SIMPLE DAMAGE MODEL FOR CRACKED THIN PLATE TENSILE TEST

MUHAMMAD IZZAT SHAZALI BIN MOHD RAMLI

A report submitted in partial fulfilment of the requirements
For the award of the degree of
Of Mechanical Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

JUNE 2012

SUPERVISOR’S DECLARATION

I hereby declare that I have checked this project report and in my opinion this project is satisfactory in terms of scope and quality for the award of the degree of Mechanical Engineering

Signature :

Name of Supervisor : PN. NORHAIDA BINTI AB. RAZAK

Position : LECTURER

Date : 27 JUNE 2012

STUDENT'S DECLARATION

I declare that this report titled "*SIMPLE DAMAGE MODEL OF CRACKED THIN PLATE TENSILE TEST*" is my result of my own research except as stated in the references. This report has not been accepted for any degree and is not concurrently submitted for award of other degree.

Signature :
Name : MUHAMMAD IZZAT SHAZALI BIN MOHD RAMLI
Id. Number : MC10025
Date : 27 JUNE 2012

IN THE NAME OF ALLAH,
THE MOST BENEFICENT, THE MOST MERCIFUL

A special dedication of this grateful feeling to my beloved late father, Mr Mohd Ramli Bin Hassan and my mother, Pn Rosnani Binti Md Nor for giving me full of moral and financial support. It is very meaningful to me in order to finish up my degree's study. Not forgetting also to all my loving brothers and sisters. Last but not least to all my colleagues and my lovely friends.

Thanks for all the support, wishes and love.

ACKNOWLEDGEMENTS

I am grateful and would like to express my sincere gratitude to my supervisor Pn Norhaida Binti Ab Razak for her germinal ideas, invaluable guidance, continuous encouragement and constant support in making this report. I appreciate her consistent support from the first day I applied to graduate program to these concluding moments. I am truly grateful for her progressive vision about my training in science, her tolerance of my naive mistakes, and her commitment to my future career.

My sincere thanks go to all my class mates and members of the staff of the Mechanical Engineering Department, University Malaysia Pahang, especially to Corrosion and Fracture Focus Group (CFRAC) who helped me in many ways and made my stay at UMP pleasant and unforgettable. Many special thanks go to member engine research group for their excellent co-operation, inspirations and supports during this study.

I acknowledge my sincere indebtedness and gratitude to my parents for their love, dream and sacrifice throughout my life. I cannot find the appropriate words that could properly describe my appreciation for their devotion, support and faith in my ability to attain my goals. Special thanks should be given to my committee members. I would like to acknowledge their comments and suggestions, which was crucial for the successful completion of this study.

ABSTRACT

Simple damage model was a preliminary study in order to identify the need of assessment of method for cracked thin plate because the important for safe operation and maintenance especially in oil and gas industry. This study was dividing into 2 step which is experimental and FEA analysis using MSC PATRAN Software. For this project, the Finite Element Analysis result will compare to the experiment and the parameter will be added to see the change in the coalescence load versus crack size diagram between experiment and simulation. For the experiment, the project scope was focus on the API Grade B Specimen that has been machining into ASTM E8 standard size of tensile test specimen. Crack size for the specimen has been varying into 3 different sizes which are 4mm, 6mm, and 8mm crack size in order to examine the maximum tensile stress, strain and load for different crack size of the specimen. Afterward, using MSC Marc Patran Simulation the parameter will be added to generate simple damage model equation accurately and the results of finite element analysis can simplify working process and reduce working time to do an experiment. In the analysis, the parameters used are 2mm, 4mm, 6mm and 8mm crack size and also included in the analysis the material yield strength and poisson ratio. Based on the experimental and analysis, stress versus strain graph obtained and the result of the maximum stress, strain and load for the uncrack specimen and other parameter are compared. Besides that, simple damage model equation was obtained from load versus crack size diagram in order to predict the maximum load from various size of crack by using crack as the x parameter and y as the value of maximum load from finite element analysis result. These finding led to the conclusion that the maximum load are proportional to the crack size which is bigger the crack size of the specimen, the maximum load are decreased.

ABSTRAK

Model kerosakan mudah untuk plat nipis yang mempunyai retakan dan ujian tegangan adalah satu kajian awal untuk menentukan perlunya untuk membuat penaksiran tentang kaedah untuk plat nipis yang mempunyai retakan kerana ia penting di dalam operasi yang selamat dan penyelenggaraan terutama di dalam industri minyak dan gas. Kajian ini telah dibahagikan kepada 2 bahagian iaitu eksperimen dan FEA analisis dengan menggunakan perisian MSC PATRAN. Untuk projek ini, FEA analisis akan dibandingkan dengan hasil kajian daripada eksperimen yang telah dilakukan dan parameter akan ditambahkan untuk melihat perubahan tautan antara beban melawan saiz retakan diantara eksperimen dan simulasi. Pertamanya, untuk eksperimen skop untuk projek ini adalah fokus kepada penggunaan bahan daripada API Gred B yang telah diproses menggunakan mesin kepada saiz yang telah ditetapkan di dalam piawaian ASTM E8 untuk sampel ujian tegangan. Saiz retakan untuk sampel – sampel yang diperlukan telah dibezakan kepada 3 saiz retakan yang berlainan iaitu 4mm, 6mm dan 8mm dalam usaha untuk mengkaji tekanan tegangan dan bebanan maksimum untuk saiz retakan yang berlainan. Selepas itu, dengan menggunakan simulasi MSC Marc Patran, parameter akan ditambah untuk menghasilkan persamaan model retakan mudah yang lebih tepat dan hasil kajian daripada simulasi ini juga dapat mengurangkan proses kerja dan mengurangkan masa kerja untuk melakukan eksperimen. Daripada analisis, parameter yang digunakan adalah 2mm, 4mm, 6mm dan 8mm saiz retakan dan di dalam simulasi ini juga disertakan ‘yield strength’ dan ‘poisson ratio’ untuk bahan yang digunakan. Berdasarkan eksperimen dan analisis, graf Stress melawan strain diperoleh dan semua data yang diperoleh akan dibandingkan antara simulasi dan juga eksperimen untuk setiap parameter. Selain itu, persamaan model kerosakan mudah telah diperolehi daripada beban melawan saiz retakan untuk meramalkan beban maksimum dari pelbagai saiz retakan dengan menggunakan retakan sebagai parameter x dan nilai beban maksimum sebagai y daripada hasil FEA analisis. Penemuan ini membawa kepada kesimpulan bahawa beban maksimum adalah berkadar terus dengan saiz retakan dimana yang semakin besar saiz retakan sesuatu sampel, beban maksimum semakin menurun.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

σ	Stress
ϵ	Strain
σe	Engineering Stress
$\sigma \tau$	True Stress
$\epsilon \tau$	True Strain
ϵe	Engineering Strain
ln	Natural Log
C	Carbon
Mn	Manganic
P	Phosphorus
S	Sulphur
Si	Silicon

LIST OF ABBREVIATIONS

API	American Petroleum Institute
MSC	MacNeal-Schwendler Corporation
HAZ	Aluminium
ASTM	American Society for Testing and Materials
FEA	Finite element analysis
EDM	Electro – discharge machining
CMD	Command Prompt

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

This chapter will explain briefly about the synopsis of this study and some background of the study about simple damage model for cracked thin plate tensile test. This chapter consist background of the study, objective, scopes and also problem statement.

1.2 BACKGROUND OF THE STUDY

It is impossible to keep petroleum and natural gas transmission pipelines free from defects in the manufacturing, installation and servicing processes. The damage might endanger the safety of pipelines and even shorten their service life. Gas or petroleum release due to defects may jeopardise the surrounding ecological environments with associated economic and life costs. Also, steel structures such as pipelines for offshore and onshore industry are prone to suffer various types of damage as they get older. Under the action of repeated loading, fatigue cracks may be initiated in the stress concentration areas of the piping. The threshold for crack initiation increases with the pre-deformation due to a strain hardening effect, while the fatigue resistant factor exhibits a maximum with pre-deformation owing to its special dependence on fracture strain and fracture strength. The result is expected to be beneficial to the understanding of the effect of damage on the safety of pipelines and fatigue life prediction. (Jiang, Y.& Chen, M., 2012)

1.3 PROBLEM STATEMENT

There are several methods to predict failure on the steel structures for cracked structure especially pipelines but sometimes accident happens and it might endanger life and harm certain parties. The need of assessment of method for cracked thin plate is important because of the structural engineering is increasing important for safe operation and maintenance especially in oil and gas industry. Even the best designed and maintained pipeline will become defective as it progresses through its design life. Therefore, operators need to be aware of the effect these defects will have on their pipeline, and more importantly be able to assess their significance in terms of the continuing integrity of the pipeline.Despite the convenience provided by simulation software, however there are still errors that arise due to material properties, technical issues and less proper procedures while performing the analysis. Besides that, the tensile tests on cracked thin plate are important in order to assume the lifespan of the pipelines.

1.4 OBJECTIVE

The objectives of this study are:

i) To determine the stress strain curve for cracked thin plate.
ii) To develop a load versus crack size diagram using a various crack size and generate simple damage model equation for cracked thin plate tensile test.

1.5 SCOPES

This project will focus on the following points:

i) Material used is API Steel 5L Grade B.
ii) Tensile test for cracked tensile specimen to obtain the stress strain curve
iii) To simulate the crack by using Software MSC Marc 2008 r1 and compare the experiment test and simulation.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

This chapter will explained briefly about the pipeline and relation to the important of tensile test on cracked thin plate. It includes material used in pipeline, defects on pipelines, ultimate tensile strength of steel plates with cracking damage, method to predict failure behaviour and also about tensile test. Literature review is important to know the previous study that related to this project

2.2 INTRODUCTION TO PIPELINES

Pipeline is important in transportation natural gas and other products in oil and gas industry either in offshore or onshore. The most defect occur are corrosion but fatigue cracking is another important factor of age related structural degradation, which has been a primary source of costly repair work of aging steel structures. Cracking damage has been found in welded joints and local areas of stress concentrations such as at the weld intersections of longitudinal, frames and girders. Fatigue cracking has usually been dealt with as a matter under cyclic loading, but it is also important for residual strength assessment under monotonic extreme loading, because fatigue cracking reduces the ultimate strength significantly under certain circumstances.(Paik, J. K., *et al.*, 2004). There are few factors that will affect pipeline failure performance such as good design, materials and operating practices. In this chapter, there are type of defect that usually occurs, material selection of common pipelines and also the method to predict failure behaviour.

Figure 2.1below shows a schematic representation of the nonlinear behaviour of cracked steel structures under monotonic loading. It is noted that for similar structures the stiffness and ultimate strength of cracked structures is smaller than those of uncracked structures.(Paik, J. K., *et al.*, 2004).

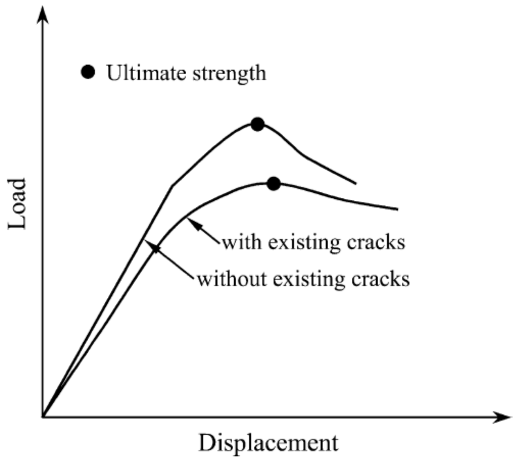

Figure 2.1: A schematic representation of the cracking damage effect on the ultimate strength behaviour of steel structures.

Source: Paik, J. K., *et al.*, 2004

2.3 MATERIAL USED IN PIPELINE

Materials used in pipelines are varying be influenced by on the type of element that will be transporting by the pipeline. Today, the X70 pipeline steel is widely used in the world, the X80 pipeline steel began to apply in some developed countries and the research and development of X100/X120 pipeline steel is being studied in the recent years. In our country, the X60 pipeline steel is widely used in the working pipelines. The X70 pipeline steel is used in the West-East Gas Pipeline Project. Most of the researches are focus on the fatigue failure of the pipeline steel, especially the fatigue crack propagation of the pipeline steels. (Jiang, Y.& Chen, M. 2012).

The researches and industrialization of pipeline steel fatigue crack propagation are summarized, especially the X60 and X70 pipeline steel after the mechanical damage and in the synthetic soil solution. (Jiang, Y.& Chen, M. 2012).Line pipe grade designations come from API Spec 5L Specification for Line Pipe. For standard pipeline, the grade are A and B but the stronger grades have the designation of X. For example, X42 until X80. Table 2.1 shows the physical properties of the line pipe.

Table 2.1: Physical properties of the line pipe

API 5L Grade	Yield Strength min. (MPa)	Tensile Strength min.(MPa)	Yield to Tensile Ratio (max.)	Elongation min.%
A	207	331	0.93	28
B	241	414	0.93	23
X42	290	414	0.93	23
X46	317	434	0.93	22
X52	358	455	0.93	21
X56	386	490	0.93	19
X60	414	517	0.93	19
X65	448	530	0.93	18
X70	482	565	0.93	17
X80	551	620 ~ 827	0.93	16

Source: www.woodcousa.com, Internet Sources

API 5L elongation figures vary with specimen dimensions. As for the elongation taken in table above, the value are for 130 mm^2.

2.4 DEFECTS IN PIPELINE

Oil and gas transmission pipelines basically have a good safety records. This is due to a combination of decent design, materials and operating observes; however, like any engineering structure, pipelines at times will fail. The most common causes of damage and failures in onshore and offshore transmission pipelines are mechanical damage which is cracks and corrosion. (Jiang, Y.& Chen, M. 2012).

2.4.1 Corrosion

Corrosion is an electrochemical process it usual appears as either general corrosion or localised corrosion. Figure 2.2 below shows the irregular length, width and depth of a typical corrosion defect. There are many different types of corrosion, including galvanic corrosion, microbiologically induced corrosion, AC corrosion, differential soils, differential aeration and cracking. It can occur on the internal or external surfaces of the pipe, in the base material, the seam weld, the girth weld, and/or the associated heat affected zone (HAZ). (Cosham, A.& Hopkins, P. 2003).

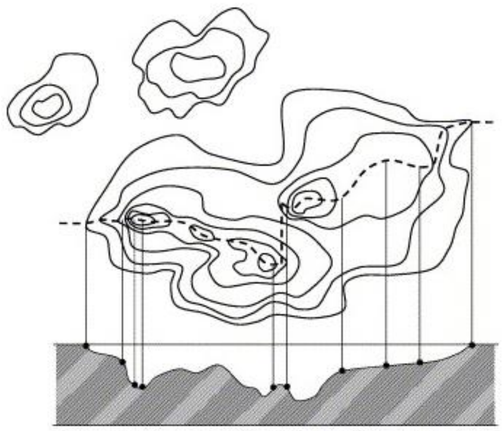

Figure 2.2: The irregular length, width and depth of a typical corrosion defect

Source: Cosham, A., & Hopkins, P. 2003

2.4.2 Dents

A dent in a pipeline is a permanent plastic deformation of the circular cross section of the pipe. A dent is a gross distortion of the pipe cross-section and Figure 2.3 show the dimension of the dent. Dent depth is defined as the maximum reduction in the diameter of the pipe compared to the original diameter. This definition of dent depth includes both the local indentation and any divergence from the nominal circular cross-section. (Cosham, A., & Hopkins, P. 2003).

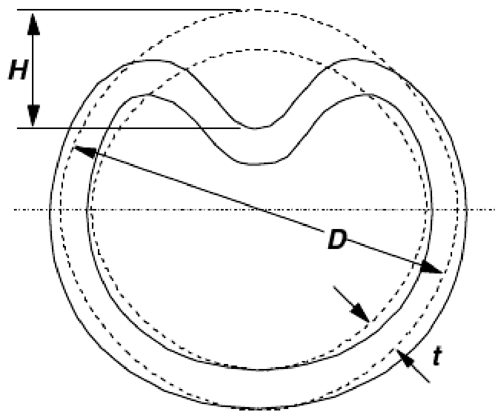

Figure 2.3: Dimension of the dents

Source: Cosham, A., & Hopkins, P. 2003

According to Cosham, A.& Hopkins, P. 2003, there are few different type of dent exists. For example, smooth dent which is a dent which causes a smooth changes in the curvature of the pipe wall in a pipelines. Besides that, kinked dent which is a dent which causes an abrupt change in the curvature of the pipe wall (radius of curvature (in any direction) of the sharpest part of the dent is less than five times the wall thickness). Another type is smooth dent that contains no wall thickness reductions (such as a gouge or a crack) or other defects or imperfections (such as a girth or seam weld), unconstrained dent which is dent free to rebound elastically (spring back) when the indenter is removed, and is free to reround as the internal pressure changes.

2.5 CAUSES OF PIPELINE FAILURE

Failure of an operating gas pipelines is a rare event. It is extremely serious event but it statistics shows that failures only occur once in a year per thousand miles of pipelines. Yet, when failure occur prevention must be apply because of the potential of losing life. It must be well analysed to prevent relapse. Figure 2.4 shows the number of gas pipelines services incident versus year of occurrence by cause.(Giedon, D.N and Smith R.B. 1980.)

Based on the Figure 2.4, over half of the operating pipelines failures are resulting from some externally applied mechanical force and also shows failure that occurs by cause which is outside force, material failure and corrosion.

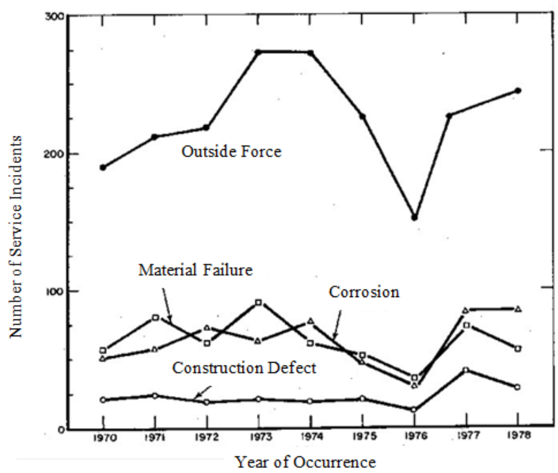

Figure 2.4: Number of gas pipeline service incidents versus year of occurrence by cause.

Source: Giedon, D.N and Smith R.B., 1980

2.5.1 Outside Force

External interference, mostly third party activity involving interference using machinery, has been recognised as a dominant failure mechanism both in gas and oil-industry pipelines. Precise records of the location and the depth of a pipeline should always be kept and communicated to any contractors before commissioning of planned work in the area. All other types of incidents appear to have some kind of connection with the activities and safety measures taken or not taken by the operator. (Papadakis, G. A. 1999)

2.5.2 Material Failure

As for material defect, it is not common causes of service failures because they are usually found before the pipe is placed in service, either during inspection of the pipe or during hydrostatic testing. (Giedon, D.N and Smith R.B., 1980)Construction and material defects (caused during processing or fabrication) are often connected with equipment associated with the pipeline. (Papadakis, G. A. 1999)

2.5.3 Corrosion

Corrosion is another major causative factor for incidents and mostly attacks pipelines as they are ageing. It can cause failures by thinning the wall over a large area or localized pitting. There also another form of corrosion which is stress-corrosion cracking that also can lead into failures. This failure is results from the accumulation of moisture on the pipe surface at imperfections in the pipe coating. Stress corrosion cracking in pipelines is identified by the distinctive intergranular nature of the crack. (Giedon, D.N and Smith R.B., 1980)

2.6 STRESS - STRAIN CURVE

One of the most common mechanical stress–strain tests is performed in tension. The tension test can be used to ascertain several mechanical properties of materials that are important in design. A specimen is deformed, usually to fracture, with a gradually increasing tensile load that is applied uniaxial along the long axis of a specimen. The tensile testing machine is designed to elongate the specimen at a constant rate and to continuously and measure the instantaneous applied load and the resulting elongations. (William D. Callister. 2006).

Typical stress- strain curve normally have four deformation changes as shown in Figure 2.5 below. It started from point a, which the elastic deformation starting to occur and it occur only to strain of about 0.005. As the material is deformed beyond this point, the stress is no longer proportional to strain and after that plastic deformation occurs. Point b the yield strength occur, that the point where the transition between elastic – plastic on the deformation. The point of yielding may be determined as the initial departure from linearity of the stress–strain curve. After yielding, the stress necessary to continue plastic deformation increases to a maximum strength at point c and then decreases to the eventual fracture, point d. The tensile strength is the stress at the maximum on the engineering stress–strain curve. This corresponds to the maximum stress that can be sustained by a structure in tension and if this stress is applied and maintained, fracture will result.(William D. Callister. 2006)

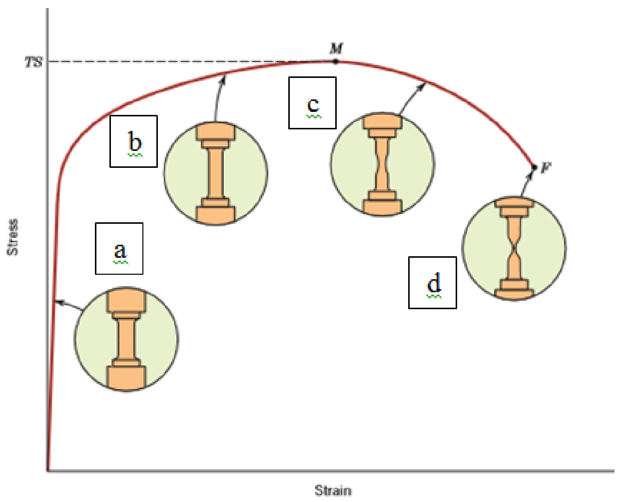

Figure 2.5: Typical engineering stress– strain behaviour to fracture

Source: William D. Callister, 2006

2.6.1 True Stress – Strain Curve Compare to Engineering Stress – Strain Curve

The engineering stress is the load taken by the sample divided by the original area. Meanwhile the true stress is the load that gets by the sample divided by a variable the instantaneous area as shown in the Figure 2.6 below. The figure shows the comparison of engineering and true stress-strain curves. Note that the true stress always rises in the plastic, whereas the engineering stress rises and then falls after going through a maximum.

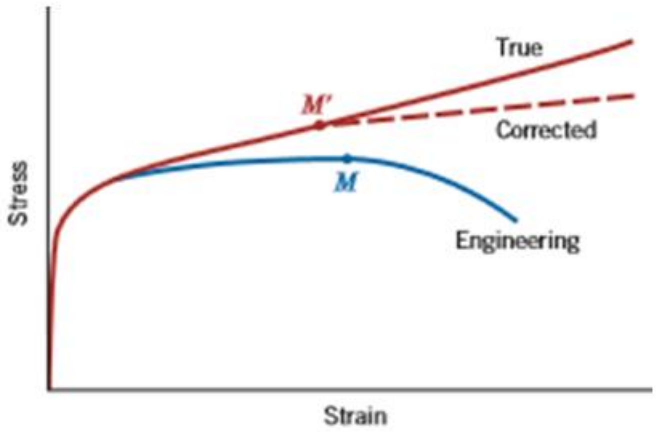

Figure 2.6: A comparison of typical tensile engineering stress–strain and true stress–strain behaviours

Source: William D. Callister, 2006

Necking begins at point M on the engineering curve, which corresponds to on the true curve. The “corrected” true stress– strain curve takes into account the complex stress state within the neck region. Engineering stress and strain can be computed from the experiment which is the engineering stress,σeget from the load measured in the tensile test divide to the original area. Meanwhile, the engineering strain,ϵecan be get from deformation divide with original length as shown in equation (2.1) and (2.2). (Ling, 1996).

$$\sigma e = \frac{P}{AO} \quad (2.1)$$

$$\epsilon e = \frac{\Delta l}{l O} \quad (2.2)$$

$$\sigma\tau = \sigma(1 + \epsilon) \quad (2.3)$$

$$\epsilon\tau = ln\ (1 + \epsilon) \quad (2.4)$$

Equations (2.3) and (2.4) are the true stress and strain that can be computed from actual load, cross-sectional area, and gauge length measurements.

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

This chapter will explain about the detail of step that had been taken in order to finish the project. In order to finish this project smoothly, the method used must be clear. Methodology also include all about process from start until end of this project. Detail of material selection, equipment, experiment procedure, preparation and also test setup. Besides that, this chapter also will cover about specimen preparation, machining process, tensile test, and finite element analysis (FEA) and etc.

3.2 OVERVIEW PROCEDURE

The main of this study are to determine the stress and strain curve for cracked thin plate and to determine the fracture strain and strain equation for cracked thin plate. This chapter show the step and procedure to get the data for cracked thin plate and to be compared to the failure behaviour that will be predicted using MSC PATRAN software. The procedures that will explain are from the beginning of the process until end of the simulation step are shown in this chapter. Figure 3.1 below shows the flow chart of the whole project measure.

3.2.1 Project Flow Chart

Project Flow below showed the step from start the project until the end.

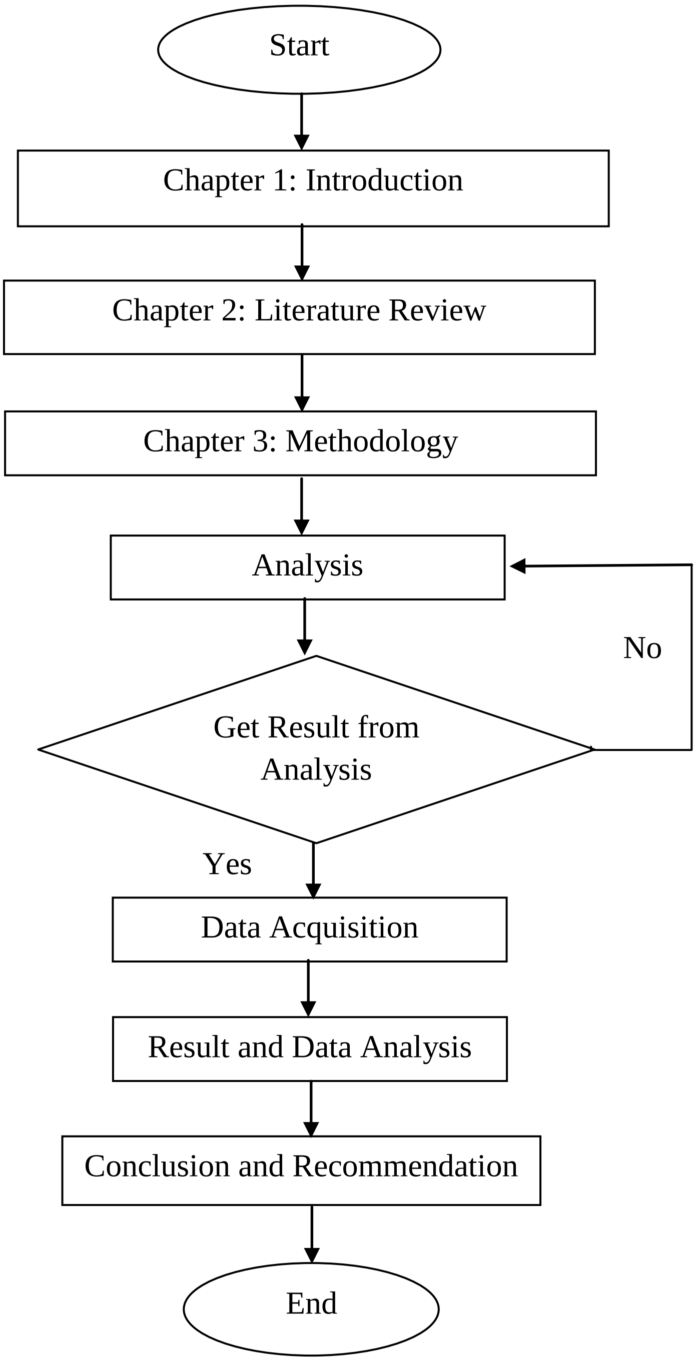

Figure 3.1: Project Flow Chart

3.2.2 Procedure

Complete procedures on this project are shown in figure 3.2 below.

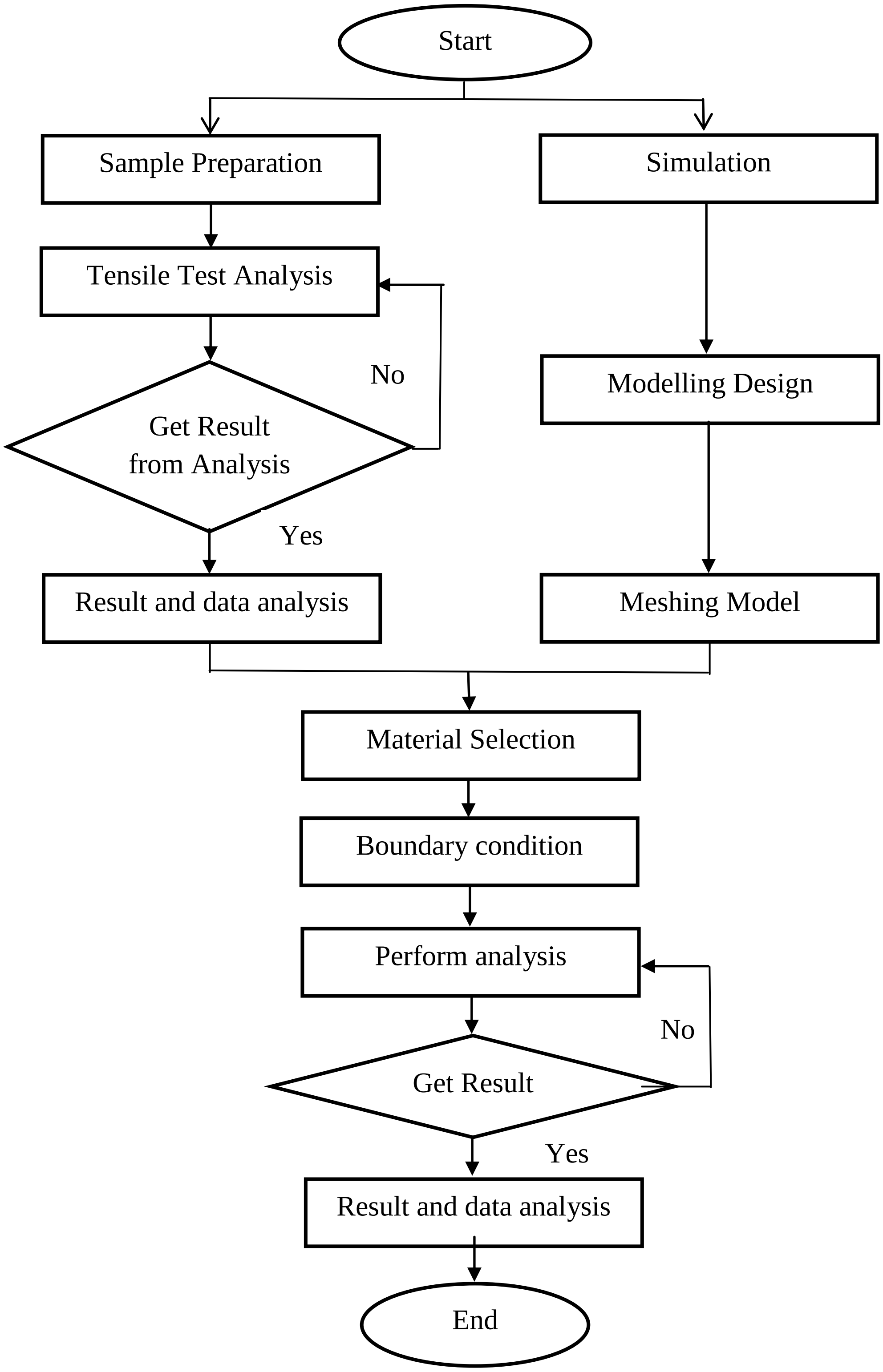

Figure 3.2: Methodology

3.2.3 Specimen Requirement and Standard

For tensile test, there are four types of specimen with different crack size have been prepared. Two specimens have been machined for each crack size. In order to reduce the experimental error, three specimens for each material have been prepared

Tensile testing requirement are specified in various standards for a wide variety of different material and products. The specimen was machined according to ASTM E8specification for plane tensile test specimen. Based on ASTM E8 that shows in Figure 3.3, the length of specimens that need to use in tensile test is 200 mm and thickness in a range from 0.127 mm – 6.35 mm. For the specimen thickness, it was decided that 2.5 mm testing specimen were used. These specifications define requirements for the test apparatus, test specimen, and test procedures. Below included figure 3.3 and table 3.1 that show the standard ASTM E8 geometry of tensile specimen.

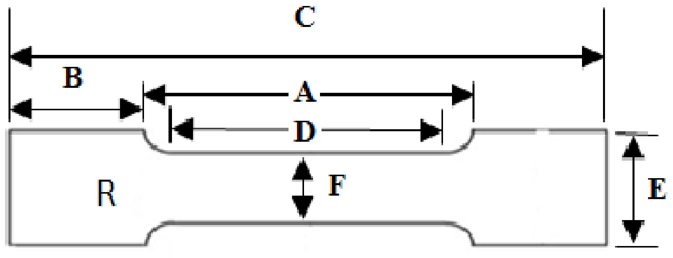

Figure 3.3: Standard ASTM geometry for tensile specimen

Source: Joseph R. Davis, 2004

Table 3.1: Dimension for the standard specimen ASTM E8

Abbreviation	Dimension	Measurement (mm)
A	Length of reduced section	101.6
B	Length of grip section	50.8
C	Overall length	203.2
D	Gauge length	50.8
E	Width of grip section	19.05
F	Width	12.7
R	Fillet radius	6.35
T	Thickness	$0.127 \leq T \leq 6.35$

Source: Joseph R. Davis, 2004

3.3 MATERIAL SELECTION

For this research, the material used in this experiment is taken from actual pipe used in pipeline industries. Chemical composition analysis was conducted in order to compare the chemical properties for current material and to know the material had been used in this experiment. Figure 3.4 below shows the raw material used in this experiment before all the processed start.

Figure 3.4: Raw Material

Initially, the pipe was cut into small two pieces in order to use in chemical composition machine test. Table 3.2shows the chemical properties for the material that has been analyse using chemical composition machine test. From the table, the analyse or experimental result for the material are in a range of Grade B and means that the material used are from material Grade B.

Table 3.2: Chemical properties for raw material

	C	**Mn**	**P**	**S**	**Si**
Experimental	0.261	0.631	0.030	0.030	0.278
API 5L Grade B	0.17- 0.24	0.35- 0.65	0.035	0.040	0.17- 0.37

3.4 SPECIMEN PREPARATION

For tensile test, the specimen has three types of crack size parameter with same material and dimension that had been prepared. The specimen was machined according to ASTM E8 specification for plate tensile test specimen as shown in Figure 3.3above.

3.4.1 Machining Process

The raw materials that in a shape of hollow pipes as shown in Figure 3.4 above was cut into small pieces using band saw machine. Earlier, the hollow pipe has a diameter of 140mm, length of the pipe is 390mm and the thickness of the pipe is 13mm. There are several machine used in this process in order to shaping the specimen into standard geometry for tensile specimen which is ASTM E8. Firstly, the hollow pipe was cut into 250mm length to make it easier to design into tensile test specimen using band saw machine as shown in Figure 3.5below.

Figure 3.5: Band saw machine

After finishing first step of the process which is cutting hollow pipe into small pieces using band saw, the next step is to shaping the specimen into rectangular shape and the thickness of the specimen reduce into 4mm. In order to do that, the milling machine is used to decrease the material to the desired shape which is 4 mm. Figure 3.6 a) shows the milling machine process and the process are use two type of tools which is end mill and face mill.

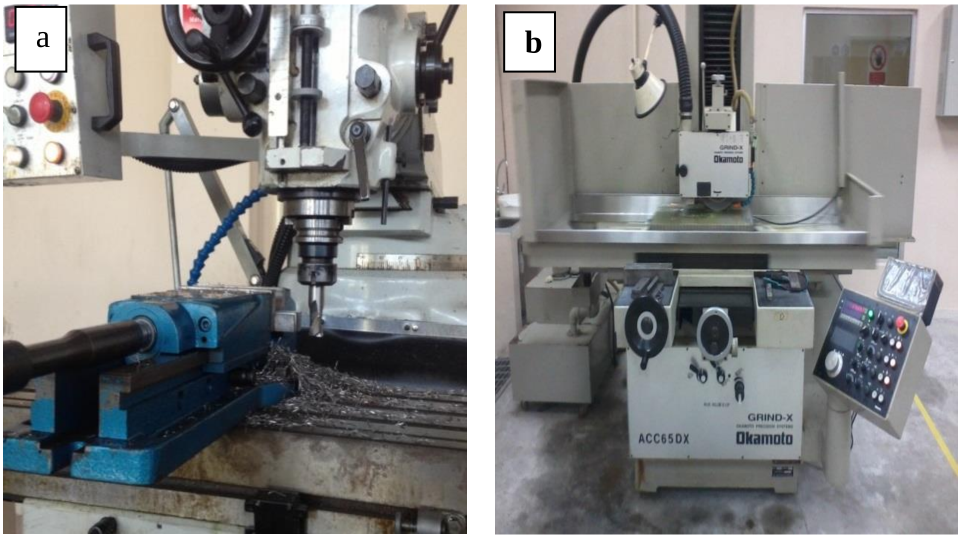

Figure 3.6: Machining process **a)** Milling Machine and **b)** Grinding Machine

After the specimen reduces to 4mm thickness, grinding machine as shown on Figure 3.6 b) is used to decrease the specimen until reduce into desired thickness and in this case is 2.5mm. Grinding machine is use over milling machine for the finishing part is because of the grinding is usually used to finish workpieces that must show high surface quality and high accuracy of shape and dimension. For milling machine, it has to run one by one and will costly time to fabricate while grinding machine can operate at once in this case which are 8 specimens. Figure 3.7above shows the specimen after finish grinding process; the material has a length of 250mm, width of 19mm and 2.5mm thickness. Although the specimen length are exceed 250mm which is the geometry standard of ASTM E8 is 200mm, it was because the shaping process using electro discharge-machining (EDM) Wirecut need to grip the extra length to hold the specimen during machining process. EDM wirecut is use to design the raw material become a tensile test specimen shape.

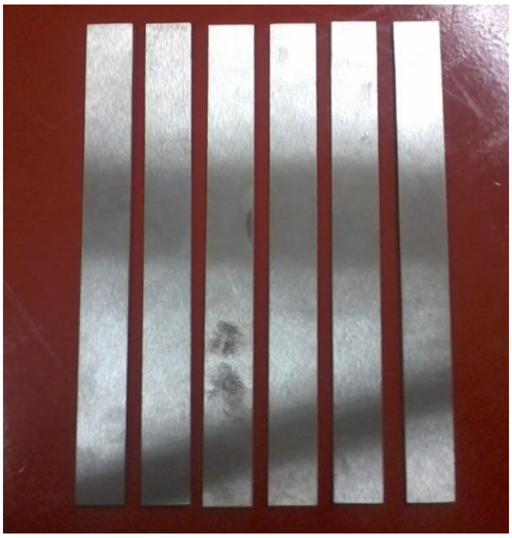

Figure 3.7: Specimen after Grinding Machine

First process is drawing the cutting line in the wire cut machine system. The Sodick machine has Sodick Linear Servo Controller software that can operate all the system in this Sodick machine. The software can read the drawing and then the wire has been used to cut the material by the shape that drawn in the system. Electric discharge machining (EDM) is a manufacturing process whereby a desired shape is obtained using electrical discharges. Material is removed from the workpiece by a series of rapidly recurring current discharges between two electrodes, separated by a dielectric liquid and subject to an electric voltage. The wire used is rotating to cut the material as shown in Figure 3.8below. After that, the specimens are shaping into plate dogbone shape as shown in Figure 3.9below in order to follow standard requirement for tensile test specimen.

Figure 3.8: Rotating cut wire

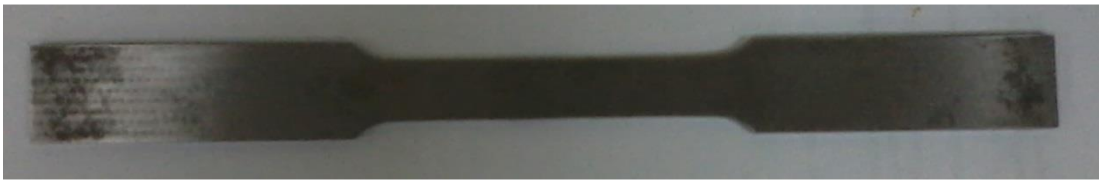

Figure 3.9: Shaping specimen

Meanwhile, for crack manufacture process, EDM die sink machine were use and the tool in making the crack are using copper electrode. EDM die sink consists of an electrode and workpiece submerged in an insulating liquid which are dielectric fluids. The electrode and workpiece are connected to a suitable power supply. The power supply generates an electrical potential between the two parts. As the electrode approaches the workpiece, dielectric breakdown occurs in the fluid, forming a plasma channel and a small spark jumps. The process in making the crack for the specimen is shown in Figure 3.10 below.

Figure 3.10: EDM Die Sink process

For this project, the tool used for making the crack are using electrode cooper with a dimension of 0.5 width and the length vary from 4mm, 6mm and 8mm. As for the depth of the crack, the entire specimen crack is set with 1mm depth during the machining process. Figure 3.11 below shows the crack of the specimen of 4mm, 6mm and 8mm that will be tested.

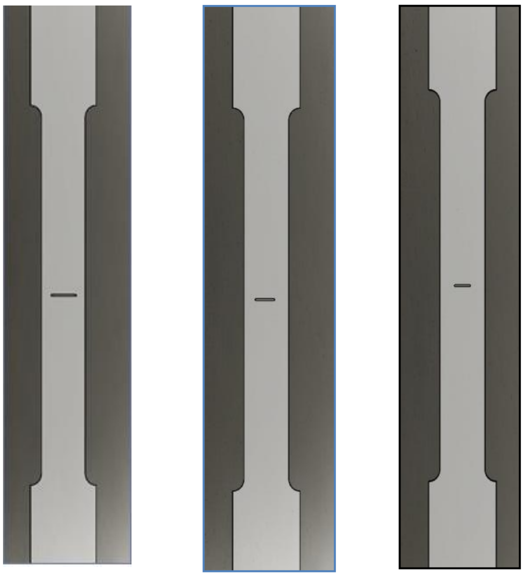

Figure 3.11: Specimen with different crack size

3.5 TENSILE TEST

Tensile test are the simple test to determine the mechanical properties for material. For tensile test, a load is applied along the longitudinal axis of a rectangular cross- section test specimen. The applied load and the resulting elongation of the member are measured and the process is repeated with increased load until the specimen breaks. Load-deformation data obtained from tensile tests do not give a direct indication of the material behaviour, because they depend on the specimen geometry. In this project, the geometry of the specimen is plane specimen with rectangular cross-section area.

This tensile test experiment has been done by using SHIMADZU machine and this machine can be used until maximum force or load about 48kN. The strain rate or speed that use for this tensile experiment is1mm/s. After that, the results of the test for specimen with different crack are measured as in Figure 3.12 and Figure 3.13. Figure 3.12 below also shows the specimen condition after tensile test for uncracked specimen and its elongation. The uncracked specimen only used in finding the mechanical properties for the specimen. It also shows that the different in elongation condition after tensile test was held. As show below, the specimen elongations are proportional with the crack size.

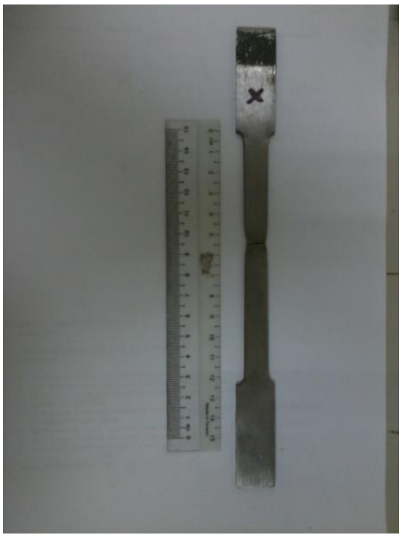

Figure 3.12: Specimen condition for uncracked specimen

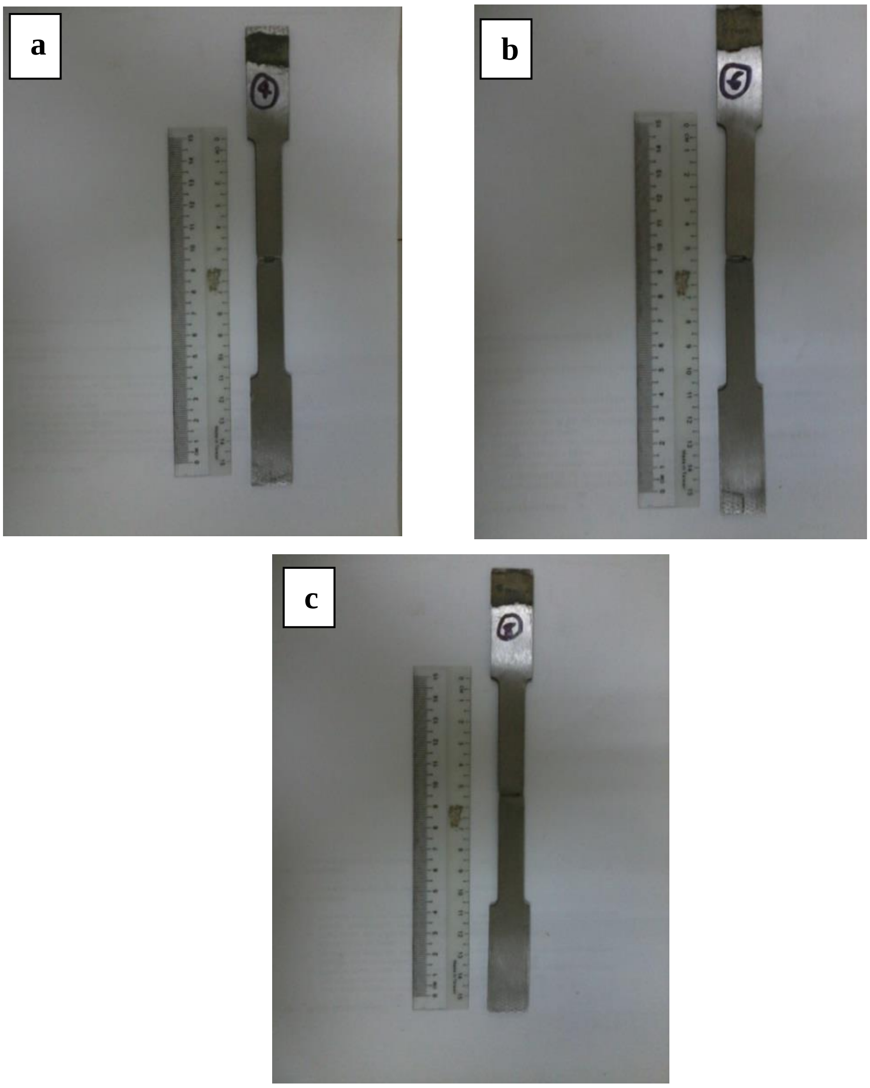

Figure 3.13: Specimen Condition for different crack size **a)** 4mm crack size **b)** 6mm crack size **c)** 8mm crack size

Besides that, the example of the process during tensile test also shows in the figure 3.14 below.

Figure 3.14: Specimen during tensile test

3.6 FINITE ELEMENT ANALYSIS (FEA)

3.6.1 Modelling and Meshing Process

Modelling design was prepared by using Finite Element Analysis software that is MSC PATRAN software. Firstly, one new folder was made for this project. All the data and result that has been done by using Finite Element Analysis was obtained in this folder. Before that, the engineering stress-strain data was converted into true plastic stress- strain data in order to run the finite element analysis in MSC Patran/Marc 2008r1 software. Then, the data was saved in excel with CSV (comma delimited) format. Then, PATRAN 2008r1 software was opened and a new file was created to start the modelling process. After that, the analysis code was change to desired code which is MSC Marc and changes the setting of preferences into mm unit. Then the modelling process has been started. Firstly, the GEOMETRY bar that shown in Figure 3.15 was taken to draw the specimen.

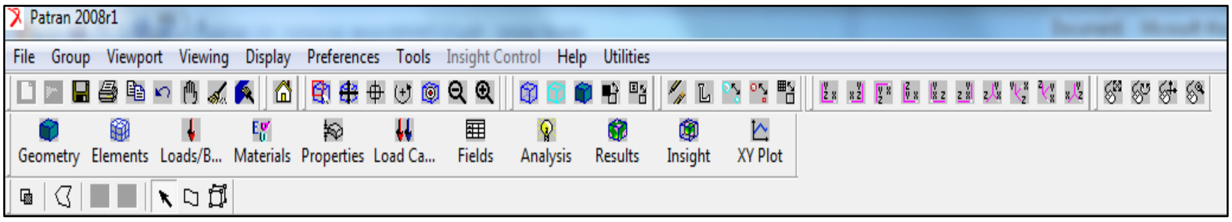

Figure 3.15: Step shown in PATRAN Software

Firstly, GEOMETRY bar as Figure 3.15 stated were chose to draw the specimen and the specimen has been drawn in 2D by using coordinate to obtain the shape of specimen in quarter. Firstly the point had been draw using coordinate like shows in Figure 3.16.a). After that, curve was created and the point combined together using curve that is stated in Figure 3.16.b). Then, surface was taken to create the surface for this shape and for this surface process, don't forget to change the direction of surface to become positive direction that already shown in Figure 3.16.c). Lastly create the solid to get the 3D specimen. The surface was extruded with the thickness needed (2.5 mm) that shown in Figure 3.17.

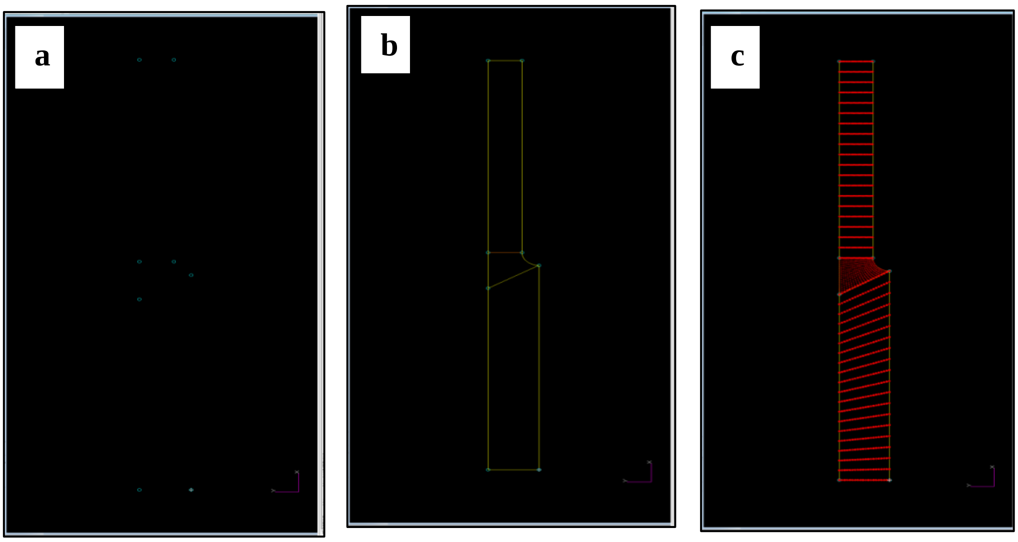

Figure 3.16: Step for Geometry **a)** Create point using coordinate **b)** Combine the point using line and create the surface **c)** Surface direction

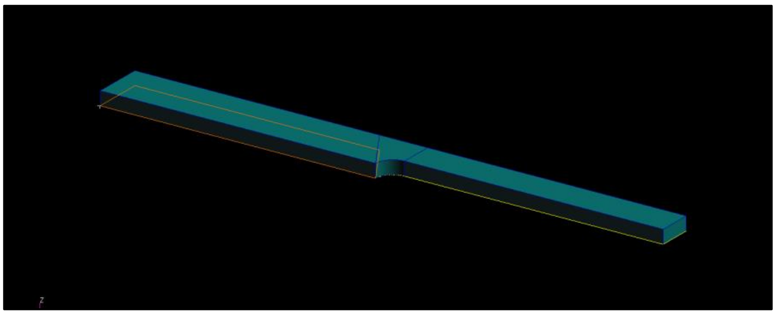

Figure 3.17: Solid part of the specimen

For the next step, the ELEMENT part for the specimen was created by using mesh. Figure 3.18.a) show the first step in ELEMENT part which is the mesh seed process for the specimen. In this project, two type of mesh seed has been used which is uniform and one way bias. One way bias used in order to make the mesh more accurate at the critical part of the specimen. Figure 3.18.b) is showing the meshing process for the specimen used in this project. The mesh in solid type and element shape of hexagon are applied. After that, all the meshing has been equivalence to connect the mesh in the solid part of the specimen. The process is shown in Figure 3.19.

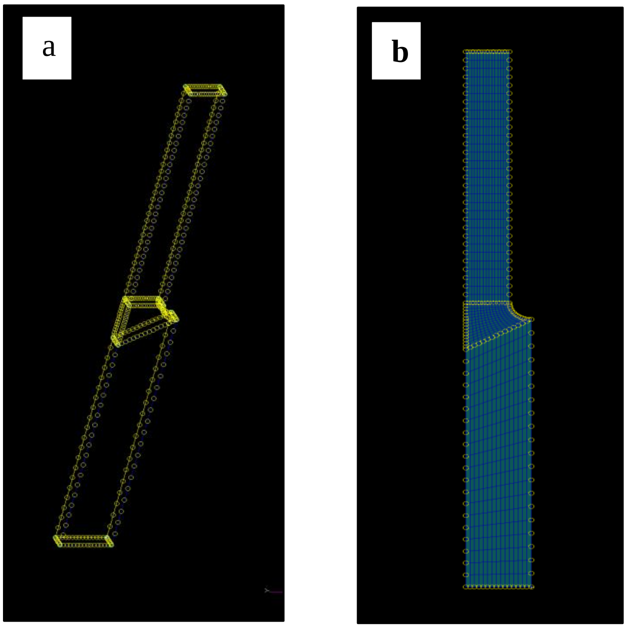

Figure 3.18: Step for element **a)** Mesh Seed Process **b)** the Meshing Process

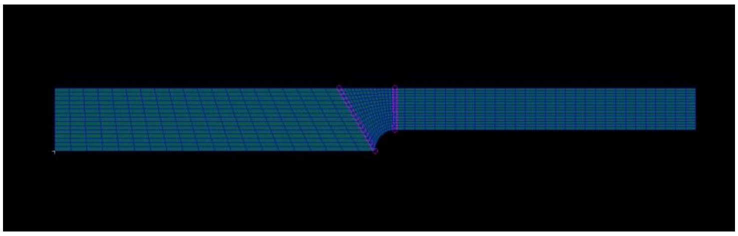

Figure 3.19: Equivalence Process

Next, LOADS/BCs (boundary condition) part be applied to get the boundary condition at the specimen. New name was set and then click input data and put the data for the load. First is the displacement (15mm) boundary condition that is representing as uniaxial tensile test in Y-direction. After that, apply boundary condition to represent as symmetry in X and Y axis. Change the translations and rotations area to give the boundary conditions that shown in Figure 3.20. This symmetry must be apply to make sure the software read there are symmetry shape at others site of the axis. Lastly, the end of the specimen has to fix. Next, the application region for all the loads is applied. Lastly, the solid region is selected as an application region.

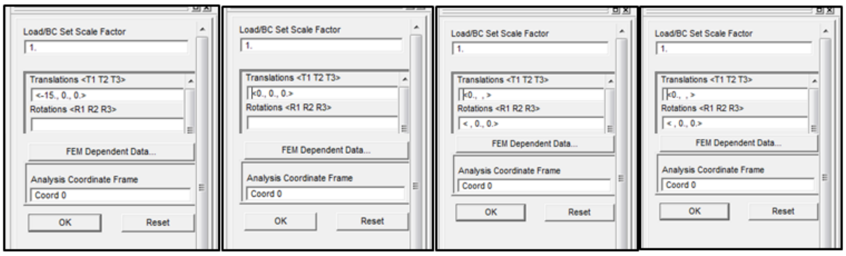

Figure 3.20: Step in creating boundary condition

After that, the FIELD part must be done to get the data before run the simulation. Put all true plastic stress-strain data that save in early process, change the object and method to material property and tabular input. Then, put any field name, click strain and input data to export the CSV file same as shown in Figure 3.21.

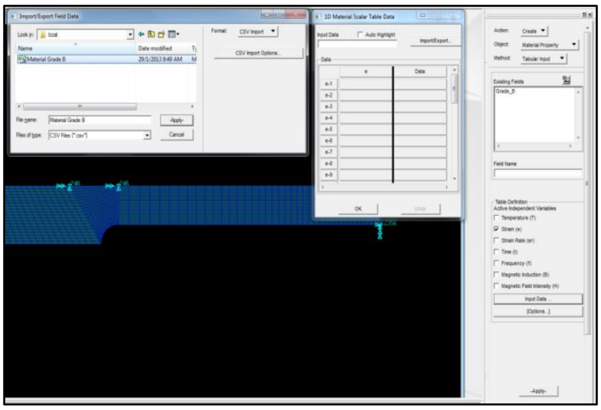

Figure 3.21: Step in Field part input data

The next steps are the MATERIAL part and name the material. Click input properties to put all the data for elastic and plastic region. There are two importance data must be obtained in this part. Firstly is elastic region data that need to fill are young modulus and poison ratio of each material used. The value of young modulus and poison ratio for the material is 207GPa and 0.30. Secondly is plastic region data that need to take from the FIELD part is true plastic stress-strain data for the material need to run. Figure 3.22 shows the material data input that has been filled on the table.

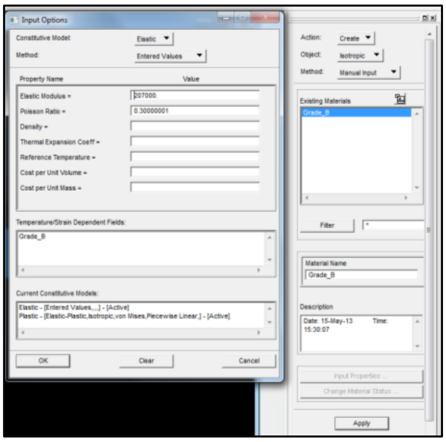

Figure 3.22: Material data input

After that, the PROPERTIES part has been done to ensure the finite element analysis PATRAN 2008r1 software can read the properties given and run the simulation successfully. The properties name has given for this property. After that, click option and take reduce integration for this part same as shown in Figure 3.23. After all the data and application region been selected, PROPERTIES step been applied.

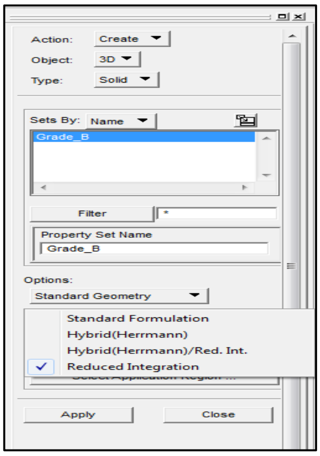

Figure 3.23: The Properties part

Then, click the ANALYSIS part. In this part all the parameter for analysis will be set up before run the analysis. The method must be changed into analysis deck and the job name part were filled same as shown in Figure 3.24. After that, click job parameter and fill all the data needed. Firstly, click solver that also shows on Figure 3.24 and then click non-positive definite.

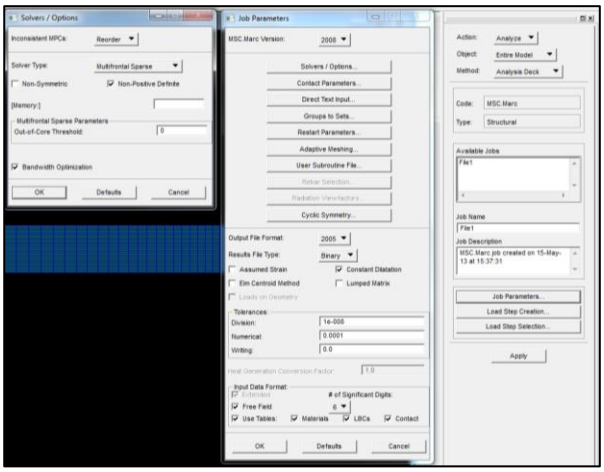

Figure 3.24: Step in job parameter part

For further analysis, select load step creation and then change the load step name. After that, click at solution parameter and tick at follower forces. Then, select load increment parameter and fill the number of step of output to 50 that shown in Figure 3.25. Iteration parameter was selected and change the value of relative residual force to 0.001 that shown in Figure 3.26. Then click OK for the entire event.

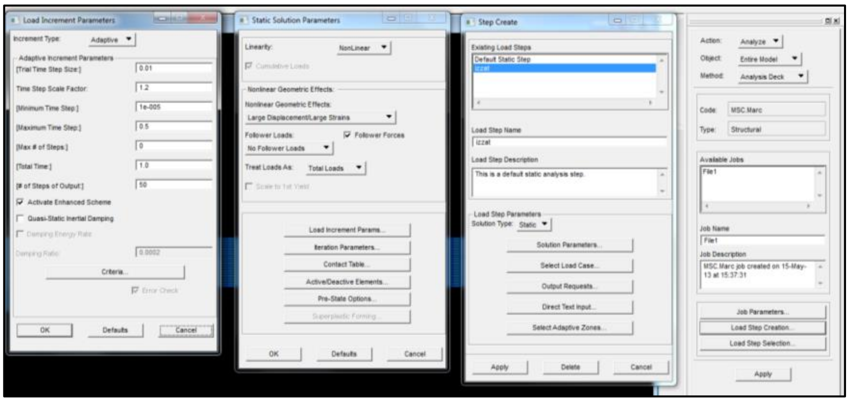

Figure 3.25: Step in load step creation part to set the load increment parameter

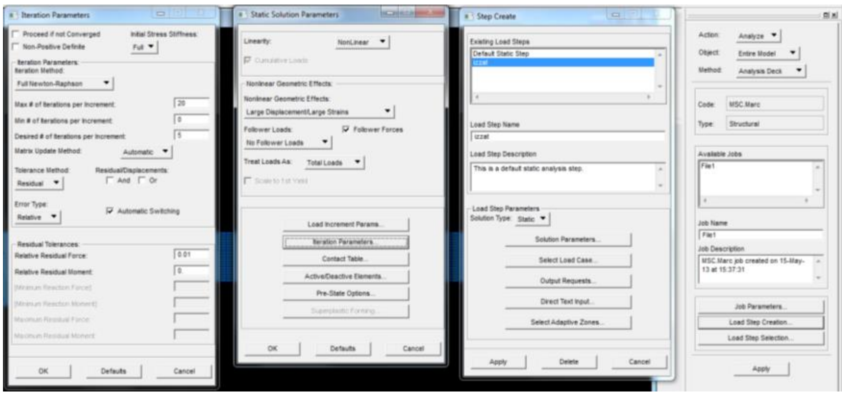

Figure 3.26: Step in load step creation part to set the iteration parameter

Lastly, run the analysis using CMD command to get the result of this project. After get the result, change the action at ANALYSIS part into read result and select the result file to take the result same as shown in Figure 3.27. Then, apply the result.

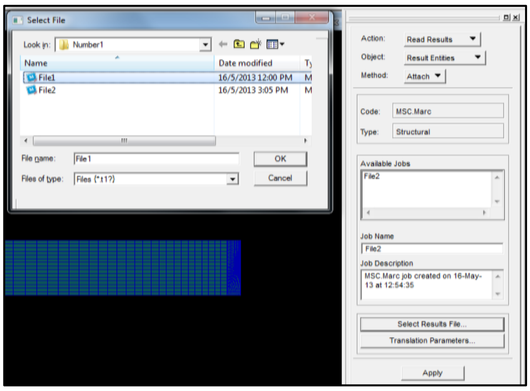

Figure 3.27: Step in read the result

The last process for PATRAN 2008r1 software is selecting the RESULT to read and see the result of simulation of the specimen. After run this specimen, go to result to play the movement animation to represent the displacement force in Y-direction. If the simulation don't have any problem and suitable with experiment result, next step is to take an important data for finite element analysis result. Next, take the suitable data in RESULT part such as the data for load and displacement. Lastly, the data has been put in EXCEL and the graph needed has been plotted.

CHAPTER 4

RESULT& DISCUSSION

4.1 INTRODUCTION

Result from tensile test experiment and finite analysis elements (FEA) were compared to determine the different of fracture strain equation for cracked thin plate between experiment and FEA. All the data from experiment and analysis result were discussed in this chapter.

Firstly, tensile test on the standard specimen are performed in order to find the mechanical properties for the material. After that, specimen with crack will also performed the tensile test and to be compared. Engineering stress – strain graph will be convert into true stress – strain in order to use when perform simulation process. Then, the simulation result compare with experimental and also coalescence load versus crack size diagram were compare between experimental and simulation.

4.2 EXPERIMENT RESULT

By doing experiment, the results were discussed to be compared to the finite element analysis (FEA) and it also shows the different of strength between specimens with vary crack size. The tensile test must to be test on the specimen in order to find material properties for the specimen. Figure 4.1 below shows the deformation of the specimen during tensile test was conducted.

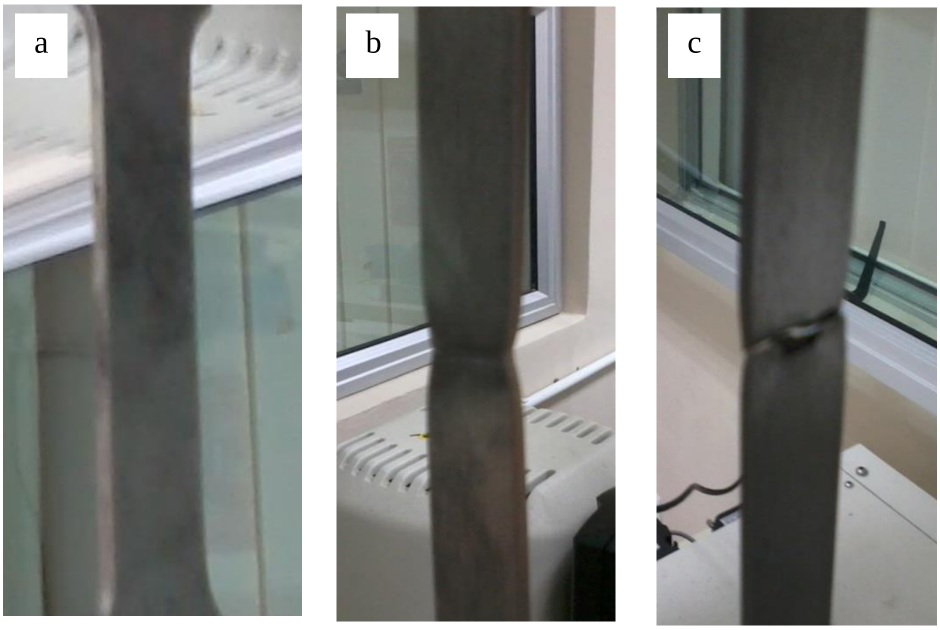

Figure 4.1: Step of deformation for the uncrack specimen **a)** Elastic Deformation **b)** Plastic Deformation (transition between elastic and plastic **c)** Fracture process

From the figure shown, the deformation of the uncrack specimen which the elastic deformation starting to occur Figure 4.1 a) and as the material is deformed beyond this point, the stress is no longer proportional to strain and after that plastic deformation occurs Figure 4.1 b). At b), the yield strength occur, that the point where the transition between elastic – plastic on the deformation and after yielding, the stress necessary to continue plastic deformation increases to a maximum strength before decreases to the eventual fracture Figure 4.1 c).

4.2.1 Mechanical Properties for Experimental Specimen

Initially, the tensile tests are performing to find the mechanical properties for the material that used for the experiment. The basic data used to know the mechanical properties are obtained from this test. The data obtained from this test including ultimate tensile test, yield strength, strength at break and maximum load.

Figure 4.2 shows the engineering stress-strain diagram for Steel API Grade B. Based on the graph, the maximum stress or ultimate tensile strength is 466.12MPa whereas maximum strain is 0.166%, and maximum force is 14.799kN. Table 4.1 also indicated the mechanical properties obtained from the tensile test. In this table the value of stress and strain at the onset of fracture was also included. The stress and strain at the corresponding point is given by 396.91MPa and 0.217% respectively. Then the break area force is 12.602kN.

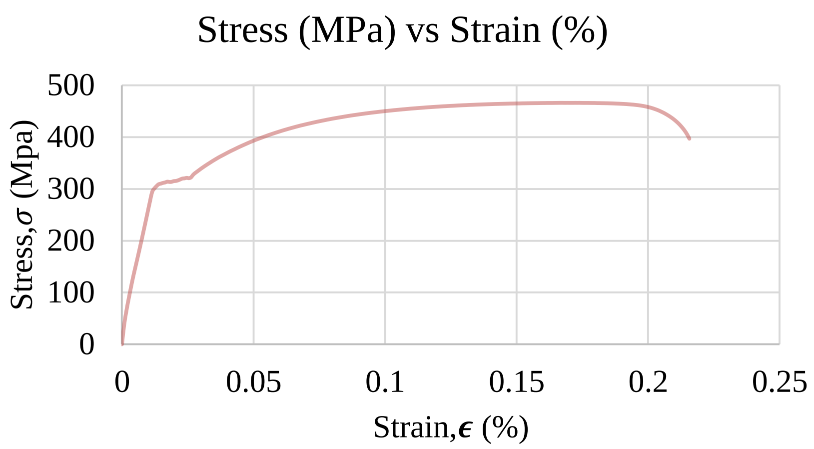

Figure 4.2: Stress- strain graph

Data from this material will be a reference and will be compared the result with the crack specimen. Crack specimen has 3 type of crack size and the different of strength will discuss in order to show the properties of the specimen. Besides that, the data also will be comparing to the finite element analysis (FEA) via PATRAN software.

Table 4.1: Mechanical properties obtained from tensile test

	Maximum	**Break**
Stress (MPa)	466.12	396.91
Strain (%)	0.166	0.217
Load (kN)	14.799	12.602

4.2.2 Tensile Test Result (Engineering Stress - Strain Graph)

The first result in this experiment is engineering stress-strain graph. This result was acquired from the tensile test experiment, from that the force versus displacement graph and stress versus strain graph were obtain. However, the stress versus strain graph is the only graph that will be used and the result will be compared with analysis by using finite element analysis (FEA) PATRAN.

Figure 4.3 shows the engineering stress-strain diagram for specimen with a 4mm crack size. Based on the graph, the maximum stress that represent as ultimate tensile strength is 431.54MPa whereas maximum strain is 0.078%, and maximum force is 13.697kN. Table 4.2 indicated the mechanical properties obtained from the tensile test. In this table the value of stress and strain at the onset of fracture was also included. The stress and strain at the corresponding point is given by 413.70MPa and 0.081% respectively. Then the break area force is 13.134kN.

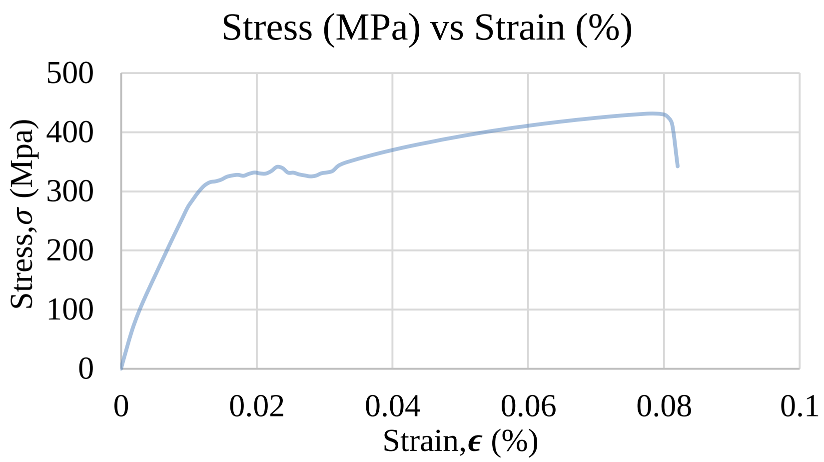

Figure 4.3: Stress- strain graph for 4mm crack specimen

Table 4.2: Mechanical properties obtained from tensile test for 4mm crack size specimen

	Maximum	**Break**
Stress (MPa)	431.54	413.70
Strain (%)	0.078	0.081
Load (kN)	13.697	13.134

Figure 4.4 shows the engineering stress-strain diagram for specimen with a 6mm crack size. Based on the graph, the maximum stress that represent as ultimate tensile strength is 410.43MPawhereas maximum strain is 0.059%, and maximum force is 13.031kN. Table 4.3 indicated the mechanical properties obtained from the tensile test. In this table the value of stress and strain at the onset of fracture was also included. The stress and strain at the corresponding point is given by 397.18MPa and 0.061% respectively. Then the break area force is 12.610kN.

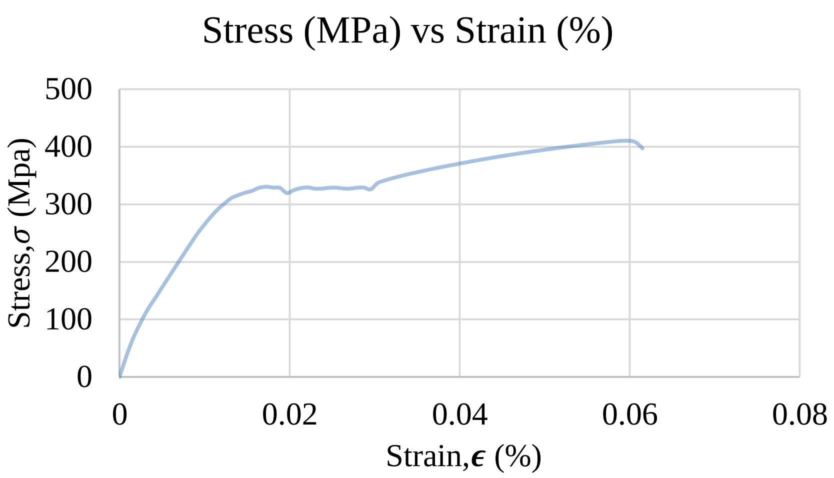

Figure 4.4: Stress- strain graph for 6mm crack specimen

Table 4.3: Mechanical properties obtained from tensile test for 6mm crack size specimen

	Maximum	**Break**
Stress (MPa)	410.43	397.18
Strain (%)	0.059	0.061
Load (kN)	13.031	12.610

Figure 4.5 shows the engineering stress-strain diagram for specimen with a 6mm crack size. Based on the graph, the maximum stress that represent as ultimate tensile strength is 392.15MPawhereas maximum strain is 0.049%, and maximum force is 12.450kN. Table 4.4 indicated the mechanical properties obtained from the tensile test. In this table the value of stress and strain at the onset of fracture was also included. The stress and strain at the corresponding point is given by 359.82MPa and 0.051% respectively. Then the break area force is 11.424kN.

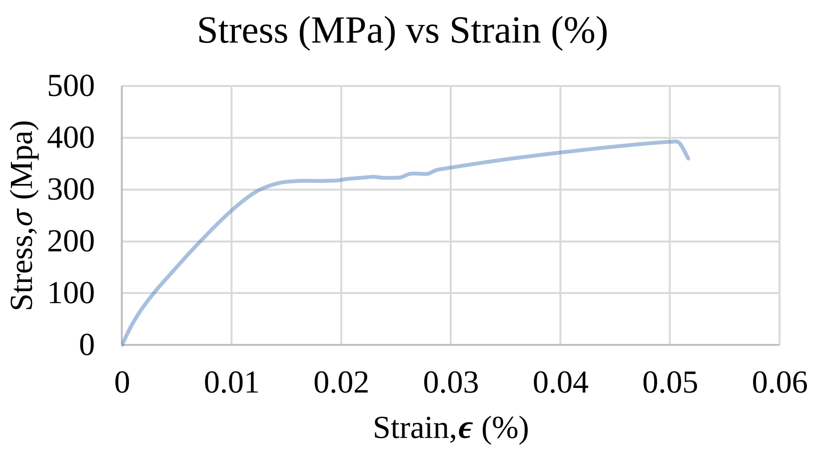

Figure 4.5: Stress- strain graph for 8mm crack specimen

Table 4.4: Mechanical properties obtained from tensile test for 8mm crack size specimen

	Maximum	**Break**
Stress (MPa)	392.15	359.82
Strain (%)	0.049	0.051
Load (kN)	12.450	11.424

Figure 4.3 until 4.5 shows the stress – strain curve that vary by different crack size which is 4mm, 6mm and 8mm. Figure 4.6 below shows the comparison between the three data and can be concluded that the material with the more crack size length will have a low ultimate tensile strength and will tend to rupture more faster than lower size of crack specimen. The maximum stress or ultimate tensile strength for 4mm crack specimen is 431.54MPa compare to other specimen which is 410.43MPa and 392.15Mpa. As shown in Figure 4.6, specimen with crack 4mm also has a higher fracture strain percentage compare to 6mm and 8mm crack specimen which is 0.081%, 0.061% and 0.051%. Besides that, the yield strength for the 3 specimen also shows that smaller crack size specimen has a higher yield strength value.

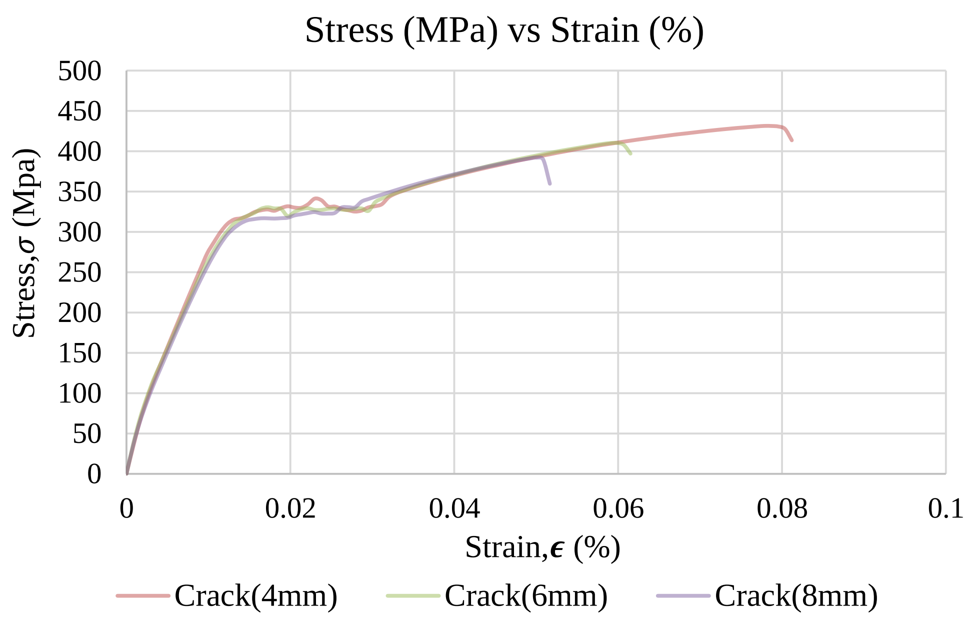

Figure 4.6: Comparison between crack sizes stress – strain curve

4.2.3 True Stress - Strain Graph

In finite element analysis (FEA), true plastic stress-strain data must be employed as an input data for the material. Therefore, it is very important to convert engineering stress- strain data into true stress-strain data. There are the steps to convert the data which is by using the equations and the equation are given by:

$$\sigma\tau = \sigma e(\ \epsilon e + 1\ ) \tag{4.1}$$

$$\epsilon\tau = ln\ (\ 1 + \ \epsilon e\ ) \tag{4.2}$$

The equation 4.1 and 4.2 is only applicable up to necking point of the material and Figure 4.7 shows the true stress strain of the smooth specimen on elastic deformation. From this, it can be concluded that the stress is increasing after the maximum point up to fracture. This is because due to the reduction of cross-section area of the material after necking was occurred.

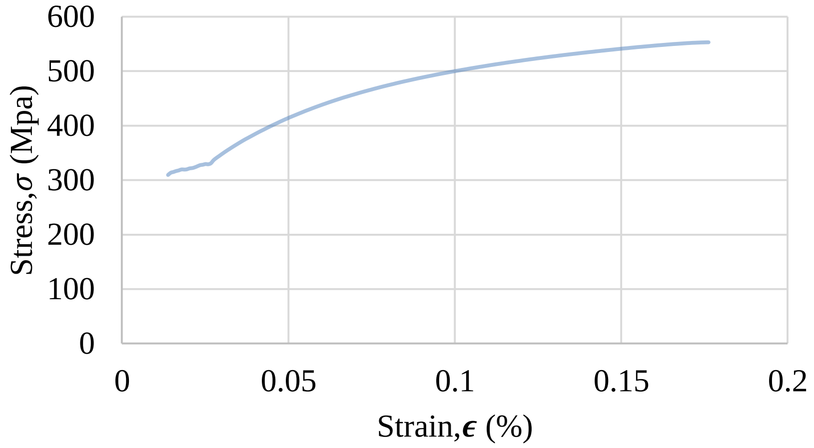

Figure 4.7: True Plastic stress – strain curve

4.3 FINITE ELEMENT ANALYSIS (FEA) RESULT

The results of finite element analysis are used to simplify working process and reduce working time to do an experiment. Furthermore, the result from finite element analysis usually can be used as an experiment result. For this research, the Finite Element Analysis result will compare to the experiment and the parameter will be add to see the change in the coalescence load versus crack size diagram between experiment and simulation. The purpose of the simulation is to indicate the area, where the state of stress on the tensioned specimen is the closest to uniaxial tension. In finite element analysis (FEA), the specimens are drawing in four quarter to simplify and minimize the specimen before run by using PATRAN software.

The colour contour shown in Figure 4.8 are represented the stress capabilities in deformation of the specimen. This happen after boundary condition applied on the specimen which is the displacement on the Y-axis direction is applied. After the elongation, the middle of plane dog bone specimen shows the highest stress that represented in red colour contour and this means that the fracture will start on the middle of the specimen based on finite element analysis.

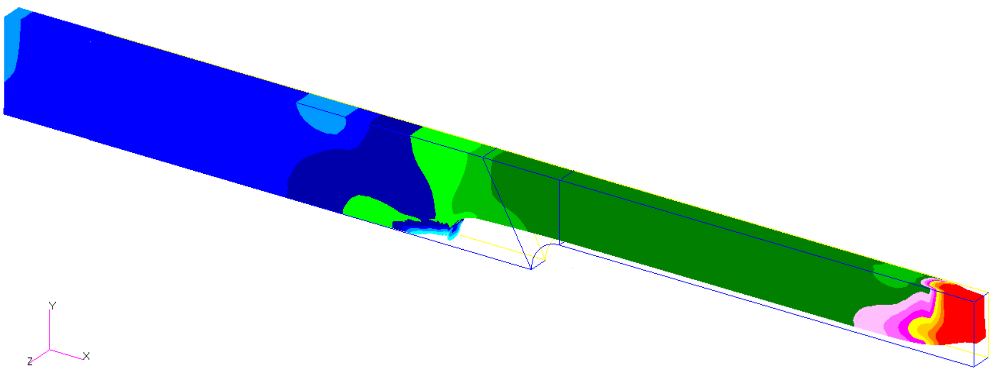

Figure 4.8: The 3D elongation of the specimen in finite element analysis

The highest stress appear at middle of the specimen because the length of material is symmetry from one side with others side. So the elongation of specimen start from middle area and stress will focus at this area until these area break after apply displacement at Y-direction. After the break occur, the thickness and width of the specimen change into smaller on the critical part. The shape changes as shown in Figure 4.9same as the experimental tensile test specimen.

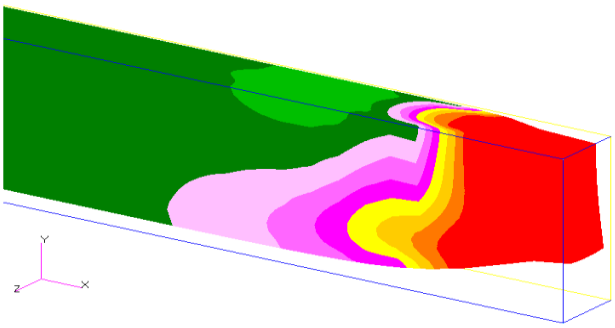

Figure 4.9: The deformation shape of the specimen in finite element analysis

Figure 4.10below shows the engineering stress-strain diagram for specimen from material API Grade B that obtained from Finite Element Analysis compared to the experimental.

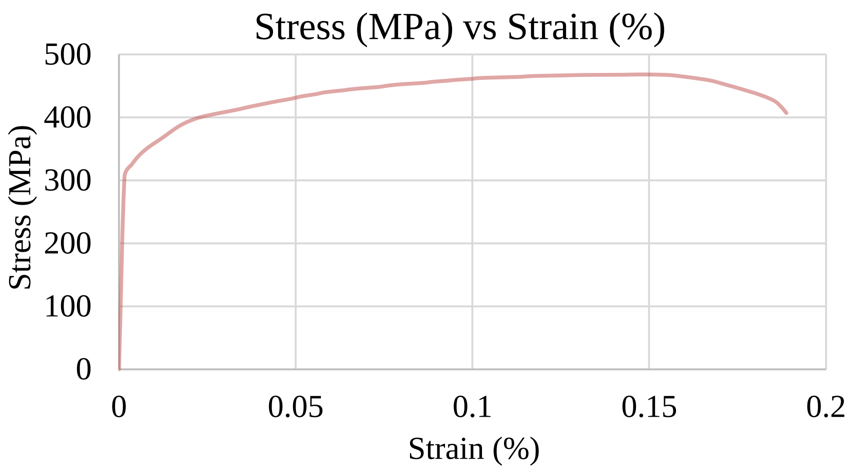

Figure 4.10: Engineering stress – strain data obtained from Finite Element Analysis for uncrack specimen

Figure 4.11 until 4.13 shows the colour contour of the specimen with a crack size 4mm, 6mm and 8mm respectively. The colour contour represented the stress level on the specimen. From the figures, the highest stress starts to appear on the crack before widespread on the entire crack before start to break. Besides that, the ultimate tensile stress fracture strain and maximum load on the specimen with different crack size are also differ depend on the size of the crack. The specimen with higher size of crack appears to have higher potential to be rupture more easily than lower crack size.

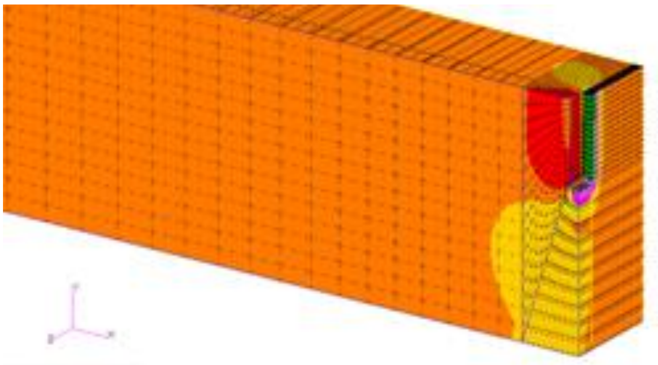

Figure 4.11: Finite element analysis model for specimen with 4mm crack size

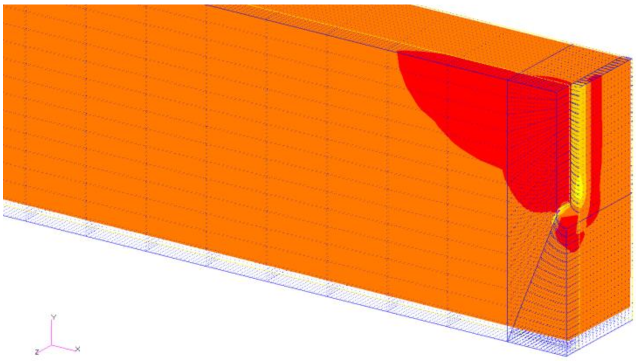

Figure 4.12: Finite element analysis model for specimen with 6mm crack size

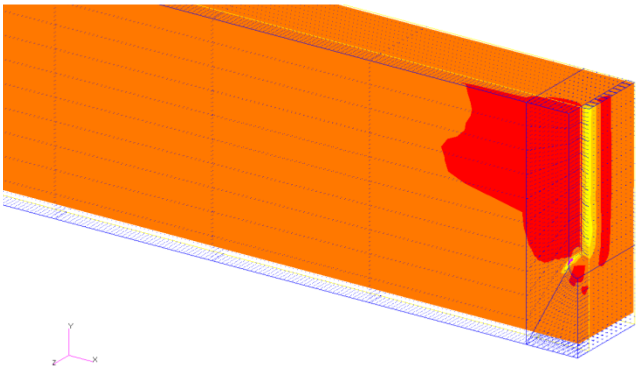

Figure 4.13: Finite element analysis model for specimen with 8mm crack size

Figure 4.14 shows the comparison of engineering stress – strain curve that obtained from finite element analysis for different crack size which is 4mm, 6mm and 8mm. Figure above shows the comparison between the three data and can be concluded that the material with the more crack size length will have a low ultimate tensile strength and will tend to rupture more faster than lower size of crack specimen. The maximum stress or ultimate tensile strength for uncrack specimen is 468.18MPa compare to other specimen which is 401.42MPa, 380.04MPa and 333.76Mpa. As shown in the figure, specimen with crack lower crack also has a higher fracture strain percentage compare to other specimen. Besides that, the yield strength for the 3 specimen also shows that smaller crack size specimen has a higher yield strength value.

Based on the graph, the maximum stress that represent as ultimate tensile strength for specimen with 4mm crack size is 401.42MPa whereas maximum strain is 0.109 %, and maximum load is 11.942kN. As for the engineering stress-strain diagram for specimen with a 6mm crack size obtained from Finite Element Analysis, the maximum stress that represent as ultimate tensile strength is 380.04MPa whereas maximum strain is 0.0413%, and maximum load is 10.926kN. Meanwhile, the graph below also shows the engineering stress-strain diagram for specimen with an 8mm crack size obtained from Finite Element Analysis which is the maximum stress that represent as ultimate tensile strength is 333.76MPa whereas maximum strain is 0.0202%, and maximum force is 9.261kN.

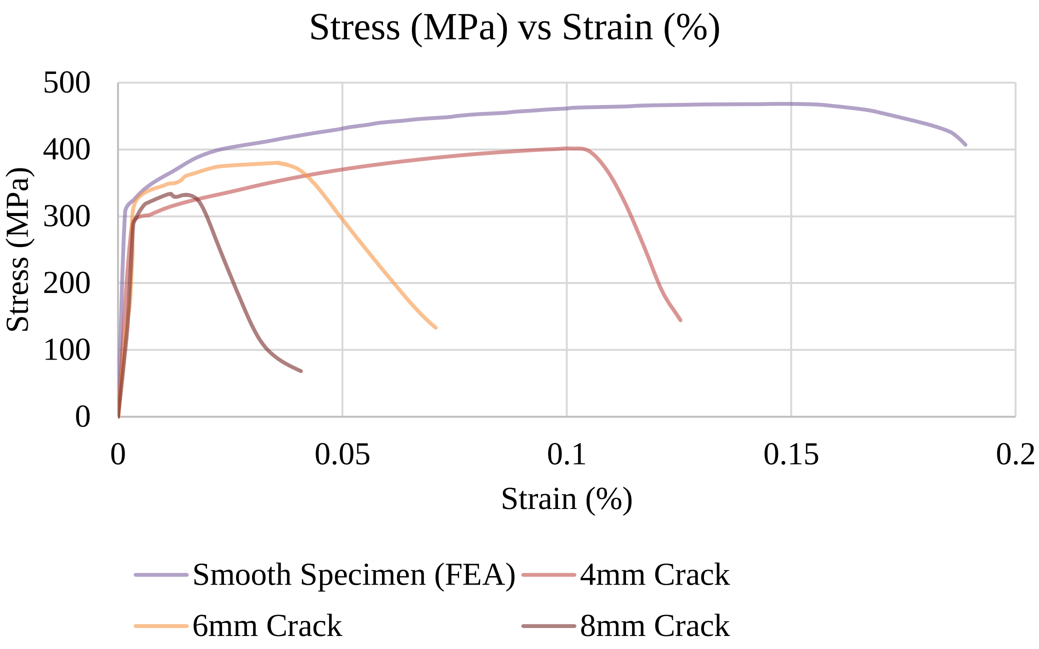

Figure 4.14: The comparison of engineering stress-strain data obtained from finite element analysis for different crack size

4.4 COMPARISON BETWEEN EXPERIMENTAL AND FINITE ELEMENT ANALYSIS RESULT

After get the result from experimental and finite element analysis, both result is compared to find the percentage of error between the two results. From the results, 4 graph were obtain based on the parameter used for both analysis. Based on the graph, the maximum stress that represent as ultimate tensile strength for experiment is 466.12MPa whereas maximum strain is 0.166 %, and maximum load is 14.79kN and for Finite element analysis the maximum stress that represent as ultimate tensile strength is 468.18MPa whereas maximum strain is 0.1498 %, and maximum load is 14.86kN.From Figure 4.15, both graph for stress are increased steadily but in a different strain percentage in elastic deformation. Meanwhile, the ultimate tensile strength for the both test are slightly different which is 466.12MPa and 468.18MPa respectively. Both result obtain are slightly different due to the ideal condition in the simulation such as the material used are flawless to be compared to the material that has been machining even though the yield strength and the poisson ratio were fixed for both test. Therefore, the slight variation in both result can be tolerate thus validate the simulation that is done in MSC PATRAN Simulation software.

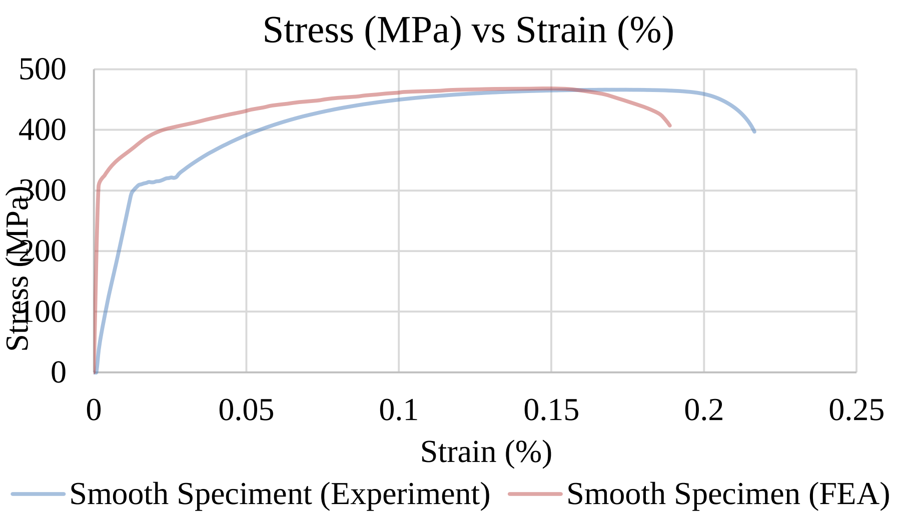

Figure 4.15: The comparison of engineering stress-strain data obtained from experimental and finite element analysis for smooth specimen (uncrack)

Figure 4.16 below shows the comparison of engineering stress-strain data obtained from experimental and finite element analysis for 4mm crack size. As shown in the graph, the elastic deformation for finite element analysis the strain occur early compared to the experiment result and the fracture point for finite element analysis for 4mm crack size are in higher strain value compared to the experimental. However, the maximum stress for experiment result is higher than finite element analysis result which is 433.54MPa and 401.42MPa respectively.

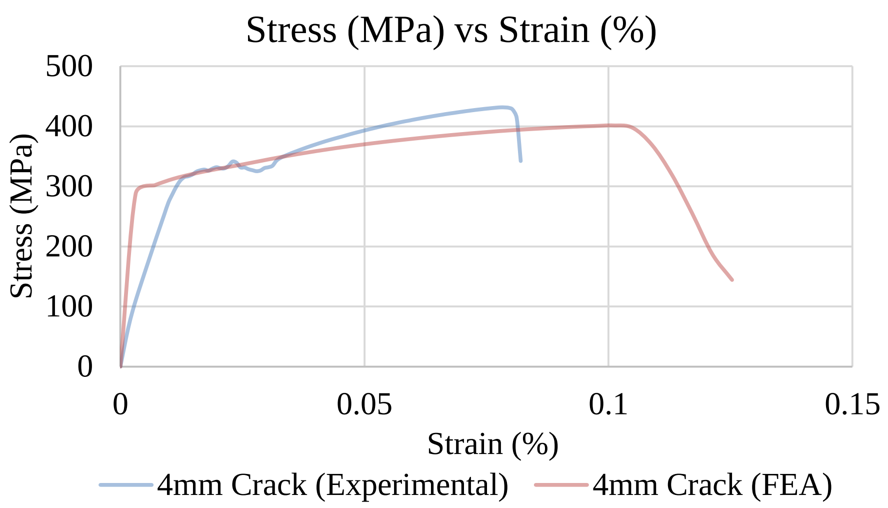

Figure 4.16: The comparison of engineering stress-strain data obtained from experimental and finite element analysis for 4mm crack size

Meanwhile, Figure 4.17 shows the comparison of engineering stress-strain data obtained from experimental and finite element analysis for 6mm crack size. Same as 4mm crack size, the graph shows that the elastic deformation for finite element analysis of the strain occurs early compared to the experiment result but the fracture point for finite element analysis for 6mm crack size are fracture early compared to the experimental. Next, the maximum stress for experiment result is higher than finite element analysis result which is 410.43MPa and 380.04MPa respectively.

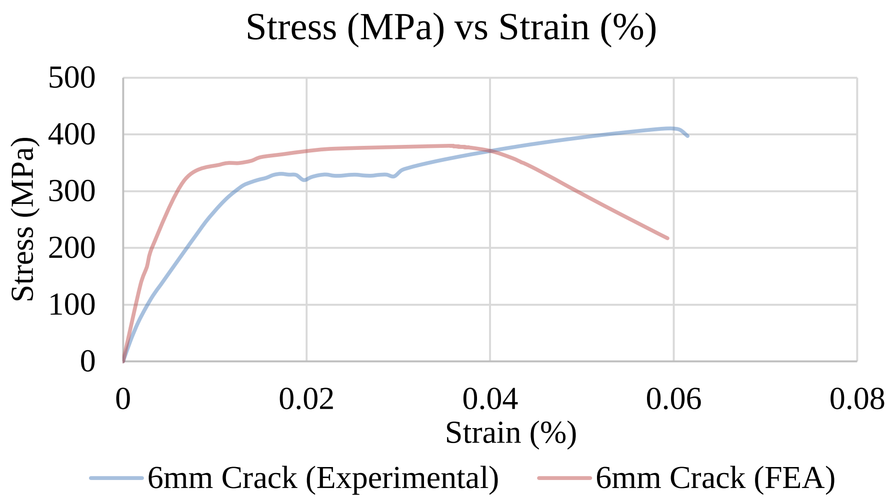

Figure 4.17: The comparison of engineering stress-strain data obtained from experimental and finite element analysis for 6mm crack size

Figure 4.18 shows the comparison of engineering stress-strain data obtained from experimental and finite element analysis for 8mm crack size. There are huge percentage error on the maximum strain value which is 0.049MPa and 0.020MPa for experimental and finite element analysis. The fracture point for both test are also has a huge different and the maximum stress for finite element analysis are higher than experimental result.

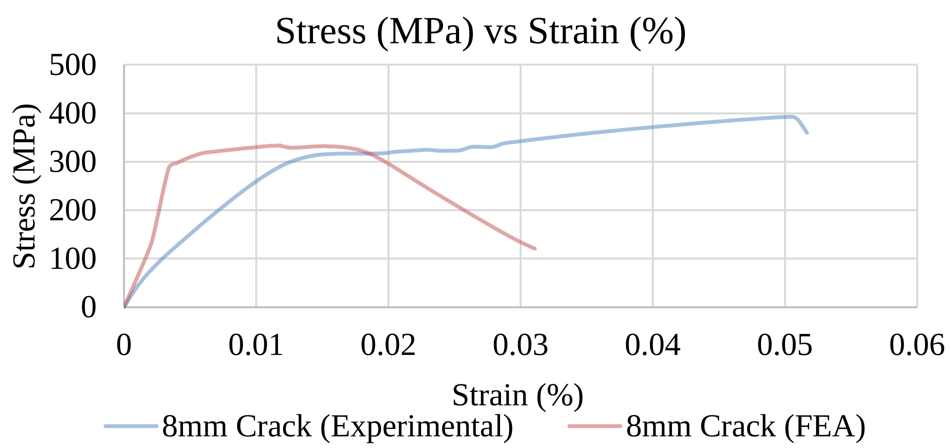

Figure 4.18: The comparison of engineering stress-strain data obtained from experimental and finite element analysis for 8mm crack size

Table 4.5 below show the overall comparison data of maximum stress, maximum strain, and maximum load from experimental and finite element analysis. Besides that, the percentage error also obtains for each of the parameter and different data for the experimental and finite element analysis. On behalf of percentage error for maximum stress, specimen with 8mm crack size has the highest percentage error compared to the two methods which is 17.48%. As for uncrack specimen, the value of percentage error is –0.44 and that means that finite element analysis result has higher maximum stress compared to the experimental result. In the meantime, the maximum strain percentage error is little bit higher for 8mm crack size which is 145%. The big different of maximum strain between the two experiments has make the percentage error became higher and this was due to the ideal condition of the simulation process. For the percentage error between experimental and finite element analysis for maximum load, the error higher proportional as the crack size increase.

Table 4.5: Comparison of Experimental and Finite Element Analysis

	Crack Size	**Experimental**	**Finite Element Analysis**	**Percentage Error (%)**
Max Stress (MPa)	Uncrack (0mm)	466.12	468.18	-0.44
	4mm	433.54	401.42	8.00
	6mm	410.43	380.04	7.99
	8mm	392.15	333.76	17.49
Max Strain (%)	Uncrack (0mm)	0.166	0.149	11.41
	4mm	0.078	0.109	-28.44
	6mm	0.059	0.041	43.90
	8mm	0.049	0.020	145.00
Max Load (MPa)	Uncrack (0mm)	14.799	14.860	-0.041
	4mm	13.697	11.942	14.69
	6mm	13.031	10.926	19.27
	8mm	12.450	9.261	34.43

4.5 SIMPLE DAMAGE MODEL EQUATION

Simple damage model equation can be obtain from linear equation of the crack size versus load diagram that can get from the data of maximum load and crack size of the specimen and the data are obtained from simulation. As state on objective, this project will compare the result and to find the simple damage model equation to predict maximum load of the other crack size without using experiment and simulation.

4.5.1 Crack Size versus Load Diagram Obtain from Finite Element Analysis

For coalescence crack size versus load diagram obtain from finite element analysis, the data were add from 1mm until 8mm crack size specimen compared to experimental data that only take 3 types of crack size. It was because the results of finite element analysis are used to simplify working process and reduce working time to do an experiment. Besides that, the data taken more because the simple damage model equation will be more accurate compared to the data that taken less.

Table 4.6: Properties obtain from Finite Element Analysis for specimen with different crack size

	Max Stress (MPa)	**Max Strain (%)**	**Max Load (kN)**
0mm (uncrack)	468.18	0.149	14.799
4mm	401.42	0.109	11.942
6mm	380.04	0.041	10.926
8mm	333.76	0.020	9.261

Table 4.6 shows the summarization of the maximum stress, strain and load for the entire specimen with different crack size. The data shows that the bigger crack size, the maximum stress, maximum strain and maximum load became smaller compare to specimen with no crack and the small crack size. The data show that maximum stress, strain and load for 8mm crack size specimen were 333.76MPa, 0.0202% and 9.261kN compared to uncracked specimen which is 468.18MPa, 0.149% and 14.799kN respectively. Besides that, the table also shows that the maximum stress, strain and load are keep decreasing as the crack size bigger.

Figure 4.19below shows the crack size versus load curve obtain from finite element analysis result for 4 types of specimen with different size of crack. Besides that, the uncracked specimen also include into the data to generate the graph. From the graph, the simple damage model equation are generate from the finite element analysis result. The equation get from the graph are given by:

$$y = -\ 0.6786x + 14.785 \quad (4.4)$$

From this equation, we can predict the maximum load from various size of crack by using crack as the x parameter and y as the value of maximum load from finite element analysis result.

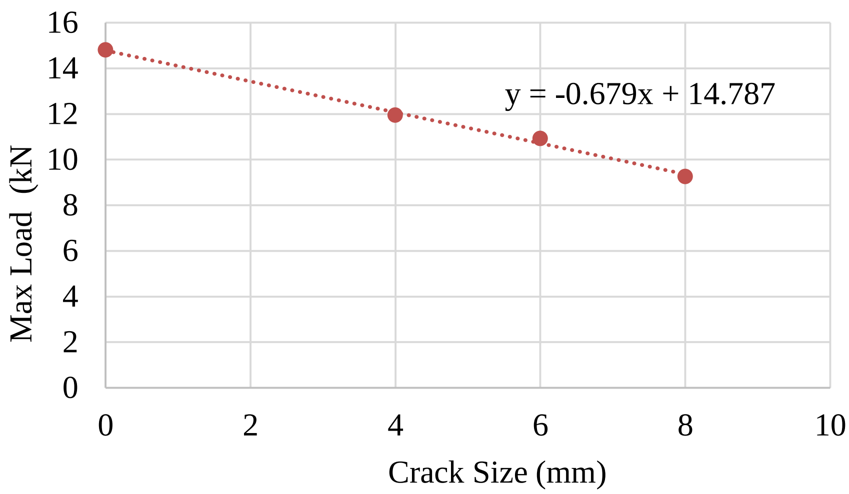

Figure 4.19: Crack Size versus Load from Finite Element Analysis Result

The graph on Figure 4.20 below show the comparison of crack size versus load between finite element analysis and experiment result. The graph shows that the result from experiment has higher maximum load compare to the simulation and the linear line of crack size versus load diagram for simulation are tend to decrease rapidly as the crack size bigger compared to the experimental linear line. This was due to the ideal condition of the result from simulation, so that the results are flawless and not related to the other factor for example machining reason, material and others. From the result, it can be said that the simulation can be used to investigate the crack size factor related to the failure material but the experimental result will get more precise result.

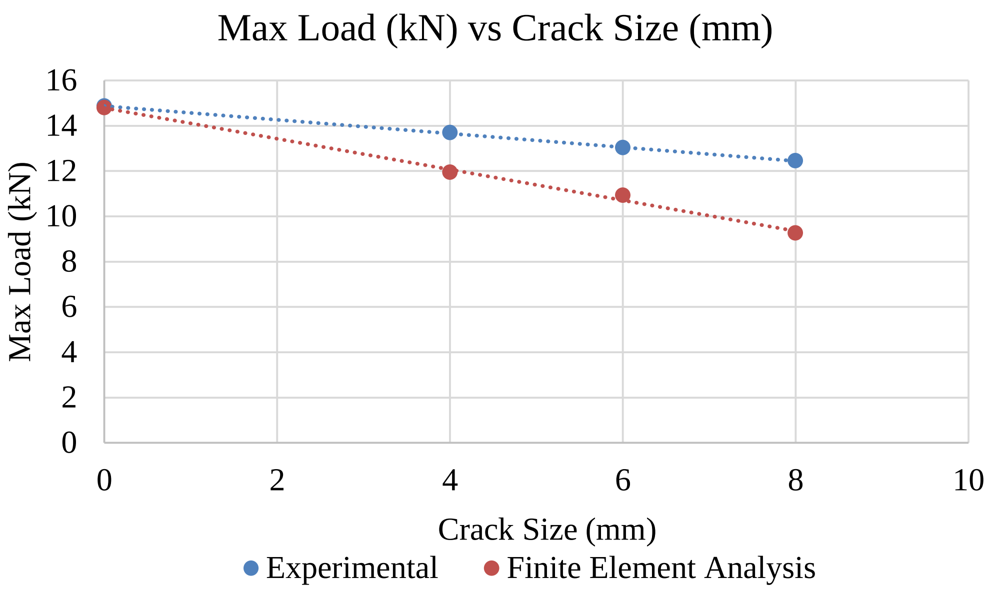

Figure 4.20: Comparison of Crack Size versus Load between Finite Element Analysis and Experiment Result

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS

5.1 CONCLUSION

The main objective of this project is to determine the stress strain curve for cracked thin plate specimen. The experiment perform shows that the specimen with bigger crack size tend to break compare the specimen with smaller crack size. As shown in Chapter 4, the specimen with 4mm crack size has the maximum value of stress, strain and load compare to 6mm and 8 mm crack size. This experiment result also have been proved by the simulation perform using MSC PATRAN software but in a various type of crack size from 1mm until 8mm. However, the value for both test are not exactly the same that because of several criteria that can be stated. For example, the simulation process is very ideal that no defect will occur to the specimen compare to the experimental specimen.

To reach the other objective of the study which is to develop a load versus crack size diagram using a various crack size and generate simple damage model equation for cracked thin plate tensile test, the simulations are performing in more various crack size. The more parameter (Crack Size) used, the equation will be more valid and the maximum load for other value of crack size can be predict. With this equation, the working time on the experimental or the simulation process can be reduced meanwhile the maximum load for vary of crack size can be predicted.

5.2 RECOMMENDATIONS

Recommendation is very important process in every project because it will affect the quality of the project for future study. Based on the findings of the present investigation, there are few recommendations can be made for further research.

Firstly, the main recommendation on this project is on the specimen preparation process. The process must be done perfectly especially in the machining part and on the dimension of the specimen. The result of the test experiment can be affect or change only because of the inaccurate dimension of the specimen. Besides that, there can also be affected from defect occurs during machining process.

Besides that, the other recommendation that can be used for further research is use of high speed camera in order to see the deformation process of the specimen during tensile test experiment. From this, the change of the specimen can be seen from start its necking until the specimen became fracture.

REFERENCES

Cosham, A., & Hopkins, P. 2003.The assessment of corrosion in pipelines– Guidance in the pipeline defect assessment manual (PDAM).*In International Colloquium Reliability of High Pressure Steel Pipelines: 1*-30

Giedon, D.N and R.B. Smith. 1980. An analysis of Reportable Incidents for Natural Gas Transmission and Gathering Lines 1970 through 1978.

Jiang, Y., & Chen, M. 2012. Researches on the Fatigue Crack Propagation of Pipeline Steel.*Energy Procedia*, *14:* 524-528.

Joseph R. Davis. 2004. Tensile Testing, 2nd Edition.

Ling, Y. 1996. Uniaxial true stress-strain after necking. *AMP Journal of Technology*, *5*: 37-48.

Paik, J. K., Satish Kumar, Y. V., & Lee, J. M. 2005.Ultimate strength of cracked plate elements under axial compression or tension. *Thin-Walled Structures*,*43*(**2**):237-272.

Papadakis, G. A. 1999. Major hazard pipelines: a comparative study of onshore transmission accidents. *Journal of Loss Prevention in the process Industries*, *12*(**1**): 91-107.

Reis, F., Malcher, L., Andrade Pires, F. M., & César de Sá, J. M. A. 2010.A modified GTN model for the prediction of ductile fracture at low stress triaxialities.*International Journal of Structural Integrity*

William D. Callister. 2006. Materials Science and Engineering an Introduction 7th Edition

Williams, K. V., Vaziri, R., &Poursartip, A. 2003.A physically based continuum damage mechanics model for thin laminated composite structures. *International Journal of Solids and Structures*, *40*(**9**): 2267-2300.

APPENDICES

APPENDICES A1

FYP 1 GANTT CHART

No.	TASKS		WEEKS													
			1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	Meeting with supervisor to get the project title, and get the logbook at FKM office	Planned														
		Actual														
2	Determine the project objectives, scopes and background project	Planned														
		Actual														
3	Literature study	Planned														
		Actual														
5	Analysis of problems	Planned														
		Actual														
6	Material properties identified	Planned														
		Actual														
7	Making draft report and send to supervisor	Planned														
		Actual														
8	Correction and submit draft report to PSM coordinator	Planned														
		Actual														
9	FYP 1 presentation	Planned														
		Actual														

APPENDICES A2

FYP 2 GANTT CHART

No.	TASKS		WEEKS														
			1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
1	Fabrication Process	Planned															
		Actual															
2	Data and Analysis	Planned															
		Actual															
3	Simulation Process	Planned															
		Actual															
4	Data and Analysis	Planned															
		Actual															
5	Making final report	Planned															
		Actual															
6	Correction and submit final report	Planned															
		Actual															
7	Preparing final year project 2 presentation	Planned															
		Actual															
8	Final year project 2 presentation	Planned															
		Actual															

APPENDIX B

CHEMICAL COMPOSITION

FOUNDRY LABORATORY
FACULTY OF MECHANICAL ENGINEERING
UNIVERSITI MALAYSIA PAHANG

Chemical Results

Date: 11/03/2013

Spectrometer Foundry-MASTER Grade :

	Fe	C	Si	Mn	P	S	Cr	Mo
1	98,5	0,267	0,354	0,564	< 0,0100	< 0,0100	0,0285	< 0,0100
2	98,8	0,249	0,288	0,552	< 0,0100	< 0,0100	0,0247	< 0,0100
3	98,8	0,257	0,286	0,562	< 0,0100	< 0,0100	0,0264	< 0,0100
Ave	98,7	0,258	0,309	0,559	< 0,0100	< 0,0100	0,0265	< 0,0100

	Ni	Al	Co	Cu	Nb	Ti	V	W
1	0,0174	0,0499	< 0,0100	< 0,0050	< 0,0050	0,0097	< 0,0050	< 0,0250
2	0,0117	0,0138	< 0,0100	< 0,0050	< 0,0050	0,0042	< 0,0050	< 0,0250
3	0,0132	0,0255	< 0,0100	< 0,0050	< 0,0050	0,0041	< 0,0050	< 0,0250
Ave	0,0141	0,0298	< 0,0100	< 0,0050	< 0,0050	0,0060	< 0,0050	< 0,0250

	Pb
1	< 0,0500
2	< 0,0500
3	< 0,0500
Ave	< 0,0500