to structural components during manufacturing processes such as forging, casting, welding, machining, or from heat treatments such as quenching. Several life enhancement processes have also been developed to induce compressive residual stresses. Compressive residual stresses are beneficial to fatigue life under low amplitude, high frequency loadings, since they retard crack initiation and propagation. Residual stresses, regardless of the manner of their introduction, are generally produced by non-uniform plastic deformation caused by mechanical or thermal loads.

Apart from macro-stresses discussed above, grain scale (intergranular) and atomic scale stresses exist within the material. Low level intergranular micro-stresses are nearly always present in polycrystalline structures because of the variations in the elastic and thermal properties of differently oriented neighboring grains. Higher intergranular stresses exist when the microstructure contains multiple phases. Atomic stresses, on the other hand, originate from dislocations and coherency at interfaces [1]. Except for understanding the microcrack growth behaviors, the grain scale and atomic micro-stresses are often ignored in crack growth life assessment analysis in a metallic
component [2]. This is because micro-stresses must balance out over the very small distance. The current study will use the term “residual stresses” to refer to macro-stresses.

The negative influence of the residual stresses on fatigue life is usually accounted for by a factor of safety, whereas the positive effects are generally not considered during the design process. Understanding the residual stresses present in a component is important to better quantify their beneficial or detrimental impacts. Numerous experimental and numerical methods have been developed to measure the residual stress field. Experimental measurement methods are typically subdivided into three groups: (a) non-destructive, (b) semi-destructive and (b) destructive.

In non-destructive methods, a workpiece remains physically unaltered, and the stress field is obtained from the relationship between the physical or crystallographic parameters and the residual stress [3]. Diffraction methods that use X-Ray, electron, or neutron beams are considered as non-destructive if the stresses are to be measured near the external surfaces. Semi-destructive methods do not substantially destroy the specimen and the damage is very localized. A hole drilling method is an example for this category. In this method, strain gages are attached to the surface, and a hole is drilled in a nearby location. Relieved strains are detected by strain gages, which are then related to residual stresses [4].

Destructive measurement methods require the material to be destroyed while the stresses are measured. Examples for destructive methods include a crack compliance method, not to be confused with an on-line crack compliance method. In the crack compliance method, a part is incrementally cut along the plane where the residual stresses are to be measured and changes in strain at a suitable location are recorded. By treating
this cut as a mathematically sharp crack, a linear elastic fracture mechanics (LEFM) approach is employed to find the relation between the incremental change in strain with respect to the crack length \( \frac{d\varepsilon}{da} \) and the stress intensity factor \( K_{irs} \) due to the residual stress

\[
K_{irs} = \frac{E'}{Z(a)} \frac{d\varepsilon}{da}
\]  
(5.1)

where \( E' \) is the generalized Young's modulus and \( Z(a) \) is an influence function [5].

Calculated \( K_{irs} \) values for different crack lengths \( a \) can then be converted to residual stresses via inverse solution methods such as incremental stress [6], series expansion [7], or pulse method [8]. For further details on the crack compliance method, the reader may refer to [5, 9, 10].

Recently, Lados [11, 12] introduced an on-line crack compliance technique, which is used to determine \( K_{irs} \) from the crack opening displacement measurements “on-line”, that is during an actual fatigue crack growth test. Hence, this method generates additional data regarding the residual stress field as a by-product after the crack growth test is carried out. This method is based on the crack compliance method and is derived from LEFM as discussed further in the next section.

With the advance in computational technology, the finite element method (FEM) has become a valuable tool to determine the residual stress fields by making it possible to simulate a wide range of life enhancement and manufacturing processes numerically (see for example [13, 14, 15]). The FEM can also be used to study the existing experimental methods of stress measurement. Prime [16] introduced residual stresses in a finite element (FE) model of a compact tension (CT) specimen by overloading the model beyond the elastic limit of the material. He then simulated the crack compliance method.
by incrementally removing the elements along the crack plane and letting the model elastically unload. He measured the strains on the back face of the model after each incremental cut. Resulting stress fields from the strain measurements compared well with the residual stress distributions produced in the FE model after the overloading event. De Swardt [17] also employed FEM to simulate the crack compliance technique on autofrettaged thick-walled high-strength steel cylinders. He progressively extended the cut in his model by modifying the nodal constraints along the line of the cut and recorded the strains on the outside wall. De Swardt compared the computed strains with the experimental strain data from the crack compliance method, and concluded that using an elastic-plastic material model incorporating the Bauschinger effect produced the best results.

In this study, the on-line crack compliance method was simulated using a FE model of a rectangular sheet with a central hole under plane stress conditions. The sheet material was an AA7075-T6 aluminum alloy. The analysis was completed in two stages. In the first stage, a residual stress field was introduced around the hole by a cold hole expansion simulation. In the second stage, crack growth simulation was performed by applying remote cyclic loads and incrementally propagating the crack in each cycle. The crack growth stage was conducted with two material behaviors: (a) purely elastic and (b) elastic-plastic. This was done to better understand the performance of the on-line crack compliance technique under more realistic conditions with plastic deformations present around the crack tip. The stress intensity factors due to the residual stress field normalized by the maximum applied stress intensity factors, $\frac{K_{irs}}{K_{imax}}$, were obtained using the on-line crack compliance method. As part of the validation process, the elastic crack growth solutions of $\frac{K_{irs}}{K_{imax}}$ were compared with the results obtained from $J$-
integral values. Convergence studies were performed to validate the elastic-plastic solutions. Finally, the influence of the crack tip plasticity is presented by comparing the results from elastic and elastic-plastic crack growth simulations for different cold working levels and applied loadings.

5.2 Methodology

5.2.1 On-line Crack Compliance Method

Figure 5.1 illustrates a close-up view of a crack with a length \( a \) advancing by an amount \( da \). A newly extended crack face is depicted with a dashed line in the figure. The crack mouth opening displacement (CMOD) of the crack \( a \) under a remote load \( P \) is given by \( \delta \). As the crack length is grown to \( a+da \), \( \delta \) will increase by an increment \( d\delta \) under the same applied load \( P \). For linear elastic materials under plane stress conditions, the Mode I stress intensity factor \( K_I \) can be expressed as (see Appendix A for derivation)

\[
K_I = \frac{E}{Z(a)} \frac{d\delta}{da}
\]  

(5.2)

Figure 5.1 Schematic illustration of crack extension
where $E$ is the Young's modulus of the material and $Z(a)$ is the influence function that depends on the geometry of the specimen as well as the location of the displacement measurement. Note that equation 5.2 is very similar to equation 5.1 of the crack compliance method, except displacements are used instead of strains. Note also that the crack opening displacement $\delta$ can be measured at any fixed point along the crack face, since the influence function $Z(a)$ changes accordingly to give the same value for $K_I$. In the present study, the CMOD is used for $\delta$.

Figure 5.2 depicts the load-displacement curves as the remote load is increased from zero to $P_{\text{max}}$ prior to and after the crack has been grown by $da$. Figure 5.2(a) is for the specimen without any residual stress field, whereas Figure 5.2(b) is for the specimen with the compressive residual stress field present in the crack growth region. In the absence of any residual stress field, the load-displacement curves are linear, and the crack starts to open at the onset of load application. The maximum applied stress intensity factor $K_{I\text{max}}$ will be

$$K_{I\text{max}} = \frac{E}{Z(a)} \frac{d\delta_{\text{max}}}{da}$$  \hspace{1cm} (5.3)
where \( d\delta_{\text{max}} \) is the incremental CMOD at the maximum load \( P_{\text{max}} \) shown in Figure 5.2(a). With a compressive residual stress field in the growth region, the crack mouth does not open until the applied load reaches a certain level. This corresponds to a vertical segment in Figure 5.2(b). As the applied load is further increased, the crack will start opening at the same rate as in the case with no residual stress field. Thus, the respective slopes of the inclined segments \( a \) and \( a+da \) in Figure 5.2(b) are the same as those of the lines \( a \) and \( a+da \) in Figure 5.2(a). However, the incremental opening displacement \( d\delta_{r\text{max}} \) shown in Figure 5.2(b) is smaller because of the presence of compressive residual stress field. This results in a lower maximum stress intensity factor, \( K_{I_{\text{r}}} \)

\[
K_{I_{\text{r}}} = \frac{E}{Z(a)} \frac{d\delta_{\text{rmax}}}{da}
\]  

(5.4)

Superposition can be employed to determine the stress intensity factor \( K_{I_{\text{r}}} \) due to the residual stress field alone from the Equations 5.3 and 5.4

\[
K_{I_{\text{r}}} = -(K_{I_{\text{max}}} - K_{I_{\text{r}}}) = \frac{E}{Z(a)} \left( \frac{d\delta_{r\text{max}} - d\delta_{\text{max}}}{da} \right)
\]  

(5.5)

The negative sign in equation 5.5 indicates the fact that the \( K_{I_{\text{r}}} \) is due to the compressive residual stresses. Normalizing \( K_{I_{\text{r}}} \) by \( K_{I_{\text{max}}} \) will result in a simple expression

\[
\frac{K_{I_{\text{r}}}}{K_{I_{\text{max}}}} = \frac{d\delta_{r\text{max}} - d\delta_{\text{max}}}{d\delta_{\text{rmax}}} = \frac{d\delta_{r\text{max}}}{d\delta_{\text{max}}} - 1
\]  

(5.6)

Hence, \( K_{I_{\text{r}}}/K_{I_{\text{max}}} \) is readily determined from the ratio of the incremental changes in the displacements \( d\delta_{r\text{max}} \) and \( d\delta_{\text{max}} \) shown in Figure 5.2. The efforts reported here involves a study of this non-dimensional parameter to investigate the influence of crack tip plastic deformation on the performance of the on-line crack compliance method.
5.2.2 J-Integral Method

The ratio \( K_{irs} / K_{I_{\text{max}}} \) in equation 5.6 can also be computed using \( J \)-integral values computed near the crack tip. For linear elastic materials under Mode I loading, \( K_I \) can be obtained from \( J \) using the relation

\[
K_I = \sqrt{JE}
\]  
(5.7)

where \( E \) is the Young's modulus of the material [18]. If \( J_{rs \text{max}} \) and \( J_{\text{max}} \) are the \( J \)-integrals computed at the maximum applied load \( P_{\text{max}} \) with and without the presence of the compressive residual stress field, respectively, the ratio \( K_{irs} / K_{I_{\text{max}}} \) can be expressed as

\[
\frac{K_{irs}}{K_{I_{\text{max}}}} = \frac{\left( \sqrt{J_{\text{max}} E} - \sqrt{J_{rs \text{max}} E} \right)}{\sqrt{J_{\text{max}} E}} = \frac{\sqrt{J_{rs \text{max}}} - \sqrt{J_{\text{max}}}}{\sqrt{J_{\text{max}}}} = \sqrt{\frac{J_{rs \text{max}}}{J_{\text{max}}}} - 1
\]  
(5.8)

This approach may be used to validate the numerical solutions of the on-line compliance technique when a linear elastic material model is used for crack growth.

5.2.3 Finite Element Model

Figure 5.3 shows the geometry and dimensions of a rectangular sheet with a circular hole at the center. A single crack perpendicular to the applied load emanates from the hole edge. The size of the sheet is 130 mm by 44.45 mm (\( h \times 2w \)) and the radius of the hole is \( r = 3.535 \) mm. A plane stress condition is assumed valid. The crack grows from an initial length \( a = 0.345 \) mm by a total amount \( da = 3.57 \) mm. The remote stress \( S \) is applied in a cyclic fashion from zero to \( S_{\text{max}} \) and back to zero in each cycle, with maximum applied loadings of \( S_{\text{max}} = 0.3 \sigma_{ys} \) and \( 0.4 \sigma_{ys} \), where \( \sigma_{ys} \) is the material yield strength.
Figure 5.3  Sheet geometry with dimensions: $h = 130$ mm, $2w = 44.45$ mm, $r = 3.535$ mm, $a = 0.345$ mm

Figure 5.4  Stress-strain curve for AA7075-T6 Aluminum Alloy [20]

A multilinear stress-strain curve used for the sheet material (AA7075-T6) is given in Figure 5.4 [20]. The Young’s modulus and the Poisson’s ratio are $E = 72.5$ GPa and $\nu = 0.3$, respectively. The yield strength of the material is $\sigma_{ys} = 483$ MPa. For the elastic
crack growth simulation, only the elastic domain of the curve is used. Thus, $E$ and $\nu$ are sufficient parameters to describe the material constitutive model. For cold working and elastic-plastic crack growth simulations, the stress-strain data in both elastic and plastic domains are used with the Von-Mises yield criterion and isotropic hardening behavior.

Figure 5.5  Finite element mesh

ANSYS 12.0 was used to conduct the FE simulations. The entire analysis consists of two major stages: (i) cold hole expansion simulation, (ii) crack growth simulation. In the first stage, a FE mesh of the model was created with an elastic-plastic material model. Figure 5.5 shows a typical FE mesh, which consists of about 13.5K nodes and 13K 4 node quadrilateral plane stress elements. Only the top half of the sheet is modeled using symmetry boundary conditions along the crack line. A compressive residual stress field is obtained by uniformly expanding the hole beyond the elastic limit of the material and
allowing it to elastically unload. For further details on the cold expansion simulation in 2D, the reader may refer to [19]. The radial interferences considered here are $\Delta r = 0.8\%$, 1.2% and 1.6%, where $\Delta r$ is the amount of the radial hole expansion.

Two different cases were considered with regard to crack growth through the residual stress field:

1. **Elastic crack growth:** the residual stress field obtained in stage 1 was transferred to another identical FE mesh but with a linear elastic material constitutive model. The crack growth simulation was performed using the new FE mesh.

2. **Elastic-plastic crack growth:** the model from stage 1 was used to continue with the crack growth simulation.

An initial crack is inserted by removing the displacement constraints and using a rigid, frictionless contact surface along the crack line to prevent crack face overlapping. Highly refined elements of equal length occupy the crack growth region as shown in Figure 5.5. At the minimum point of each load cycle, the crack tip is extended by one element length. The crack mouth opening displacements $\delta$ at the maximum load of each cycle is recorded from the cold-worked and non-cold-worked models to calculate the normalized stress intensity factor due to compressive residual stress field $K_{Irs}/K_{I_{\text{max}}}$ using equation 5.6. $J$-integrals were also computed for different crack lengths to obtain $K_{Irs}/K_{I_{\text{max}}}$ from equation 5.8.
5.3 Results and Discussion

Figure 5.6 presents hoop residual stress fields around the hole after uniform hole-expansion simulation with different radial interferences. Normalized stress results ($\sigma_{\theta\theta}/\sigma_{ys}$) are plotted versus the normalized distance from the hole center $x/r$ in the figure. It is observed that the greater radial interference increases both the magnitude and the depth of the resulting residual stress field.

An example of normalized stress intensity factors due to the residual stress field from the on-line crack compliance (equation 5.6) and the $J$-integral (equation 5.8) methods are shown in Figure 5.7 for the case of elastic crack growth. The $K_{irs}/K_{I_{max}}$ values are plotted versus the normalized crack length $(a+r)/w$ in the figure. The hole is cold worked with 1.2% radial interference, and the applied maximum load is $S_{max}/\sigma_{ys} = 0.4$. It is observed that the two methodologies produce nearly identical solutions. Thus,
the on-line crack compliance technique and the more traditional $J$-integral methods are equivalent when the crack is grown under purely elastic conditions. Similar observations were also made for other applied loadings and cold working levels, but their results are omitted here for brevity. Note that no additional residual stress field is introduced to the numerical model during elastic crack growth. Therefore, the information obtained from the on-line crack compliance and the $J$-integral methods pertain to the original residual stresses produced from the cold working simulation. This may no longer be true when material plasticity is included during crack growth, since the plastic deformation occurring near the crack tip may alter the existing residual stress field.

Figure 5.7  Comparison of $K_{irs}/K_{I_{max}}$ solutions from crack compliance technique and $J$-Integral method for elastic crack growth
Figure 5.8  Convergence study for elastic-plastic crack growth results

Figure 5.8 presents an example of the convergence study performed for elastic-plastic crack growth with 1.2% cold work and $S_{\text{max}}/\sigma_{ys} = 0.4$ applied load. The $K_{Irs}/K_{I_{\text{max}}}$ values computed using the crack compliance method (equation 5.6) are plotted versus the normalized crack length $(a+r)/w$. Three levels of mesh refinement were made with the element lengths $da = 0.12$ mm, 0.06 mm and 0.03 mm along the crack growth line. Solutions did not change significantly with the level of mesh refinement, although some noisy behavior was observed when smaller elements were used. The convergence behavior for other cold working levels and applied loadings considered were similar. Solutions of the on-line crack compliance technique presented next are obtained from the models with the element size $da = 0.06$ mm in the crack growth region.
Figure 5.9  Comparison of $K_{irs}/K_{i,max}$ solutions from elastic and elastic-plastic crack growth simulations with 0.8% cold work and $S_{max}/\sigma_{ys} = 0.3$.
Figure 5.10  Comparison of $K_{I_{rs}}/K_{I_{max}}$ solutions from elastic and elastic-plastic crack growth simulations with 1.2% cold work and $S_{max}/\sigma_{ys} = 0.4$

Figure 5.9 compares the $K_{I_{rs}}/K_{I_{max}}$ results from the elastic and elastic-plastic crack growth simulations with 0.8% cold work and $S_{max}/\sigma_{ys} = 0.3$ applied load. It is observed that the $K_{I_{rs}}/K_{I_{max}}$ values from both simulations initially exhibit the same level. As the crack is further grown, the two solutions slightly bifurcate, and they merge back towards the end of the simulation. Within the bifurcation zone the elastic-plastic model generated lower $K_{I_{rs}}/K_{I_{max}}$ magnitudes with a maximum difference of about $\Delta K_{I_{rs}}/K_{I_{max}} = 9\%$ near the crack length $(a+r)/r=0.19$.

Figure 5.10 presents the results from the elastic and elastic-plastic crack growth simulations with 1.2% cold work and $S_{max}/\sigma_{ys} = 0.4$ applied load. Higher magnitudes and greater depth of the compressive residual stress field from the increased cold working level is clearly reflected in the $K_{I_{rs}}/K_{I_{max}}$ solutions when compared with the previous plot (Figure 5.9). Furthermore, a greater deviation is observed from the purely elastic crack
growth solutions when the material plasticity is included. The maximum difference of about $\Delta K_{I_{rs}} / K_{I_{max}} = 15\%$ occurs near the normalized crack length $(a+r)/r = 0.21$. It is of interest to know whether the increase in the variation is caused by the higher residual stress magnitudes or the higher applied load level. To shed some light on this, another set of simulations were conducted with a greater cold working amount (1.6%) while keeping the applied load the same ($S_{\text{max}} / \sigma_{ys} = 0.4$).

Normalized stress intensity factors $K_{I_{rs}} / K_{I_{max}}$ due to the residual stress field are shown in Figure 5.11 in the same format of the previous two plots. The absolute $K_{I_{rs}} / K_{I_{max}}$ values increased further because of the higher compressive stress magnitudes produced from 1.6% cold hole expansion (see Figure 5.6). The maximum variation between the elastic and elastic-plastic crack growth is nearly $\Delta K_{I_{rs}} / K_{I_{max}} = 15\%$ at $(a+r)/r = 0.23$. Thus, difference in $K_{I_{rs}} / K_{I_{max}}$ did not change significantly with the increased level of cold work. Nevertheless, the region of variation is shifted slightly to the right (compare Figure 5.10 and 5.11). These observations suggest that it is the higher applied loading, not the residual stress magnitude that increases the deviation between the solutions of elastic and elastic-plastic crack growth. Residual stress magnitudes seem to affect the location of the bifurcation zone. It must be noted however that high compressive residual stresses present within the growth region may not allow the crack faces to fully open if the applied load is too low. This imposes an additional restriction on the maximum applied load levels, since the on-line crack compliance technique requires crack face displacements measured for fully open cracks.
5.4 Conclusions

The on-line crack compliance technique was studied using the 2D FE model of a rectangular sheet with a single crack emanating from a central hole under plane stress conditions. The residual stress fields were produced in the crack growth region by uniformly cold working the hole to three different levels: 0.8%, 1.2%, and 1.6%. The applied maximum loads considered were $S_{\text{max}}/\sigma_{\text{ys}} = 0.3$ and 0.4. The crack growth simulations were performed under purely elastic and elastic-plastic conditions. As part of the validation process, the normalized stress intensity factors $K_{Irs}/K_{I\text{max}}$ calculated using the on-line crack compliance technique are compared with the solutions obtained via $J$-integral method for elastic crack growth. Results were in good agreement indicating that the two methods were equivalent under elastic conditions. Also, the $K_{Irs}/K_{I\text{max}}$ values
obtained pertain to the original stress field induced by cold hole expansion simulation since no additional residual stresses are produced throughout elastic crack growth.

Convergence studies were performed to validate the results of the on-line crack compliance technique using the elastic-plastic crack growth model. Generated $K_{I_{rs}} / K_{I_{max}}$ magnitudes were generally lower than those produced from purely elastic crack growth. The deviation from the elastic solutions grew larger with increased applied maximum loading. Therefore, the lowest possible load levels must be used to obtain the more accurate data regarding the original residual stress field present within the component. However, high compressive residual stresses may prevent the crack faces from fully opening if the applied maximum load is too low. This must also be considered when selecting the load level, because the on-line crack compliance technique requires crack face displacements measured for fully open cracks.
5.5 References


CHAPTER VI
CONCLUSIONS AND FUTURE WORK

In the first part of this study, three dimensional finite element simulations of plasticity induced fatigue crack closure was performed using a model of a single edge-cracked tension specimen under constant amplitude cyclic loading with $R = 0$. The crack closure parameter $U$ was obtained using four different assessment methodologies: (a) node displacement method, (b) contact stress method, (c) compliance offset and (d) ACR. The node displacement method with the first node behind the crack tip and the contact stress method produced the lowest $U$ levels and these two methods compared well with one another. Using the second node behind the crack tip produced higher $U$ values and exhibited larger through-thickness variation especially at lower load levels. The compliance offset method generated the next higher $U$ levels. Also, improved agreement was observed between the second node displacement method and the compliance offset technique when small offset values were used. The lowest compliance offset considered was 0.5%, whereas the opening loads must be ideally computed as soon as the instantaneous compliance starts to diverge from the open crack compliance. Reducing this offset value may further improve the agreement with the second node displacement method. The highest $U$ levels were obtained for all considered applied loadings when ACR technique was employed. The computed ACR $U$ values were close to unity, indicating that the ACR solutions for $R = 0$ are not significantly different from the LEFM conditions. In future research, the influence of different $R$ ratios on $U$ levels needs to be
studied. Lower load levels must be considered, which inherently requires more computational power. The ACR method, in particular, is known to perform well in fatigue threshold measurements. Thus, a set of load shedding simulations should be conducted to further evaluate this method.

In the second part of this research, a detailed two dimensional stress analysis of a single pin-loaded hole in a rectangular sheet under plane stress conditions was conducted. First, the residual stress free hole was pin-loaded to different levels. The influence of the material nonlinearity and friction on the local radial and hoop stress distributions was studied. The maximum hoop stresses occurred at the hole edge. The values of the peak hoop stress were reduced by a significant amount when the material non-linearity was used. Friction increased the peak hoop stresses for the most part. Next, a compressive residual stress field was introduced around the hole by a simplified cold working simulation. Friction increased the peak hoop stress levels along the hole circumference. However, the absolute maximum hoop stress was located in the interior region, where the preexisting residual stress field from cold hole expansion was tensile. This indicates that the cracks near the cold worked holes may initiate at the locations away from the hole surface. The current study used two dimensional plane stress model by ignoring the out of plate bending effects. This may not be appropriate for single lap pin-joints. In the future, a three dimensional models need to be employed to include sheet bending effects. With the three dimensional models, one may also consider a more realistic residual stress field from the cold hole expansion simulation by explicitly modeling the tapered mandrel and the sleeve. Unlike the uniform hole expansion approach in two dimensional models, simulating the cold working process by pulling a tapered mandrel through the hole creates a residual stress field that varies along the plate thickness in three dimensional
models. Furthermore, the influence of bypass loads may be considered which may be a significant portion of the applied loading, particularly for the structural components with multiple pin-joints.

In the third part of this work the beneficial influence of cold working process was quantified by computing the Mode I stress intensity factors for a single radial crack emanating from a side of the loaded hole. Two different loading configurations were considered: (a) an open hole in tension, (b) a pin-loaded hole. The $K_I$ values were obtained for the normalized crack lengths from $(a+r)/w = 0.175$ to $0.75$ using the $J$ integral approach and the weight function technique. Solutions of these two approaches correlated well with one another. Significant reductions in the $K_I$ levels were observed when the hole is cold-worked as opposed to the non-cold worked hole, and the amount of reduction depended on the level of interference. However, the beneficial effect of the cold working process diminished after the crack size reached a certain length. This work assumed that the residual stress field around the hole does not alter during the crack growth process, whereas plastic deformation occurring near the growing crack tip may cause stress redistribution. In the future, the consequence of stress alteration during elastic-plastic crack growth analysis on the computed $K_I$ levels around the cold worked hole may be studied.

In the final part of the study, the on-line crack compliance technique was investigated using the two dimensional finite element model of crack growth under plane stress conditions. The on-line crack compliance is a rather new experimental technique used to measure the existing residual stress field along the growing crack line. Its theoretical foundation is based on linear elastic fracture mechanics principles as derived in Appendix A1. The main idea of the on-line compliance technique is that there is a
direct relation between the applied Mode I stress intensity factor $K_i$ and the incremental crack face opening displacements with respect to the incremental crack growth $d\delta/da$.

The function relating the two is called an influence function, which can be readily obtained if the Green’s function is known for the crack configuration. The present study used a model of a finite rectangular sheet with a single crack emanating from a side of a central hole in the direction perpendicular to the applied loading. The residual stress field was introduced around the hole by cold working simulation with different interference levels. The crack growth was simulated with purely elastic and elastic-plastic material constitutive models. Normalized residual stress intensity factors $K_{frs}/K_{f max}$ were calculated from the incremental crack mouth opening displacements. As part of the validation, the $K_{frs}/K_{f max}$ solutions from the on-line crack compliance method were compared with those obtained using the $J$-integral approach under purely elastic conditions. The results were nearly identical indicating that the on-line crack compliance method is as accurate as the more traditional $J$-integral approach when the material is linear elastic. The elastic-plastic solutions of $K_{frs}/K_{f max}$ were generally lower than those produced from purely elastic crack growth. The variation between the two grew larger with higher applied maximum loading. It must be noted that the elastic results of $K_{frs}/K_{f max}$ pertain to the original residual stress field from cold-working since no additional residual stresses are introduced during elastic crack growth. This is no longer true for elastic-plastic crack growth, since the crack tip plasticity may alter existing residual stress field. Therefore, during experimental measurements, the lowest possible load levels must be used to mitigate the effect of crack tip plasticity and to obtain the more accurate data regarding the original residual stress field present within the component. However, if high compressive residual stresses exist in the crack growth
region, the crack faces may not fully open when the applied maximum load is too low. This must also be considered when selecting the load level, because the on-line crack compliance technique requires crack face displacements measured for fully open cracks.

The variation between elastic and elastic-plastic results is explained by recalling the fact that the on-line crack compliance technique was derived using the LEFM principles. Thus, there is no doubt that the existence of plastic deformation near the crack tip during the fatigue crack growth testing interferes with the method results. Further detailed study needs to be conducted to better understand the influence of residual stress evolution during the elastic-plastic crack growth and the effect of the material constitutive behavior such as strain hardening on the performance of this technique.
APPENDIX A

THE ON-LINE CRACK COMPLIANCE METHOD
A.1 The On-line Crack Compliance Technique

The on-line crack compliance technique was first introduced by Lados [1, 2]. This methodology allows the computation of applied stress intensity factors using the incremental crack opening displacements. This section presents a comprehensive theoretical background of the technique for a plane stress condition. Although parts of this discussion are presented elsewhere [1], they are repeated here for the reader’s convenience.

Consider a cracked plate subjected to a fixed force load $P$ as shown in Figure A.1. The energy release rate of the plate is defined as the rate of change in its total potential energy $\Pi$ with respect to a newly formed crack surface $A$ [3]

$$G = -\frac{d\Pi}{dA} \tag{A.1}$$

The total potential energy is given by

$$\Pi = U - W \tag{A.2}$$

where $U$ is the strain energy stored in the cracked body and $W$ is the work done by external forces. Since the applied point force $P$ is fixed (i.e. the specimen is load controlled), $U$ and $W$ will be

$$U = \frac{P\Delta}{2} \tag{A.3}$$

$$W = P\Delta \tag{A.4}$$

Then the total potential energy $\Pi$ and the energy release rate $G$ will become

$$\Pi = -\frac{P\Delta}{2} = -U \tag{A.5}$$

$$G = -\frac{d\Pi}{dA} = \frac{1}{B} \frac{dU}{da} = \frac{1}{2B} \frac{d(P\Delta)}{da} \tag{A.6}$$
where $B$ is the thickness of the specimen. For a constant applied load $P$, the equation A.6 can further be simplified to

$$ G = \frac{P}{2B} \frac{d\Delta}{da} \quad (A.7) $$

The mode I stress intensity factor $K_{IP}$ is related to the energy release rate via the modulus of elasticity $E$ of the material

$$ K_{IP}^2 = EG = \frac{EP}{2B} \frac{d\Delta}{da} \quad (A.8) $$

We recognize that the stress intensity factor $K_{IP}$ is a linear function of $P$, so it may be expressed as

$$ K_{IP} = P \frac{f_P(a,w,...)}{B} \quad (A.9) $$

where $f_P$ is a function that depends on the geometry and crack size. By combining equations A.8 and A.9 we get

$$ K_{IP} = \frac{E}{2f_P(a,w,...)} \frac{d\Delta}{da} \quad (A.10) $$

Figure A.1  Cracked plate subjected to fixed load $P$
Thus far, we were able to relate the rate of change of the load point displacement \( d\Delta/da \) to the stress intensity factor due to the applied force load \( P \) via a geometry function \( f_p(a,w,\ldots) \) and the modulus of elasticity \( E \). However, the on-line crack compliance method uses an incremental crack face opening displacements \( d\delta \) instead of \( d\Delta \). Therefore, \( d\Delta/da \) will next need to be related to \( d\delta/da \). We will use the approach presented in Appendix B of Tada and Paris [4]. Let \( F \) be a virtual pair force applied at a point \( a_F \) along the crack face, where the crack face opening displacement \( \delta \) is being measured (see Figure A.2). If \( K_{IF} \) is a stress intensity factor due to \( F \), then by superposition the total stress intensity factor is

\[
K_I = K_{IP} + K_{IF} \tag{A.11}
\]

The strain energy of the cracked body can be decomposed into two parts

\[
U = U_{no\,crack} + dU = U_{no\,crack} + \int_0^a \frac{\partial U}{\partial a} da \tag{A.12}
\]

where \( U_{no\,crack} \) is the strain energy corresponding to the applied forces with no crack present, and \( dU \) is due to introducing a crack \( a \) while holding the forces constant [4]. Then, using the equations A.6, A.8, A.11, and A.12

\[
U = U_{no\,crack} + B \int_0^a G\,da = U_{no\,crack} + \frac{B}{E} \int_0^a (K_{IP} + K_{IF})^2 da \tag{A.13}
\]
For linear elastic materials, Castigliano’s theorem may be employed to determine the displacements $\Delta$ and $\delta$ by differentiating the strain energy $U$ with respect to the corresponding forces $P$ and $F$ and by setting the virtual force $F$ equal to zero.

\[ \Delta = \frac{\partial U}{\partial P} = \Delta_{\text{no crack}} + \frac{2B}{E} \int_0^a K_{IP} \frac{\partial K_{IP}}{\partial P} \, da \]  
(A.14)

\[ \delta = \frac{\partial U}{\partial F} = \frac{2B}{E} \int_{a_F}^a K_{IP} \frac{\partial K_{IP}}{\partial F} \, da \]  
(A.15)

Where $\Delta_{\text{no crack}} = \partial U_{\text{no crack}}/\partial P$ is the displacement of the uncracked body due to the applied force $P$. Note the equation A.15 does not have a the similar term $\delta_{\text{no crack}}$ since, with the absence of the crack, the opposite forces $F$ are applied at the same point resulting in $\delta_{\text{no crack}} = 0$.

By differentiating the above expressions with respect to the crack length $a$ and recognizing that $\Delta_{\text{no crack}}$ has no dependence on $a$ we get

\[ \frac{d \Delta}{da} = \frac{2B}{E} K_{IP} \frac{\partial K_{IP}}{\partial P} \]  
(A.16)
\[
\frac{d\delta}{da} = \frac{2B}{E} K_{IF} \frac{\partial K_{IF}}{\partial F} \tag{A.17}
\]

We may express \(K_{IF}\) in a similar manner to equation A.9 as

\[
K_{IF} = F \frac{f_F(a,w,...)}{B} \tag{A.18}
\]

where \(f_F(a,w,...)\) is another function that depends on the geometry, crack length as well as the location of the point load. By substituting equations A.18 and A.9 into A.16 and A.17, the following expressions for the displacement rates can be obtained

\[
\frac{d\Delta}{da} = \frac{2}{E} P f_F(a,w,...)^2 \frac{1}{B} \tag{A.19}
\]

\[
\frac{d\delta}{da} = \frac{2}{E} P f_F(a,w,...) f_F(a,w,...) \frac{1}{B} \tag{A.20}
\]

Thus, the load point and crack face displacement rates \((d\Delta/da\) and \(d\delta/da\) are related with one another as follows

\[
\frac{d\Delta}{da} \frac{1}{\frac{d\delta}{da}} = \frac{f_F(a,w,...)}{f_F(a,w,...)} \tag{A.21}
\]

or

\[
\frac{d\Delta}{da} = \frac{f_F(a,w,...)}{f_F(a,w,...)} \frac{d\delta}{da} \tag{A.22}
\]

Finally from equation A.10, the mode I stress intensity factor due to applied fixed point load \(P\) will be

\[
K_{IF} = \frac{E}{2f_F(a,w,...)} f_F(a,w,...) \frac{d\delta}{da} = \frac{E}{2f_F(a,w,...)} \frac{d\delta}{da} \tag{A.23}
\]

And by defining the influence function to be

\[
Z(a,w,...) = 2f_F(a,w,...) \tag{A.24}
\]

The equation A.10 can be written as

\[
K_{IF} = \frac{E}{Z(a,w,...)} \frac{d\delta}{da} \tag{A.25}
\]
Thus, the applied stress intensity factor can be represented in terms of the rate of change of the crack face displacement, the modulus of elasticity of the material, and the influence function specific to the crack face measurement location. Since, the influence function is simply $Z(a,w,...) = 2f_{F}(a,w,...)$, there is a direct relation between $Z(a,w,...)$ and a Green’s function for a pair of point loads applied at the crack surface, where the opening displacement $\delta$ is measured.
A.2 References


