Stress Intensity Factor on a Surface Crack of Cylindrical Pressure Vessel

By

Muhammed Raqib bin Mohamed Razali

A project dissertation submitted in partial fulfillment of
the requirements for the
Bachelor of Engineering (Hons)
(Mechanical Engineering)

JAN 2009

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Stress Intensity Factor on a Surface Crack of Cylindrical Pressure Vessel

by

Muhammed Raqib bin Mohamed Razali

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)
Project Supervisor

UNIVERSITI TEKNOLOGI PETRONAS

TRONOH, PERAK

January 2009

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.

____

MUHAMMED RAQIB BIN MOHAMED RAZALI

ABSTRACT

Cylindrical Pressure Vessel is widely used in the industries. The wide application of this vessel has made the studies and engineering design to be more important than before. The best design need to be obtained in order to ensure the safety, performance and reliability of the vessel. Because of wrong selection of material, design and overloading cracks can occur at the surface of the pressure vessel. This is due to high Stress intensity Factor (SIF) that occurs at the crack. This report basically discusses the research done and basic understanding of the chosen topic, which is **Stress Intensity Factor on a Surface Crack of Cylindrical Pressure Vessel**. The objective of the project is to do the investigation and understand the surface crack propagation in a cylindrical pressure vessel. A standard practice of the fracture testing is referred to the ASTM Standard Test Practice for Fracture Testing with Surface-Crack Tension Specimens (E 740 – 88). The analysis used A516 Grade 60 mild steel (0.25 carbon) as a reference of material of this project. It is based from *Pressure Vessel Design Manual, Third Edition* by Dennis R. Moss where it states the material for the surface material for cylindrical pressure vessel. Software called ANSYS had been used to analyze the effect of surface crack to the Stress Intensity Factor (SIF). Theorically, if the crack size and crack depth increase the value of Stress Intensity Factor will also increase at the crack tip of the crack. This has been proves with the result that the author get. From the analytical approach the crack length, c and crack depth, a influences the value of SIF. This result can be approved with the ANSYS analysis where the value of SIF will increase with the increasing of crack depth and crack size.

ACKNOWLEDGEMENT

First and foremost, thanks to God because of His bless, the Final Year Project is completed successfully within the designated time frame. The experiences, knowledge and skills gained from this project are very useful and beneficial in a journey to be a wise and charismatic engineer in future.

Next, a special gratitude is expressed to Dr. Khairul Fuad for his guidance, patience and supervision in completing this project. This special gratitude is also forwarded to Mr. Julendra Bambang Ariatedja for sharing his experiences and lends me their expertise especially in ANSYS.

Not to be forgotten, to the entire author’s friends who had shared their experiences and giving a lot of advices for the whole period of project. To those who had assist direct or indirectly in making this project into success, highest appreciation is given to all of them.

Finally, the author dedicates this work to family for their support, encouragement and understanding until completion of this dissertation report. Wassalam.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLE

CHAPTER 1
INTRODUCTION

1.1 BACKGROUND OF STUDY

Cylindrical Pressure Vessels are commonly used in industry to serve as boilers or tanks. It is designed to hold gases or liquids at a pressure different from the ambient pressure. The pressure differential is potentially dangerous and many fatal accidents have occurred in the history of their development and operation.

Consequently their design, manufacture and operation are regulated by engineering authorities backed up by laws. For these purposes the definition of a pressure vessel varies from country to country, but involves parameters such as the maximum safe operating pressure and temperature.

Pressure vessels are used in a variety of applications. These include the industry and the private sector. They appear in these sectors respectively as industrial compressed air receivers and domestic hot water storage tanks, other examples of pressure vessels are: diving cylinder, recompression chamber, distillation towers, rail vehicle airbrake reservoir, and road vehicle airbrake reservoir and storage vessels for liquefied gases such as ammonia, chlorine, propane, butane and LPG.

In cylindrical pressure vessel, there are many categories in failures. Based on *Pressure Vessel Design Manual*, Third Edition by Dennis R. Moss he stated that Material, Design, Fabrication and Service are the common failures for the Pressure Vessel. In *material*, the improper selection of material can influence the characteristic of the vessel. Besides that, incorrect *design* data and inaccurate design method may cause failures in Pressure Vessel. For *fabrication*, improper fabrication procedures including welding; heat treatment of forming method can make the vessel less efficiency in production. As for *service*, inexperienced operations or maintenance personnel or human error can cause failures in Pressure Vessel.

1.2 PROBLEM STATEMENT

1.2.1 Problem Identification

In Pressure Vessel, there are many types of failures. Dennis R. Moss noted that *Elastic Deformation, Brittle Fracture, Excessive Plastic Deformation, Stress Rupture, Plastic Instability, High Strain, Stress Corrosion an Corrosion Fatigue* are the types of failures that always occur to the Pressure Vessel. In dealing with these various modes of failures, the designer must know how to tackle the problems. For this project, I will stress on a surface crack of the Cylindrical Pressure Vessel. Although cracks can never be eliminated their damaging effects should be alleviated by accurately predicting the strength of the Pressure Vessel material.

The experimental studies of crack propagation in Pressure Vessel essentially try to find association between environment, temperature or fatigue loading parameter and the crack growth behaviour. The results are ample knowledge, although, these studies require great resources. Moreover, not all phenomena of crack propagation can be explored experimentally, e.g. the surface crack propagation, one of the major problems. Causing this difficulty, the analytical/numerical study becomes a preferred alternative.

1.2.2 Significance of Project

The concern of this project is to conduct mesh analysis using ANSYS on the model of surface cracks. The thickness of the cylindrical pressure vessel will be based on *Pressure Vessel Design Manual, Third Edition* by Dennis R. Moss while the standard for the specimen will be based on ASTM Standard Test Practice for Fracture Testing with Surface-Crack Tension Specimens (E 740 – 88). Here we will analyze and simulate based on the ANSYS. This step is one of the most important of entire analysis, for the decision make at this stage in the model development will profoundly affect the accuracy and economy this analysis.

1.3 OBJECTIVE AND SCOPE OF STUDY

1.3.1 Objectives

The main objectives of this research are:

- To simulate the finite element analysis of the Stress Intensity Factor using ANSYS Software.
- To investigate and determine the Stress Intensity Factor on a surface crack specimen.

.

1.3.2 Scope of Study

The scope of work is divided into two semesters. For the first semester, I will, concentrate in studying first about finite element method for the calculation of Stress Intensity Factor, *K* and cracks where I will be guided by Mr. Julendra. It comprises on the concept of the crack itself, factors that influence the *K* value, the 3-D crack problem and numerical method for *K* determination. In the second semester, I will start to do a problem that relates to the project. Instead of that, I will start to do a simulation using ANSYS where the thickness of the specimen will be based on *Pressure Vessel Design Manual, Third Edition* by Dennis R. Moss while the standard for the specimen will be based on ASTM Standard Test Practice for Fracture Testing with Surface-Crack Tension Specimens (E 740 – 88). Analysis will be conducted after getting the results.

CHAPTER 2
LITERATURE REVIEW

2.1 CYLINDRICAL VESSEL

A cylindrical pressure with wall thickness, *t*, and inner radius, *r* is considered, when vessel wall is "thin", the stress distribution throughout its thickness will not vary significantly and so we will assume that it is uniform. By definition, thin wall have a ratio of inner radius, *r* to wall thickness, *t* of $r/t \geq 10$. Using this assumption, we will now analyze the state of stress in thin-walled cylindrical pressure vessels. The pressure vessel is a gauge pressure, since it measures the pressure above atmospheric pressure, which is assumed to exist both inside and outside the vessel's wall.

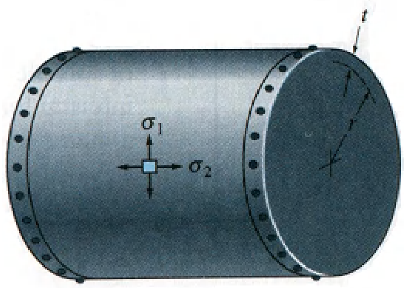


Figure 2.1 Cylindrical Thin-Walled Pressure Vessel

A gauge pressure *p* is developed within the vessel by a contained gas or fluid, which is assumed to have negligible weight. Due to uniformity of this loading, an element of the vessel that is sufficiently removed from the ends and oriented is subjected to two types of normal stresses: hoop, σ_H or σ_1, and longitudinal, σ_L or σ_2, that both exhibit tension of the material. These components exert tension on the material.

Several assumptions are made in this method.

1. Plane sections are remain plane
2. $r/t \geq 10$ with t being uniform and constant
3. The applied pressure, p is the gauge pressure (note that p is the difference between the absolute pressure and the atmospheric pressure)
4. Material is linear-elastic, isotropic and homogeneous
5. Stress distributions throughout the wall thickness will not vary
6. Element of interest is remote from the end of the cylinder and other geometric discontinuities
7. Working fluid has negligible weight

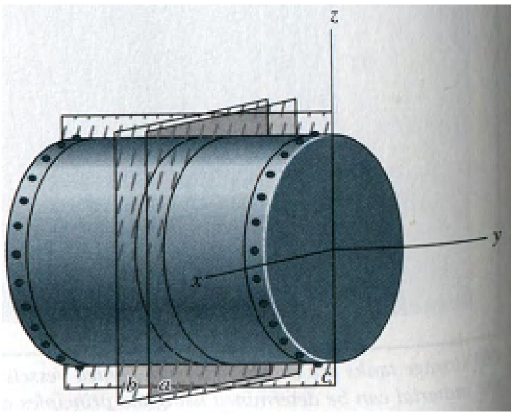


Figure 2.2 Cylindrical Thin-Walled Pressure Vessel Showing Coordinate Axes and Cutting Planes (a, b and c)

For Hoop Stress, we consider the pressure vessel sectioned by planes a, b and c for Figure 2.2. A free body diagram of a half segment along with the pressurized working fluid in Figure 2.3. Note that the x-direction with equilibrium and only the loading in the x-direction is shown and the internal reactions in the material are due to hoop stress acting on incremental areas, A, produced by the pressure acting on projected area, A_P. And these loading are developed by the uniform hoop stress σ_H, acting throughout the vessel's wall, and the pressure acting on the vertical face of the sectioned gas or liquid. We require:

$$\Sigma F_x = 0; \qquad 2[\sigma_H A] - pA_P = 0$$

$$2[\sigma_H(t\,dy)] - p(2r\,dy) = 0 \qquad (2.1)$$

$$\sigma_H = \text{—}$$

where *dy* = incremental length, *t* = wall thickness, *r* = inner radius, *p* = gauge pressure

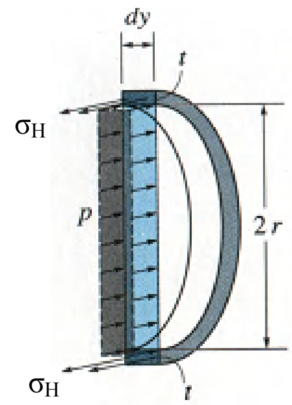


Figure 2.3 Free-body Diagram of Segment of Cylindrical Thin-Walled Pressure and Internal Hoop Stresses

To obtain the longitudinal stress, σ_L we consider the left portion of section b of the cylinder pressure vessel in Figure 2.2. A free body diagram of a half segment along with the pressurized working fluid is shown in Figure 2.3. Note that the Longitudinal Stress, σ_L acts uniformly throughout the wall, and p acts on the section of fluid. Since the mean radius is approximately equal to the vessel's inner radius, equilibrium in y-direction requires:

$$\Sigma F_y = 0; \qquad \sigma_L A - pA_e = 0$$

$$\sigma_L \pi (r_o^2 - r^2) - p\pi r^2 = 0$$

Or solving for σ_L

$$\sigma_L = \frac{p\pi r^2}{\pi(r_o^2 - r^2)}$$

Substituting $r_o = r + t$ gives

$$\sigma_L = \frac{p\pi r^2}{\pi[(r+t)^2 - r^2)]} = \frac{p\pi r^2}{\pi(r^2 + 2rt + t^2 - r^2)}$$

$$\sigma_L = \frac{pr^2}{(2rt + t^2)}$$

Since this a thin wall with a small t, t^2 is smaller and can be ignored such that after simplification

$$\sigma_{L=} — \tag{2.2}$$

Where r_o = inner radius and σ_L is the Longitudinal Stress.

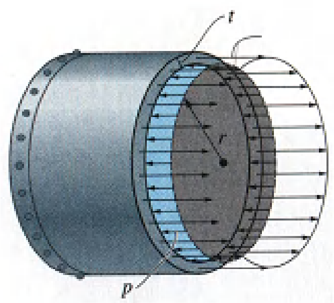


Figure 2.4 Free-Body Diagram of End Section of Cylindrical Thin-Walled Pressure Vessel Showing Pressure and Internal Longitudinal Stresses

From the equation of hoop stress and longitudinal stress, it should be noted that the hoop stress is twice as large as the longitudinal stress.

2.2 FRACTURE MECHANICS

Fracture mechanics is the discipline concerned with the behavior of materials containing cracks or small flaws. Flaws refer to features as small pores (holes), inclusion, or micro-cracks. Fracture toughness measures the ability of a material containing a flaw to withstand an applied load. The stress applied to the material is intensified at the flaw, which acts as a stress raiser. The stress applied to the material is intensified at the flaw, which act as a stress raiser. Stress Intensity Factor (SIF), K is

$$K = f\,\sigma\,\sqrt{\pi a}, \qquad (2.3)$$

Where f is a factor of geometry for the specimen and flaw, σ is the applied stress, and a is the flaw size. The analytical expression for K changes with the geometry of the crack body on its size, and on the mode and magnitude of loading. Since the applied stress, σ is perpendicular to the crack plane; the linear elastic bodies must undergo proportional stressing. It means all stress components at all locations increase in proportion to the remotely applied forces. Thus the crack tip stresses must be proportional to the applied stress, and $K \alpha \sigma$.

The deformation fields produced by the symmetric and anti-symmetric stress field results in characteristically different motions of the crack faces. The symmetric part results in an opening displacement of the crack faces, whereas the anti-symmetric part imparts a shearing motion. For K_I, it is the *opening mode* K_{II}, *forward shear mod*e and K_{III}, *anti shear mode* respectively. The third deformation mode produces a scissoring motion of the crack faces.

In opening mode, the linear elastic bodies must undergo proportional stressing, i.e., all the stress components at all locations increase in proportion to the remotely applied forces.

For opening mode, K_I

$$\begin{aligned}
\sigma_x &= \frac{K}{\sqrt{2\pi r}}\cos\frac{\theta}{2}\left[1 - \sin\frac{\theta}{2}\sin\frac{3\theta}{2}\right] + \sigma_{ox} \\
\sigma_y &= \frac{K}{\sqrt{2\pi r}}\cos\frac{\theta}{2}\left[1 + \sin\frac{\theta}{2}\sin\frac{3\theta}{2}\right] \qquad (2.4) \\
\tau_{xy} &= \frac{K}{\sqrt{2\pi r}}\cos\frac{\theta}{2}\sin\frac{\theta}{2}\cos\frac{3\theta}{2}
\end{aligned}$$

From the equations we can stated that the distribution of the stresses, strains and displacements in some small region around a crack tip are always the same for any cracked body in which the stress state around the crack tip is dominated by the inverse-square-root singularity.

For forward shear mode, K_{II}

$$
\begin{aligned}
\sigma_x &= \frac{K_{\mathrm{II}}}{\sqrt{2\pi r}} \sin\frac{\theta}{2}\left[2 + \cos\frac{\theta}{2}\cos\frac{3\theta}{2}\right] \\
\sigma_y &= \frac{K_{\mathrm{II}}}{\sqrt{2\pi r}} \sin\frac{\theta}{2}\cos\frac{\theta}{2}\cos\frac{3\theta}{2} \qquad (2.5) \\
\tau_{xy} &= \frac{K_{\mathrm{II}}}{\sqrt{2\pi r}} \cos\frac{\theta}{2}\left[1 - \sin\frac{\theta}{2}\sin\frac{3\theta}{2}\right]
\end{aligned}
$$

From the equations above K give a significant role in the fracture mechanics. It results the concept that a characteristic feature of crack-tip stress fields is that the distribution of the stress in the neighborhood of a crack tip is universal and that only the magnitude singular term (i.e., K) varies with the geometry and the type of loading.

Because of the importance of this concept to the formulation of a stress-based theory of fracture, it is intrusive to estimate the size of the region around the crack tip for which singular term describes the distribution of the stresses to a reasonable degree of accuracy.

As we stressed before, the Stress Intensity Factor, K is proportional to the applied stress. This relationship is a direct consequence of the linear nature of the theory of elasticity. Second, K contains the crack length as a parameter. Therefore, the stress intensity factor is size dependent. In addition, the stress intensity factor must be function of the geometry of the body.

Therefore, we can infer that, in general, the stress intensity factor must be of the form

$$= \sqrt{\quad} \; . \qquad - \tag{2.6}$$

where *Y (a / W)* is a dimensionless shape factor that embodies the effects of all of the geometric parameters and *W* is any characteristic in-plane dimension (often the width of the body). For the central-crack problem, *Y (a / W) = 1*.

In one case, if there are σ_y at the crack plate, there will be a singular stress at the respective crack. Even though, there are singular stress happens there it doesn't mean that the crack will grow. This is because there is another parameter that affects the crack which is Stress Intensity Factor, *K*.

2.2.1 THE THREE DIMENSIONAL CRACK PROBLEM

In three dimensional problems, thumbnail crack often leads to failure in structures that lack local stress raiser (such as threads, holes, or notches). Examples of this problem include welded pressure vessels, pipe and ship hulls.

This problem is often called the penny-shaped crack problem. Below representing the parametric of a point on an ellipse. We can see that all points along the flaw border are obtained by the horizontal and vertical projections of points on the radial intercept of parametric angle, ø with circles whose radii represent the semiminor and semimajor axes of the ellipse. Note that only when ø=0, and π does the parametric angle agree with the polar-angle measure of the crack perimeter position.

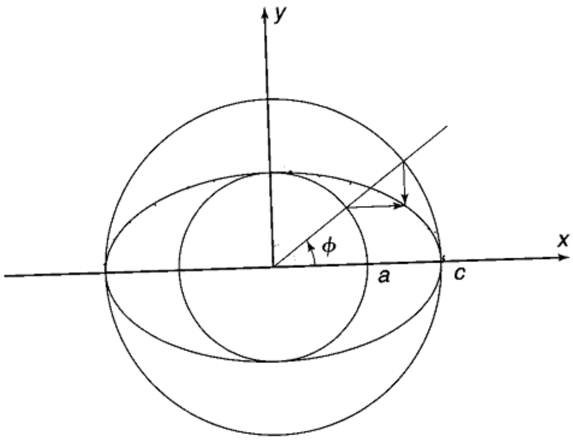


Figure 2.5 Parametric representation of a point on an ellipse.

Therefore, the expression for the geometric stress intensity factor at any point around the perimeter of the crack is

$$K = \frac{\sigma\sqrt{\pi a}}{\Phi} \tag{2.7}$$

where

$$\Phi = \int_0^{\frac{\pi}{2}} \left[1 - \left(\frac{c^2 - a^2}{c^2}\right)\sin^2\phi\right]^{\frac{1}{2}} d\phi \tag{2.8}$$

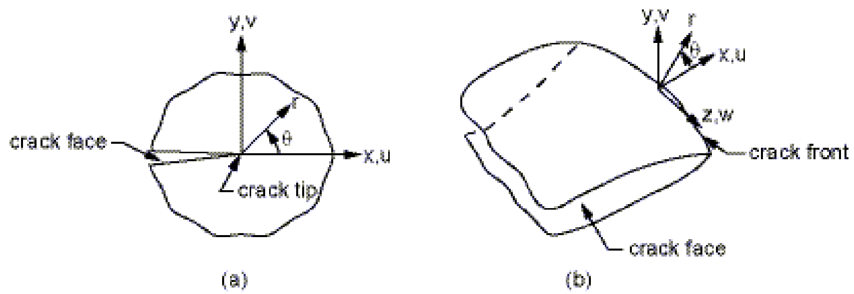
From it, we observe that the stress intensity factor is not constant along the crack border, but rather varies from a minimum along the major axis (ø = 0, π) to a maximum along the minor axis (ø =–).

2.3 STRESS INTENSITY FACTOR CALCULATION IN ANSYS

In ANSYS, it calculates the mixed-mode stress intensity factors K_I, K_{II} and K_{III}. The command is limited to linear elastic problem with a homogeneous, isotropic material near the crack region. Here, we use KCALC command where the analysis uses a fit of the nodal displacements in the vicinity of the crack. To use KCALC, the steps are below

1. **Define a local crack-tip or crack-front coordinate system.**

   The X axis must be parallel to the crack face (perpendicular to the crack front in 3-D models) and the Y axis perpendicular to the crack face.



(a) 2-D model (b) 3-D model

Figure 2.6 Crack Tip and Crack Front

   This coordinate system must be the active model coordinate system (**CSYS**) *and* results coordinate system (**RSYS**) when **KCALC** executes.

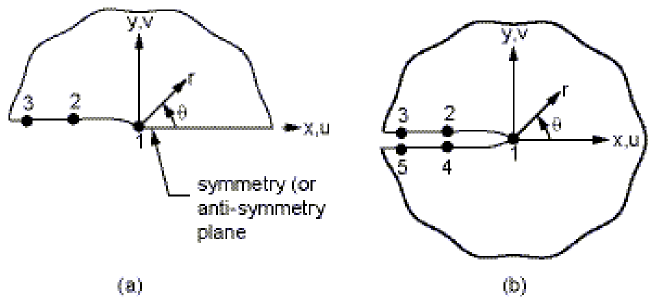
Command(s):	**LOCAL** (or **CLOCAL**, **CS**, **CSKP**, etc.
GUI:	**Utility Menu> WorkPlane> Local Coordinate Systems> Create Local CS> At Specified Loc**

2. **Define a path along the crack face.**

   The first node on the path should be the crack-tip node. For a half-crack model, two additional nodes are required, both along the crack face. For a full-crack model, where both crack faces are included, four additional nodes are required: two along one crack face and two along the other.

Command(s):	**PATH** (or **PPATH**)
GUI:	**Main Menu> General Postproc> Path Operations> Define Path**

The following figure illustrates the two cases for a 2-D model.



(a) half-crack model and (b) full-crack model

Figure 2.7 Typical Crack Face Path Definitions

3. **Calculate K_I, K_{II}, and K_{III}.**

   The KPLAN field on the **KCALC** command specifies whether the model is plane-strain or plane-stress. Except for the analysis of thin plates, the asymptotic or near-crack-tip behavior of stress is usually thought to be that of plane strain. The KCSYM field specifies whether the model is a half-crack model with symmetry boundary conditions, a half-crack model with antisymmetry boundary conditions, or a full-crack model.

   Command(s): KCALC
   GUI: Main Menu> General Postproc> Nodal Calcs> Stress Int Factr

The actual displacements at and near a crack for linear elastic materials are:

$$u = \frac{K_I}{4G}\sqrt{\frac{r}{2\pi}}\left((2\kappa-1)\cos\frac{\theta}{2}-\cos\frac{3\theta}{2}\right)-\frac{K_{II}}{4G}\sqrt{\frac{r}{2\pi}}\left((2\kappa+3)\sin\frac{\theta}{2}+\sin\frac{3\theta}{2}\right)+0(r)$$

$$v = \frac{K_I}{4G}\sqrt{\frac{r}{2\pi}}\left((2\kappa-1)\sin\frac{\theta}{2}-\sin\frac{3\theta}{2}\right)-\frac{K_{II}}{4G}\sqrt{\frac{r}{2\pi}}\left((2\kappa+3)\cos\frac{\theta}{2}+\cos\frac{3\theta}{2}\right)+0(r) \qquad (2.9)$$

$$w = \frac{2K_{III}}{G}\sqrt{\frac{r}{2\pi}}\sin\frac{\theta}{2}+0(r)$$

where;

u, v, w = displacement in a local Cartesian coordinate system as shown in Figure 2.8: " Local Coordinates Measured From a 3-D Crack Front".

r, θ = coordinates in a local cylindrical coordinate system as shown in Figure 2.8: " Local Coordinates Measured From a 3-D Crack Front".

G = shear modulus

K_I, K_{II}, K_{III} = stress intensity factors relating to deformation shapes as shown in Figure 2.5: "The Three Basic Mode of Fracture"

$$\kappa = \begin{cases} 3 - 4\nu & \text{if plane strain or axisymmetric} \\ \dfrac{3\nu}{1+\nu} & \text{if plane stress} \end{cases} \tag{2.10}$$

ν = Poisson's ratio

0(r) = terms of order r or high

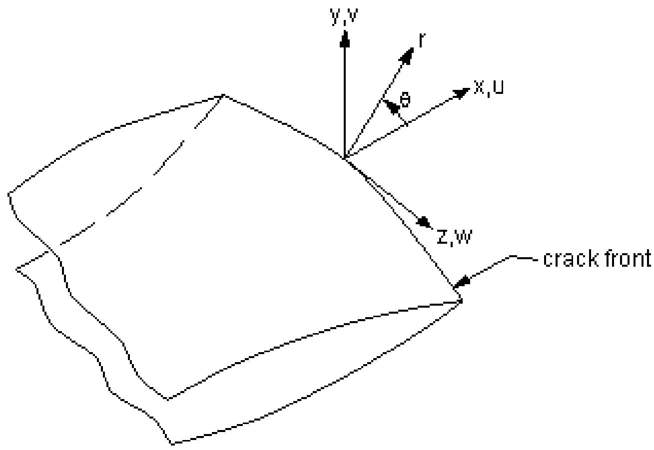


Figure 2.8 Local Coordinates Measured From a 3-D Crack Front

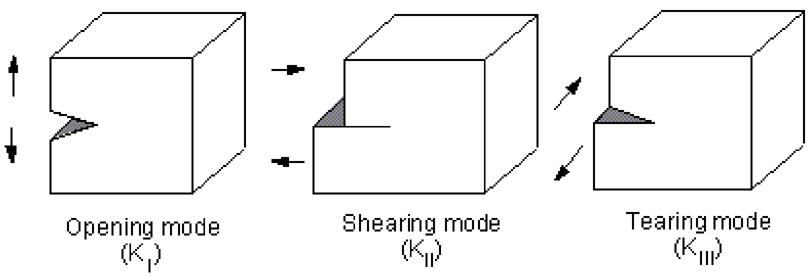


Figure 2.9 The Three Basic Modes of Fracture

For models symmetric about the crack plane (half-crack model, Figure 2.6: "Nodes Used for the approximate Crack-Tip Displacements" (a)) can be reorganized to give:

$$K_I = \sqrt{2\pi}\,\frac{2G}{1+\kappa}\,\frac{|v|}{\sqrt{r}}$$
$$K_{II} = \sqrt{2\pi}\,\frac{2G}{1+\kappa}\,\frac{|u|}{\sqrt{r}} \qquad (2.11)$$
$$K_{III} = \sqrt{2\pi}\,2G\,\frac{|w|}{\sqrt{r}}$$

and for the case of no symmetry (full-crack model, Figure 2.10: "Nodes Nodes Used for the approximate Crack-Tip Displacements" (b)),

$$K_I = \sqrt{2\pi}\,\frac{G}{1+\kappa}\,\frac{|\Delta v|}{\sqrt{r}}$$
$$K_{II} = \sqrt{2\pi}\,\frac{G}{1+\kappa}\,\frac{|\Delta u|}{\sqrt{r}} \qquad (2.12)$$
$$K_{III} = \sqrt{2\pi}\,\frac{G}{1+\kappa}\,\frac{|\Delta w|}{\sqrt{r}}$$

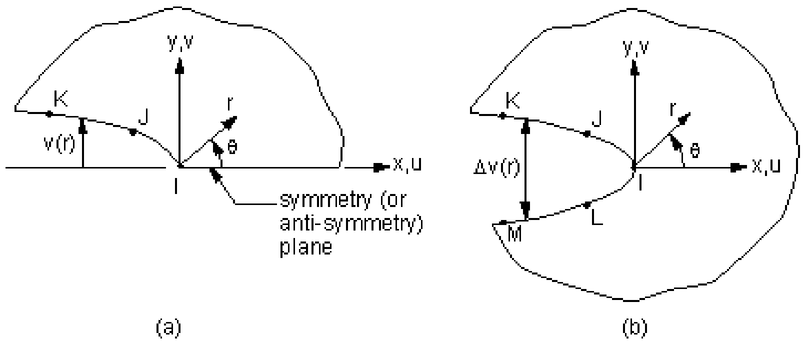
where Δv, Δu, and Δw are the motions of one crack face with respect to the other.

As the above six equations are similar, consider only the first one further. The final factor is $\frac{|\ |}{\sqrt{}}$, which needs to be evaluated based on the nodal displacement and locations. As shown in Figure 2.10: "Nodes Used for the Approximate Crack-Tip Displacements" (a), three points are available. *v* is normalized s that *v* at node I is zero. Then A and B are determined so that

$$\frac{|\ |}{\sqrt{}} = A + Br \qquad (2.13)$$

at point J and K. next, let r approach 0.0:

$$\lim_{r \to 0} \frac{|v|}{\sqrt{r}} = A \qquad (2.14)$$



(a) Half model (b) Full Model

Figure 2.10 Nodes Used for the Approximate Crack-Tip Displacements

Thus, the equation becomes:

$$K_I = \sqrt{2\pi}\,\frac{2GA}{1+\kappa} \tag{2.15}$$

CHAPTER 3
METHODOLOGY

3.1 PROJECT IDENTIFICATION

3.1.1 Project Activities

This project will involve the design and development activities as shown in Figure 3.1.

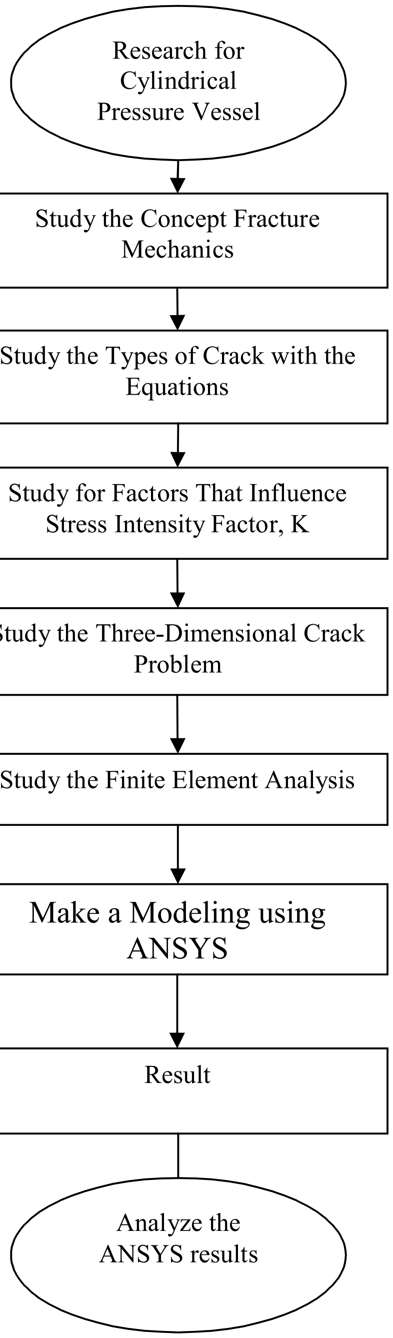


Figure 3.1 Flow Process of entire project

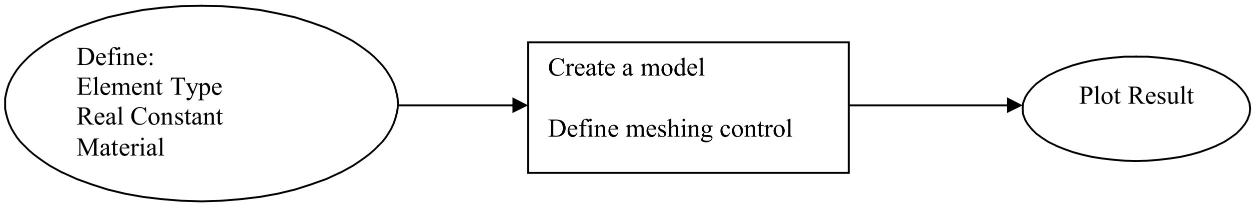


Figure 3.2 Process flow of ANSYS

3.2 ANSYS Analysis Flow

Figure 1.3; show a few steps that used during the simulation of the thin plate. The analysis is divided into two separate preprocessing, solution, and post processing steps.

3.2.1 Preprocessing Phase

Table 3.1 Preprocessing Phase

Element Type	PLANE82 and SOLID95 is a higher order version of the 2-D, provides more accurate results for mixed (quadrilateral-triangular) automatic meshes and can tolerate irregular shapes without as much loss of accuracy. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities.
Real Constant	Element real constant are properties that depend on the element type, such as Poisson ratio of an element.
Meshing	Meshing is creating a finite element model by dividing the geometry into nodes and element. The elements attributes include elements type, real constant and material properties. The element sizes control the finesse of the mesh. The smaller the element size, the finer the mesh.
Material Properties	Material properties can be linear or non-linear. Linear material properties can be constant or temperature dependant and isotropic. As with element types and real constants, each set of material properties has reference numbers.
Modeling	There are two approaches to constructing a finite model's geometry that are direct generation and solid modeling approach.

3.2.2 Solution Phase

Define Load

Apply boundary condition, initial condition and loading such as pressure for this study.

Solution

Solve a set of linear and non-linear algebraic equation and loading a pressure for this case.

3.2.3 Post Preprocessing Phase

Read Result

List the result in tabular form. From the result we can make comparison with another case.

Plot Result

Contour display is used to see the distribution of certain variables, such as component of stress. The most important thing is we can get the information about the von misses stress.

3.3 Tools

ANSYS – Generate mesh element analysis and computational fluid dynamics software. Make accurate results in solving the thermal distribution and stress distribution using mesh analysis.

CHAPTER 4

RESULT AND DISCUSSION

4.1 SPECIMEN IDENTIFICATION

4.1.1 Specimen Description

The dimensions for the specimen are based on the ASTM E 740 – 88 (1995). It stated that the specimen width, W should be 5 times the crack length 2c and the specimen test section length, L should be twice the width, W. During the analysis, only a quarter of the specimen was modeled due to symmetrical shape of the specimen. For this specimen, the parameters are as below:

Width, W = 5 inches (0.127m);

Length, L = 10 inches (0.254m);

Crack depth, a = 0.1 inch (0.00254m), 0.15 inch (0.00381m), 0.20 inch (0.00508m), 0.25 inch (0.00635m) and 0.30 inch (0.00762m).

For the thickness of the specimen, it is based on *Pressure Vessel Design Manual*, Third Edition by Dennis R. Moss. It stated that if the vessel diameter is 72 inches (1.8288m) with the allowable stress is 20,000 psi (137MPa), joint efficiency is 1.0, and then the vessel thickness will be 1 inch (0.0254m).

Besides that, the material of the specimen is a mild steel A516 Grade 60 (0.25 carbons). The properties for the specimen are:

Poisson Ration = 0.3;

Modulus of Elasticity = 30 x 10^6 psi (206 x 10^9 Pa)

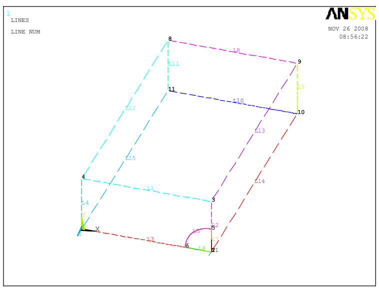


Figure 4.1 Quarter specimen

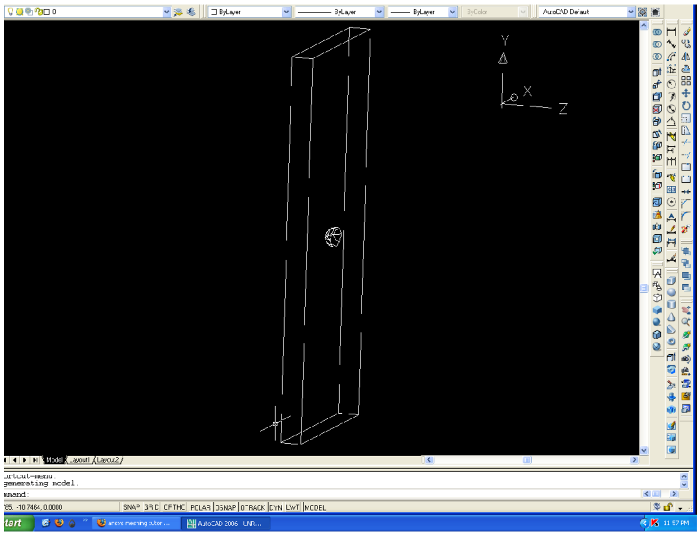


Figure 4.2 Actual size specimens

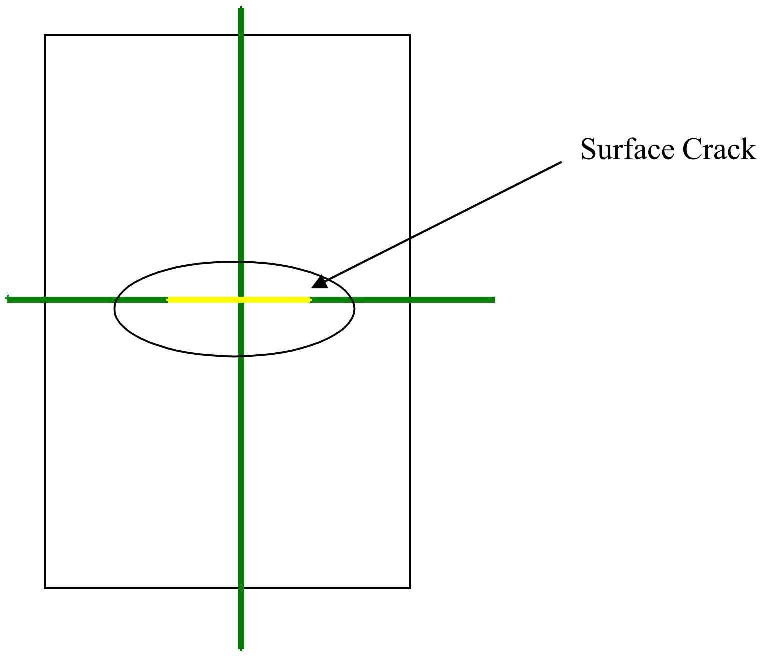


Figure 4.3 The actual size of the specimen will be divided into 4 segments.

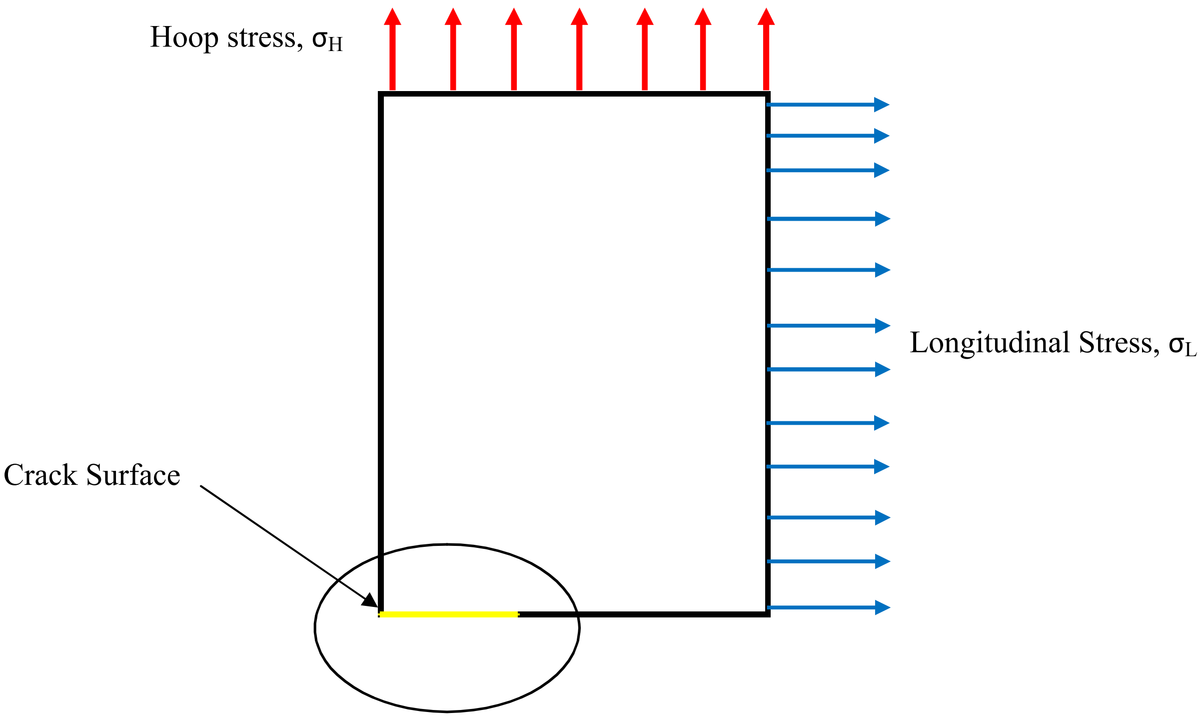


Figure 4.4 Stresses around the specimen

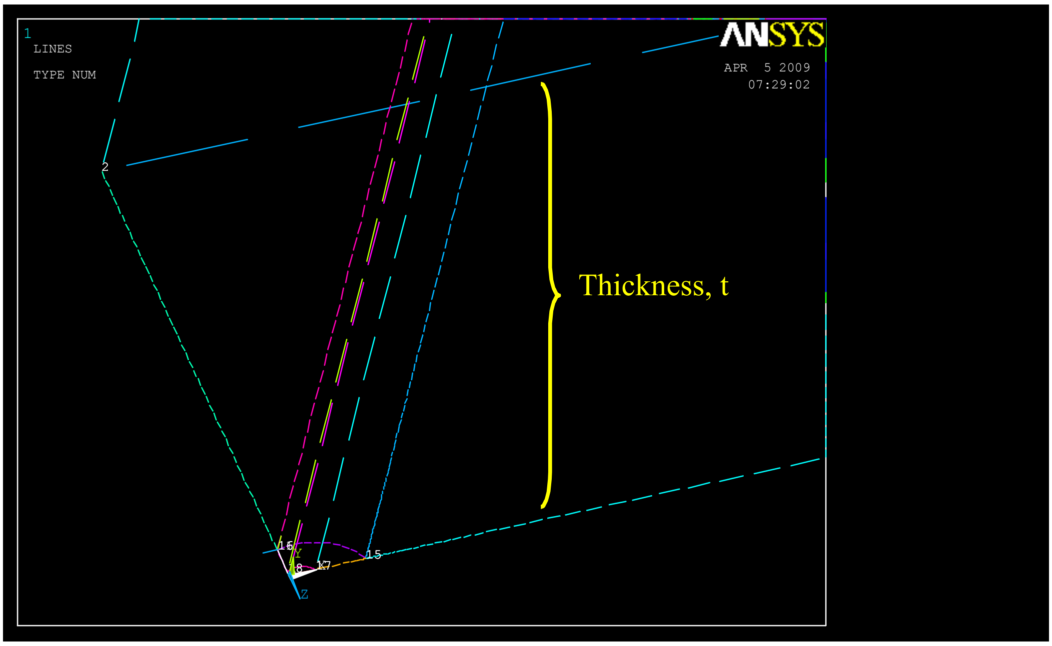


Figure 4.5 Thickness, t of the specimen

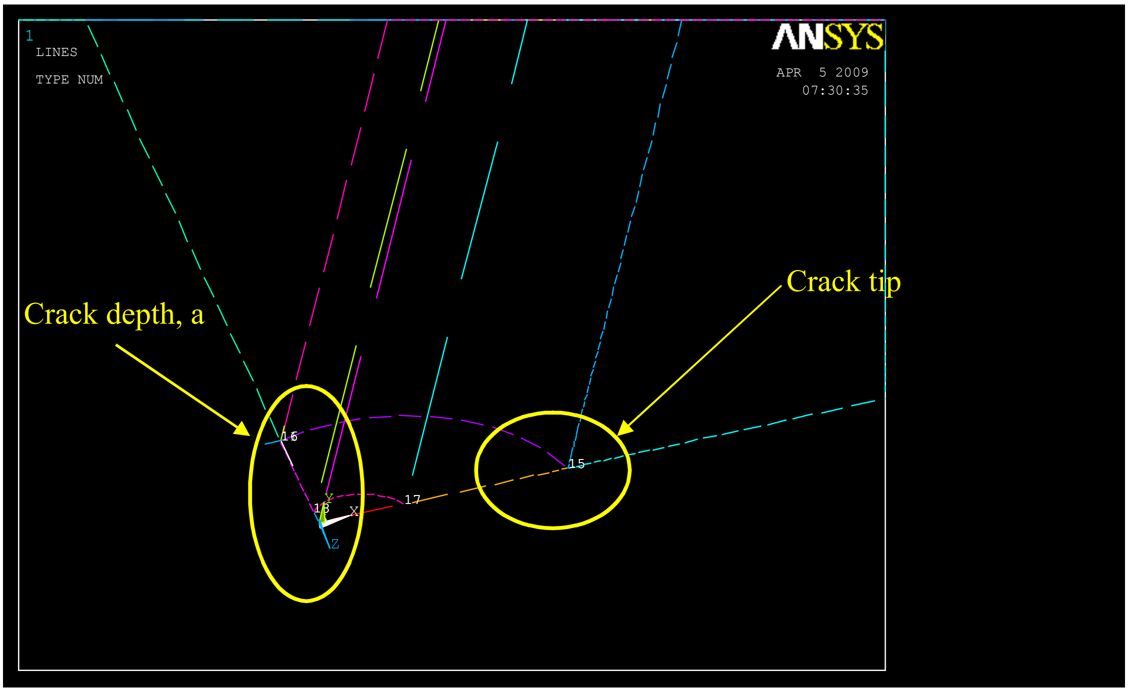


Figure 4.6 The locations for crack depth, a and the crack tip

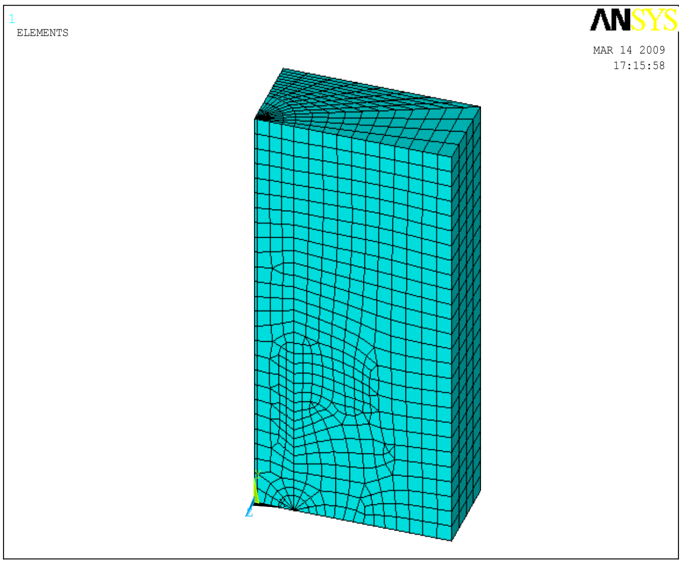
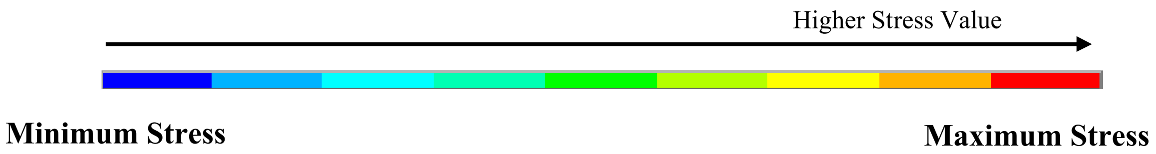


Figure 4.7 Quarter size specimens after mesh

The specimen has a 0.005m in element radius on a crack tip, the number of element around it will be 4, and the radius ratio is 0.05.

4.2 RESULT AND DISCUSSION

In ANSYS, the author had put a load at the surface of the model. From the Finite Element Analysis (FEA) using ANSYS, Figure 4.1, 4.2 and 4.3 shows the stress field around the crack tip. We can see that the stress concentration is higher at the crack tip compared to other area of the region.



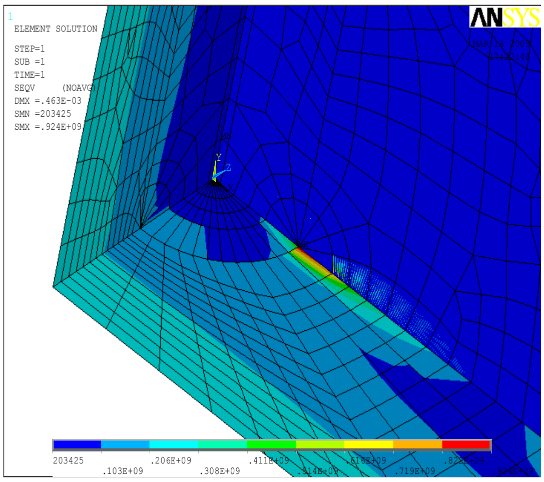


Figure 4.8 Stress around the crack

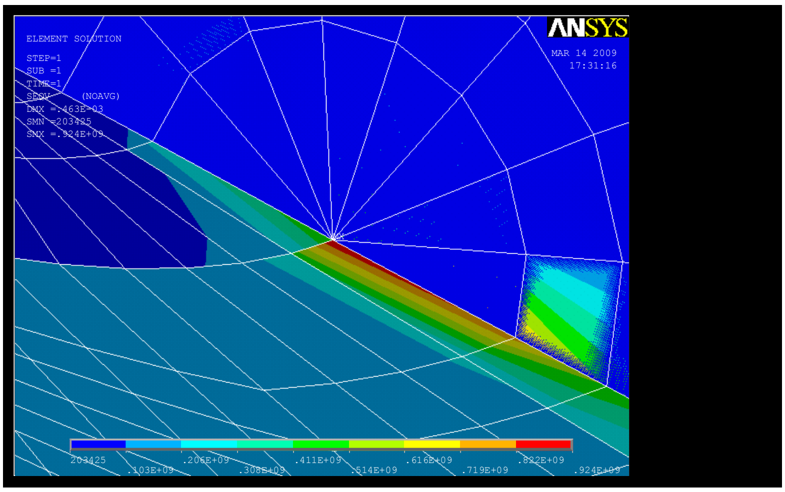


Figure 4.9 Maximum stress on a crack tip

The crack size is called a penny-shaped crack. Based on *Principles of Fracture Mechanics* by R. J. Sanford, it was shown that for the penny-shaped crack geometry

$$= - \quad \sqrt{\ } \quad = 0.64 \ \sqrt{\ } \qquad (4.1)$$

; a = crack depth

He also noted that the expression for the geometric stress intensity factor at any point around the perimeter of the crack is:

$$K = \sigma\sqrt{\pi a} \,/\, \Phi. \qquad (4.2)$$

; a = crack depth,
Φ= angle from the center of the flaw to the perimeter of the crack.

From here, we observe that the stress intensity factor is not constant along the crack border, but rather varies from a minimum along the major axis (ø = 0, π) to a maximum along a minor axis (ø=π/2).

In this project the author will make a comparison of Stress Intensity Factor, *K* with different crack depth, *a* which are 0.1in, 0.15in, 0.2in, 0.25in and 0.30in with a different ratio between the crack size and crack depth, c/a. The value for it will be 1.4, 1.5 and 1.6. Based on the equation above, we can see that once the crack depth increase the value of *K* will also increase. This is due to a high stress concentrate at the crack tip. Below is the graph for c/a=1.4, 1.5, 1.6.

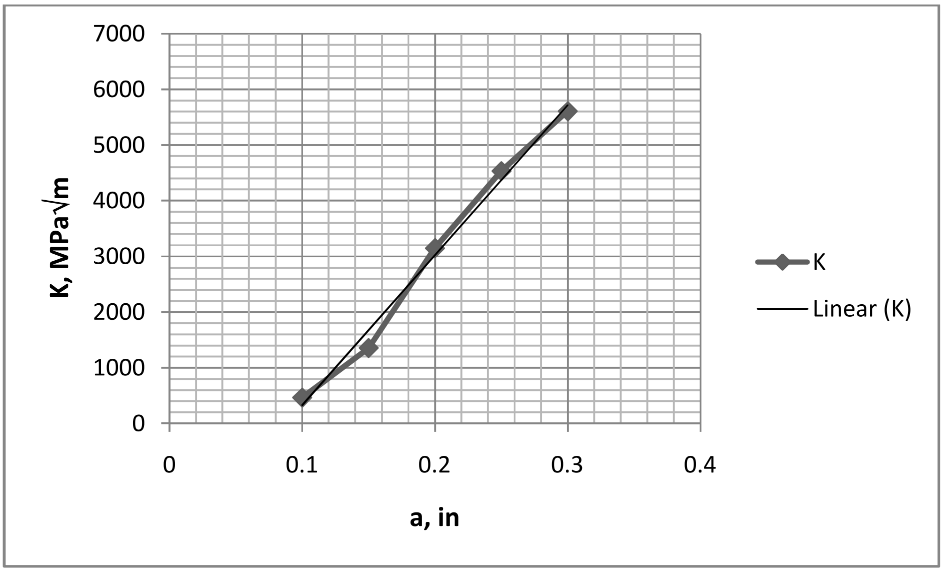


Figure 4.10 Graph K vs a for c/a=1.4

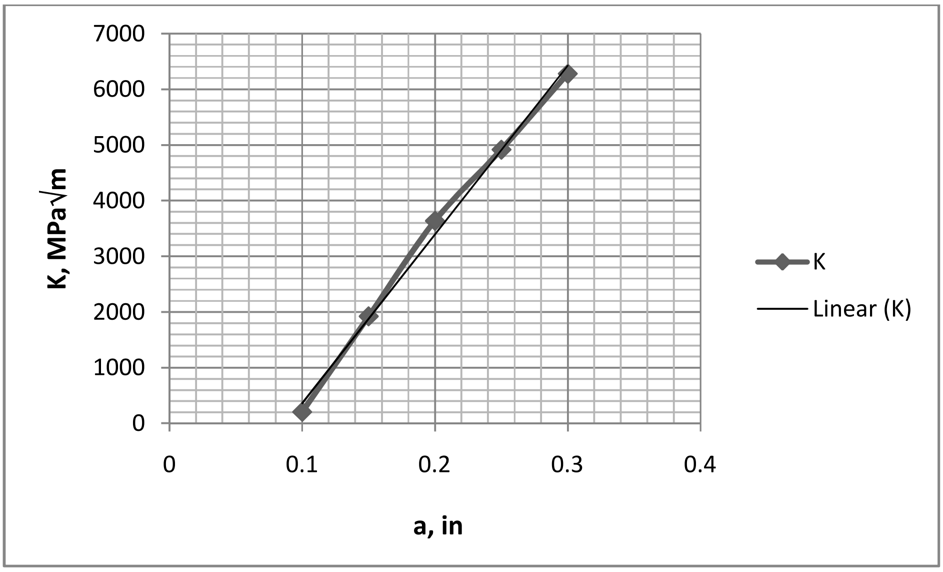


Figure 4.11 Graph K vs a for c/a=1.5

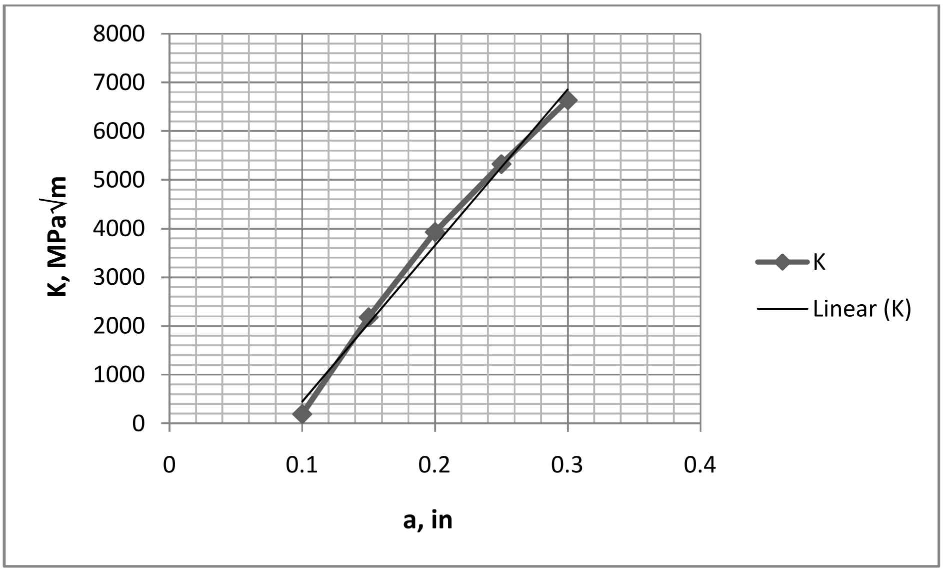


Figure 4.12 Graph K vs a for c/a=1.6

From the author observed, we can say that when crack depth increases the value of Stress Intensity Factor, K will also increase. Based on *Fracture Mechanics, Fundamentals and Applications Third Edition* by T.L. Anderson it stated that since a tensile stress cannot be transmitted through a crack, the lines of force are diverted around the crack, resulting in a local stress concentration. In the infinite plate, the line of force at a distance W from the crack centerline has force components in the x and y directions. If the plate width is restricted to 2W, the x force must be zero on the free edge; this boundary condition causes the lines of force to be compressed, which results in higher stress intensification at the crack tip.

Due to that, the factor of the crack depth influences the stress intensity factor. With the increase of the crack depth, *a* there will be an increment for stress intensity factor at the crack tip.

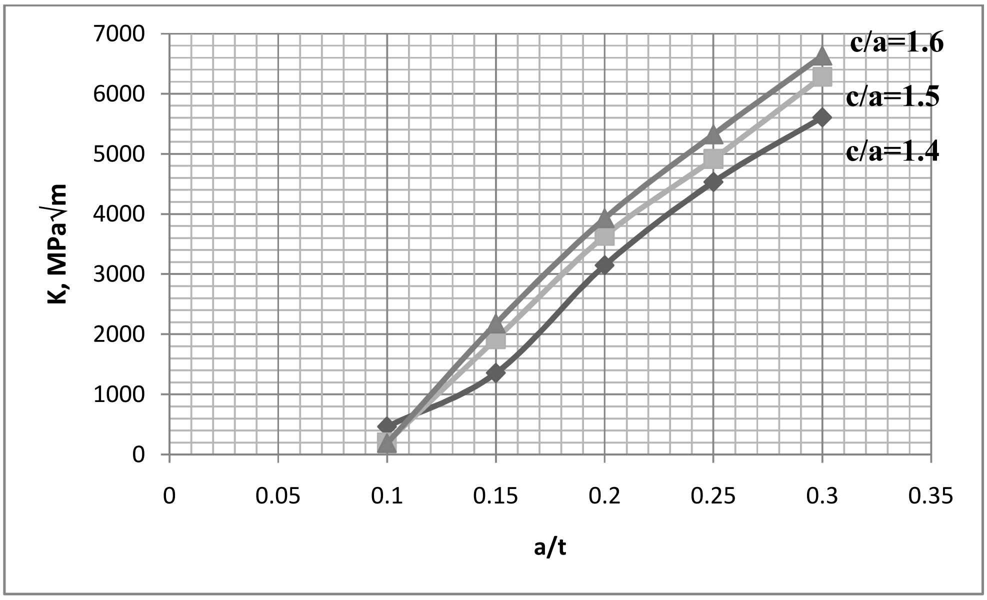


Figure 4.13 Graph K vs a/t

With a constant thickness, *t* of 1 inch for the specimen the author would like to make a ratio between the crack depth, *a* and the thickness, *t*. By that there will be dimensionless function. From the above graph, we can see that the value for stress intensity factor for c/a=1.6 are much higher than stress intensity factor for c/a=1.5 and c/a=1.4. We can say here that the crack length, c also influence the value of stress intensity factor. It is based on the equation from *Fracture Mechanics, Fundamentals and Applications Third Edition* by T.L. Anderson where Q, flaw shape parameter is:

$$Q = 1 + 1.464(-)^{1.65} \tag{4.3}$$

And the stress intensity factor, K is:

$$K = \lambda_s \sigma(\ \overline{}\ -\ )\, f(\varnothing) \tag{4.4}$$

From here, we can see that crack length also influence the value of *K*. If the crack length increase, the value of *K* will also increase. That is why the value of *K* for c/a=1.6 are much higher than c/a=1.4 and 1.5.

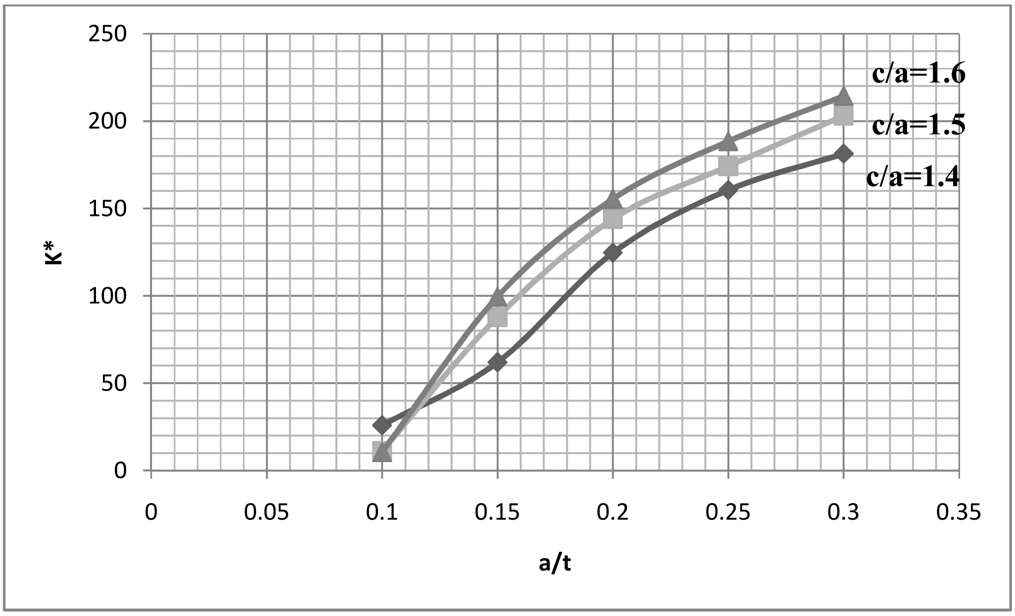


Figure 4.14 Graph K* vs a/t

From the table above, the author make a dimensionless function for the value of stress intensity factor, K^*. The equation for K^* is:

$$K^{*}=K_{actual} / \sigma\sqrt{} \tag{4.5}$$

The K_{actual} will be from the ANSYS value calculation while the value of crack depth, a will be 0.1in, 0.15in, 0.20in, 0.25in, and 0.3in. All of them will be converted to the (International Standards) SI unit. From here, we can see that the stress intensity factor is high for c/a=1.6 rather than stress intensity factor for c/a=1.5 and 1.4.

4.3 EXPERIMENTAL ERROR

4.3.1 Meshing Consistency

From the analysis, several data obtained from the different types if mesh. This situation is due the certain geometry that could not be compute using certain types of mesh and also the maximum stress value that are not constantly increase or decrease. Therefore, different type of mesh been tried to ensure that the point a closely on the curve.

CHAPTER 5
CONCLUSION AND RECOMMENDATION

5.1 Conclusion

From the results obtained, we can see that the results are on the same current with the equation which are if $ø = 0, \pi$ then the *K* will be in high while $ø=\pi/2$, then the *K* will be in minimum. Besides that, from the analytical approach the crack length, c and the crack depth, *a* influences the value of *K*. When the value of c or a increase, the value of *K* will also increase. Hence, the elliptical integral, Φ also influence the value of K around the parameters of the crack and the maximum value of *K* will be at the crack tip and the minimum value of *K* around the crack parameter will be at the crack depth. This is suitable with the results what the author get.

5.2 Recommendation

In future, the recommendation is done by the dimension of the model need to take account to get the accurate results in ANSYS.

5.3 GANTT CHART: Final Year Project 2 (Key Milestone)

NO	TRAINING ACTIVITIES	WEEK NO													
		1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	Project Work Continue														
2	Submission of Progress Report 1														
3	Project Work Continue														
4	Submission of Progress Report 2														
5	Seminar														
6	Project Work Continue														
7	Poster Exhibition														
8	Submission of Project Dissertation														
9	Oral Presentation														
10	Submission of Project Dissertation														

REFERENCES

1. T. L. Anderson, Department of Mechanical Engineering, Texas A&M University College Station, Texas, 1995, "Fracture Mechanics Fundamentals and Applications", Second Edition , CRC Press

2. R. J. Sanford, Professor Emeritus Mechanical Engineering, University of Maryland, College Park, 2003, "Principles of Fracture Mechanics", Prentice Hall

3. Dominique P. Miannay, 1998, "Fracture Mechanics, Mechanical Engineering Series",Springer

4. Standard Practice for Fracture Testing with Surface-Crack Tension Specimens, ASTM E 740 – 88, 1995.

5. R. C. Hibbeler, 2005, "Mechanics of Materials" SI Second Edition, Pearson, Prentice Hall

6. Donald R. Askeland and Pradeep P. Phule, 2003, "The Science and Engineering of Materials", Fourth Edition, Thomson Brooks/Cole

7. Dennis Moss, 2004, "Pressure Vessel Design Manual", Third Edition, Elsevier

8. James R. Farr and Maan H. Jawad, 1998, "Guidebook for the Design of ASME Section VIII Pressure Vessel, ASME PRESS

APPENDICES

1. Table for K vs c/a (from ANSYS)

For c/a=1.4

a, inch	K, MPa √m
0.10	463.55
0.15	1355.7
0.20	3148.3
0.25	4533.1
0.30	5608.4

For c/a=1.5

a, inch	K, MPa √m
0.10	202.89
0.15	1921.3
0.20	3636.9
0.25	4916.3
0.30	6281.8

For c/a=1.6

a, inch	K, MPa √m
0.10	188.26
0.15	2177.2
0.20	3927.5
0.25	5322.5
0.30	6633.7

2. Table for K using analytical approach at certain crack depth, *a*

Using equation $K = \sigma\sqrt{\quad}$

a, inch	K, MPa √m
0.10	17.865
0.15	21.881
0.20	25.266
0.25	28.248
0.30	30.944

3. Table for K* vs a/t (dimensionless function)

For c/a=1.4

a/t	K*
0.1	25.947
0.15	61.958
0.2	124.61
0.25	160.48
0.3	181.24

For c/a=1.5

a/t	K*
0.1	11.36
0.15	87.81
0.2	143.94
0.25	174.05
0.3	203.01

For c/a=1.6

a/t	K*
0.1	10.54
0.15	99.5
0.2	155.45
0.25	188.42
0.3	214.38

1. ANSYS Command

PATH - The **PATH** command is used to define parameters for establishing a path.

PPATH - Defines a path by picking or defining nodes, or locations on the currently active working plane, or by entering specific coordinate locations.

Example: **Main Menu>General Postproc>Path Operations>Define Path>By Location**

LOCAL - Defines a local coordinate system by a location and orientation. The local coordinate system is parallel to the global Cartesian system unless rotated. Rotation angles are in degrees and redefine any previous rotation angles.

CLOCAL - Defines and activates a local coordinate system by origin location and orientation angles relative to the active coordinate system. This local system becomes the active coordinate system, and is automatically aligned with the active system. Example: x is radial if a cylindrical system is active, etc.

CS - Defines and activates a local right-handed coordinate system by specifying three existing nodes: to locate the origin, to locate the positive x-axis, and to define the positive x-y plane. This local system becomes the active coordinate system.

CSKP - Defines and activates a local right-handed coordinate system by specifying three existing keypoints: to locate the origin, to locate the positive x-axis, and to define the positive x-y plane. This local system becomes the active coordinate system.

CSYS - The **CSYS** command activates a previously defined coordinate system for geometry input and generation.

RSYS - Activates a coordinate system for printout or display of element and nodal results.

KCALC - Calculates the stress intensity factors (K_I, K_{II}, and K_{III}) associated with homogeneous isotropic linear elastic fracture mechanics.

MAIN MENU - A menu path represents the complete location of a particular function in the Graphical User Interface (GUI) . The first part of the path (Main Menu) determines where the function is found. It is usually either the Main Menu or the Utility Menu.

GENERAL POSTPROC - Once the desired results data are stored in the database, you can review them through graphics displays and tabular listings.

PATH OPERATIONS - Describes basic ANSYS operations such as starting, stopping, interactive or batch operation, using help, and use of the graphical user interface (GUI).

WORKPLANE - By default, when you initiate your ANSYS session, there is a working plane located on the global Cartesian X-Y plane, with its x and y axes colinear with the global Cartesian X and Y axes.

LOCAL COORDINATE SYSTEMS - In many cases, it may be necessary to establish your own coordinate system, whose origin is offset from the global origin, or whose orientation differs from that of the predefined global systems.