EXPERIMENTAL AND SIMULATION STUDY ON THE EFFECT OF FRICTION IN A FOUR-POINT BENDING TEST (4PB)

ABDUL ZARIF BIN ABDUL MALEK

Report submitted in partial fulfilment of the requirements
for the award of Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

JUNE 2012

UNIVERSITI MALAYSIA PAHANG
FACULTY OF MECHANICAL ENGINEERING

I certify that the project entitled "Experimental and simulation study on the effect of friction in a four-point bending test (4-PB)" is written by Abdul Zarif Bin Abdul Malek. I have examined the final copy of this project and in our opinion; it is fully adequate in terms of scope and quality for the award of the degree of Bachelor of Engineering. I herewith recommend that it be accepted in partial fulfilment of the requirements for the degree of Bachelor of Mechanical Engineering.

Examiner Signature

Dr. Akhtar Razul B. Razal

SUPERVISOR'S DECLARATION

I hereby declare that I have checked this project report and in our opinion this project is satisfactory in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering with "specialization".

Signature : 

Name of Supervisor : MR MOHD ZAIDI BIN SIDEK

Position : FACULTY OF MECHANICAL ENGINEERING LECTURER

Date : 22 JUNE 2012

STUDENT'S DECLARATION

I hereby declare that the work in this report is my own except for quotations and summaries which have been duly acknowledged. The report has not been accepted for any degree and is not concurrently submitted for award of other degree.

Signature :

Name : ABDUL ZARIF BIN ABDUL MALEK

ID Number : ME08007

Date : 22 JUNE 2012

ACKNOWLEDGEMENTS

I would like to thank all people who have helped and inspired me during completed this studies. I especially want to thank my supervisor, Mr Mohd Zaidi Bin Sidek, for his guidance during completed this project. His perpetual energy and enthusiasm in supervising had motivated his entire student, including me. In addition, he was always accessible and willing to help her students with their project. As a result, this project can be completed.

All of my friends at the Universiti Malaysia Pahang made it a convenient place to study. In particular, I would like to thank my lab mates, Mohd Zaki Bin Che M. Nor, Nur Syafiqah Binti Mohd Nawawi, Mohd Shahril Bin Mohd Hassan, Mohd Adzlan Bin Mohd Azmi, Mohd Jaafar Bin Mohd Nor, and all of the staff of the Mechanical Engineering Department, UMP who helped me in many ways.

My deepest gratitude goes to my family for their unflagging love and support throughout our life; this dissertation is simply impossible without them. Last but not least, thanks be to God for giving us living in this universe and provide us with all necessary things.

ABSTRACT

Finite element evaluation is one of the methods in predicting the strain value in sheet metal bending. Predicting the strain value is sometimes used in areas of plastic deformation, which is the integral of the ratio of the incremental change in length to the instantaneous length of a plastically deformed. This thesis aims to evaluate the reliability of finite element method by comparing the results with experimental results. The effect of parameters such as length of displacement after bend also has been studied. Abaqus software has been used to simulate the bending process and the mechanical properties provided from the Solidwork data will be used to run the simulation. In the four-point bending experiment, the test rig was clamped on Shimadzu machine and the mild steel sheets was assembled with strain gauge before bend process was run. Strain value being measured with Dasylab software. The results from the experiment and simulation is slightly different for value of strain, which the simulation shows the value of strain higher than strain value that was get from the experiment. For the free force four-point bending, value length displacement after bend almost the same on the simulation and experimental. Finite element method can be used to make comparison since the pattern of the graphs are nearly the same and percentages of error are below 10 %. The further study on parameters that effected bending process will make the finite element method is important in the future.

ABSTRAK

Kaedah analisis simulasi merupakan salah satu kaedah untuk meramal nilai terikan dalam pembengkokan kepingan logam. Ramalan nilai terikan kadang-kadang digunakan dalam bidang ubah bentuk plastik, yang penting nisbah perubahan pertambahan panjang panjang ketika plastik cacat. Laporan ini bertujuan untuk menilai kebolehan kaedah simulasi dengan membandingkan keputusan simulasi dengan keputusan eksperimen. Kesan parameter seperti panjang anjakan selepas selekoh juga dikaji. Perisian Abaqus telah digunakan untuk mensimulasikan proses lenturan dan sifat-sifat mekanik yang disediakan dalam data perisian Solidwork akan digunakan untuk menjalankan simulasi. Dalam eksperimen lentur empat mata, rig ujian telah dikepit pada mesin Shimadzu dan kunci keluli lembut telah dipasang dengan tolok terikan sebelum proses lenturan dijalankan. Nilai terikan diukur dengan memggunakan perisian Dasylab. Hasil dari eksperimen dan simulasi adalah sedikit berbeza untuk nilai terikan, di mana simulasi menunjukkan nilai ketegangan yang lebih tinggi daripada nilai terikan yang telah didapati dari eksperimen. Untuk tenaga bebas empat mata lenturan, nilai anjakan panjang selepas selekoh yang hampir sama pada simulasi dan eksperimen. Kaedah simulasi boleh digunakan untuk membuat pembandingan kerana corak graf adalah hamper sama dan peratusan ralat di bawah 10 %. Kajian lanjut mengenai parameter yang mempengaruhi proses pembengkokan adalah penting pada masa akan datang.

TABLE OF CONTENTS

LIST TABLE

LIST OF FIGURES

LIST OF SYMBOLS

σ^2	Variance
Pmax	Max bending load
μ	Mean
η	Structural efficiency
L	Distance between load support
R	Normal anisotropic value
v	Poisson's ratio
E	Young's modulus
t	Sheet thickness
ρ	Neutral axis
I	Inertia moment of cross-section per unit width
$M(\alpha)$	Bending moment along the bending surface
F_s	Shear Force
K	Ultimate tensile strength
t	Sheet thickness
ΔK	Spring back curvature
M	Bending moment
Re	Yield Strength
L	Inertia moment of cross-section
W	Elastic Section Modulus
h	Specimen width

LIST OF ABBREVIATIONS

AISI	American Iron and Steel Institute
ASTM	American Society for Testing and Material
TRIP	Transformation Induced Plasticity
CNC	Computer Numerical Control
UTS	Ultimate Tensile Strength
SHS	Square Hollow Sections
EHS	Element Hollow Sections
FEA	Finite Element Analysis
LVDT	Linear Variable Displacement Transducers
PDE	Partial Differential Equations
FEM	Finite Element Method
CHS	Circular Hollow Sections
RHS	Rectangular Hollow Sections
CAD	Computer-Aided Design
EDM	Electric Discharge Machining
3PBT	Three Point Bend Test
4PBT	Four Point Bend Test
DAQ	Data Acquisition System

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND

Steel, one of the vital materials used in the construction of roads, railways, other infrastructure, appliances, and buildings. Steel is used in a variety of other construction materials, such as bolts, nails, and screws. In this project, the effect of friction in four- point bending test study, be studied; and the mild steel was prepared as a workpiece. The four-point bending test was conducted in order to determine the effect of friction on the mild steels workpiece on flexural strength. The experiment also conducted by using finite element application which to simulate the effect of friction on the mild steel workpiece with a different value of coefficient of friction.

Four-point bending test is broadly used because it is suitable for testing large sample size of packages at similar loading condition (bending moments) between the inner load span regions. In contrast with other commonly used mechanical test methods such as tension, plane bending and torsion, four-point bending has its own advantages for characterizing the mechanical properties of materials.

Firstly, it produces a uniform moment between the two inner loading rollers in the specimen which gives ascend to a uniform maximum tensile stress in the specimen surface. Secondly, no special sample gripping is required for the four-point bend test, which makes it possible to test brittle materials in tension and makes sample preparation rather simple since a specimen with a uniform rectangular cross-section is usually used in the test. Thirdly, sample mounting and dismounting are fairly straightforward in four-point bend, which is particularly suitable for high temperature tests in which sample mounting and dismounting can be a non-trivial matter in bending test.

Furthermore, using four-point bend a pure shear stress can be applied to a test piece by asymmetrically loading the specimen. Because of these advantages four-point bending can be a convenient test method for bending studies, especially for studying on the effect of friction since the specimen for four-point bend has a flat surface on which a uniform tensile stress is applied.

1.2 PROBLEM STATEMENTS

The three-point bending end-notched flexure (3ENF) and, more recently, four-point bend end-notched flexure (4ENF) tests are often used to determine the mode IIdelamination toughness, GIIc, of laminated composites. The 3ENF, as shown has been used more extensively, but the 4ENF, is gaining popularity due to the stable nature of crack advance. This allows for the determination of non-pre-cracked and pre-cracked toughness's, as well as resistance curves, from each specimen tested.

Bending test is to measure the strength of a material when a material is in force. The focus is where the point of material to start bending and to see and determine the friction effect on the material. Experimental and simulation result need to be compare.

1.3 OBJECTIVES OF PROJECT

The objectives of the project are as follows

(i) To design and fabricate test rig for the four point bending test (4-PB).
(ii) To establish experiment concentrates on the effect of friction in 4-PB test.
(iii) To compare between the experiment and the stimulation by using FEM software.

1.4 SCOPE OF PROJECT

Before performing the simulation, the structural modeling of the bending test-rig needs to be developed by using computer-aided design (CAD) software. The structure is modeled and then imported onto computer-aided engineering (CAE) software to mesh the test-rig and the specimens. The finite element modeling (FEM) processes were performed by using ABAQUS version 6.7. Thus, producing the result of stress, strain and displacement where it was used to analyze the critical area of the specimen. Finally the simulation took place and the result is used to compare with the experiment results.

This research is focusing in:

(i) To design and fabricate fixtures for four point bending test on Shimadzu Machine

(ii) The tested material is mild steel.

(iii) Experiment was conducted to validate of all input parameters.

(iv) To conduct simulation using the finite element software like Abaqus, Algor or patran.

(v) The effect of the known parameters was tested

(vi) Similar test condition was used to compare the result

1.5 OUTLINE OF REPORT

Chapter 1 introduces the background, problem statement and the scopes of this study. Chapter 2 presents the literature study about material used, finite element method and optimization of the bending test. Chapter 3 discusses the methodology from designing until how to run the experiment, finite element modeling and the optimization technique. Chapter 4 discusses the results and analysis of the four-point bending test experiment, finite element analysis, modal analysis and optimization of the bending test. Chapter 5 presents the conclusion and recommendation of the future work.

1.6 RESEARCH FLOW

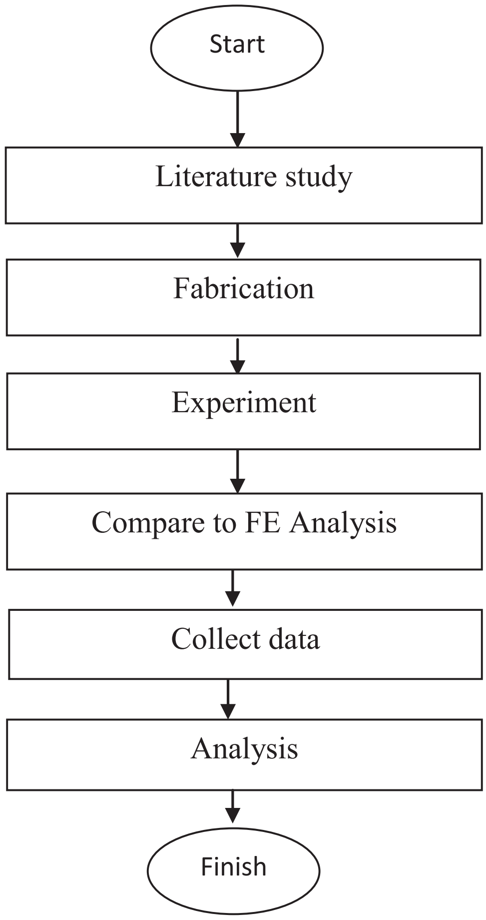


Figure 1.0: Research Flow

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

The purpose of this chapter is to provide information which related to the material used; mild steel, finite element analysis (FEA) and also about bending test.

2.2 MILD STEEL

Mild steel is a type of steel alloy that contains a high amount of carbon as a major element. An alloy is a mixture of metals and non-metals, designed to have specific properties. Alloys make it possible to compensate for the shortcomings of a pure metal by adding other elements. To get what mild steel is, one must know what the alloys that are combined to make steel are.

Steel is any alloy of iron, consisting of 0.2% to 2.1% of carbon, as a hardening agent. Besides carbon, there are many metal elements that are a part of steel alloys. The elements other than iron and carbon, used in steel are chromium, manganese, tungsten and vanadium. All these elements along with carbon, act as hardening agents. That is, they prevent dislocations from occurring inside the iron crystals and prevent the lattice layers from sliding past each other. This is what makes steel harder than iron. Varying the amounts of these hardening agents creates different grades of steel. The ductility, hardness and mild steel tensile strength are a function of the amount of carbon and other hardening agents, present in the alloy. The amount of carbon is a deciding factor, which decides hardness of the steel alloy. A steel alloy with a high carbon content is mild steel, which is in fact, much harder and stronger than iron. Though, increased carbon content increases the hardness of the steel alloy.

2.2.1 Types of Mild Steel

(a) 1018 Mild Steel

Alloy 1018 is the most commonly available of the cold-rolled steels. It is generally available in round rod, square bar, and rectangle bar. It has a good combination of all of the typical traits of steel strength, some ductility, and comparative ease of machining. Chemically, it is very similar to A36 Hot Rolled steel, but the cold rolling process creates a better surface finish and better properties.

Table 2.1: Properties of 1018 Mild (low-carbon) Steel

Minimum Properties	Ultimate Tensile Strength,psi	63,800
	Yield Strength, psi	53,700
	Elongation	15.0%
	Rockwell Hardness	B71
Chemistry	Iron (Fe)	98.81-99.26%
	Carbon (C)	0.18%
	Manganese (Mn)	0.6-0.9%
	Phosphorus (P)	0.04% max
	Sulfur (S)	0.05% max

Source: http://www.onlinemetals.com/alloycat.cfm?alloy=1018

(b) A36 Mild Steel

ASTM A36 steel is the most commonly available of the hot-rolled steels. It is generally available in round rod, square bar, rectangle bar, as well as steel shapes such as I-Beams, H-beams, angles, and channels. The hot roll process means that the surface on this steel will be somewhat rough. Note that its yield strength is also significantly less than 1018 this means that it will bend much more quickly than will 1018. Finally, machining this material is noticeably more difficult than 1018 steel, but the cost is usually significantly lower.

Table 2.2: Properties of A36 Mild Steel

Minimum Properties	Ultimate Tensile Strength,psi	58,000-79,800
	Yield Strength, psi	36,300
	Elongation	20.0%
Chemistry	Iron (Fe)	99%
	Carbon (C)	0.26%
	Manganese (Mn)	0.75%
	Copper (Cu)	0.2%
	Phosphorus (P)	0.04% max
	Sulfur (S)	0.05% max

Source: http://www.onlinemetals.com/alloycat.cfm?alloy=A36

2.3 FINITE ELEMENT METHOD

Finite element analysis (FEA) is one of CAE tools and has become humdrum in recent years. Numerical solutions to even very complicated stress problems can now be obtained routinely using FEA, and the method is so important that even preparatory treatments of Mechanics of Materials such as these projects should outline its principal features. The finite element method (FEM) (its practical application often known as FEA) is a numerical technique for verdict approximate solutions of partial differential equations (PDE) as well as integral equations. The solution approach is based either on eliminating the differential equation completely (steady state problems), or interpretation the PDE into an approximating system of ordinary differential equations, which are then numerically integrated using standard techniques such as Euler's method, Runge-Kutta.

In solving partial differential equations, the primary challenge is to create an equation that approximates the equation to be studied, but is numerically stable, meaning that errors in the input and intermediate calculations do not accumulate and cause the resulting output to be meaningless. There are many ways of doing this, all with advantages and disadvantages. The finite element method is a good choice for solving partial differential equations over complicated domains (like cars and oil pipelines), when the domain changes (as during a solid state reaction with a moving boundary), when the desired precision varies over the entire domain, or when the solution lacks smoothness. For instance, in a frontal crash simulation it is possible to increase prediction accuracy in "important" areas like the front of the car and reduce it in its rear (thus reducing cost of the simulation). Another example would be in Numerical weather prediction, where it is more important to have accurate predictions over developing highly-nonlinear phenomena (such as tropical cyclones in the atmosphere, or eddies in the ocean) rather than relatively calm areas.

2.4 THEORY OF BENDING TESTS

In engineering mechanics, bending (also known as flexure) characterizes the behavior of a slender structural element subjected to an external load applied perpendicularly to a longitudinal axis of the element. The structural element is assumed to be such that at least one of its dimensions is a small fraction, typically 1/10 or less, of the other two. When the length is considerably longer than the width and the thickness, the element is called a beam. A closet rod sagging under the weight of clothes on clothes hangers is an example of a beam experiencing bending. On the other hand, a shell is a structure of any geometric form where the length and the width are of the same order of magnitude but the thickness of the structure (known as the 'wall') is considerably smaller. A large diameter, but thin-walled, short tube supported at its ends and loaded laterally is an example of a shell experiencing bending.

In the absence of a qualifier, the term bending is ambiguous because bending can occur locally in all objects. To make the usage of the term more precise, engineers refer to the bending of rods, the bending of beams, the bending of plates, the bending of shells, and so on.

Figure 2.1: Bending of an I-beam

Typically the bend test measures ductility, the ability of a material to change form under pressure and keep that form permanently. In certain cases the bending test can determine tensile strength. When using the bend test for this purpose, testers examine which side of the material breaks first to see what type of strength the material has. It also lets them know what kinds of pressure it holds up against and what kinds it doesn't. Ductility describes how well a material, usually metal, can be stretched and keeps its new shape. Steel, for example, is highly ductile. If pressure is applied that stretches the steel into a new shape, it will keep this shape even after the pressure has been removed. This characteristic is referred to as ductility and is a desirable characteristic for metals and other building materials.

To determine how ductile a material is, a bending test is used. Force is applied to a piece of the material at a specific angle and for a specific amount of time. The material is then bent to a certain diameter using force. After the bending test is over, the material is examined to see how well it held its shape once the pressure was removed, and whether or not the material cracked when pressure was applied. Bending tests are used for determining mechanical properties of unidirectional composite materials. Due to the important influence of shear effects in the displacements, great span-to-depth ratios are used in order to eliminate these effects.

Four-point test configurations are used in order to obtain flexural strength and flexural modulus. The rotation of the cross sections in the deformation process leads to the contact zone between specimen and cylindrical supports changing in a three-point bending test. Furthermore, in a four-point bending test the contact between specimen and cylindrical loading noses also changes.

2.4.1 Flexural Strength

Flexural strength, also known as modulus of rupture, bend strength, or fracture strength, a mechanical parameter for brittle material, is defined as a material's ability to resist deformation under load. The transverse bending test is most frequently employed, in which a rod specimen having either a circular or rectangular cross-section is bent until fracture using a three point flexural test technique. The flexural strength represents the highest stress experienced within the material at its moment of rupture. It is measured in terms of stress, here given the symbol σ.

When an object formed of a single material, like a wooden beam or a steel rod, is bent (Figure 2.2), it experiences a range of stresses across its depth (Figure 2.3). At the edge of the object on the inside of the bend (concave face) the stress will be at its maximum compressive stress value. At the outside of the bend (convex face) the stress will be at its maximum tensile value. These inner and outer edges of the beam or rod are known as the 'extreme fibers'. Most materials fail under tensile stress before they fail under compressive stress, so the maximum tensile stress value that can be sustained before the beam or rod fails is its flexural strength.

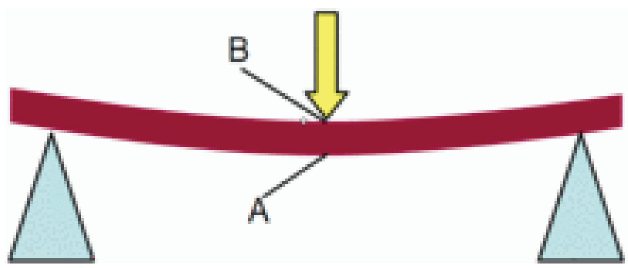


Figure 2.2: Beam of material under bending. Extreme fibers at B (compression) and A (tension)

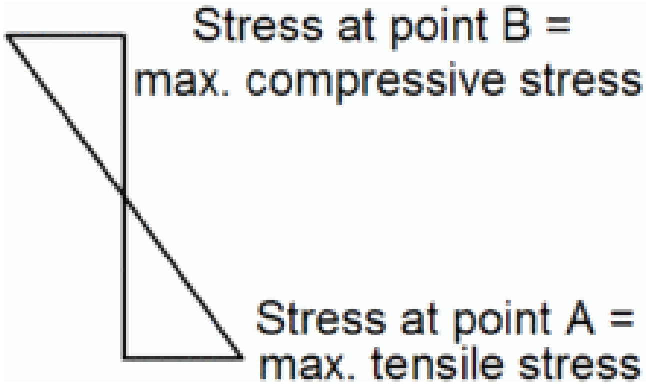


Figure 2.3: Stress distribution across beam

2.4.1.1 Flexural Versus Tensile Strength

The flexural strength would be the same as the tensile strength if the material was homogeneous. In fact, most materials have small or large defects in them which act to concentrate the stresses locally, effectively causing a localized weakness. When a material is bent only the extreme fibers are at the largest stress so, if those fibers are free from defects, the flexural strength will be controlled by the strength of those intact 'fibers'. However, if the same material was subjected to only tensile forces then all the fibers in the material are at the same stress and failure will initiate when the weakest fiber reaches its limiting tensile stress. Therefore it is common for flexural strengths to be higher than tensile strengths for the same material. Conversely, a homogeneous material with defects only on it surfaces (e.g. due to scratches) might have a higher tensile strength than flexural strength. If we don't take into account defects of any kind, it is clear that the material will fail under a bending force which is smaller than the corresponding tensile force. Both of these forces will induce the same failure stress, whose value depends on the strength of the material.

For a rectangular sample, the resulting stress under an axial force is given by the following formula:

$$\sigma = \frac{F}{bd} \tag{2.1}$$

This stress is not the true stress, since the cross section of the sample is considered to be invariable (engineering stress).

- F is the axial load (force) at the fracture point
- b is width
- d is thickness

The resulting stress for a rectangular sample under a load in a three-point bending setup (Figure 2.4) is given by the formula below (see "Measuring flexural strength"). The equation of these two stresses (failure) yields:

$$F = \frac{3FL}{2d} \tag{2.2}$$

Usually, L (length of the support span) is much bigger than d, so the fraction $\frac{3L}{2d}$ is bigger than one.

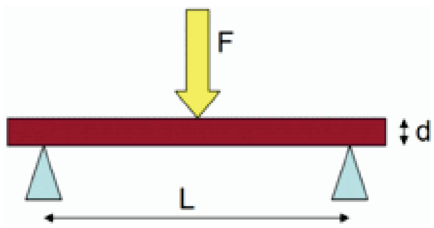


Figure 2.4: Beam under 3 point bending

2.4.1.2 Measuring Flexural Strength

For a rectangular sample under a load in a three-point bending setup (Figure 2.4):

$$\sigma = \frac{3FL}{2bd^2} \tag{2.3}$$

- *F* is the load (force) at the fracture point
- *L* is the length of the support span
- *b* is width
- *d* is thickness

For a rectangular sample under a load in a four-point bending setup where the loading span is one-third of the support span:

$$\sigma = \frac{FL}{bd^2} \tag{2.4}$$

- *F* is the load (force) at the fracture point
- *L* is the length of the support (outer) span
- *b* is width
- *d* is thickness

For the 4 pt bend setup, if the loading span is 1/2 of the support span (i.e. L_i = 1/2 L in Figure 2.5):

$$\sigma = \frac{3FL}{4bd^2} \tag{2.5}$$

If the loading span is either 1/3 or 1/2 the support span for the 4-point bending setup (Figure 2.5):

$$\sigma = \frac{3F(L-L_i)}{2bd^2} \tag{2.6}$$

- L_i is the length of the loading (inner) span

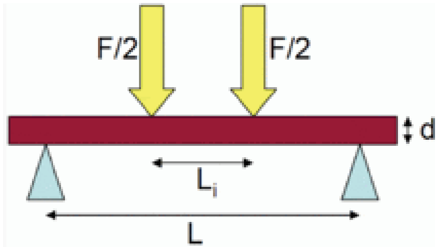


Figure 2.5: Beam under 4 point bending

2.4.2 Bending Test

Two principal symmetrical testing configuration four point bending and three-point bending were employed. The primary aim of adopting these two bending configurations was to investigate the cross-section response under constant moment (four-point bending tests) and a moment gradient (three-point bending tests). In the four-point bending tests, the beam was of span 4.5 m while for the three-point bending tests, the span was 3 m. This provided a consistent length of 1.5 m between the end support and the loading point. These two statically determinate configurations provide fundamental information, which may be used to simplify the analysis of redundant structures. A total of 18 in-plane bending tests were conducted. Full moment–curvature and moment–rotation histories were derived for the four-point bending and three point bending tests respectively.

2.4.3 Four-point Bending Test Analysis

2.4.3.1 Stress distribution with linear material properties

The specimen geometry for the four-point beam test is shown in Figure 2.6. Only half of the specimen needs to be modeled as a result of symmetry. The finite element model of the four-point bending specimen is shown in Figure 2.7. The specimen modeled is 32 plies thick. The nominal value of the thickness of 4-062 mm is used (Figure 2.7). T~ = T3 =0.254mm, with five elements in each region. T2 = 3.554 mm with 10 elements in this region chosen. The total length of the specimen is 50 mm. The span is 40 mm. The distance between the two loading rollers is 20 mm.

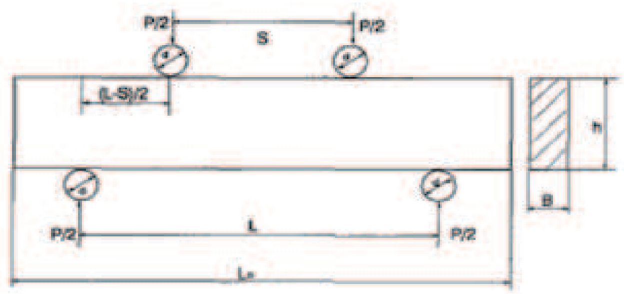


Figure 2.6: Four-point bending test geometry

Source: (Wei Cheng Cui & Michael R. Wisnom, 2011)

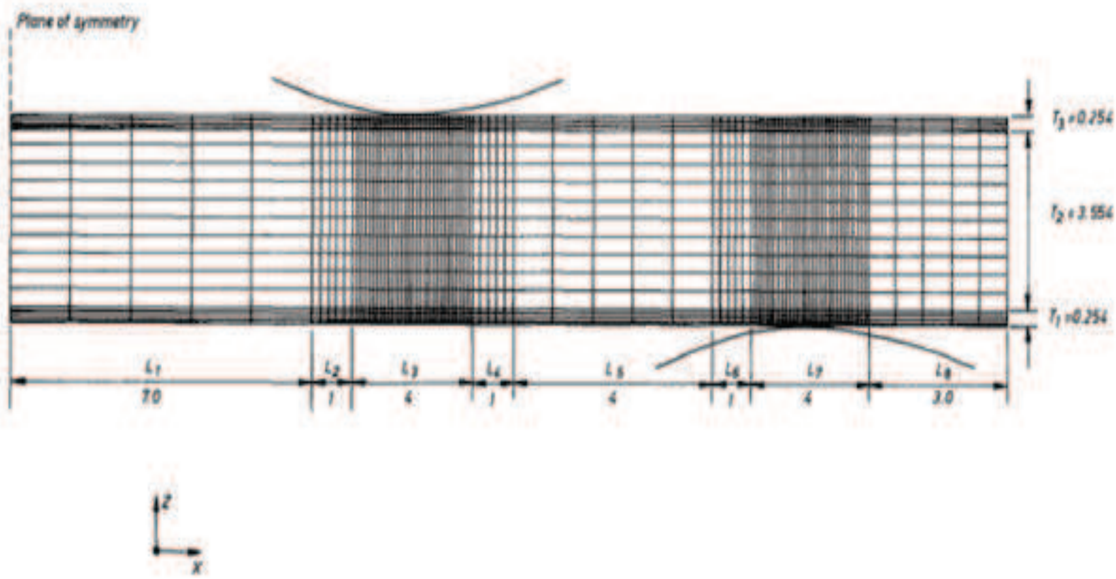


Figure 2.7: The finite element model of four-point bending specimen (dimensions in mm)

Source: (Wei Cheng Cui & Michael R. Wisnom, 2011)

2.4.3.2 Stress distribution with nonlinear material properties

Nonlinear analysis of the four-point bending specimen has also been carried out. The material properties are the same as for the three-point bending case. The finite element mesh and other details are the same as for the linear material case. It can be seen that the influence of nonlinearity on the maximum shear stress is significant. Therefore, this influence should be taken into account in the stress analysis. The shear stress contours for the four-point bending specimen are shown in Figure 2.8. The shear stress distributions at various sections are compared with beam theory in Figure 2.9.

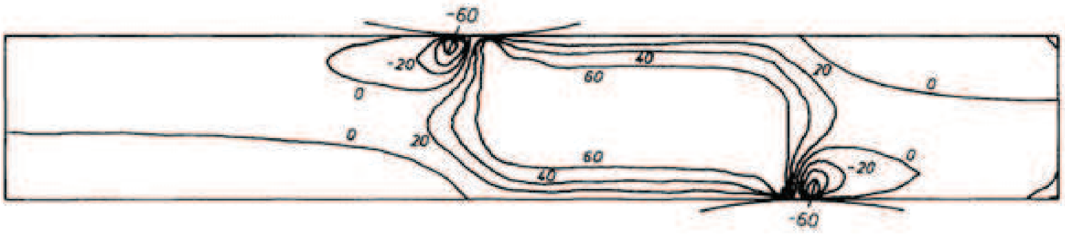


Figure 2.8: Shear stress contours (Mpa) in four-point bending specimen (nonelinear material properties, roller diameter 20mm).

Source: (Wei Cheng Cui & Michael R. Wisnom, 2011)

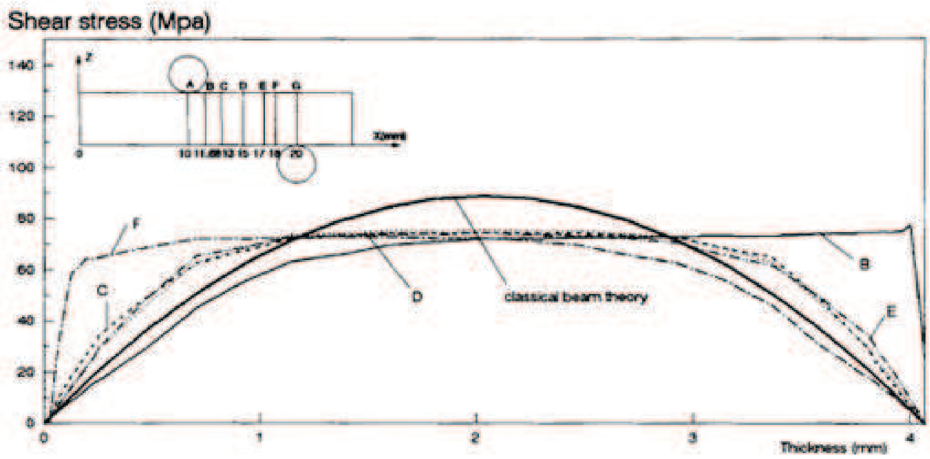


Figure 2.9: Shear stress distributions in four-point bending specimen (nonelinear material properties, roller diameter 20mm).

Source: (Wei Cheng Cui & Michael R. Wisnom, 2011)

CHAPTER 3

METHODOLOGY

3.1. INTRODUCTION

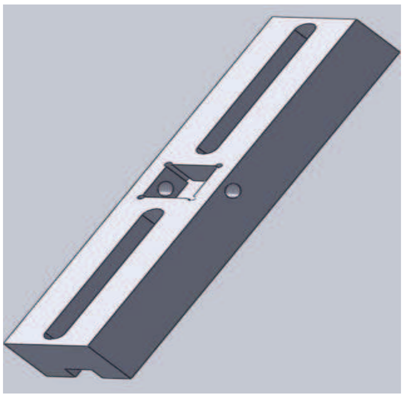
This project is mainly about design and fabrication four-point bending test rig for experimental on the effect of friction on parts performance. In general, this project used the four point bending test method like other researchers. With different apparatus and approach, the experiment was conducted in order to get the result and achieve the objective. The aim of this chapter is to develop a methodology on how to make the test rig for experimental use, the setup of the experiment itself and the simulation of finite element method.

3.2 DESIGN

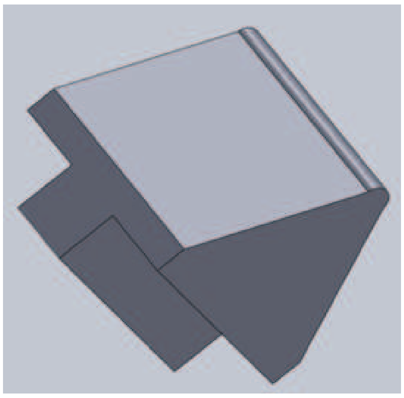
SolidWorks software was used to design the test rig components. SolidWorks is a 3D mechanical CAD (computer-aided design) program that runs on Microsoft Windows.SolidWorks is a Parasolid-basedsolid modeler, and utilizes a parametric feature-basedapproach to create models and assemblies.

These are the components for the test rig with dimensions;

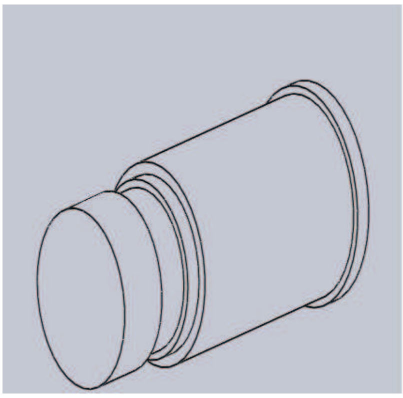
(a) Beam = 300mm length x 50 mm width x 40 mm height
(b) Striker = 50mm length x 50 mm width x 40 mm height
(c) Coupler = 70mm length with 32mm radius



(a) Beam



(b) Striker



(c) Coupler

Figure 3.1:(a) beam, (b) striker, and (c) coupler are the four-point bending test experiment components

3.3 FABRICATION

3.3.1 Cutting Process

Band-saw cutting machine was used to cut the raw material. The saw blade is movable, allowing the user to adjust the angle at which the blade comes in contact with the raw material. The blade can be moved up and down, thereby allowing the user to cut raw material at different depths according to their needs. Figure 3.2 show band-saw cutting machine.

Figure 3.2: Band-saw cutting machine

3.3.2 Squaring Process

Conventional milling machine used to clean up the raw material from the carbon layer and also to minimize the size of the material. Milling cuttersare cutting tools typically used inmilling machines or machining centersand occasionally in othermachine tools. They remove material by their movement within the machinery directly from the cutter's shape. Figure 3.3 show the conversional milling machine.

Figure 3.3: Conventional milling machine

3.3.3 Grinding

A grinding machine, often shortened to grinder, is a machine tool used for grinding, which is a type of machining using an abrasive wheel as the cutting tool. Grinding is used to finish work pieces which must show high surface quality (e.g., low surface roughness) and high accuracy of shape and dimension. For this project, I used the grinding machine for surface finish. Figure 3.4 show the grinding machine.

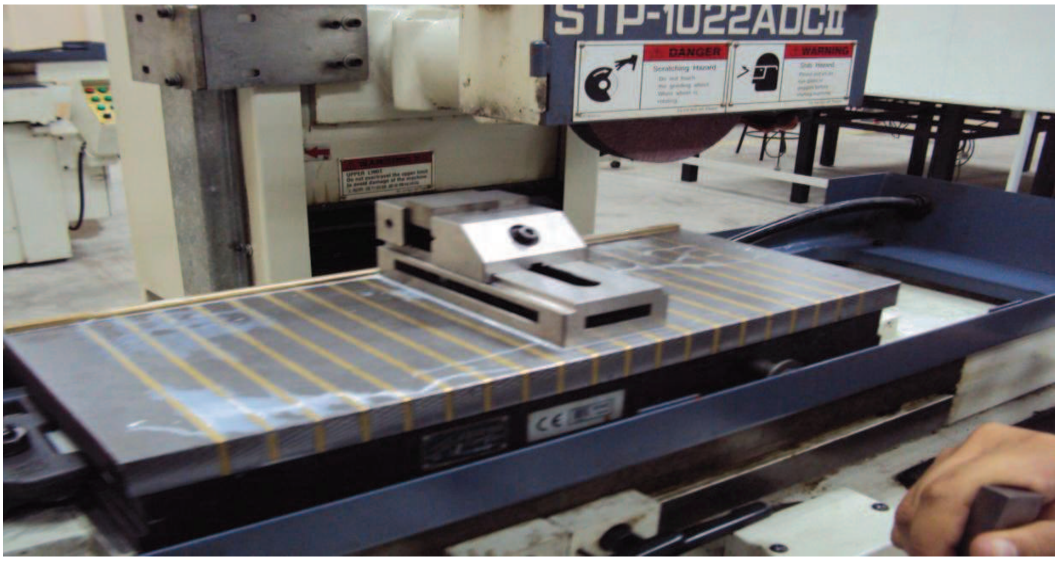


Figure 3.4: Grinding machine

3.3.4 CNC Machine

Before do the cutting part, we need to draw the design by using Mastercam. Mastercam's comprehensive set of predefined tool paths which including contour, drill, pocketing, face, peel mill, engraving, surface high speed, advanced multiaxis, and many more and enable us to cut parts efficiently and accurately.

3.4.6 Tapering

Process fabricate the fixture continue by using Electric discharge machining (EDM) wire cut. Wire-cut EDM is typically used to cut plates as thick as 300mm and to make punches, tools, and dies from hard metals that are difficult to machine with other methods.Wire-cutting EDM is used when low residual stresses are desired, because it does not require high cutting forces for removal of material. For this project, it used to produce taper angles of up to +/- 30º and R2 at edge of the striker.

3.4 EXPERIMENT AND SIMULATION

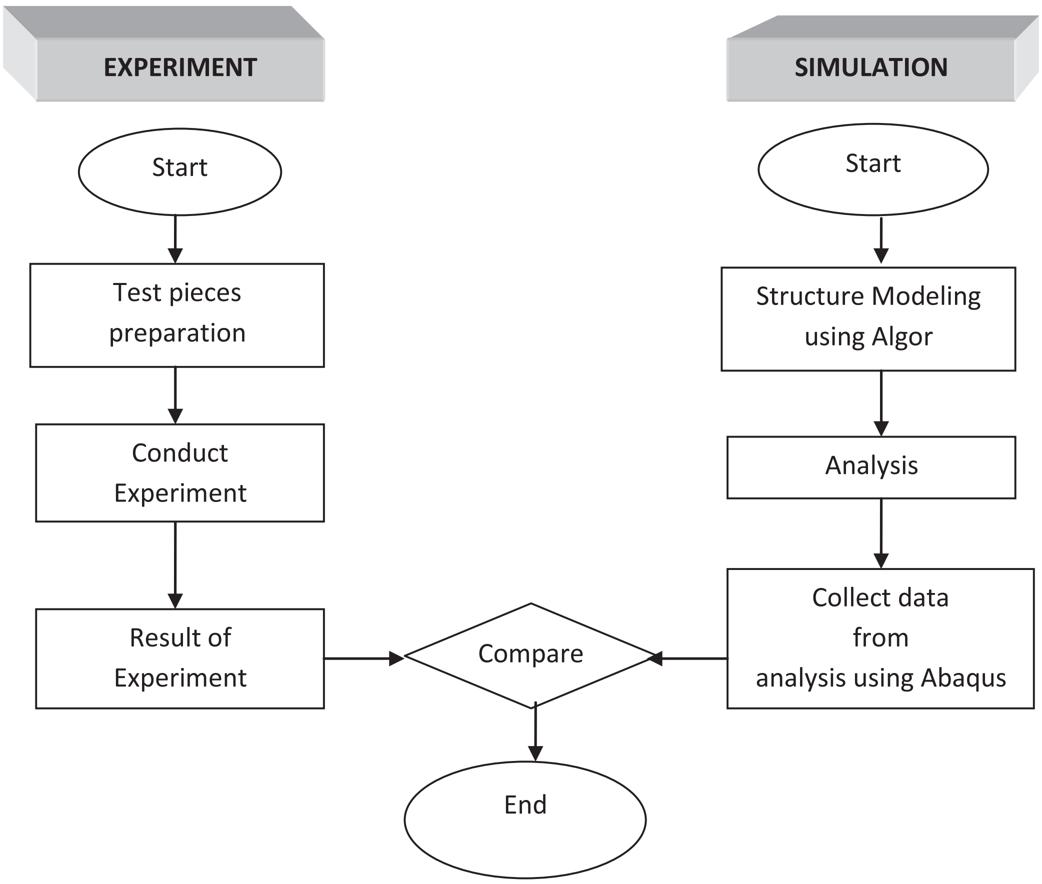


Figure 3.5: Experiment and simulation flow chart

3.4.1 Specimen preparation

First, thegalvanized steelis cut according to ASTM Standards into a rectangular size of 220mm x 30mm x 1mm by using shearing band-saw.

Figure 3.6: Shearing Band-saw

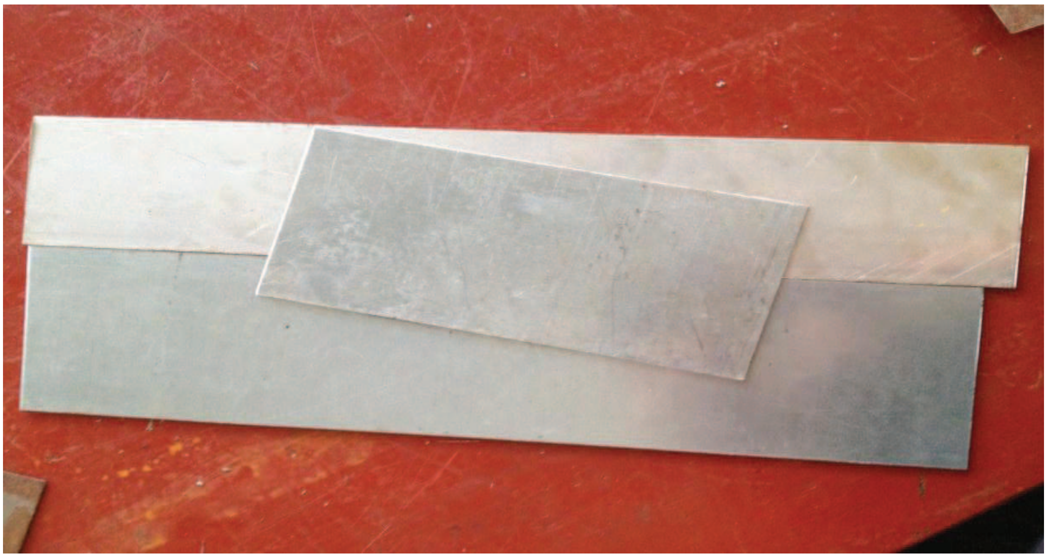


Figure 3.7: Plates of galvanized steel (220mm x 33 mm x 1mm)

3.4.2 Four-point Bending Test Experiment

Material that has been used in this project galvanized steel. The tensile test is used to evaluate the strength of metal alloys. In this test a galvanized steel sample is pulled to failure in a relatively short time at a constant rate. The bending test is to determine strength of a material by applying force to the item in question and seeing how it reacts under pressure. During this test process, force-extension data, a quantitative measure of how the test specimen deforms under the applied bend force, usually monitored and recorded.

Figure 3.8: Shimadzu Bending Test machine

This experiment offers a useful method for accessing a real performance of the thick sheet metal plate under the effect of inner span. The main objective is to study the bending outcome from the 4PBT under the difference of inner span. Based on the experimental results, it will be presented compared with the simulation analysis.

3.4.3 Test Setup

The tests were performed using a 100kN Shimadzu tension/compression machine. The machine is computer controlled with the ram movement is controlled from the control panel. The test rig was properly aligned on the Shimadzu machine by clamping onto the base at the center place of the test machine. The experimental setup used for the 4PBT is shown in Figure 3.9 to Figure 3.11.

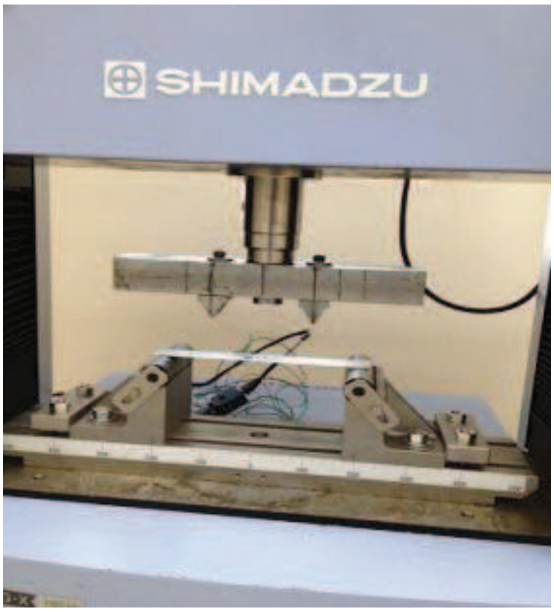


Figure 3.9: Four point bending test setup

Figure 3.10: Strain gauge connected to data logger

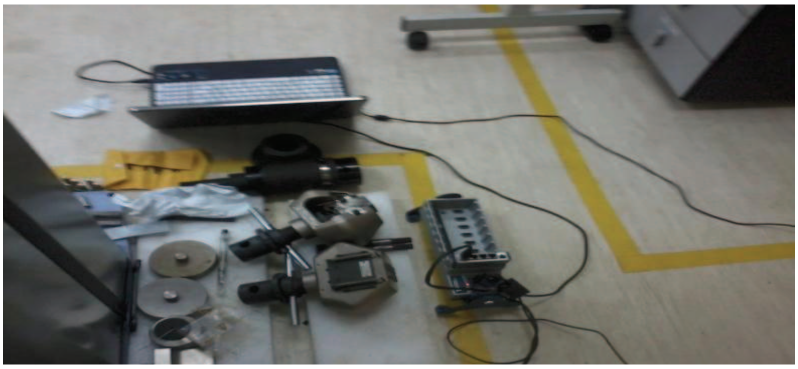


Figure 3.11: Data logger connected to laptop

For data collection, there are two data acquisition systems have been used in this experiment i.e. from the machine and PC-based a data acquisition system (DAQ). The data from the machine's DAQ is used to gather the bending speed during the 4PBT. Meanwhile, PC-based DAQ is purposely used for measurement of strain on the workpiece during the process. Strain gauge location on the specimen was show in Figure 3.12.

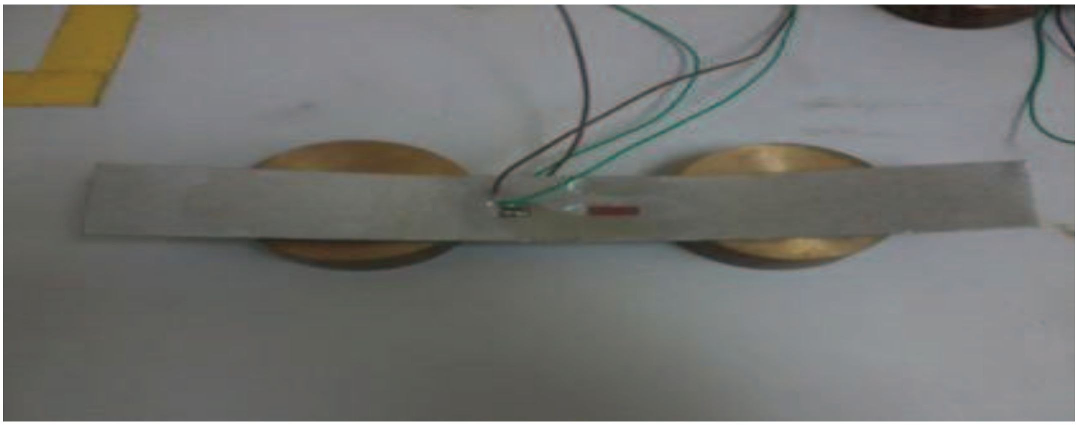


Figure 3.12: Strain gauge location on specimen during 4-PBT

There are a total of 9 specimens were used in this investigation. Specimens that used for this research is galvanized steel. In each process setting, three specimens have been used. For specimen preparation, the 1 mm thick in-received galvanized steel were cut with a shearingmachine to the specified workpiece size 220 mm x 33 mm. Among the consideration in cutting the workpiece is to standardize the workpiece width at 33 mm to accommodate plane strain analysis condition as implemented in the finite element analysis. All workpiece were marked on the sidewall to confirm a right setting during the bending process. Strain value was measured with a PC-based DAQ by using strain gauge. Strain gauges were spot welded at selected locations on the workpiece as shown in Figure 4.11. The signals were converted from analogue to digital by NI's data logger connected to the laptop.

3.4.4 Test Procedure

A total of three experiments were conducted on nominally identical workpiece where the centre line was located within the focal line region as shown in Figure 4.12. Initially, the used inner was 100mm and outer spans were fixed at 200 mm respectively. The test condition was changed with different inner span to 120mm and 140 mm.

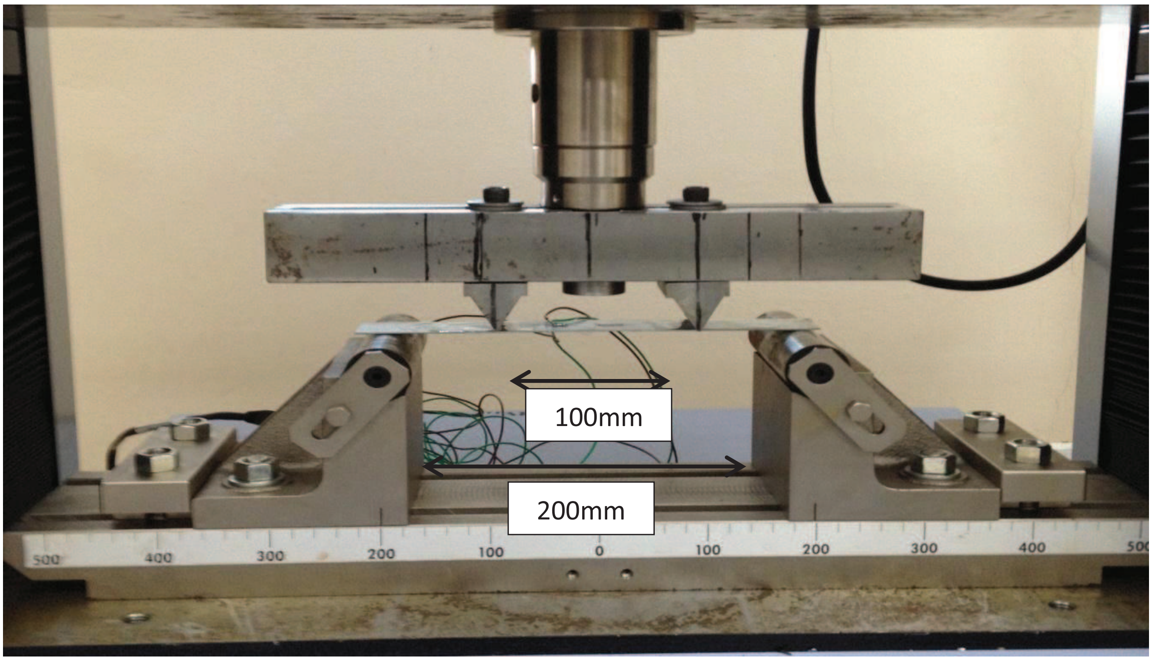


Figure 4.12: Specimen positioning inside the test rig during 4PBT

There are three main steps was set during the 4PBT. Initial setting had been done before the actual test was commenced. The workpiece with strain gauge ready were placed on the bending support. The machine ram was moved down to make an initial contact between the striker and workpiece top surface. Most importantly, the centre section of the blank must be positioned exactly at the middle between both of the bending supports. Displacement of striker in right and left must be same from the center.

The complete experiments were conducted in the following steps:

Step 1-The workpiece with strain gauge ready were placed on the bending support. Striker was set up 100mm and the support was fixed at 200mm. The stroke limit was setup fixed at 30mm from the center of the beam.

Step 2-It was followed by setting at the pc used the trapezium x software. In addition workpiece was connected to the data logger used the Dasylab software at the laptop.

Step 3-The final step was used for the four point bending process. The striker movement was calculated based on the plotted curve. In this experiment, the bending speed has been specified at 1 ms-1.Make sure run the four bending process from the press machine and Dasylab software from the laptop simultaneously.

The load on the specimen being tested is measured by the load cell while the strain is obtained from the extensometer attached to the specimen and the data is collected in a computer control software package.

CHAPTER 4

RESULTS AND DISCUSSION

4.1 INTRODUCTION

This chapter concentrates on the experimental and simulation work of the four-point bending tests. This chapter is started with discuss about a fundamental concept of flexural bending with concentration on four-point bending test. It is followed by explanation about practical works that involves in fabrication test rig, experimental work and finally gets the final result to make a discussion and analysis. Then, the finite element analysis has been adopted to simulate the similar four-point bending test. From the simulation analysis, it provides better understand on the related problem in the experiment. In the same time, it has been used to validate reliability of all the input parameters of the finite element simulation model.

4.2 TEST RIG FABRICATION

A complete test rig was built by considering it compatible to suit the space area to be fitted inside the Shimadzu machine. On these requirements, the completed 4-PBT test rig was designed and fabricated as shown in Figure 4.1. Size of the fabricated test rig is 300mm length x 50 mm width x 40 mm height for beam, 50mm length x 50 mm width x 40 mm height for striker and is 70mm length with radius 32mm for coupler.

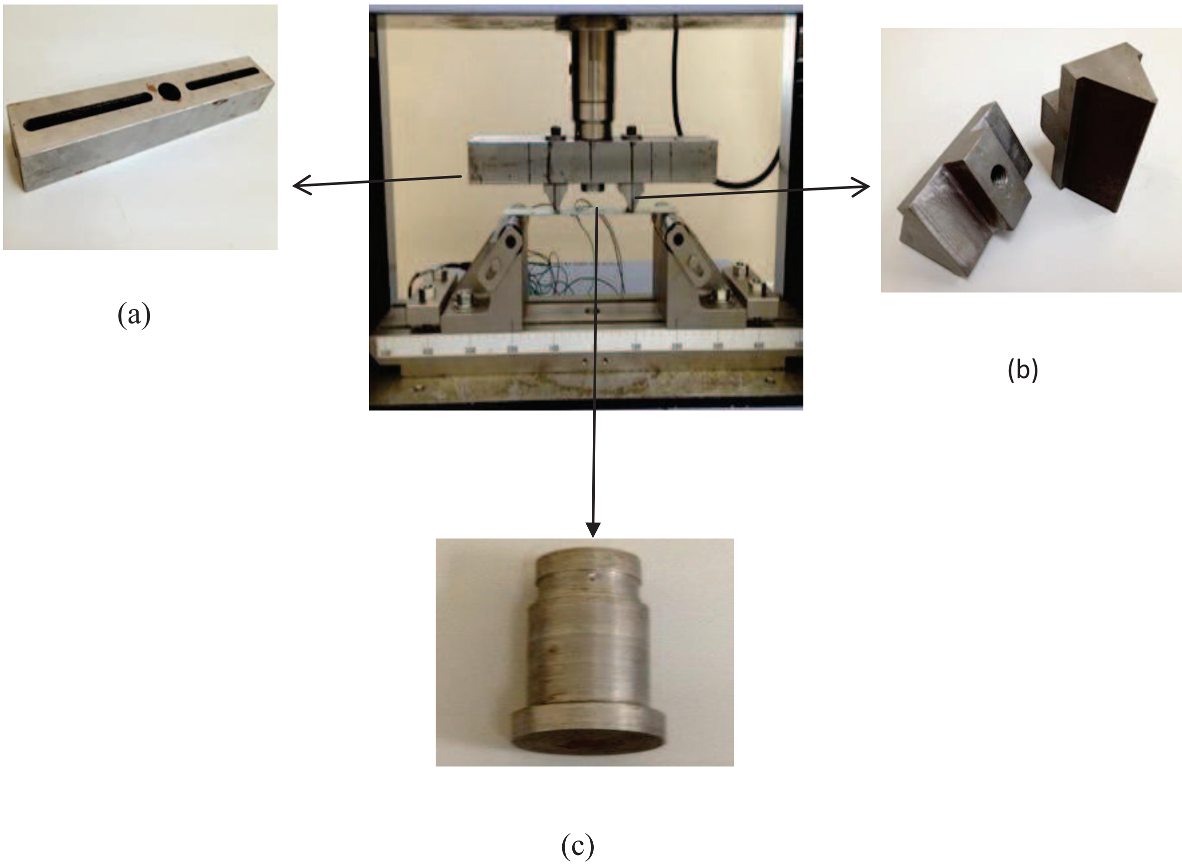


Figure 4.1: (a) beam, (b) striker, and (c) coupler

There are several important factors were considered during test rig fabrication stages are:

(a) Beam

It provides flat surface for uniform forces from the machine ram during bending process. The hollow shape is designed on the top plate surface for adjustable striker.

(b) Striker

Both top surface of striker are screwed to the plate.

(c) Coupling

There are four stage must be consider.

4.3 TEST RESULT

This subtopic discuss about the summarized result was get from the experiment.. Before proceed with a detail discussion about the experimental results as outlined in Table 3.1 there are some issues related the four-point bending tests. First, results that get from experiment differ from the scope of objective to make of “U” bending shape. To overcome that problem, a lot of experiments have been done with various improvements but still failed to produce the aimed results. Among the improvement in the experiments are;

i. Using the smallest inner span length
ii. Used different bending speed setting
iii. Used of smallest length of specimen

Considering all the problems, the experiment objective has been changed on identifying the material response under one test conditions inner span setting as presented in Table 5.1. Reason for adopting the inner span as a testing parameter is to standardize the test condition. In running the experiments, the outline of the test is summarized in Table 4.1.

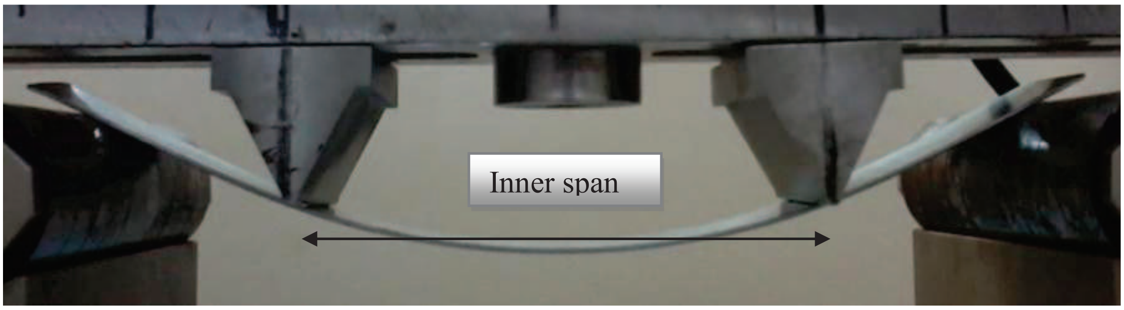
Table 4.1: The 4-PBT test program

Test Experimental	Thickness (mm)	Bending speed (m/s)	Inner span (mm)	Support (mm)	Specimen width (mm)
With Strain Gauge	1.0	1.0	100	200	33
With Strain Gauge	1.0	1.0	120	200	33
With Strain Gauge	1.0	1.0	140	200	33
Without Strain Gauge	1.0	1.0	100	200	33
Without Strain Gauge	1.0	1.0	120	200	33
Without Strain Gauge	1.0	1.0	140	200	33

In this discussion, the test condition is separated into two test arrangements; i.e. with strain gauge and without strain gauge. The bending outputs for the three inner setting conditions are shown in figure 4.2. The test planning is shown in table 4.2.

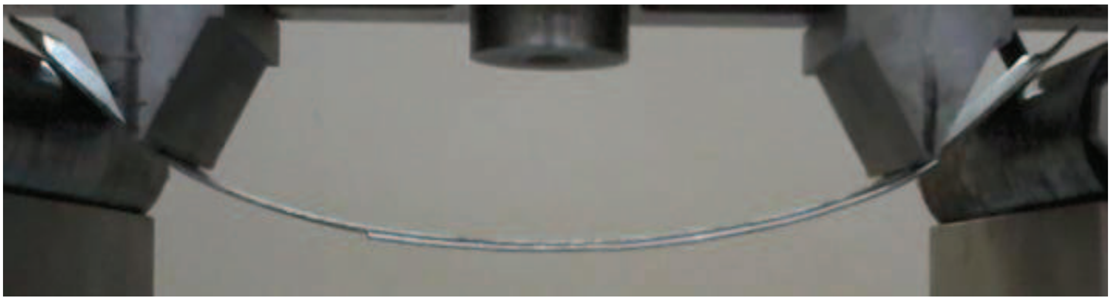
Table 4.2: The 4PBT test planning

Experimental	Displacement Inner Span (mm)		
	100	120	140
Strain Value			
Length displacement after bend value			



(a)

(b)



(c)

Figure 4.2: (a) inner span = 100mm, (b) inner span = 120mm, (c) inner span = 140mm, condition inner setting

As can be seen above, it shows how the bending configuration has an obvious effect on the stress evolution during the four-point bending process. It also has shown how the bending affects after change the inner span.

4.3.1 Different Inner Spans

For this section there are two cases or experiment shown as a result for this research. First case is to find strain value of four point bending test with three different inner spans. For second case is to find length specimen bend after bending test process. Strain value was taken in 1 second.

Case #1: Specimen was welded by strain gauge

One main purpose of this experimental work is to find the strain value in four-point bending test. Based on identical process output from the both analysis methods, reliability of the input parameters in the finite element model can be validated. Figure 4.3 and figure 4.4 below was shown condition before and after bending process.

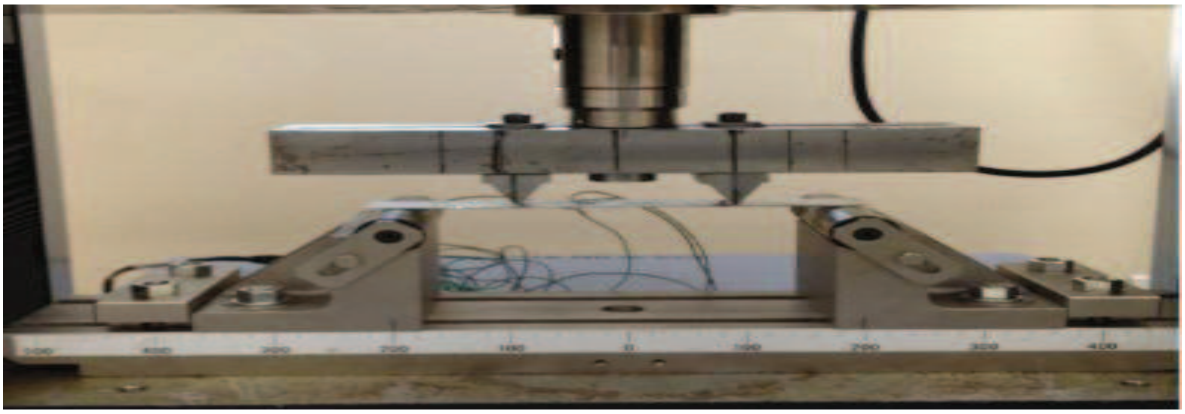


Figure 4.3: Condition specimen before bending process

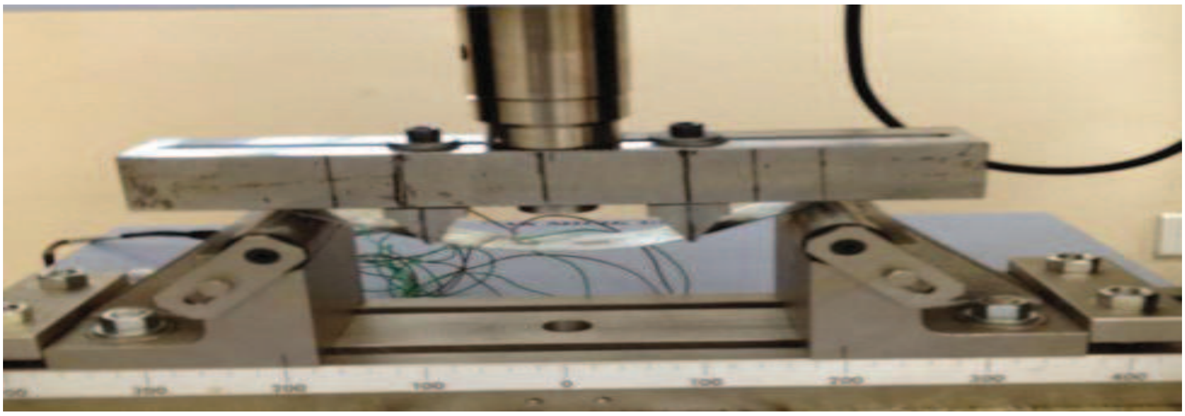


Figure 4.4: The bended workpiece from inner span 100mm.

Based on the four point bending test above, the time-strain graph can be plotted. The graph can be plotted by using the engineering strain. Figure 4.5 and Figure 4.6 below shown times-Strain graph for galvanized steel material having a thickness of 1.0mm with different inner span. The data of strain value was shown at the appendix C.

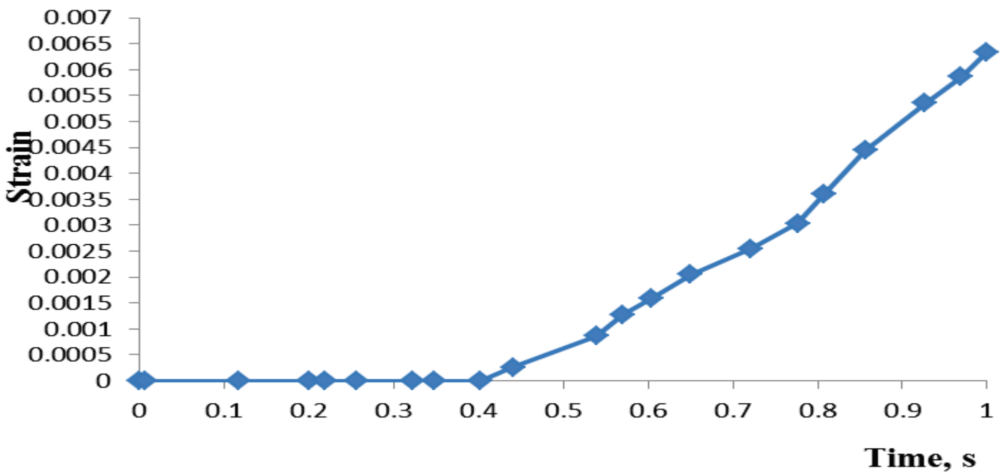


Figure 4.5: Graph times versus strain value for inner span 100mm

From the graph, time versus strain value graph above, the maximum strain for experiment inner span 100 mm is 0.006335. In addition, from time 0s to o.4s the strain value for this condition is zero. Then value of strain increase with time until the four-point bending test finish.

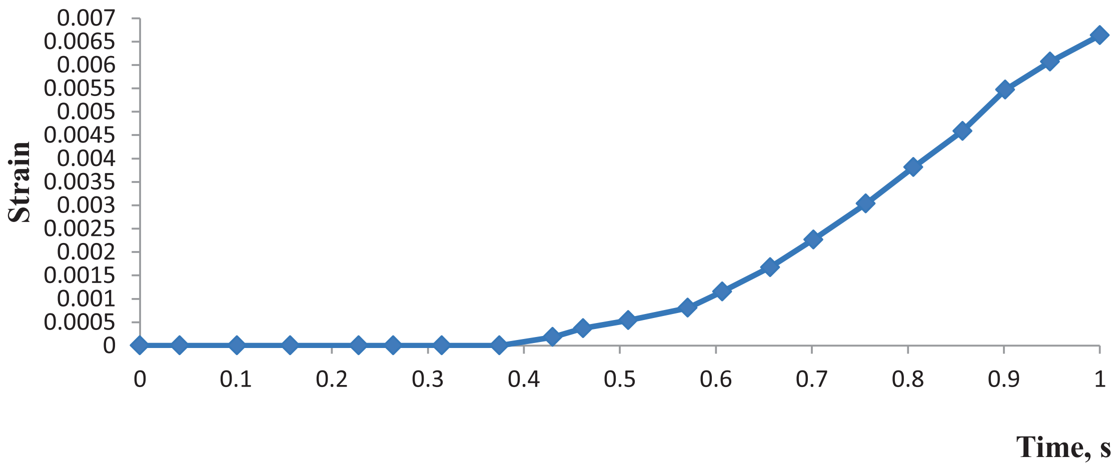


Figure 4.6: Graph times versus strain value for inner span 120mm

From the graph, time versus strain value graph above, the maximum strain for experiment inner span 100 mm is 0.006633. For this condition value of strain is zero from 0 to 0.37s.

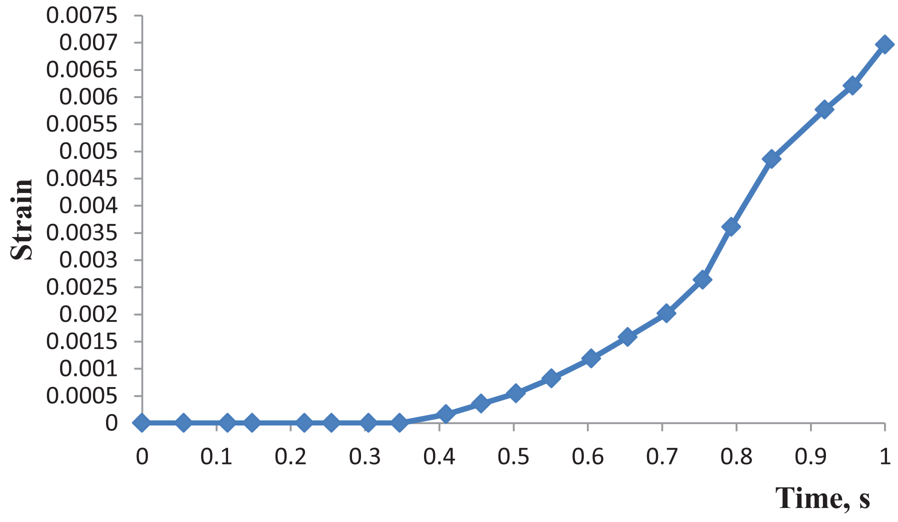


Figure 4.7: Graph times versus strain value for inner span 140mm

From the graph, time versus strain value graph above, the maximum strain for experiment inner span 100 mm is 0.0069626. For this condition value of strain is zero from 0 to 0.34s.

Case #2: Free four point bending test

The second case of this experimental work is to find length of displacement after bending test process. The basic arrangement is similar to first tests but different is specimens for this case not use strain gauge. This experimental result will be used as comparison to the finite element simulation output. Based on identical process output from the both analysis methods, reliability of the input parameters in the finite element model can be validated. Figure 4.8 and Figure 4.9 below was shown condition before and after bending process.

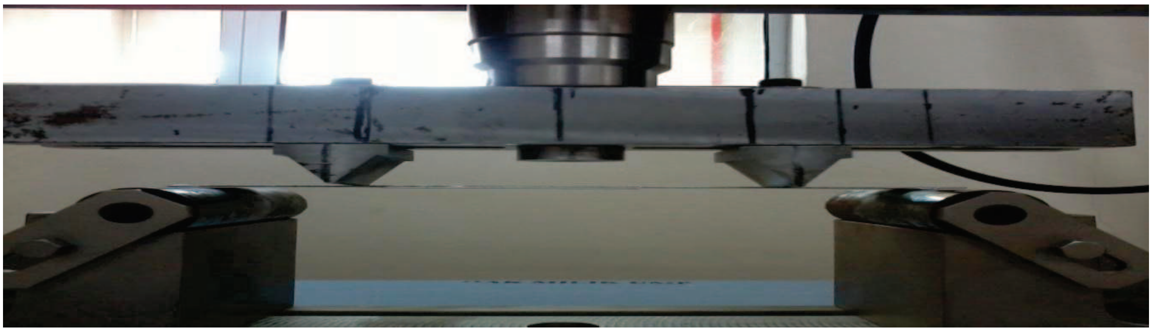


Figure 4.8: Condition specimen before bending process for free bend

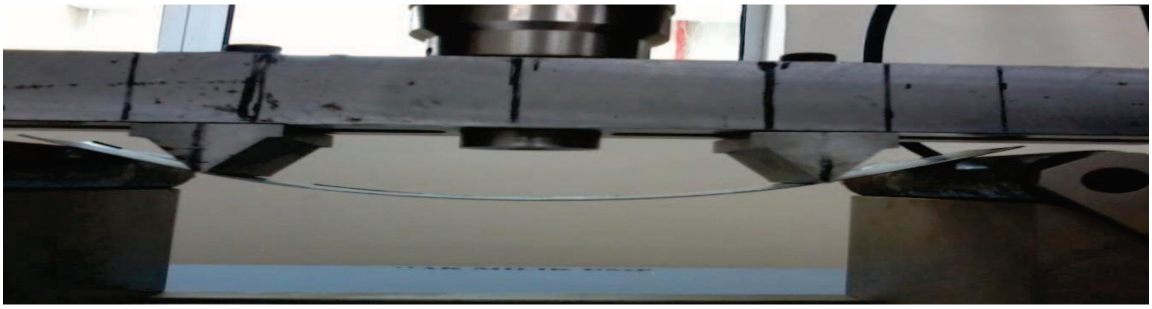
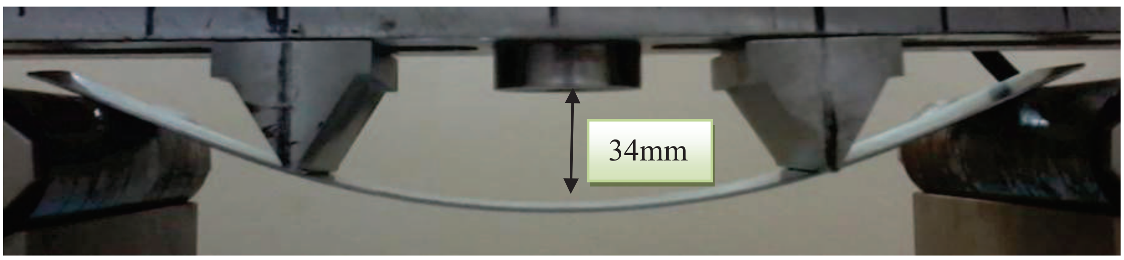
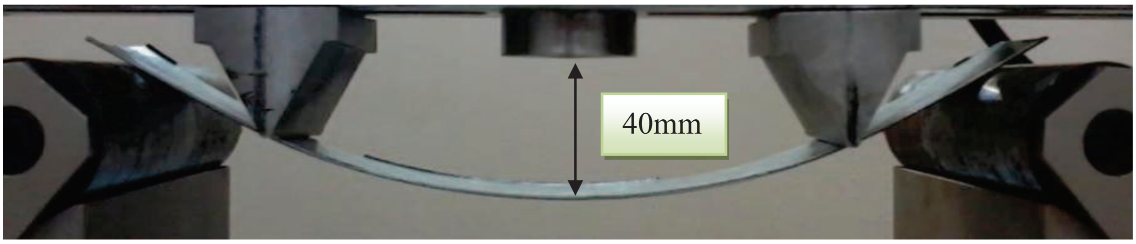


Figure 4.9: Condition specimen after bending process

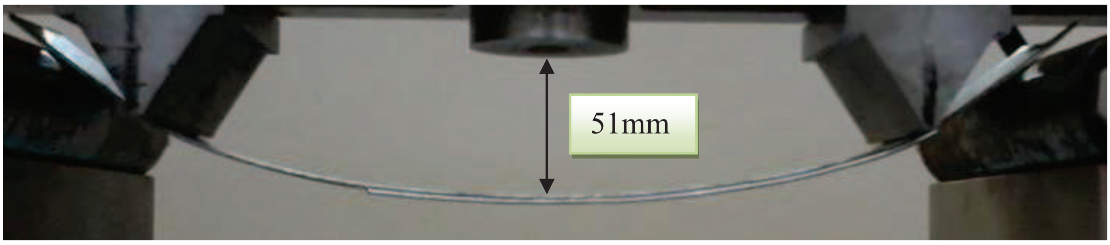
The four point bending test outputs for the three inner span setting conditions are shown in Figure 4.10.



(a) Inner span = 100mm



(b) Inner span = 120mm



(c) Inner span = 140mm

Figure 4.10: (a) inner span = 100mm, (b) inner span = 120mm, (c) inner span = 140mm, result experimental for free bending

As can be seen above, it shows how the bending configuration has an obvious effect on the stress evolution during the four-point bending process. The specimen with the biggest inner span 140 mm has maximum length.

4.4 FINITE ELEMENT MODEL FOUR POINT BENDING

The elastically driven change of shape in sheet metal has been simulated with ABAQUS code. Due to plane symmetry, only half of the process was modeled (Figure 4.11). The problem consists of the surface contact between steel blank strip and the tools such as the support and striker that is a basic aspect of the stamping operations. The tools can be modeled as rigid surfaces because they are much stiffer than the blank. Figure 4.11 shows the basic arrangement of the components considered in FEM model.

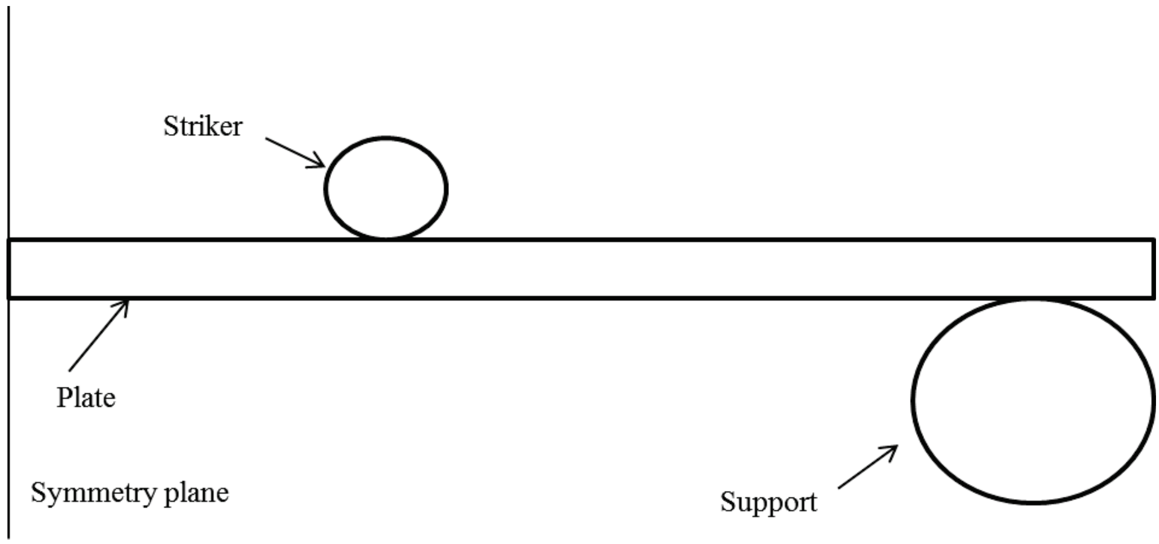
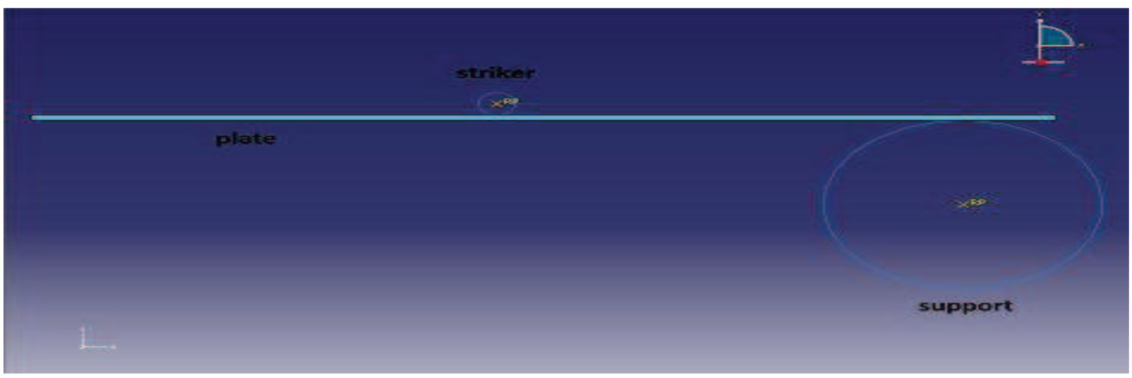
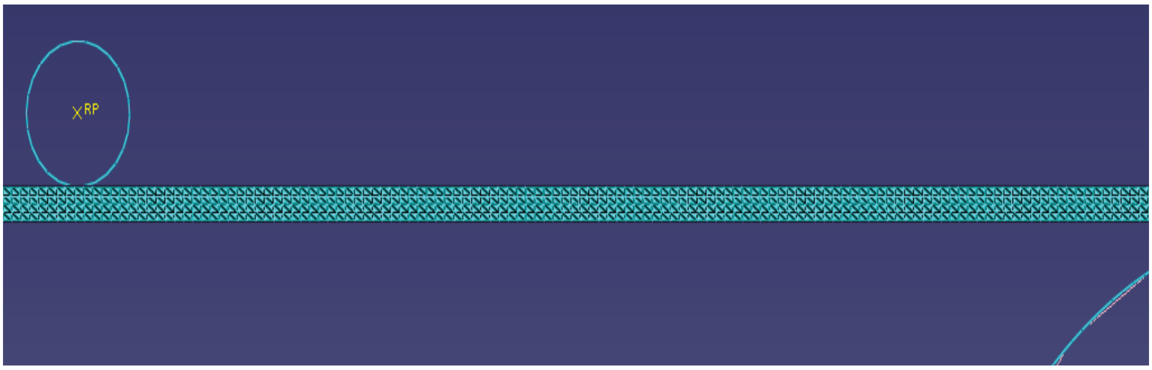


Figure 4.11: Geometrical description of the simulation model

The main objective of this analysis is to investigate the reliability of all the material properties and setting parameters of the studied system. From this task, it able to determine reliability of all the input parameters in the finite element model based on their good agreement with the experiment results. The model was developed to compare result simulation with experimental result. The simulation condition was setup at three different cases that based on the inner span length; i.e. 100 mm, 120mm and 140mm. The geometrical and meshes of the model are shown in the Figure 4.12 (a) and (b). The same configuration will be used for the analysis at static condition.



(a)



(b)

Figure 4.12: (a) The geometrical model for 4PBT (b) The meshed model for simulation

The present analysis involves implicit ABAQUS methods. The implicit solution algorithm applied for the plain strain logarithm. First step of this analysis concentrates on determination of the strain distribution inside the workpiece. The implicit method was adopted for easy to run the simulation. Since then, the plane strain method based implicit algorithm was the most suitable solution to generate data in the fastest time. In the plane strain analysis, continuum four point element (C4PE) has been adopted.

4.4.1 The Material Properties

In developing the finite element model, the workpiece was assumed as a continuum body, isotropic and homogeneous which represent the condition of the material properties do not vary with direction or orientation and identical properties at all points. The material specifications for the current analysis were obtained from the references and the experiment results. The full details of the material properties and their source of reference can be found in the Appendix E.

4.3.2 Boundary Condition

In addition the fundamental material characteristic, the other input parameters must be defined to complete the model. Among the important input parameters are an initial condition and interaction properties. The initial settings for the model were set based on suitability of the model in respect to the actual test conditions. Meanwhile, the interaction properties were mainly obtained from the related references from the other research studies. In the implicit analysis, different process setting was used to comply with their process requirements.

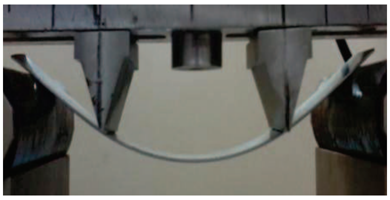
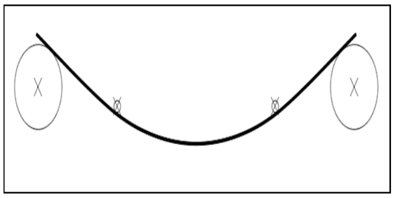
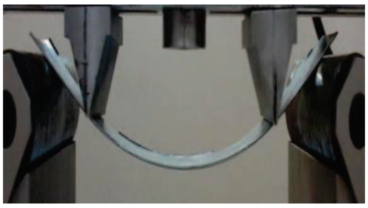
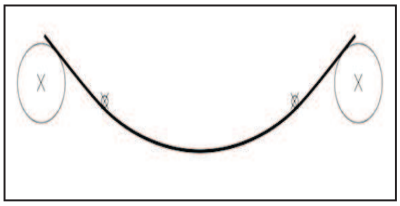
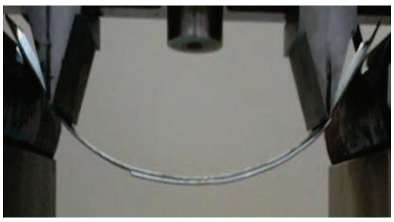
4.4.3 Process Step

Bending stage: The simulation has been carried under different inner span. The displacement of the rigid striker which has been used to push the workpiece downward metal plate were set equivalent to the inner span distance as indicated in the Table 4.1.

4.4.4 Test result

This section presents example the finite element results for the four-point bending test to compare with the experimental results. To precede the discussion on the bending outputs, it is differentiated of inner span. Result simulation for this research are shown in table 4.3

Table 4.3: Comparison finite element analysis with experimental

Test type / Test condition	Experiment	Simulation
Inner span =100 mm		
Inner span =120 mm		
Inner span = 140 mm		

4.4.5 Simulation of four point bending

For this topic shown how process simulation for four point bending test was doing. It can see in Figure 4.13 to Figure 4.15.

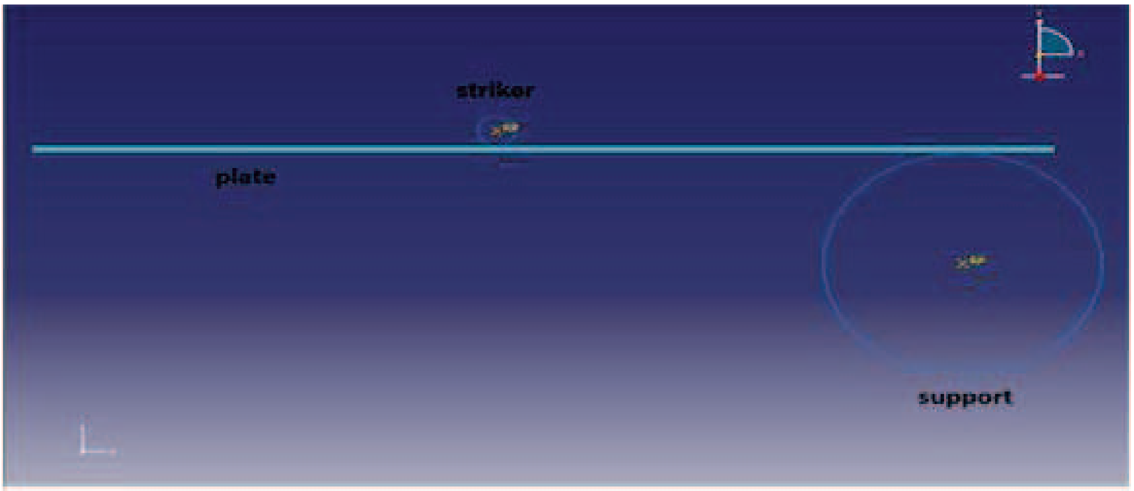


Figure 4.13: Striker start touching the workpiece

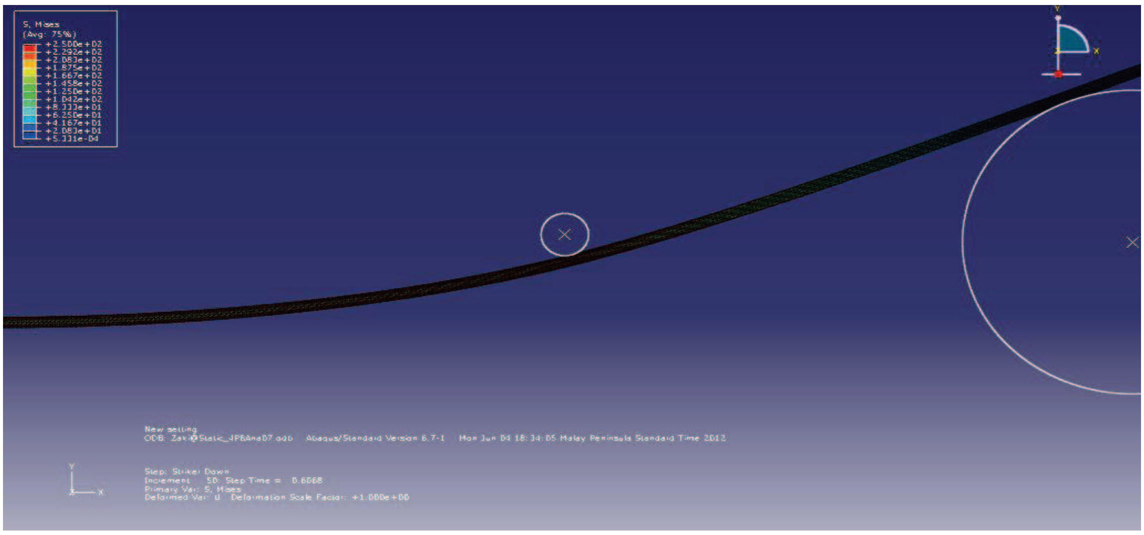


Figure 4.14: During bending process

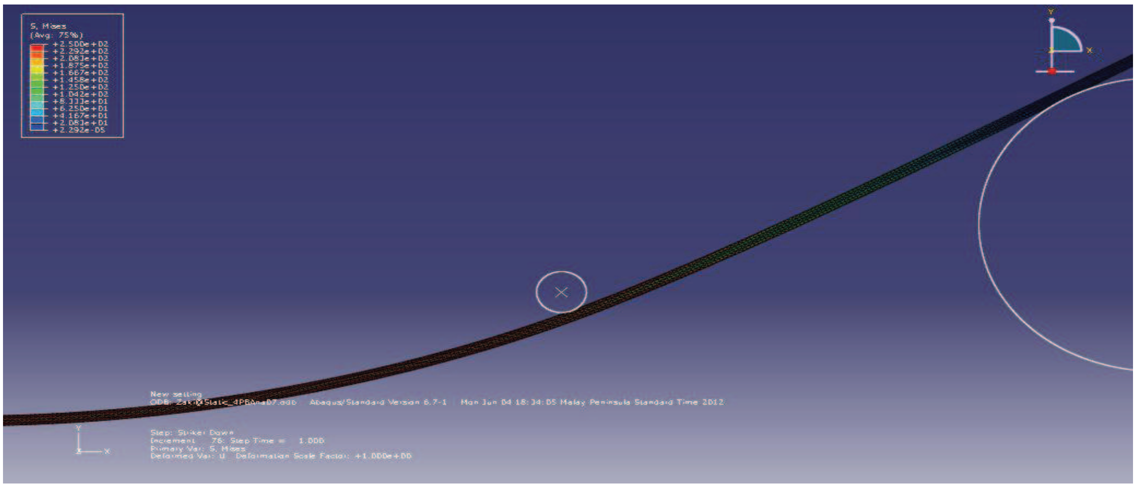
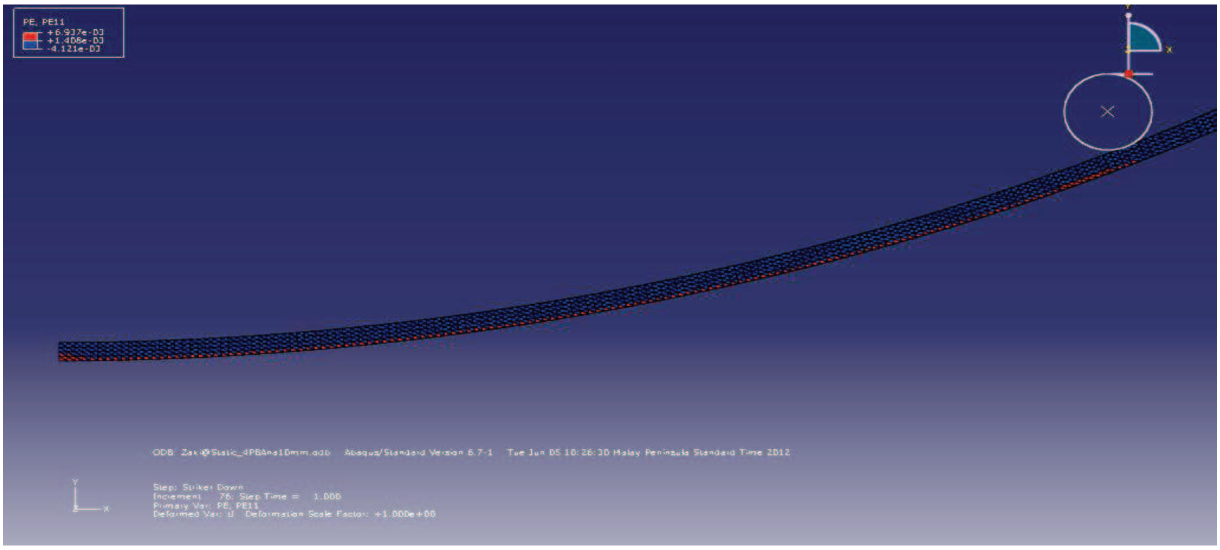


Figure 4.15: After bending process

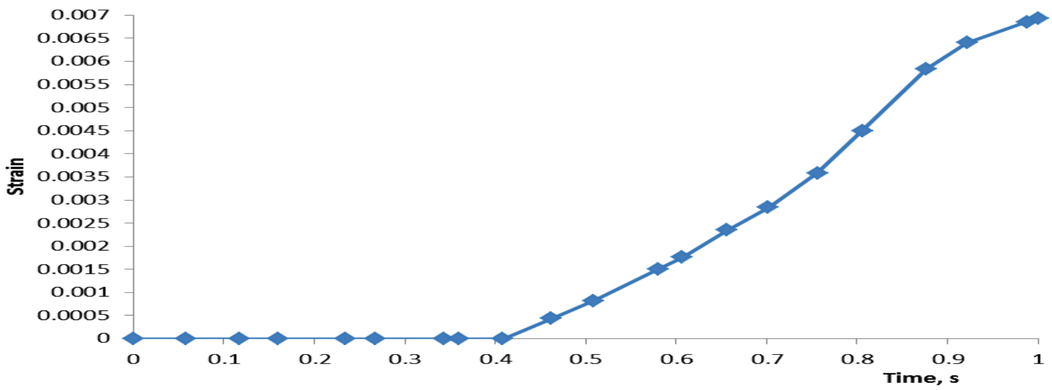
4.4.6 Result simulation

<u>Case #1: To find strain value</u>

The first discussion will be concentrated on final bending output to find value of strain. Based on the simulation four point bending test above, the time-strain graph can be plotted. Figure 4.16 to Figure 4.18 below shown times-Strain graph for galvanized steel material having a thickness of 1.0mm with different inner span. For this chapter also show condition strain of galvanized steel on simulation. The data of strain value was shown at the appendix D.



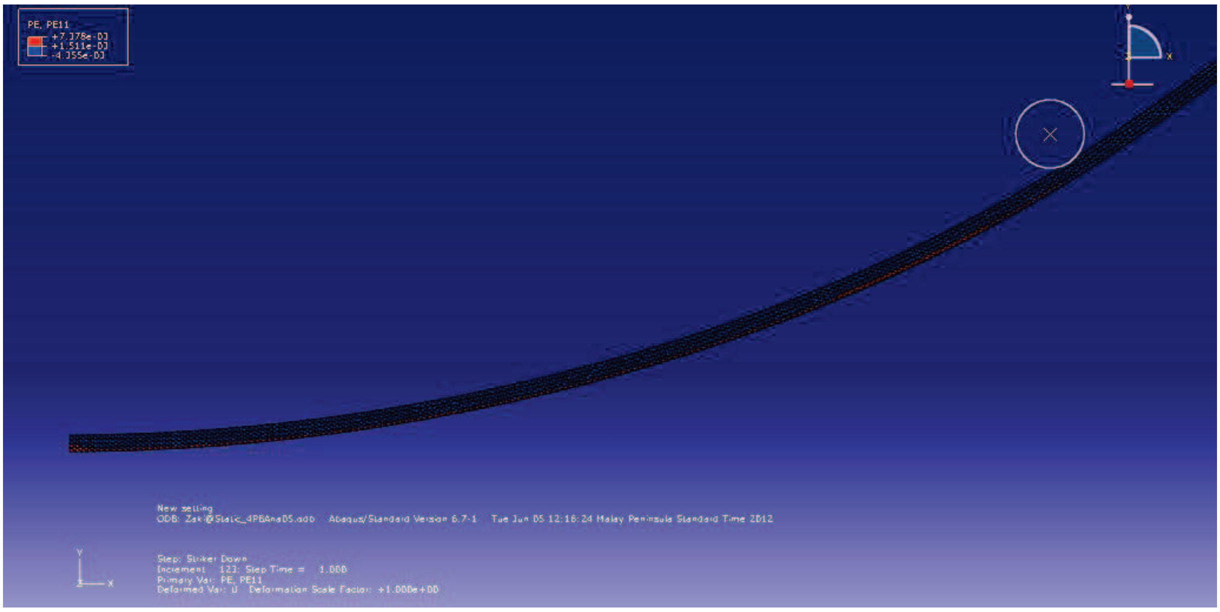
(a) Simulation inner span 100mm



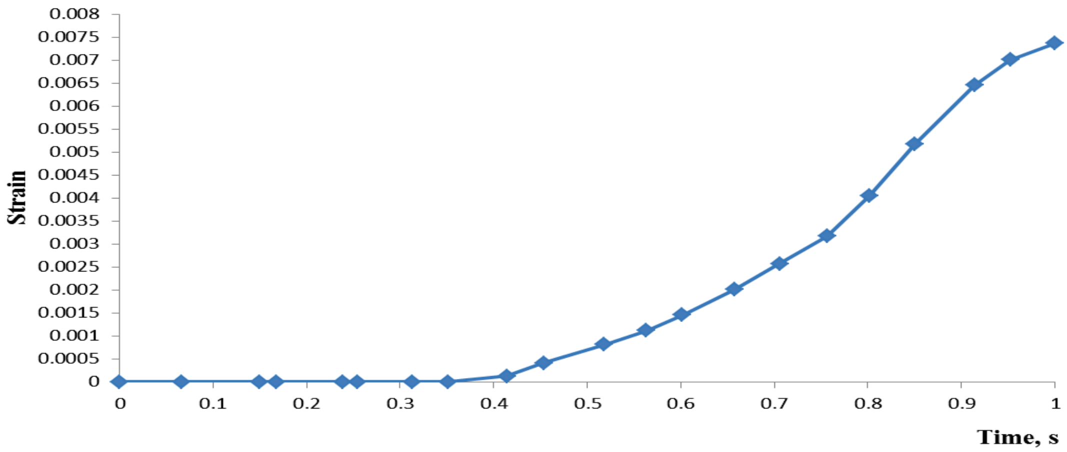
(b) Graph times versus strain value for inner span 100mm

Figure 4.16: Result simulation for inner span 100mm

From the graph, time versus strain value graph above, the maximum strain for experiment inner span 100 mm is 0.006937.



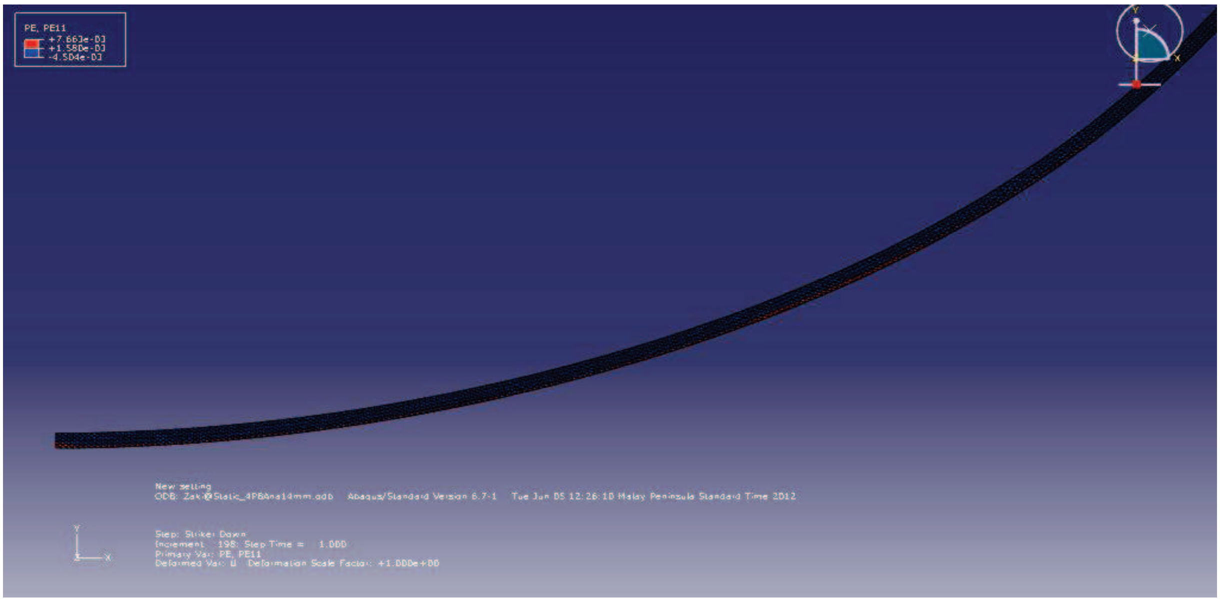
(a) Simulation for inner span 120mm



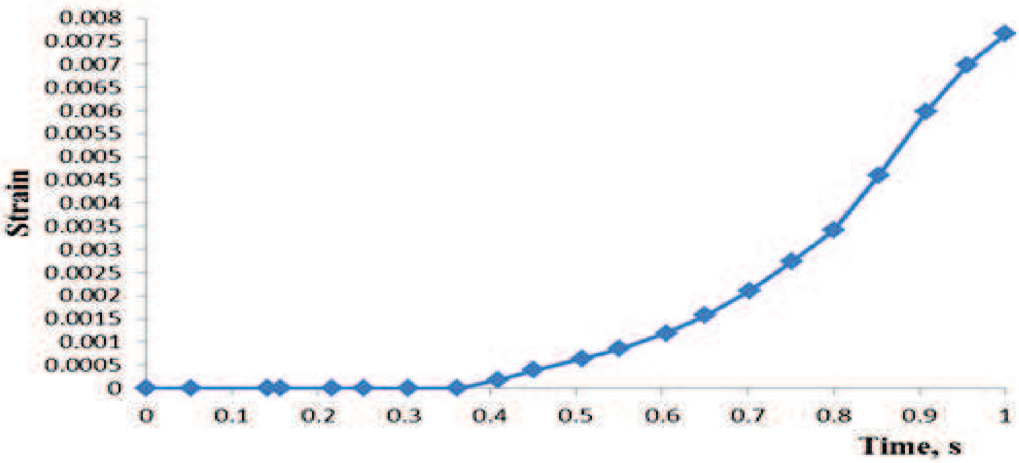
(b) Graph times versus strain value for inner span 120mm

Figure 4.17: Result simulation for inner span 120mm

From the graph, time versus strain value graph above, the maximum strain for experiment inner span 120 mm is 0.007366.



(a) Simulation for inner span 140mm



(b) Graph times versus strain value for inner span 140mm

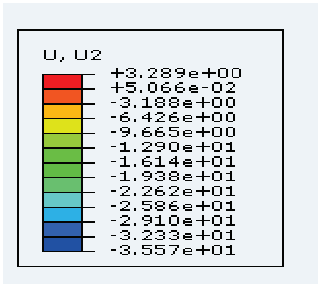
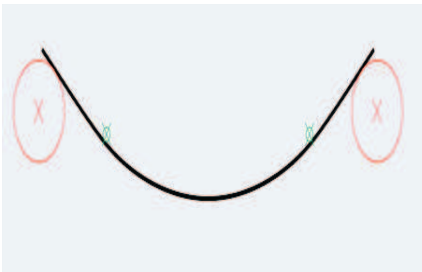
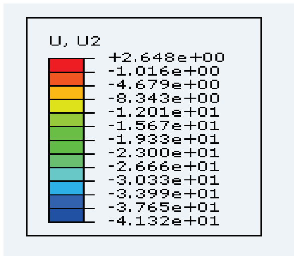
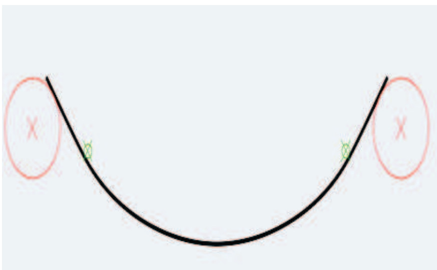
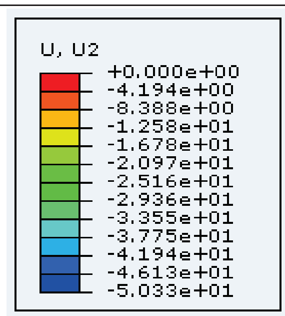
Figure 4.18: Result simulation for inner span 140mm

From the graph, time versus strain value graph above, the maximum strain for experiment inner span 140 mm is 0.007663.

Case #2: find length displacement after bend

The second case will be concentrated on final bending output to find value of length of displacement after bending process. Based on the simulation four point bending test above, value of length will get. Condition specimen after simulation and length value for galvanized steel material with thickness of 1.0mm was shown in Table 4.3.

Table 4.3: Result simulation

Test condition	Simulation	Length value
Inner span =100 mm		
Inner span =120 mm		
Inner span = 140 mm		

4.5 COMPARISON OF FINITE ELEMENT SIMULATION AND EXPERIMENTAL

Result of value of strain for different inner span for both FE Simulation and experimental was shown in Table 4.4. Graph in figure 4.19 was shown the comparison of strain value between experimental and simulation. Result of value of length displacement in simulation was compared with result for experimental and shown in Figure 4.20.

Table 4.4: Comparison strain value experimental and finite element simulation

Inner Span	100mm	120mm	140mm
Experimental	0.006335	0.006633	0.006962
FE Simulation	0.006937	0.007366	0.00766
Percentage of error POE (%)	8.68%	9.95%	9.11%

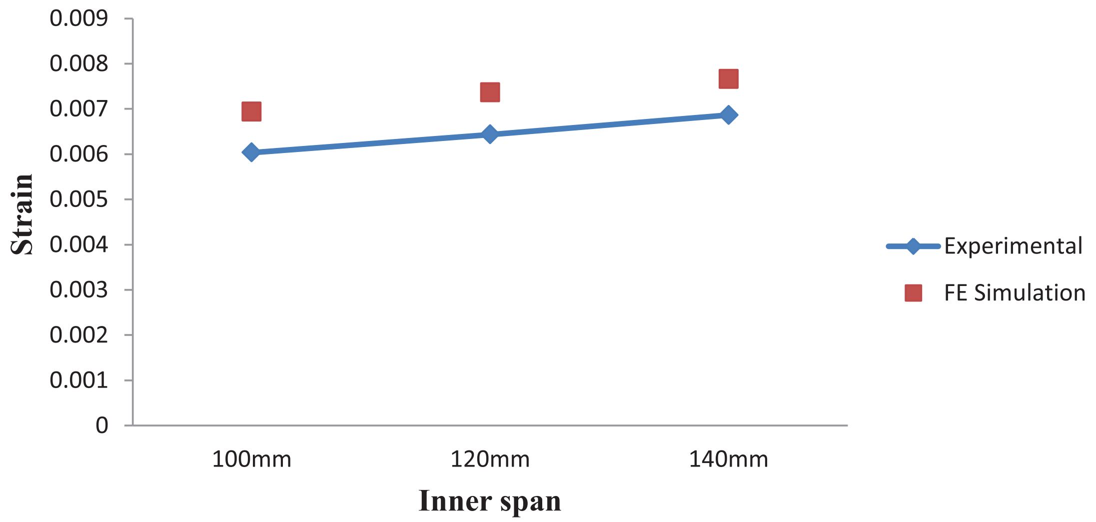
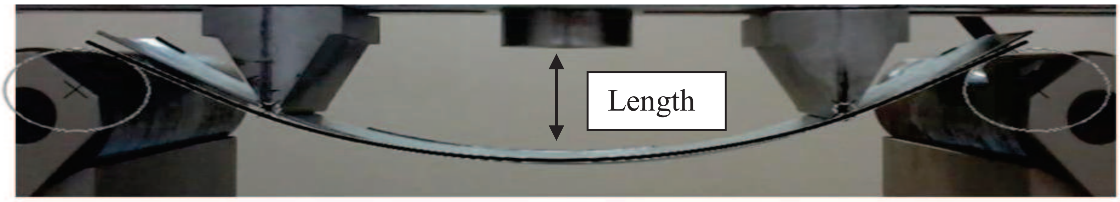
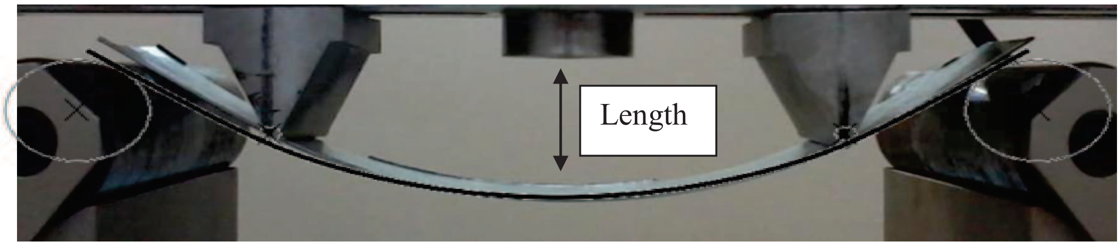


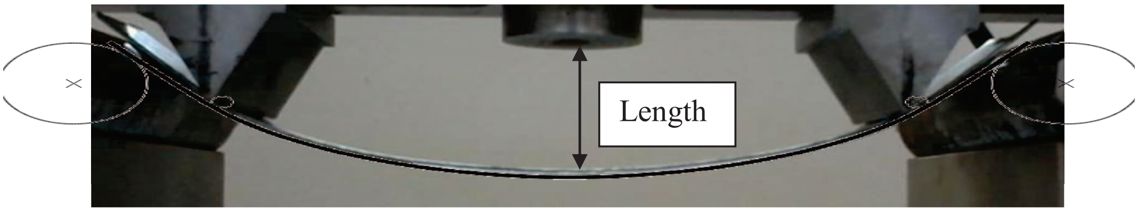
Figure 4.19: Graph comparison strain value experimental and FE simulation



(a) Inner Span = 100mm



(b) Inner Span = 120mm



(c) Inner Span = 140mm

Figure 20: Comparison length displacement after bend of experimental and FE simulation for inner span 100mm, 120mm and 140mm.

4.6 DISCUSSION

For this subtopic, it was review about discussion comparison of simulation with experimental**.** According to the objectives, this research only to do comparison analysis FE simulation with experimental four point bending test.

Figure 4.20 shows the experimental and simulation effect of strain value on the different inner span of the galvanized steel for 1mm thickness. Comparing the result of the strain value for different inner span, it is noted that inner span affected the strain value. From strain value data from the simulation, by increasing the inner span, will increase the strain value. It is same with result experimental where strain value also increase when inner span increase.

From the figure 4.20, result from experimental has the lowest values than result from simulation. From FEA, values have higher values mainly because FEA analysis does not included safety factor in the calculation. For experimental, safety factor were included in their calculation. The lowest value in experimental that means that higher safety factor was included in the calculation.

Comparing the result of length displacement after bending process for different inner span, it is noted that inner span was effected the length. From the simulation data, by increasing the inner span, the length displacement will increase. It also same with result from experiment where length displacement after bend process increase due to inner span increase. What can conclude from this figure is value of length displacement after bend from result experiment and simulation data is almost the same.

CHAPTER 5

CONCLUSION

5.1 INTRODUCTION

Generally this chapter concludes the study research. Besides that, the objective is also reviewed in this chapter to determine if it achieved or not. The contribution of this study, the limitation are also been discussed in this chapter.

5.2 CONCLUSION

Based on the study, the following remarks are drawn :

1. Inner span are strongly affected the amount of strain value where increasing the inner span will increase the strain value.
2. Inner span was also affected by the length after bend which increases the inner span.
3. Finite Element Analysis can be used to do comparison with experimental. It because result of the graphs compared are nearly the same and the percentage of errors, (POE) are less than 10 %.

5.3 RECOMMENDATIONS

For the improvement of the study, there are several matters can be done:

(i) Using a variety of materials in the experiment and simulation such as Aluminum, Stainless Steel and so on to investigate which material that have a higher strain value.

(ii) Using different thickness for every material to investigate the effect of strain value with fixed striker.

(iii) Using a variety of method of experiment and simulation such as using different temperature to test the material.

(iv) Study the meshing effect to predict the strain value by using a different mesh in the simulation and choose the result that has a nearest value with the experiment.

REFERENCES

P.F. Zhao et al. (2010). “Fracture toughness measurements of plasma-sprayed thermal barrier coatings using a modified four-point bending method”, Surface & Coatings Technology, 204: 4066–4074

T.M. Chan et al., (2008), “*Bending strength of hot-rolled elliptical hollow sections*”, Journal of Constructional Steel Research, 64: 971–986

F.Mujika , (2006), “On the difference between flexural moduli obtained by three-point and four-point bending tests” , Polymer Testing ,25:214–220

L. Zheng et al. (2005), “Fracture prediction in 4-point bending of an extruded aluminum panel”, Thin-Walled Structures , 43: 565–590

Pizhong Qiao et al. (2011), “Fracture characterization of Carbon fiber-reinforced polymer-concrete bonded interfaces under four-point bending”, Engineering Fracture Mechanics 78 :1247–1263

Y. Wang et al. (2006), “*Four-point bending fatigue behavior of an iron-based laser sintered material*”, International Journal of Fatigue 28 :1705–1715

Sinval A. Rodrigues Junior et al. (2008), “Flexural strength and Weibull analysis of a microhybrid and a nanofill composite evaluated by 3- and 4-point bending tests” , dental materials 2 4: 426–431

L. Lagunegrand et al. (2006),. ”Design of an improved four point bending test on a sandwich beam for free edge delamination studies” Composites: Part B 37 :127–136

T.Zhai et al.(1999), "*A self-aligning four-point bend testing rig and sample geometry effect in four-point bend fatigue*" International journal of fatigue of fatigue 21 (1999); 889-894

Afred Scholz et al. (2008), "*Experiences in the determination of TMF, LCF and creep life of CMSX-4 in four-point bending experiments*" International journal of fatigue of fatigue 30 (2008), 357-362

D.Sturm et al. (1986) " The behaviour oof dynamically loaded pipes wth circumferential flaws under internal pressure and external bending loads" Nuclear Engineering and Design 96: 99-133

A.B. de Morais et,al (2009) "Mode III interlaminar fracture of carbon/epoxy laminates using a four-point bending plate test" Composites: Part A 40 (2009); 1741–1746

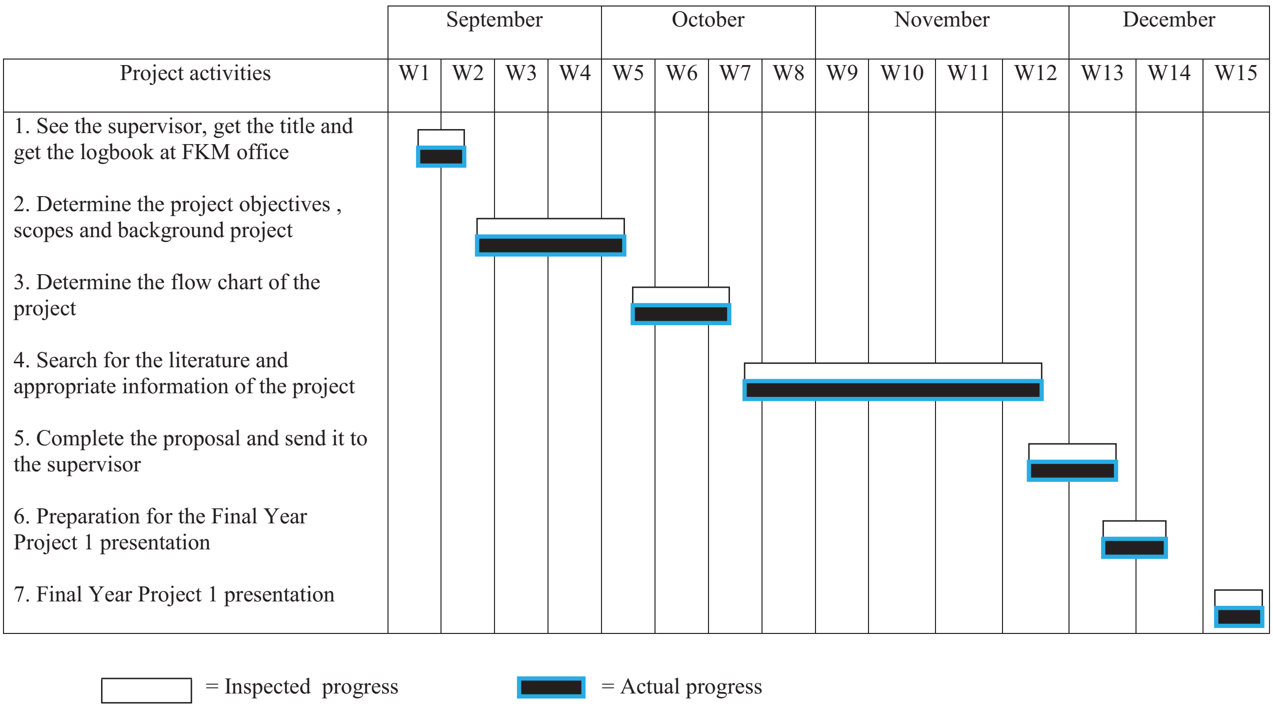
A.Kucuk et al (2000). "Influence of plasma spray parameters on mechanical properties of yttria stabilized zirconia coatings. I: Four-point bend test" Materials Science and Engineering A284; 29–4

ASTM D790M-93,'Standard test methods for flexural properties of unreinforced and reinforced plastics and electrical insulating materials', American Society for Testing and Materials, Annual Book of ASTM Standards,Vol. 08.01, 1993.

P t curtis (ed), crag test methods for the measurement of the engineering properties of fibre-reinforced plastics, royal aircraft establishment, technical report 88012, february 1988.

APPENDIX A

GANTT CHART FOR FINAL YEAR PROJECT 1



APPENDIX B

GANTT CHART FOR FINAL YEAR PROJECT 2