THREE-DIMENSIONAL FATIGUE CRACK GROWTH ANALYSIS USING FAILURE MECHANISM APPROACH IN ANSYS

By:

MUHAMMAD DHIRAR AS SUHAIMI BIN ISMAIL

(Matric No.: 128954)

Supervisor:

Ir. Dr. Feizal Yusof

May 2019

This dissertation is submitted to
Universiti Sains Malaysia
as partial fulfillment of the requirement to graduate with honors degree in
(BACHELOR OF ENGINEERING (MECHANICAL ENGINEERING)

UNIVERSITI SAINS MALAYSIA

School of Mechanical Engineering
Engineering Campus
Universiti Sains Malaysia

DECLARATION

This work has not previously been accepted in substance for any degree and is not being concurrently submitted in candidature for any degree.

Signed…………………………………………..
(MUHAMMAD DHIRAR AS SUHAIMI BIN ISMAIL)
Date……………………………………………..

STATEMENT 1
This thesis is the result of my own investigations, except where otherwise stated.
Other sources are acknowledged by giving explicit references.
Bibliography/references are appended.

Signed…………………………………………..
(MUHAMMAD DHIRAR AS SUHAIMI BIN ISMAIL)
Date……………………………………………..

STATEMENT 2
I hereby give consent for my thesis, if accepted, to be available for photocopying and for interlibrary loan, and for the title and summary to be made available outside organizations.

Signed…………………………………………..
(MUHAMMAD DHIRAR AS SUHAIMI BIN ISMAIL)
Date……………………………………………..

ACKNOWLEDGEMENT

The completion of this project is not possible without the help from many people. First of all, I would like to express my gratitude to Universiti Sains Malaysia for giving me the chance to complete this four-year course in Mechanical Engineering.

I would like to express my gratitude to my dedicated supervisor, Dr. Feizal Yusof who has given me the chance to carry out this interesting and yet challenging project. Beside this, thanks for giving constant guidance and advice throughout this whole project. It would not be possible for the projects to go this far without your patience and enthusiasm encouragement and guidance.

Next, I would like to extend my gratitude to the staffs of the School of Mechanical Engineering, especially Mrs. Jamari Sadli for providing the access to high-performance personal computer in Computer Aided Design (CAD) lab. Apart from that, I would like to thank all my fellow friends who provide me the encouragement and assistant whenever needed.

Last but not least, I would like to thank my dearest family and friends for their continuous support along the journey of completion of this final year project.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATION

SIF	Stress Intensity Factor
SMART	Separating, Morphing, Adaptive, and Remeshing Technology
LC	Life Cycle Prediction
LEFM	Linear Elastic Fracture Mechanic
NLEFM	Non-Linear Elastic Fracture Mechanic

LIST OF SYMBOLS

SYMBOL	Description	Unit
K	Stress Intensity Factor	MPa $(m)^{0.5}$
K_c	Critical Stress Intensity Factor	MPa $(m)^{0.5}$
K_I	Stress Intensity Factor in Mode I	MPa $(m)^{0.5}$
K_{II}	Stress Intensity Factor in Mode II	MPa $(m)^{0.5}$
$(K_I)_{max}$	Maximum Stress Intensity Factor in Mode I	
$(K_{II})_{max}$	Maximum Stress Intensity Factor in Mode II	
G	Strain Energy Release Rate	N/mm
G_c	Critical Strain Energy Release Rate	N/mm
J	J - Integral	J/m^2
J_c	Critical J - Integral	J/m^2
Γ	Line Integral	-
υ	Poisson's Ratio	-
E	Young's Modulus	Pa
σ_f	Fracture Stress	Pa
a	Crack length	mm
γ_s	Elastic surface energy	N/m
γ_P	Plastic deformation energy	N/m
Q	Point load	N
T	Traction vector	N/m^2

ABSTRAK

Keretakan kritikal ada terutama pada permukaan komponen mekanikal dalam kejuruteraan. Untuk meramalkan penyebaran retakan lesu, hayat pertumbuhan retakan lesu, dan pencirian retakan permukaan melalui alat simulasi yang berbeza dilaksanakan secara meluas. Daripada analisis fraktur, diketahui bahawa medan tekanan dalam retakan permukaan 3D berbeza dari medan hujung terikan konvensional. Faktor keamatan tekanan (SIF) bergantung pada kelengkungan depan retak, tepi retakan ke permukaan bebas, dan konfigurasi struktur. Terutama, SIF mungkin berubah secara tidak seimbang dengan pertumbuhan retak. Keterangkapan dan ketepatan mekanik retak ke retakan permukaan 3D masih menjadi isu menarik bagi banyak bahagian mekanikal di bawah geometri dan aplikasi beban yang berbeza. Jadi, tesis ini mencadangkan Pemisahan, Perubahan Bentuk, Penyesuaian dan Penyiratan Teknologi (SMART) yang merupakan ciri baru yang terdapat di dalam ANSYS versi 19.0 dan ke atas untuk menganalisis pertumbuhan retakan lesu di bawah beban kitaran dalam analisis 3D. Kajian ini adalah untuk menilai hayat pertumbuhan retakan lesu (FCG) dan kesan variasi ketidaksambungan geometri dan aplikasi beban yang berbeza pada trajektori retak masing-masing. Dengan menggunakan kaedah Ramalan Kitaran Hayat (LC) dalam SMART, Analisis Unsur Terhingga (FEA) dilakukan pada spesimen yang diambil dari kajian terdahulu untuk mendapatkan arah pergerakan retakan di bawah bebanan lesu. Analisis dilakukan menggunakan ANSYS 19.2 di bawah platform *workbench* mekanikal. Penyebaran retak di dalam spesimen telah disimulasikan dengan menggunakan ciri retak SMART di bawah kaedah Ramalan Kitaran Hayat (LC). Tesis ini mengesahkan kesahihan ciri baru ini untuk analisis pertumbuhan retak dan menunjukkan kesan variasi ketidaksambungan geometri dan aplikasi beban yang berbeza pada sudut penyebaran retak. Didapati bahawa kecacatan geometri dan variasi aplikasi beban memberi kesan utama pada sudut penyebaran retak dan kitaran hayat lesu spesimen. Tesis ini mewajarkan bahawa sudut retak dan hayat spesimen dipengaruhi oleh kedudukan aplikasi beban, ketidaksambungan geometri, kecacatan yang wujud dalam spesimen termasuk kecacatan specimen.

ABSTRACT

Critical cracks exist mainly on surfaces of mechanical components in engineering. To predict fatigue crack propagation, fatigue crack growth life, and characterization of a surface crack through different simulation tool are widely being implemented. From fracture analysis it is known that the stress field in a 3D surface crack differs from the conventional plane strain tip field. The stress intensity factor (SIF) depends additionally on crack front curvature, crack edge to free surface, and structure configuration. Especially, SIF may vary non-proportionally with crack growth. Predictability and accuracy of the fracture mechanics to 3D surface crack are still interesting issue for many mechanical parts under different geometry and loading conditions. So, this thesis proposes Separating, Morphing, Adaptive, and Remeshing Technology (SMART) which is a new feature available in ANSYS version 19.0 and above to analyze fatigue crack growth under cyclic loading in 3D analysis. This research is to evaluate the fatigue crack growth (FCG) life, effect of geometric discontinuity variation and loading position different on the crack trajectory and fatigue life respectively. Finite element analysis (FEA) was performed on a test specimen taken from a published research to obtain the direction of crack propagation under fatigue loading. The analysis was executed using ANSYS 19.2 under mechanical workbench platform. The crack propagation on the specimen was simulated using Life-Cycle Prediction (LC) method under SMART feature. This thesis verifies the validity of this new feature for crack growth analysis and to portray the effect of geometric discontinuity variation and different loading application on the crack propagation angle and specimen fatigue life respectively. It was found that, geometric discontinuity and loading application variation give a major effect on the crack propagation angle and specimen's fatigue life cycle. This also justify that the crack angle and fatigue life is influenced by loading position, geometric discontinuity and defect that present in the specimen.

CHAPTER 1: INTRODUCTION

1.1 Background of Research

Study of crack propagation analysis is still actively pursued. The stresses and strains field analysis near the crack-tip are the basis for understanding the behavior of cracks. Stress Intensity Factor (SIF) is an important fracture parameters used to estimate the structures behavior containing crack and surrounding of crack-tip [1]. Although a plastic and damaged zone is present at the crack-tip, the linear elastic analysis provides an accurate mapping of reality for materials such as steel. For high ductility materials or extreme loads, elastic-plastic behavior laws are taken into account [2].

Common three modes of fracture are shown in Figure 1.1. Mode I fracture called opening mode where a tensile stress applied in normal direction to the plane of the crack (a traction mode). Mode II fracture is Sliding mode where a shear stress acting parallel to the plane of the crack and perpendicular to the crack front (a shear mode). Mode III fracture is Tearing mode where a shear stress acting parallel to the plane of the crack and parallel to the crack front (a torsion mode). A fracture body can be applied with one or combination of two or three modes [1].

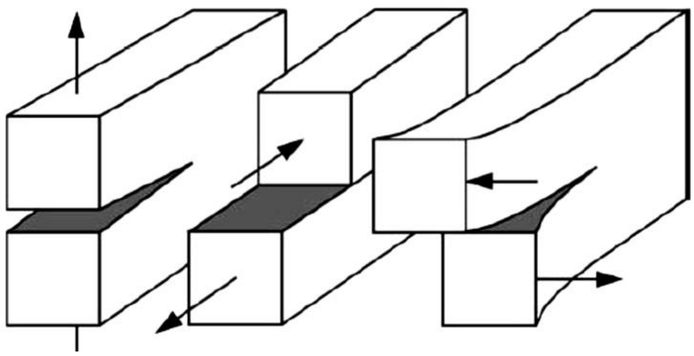


Figure 1.1: Loading Modes I, II, and III [1].

Fatigue is the most common cause of crack initiation and crack growth to critical size, at which sudden fracture takes place. It is well known that the fatigue crack growth is influenced by many factors such as loading ratio, environment, loading sequence, microstructure, geometry, thermal activation, etc. Fatigue occurs when a metal or nonmetal (plastic or composite) component is subject to fluctuating or cyclic stress and strain and ultimately results in cracks. Metal fatigue is a failure or fracture mechanism that lead to cracks in the components. Typically, a fatigue crack starts at a point, usually on the surface of the component, and then grows into the metal as the component is subjected to the cyclic stress [3]. Fig 1.2 shows the various stages of the crack before failure occurs.

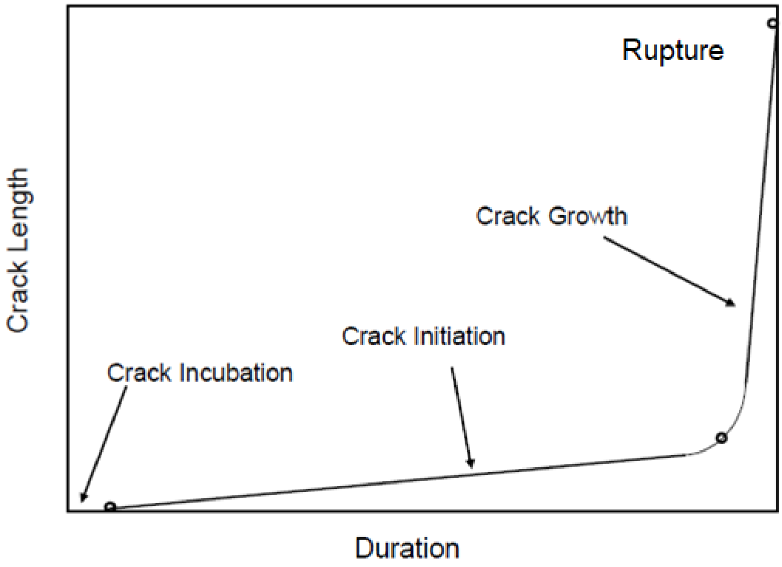


Figure 1.2**:** Crack Length as Function of Time[3]

Crack growth depends on different condition and materials behavior, the crack growth is expressed in three stages, before final failure occur, schematic showed in Figure 1.3.

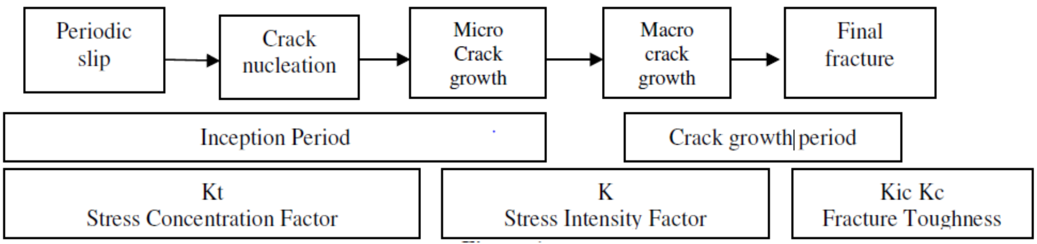


Figure 1.3**:** Different Stages of Crack Fatigue Crack Growth Life [2]

In the initiation phase, cracks are very short and grow irregularly. For instance, the crack growth may become retard for an unknown amount of time as the crack hit a grain boundary or other barrier. As these barriers is overcome by the crack, a stable mode of crack growth can be reached. This steady phase of crack growth is thought to be well modelled by the Paris law. The crack initiation stage is vital for the next part of the crack's life. The differences between these two regions of a crack's life are very important when determining the fatigue life of a specimen. It was realized that crack extension takes place due to stress concentration at the crack tip and due to failure of material during cyclic loading; an effort has been constructed to relate the crack growth with stress intensity factor "K" at the crack tip. A well-established relationship was given by Paris and Erdogan [4] and takes the following form:

$$\frac{da}{dN} = C\ (\Delta K)^m \tag{1.1}$$

Where C, m are material constants, ΔK is the stress intensity factor amplitude, a is the crack size and N is the cycle number of periodic loads. A typical fatigue rate curve for fatigue crack growth, commonly referred to as a da/dN versus ΔK curve, is illustrated by Figure 1.3. The curve is defined by regions I, II and III..

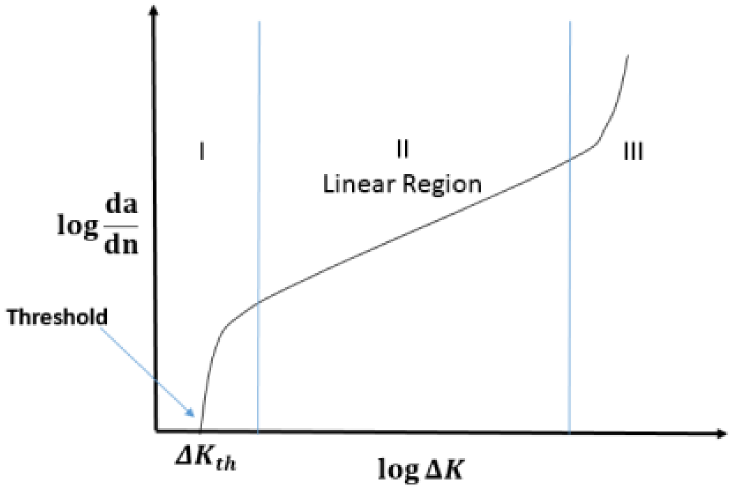


Figure 1.4: Variation of Crack-Growth Rate vs. ΔK for Metal [5]

There are has three stages of crack growth which are crack initiation, subcritical crack growth and unstable fracture. When ΔK is lower than threshold value ΔKth, the crack is not extended. When ΔK> ΔKth, it comes to the stage II, which is the subcritical crack growth stage, and the component fractured when it comes to the stage III. The subcritical crack growth stage has good log-linear relationship, and this stage constitutes the main part of the fatigue life under low stress periodic load. Thus, stage II is key point of the fatigue crack growth simulation [6].

For the direction of crack propagation, maximum circumferential stress criterion was introduced by Erdogan and Sih [7] for elastic materials. It states that the crack propagates in the direction where the circumferential stress $\sigma_{\theta\theta}$ is maximum. It is a local approach since it is based on local stress fields around the crack front (see fig 1.5). The circumferential stress field at the evaluation points around the tips of the crack front can be formulated in terms of the stress intensity factors. Once the mixed-mode stress intensity factors have been found the following equation can be used to find the angle of crack growth, θ_c.

$$\sigma_{\theta\theta} = \frac{1}{\sqrt{2\pi r}} \frac{K_I}{4}\left(3\cos\frac{\theta}{3} - \cos\frac{3\theta}{2}\right) + \frac{K_{II}}{4}\left(-3\sin\frac{\theta}{2} - 3\sin\frac{3\theta}{2}\right) \tag{1.2}$$

$$K_{max} = {\sigma_{\theta\theta}}^{max}{}_{\sqrt{2\pi r}} \tag{1.3}$$

Solving θ_c

$$\theta_c = \cos^{-1}\frac{3(K_I)_{max} + (K_I)_{max}\sqrt{(K_I)^2{}_{max} + 8(K_{II})^2{}_{max}}}{(K_I)^2{}_{max} + 9(K_{II})^2{}_{max}} \tag{1.4}$$

Where θ_c is the angle that will follow the crack for each of the crack increment. θ_c is measured with respect to a local polar coordinate system with its origin at the crack tip and aligned with direction of the existing crack. Once the crack growth orientation is determined, a propagation increment Δa is added to the existing crack geometry and the analysis procedure is repeated.

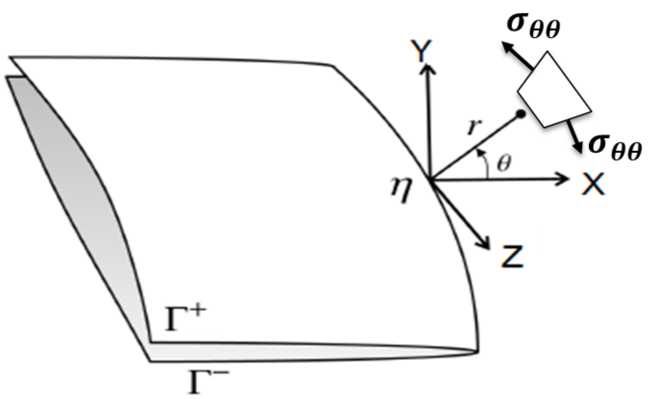


Figure 1.5: Local stress field at crack front

Fatigue crack growth has been studied in various types of structural materials with its analysis approach can be experimental, analytical or numerical. Finite element method (FEM) is one of the numerical methods that is frequently used for simulating crack growth under fatigue loading. Through finite element analysis (FEA), propagation of crack growth can be represented under realistic conditions and with the development in fracture mechanics, the accuracy of FEM is highly appreciable. Fatigue crack growth simulation consists of two key parts: Numerical analysis and mesh generation [8].

In recent years, the SMART crack growth which stands for Separating, Morphing, Adaptive, and Remeshing Technology (SMART) has been introduced in ANSYS version 19.0 as a powerful numerical procedure for the analysis of crack problems. SMART is a remeshing-based method for crack growth simulation. This method automatically uses a combination of morphing, adaptive, and remeshing methods to update mesh changes to simulate static or fatigue crack growth during the solution process. A SMART crack-growth simulation is assumed to be quasi-static. This method can perform in both static and fatigue crack-growth simulation [9].

SMART updates the mesh from crack-geometry changes due to crack growth automatically at each solution step. Mesh updates occur around the crack-front region only and are integrated into the solver without exiting and re-entering the solver, resulting in a computationally efficient solution of the crack-growth problem. Crack-growth mechanics include various fracture criteria for static crack growth and Paris' Law for fatigue crack growth [9] .

Development of a fatigue crack growth model by using numerical modelling is important in characterizing the susceptibility of structures to damage and failure due to fatigue loading as prediction of the crack propagation process is vital for many fracture mechanical issues. Numerical simulation has the ability to provide unique possibilities and has become a powerful tool in developing and executing this task. There is high technical interest in modelling subcritical growth of fatigue cracks, stable crack growth in ductile materials and unstable dynamic fracture processes. Fracture mechanics provides criteria and principles for these cases that specify:

(1) Initiation of crack propagation at certain load level and cycle.

(2) Direction of crack propagation growth, θc

(3) Scale and size of crack propagation.

Thus, the object of numerical simulation is to implement these laws in the context of the finite element method by appropriate solution algorithms. This steady phase of crack growth is thought to be well modelled by the Paris law. The crack initiation stage however is thought to account for a large part of the crack's life. The differences between these two regions of a crack's life are very important when determining the fatigue life of a specimen [3]. SMART based crack growth analysis will be used in Ansys Mechanical Workbench platform. The SMART crack-growth method uses stress-intensity factors (SIFS) as the main fracture parameter (driving force) and the criteria for crack-growth calculation [9]. Fracture mechanics deals with cracks (defects), and a singularity always exists around the crack tip/front. The crack-tip/-front mesh is therefore of utmost importance in a crack analysis, as stress-analysis and fracture-parameters calculation accuracy depend on the crack mesh. Size and shape differences in the elements ahead of and behind the crack tip/front affect the accuracy of the fracture-parameters calculation, and therefore the crack-growth simulation [9].

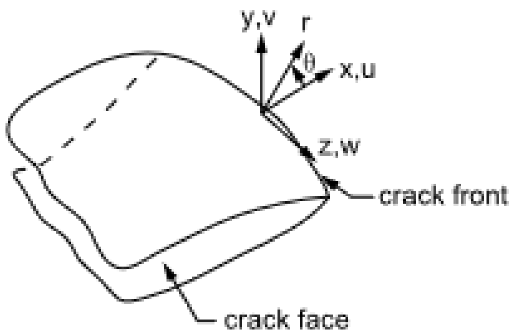


Figure 1.6: Crack Tip Region [10]

1.2 Problem Statement

Cracks and flaws occur in many structures and components for several reasons. The material may be inherently defective. Cracks may be introduced during the manufacturing stage, or later as a result of environmental conditions. The presence of such cracks or flaws can significantly degrade the structural integrity of a component under the action of applied loads and environmental conditions. Cracks frequently change their shape at geometric discontinuities such as corners or holes. Nearly all structures exhibit some kind of geometric discontinuities. If a crack initiates in the neighborhood of such a discontinuity the question naturally arises how this geometric feature will influence the crack propagation. Geometric discontinuity always presence in a structure where it will influence the stress flow and distribution when load is applied. Due to concentrated stress flow in a structure, it will affect the direction of the crack propagation originated from a crack.

There are many literatures focusing on the impact of structure defect, flaw and geometric discontinuity on crack growth propagation. Inspection are used to look for cracks, and if a crack is found the component can be fixed or replaced as necessary. Inspection intervals could then be set at some fraction of the fatigue life. Fatigue fracture life is calculated as the number of cycles required to grow a crack from a minimum detectable size to a critical size when the structure fails. Numerical fatigue crack growth model is available in (ANSYS 19.2) for many generalized geometries and crack problems including edge crack, arbitrary crack and semi-elliptical crack and etc. As geometry, loading and boundary condition become more complex, the analytical solution is not reliable as the problem become more complicated and it may be difficult to account for all effects on crack propagation. The Finite Element Method (FEA) has been used for decades to assist engineers in the analysis of complex cracked structures.

Until recently, cracks had to be modelled as part of the geometry. As the crack grew, the model would be rebuilt, requiring significant user interaction or specialized program. Separating, Morphing, Adaptive and Remeshing Technology (SMART) is new crack analysis mechanism which cover simulation of both static and fatigue crack growth in engineering structures. SMART updates the mesh from crack-geometry changes due to crack growth automatically at each solution step. Mesh updates occur around the crack-front region only and are integrated into the Mechanical APDL solver without exiting and re-entering the solver, resulting in a computationally efficient solution of the crack-growth problem. Therefore, the main aim of this project is to take a look at the ability of SMART to evaluate crack growth.

1.3 Objectives of Project

The primary goal of this project is to model three-dimensional (3D) fatigue crack growth analysis using SMART method in ANSYS. The following objective are to be met to realize these goals.

i. To develop fatigue crack growth (FCG) life and to illustrate graphically the crack propagation of the specimen which contain a defect and geometric discontinuity using SMART method in ANSYS.

ii. To study the effect of geometric discontinuity and load configuration on the crack trajectory and fatigue life of the specimen respectively under 3D analysis using Life-Cycle Prediction (LC) method in SMART feature.

1.4 Scope of Work

The scope of the study presented in this thesis is to investigate the effect of geometric discontinuity and load configuration on the crack trajectory in three-dimensional plate and obtaining the FCG life of the specimen. The test specimen was chosen from previous study of crack analysis simulation by A. Meyer and F. Rabold [11] see Fig 1.6. Simulating a single edge crack in Annealed AISI 4140 Alloy Steel plate with different geometrical and loading configuration. Material properties and Paris's constant value used in the simulation is taken from experimental data done by Ramasagara Nagarajan [12] in his fatigue crack growth experiment on Annealed AISI 4140 Alloy Steel plate. The specimen is set to different geometrical and loading configuration which result in four geometry type and two loading configurations in total. In the crack simulation, Life-Cycle Prediction (LC) method under SMART was used to compute SIF and FCG life. From the SIF value, crack propagation angle is obtained through manual calculation. The equation of the crack propagation angle used is extracted from ANSYS Fatigue Crack Growth analysis manual guide. All the crack propagation angle is calculated for all the specimen configuration. The effect of the configuration on the crack propagation angle and FCG life is are compared between the all the geometrical and loading type.

CHAPTER 2: LITERATURE REVIEW

2.1 Introduction

Before proceeding to the research and analysis of the proposed technique, earlier studies are important to be understood. In addition, previous studies and methods used from other research works are reviewed in this chapter. Fracture studies were carried out by many researchers to explain the crack analysis by various methods and to enhance the formulation. Stress intensity factor (SIF) and energy is parameters used to analyze failure of the structure. Calculations of crack propagation angle with different approaches in conjunction with maximum circumferential stress criterion are reviewed. Three-dimensional (3D) fatigue crack analysis concept is reviewed in aspect of numerical method. Furthermore, SMART is the recent feature utilized by ANSYS for investigating crack analysis in engineering industry. The reviews enable the development and execution of the proposed method.

2.2 Review of fracture mechanics

From the earlier study, the rupture phenomena of internal cracks and flaws has been discovered by Griffith [13] which having an important role in initiation and propagation of fracture. The study established, the relationship between the fracture strength and the crack size in terms of the total energy change during cracking process.

$$\sigma_f = \left(\frac{2E\gamma_s}{\pi a}\right)^{\frac{1}{2}} \qquad (2.1)$$

The general expression of stress intensity factor derived from Griffith is shown below.

$$K = \sigma_f \sqrt{\pi a} \qquad (2.2)$$

Westergaard [14] take a different approach to derive solution based on the stress field near a sharp crack-tip. The equation used in computation of vertical stress due to the point load at the surface suggested by Westergaard is given as below.

$$\sigma_z = \frac{Q}{z^2\pi}\frac{1}{\left[1+2(\frac{r}{z})^2\right]^{\frac{3}{2}}} \quad (2.3)$$

Irwin [15] independently modified the Griffith expression to account for material that are capable of plastic flow.

$$\sigma_f = \left(\frac{2E(\gamma_s+\gamma_p)}{\pi a}\right)^{\frac{1}{2}} \quad (2.4)$$

The revised expression is given by (2.4) where γ_p is the plastic work per unit area of surface created and is typically much larger than γ_s

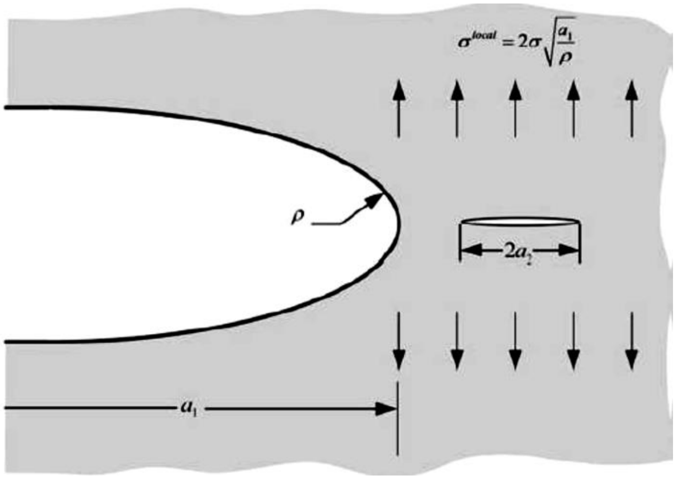


Figure 2.1: A sharp microcrack at the tip of a macroscopic crack

Rice (Rice, 1968) introduced non-linear energy release rate being given by a path independent line integral called J-integral. The definition of J-integral around the crack-tip is as shown in Fig 2.2, where the energy release rate is calculated along the curve Γ using a line integral. Then, the general finite element solutions of fracture mechanics problems were started by Rice and Levy [16]. The J-integral was defined as a path independent contour integral which equals the potential energy rate of change for elastic nonlinear solid during crack extension. Performing an analysis of the potential energy balance of a cracked body, Rice proved that, under quasi-static conditions, the J integral is equivalent to the energy release rate G.

$$J \equiv G \tag{2.5}$$

This very powerful equality allows using the J integral for practical numerical evaluation of the energy release rate. Using the Irwin's formula, the stress intensity factors can also be linked to the J integral. However, under mixed mode solicitations, this integral does not allow to differentiate the three different stress intensity factors value from the global expression of J.

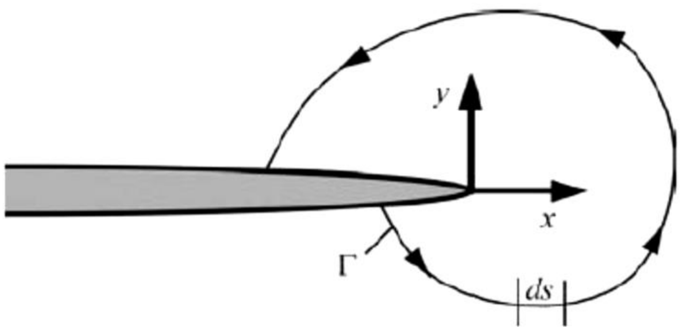


Figure 2.2: Arbitrary contour around the crack-tip [17]

Discussed in ANSYS Fracture Mechanic Analysis manual guide [18] was nonlinear fracture mechanics (NLFM), where the material behavior is described via the general Ramberg-Osgood relation, the J-integral characterizes the stress at the crack tip. It can therefore be used as a crack-growth criterion similar to the stress-intensity factor. A simple criterion based on the energy-release rate can be expressed as

$$J = J_c \tag{2.6}$$

The most general approach of crack initiation is the energy-release-rate method. Driving the crack growth requires an increase in the surface energy to separate the two crack surfaces. When the surface energy reaches the critical value, the crack surfaces separate, and the crack grows. A simple criterion based on the energy-release rate can be expressed as:

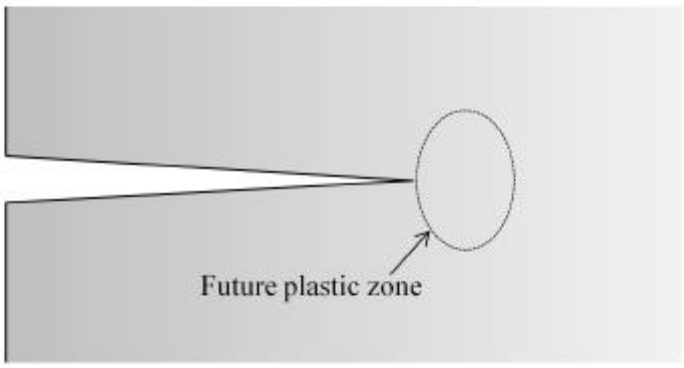
$$G = G_c \tag{2.7}$$

where G_c is the so-called critical fracture energy required to separate the two crack surfaces. It is considered to be a material property, independent of the applied loads and the geometry of the body and is often referred to as the fracture energy [18]. For isotropic, perfectly brittle, linear elastic materials, the J-integral can be directly related to the fracture toughness if the crack extends straight ahead with respect to its original orientation. For plane strain, under Mode I, II, III loading conditions, this relation is

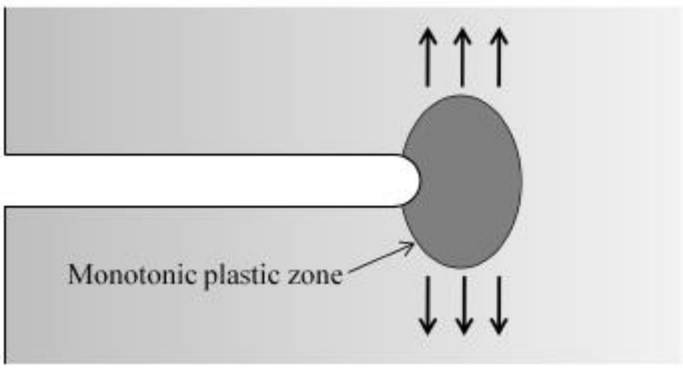
$$J_c = G_c = K_c^{\,2}\left(\frac{1 - v^2}{E}\right) \tag{2.8}$$

2.3 Stress Intensity Factors Computation

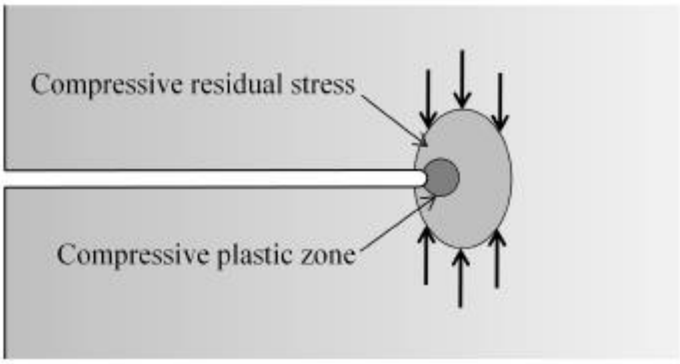
The computation of the stress intensity factors and of the energy release rate in arbitrary tri-dimensional structures was reviewed by Florent Galland in research of an adaptive model reduction approach for 3D fatigue crack growth [10] and it is found to be a challenging task in term of its computation method. Several methods have been proposed over the years, such as the elemental crack advance, the stiffness derivative method [19], or the displacement matching method.



a)



b)



c)

Figure 2.3: Deformation at a crack tip during a single load-unload cycle. A crack in virgin material (a) is submitted to a tensile load resulting in tip blunting and monotonic plastic zone appearance (b). Then the load is removed, and the compressive residual stresses lead to the formation of a compressive yielded zone, the cyclic plastic zone (c) [17]

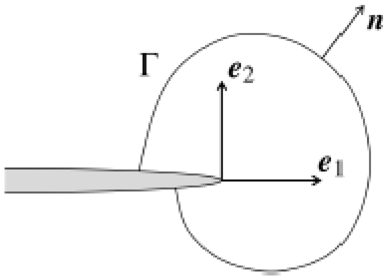


Figure 2.4: Crack tip integration contour

The last method has been utilized for quite time in the industry, in particular because of its simplicity of implementation. It consists in projecting the nodal displacements obtained by a finite element computation on the singular solutions of Table 2.1. After that, the extraction of the stress intensity factors values is straightforward. Since the asymptotic solutions are valid only in a restricted zone near the crack tip, the nodal displacement is to be taken as close as possible to the front. Energetic method is used with displacement matching method to compensate the accuracy problem cause by saturated singularity. This method relies on the integration domain independence property of some class of integrals, hence allowing to use fields computed far from the crack front for the extraction of the stress intensity factors.

Table 2.1: Stress and displacement fields around a crack tip for mode I and mode II (linear elastic, isotropic material behavior) [1].

	Mode I	Mode II	T-stress
σ_{11}	$\frac{K_I}{\sqrt{2\pi r}}\cos\left(\frac{\theta}{2}\right)\left[1-\sin\left(\frac{\theta}{2}\right)\sin\left(\frac{3\theta}{2}\right)\right]$	$-\frac{K_{II}}{\sqrt{2\pi r}}\sin\left(\frac{\theta}{2}\right)\left[2+\cos\left(\frac{\theta}{2}\right)\cos\left(\frac{3\theta}{2}\right)\right]$	$+T$
σ_{22}	$\frac{K_I}{\sqrt{2\pi r}}\cos\left(\frac{\theta}{2}\right)\left[1+\sin\left(\frac{\theta}{2}\right)\sin\left(\frac{3\theta}{2}\right)\right]$	$+\frac{K_{II}}{\sqrt{2\pi r}}\sin\left(\frac{\theta}{2}\right)\cos\left(\frac{\theta}{2}\right)\cos\left(\frac{3\theta}{2}\right)$	
σ_{33}	$\frac{K_I}{\sqrt{2\pi r}}2\text{v}\cos\left(\frac{\theta}{2}\right)$	$-\frac{K_{II}}{\sqrt{2\pi r}}2\text{v}\cos\left(\frac{\theta}{2}\right)$	$+T_{33}$
σ_{12}	$\frac{K_I}{\sqrt{2\pi r}}\cos\left(\frac{\theta}{2}\right)\sin\left(\frac{\theta}{2}\right)\sin\left(\frac{3\theta}{2}\right)$	$+\frac{K_I}{\sqrt{2\pi r}}\cos\left(\frac{\theta}{2}\right)\left[1-\sin\left(\frac{\theta}{2}\right)\sin\left(\frac{3\theta}{2}\right)\right]$	
u_1	$\frac{K_I}{2\mu}\sqrt{\frac{r}{2\pi}}\cos\left(\frac{\theta}{2}\right)\left[k+1-2\sin^2\left(\frac{\theta}{2}\right)\right]$	$-\frac{K_{II}}{2\mu}\sqrt{\frac{r}{2\pi}}\sin\left(\frac{\theta}{2}\right)\left[k+1+2\cos^2\left(\frac{\theta}{2}\right)\right]$	
u_2	$\frac{K_{II}}{2\mu}\sqrt{\frac{r}{2\pi}}\sin\left(\frac{\theta}{2}\right)\left[k-1-2\cos^2\left(\frac{\theta}{2}\right)\right]$	$-\frac{K_{II}}{2\mu}\sqrt{\frac{r}{2\pi}}\cos\left(\frac{\theta}{2}\right)\left[k-1-2\sin^2\left(\frac{\theta}{2}\right)\right]$	

2.3.1 J-Integral

Interaction integral is a specific integral that keeps the interesting properties of the standard J integral such as the domain independence, and that enables the extraction of the stress intensity factors from the solution fields of a crack solicited in mixed mode loading.

Volume integral is used under 3D analysis for crack body and it was discussed by Florent Galland in his research for 3D fatigue crack growth [10]. The discussion considers a cracked body at an equilibrium state. Its stress and deformation tensors σ an ε ", as well as its displacement field u, fulfil the boundary conditions. A second state is then introduced. This additional state represents in fact an auxiliary, fictitious state and all its corresponding quantities will be denoted by the subscript aux. Applying the superposition principle, and assuming the Einstein summation convention, the J volume integral see Fig 2.2 of these two states writes:

$$J^{total} = -\int_V \left[W^{total}\delta_{kj} - (\sigma_{kj} + \sigma_{ij}{}^{aux})\frac{\partial(u_i + u_i{}^{aux})}{\partial_{xk}}\right]\frac{\partial q_k}{\partial x_j}dV \qquad (2.9)$$

where the total deformation energy is given by

$$W^{total} = \frac{1}{2}(\sigma_{ij} + \sigma_{ij}{}^{aux})(\varepsilon_{ij} + \varepsilon_{ij}{}^{aux}) \tag{2.10}$$

By reordering the terms, the total J integral can be expressed as the following sum:

$$J^{total} = J + J^{aux} + I$$

Rearrange,

$$I = J^{total} - J - J^{aux} \tag{2.11}$$

From 2.5 (Irwin's Expression)

$$I_{Ic} = K_{Ic}{}^{2}(\frac{1 - v^{2}}{E}) \tag{2.12}$$

Again, considering that the pointwise value of this integral does not vary significantly in the integration volume, the mode I stress intensity factor K_I at any point of the front enclosed in the integration domain is deduced as:

$$K_{Ic} = \frac{E}{2(1 - v^{2})} \frac{I_{Ic}}{\int_{C}^{\cdot} q(\zeta) d\zeta} \tag{2.13}$$

For isotropic, perfectly brittle, linear elastic materials, the J-integral can be directly related to the fracture toughness if the crack extends straight ahead with respect to its original orientation. For plane strain, under Mode I, II, III loading conditions, the relation is

$$J_c = G_c = K_c{}^{2}(\frac{1 - v^{2}}{E}) \tag{2.14}$$

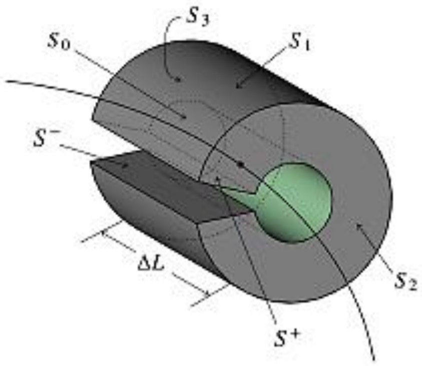


Figure 2.5**:** Volume Integration Domain Around a Tri-dimensional Crack Front **[10]**

2.4 Numerical methods for 3D crack growth analysis

In general, the numerical simulation of crack growth is resolved in three main stages see Fig 2.6. In a first step, the structural mechanics problem is solved, that is, the partial derivative equations modelling the mechanical behaviour of the cracked structure associated with the appropriate boundary conditions.

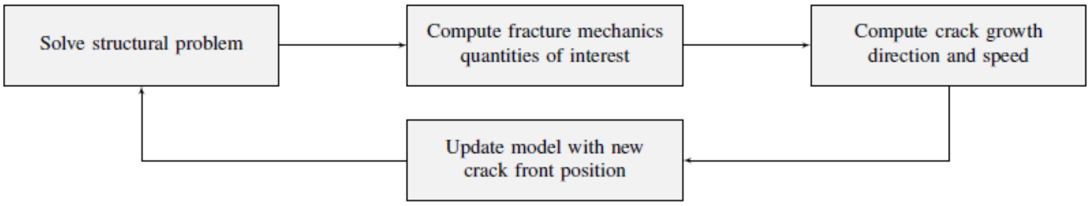


Figure 2.6**:** General strategy for the numerical simulation of crack propagation [10]

Finite element method (FEM) is a well-known numerical method for resolving structural mechanics problems as it can deal with very complex geometry, material behaviours and contact conditions. The size of the problem to solve depend on the mesh used to discretize the considered mechanical system. When dealing with problems of evolving geometries, such as crack growth, using a specific mesh morphing technique for the geometrical updates of the model allows discretizing several different problems with meshes of the same dimension. Hence the problems to solve are of the same dimension as well, providing a straightforward coupling with reduced basis and model reduction techniques in general. For this reason, it is the chosen numerical method for this work. All the associated developments and computations were performed in the ANSYS software, taking advantage of very powerful solver and elements.

Displacement method is one of the FEM which has been used over the years. Wong and EE Fun [1] which explain that stress intensity factor K is the parameter used in fracture mechanics to predict the stress intensity near a crack-tip caused by loading or residual stresses. It is a technique which involves a correlation of the finite element nodal point displacements with the crack-tip displacement. Chan [20] studied the computation of crack-tip stress intensity factors using finite element method. A mesh size effects study of an edge crack specimen had been simulated. Typically, the independent refinement in mesh size is incorporated in the crack-tip area. The method showed a good result in KI value calculation compared to the stress method and line integral method.

Alshoaibi [21] studied finite element program developed in elastic-plastic crack propagation simulation using displacement extrapolation method. An arrangement of quarter-point elements is constructed around the crack-tip to facilitate the prediction of crack growth based on the maximum normal stress criterion and to calculate stress intensity factors under plane stress and plane strain conditions. The adaptive mesh refinement around the crack-tip enables to maintain a good precision in the vicinity of the crack. Re-meshing technique is needed to avoid the distortion at the crack-tip and continuing with a new well-suited mesh to propagate through the inter-element in the mesh. The displacement extrapolation method used to calculate stress intensity factor is shown in Eq (2.15) and (2.16).

$$K_I = \frac{E}{3(1+v)(1+k)}\sqrt{\frac{2\pi}{L}}\left[4(v_b - v_d) - \frac{(v_c - v_e)}{2}\right] \tag{2.15}$$

$$K_I = \frac{E}{3(1+v)(1+k)}\sqrt{\frac{2\pi}{L}}\left[4(u_v - u_d) - \frac{(v_c - v_e)}{2}\right] \tag{2.16}$$

Where E is modulus of elasticity, υ is Poisson's Ratio and parameter κ = (3 - 4υ) for plane strain and κ = (3 - υ)/ (1 + υ) for plane stress. The displacement u and v are components in the x and y directions where the subscript refers to the respective node of a six-node triangular element as depicted in Fig 2.7. The element length L is also shown in the figure.

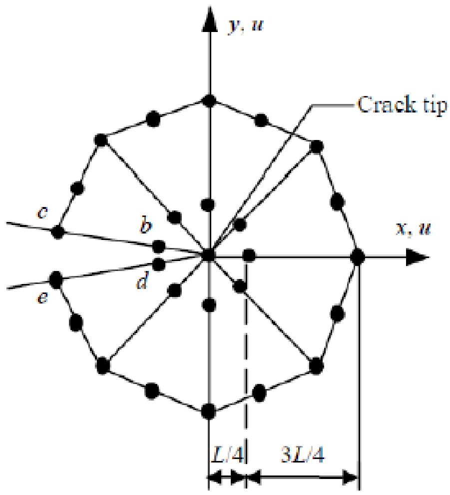


Figure 2.7**:** Quarter-point triangular elements around the crack-tip

In order to represent the field singularity correctly at the crack-tip, the singular elements have to be constructed. To obtain an optimal mesh, the adaptive re-meshing process is carried out by the posteriori stress error norm scheme. This scheme is based on adaptive Delaunay triangulation as mesh generator, which is obtained from the solution from the previous mesh. This study was done by [22] The strategy used to refine the mesh during the analysis process is adopted from Alshoaibi and Kamal [21]. The number of elements depends on the distributed nodes around the crack-tip as shown in Fig. 2.8.

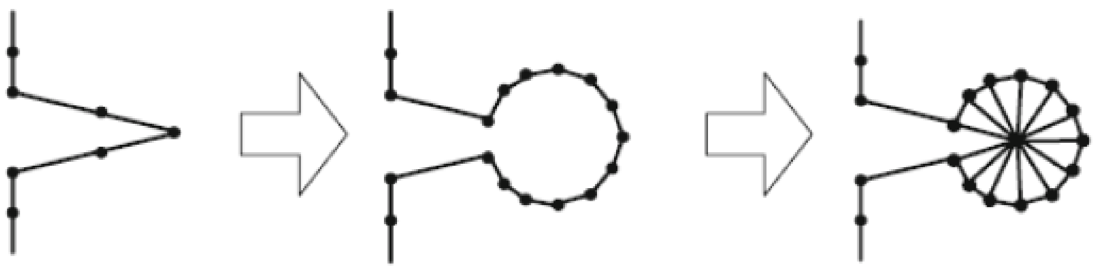


Figure 2.8**:** The cut and patch procedure of generating singular elements around a crack-tip [1].

2.5 Fatigue Crack Growth Overview

In fatigue loading, the specimen is applied with constant magnitude of load whereas fatigue loading the load is applied in cycles. Experiment have shown that the crack length, a is an exponential function of the number of cycles N. Proposed by Paris and Erdogan [4], the basic hypothesis of the crack modelling procedure is considering that the crack front advances by fatigue which follow the Paris law.

$$\frac{da}{dN} = C\,(\Delta K)^m \tag{2.17}$$

Paris's law take place at region II as shown in Fig 2.9. The crack growth behaves linearly until region III where the fatigue limit is reached, and fracture may take place further beyond. The differences between these two regions of a crack's life are very important when determining the fatigue life of a specimen. It was realized that crack extension takes place due to stress concentration at the crack tip and due to failure of material during cyclic loading; an effort has been made to relate the crack growth with stress intensity factor "K" at the crack tip.

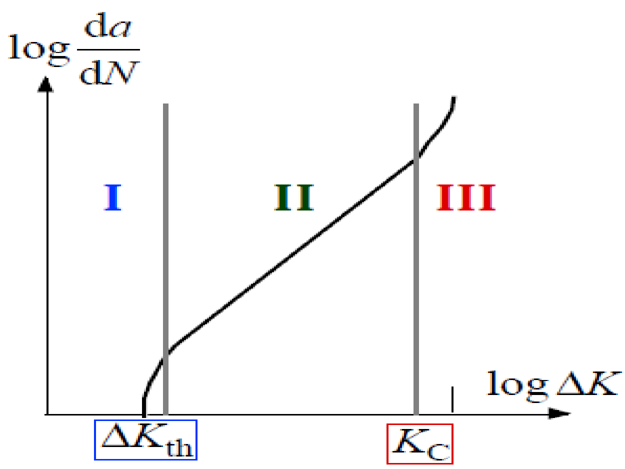


Figure 2.9**:** Variation of crack-growth rate vs ΔK for metals

Crack growth is very slow until the final stage in the fatigue life, where a relative short number of cycles will result in fast crack growth leading to failure. The initial fatigue crack length a_i seem to be a very important parameter for the fatigue life N_f.

Shripad T. Revankar [3] in his research, has discussed on the stages that exist in the crack growth as it starts at incubation period before the crack initiation stage (see fig 2.10). Once the crack initiates, it propagates at different rates a slow growth followed by rapid growth leading to failure. Initiation time for the crack can be very short, if a surface defect due to use and/or abuse, manufacturer's design, welding is present, or it can be longer; a crack can take millions of cycles of stress, even years, in some situations. The crack will grow or extend if the component is subject to a tensile or pulling stress.

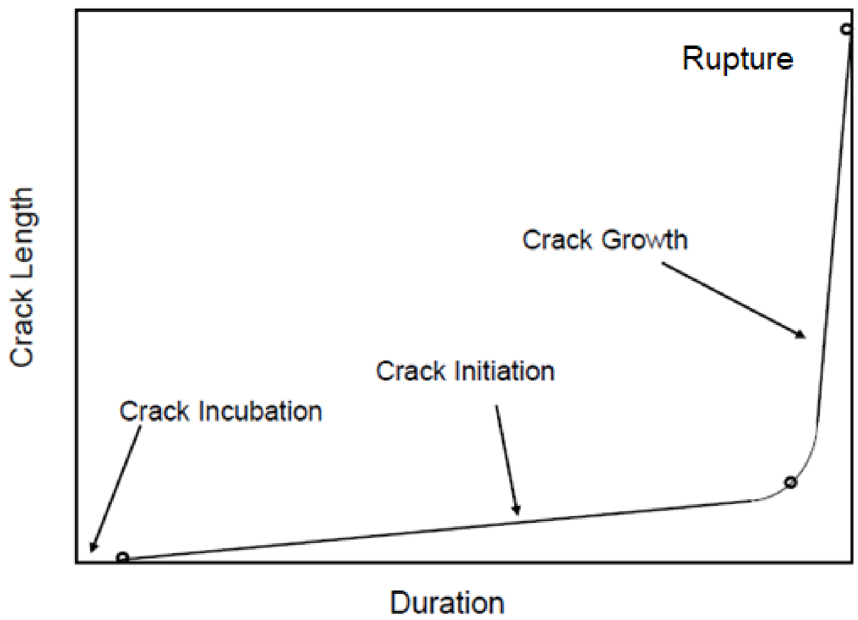


Figure 2.10**:** Crack Length as Function of Time

Crack growth process was explained in [23]. For an initially undamaged material, it takes N_i cycles to initiate a crack by dislocation movement. At this fatigue crack initiation life, the initial crack has been formed, but in most cases, it is so small that it cannot be detected. In this stage I, the crack propagation rate is very slow, typically < 0.25nm/cycle. After N_i cycles, in stage II of crack growth, crack propagation is faster, typically µm's per cycle. The crack growth is triggered by tensile stresses and involves plastic slip on multiple slip planes at the crack front, resulting in striations.

After a large number of cycles the crack reaches a length a1 see Fig 2.11, which can be detected by non-destructive techniques. The crack growth is now much faster and after the fatigue life N_f is reached its crack length is a_f and after a few more cycles a_c the critical crack length is reached, and failure occurs. For higher loading amplitudes, the crack growth will be faster. After N cycles, the cycle to go until failure at N_f, is indicated as N_r. The rest-life is the ratio of N_f and N_r.

$$\frac{N_r}{N_f} = 1 - \frac{N}{N_f} \tag{2.18}$$

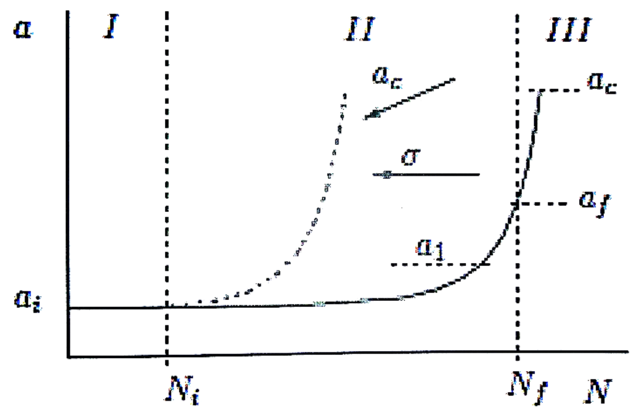


Figure 2.11**:** Crack Length vs Number of Cycle Graph [24**]**

2.5.1 Fatigue Life

In the study of diesel engine system design by Qianfan Xin, he provided a brief explanation regarding fatigue life [25].Fatigue life is defined as the number of loading (stress) cycles of a specified character that a specimen sustains before failure of a specified nature occurs. Fatigue life is affected by cyclic stresses, residual stresses, material properties, internal defects, grain size, temperature, design geometry, surface quality, oxidation, corrosion, etc. The fatigue life of a component under the following different fatigue mechanisms can be ranked from low to high as: thermal shock, high temperature LCF, low temperature LCF, and HCF. In the assessment of the risk of fatigue failure, it may be assumed that the component is safe for an infinite number of cycles if it does not show failures after more than ten million cycles. The total fatigue life is equal to the life of crack formation and crack propagation [25].

Fatigue life is dependent on the cycle history of the loading magnitude since crack initiation requires a larger stress than crack propagation. The fatigue life of the component can be determined by the strain, stress, or energy approach. Fatigue is a very complex process affected by many factors. It is usually more effective to use a macro phenomenological method to model the effects of fatigue mechanisms on fatigue life rather than using a microscopic approach [25].

To predict the fatigue life of structures, there several methods used over the years such as Basquin form in the stress-based method and the Coffin-Manson relationship in the strain-based method. To predict the fatigue life, crack growth model has been proposed, which relate growth rate da/dN to load amplitude or maximum load, which can be expressed in the SIF, K because we assume to be in the high cycle fatigue regime, where stresses are low considered for the material used. So, it is possible to use Eq (1.1) to predict fatigue lifetimes. From expression (2.2) letting the initial crack size in the material be a_o, then the number of cycles N_f to cause this crack to grow to a size a_f at which would occur in one application of stress can be obtained by integrating Eq (1.1).

$$N_f = \frac{2}{C(Y.S_R)^m \pi^{\frac{m}{2}}(2-m)}\left(a_f^{1-\frac{m}{2}} - a_o^{1-\frac{m}{2}}\right) \qquad (2.19)$$

Assuming that $m \neq 2$ and Y does not vary with a.

So, fatigue crack growth of metal materials has been a research focus in engineering field for a long time. It’s important for guiding safety production and preventing major accidents to find crack phenomenon, study the prerequisite of initiation and unstable propagation of cracks, fatigue limit and life, understand the progress of stable propagation and control the propagation of crack.

2.6 Direction of Crack Propagation

Many models are available for predicting the direction of crack extension based on either stress, strain, energy, or any combination of these. Appropriate models should be selected based on material and loading condition. In ANSYS we have to obtain maximum value of stress intensity factor (SIF) in mode I and II as an input in determine the crack propagation angle. There are three (3) ways the crack can extend: at a direction normal to the direction of the maximum tangential stress, based on Maximum Energy Release Rate (MERR) criterion, and $K_{II} = 0$ criterion. It is set defined by default the crack propagates normal to the direction of the maximum tangential stress. The one model presented here were from literature [11]

$$\theta_c = arcos\left(\frac{3{K_{II}}^2 + \sqrt{{K_I}^4 + 8{K_I}^2{K_{II}}^2}}{8{K_I}^2 + 9{K_{II}}^2}\right) \tag{2.20}$$

Where θ_c is the angle that will follow the crack for each of the crack increment. θ_c is measured with respect to a local polar coordinate system with its origin at the crack tip and aligned with direction of the existing crack. The sign convection is such that $\theta_c < 0$ when $K_{II} > 0$ and vice versa. Once the crack growth orientation is determined, a propagation increment Δa is added to the existing crack geometry and the analysis procedure is repeated.

Maximum Circumferential Stress Criterion (MCSC) was proposed by Erdogan and Sih [7] which states that the crack extension will happen radially from the crack tip and perpendicular to the maximum applied tensile load [23]. These two criterions are met when the circumferential stress, $\sigma_{\theta\theta}$, is maximized and the shear stress, $\tau_{r\theta}$ is zero. It is a local approach since it is based on local stress fields around the crack front. The circumferential stress field at the evaluation points around the tips of the crack front can be formulated in terms of the stress intensity factors, K [7].

$$\sigma_{\theta\theta} = \frac{1}{\sqrt{2\pi r}}\frac{K_I}{4}\left(3\cos\frac{\theta}{3} - \cos\frac{3\theta}{2}\right) + \frac{K_{II}}{4}\left(-3\sin\frac{\theta}{2} - 3\sin\frac{3\theta}{2}\right) \tag{2.21}$$