Determination of Stress Intensity Factor for Arc-shaped Tension Specimen by Using Finite Element Analysis

By

Nurulain bte Marjono

Dissertation submitted in partial fulfilment of
the requirements for the
Bachelor of Engineering (Hons)
(Mechanical Engineering)

DECEMBER 2008

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Determination of Stress Intensity Factor for Arc-shaped Tension Specimen by Using Finite Element Analysis

by

Nurulain bte Marjono

A project dissertation submitted to the
Mechanical Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfilment of the requirement for the
BACHELOR OF ENGINEERING (Hons)
(MECHANICAL ENGINEERING)

Approved by,

(Dr. Khairul Fuad)

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

July 2008

CERTIFICATION OF ORIGINALITY

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

NURULAIN BTE MARJONO

ABSTRACT

Fracture mechanics is a method for predicting failure of a structure containing a crack which divided into linear elastic fracture mechanics and nonlinear fracture mechanics. The most important parameter in fracture mechanics, Stress Intensity Factor (SIF), K, is used to more accurately predict the stress intensity near the tip of a crack caused by a remote load or residual stresses. Fracture Toughness, K_C is a property which describes the ability of a material containing a crack to resist fracture when load is applied to the material. Fracture occurs when $K_I \geq K_C$.

The crack tip parameter K is useful tool to calculate the crack growth up to failure for different crack geometry and loading conditions. The advancement of finite element source codes allows the crack tip parameters (SIF and stress distribution) to be computed nearly accurate without being overly dependent on purely experimental work which is time and cost consuming.

The aim of the present work is to investigate the SIF of Mode-I crack type specimen used in general applications. The SIF for different loading value and varied parameter which obtained from ANSYS software will be analyzed and compared with the calculation method. The modeling work is carried out using Arc-shaped Tension specimens.

In this study, the Arc-shaped Tension specimen was modeled to carry aluminum properties, which exhibit linear-elastic behavior under stresses. This specimen was modeled to be plane-strain and finite-width exerted with opening mode loading (Mode-I). The ANSYS finite element software is employed to model this specimen and obtain the value of SIF to compare and validate through theoretical calculation.

The modeling results have exhibit proper linear-elastic behavior and stress distribution. From the numerical data and theoretical considerations, it shown that the Mode-I SIF are dependent of the loading applied to the specimens. All of all, the results show that increasing the load applied, ring segment distance, crack length or decreasing the thickness can significantly increase the SIF.

ACKNOWLEDGEMENT

I would like to take this opportunity to acknowledge and thank everyone that has given me all the supports and guidance throughout the whole period of completing the final year project. Firstly, many thanks to the university and the Final Year Project coordinators that have coordinated and made the necessary arrangements for this study.

I must also acknowledge the endless help and support received from my supervisor, Dr. Khairul Fuad throughout the whole period of completing the final year project. His guidance and advices are very much appreciated. Apart from that, many thanks to the graduate assistances, Mr. Irfan and Mr. Julendra for assisting me in the completion of the modelling in ANSYS software.

I would also like to express gratitude to my examiners for their comments and evaluation for me to improve my progress for this study. Last but not least, many thanks to my fellow colleagues for their help and ideas throughout the completion of this study. Thank you all.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATIONS AND ACRONYMS

σ	Remote stress applied to component
a	Crack length
β	Correction factor that depends on specimen and crack geometry
r	Crack tip radius
θ	Degree of crack tip from normal axis
P	Load applied
B	Specimen thickness
X	Loading hole offset
W	Specimen width
r_1/r_2	Ratio of inner to outer radii
α	Fatigue pre-crack length
ASTM	American Society for Testing Materials
CAE	Computer Aided Engineering
FEA	Finite Element Analysis
FEM	Finite Element Method
LEFM	Linear Elastic Fracture Mechanics
SIF	Stress Intensity Factor

CHAPTER 1
INTRODUCTION

1.1 Background of Study

Fracture in science and technology is understood as the total or partial separation of an originally intact body or structure. Fracture mechanics is a method for predicting failure of a structure containing a crack by approaches requires that an initial crack size be known or assumed. Fracture mechanics may be classified from different points of view. Usually it is divided into linear elastic fracture mechanics and nonlinear fracture mechanics.

Linear Elastic Fracture Mechanics (LEFM) first assumes that the material is isotropic and linear elastic. Based on the assumption, the stress field near the crack tip is calculated using the theory of elasticity. When the stresses near the crack tip exceed the material fracture toughness, the crack will grow. In LEFM, most formulas are derived for either plane stresses or plane strains, associated with the three basic modes of loadings on a cracked body; opening, sliding and tearing.

Stress Intensity Factor (SIF), K, defines the magnitude of the local stresses around the crack tip [4]. SIF is used to more accurately predict the stress state or stress intensity near the tip of a crack caused by a remote load or residual stresses [5]. Fracture Toughness, K_C is a property which describes the ability of a material containing a crack to resist fracture when load is applied to the material [6]. The subscripts I, II, and III denote the mode of the crack. Fracture occurs when $K_I \geq K_C$.

Finite element analysis (FEA) is a computer simulation technique used in engineering analysis by using a numerical technique called the finite element method (FEM). Development of the FEM in structural analysis can help one to study on the crack behavior and propagation, and to obtain the value of SIF.

1.2 Problem Statement

The characterization of crack tip stresses when applying the LEFM concept requires the use of a SIF parameter, *K*, which can address important cracking problems in many industries where the safety of people and structure is of utmost concern. The crack tip parameter *K* is useful tool to calculate the crack growth up to failure for different crack geometry and loading conditions when the crack tip plasticity is confined to a small region at the crack tip.

The tensile opening mode (Mode-I) type of failure represents the most frequent type of separation that engineers design against and be prevented. Theoretically different loading modes will give different value of SIF. The experimental analysis for crack specimen is tedious and expensive. Therefore, this project will aim to design test specimens from Mode-I type of failure using ANSYS software and compare the result with theoretical method.

1.3 Objective and Scope of Study

Upon completing the project, a few objectives need to be achieved. The objectives of study are as follows:

- To analyze and determine the Mode-I SIF for Arc-shaped Tension specimen by using FEA with different loading modes.
- To compare the SIF values obtain by using calculation method and FEA for varied parameters.

The scope of study is divided into 4 major parts as follows:

- To do literature review of fracture mechanics basis, Mode-I crack specimen and the factors affected SIF.
- To model the Arc-shaped Tension specimen using FEM. The specimen will be modeled using suitable scale and specifications.
- To apply appropriate analytical analysis as a result comparison.
- To analyze the result obtained from the modeling.

CHAPTER 2
LITERATURE REVIEW AND THEORY

2.1 Literature Review

2.1.1 Fracture Mechanics

Fracture mechanics is a method for predicting failure of a structure containing a crack by approaches requires that an initial crack size be known or assumed. In real structure of material, cracks or other defects of different size which possible evolve to cracks are virtually always present.

To analyze the relationship among stresses, cracks, and fracture toughness, Fracture Mechanics was introduced. Since cracks can lower the strength of the structure beyond that due to loss of load-bearing area a material property, above and beyond conventional strength, is needed to describe the fracture resistance of engineering materials. This is the reason for the need for fracture mechanics; the evaluation of the strength of cracked structures.

The existence of cracks tends to lower the experimental value of fracture strength. That explains why the theoretical value of fracture strength always higher than the experimental value. These cracks may be affected by material defects (dislocation, impurities), discontinuities in assembly and/or design (sharp corners, grooves, nicks, and voids), harsh environments (thermal stress, corrosion) and damages in service (impact, fatigue, unexpected loads). Most microscopic cracks are arrested inside the material but it takes one run-away crack to destroy the whole structure.

The development of fracture mechanics is driven to a large extent by the ambition to prevent failure of technical constructions and components. Therefore, fracture mechanics is used as a design tool in all fields where fracture and an accompanying failure of a component with serious, or in the worst case, catastrophic consequences must be prevented.

2.1.2 Linear Elastic Fracture Mechanics

Linear Elastic Fracture Mechanics (LEFM) forms the basis of fracture mechanics. Linear-elastic materials generally have lower fracture toughness which linearly proportional to the failure stress. At that level, the fracture will be brittle.

The objective of LEFM is to predict the critical loads that will cause a crack in a solid to grow. LEFM can deal with only limited crack tip plasticity, i.e. the plastic zone must remain small compared to the crack size and the cracked body as a whole must still behave in an approximately elastic manner.

There are generally three modes of loading, which involve different crack surface displacements as shown in Figure 2.1. The three modes are:

Mode-I: Opening or tensile mode (the crack faces are pulled apart)

Mode-II: Sliding or in-plane shear (the crack surfaces slide over each other)

Mode-III: Tearing or anti-plane shear (the crack surfaces move parallel to the leading edge of the crack relative to each other.

Mode-I is the predominant loading mode in most engineering applications. Similar treatments can readily be extended to Modes-II and III.

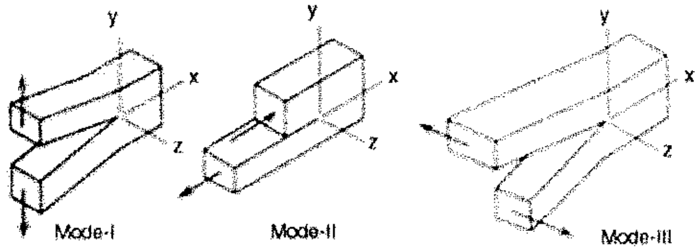


Figure 2.1. Three loading modes.

2.1.3 Stress Intensity Factor

Stress Intensity Factor (SIF), K, is used in fracture mechanics to more accurately predict the stress state (stress intensity) near the tip of a crack caused by a remote load or residual stresses. When this stress state becomes critical a small crack grows (extends) and the material fails. The load at which this failure occurs is referred to as the fracture strength. The experimental fracture strength of solid materials is 10 to 1000 times below the theoretical strength values, where tiny internal and external surface cracks create higher stresses near these cracks, hence lowering the theoretical value of strength.

The SIF, K, defines the magnitude of the local stresses around the crack tip. This factor depends on loading, crack size, crack shape, and geometric boundaries. The subscript I arises because of the different ways of loading a material to enable a crack to propagate. It refers to loading via Mode-I which is the most common form of loading. General equation for SIF with refers to Hertzberg [4] is:

$$K_I = \beta\sigma\sqrt{\pi a} \tag{2.1}$$

As the SIF reaches a critical value (K_C), unstable fracture occurs. This critical value of the SIF is known as the fracture toughness of the material. The fracture toughness can be considered the limiting value of stress intensity just as the yield stress might be considered the limiting value of applied stress.

2.1.4 Fracture Toughness

Fracture Toughness, K_C is defined as the resistance of the material to unstable crack growth in a non-corrosive environment. This parameter characterizes the intensity of the stress field at the locality of the crack tip when unstable cracking takes places. The plane strain fracture toughness, K_I is believed to be crack length and thickness independent; however, it can vary as the function of temperature, type of material and strain rate.

Fracture occurs when $K_I \geq K_C$. For the special case of plane strain deformation, K_C becomes K_{IC} and is considered a material property. As stated by Hertzberg [4] the fracture toughness can be defined in terms of the stress intensity factor, K, but at a critical stress state as:

$$K_{IC} = \beta\sigma_C\sqrt{\pi a} \tag{2.2}$$

Where: σ_C = remote stress applied to component

a = crack length

β = correction factor that depends on specimen and crack geometry

2.2 Theory

2.2.1 Stress Analysis at Crack Tip

Based on linear elasticity theories, the stress field near a crack tip is a function of the location, the loading condition, and the geometry of the specimen or object. In practice, mostly the SIF is calculated based on the stress field at the crack tip and compare it against the known fracture toughness of the material. According to Janssen et al [1] the crack tip stress field is a function of the location, loading, and geometry:

$$\sigma_{ij}^{Tip} \equiv \sigma_{ij}^{Tip}(Location, Loading, Geometry)$$
$$\equiv \sigma_{ij}^{Tip}(r, \theta, K) \tag{2.3}$$

Where location can be represented by r and θ using the polar coordinate system whereas the loading and geometry terms can be grouped into a single parameter K, called the ***stress intensity factor.***

$$K \equiv K(\sigma^{Loading}, Geometry) \tag{2.4}$$

For Mode-I loading, linear elastic bodies will bear a stress distribution as show in Figure 2.2 that can be characterized by the stress equation:

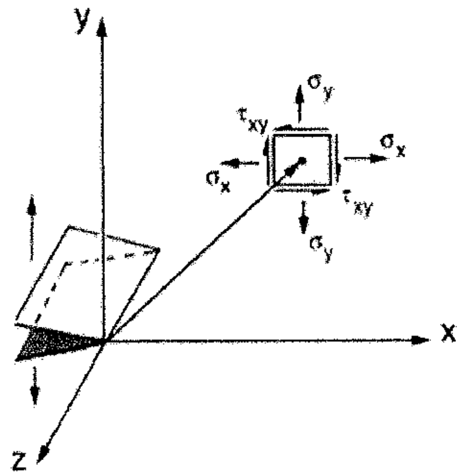


Figure 2.2. Distribution of stresses in vicinity of crack tip.

$$\sigma_{xx} = \frac{K_1}{\sqrt{2\pi r}} \cos\left(\frac{\theta}{2}\right)\left[1 - \sin\left(\frac{\theta}{2}\right)\sin\left(\frac{3\theta}{2}\right)\right] \tag{2.5}$$

$$\sigma_{yy} = \frac{K_1}{\sqrt{2\pi r}} \cos\left(\frac{\theta}{2}\right)\left[1 + \sin\left(\frac{\theta}{2}\right)\sin\left(\frac{3\theta}{2}\right)\right] \tag{2.6}$$

$$\sigma_{xy} = \begin{cases} 0 & (planestress) \\ v(\sigma_{xx} + \sigma_{yy} & (planestrain) \end{cases} \tag{2.7}$$

$$\tau_{xy} = \frac{K_1}{\sqrt{2\pi r}} \cos\left(\frac{\theta}{2}\right)\sin\left(\frac{\theta}{2}\right)\sin\left(\frac{3\theta}{2}\right) \tag{2.8}$$

$$\tau_{yz} = 0 \tag{2.9}$$

$$\tau_{zx} = 0 \tag{2.10}$$

These expressions shows that the stress components at the crack tip tend to infinity at the crack tip (r=0), a so-called $1/\sqrt{r}$ singularity. The stress fields near a crack tip of an isotropic linear elastic material can be expressed as a product of $1/\sqrt{r}$ and a function of θ with a scaling factor K.

CHAPTER 3
METHODOLOGY

3.1 Project Identification

The procedure of the project implementation is illustrated as Figure 3.1. The software required is AutoCAD and ANSYS.

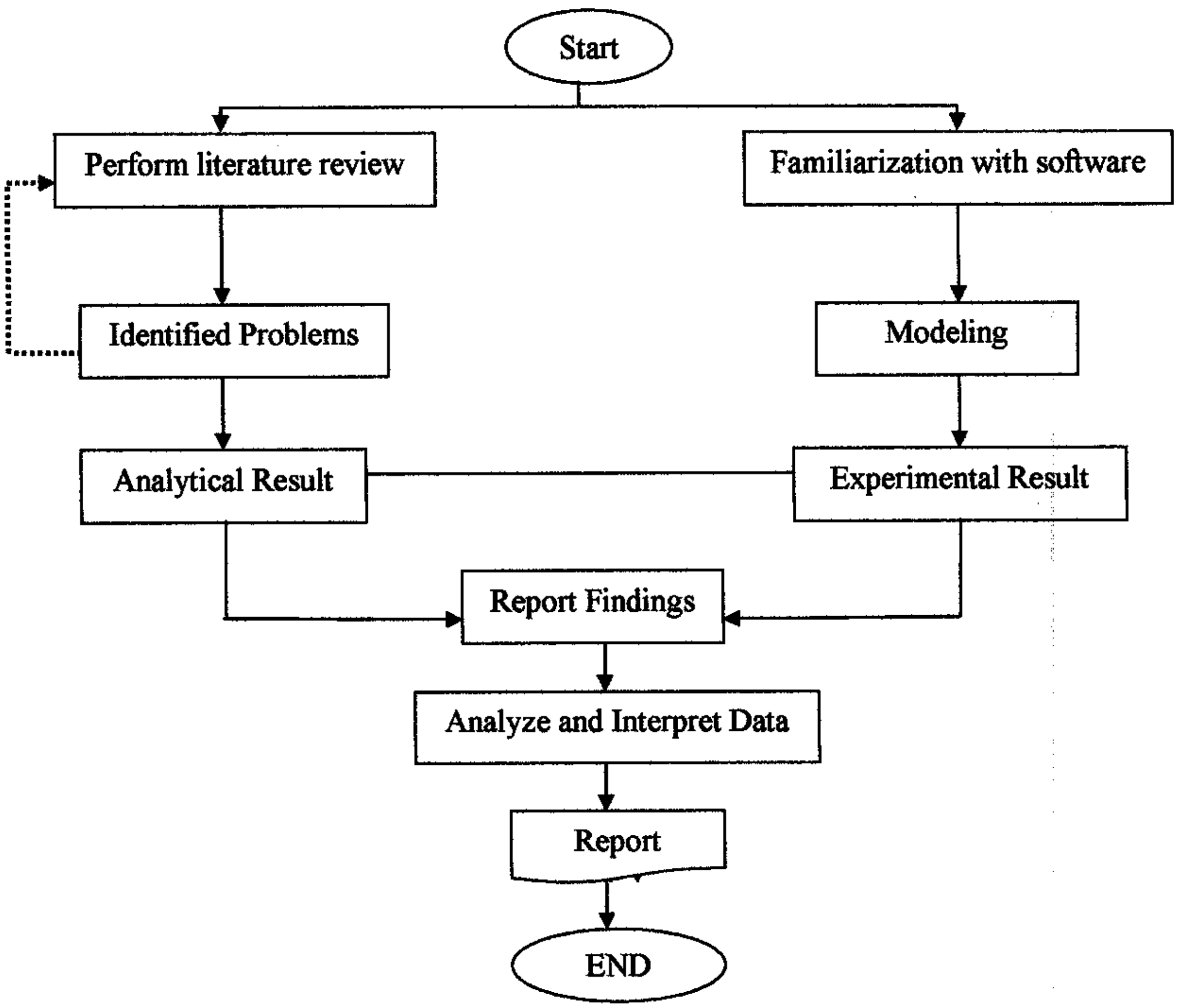


Figure 3.1. Schematic flow diagram of the project.

ANSYS software has been chosen to evaluate and determine the SIF. ANSYS has analysis procedures which are Preprocessor, Solution, and Post Processor. Preprocessor contain the command needed to create the finite element model such as analysis and element type, material properties, modeling and mesh generation. The solution and Post Processor are the steps to apply the boundary condition, loading and get the result from ANSYS analysis. The general procedures are shown in the flow diagram in Figure 3.2.

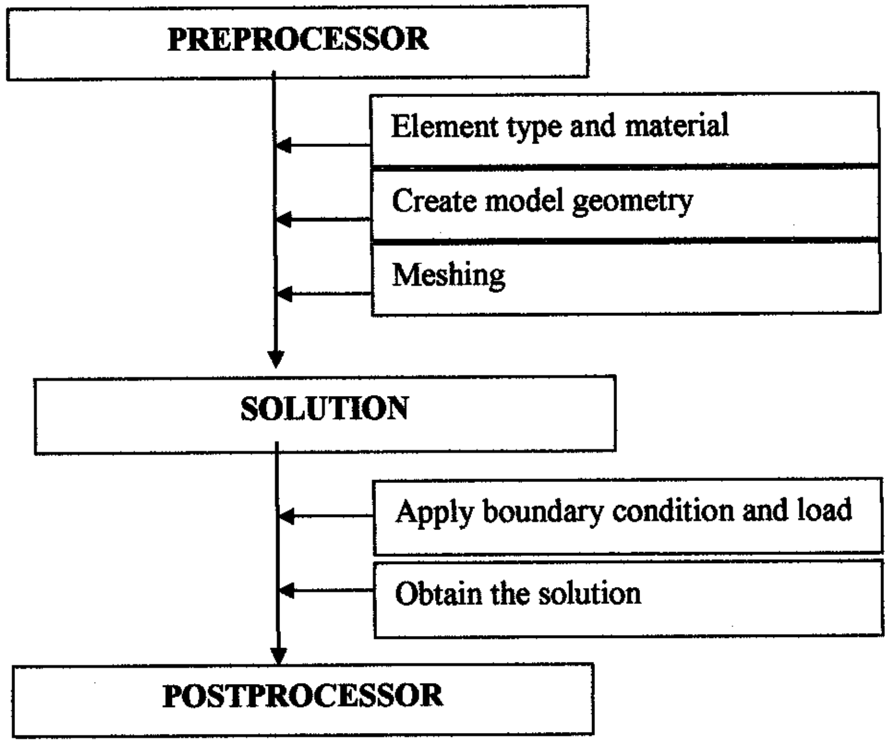


Figure 3.2. Schematic flow diagram ANSYS.

3.2 Arc-shaped Tension Specimen and Material Identification

The specimen is Arc-shaped Tension specimen. The Arc-shaped Tension specimen is one of the most well defined specimens for evaluation of the Mode-I fracture toughness. The specimen represent a configuration that ca be cut from a ring. It is a single edge-notched and fatigue cracked ring segment loaded in tension. The arc-shaped tension specimen is intended to measure the fracture toughness so that the normal to the crack plane is in the circumferential direction and the direction of crack propagation is in the radial direction. The general proportion of design of the specimen is shown in Figure 3.3.

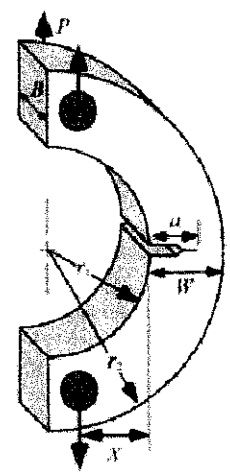


Figure 3.3. Arc-shaped Tension Specimen.

The material that has been used is Aluminum 7075-T6 Alloy. Aluminum is a metal that exhibits linear-elastic behavior and experience brittle fracture. In aluminum alloy, brittle compound such as oxides, sulfides, and silicates may be encountered, either as nonmetallic inclusions in alloys or as surface coatings. The material properties are as Table 3.1.

Table 3.1. Material Properties of Aluminum 7075-T6 Alloy.

Properties	**Value**
Modulus of Elasticity, E	69 GPa
Poisson's Ratio, v	0.33
Yield Strength	495 MPa
Critical Stress Intensity Factor, K_I	34 MPa.m$^{1/2}$

3.3 Theoretical Expression

To determine the stress intensity factor, theoretical expression in equation (3.1) and (3.2) are used with refer to ASTM E-399:

$$K_I = \frac{P}{BW^{1/2}}\left[3\frac{X}{W} + 1.9 + 1.1\frac{a}{W}\right]\left[1 + 0.25\left(1 - \frac{a}{W}\right)^2\left(1 - \frac{r_1}{r_2}\right)\right] f\left(\frac{a}{W}\right) \quad (3.1)$$

Where:

$$f\left(\frac{a}{W}\right) = \frac{\left(\frac{a}{W}\right)^{1/2}\left[3.74 - 6.30\left(\frac{a}{W}\right)^2 - 2.43\left(\frac{a}{W}\right)^3\right]}{\left(1 - \frac{a}{W}\right)^{3/2}} \quad (3.2)$$

Where:

P = load applied.

B = specimen thickness.

X = loading hole offset.

W = specimen width.

a = crack length.

r_1/r_2 = ratio of inner to outer radii.

3.4 Finite Element Modeling

As recommend by ANSYS tutorial, half of the specimen had been modeled to take advantage of the symmetrical geometry. Omitting the symmetrical half of the entire model reduces the complexity of the model. Generally, the most important part of a crack model is the crack tip itself. The element mesh around the crack tip has to have collapsed quadrilateral elements, so that the collapsed nodes have the same geometric position at the crack tip. Besides the crack tip, the remainder of the model is meshed via free mesh. The two-dimensional surface was meshed and the extruded to the desired specimen thickness to produce the three dimensional model.

The model was loaded with forces at nodes. The surface of symmetry was constrained from deformation. Solution of the model was also conducted at steady-state.

3.4.1 Three-dimensional Modeling Procedure

Before the modeling work was initiated, preliminary preparation was made to determine the dimensions of the model and the coordinates of each corners and edge of the model.

Key points were created in the active coordinate system. The key points define the important edges and points of the model. Based on the key points, the cross sectional area of the model was then created.

The stress concentration point was first identified and created in the model. The command is as follow:

Main Menu > Preprocessor > Meshing > Size Cntrls > Concetrat KPs – Create

The choice of the element size is important to ensure successful meshing. The recommended number of elements around the crack tip is 10 to 12, producing elements with approximately 30° angles. The number of meshing segments on the sides of the model was the specified. The two-dimensional model was then meshed before extruded to produce three-dimensional mesh. Prior to the extrusion, the two-dimensional mesh was prepared to be deleted aster the extrusion was completed. This was very crucial part to ensure that the model behaved properly when loaded. The importance and consequence of this modeling step is further elaborated in Chapter 4.

CHAPTER 4
RESULT AND DISCUSSION

4.1 Theoretical Result

The SIF for the main model of Arc-shaped Tension specimen resulting from Equation (3.1) and (3.2) are listed in Table 4.1:

Table 4.1. Calculation result for main model.

P (N)	*a* (m)	*W* (m)	f(*a/W*)	K_{1c} Mpa√m
10000	0.025	0.05	3.7325	27.19793
10500	0.025	0.05	3.7325	28.55783
11000	0.025	0.05	3.7325	29.91772
11500	0.025	0.05	3.7325	31.27762
12000	0.025	0.05	3.7325	32.63752

4.2 Specification of the Model

The modeling work involved in this project was based on the reference material obtained from American Society for Testing Materials (ASTM) E399 Standard Test Method for Plane-Strain Fracture Toughness of Metallic Materials. This is to ensure the procedural and dimensional accuracy of the model are preserved.

To ensure that the model meet the plane-strain requirement, the thickness and the crack length were calculated as follow:

$$\text{Crack Tip, Thickness} \geq 2.5\left(\frac{K_{1C}}{\sigma_{YS}}\right)^2 \qquad (4.1)$$

Where K_{1C} and σ_{YS} are the fracture toughness and the yield strength respectively. Other parameter like the notch dimensions, the position of the loading holes, the width of the model and also, the length of the fatigue pre-crack were all determined from the ASTM E399 standard (refer Appendix A). Note that all of the dimensions are function of *W* (width). *W*=25mm were chosen as the benchmark since $25mm \approx 1inch$.The dimensions outlined in the ASTM E399 are for finite-width specimens. Figure 4.1 shows the drawing of the model using AutoCAD. The critical dimensions for this model were fixed as shown in Table 4.2. These dimensions will ensure that the model was plane-strain, finite-width, and suitable for fracture mechanics analysis.

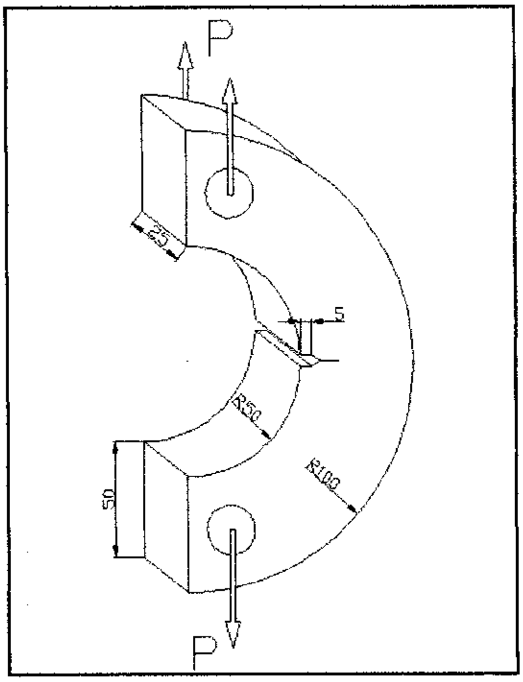


Figure 4.1. Drawing of Arc-shaped Tension model using AutoCAD.

Table 4.2. Critical Dimension of a Crack Specimen.

Dimensions	Details
Crack Length, *a*	25mm
Fatigue Pre-crack Length, α	5mm
Specimen Width, *W*	50mm
Specimen Thickness, *B*	25mm
Notch Type	Straight Through Notch

4.3 ANSYS – Meshing Technique

Meshing is the next step in creating a finite element model to divide the geometry into nodes and element. The ANSYS program can automatically generate the nodes and elements provided the specified *element attributes* and the *element size*:

1. The element attributes include element type(s), real constant and material properties.
2. The element size controls the fineness of the mesh. The smaller the element size, the finer the mesh.

Basically mesh generation has two types which are *free meshing* and *mapped meshing*. A free mesh has no restrictions in terms of element shapes, and has no specified pattern applied to it. Compared to a free mesh, a mapped mesh is restricted in terms of the element shape it contains and the pattern of the mesh. According to the model geometry, free mesh is more suitable for this model.

4.3.1 Default Mesh

The simplest meshing technique is the default mesh. By applying the auto mesh generation, for area meshing, it can consist of only quadrilateral elements, only triangular elements, or a mixture of the two. For volume meshing, a free mesh is usually restricted to tetrahedral elements. Two approaches have been identified in order to solve the problem. First, the mesh generation is done at the area of two-dimensional model before extruded to three-dimension (Figure 4.1). Second approach is the specimen modeled in two-dimension and extruded into three-dimension geometry, and then the mesh generation is applied to the volume (Figure 4.2).

The element was distribute almost evenly at the outer surface of the volume but the other element inside the body and at the crack tip is not appropriately distributed. Hence ANSYS failed to gain the solution due to the existence of unstable of element and discontinuity within the finite element. It can be concluded that default mesh is only suitable for simple model and configuration.

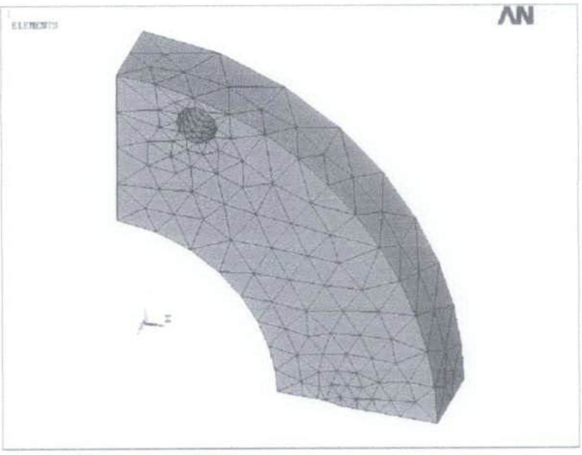


Figure 4.2. Default Mesh by Volume.

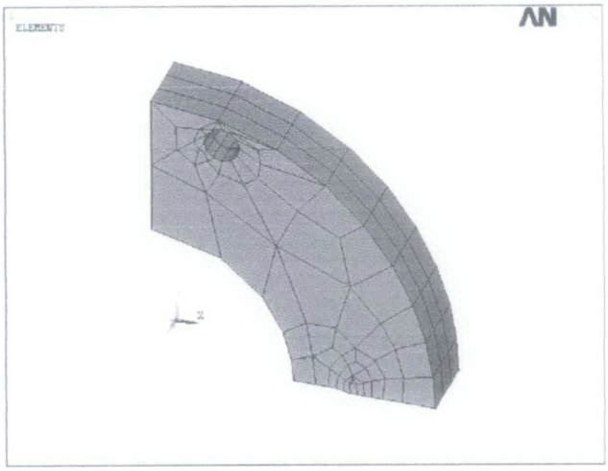


Figure 4.3. Default Mesh by Area.

4.3.2 Mesh Generation for Crack Model

For typical problems, the most efficient mesh design for the crack tip region has proven to be the "spider-web" configuration, which consists of concentric rings of quadrilateral elements that are focused toward the crack tip. The singular elements in the innermost ring are degenerate to triangles. Since the crack tip region contain steep stress and strain gradients, the mesh refinement should be greatest at the crack tip. The spider-web design facilities a smooth transition from a fine mesh at a crack tip to a coarser remote from the tip. Figure 4.3 shows an example of spider-web meshing configuration.

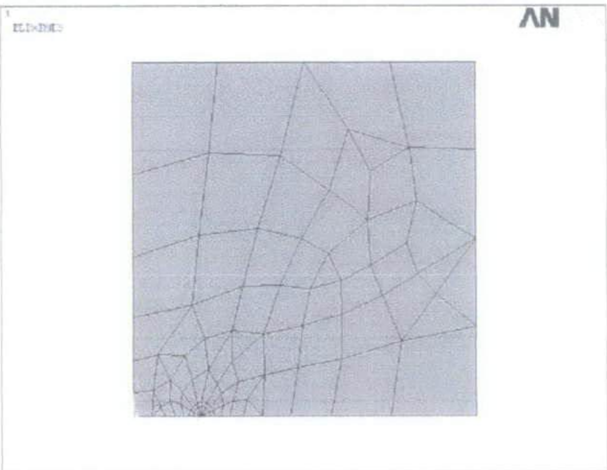


Figure 4.4. Example of "Spider-web" Mesh configuration.

4.3.3 Predefine Mesh – Triangular Element and Singular Element

The elements used are PLANE2 and SOLID95. PLANE2 is 6-node triangular element. The element has a quadratic displacement behavior and is well suited to model unequal meshes while SOLID95 is 20-nodes 3-D solid element which can tolerate irregular shapes without as much loss of accuracy especially for the Arc-shaped Tension specimen.

To pick up the singularity in the strain, the crack faces should be coincident, and the elements around the crack tip (or crack front) should be quadratic. The circular around the crack tip is the generated singular element. This meshing technique improves the stress distribution at the crack tip as shown in Figure 4.4.

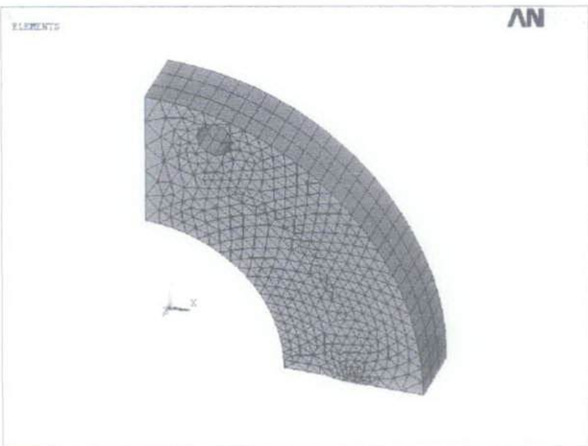


Figure 4.5. Predefine Mesh- Example of Triangular element and Singular element.

4.3.4 Predefine Mesh – quadrilateral Element and Singular Element

Another meshing technique is using quadrilateral element. Basically the method is similar with triangular element except PLANE82 is applied instead of PLANE2. PLANE82 is an eight-node quadrilateral element offers more accuracy when modeling problems with curved boundaries.

To get more accurate result, the first row of elements around the crack front should be singular elements. The generated singular element is the circular around the crack tip. The result from this technique is better the mesh generation by triangular and singular element. The result of K_I has been reduced.

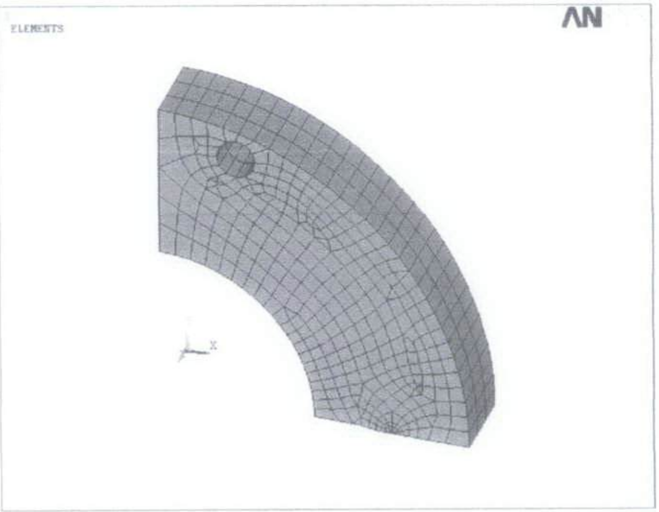


Figure 4.6. Predefine Mesh- Example of Quadrilateral element and Singular element.

4.3.5 Stress Intensity Factor by Different Meshing Technique

The following table shows the percentage different of K_I value by different meshing technique. Applying more elements and nodes result in failure of ANSYS software to proceed due to lack of computer capability.

Table 4.3. Percentage different for Triangular Mesh and Quadrilateral Mesh.

Load (N)	**Calculation** $K_I Mpa\sqrt{m}$	**Triangular mesh** $K_I Mpa\sqrt{m}$	**% Difference**	**Quadrilateral Mesh** $K_I Mpa\sqrt{m}$	**% Difference**
10000	27.1979	31.6768	16.6768	29.1186	7.0619
11000	29.9177	34.8277	16.4117	32.0391	7.0907
12000	32.6375	38.0038	16.4421	34.9602	7.1167

4.4 Main Model

4.4.1 Stress Distribution Analysis for Main Model

Under a load, the crack tip of a linear-elastic specimen will exhibit a stress distribution that is proportional to the inverse, square root of *r*, which is the distance from the crack tip. Regardless of the model configuration, the stress will vary in the above said relation. But the magnitude of the stress at the crack tip varies between applied loads, depending on the SIF.

Figure 4.6 and 4.8 shows the stress distribution on the specimen but having different load; 10000N and 11000N. Both figures show similar stress distribution except for amount of maximum stress that occurs at the crack tip.

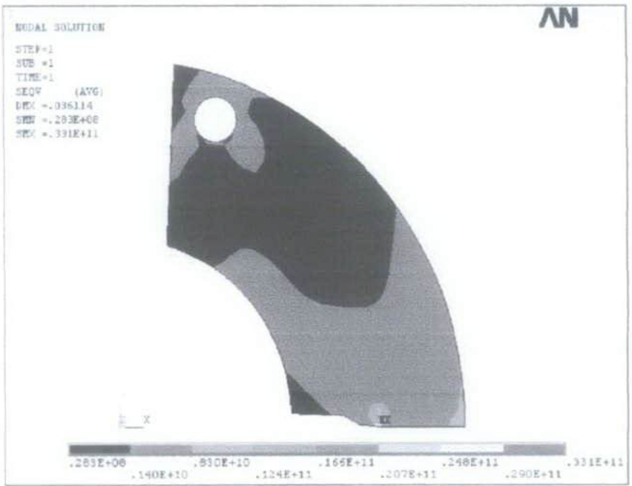


Figure 4.7. The stress distribution for load 10000N.

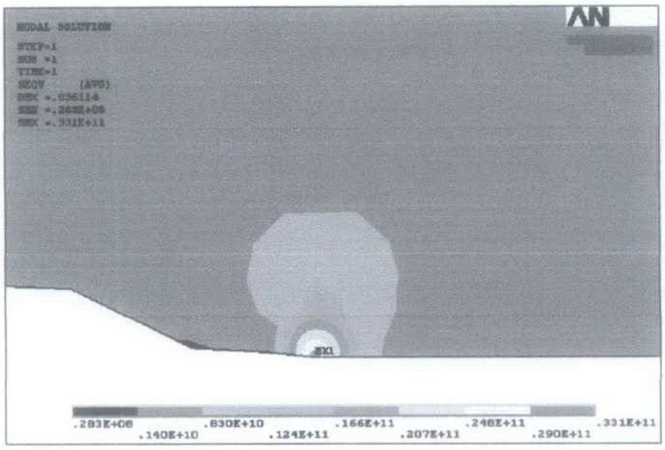


Figure 4.8. The stress distribution around the crack tip for load 10000N.

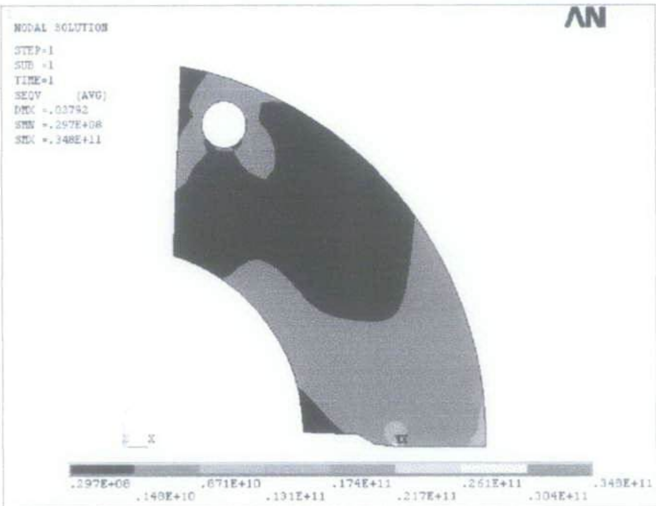


Figure 4.9. The stress distribution for load 10500N.

As observed, a common stress level (cyan colored) is distributed across the area between the loading holes to the crack tip. This is due to the lack of material to resist deformation. It ca be observed that the back portion of the Arc-shaped Tension Specimen underwent compressive stresses as well when the model is loaded. Also, crack mouth opening is not large because the model carries the brittle properties of aluminum as well.

4.4.2 Stress Intensity Factor for Various Applied Load

The stress intensity factor, K_I of a model can be derived from the ANSYS software. The modeling results are then compared with the theoretical value, determined from

Equations (3.1) and (3.2). While Figure 4.9 shows the relationship between load applied and SIF, K_1.

Table 4.4. SIF for various applied load

Load (N)	Calculation $K_I\,Mpa\sqrt{m}$	Quadrilateral mesh $K_I\,Mpa\sqrt{m}$	% Difference
10000	27.1979	29.1186	7.0619
10500	28.5578	30.5993	7.1487
11000	29.9177	32.0391	7.0907
11500	31.6375	33.4986	7.1009
12000	32.6375	34.9602	7.1167

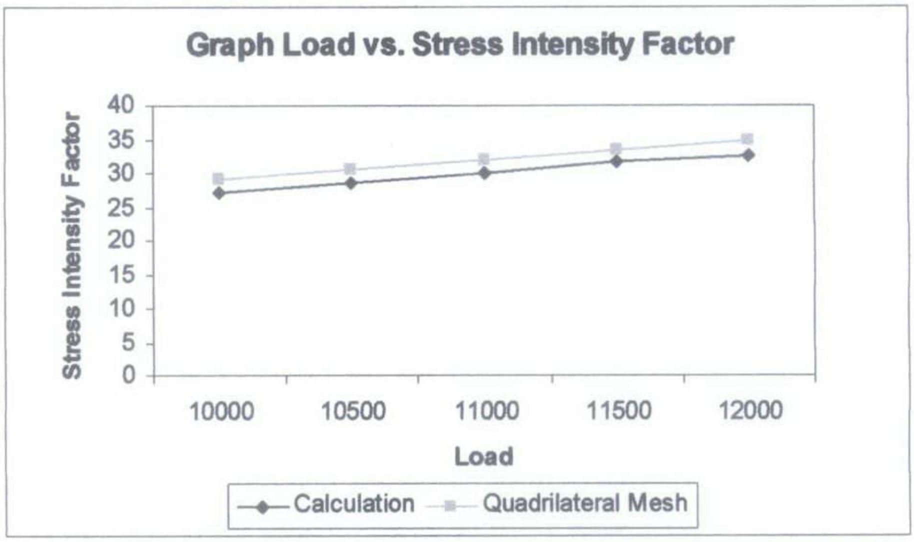


Figure 4.10. Graph load vs. SIF for various load applied.

It is very straightforward since the load is linearly proportional with the stress. In general equation, $\sigma = \frac{F}{A}$, given the area, A is constant; stress, σ will increase with the increasing of Load, F. So that, the stress intensity factor will increase as well proven that the magnitude of stress at the crack tip of load 10500N is higher than 10000N (refer Figure 4.8).

4.5 Arc-shaped Tension Specimen with Various Ring Segment

The main model is the specimen with $X/W = 0.5$ represents a half ring segment. The specimen with $X/W = 0$ represents the smallest specimen of this configuration that can be cut from a ring. Additional specimen with $X/W = 0.25$ is modeled to show the trend of varying ring segment with stress intensity factor, *K*. The ring segment determines the location of the loading holes.

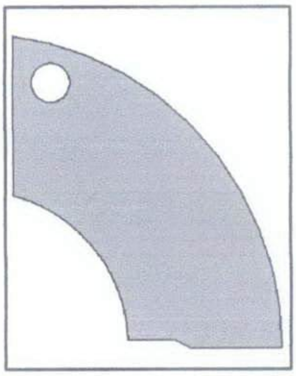
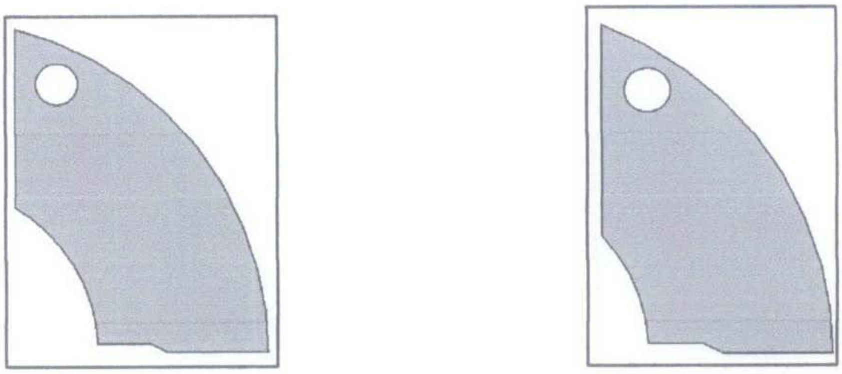


Figure 4.11. Main model of Arc-shaped Tension specimen, $X/W = 0.5$



(a) (b)

Figure 4.12. Arc-shaped Tension Specimen with different ring segment,

(a) $X/W = 0.25$; (b) $X/W = 0$

4.5.1 Stress Distribution Analysis

Following figures are the stress distribution of Arc-shaped Tension Specimen which having different ring segment. As observed, the maximum stress is occurs at the crack tip. Since the ring segment of the main model is the biggest, it has higher value of maximum stress at the crack tip. Although, the stress distribution of the third specimen, $X/W = 0$ is more obvious since the loading hole is the nearest to the crack tip. Stress is distributed smoothly resulted in low stress at the crack tip.

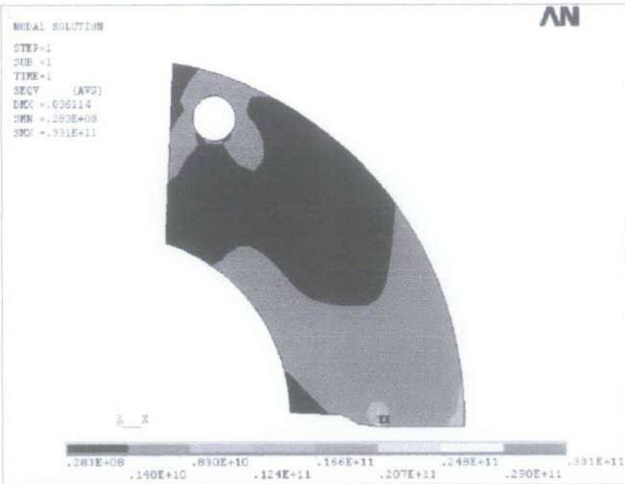


Figure 4.13. Stress distribution of specimen with $X/W = 0.5$.

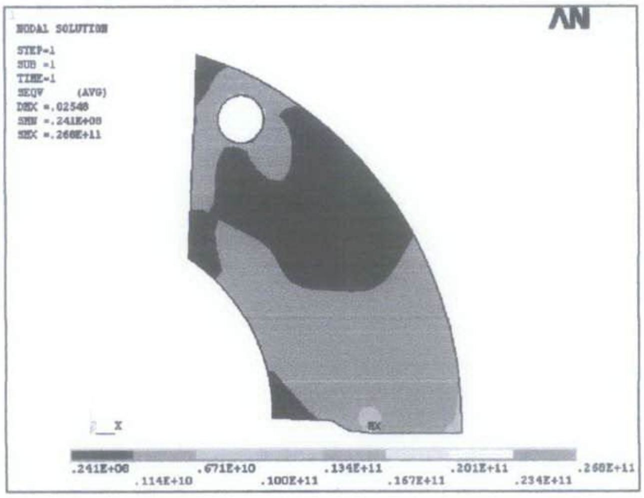


Figure 4.14. Stress distribution of specimen with $X/W = 0.25$.

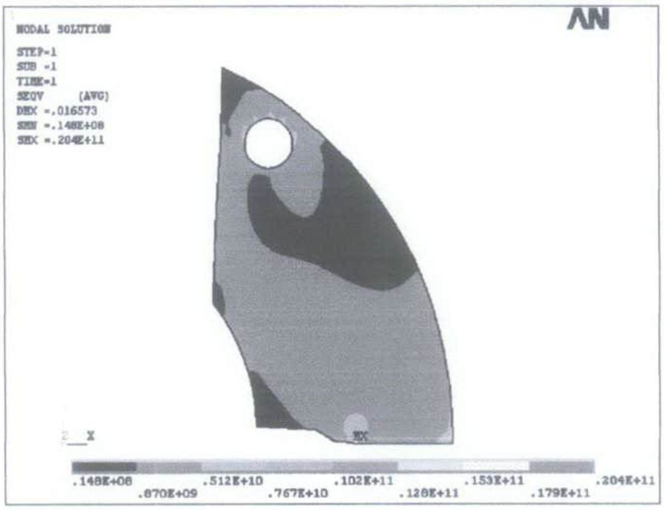


Figure 4.15. Stress distribution of specimen with $X/W = 0$.

4.5.2 Stress Intensity Factor

The calculation and ANSYS result for different ring segment are shown in the following tables and graph. Figure 4.18 show the comparison for all specimens. The nearest the loading hole to the crack tip, the lower stress intensity factor it has.

Table 4.5. SIF for ring segment $X/W = 0.5$.

Load (N)	Calculation $K_I Mpa\sqrt{m}$	ANSYS $K_I Mpa\sqrt{m}$	% Difference
10000	27.19793	29.1186	7.061821
10500	28.55783	30.5893	7.113541
11000	29.91772	32.0251	7.043906
11500	31.27762	33.4816	7.046507
12000	32.63752	34.9432	7.064518

Table 4.6. SIF for ring segment $X/W = 0.25$.

Load (N)	Calculation $K_I Mpa\sqrt{m}$	ANSYS $K_I Mpa\sqrt{m}$	% Difference
10000	22.03377	20.5102	6.9147
10500	23.13545	21.5156	7.0017
11000	24.23714	22.5427	6.9911
11500	25.33883	23.8135	6.0198
12000	26.44052	24.5894	7.0009

Table 4.7. SIF for ring segment $X/W = 0$.

Load (N)	Calculation $K_I Mpa\sqrt{m}$	ANSYS $K_I Mpa\sqrt{m}$	% Difference
10000	16.8696	15.7064	6.8952
10500	17.71308	16.4883	6.9147
11000	18.55656	17.2792	6.8837
11500	19.40004	18.0416	7.0024
12000	20.24352	18.8315	6.9751

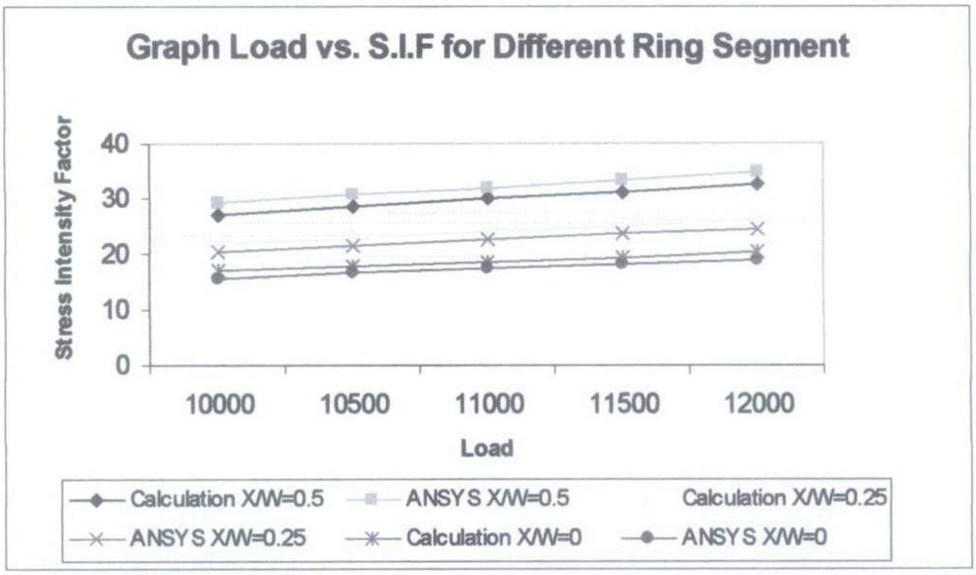


Figure 4.16. Graph Load vs. SIF for different ring segment.

The different segment ring indicates that each model has different distance between the loading hole and the crack tip. The highest distance $X/W = 0.5$ has the highest stress intensity factor since it undergo the greatest bending moment. In general equation, Moment is function of Load and Distance, $M = Fx$. The longer the distance is, the Moment will be greater. It is proven from the Von Mises Stress analysis where the specimen with $X/W = 0.5$ endure higher stress at the crack tip compared to specimen with $X/W = 0.25$ and $X/W = 0$ (refer Figure 4.12, 4.13 and 4.14).

4.6 Arc-shaped Tension Specimen with Different Thickness

Another parameter to be varied is the specimen's thickness. Beyond a certain thickness, when the material is predominantly in plane strain and under maximum constraint, the value of K_C tends to a constant lower limit, K_{IC}, the plane strain fracture toughness. K_{IC} may be considered a material property, but does depend on the test temperature and loading rate [1].

A plastic zone at the crack tip of a through-thickness crack will inevitably tend to contract in the thickness direction along the crack front. If the plate thickness is of the order of the plastic zone size or smaller, this contraction can occur freely and a *plane stress* state will prevail. On the other hand, if the plate thickness is much larger than the plastic zone size, contraction is constrained by the elastic material surrounding the plastic zone. The strain in the thickness direction will then be small, meaning that a *plane strain* state is present, as if in this specimen.

The main model has thickness; $B_1 = 0.5W = 25mm$. Comparative specimens are having $B_2 = 0.4W = 20mm$ and $B_3 = 0.3W = 15mm$.

4.6.1 Stress Distribution Analysis

Figures 4.19, 4.20 and 4.21 are the stress distributions of the main model of Arc-shaped Tension specimen having different thickness. All three specimens show similar pattern of stress distribution except the value of the maximum stress at the crack tip is different. The least thickness, $B_3 = 0.3W$ having the highest stress.

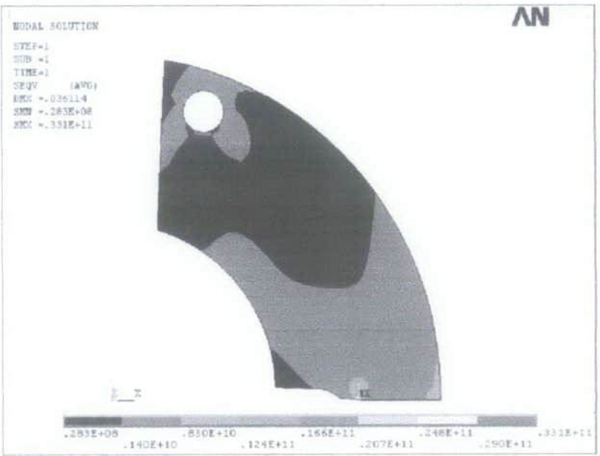


Figure 4.17. Stress distribution of specimen with $B = 0.5W$.

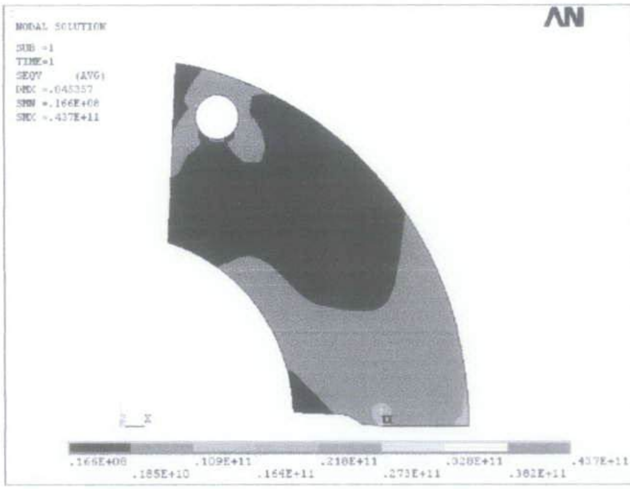


Figure 4.18. Stress distribution of specimen with $B = 0.4W$.

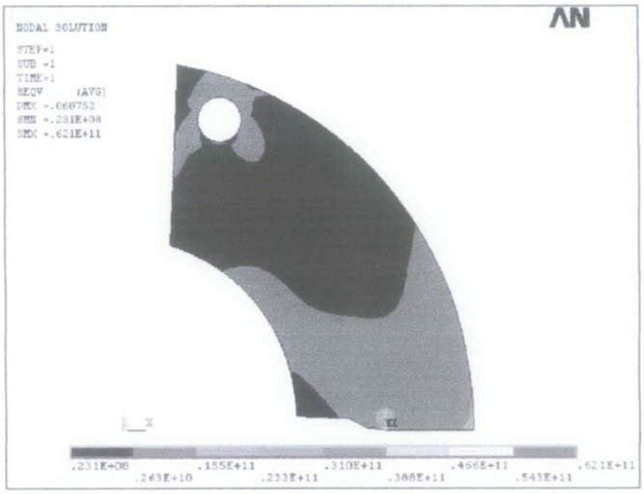


Figure 4.19. Stress distribution of specimen with $B = 0.3W$.

4.6.2 Stress Intensity Factor

SIF derived from the equation and ANSYS is shown in Table 4.8 and graph in Figure 4.22..

Table 4.8. SIF for thickness $B_1 = 0.5W = 25mm$.

Load (N)	Calculation $K_1 Mpa\sqrt{m}$	ANSYS $K_1 Mpa\sqrt{m}$	% Difference
10000	27.1979	29.1186	7.061821
10500	28.5578	30.5893	7.113541
11000	29.9177	32.0251	7.043906
11500	31.2776	33.4816	7.046507
12000	32.6375	34.9432	7.064518

Table 4.9. SIF for thickness $B_2 = 0.4W = 20mm$.

Load (N)	Calculation $K_1 Mpa\sqrt{m}$	ANSYS $K_1 Mpa\sqrt{m}$	% Difference
10000	33.9974	31.85146	6.3121
10500	35.6973	33.39777	6.4417
11000	37.3972	35.00553	6.3952
11500	39.097	36.61096	6.3587
12000	40.7969	38.18283	6.4075

Table 4.10. SIF for thickness $B_3 = 0.3W = 15mm$.

Load (N)	Calculation $K_1 Mpa\sqrt{m}$	ANSYS $K_1 Mpa\sqrt{m}$	% Difference
10000	45.3299	42.42351	6.4116
10500	47.5964	44.49114	6.5241
11000	49.8629	46.64058	6.4623
11500	52.1294	48.74435	6.4935
12000	54.3959	50.87019	6.4815

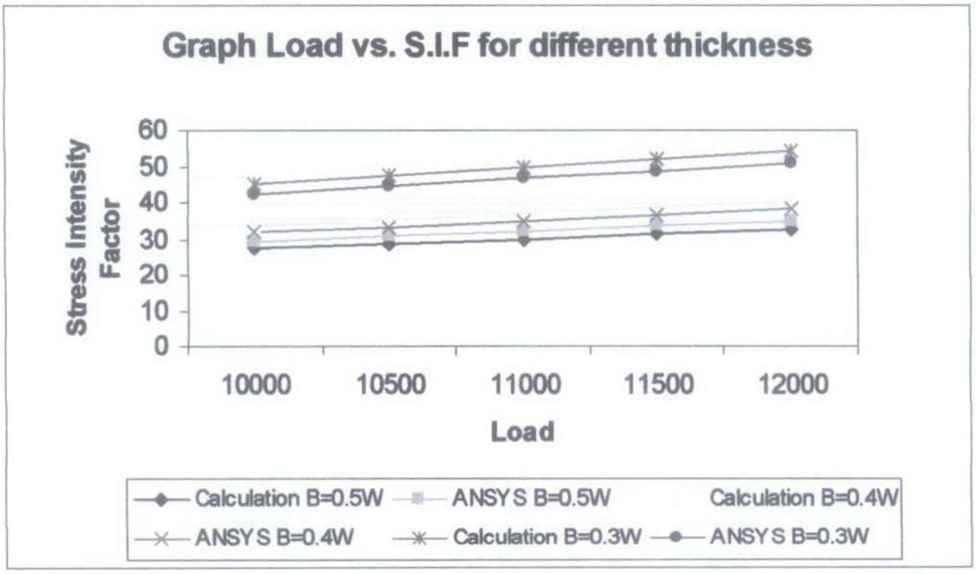


Figure 4.20. Graph Load vs. SIF for different thickness.

It can be seen clearly that the less thickness of the specimen is, the higher the stress intensity factor is has. The dependable of stress intensity factor to the specimen's thickness is due to the state of stress acting at the crack tip. Note that if the sample were made thinner, the toughness would gradually decrease because less material would be available for plastic deformation energy absorption.

4.7 Arc-shaped Tension Specimen with Different Crack Length

In the context of this fracture mechanics course the division into micro-crack and macro-crack (or long crack) growth periods is of basic importance. Virtually the whole life of some structures can be occupied from flaws. In this specimen, the crack tip acts as the stress concentration while crack length is as function of thickness.

4.7.1 Stress Distribution Analysis

The stress distribution of these three models clearly shows that the model with smaller crack length undergo more even stress distribution compared the other with larger crack length. The value of highest stress (red area) of *a=0.02* is lower indicate that the stress concentration at that point is slightly smaller than *a=0.025* and *a=0.03.*

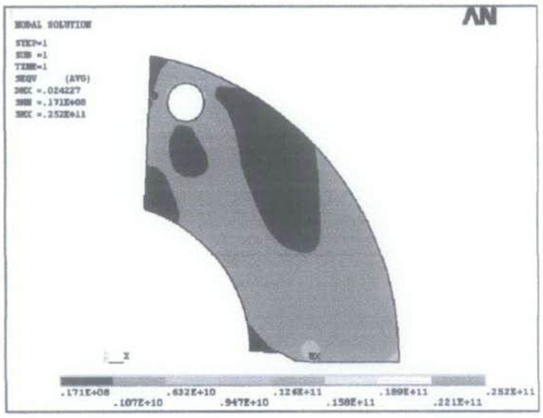


Figure 4.21. Stress distribution of specimen with $a = 0.02$.

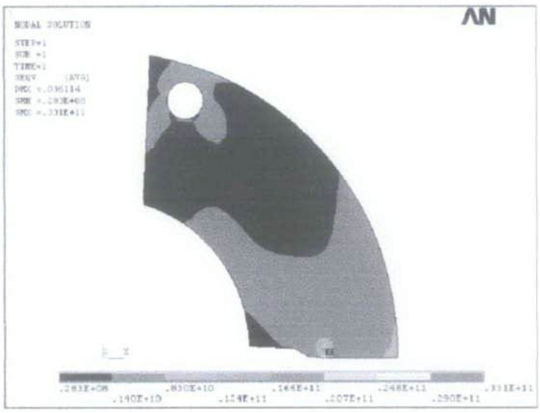


Figure 4.22. Stress distribution of specimen with $a = 0.025$.

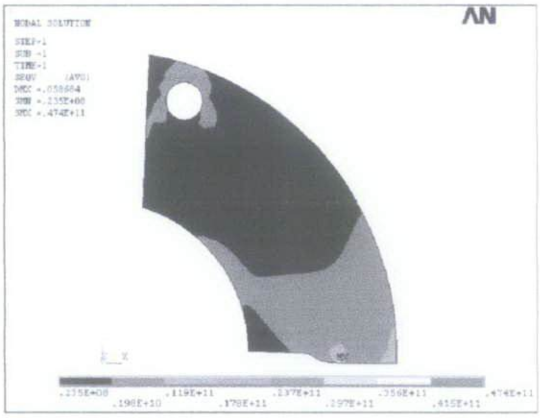


Figure 4.23. Stress distribution of specimen with $a = 0.03$.

4.7.2 Stress Intensity Factor

Table 4.11 shows the value of SIF gain from ANSYS software and theoretical analysis.

Table 4.11. SIF for crack length $a = 0.02$.

Load (N)	Calculation $K_1 Mpa\sqrt{m}$	ANSYS $K_1 Mpa\sqrt{m}$	% Difference
10000	20.2762	22.0024	8.513565
10500	21.2900	22.9856	7.964386
11000	22.3038	24.1207	8.146183
11500	23.3176	25.2187	8.153065
12000	24.3314	26.3834	8.433504

Table 4.12. SIF for crack length $a = 0.025$.

Load (N)	Calculation $K_1 Mpa\sqrt{m}$	ANSYS $K_1 Mpa\sqrt{m}$	% Difference
10000	27.1979	29.1186	7.061821
10500	28.5578	30.5893	7.113541
11000	29.9177	32.0251	7.043906
11500	31.2776	33.4816	7.046507
12000	32.6375	34.9432	7.064518

Table 4.13. SIF for crack length $a = 0.03$.

Load (N)	Calculation $K_1 Mpa\sqrt{m}$	ANSYS $K_1 Mpa\sqrt{m}$	% Difference
10000	38.7940	41.5835	7.190632
10500	40.7337	43.9439	7.881032
11000	42.6734	45.8876	7.53218
11500	44.6131	48.0257	7.649411
12000	46.5528	49.7698	6.91052

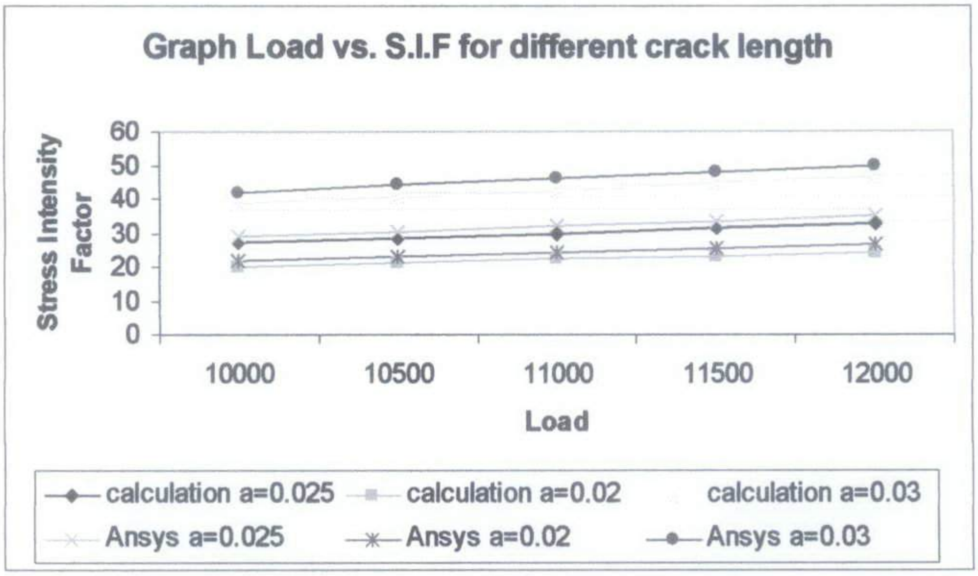


Figure 4.24. Graph Load vs. SIF for different crack length.

4.8 Analysis of Stress Distribution at the Crack Face

Additional analysis has been done to determine whether there is a jump of stresses in the specimen. Basically, any structure having imperfection like voids or crack in the microstructure will have sudden jump of stress when load applied due to the imperfection which acts as stress concentration factor. Figure 4.27 and 4.28 shows graph of stress versus distance at the crack face obtained from ANSYS for $a = 0.02$ and $a = 0.03$.

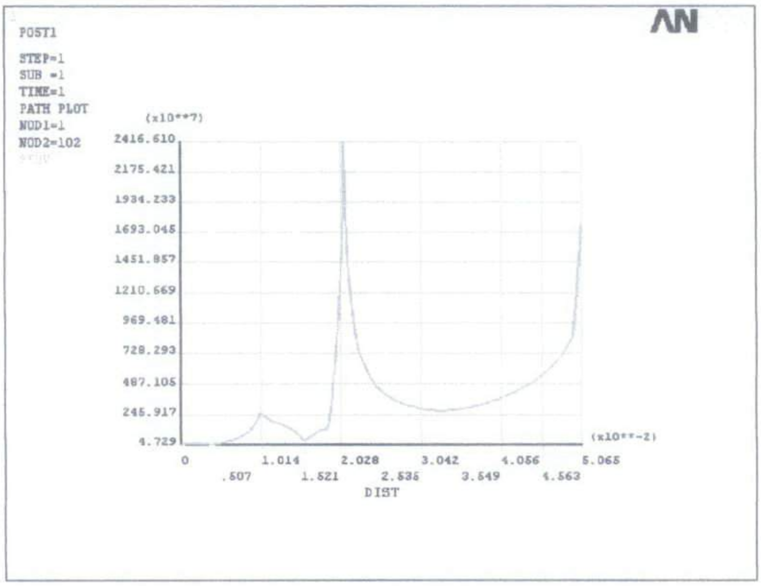


Figure 4.25. Graph stress vs. distance at the crack face of $a = 0.02$

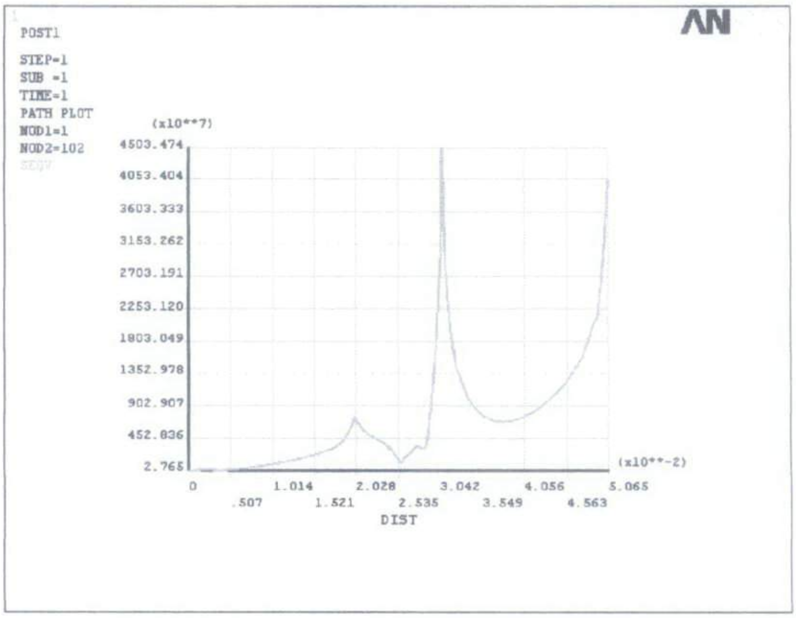


Figure 4.26. Graph stress vs. distance at the crack face of $a = 0.03$

From these two graphs, it is clearly seen that essentially both graph show similar pattern of stress distribution, only that the distance of the crack tip is different. There is sudden in the value of the stress and the highest value occurs at the crack tip. Here, crack tip acts as the stress concentration factor. The value of stress also high at the end of the specimen, that indicates that the specimen underwent compressive stress when the model is loaded. The compressive stress is resulted as reaction of the specimen to resist deformation since the specimen restricted with the boundary condition.

CHAPTER 5
CONCLUSIONS AND RECOMMENDATIONS

5.1 CONCLUSIONS

Meshing is the most crucial part in ANSYS analysis. Extra precaution need to be taken care since the mesh generation will totally affect the result of SIF and stress distribution. For fracture mechanics (crack) problem, Singular Element is the most important in the mesh generation. There are two meshing element that have been identified. Comparing these two techniques, Quadrilateral Element produce better result and is used for other parameters.

From the main model, it can be concluded that the load applied affecting the value of SIF. Since stress is linearly proportional with load; given same surface area, thus, higher load yield higher SIF. In term of geometry, having different ring segment also affect the SIF. The ring segments determine the position of the loading hole from the crack tip. Longer distances undergo higher bending moment which means larger ring segment resulted in larger SIF. The specimen's thickness and crack length also varied to study its effect with SIF. If the sample were made thinner, the toughness would gradually decrease because less material would be available for plastic deformation energy absorption, hence resulted in higher SIF. Larger crack length resulted in higher SIF due to higher bending moment experienced

As a conclusion, the results show that increasing the load applied, ring segment distance, crack length or decreasing the thickness can significantly increase the SIF.

5.2 RECOMMENDATIONS

The finite element model was able to exhibit linear-elastic behavior. The stress distribution has been observed to approximate the stress-crack tip singularity where the area of high stress is small. The minimal deformation observed in the model clearly suggests that the model adhere to the brittle properties of aluminum. The SIF value obtained show significant error. There is effort that can be taken to improve the accuracy of the result.

Generally, the accuracy of a finite element model can be significantly improved by increasing the number of elements in the model which means to employ a finer mesh. Although, as experienced in this project, the processing power of the hardware becomes a constraint. This means that only the critical areas of the model are refined, ad not the entire model.

The procedures for three-dimensional, linear-elastic crack tip have been established. Therefore, the work of finite element modeling can be taken further through the implementation of the recommended plans:

1. Using ANSYS software, Computer Aided Engineering (CAE) associates had found several ways on predicting and determination the SIF. The methods are:
    a. Hand calculation.
    b. ANSYS special crack tip elements.
    c. ANSYS J-integral method.
    d. ANSYS direct method.

   The methods been used in this project are Hand calculation and ANSYS special crack tip elements using singular element at the crack tip. Another two methods is recommended to be explored although ANSYS J-integral method quite complicated and tedious, while ANSYS direct method is continuation from ANSYS J-integral elements at the crack tip. In real fracture mechanics problem, all those methods may be necessary to be used for more accurate results.

2. Another aspect that can be explored is the modeling of different crack loading modes. This study was devoted to only Mode-I loading. Modeling work can hence be extended to Mode-II and Mode-III loadings, and even mixed mode loadings. This will be significant because in reality, components are often subjected to mixed mode loadings.

3. The simulation of real component using FEA methods can be further expanding. Component and parts with complex geometries can be modeled to identify the stress concentration sites and analyze the stress distribution throughout the entire component. One may request to any company to study on any component that is failed, model and simulate using FEA, and later compare the result with the available data.

REFERENCES

1. Michael Janssen, Jan Zuidema, Russell Wanhill,, *Fracture Mechanics* (2nd Edition), Spon Press (2004): New York.

2. Bahram Farahmand, *Fracture Mechanics of Metals, Composites, Welds, and Bolted Joints* (1st Edition), Kluwer Academic Publishers (2000): United States.

3. Dietmar Gross, Thomas Seelig, *Fracture Mechanics, with an Introduction to Micromechanics* (4th Edition), Springer (2006): New York.

4. Hertzberg, R.., *Deformation and Fracture Mechanics of Engineering Materials* (4th Edition), John Wiley and Sons (1996): New York..

5. William D. Callister. Jr., *Materials Science and Engineering an Introduction* (7th Edition), John Wiley and Sons (2006): New York.

6. American Society for Testing and Materials (ASTM), *Annual Book of ASTM Standards, Section 3: Metal Test Methods and Analytical Procedures,* E399 Standard Test Method for Plane-Strain Fracture Toughness of Metallic Materials, Standard 2001, Volume 03.01.

7. Saeed Moaveni, *Finite Element Analysis Theory and Application with ANSYS* (2nd Edition), Pearson Education (1999): New York.

8. Ted L. Anderson, *Fracture Mechanics: Fundamentals and Applications* (2nd Edition), CRC Press (2005): United States of America.

9. KTH Solid Mechanics, *Tutorial 3:2D Crack Problem,* Spring (2006)

10. Website refer to <http://www.key-to-steel.com/Articles/Art45.htm>

11. Website refer to <http://en.wikipedia.org/wiki/Stress_intensity>

12. Website refer to <http://en.wikipedia.org/wiki/Fracture_toughness>

APPENDIX A

American Society for Testing Materials (ASTM) E399 Standard Test Method for Plane-Strain Fracture Toughness of Metallic Materials calculate the normalized crack length as follows (see Note A4.3):

$$a/W = 1.000 - 4.500U + 13.157U^2 - 172.551U^3 + 879.944U^4 - 1514.671U^5 \quad (A4.5)$$

where:

$U = 1/[1 + (E'BV_m/P)^{1/2}]$

NOTE A4.3—This expression fits the equation in A4.5.5 within ±0.01 % of W for $0.2 \leq a/W \leq 0.8$ (25). This expression is valid only for crack mouth displacements measured at the location of the integral knife edges shown in Fig. 5. If attachable knife edges are used, they must be reversed or inset to provide the same measurement point location.

A5. SPECIAL REQUIREMENTS FOR THE TESTING OF THE ARC-SHAPED TENSION SPECIMEN

A5.1 Specimen

A5.1.1 The arc-shaped tension specimen is a single edge-notched and fatigue cracked ring segment loaded in tension. The general proportions of two designs of the specimen are shown in Fig. A5.1. The value of the radius ratio r_1/r_2 is not specified, so that specimens can be taken from any cylindrical geometry. However, it should be noted that specimens with $r_1/r_2 = 0$ (that is, from a solid cylinder) do not make the best possible use of the test material because the definition of W was chosen to accommodate hollow cylinders. The disk-shaped specimen should be used for tests on solid cylinders (see Annex A6).

A5.1.2 The arc-shaped tension specimen is intended to measure the fracture toughness so that the normal to the crack plane is in the circumferential direction and the direction of crack propagation is in the radial direction. This is the *C-R* orientation as defined in 5.1.3. For other orientations, a bend (Annex A3) or a compact (Annex A4) specimen should be used.

A5.1.3 The specimen with $X/W = 0.5$ (Fig. A5.1a) represents a half ring segment. The specimen with $X/W = 0$ (Fig. A5.1b) represents the smallest specimen of this configuration that can be cut from a ring.

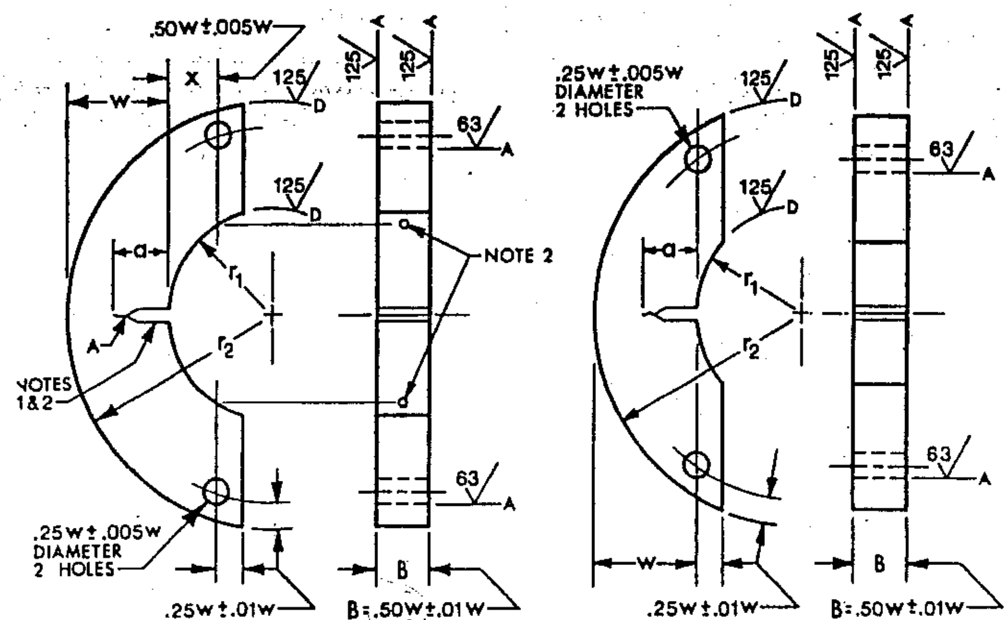
A5.1.4 Alternative specimens may have $2 \leq W/B \leq 4$ but with no change in other proportions. The use of alternative specimen proportions can be advantageous because in many cases it is possible to test ring segments with no machining of the inner and outer radii, that is, with no change in W.

A5.2 Specimen Preparation

A5.2.1 For generally applicable specifications concerning specimen size and preparation see Section 7.

A5.3 Apparatus

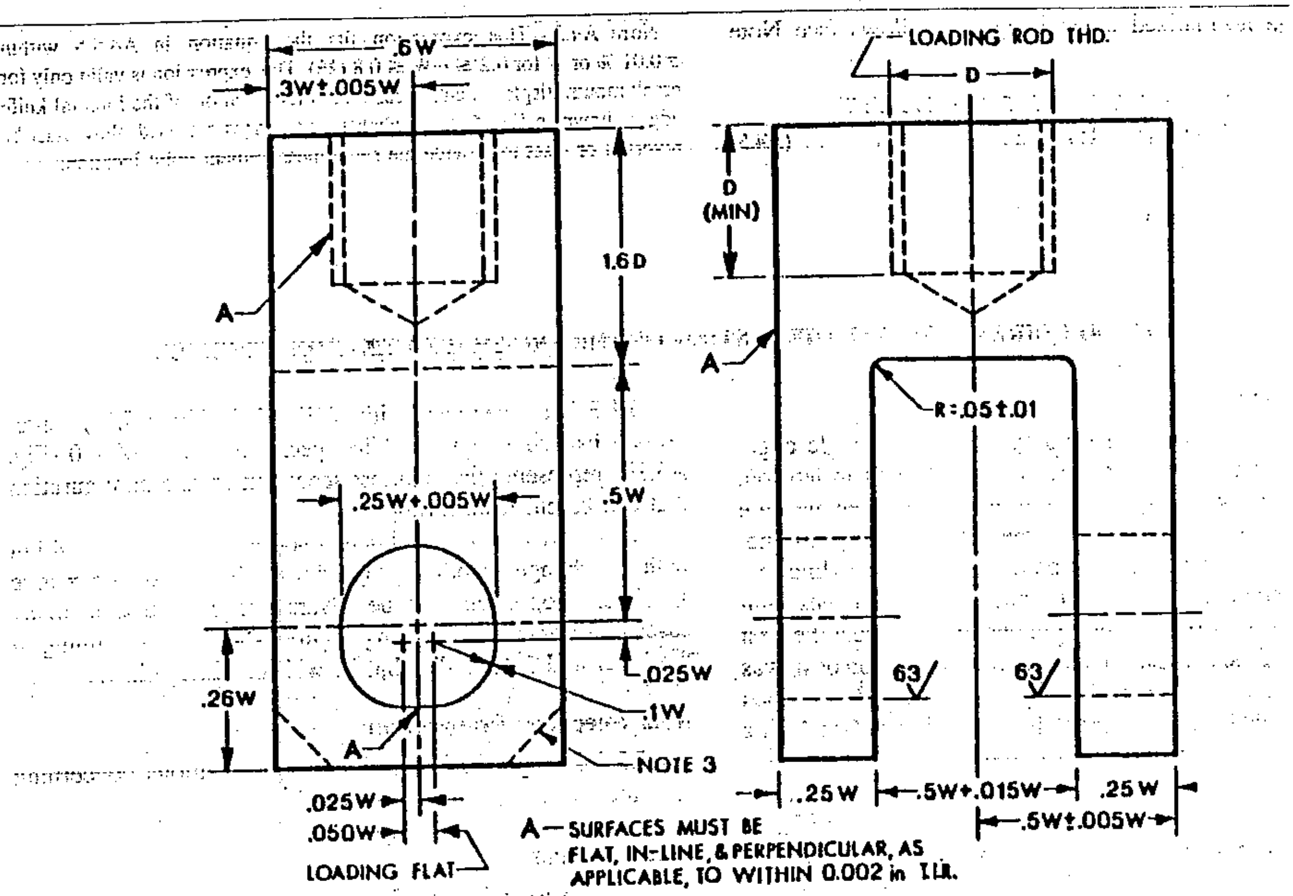
A5.3.1 *Tension Testing Clevis*—A loading clevis suitable for testing arc-shaped tension specimens is shown in Fig. A5.2. Both ends of the specimen are held in such a clevis and loaded through pins, in order to allow rotation of the specimen during



(a) $X/W = 0.5$ (b) $X/W = 0$

NOTE 1—For starter notch and fatigue crack configurations see Fig. 7.
NOTE 2—Alternative displacementgage reference points (see A5.4.1.1 for calculation of (a)).
NOTE 3—Axis of holes to be tangent to inner radius within 0.005W.
NOTE 4—*A* surfaces to be perpendicular and parallel as applicable to within 0.002 W TIR. *D* surfaces to be perpendicular or parallel as applicable to *A* surfaces to within 0.02 W TIR (see A5.4.1).

FIG. A5.1 Arc-Shaped Specimen Designs A (T) Standard Proportions and Tolerances



NOTE 1—Pin diameter = 0.24 W (+ 0.000 W/−0.005 W). For specimens with σ_{YS}> 200 ksi (1379 MPa) the holes in the specimen and in the be 0.3 W (+ 0.005 W/−0.000 W) and the pin diameter 0.288 W (+ 0.000 W/−0.005 W).

NOTE 2—0.002 in. = 0.051 mm.

NOTE 3—Corners of the clevis may be removed if necessary to accommodate the clip gage.

FIG. A5.2 Tension Testing Clevis Design

testing. In order to provide rolling contact between the loading pins and the clevis holes, these holes are provided with small flats on the loading surface (4). Other clevis designs may be used if it can be demonstrated that they will accomplish the same result as the design shown.

A5.3.1.1 The critical tolerances and suggested proportions of the clevis and pins are given in A5.2. These proportions are based on specimens having W/B = 2 for B > 0.5 in. (12.7 mm) and W/B = 4 for B= 0.5 in. (12.7 mm). If a 280 000-psi (1930-MPa) yield strength maraging steel is used for the clevis and pins, adequate strength will be obtained for testing the specimen sizes and σ_{YS}/E ratios given in 7.1.3. If lower-strength grip material is used, or if substantially larger specimens are required at a given σ_{YS}/E ratio than those shown in 7.1.3, then heavier grips will be required. As indicated in Fig. A5.2, the clevis corners may be cut off sufficiently to accommodate seating of the clip gage in specimens less than 0.375 in. (9.5 mm) thick.

A5.3.1.2 Careful attention should be given to achieving as good alignment as possible through careful machining of all auxiliary gripping fixtures.

A5.3.2 *Displacement Gage*—For generally applicable details concerning the displacement gage see 6.3.

A5.3.2.1 An alternative means of measuring the displacement is allowed for the specimen with X/W = 0.5. Conical center-punch-type indentations are provided on the face of the specimen at mid-thickness and in the pl center line of the loading holes as shown in Fig. A load-point displacement of the specimen is measure points using a displacement gage fitted with points ar the requirements described in 6.4.1.

A5.3.2.2 The displacements will be essentially in of the gage length for the arc-shaped specimen pro gage length is equal to or less than $W/2$.

A5.4 Procedure

A5.4.1 *Measurement*—Before testing an arc-sha men, measure ($r_1 - r_2$) to the nearest 0.001 in. (0.0 to 0.1 %, whichever is greater at mid-thickness pc both sides of and immediately adjacent to the cr notch mouth. Record the average of these two read Also measure ($r_1 - r_2$) at four positions, two as possible to the loading holes and two at approximatel the circumferential distance between the loading hol crack plane. If any of these four measurements diff by more than 10 %, the specimen should be di reworked. Next, measure to the nearest 0.001 in. (0.0 to 0.1 %, whichever is greater, the distance be loading hole centers and the outside surface of the s the notch plane. This measurement should be mad of the specimen by referencing the loading holes. Subtract W from the average of these two measurements and record the result as X. Measure within 5 % the outer radius, r_2; if this is not possible, determine the average value of r_2 as follows (see Note A5.1): Measure within 5 % the length, L, of the chord of the outer surface, which chord passes through the loading hole centers (see Fig. A5.3). Using this measurement, calculate:

$$r_2 = \frac{L^2}{8(W+X)} + \frac{(W+X)}{2} \quad \text{(A5.1)}$$

Then $r_1/r_2 = 1 - W/r_2$.

NOTE A5.1—A 10 % variation of the ratio r_1/r_2 will affect the value of the stress intensity factor by 1 % or less, providing that the relative crack length a/W is not less than 0.3. However, the stress analysis is based on the assumption that the specimens are to be cut from stock of uniform, axisymmetric cross section. If inspection shows that the stock deviates from axisymmetry by more than 10 %, it should be reworked to within this tolerance.

A5.4.1.1 After fracture, measure the crack length in accordance with 8.2.2, but a special procedure is necessary for the arc-shaped tension specimen due to its curvature. Thus, a length measurement, m, made from a reference point adjacent to the crack mouth to a point on the crack front will be greater than the corresponding distance from the virtual point of intersection between the crack plane and the inside circumference of the specimen (see Fig. A5.3). The error, e, may be computed from the following expression:

$$e = r_1 - \left[r_1^2 - \frac{g^2}{4}\right]^{1/2} \quad \text{(A5.2)}$$

where g is the distance across the crack mouth at the reference points for measurement of the crack length. It should be noted that g may be equal to N (Fig. 7) or larger than N if machined knife edges are used to hold the clip gage (for example, $g = 0.25$ in. (6.4 mm) as in Fig. 5). If the relative error $e/m < 0.01$, then record m as the crack length; otherwise e should be subtracted from m and the result recorded as the crack length.

A5.4.2 *Arc-Shaped Tension Specimen Testing*—When assembling the loading train (clevises and their attachments to the tension machine) care should be taken to minimize eccentricity of loading due to misalignments external to the clevises. To obtain satisfactory alignment keep the centerline of the upper and lower loading rods coincident within 0.03 in. (0.76 mm) during the test and center the specimen with respect to the clevis opening within 0.03 in. (0.76 mm).

A5.4.2.1 Load the arc-shaped specimen at such a rate that the rate of increase of stress intensity is within the range 30 to 150 ksi·in.$^{1/2}$/min (0.55 to 2.75 MPa·m$^{1/2}$/s). The corresponding loading rates for a standard ($W/B = 2$) 1-in. thick specimen are: (*1*) for the specimen with an $X/W = 0.5$ between 2800 and 14 000 lbf/min (0.2 to 1.0 kN/s) and (*2*) for the specimen with an $X/W = 0$ between 4500 and 22 500 lbf/min (0.34 to 1.7 KN/s).

A5.4.2.2 For details concerning recording of the test record see 8.4.

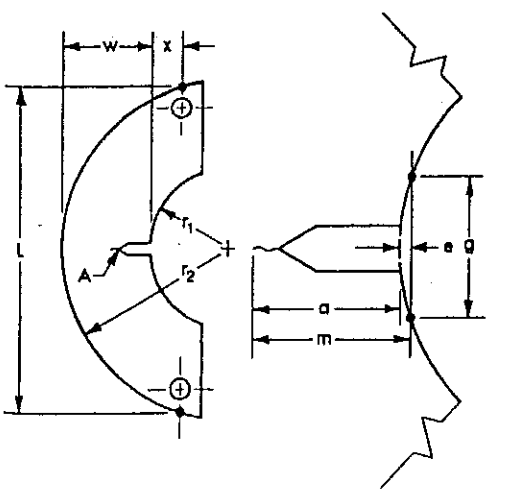


FIG. A5.3 Measurement of Outer Radius (r_2) and Crack Length for the Arc-Shaped Specimen (see A5.4.1)

A5.5 Calculations

A5.5.1 For general requirements and procedures in interpretation of the test record see 9.1.

A5.5.2 For a description of the validity requirements in terms of limitations on P_{max}/P_Q and the specimen size requirements see 9.1.2 and 9.1.3.

A5.5.3 *Calculation of K_Q*—For the arc-shaped specimen calculate K_Q in units of ksi·in.$^{1/2}$ (MPa·m$^{1/2}$) from the following expression (Note A5.2):

$$K_Q = (P_Q/BW^{1/2})[3X/W + 1.9 + 1.1a/W] \times [1 + 0.25(1 - a/W)^2(1 - r_1/r_2)]f(a/W) \quad \text{(A5.3)}$$

where:

$$f(a/W) = [(a/W)^{1/2}/(1 - a/W)^{3/2}] \times [3.74 - 6.30a/W + 6.32(a/W)^2 - 2.43(a/W)^3]$$

where:

P_Q = load as determined in 9.1.1, klbf (kN),
B = specimen thickness as determined in 8.2.1, in. (cm),
X = loading hole offset as determined in A5.4.1, in. (cm),
W = specimen width as determined in A5.4.1, in. (cm),
a = crack length as determined in 8.2.2 and A5.4.1.1, in. (cm), and
r_1/r_2 = ratio of inner to outer radii as determined in A5.4.1.

NOTE A5.2—The accuracy of this expression for all values of r_1/r_2 is considered to be as follows: (*1*) ±1 % for 0.45 ≤ a/W ≤ 0.55 and X/W of 0 or 0.5, (*2*) ±1.5 % for 0.2 ≤ a/W ≤ 1 and X/W of 0 or 0.5, and (*3*) ±3 % for 0.2 ≤ a/W ≤ 1 and 0 ≤ X/W ≤ 1 (14).

A5.5.3.1 To facilitate calculation of K_Q, values of $f(a/W)$ are tabulated in the following table for specific values of a/W:

a/W	f(a/W)	a/W	f(a/W)
0.450	3.23	0.500	3.73
0.455	3.27	0.505	3.79
0.460	3.32	0.510	3.85
0.465	3.37	0.515	3.91
0.470	3.42	0.520	3.97
0.475	3.47	0.525	4.03
0.480	3.52	0.530	4.10
0.485	3.57	0.535	4.17
0.490	3.62	0.540	4.24
0.495	3.68	0.545	4.31
...	...	0.550	4.38

A5.5.4 *Calculation of R_{sa}*—For the arc-shaped tension specimen calculate the specimen strength ratio (which is dimensionless and has the same value in any consistent system of units) as follows:

$$R_{sa} = \frac{2P_{max}(3X + 2W + a)}{B(W - a)^2 \sigma_{YS}} \quad (A5.4)$$

where:

P_{max} = maximum load that the specimen was able to sustain,
B = thickness of the specimen as determined in 8.2.1,
X = loading hole offset as determined in A5.4.1,
W = width of the specimen as determined in A5.4.1,
a = crack length as determined in 8.2.2 and A5.4.1.1, and
σ_{YS} = yield strength in tension (offset = 0.2 %) (see Methods E 8).

A5.5.5 *Calculation of Crack Mouth Opening Compliance Using Crack Length Measurements*—For arc-shaped tension specimens, calculate the crack mouth opening compliance, V_m/P, in units of m/N (in./lb) as follows (see Note A5.3): for the specimen with $X/W = 0$:

$$V_m/P = [p_1(a/W)/E'B]\cdot[0.43(1 - r_1/r_2) + q_1(a/W)] \quad (A5.5)$$

where:

$p_1(a/W) = (1 + a/W)/(1 - a/W)^2$
$q_1(a/W) = 0.542 + 13.137(a/W) - 12.316(a/W)^2 + 6.576(a/W)^3$

or, for the specimen with $X/W = 0.5$,

$$V_m/P = [p_2(a/W)/E'B]\cdot[0.45(1 - r_1/r_2) + q_2(a/W)] \quad (A5.6)$$

where:

$p_2(a/W) = (2 + a/W)/(1 - a/W)^2$
$q_2(a/W) = 0.399 + 12.63(a/W) - 9.838(a/W)^2 + 4.66(a/W)^3$, and

where:

V_m = crack mouth opening displacement, m (in.),
P = applied load, kN (klbf),
E' = Effective Young's Modulus (= E for plane stress, Pa (psi); = $E/(1 - \nu^2)$ for plane strain, Pa (psi),
ν = Poisson's Ratio, and

X, B, W, a, and (r_1/r_2) are as defined in A5.5.3.

NOTE A5.3—These expressions are considered to be accurate to within ±1.4 % ($X/W = 0$) or ±1.6 % ($X/W = 0.5$) for $0.2 \leq a/W \leq 0.8$ and $(r_1/r_2) \geq 0.4$ (**25**). These expressions are valid only for crack mouth displacements measured at the location of the integral knife edges comparable to that shown in Fig. 5. If attachable knife edges are used, they must be reversed or inset to provide the same measurement

A5.5.5.1 To facilitate the calculation of crack i
ing compliances, values of $p_1(a/W)$, $p_2(a/W)$, q
$q_2(a/W)$ are given in the following table for speci
a/W:

a/W	p_1(a/W)	q_1(a/W)	p_2(a/W)	q_2(a/
0.450	4.79	4.56	8.10	4.5
0.455	4.90	4.59	8.27	4.5
0.460	5.01	4.62	8.44	4.5
0.465	5.12	4.65	8.61	4.6
0.470	5.23	4.68	8.79	4.6
0.475	5.35	4.71	8.98	4.6
0.480	5.47	4.74	9.17	4.7
0.485	5.60	4.77	9.37	4.7
0.490	5.73	4.80	9.57	4.7
0.495	5.86	4.82	9.78	4.8
0.500	6.00	4.85	10.00	4.8
0.505	6.14	4.88	10.22	4.8
0.510	6.29	4.91	10.45	4.9
0.515	6.44	4.94	10.69	4.9
0.520	6.60	4.97	10.94	4.9
0.525	6.76	5.00	11.19	4.9
0.530	6.93	5.02	11.45	5.0
0.535	7.10	5.05	11.72	5.0
0.540	7.28	5.08	12.00	5.0
0.545	7.46	5.11	12.29	5.1
0.550	7.65	5.14	12.59	5.1

A5.5.6 *Calculation of Crack Lengths Using (*
Opening Compliance Measurements—For arc-sl
mens, calculate the normalized crack length as
Note A5.4):

for the specimen with $X/W = 0$:

$a/W = 0.989 - 3.463U - 0.171U^2 + 24.354U^3 - 72.805$

where:

$U = 1/\{1 + [(E'BV_m/P)(1 + 0.101(1 - r_1/r_2))]^{1/}$

or, for the specimen with $X/W = 0.5$:

$a/W = 0.986 - 4.082U - 5.0650U^2 + 86.819U^3 - 3$
$+ 429.101U^5$

where:

$U = 1/\{1 + [(E'BV_m/P)(1 + 0.108(1 - r_1/r_2))]^{1/}$

NOTE A5.4—This expression fits the equations in
±0.003W for $0.2 \leq a/W \leq 0.8$, $(r_1/r_2) \geq 0.4$, and $X/W =$ (
(**25**). This expression is valid only for crack mouth
measured at the location of the integral knife edges com
shown in Fig. 5. If attachable knife edges are used, they m
or inset to provide the same measurement point location.

A6. SPECIAL REQUIREMENTS FOR THE TESTING OF THE DISK-SHAPED COMPACT SPECIMEN

A6.1 Specimen

A6.1.1 The standard disk-shaped compact specimen is a single edge-notched and fatigue cracked disk segment loaded in tension (**16**). The general proportions of this specimen configuration are shown in Fig. A6.1.

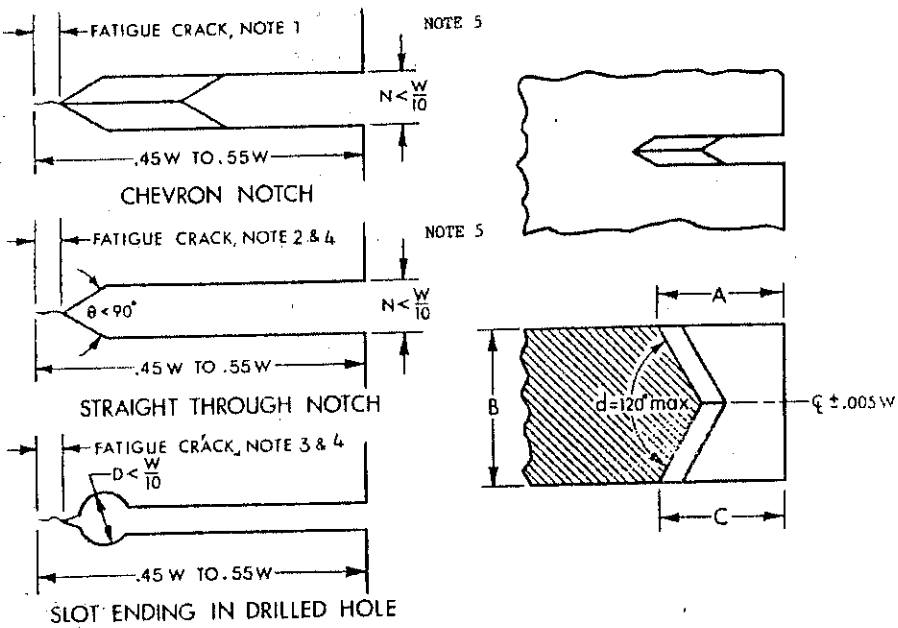
A6.1.2 Alternative specimens may have $2 < W/B < 4$ but with no change in other proportions.

A6.2 Specimen Preparation

A6.2.1 For generally applicable specification
specimen size and preparation see Section 7.

A6.3 Apparatus

A6.3.1 *Tension Testing Clevis*—A loading clev
testing disk-shaped compact specimens is shown
Both ends of the specimen are held in such a clev



(*a*) Starter Notches and Fatigue Cracks

NOTE 1—For a chevron crack starter notch the fatigue crack shall emerge on both surfaces of the specimen.

NOTE 2—Fatigue crack extension on each surface of the specimen containing a straight-through notch shall be at least 0.025 *W* or 0.050 in. (1.3 mm), whichever is larger.

NOTE 3—Fatigue crack extension on each surface of the specimen from the stress raiser tipping the hole shall be at least 0.5 *D* or 0.050 in. (1.3 mm), whichever is larger.

NOTE 4—Crack starter notch shall be perpendicular to the specimen surfaces and to the intended direction of crack propagation within ±2°.

NOTE 5—Notch width *N* need not be less than 1/16 in. (1.6 mm).

(*b*) Detail of Chevron Notch

NOTE 1—A ≈ C within 0.010 W

NOTE 2—Cutter tip angle 90° max

NOTE 3—Radius at chevron notch bottom 0.010 in. (0.25 mm) max

Figure C.5 : The notch dimensions as outlined in the ASTM E399 standard

PREPROCESSING

1. Give the Job a Name

Utility Menu >File >Change Jobname ...

- The window comes up. Enter a name, for example `CentralCrack', and click on OK.

2. Select Model Preferences

Main Menu >Preferences

- Click on Structural, and then click OK.

3. Define Element Type

Main Menu >Preprocessor >Element Type >Add/Edit/Delete

- This brings up the 'Element Types' window. Click on the Add... button.
- The 'Library of Element Types' window appears. Highlight `Solid', and `8node 82', as shown. Click on Apply.
- Under 'Solid', highlight '20node 95', Click on OK.
- You should see `Type 1 PLANE82' and 'Type 2 SOLID95' in the `Element Types' window.
- Click on the Options... button in the above window. Select `Plane strain' for `Element behavior K3' and click OK.
- Click on the Close button in the `Element Types' window.

4. Define Material Properties

Main Menu >Preprocessor >Material Props >Material Models

- In the right side of the `Define Material Model Behavior' window that opens, double click on `Structural', then `Linear', then `Elastic', then finally `Isotropic'.
- A window comes up. Enter in values for the Young's modulus (EX = 69E9) and Poisson's ratio (PRXY = 0.33) of the arc-shaped material.
- Click OK, then close the `Define Material Model Behavior' window.

5. Define Keypoints

Main Menu >Preprocessor >Modeling>Create>Keypoints>In Active CS

- We are going to create 5 keypoints given in the following table:

keypoint	X	Y
1	0	0
2	0.04994	0.0025
3	0.0125	0.04841
4	0.0125	0.0992
5	0.1	0
6	0.075	0
7	0.07	0
8	0.065	0.0025

- To create keypoint #1, enter `1' as keypoint number, and `0' and `0' as the X and Y coordinates in the following window. Click on Apply.
- Repeat the above step for keypoints #2 through #5. Note that you must click on OK instead of Apply after entering data of the final keypoint.

6. Define Line Segments

Main Menu >Preprocessor >Modeling >Create >Lines >Arcs >By End KPs and Rad

- Pick keypoint #2 then keypoint #3 as End KPs and keypoint#1 as center-of-curvature side. The radius is 0.05.
- Repeat the previous step to create line connecting keypoint#4 and #5 which having radius of 0.1.

Main Menu >Preprocessor >Modeling >Create >Lines >Lines >StraightLine

- Pick keypoint #3 then keypoint #4 to create a line connecting them.
- Repeat the previous step to create lines connecting keypoints #5 and #6, keypoints #6 and #7, keypoints #7 and #8, and keypoints #8 and #2.
- Click on OK to close the `Create Straight Line' window (picking window).
- Turn on the numbering by selecting **Utility Menu >PlotCtrls >Numbering** .... Check the boxes for `Keypoint numbers' and `Line numbers', then click on OK.
- Select **Utility Menu >Plot >Lines.**

7. Create the Area

Main Menu >Preprocessor >Modeling >Create >Areas >Arbitrary >By Lines

- Pick all lines (L1 through L7). Click OK in the picking window.

8. Create the Loading Hole

Main Menu>Preprocessor>Modeling>Create>Areas>Circle>Solid Circle

- In the 'Solid Circular Area' window, enter below specification for X and Y coordinate of the circle center and the radius. Click OK.

9. Subtract the Solid Circle Area

MainMenu>Preprocessor>Modeling>Operate>Booleans>Subtract>Area

- To create the loading hole, first pick the arc-shaped as the base area and then the solid circle as area to be subtracted. Click on OK.

10. Create the Concentration Keypoint (Crack Tip)

Main Menu >Preprocessor >Meshing >Size Cntrls >Concentrat KPs >Create

- Pick keypoint #6, then click OK in the picking window.
- In the window that appears, you should see `6' as `Keypoint for concentration'.
- Enter `0.002' for `Radius of 1st row of elems', '0.5' for 'Radius of 1st Row of Elems', input `8' for `No of elems around circumf', and select `Skewed 1/4pt' for `midside node position'. Click OK.

11. Discretize all Lines

Main Menu >Preprocessor >Meshing >Size Cntrls >ManualSize >Lines >Picked Lines

- Pick lines #1. Click on the OK button in the picking window.
- The window opens. Enter `35' for 'No. of element divisions', '4' for 'Spacing Ratio' then click Apply.
- Repeat the previous two steps for below specification.

Line#	1	2	3	4	5	6	7	8	9	10	11
NDIV	35	40	15	15	5	5	10	6	6	6	6
SPACE	4	0.2	1	0.2	1	1	4	1	1	1	1

12. Mesh the Model

Main Menu>Preprocessor>Meshing>Mesh>Areas>Free

- Pick the area. Click OK in the picking window.
- Close the `Warning' window.

13. Extrude the Model.

Main Menu>Preprocessor>Modeling>Operate>Extrude>Elem Ext Opts

- In the window 'Element Extrusion Option', select 'SOLID95' as 'Element type number', Enter '6' as 'No. Elem divs', and '1' in 'Spacing ratio'.

Main Menu>Preprocessor>Modeling>Operate>Extrude>By XYZ Offset

- Select the area to be extruded. Enter in values '0' for 'DX' and 'DY', '0.025' for 'DZ', and '1' for 'RX','RY' and 'RZ'. Click on OK.

14. Delete the Area of 2D-model.

- Turn on the numbering by selecting **Utility Menu>PlotCtrls>Numbering ....** Check the boxes for `node number', 'Element Type number', then click on OK.

Main Menu>Preprocessor>Meshing>Clear>Areas

- Select on the area which having element type number 2.

15. Apply Boundary Conditions

MainMenu>Preprocessor>Loads>DefineLoads>Apply>Structural>Displacement>Symmetry B.C.>On Areas

- Pick the symmetry area. Click OK

MainMenu>Preprocessor>Loads>DefineLoads>Apply>Structural>Displacement>On Lines

- Pick line of the arc-shaped boundary. Select 'All DOF' and '0' for 'Displacement value'. Click OK.

16. Apply Loads

Main Menu >Preprocessor >Loads >Define Loads >Apply >Structural >Force/ Moment >On Nodes

- Pick 5 lines of nodes. Click OK. _In the window that comes up, select 'FY' as the direction of force and insert '10000' as the force value. Click OK..

SOLUTION

Main Menu>Solution>Analysis Type>New Analysis

- Make sure that `Static' is selected. Click OK.

Main Menu>Solution>Solve>Current LS

- Check your solution options listed in the `/STATUS Command' window.
- Click the OK button in the `Solve Current Load Step' window.
- Click the Yes button in the `Verify' window.
- You should see the message `Solution is done!' in the `Note' window that comes up. Close the `Note' and `/STATUS Command' windows.

POSTPROCESSING

1. Plot the Stress Distribution

General Posproc>Plot Results>Contour Plot>Nodal Solu

- A window will appear. Highlight on 'Nodal Solution', then 'Stress', and then 'Von Mises Stress'

2. Zoom the Crack-Tip Region

Utility Menu>PlotCtrls>Pan Zoom Rotate ...

- In the appeared window, click on the Win Zoom button and zoom the crack-tip region, then click on the Close button to close the window.
- Plot the nodes by selecting **Utility Menu>Plot>Nodes.**

3. Define Crack-Face Path

Main Menu>General Postproc>Path Operations>Define Path>By Nodes

- Pick the crack-tip node, then the quarter-point node, and finally the third node on the crack face. Click OK.
- Enter `K1' for `Define Path Name:', then click OK.
- Close the `PATH Command' window.

4. Define Local Crack-Tip Coordinate System

Utility Menu>WorkPlane>Local Coordinate Systems>Create Local CS>By 3Nodes

- Pick the crack-tip node, then node in the x-direction with the crack tip, and finally node in the y-direction parallel with the crack tip.
- Note from the window that the reference number of the crack-tip coordinate system is 11. Click on the OK button.

5. Activate the Local Crack-Tip Coordinate System

Utility Menu>WorkPlane>Change Active CS to>Specified Coord Sys ...

- In the window that comes up, enter `11' for `Coordinate system number', then click OK.
- To activate the crack-tip coordinate system as results coordinate system, select **Main Menu>General Postproc>Options for Outp**. In the window that appears (as shown at the top of next page), select `Local system' for `Results coord system' and enter `11' for `Local system reference no.'. Click OK.

6. Determine the Mode-I Stress Intensity Factor using KCALC

Main Menu>General Postproc>Nodal Calcs>Stress Int Factr

- In the window that opens, select `Plain strain' for `Disp extrapolat based on' and `Half-symm b.c.' for `Model Type'.
- Click on OK. The window shown at the top of next page appears and it shows that the SIFs at the crack tip are KI = 0.291186 ; KII = 0 ; KIII = 0
- Close the `KCALC Command' window.
- You may want to recover the whole meshed model by selecting **Utility Menu>PlotCtrls>Pan Zoom Rotate ...**, then click on the Fit button and close the `Pan-Zoom-Rotate' window; selecting **Utility Menu>Plot>Elements.**

7. Exit ANSYS, Saving All Data

Utility Menu>File>Exit ...

- In the window that opens, select `Save Everything' and click on OK.