UNIVERSITI MALAYSIA PAHANG

BORANG PENGESAHAN STATUS TESIS♦

JUDUL: **EXPERIMENTAL AND FINITE ELEMENT EVALUATION OF BENDING FOR ALUMINIUM**

SESI PENGAJIAN: 2011/2012

Saya **MUHAMMAD HAFIZ BIN AZAM (MA08023)**
(HURUF BESAR)

Mengaku membenarkan tesis (SarjanaMuda/~~Sarjana~~ /~~DoktorFalsafah~~)* ini disimpan di Perpustakaan dengan syarat-syarat kegunaan seperti berikut:

1. Tesis adalah hak milik Universiti Malaysia Pahang (UMP).
2. Perpustakaan dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Silatandakan (√)

[ ]	**SULIT**	(Mengandungimaklumat yang berdarjahkeselamatan ataukepentinganMalaysiaseperti yang termaktub didalam AKTA RAHSIA RASMI 1972)
[ ]	**TERHAD**	(Mengandungimaklumat TERHAD yang telahditentukan olehorganisasi/badan di manapenyelidikandijalankan)
√	**TIDAK TERHAD**	

Disahkanoleh:

(TANDATANGAN PENULIS) (TANDATANGAN PENYELIA)

AlamatTetap:

LOT 1306, LORONG SENANGGIN
KAMPUNG SAUJANA,
22110, SETIU, TERENGGANU

JASRI BIN MOHAMAD
(NamaPenyelia)

Tarikh**: 08 JUNE 2012** Tarikh: **08 JUNE 2012**

CATATAN:
* Potong yang tidak berkenaan.
** Jika tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali tempoh tesisini perlu dikelaskan sebagai atau TERHAD.
♦ Tesis dimaksudkan sebagai tesis bagi Ijazah doctor Falsafah dan Sarjana secara Penyelidikan, atau diserta sebgai pengajian secara kerja kursus dan penyelidikan, atau Laporan Projek Sarjana Muda (PSM).

UNIVERSITI MALAYSIA PAHANG

FACULTY OF MECHANICAL ENGINEERING

I certify that the project entitled "Experimental and Finite Element Evaluation of Bending for Aluminium" is written by Muhammad Hafiz Bin Azam. We 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. We were here with recommend that it be accepted in partial fulfillment of the requirements for the degree of Bachelor of Mechanical Engineering.

(MOHD FADHLAN BIN MOHD YUSOF)

EXAMINER SIGNATURE

EXPERIMENTAL AND FINITE ELEMENT EVALUATION OF BENDING FOR ALUMINIUM

MUHAMMAD HAFIZ BIN AZAM

Thesis submitted in fulfillment of the requirements
for the awards of the degree of
Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

JUNE 2012

SUPERVISOR'S DECLARATION

I hereby declare that I have checked this project proposal and in my opinion, this project proposal is adequate in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering.

Signature:
Name of Supervisor: EN. JASRI BIN MOHAMAD
Position: Pensyarah, Universiti Malaysia Pahang
Date:

STUDENT'S DECLARATION

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

Signature:
Name: MUHAMMAD HAFIZ BIN AZAM
ID Number: MA08023
Date:

Dedicated to my father, Azam Bin Omar,

my beloved mother, Normah Binti Abd. Ghani,

my brothers, my sister , my love one

and last but not list to all my fellow friends.

ACKNOWLEDGEMENTS

First of all, I want to thank The Almighty Allah SWT for the beautiful life that has been given to me in the past 22 years and the present. I am very thankful to be given the time and chances to finally complete this thesis. I am grateful and would like to express my sincere gratitude to my supervisor En. Jasri Bin Mohamad for his ideas, invaluable guidance, continuous encouragement and constant support in making this thesis possible. He has always impressed me with his outstanding professional conduct, his strong conviction for science, and his belief that a Bachelor program is only a start of a life-long learning experience. I appreciate his consistent support from the first day I applied to PSM course to these concluding moments. I am truly grateful for his progressive vision about my work progressing, his tolerance of my naive mistakes, and his commitment to my future career. I also sincerely thanks for the time spent proofreading and correcting my many mistakes.

I also would like to express my gratitude to the Faculty of Mechanical Engineering and Universiti Malaysia Pahang, for their assistance in supplying the relevant literatures. I am also obliged to express my appreciation towards my beloved mom and dad and also my family members for their enduring patience, moral and financial supports. My fellow friends should also be recognized for their support. My sincere appreciation also extends to all my colleagues and others who have provided assistance at various occasions. Their views and tips are useful indeed. Unfortunately, it is not possible to list all of them in this limited space. Thank you to all. Thank you for everything. God bless all of you.

ABSTRACT

This thesis studies the effect of anisotropy to the deformation of springback in sheet metal forming and reliability of finite element method in sheet metal bending of Aluminium 1100 with the thickness 1 mm. This thesis focuses on tensile test experiment, bending experiment and finite element experiment. Tensile test experiment was done to determine the material properties of aluminium to put in the finite element simulation. The bending process was done using U-die to define the angle of springback in aluminium. The finite element simulation was done by using ABAQUS 6.7 to determine the angle of springback. The results of the experiments were analysed in order to determine the reliability of finite Element. The results show springback increased when the angle of rolling direction is higher. The lowest angle of springback was occurs at the 0° of rolling direction. Based on project, it was concluded that the parameters of anisotropy affect springback angle. It was found the 0° provides a suitable condition in U-die bending process. The finding from this project is the finite element method might be use in order to predict the springback angle of die designing.

ABSTRAK

Tesis ini mengemukakan kajian mengenai kesan anisotropi terhadap bahan yang berubah daripada bentuk yang mengikut acuan kepada satu bentuk yang berlainan dalam pembentukan lembaran logam.selain itu, kebolehpercayaan simulasi dalam analisis pembengkokan kepingan logam Aluminium 1100 dengan tebal 1mm turut dikemukakan. Dalam kajian ini tumpuan telah diberikan terhadap beberapa eksperimen seperti eksperimen ujian tegangan, eksperimen lentur dan eksperimen simulasi. Ujian tegangan telah dilakukan untuk menentukan sifat bahan aluminium untuk dijadikan input kepada ujian simulasi. Proses lentur telah dilakukan dengan menggunakan acuan berbentuk U untuk menentukan sudut membidas dalam aluminium. Simulasi telah dilakukan dengan menggunakan perisian ABAQUS 6.7 untuk meramalkan sudut membidas. Hasil keputusan eksperimen dianalisis untuk menentukan kebolehpercayaan simulasi. Hasil keputusan menunjukkan sudut membidas itu bertambah apabila sudut arah pinda adalah lebih tinggi. Sudut Membidas terendah telah berlaku di 0° arah mampatan. hasil daripada tesis ini adalah parameter anisotropi mempengaruhi sudut membidas. Diketahui bahawa 0 ° kepada arahan mampatan adalah terbaik dalam menyediakan suatu keadaan yang sesuai dalam acuan berbentuk U. hasil daripada kajian ini menunjukkan proses simulasi boleh digunakan untuk menjangka perubahan sudut didalam proses membuat acuan.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

r	Strain Ratio
A_0	Cross Sectional Area
L_0	Original Gage Length
σ	Yield Stress
υ	Poisson Ratio
ε	Plastic Strain

LIST OF ABBREVIATIONS

ASTM	American Society for Testing and Materials
FEM	Finite Element Method
UTS	Ultimate Tensile Strength
BHF	Blank Holder Force
POE	Percentage of Error

CHAPTER 1

INTRODUCTION

1.1 Introduction

This chapter discusses some of the overview flow of the entire projects that consists of project background, problem statement, objectives, scope of the works and organization of the thesis.

Nowadays, Finite Element Method (FEM) has been used extensively in sheet metal bending analysis. However, the reliability to use this method depends on the input data. This method is getting applicable and useful from time to time. Thus there is a need to evaluate of the data by comparing the results from simulation against experimental results. By mastering in this method will make the industry especially manufacturing industry know about the properties of the element, the condition of materials and appropriate bending process for such material in order to manage the company.

Tensile test has been used in this project to determine the properties of the material include tensile strength σ_{ts}, yield strength,σ_y and Young's Modulus of Elasticity, E. Then, the data from the tensile test are used in the finite element method for si mulationprocess. The results from the tensile test experimental was needed because the exact value will make this project working smoothly and getting the successfully results.

After that, the experimental of U-bending will be conducted using press brake machine. Nine specimens are use; three specimen for each orientation 0º, 45º and 90° in this experimental. Specimens from the U-bending were measured by using scanner to get the shape of the final specimens and then the springback were measured using solidworks. The result from the experimental will be compared to the result of simulation.

1.2 Problem Statement

One of the problems in the sheet metal industry is to design die that will produce final part exactly same as the shape of die. Usually the final parts of product are different with its die because of the material behavior. This phenomenon is called springback.. Several solutions have been proposed to predict springback angle. Using finite element is one of the solutions that provided. However, the reliability of finite element needs to justify in order proving it.

1.3 Project Objectives

The objectives of study in this project are:

i. To determine the effect of anisotropy to the deformation of springback in sheet metal forming.
ii. To determine the springback measurement of aluminium.
iii. To determine reliability of finite element method in sheet metal bending analysis of aluminum by comparing with experimental results.

1.4 Scope of Works

In order to finish this project successfully, flow of project have focus on some of scope of works to achieve objective the project. Basically, these thesis scopes of works are:

i. Conduct the literature review for overview of previous works
ii. Conduct the tensile test experiment that determines the mechanical properties of Aluminium.

iii. Conduct U-Bending experiment in order to determine the springback angle.
iv. Conduct FE simulation for predicting springback angle of aluminium.
v. Compare FE results with experiment results for reliability evaluation of FE.

1.5 Thesis Organisation

This thesis will provide five chapters:

Chapter 1 has provided clearly about overview of the entire project include introduction, problem statement, objectives, scope of the works and organization of the thesis.

Chapter 2 has presented the literature review based on journals that have related to this project. The background of the system including the material preparation, material properties, anisotropy parameter, tensile test, bending test, finite element method and measuring of springback from the previous journals will be discuss.

Chapter 3 has discussed of methodology that use in this project based on chapter 2. It includes a flow chart of this thesis, such as tensile test experiment to define the material properties, finite element method to predicting the angle of springback and u- die bending process as comparison for validity of FE in predicting springback angle.

Chapter 4 has discussed the results of tensile test, finite element method and U-bending experiment. The result from finite element method was compared with U-bending result in term of springback angle.

Chapter 5 has concluded this thesis and has answered the objective have been listed in chapter 1. Besides that, the outcomes from this project are listed.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

This chapter provides limited literature review on previous research and it to be as guideline to run this project. Some of previous literature review discussion about the material, U-bending, numerical method, and tensile test. There are many previous researcher works has been done in order to reduce spring-back value of sheet metal in U-bending process. The results have been compared between simulation of finite element and experiment. The source of this chapter is from journal, articles and some from books.

According to M.Kadkhodayan (2008), one of the most widely used sheet metal forming processes is bending, which is employed in industry. Bending is a process by which metal has deformed by plastically deforming the material and changing its shape following it die. The surface area of the material does not change much but it will affect the quality of the final parts. One of the problems in bending process is to predict the shape of the metal after loads have been removed. This phenomenon is called as springback. Gau J. A (1999) has defined the springback are the change in the shape of a sheet metal part upon the removal of stamping tooling. According to M.Kadkhodayan (2008) the springback is normally measured by determined the change in radius of curvature due to elastic recovery. It was influenced by a combination of various process parameters such as tool shape and dimension, contact friction condition, material properties, and thickness. M.Firat (2009) has give that the important of predicting the amount of springback deformations in sheet metal parts comes from the die-face design and process planning. However, the produced parts do not conform to the design geometry within the required dimensional tolerances right at the first time. According to Micari F (1997) works, his presented that the sheet metal slides over a die shoulder, it undergoes bending– unbending deformations developing cyclic bending loads on the sheet sections, and as a result an unbalanced stress distribution is developed over the thickness. Bending experiment was done using press brake machine. Press Brake normally has a capacity of 20 to 200 tons to accommodate stock from 1m to 4.5m (3 feet to 15 feet). Larger and smaller presses are used for specialized applications. Programmable back gages, and multiple die sets available currently can make for a very economical process based on Boljanovic (2004). Figure 2.1 show the press brake machine that use in bending experiment.

Figure 2.1: Press Machine for U-Bending Experiment

Source: Wang Hao *et. al* (2010)

2.2 Material

In engineering application, material selection is important term to make sure that material is suitable for the produce products. Before the material are chosen, there are many experiment and procedure that must be follow to avoid the unexpected cases happen after publishes. In this study, the Aluminium sheet metal has been selected as a material to perform the experimental and finite element simulation for bending test.

2.2.1 Material Preparation

Material characteristic of the sheet metal was needed as an input in the simulation finite element. Generally, to characterize the material properties and anisotropy, the different orientation angle of specimen sheet metal are provided in order to define effect of anisotropy to springback angle. From the author M. Bakhshi-Jooybari (2009), his worked has used different specimens of 25x100 mm CK67 steel sheets which were in different orientation of the rolling directions (0º, 45º and 90º), and different thickness of 0.5, 0.7, and 1 mm. All those specimens were examined with bending test. Tensile test were used to determine the stress-strain curves and the sheet metal anisotropy parameters, r-values. In another work, V.Esat (2001) had used Aluminium sheets with a length of 400 mm, a width of 300 mm, and thickness of 3, 4, and 5 mm to study the 45º angular bending. Wipe bending was analysed using aluminium sheets with a length of 250 mm, a width of 300 mm, and thickness of 3 and 4 mm. In this project the sheet metal are use is Aluminum with length of 150mm and wide 50 mm and the thickness of 1mm were analysed with different orientations to rolling direction (0º, 45º and 90º).

2.2.2 Material Properties

According to Aljibori (2009), there is two types of isotropic material involve in his model. That's type are linear elastic-plastic (for sheet metal) and linear elastic (for the punch and die). The materials properties for each type are shown in table 2.1.

Table 2.1: Materials Properties Aluminium

Sheet Metal (Aluminium)		
Linear Elastic-Plastic		
Material Properties	Symbol	Properties Value
Young's Modulus	E	78 GPa
Uniaxial yield strength	σ_y	550 MPa
Poisson Ratio	N	0.3
Material Density	P	2700 kg/m^3
Punch and Die (Steel)		
Linear Elastic		
Young's Modulus	E	400 GPa
Uniaxial yield strength	σ_y	4000GPa
Poisson Ratio	v	0.3
Material Density	ρ	7800 kg/m^3

Source: Hakim S.Sultan Aljibori *et. al.* (2009)

2.2.3 Sheet Anisotropic

Tensile test were used to determine the stress-strain curves and the sheet metal anisotropy parameters, r-values. M. Bakhshi-Jooybari (2009) said that the plastic properties of rolled sheets differ from the through thickness direction, normal anisotropy and vary with orientation in the plane of the sheet, planar anisotropy. M.Kadkhodayan (2008), was investigated the effect of sheet anisotropy on the springback of stainless steel 410 draw-bend specimens experimentally as well as through finite element simulations. Moreover, they studied the influence of blank holder force and coefficient friction on the amount of the final springback. Vial.C (1983) had stated that the anisotropy of sheet metal deformations is represented with the initial (texture-induced) anisotropy due to the sheet production process and the plastic anisotropy developed during the course of the elastic–plastic deformations.

Based on previous work by Jianfeng Wang (2004), Sheet metals naturally exhibit mechanical anisotropy because of the preferred grain orientation. The detailed crystallographic texture is determined by the thermo- mechanical manufacturing history (e.g., hot/cold rolling and annealing). Typically, sheet metals are orthotropic, with mirror symmetry axes aligned with the sheet rolling (RD), transverse (TD) and normal (ND) directions, as shown in Figure 2.2.

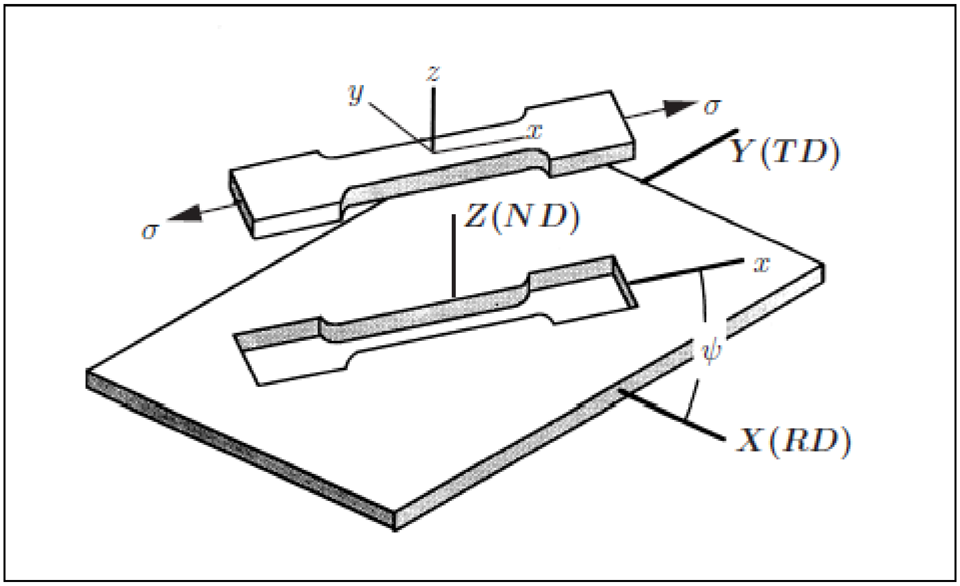


Figure 2.2: Schematics of coordinate systems of a sheet (*X*-*Y* -*Z*)

Source: Jianfeng Wang *et. al* 2004

To characterize the plastic anisotropy of the sheet, the *Lankford* parameter or the plastic strain ratio is generally adopted. The plastic strain ratio, also called *r-value*, is defined as follows:

$$r_{\psi} = \frac{\varepsilon_2^p}{\varepsilon_3^p}, \tag{2.1}$$

Where, ε_2^p and ε_3^p are true plastic strains in the sample width and thickness directions, respectively. The *r-values* can be measured by uniaxial tension tests for samples at different angles to the sheet rolling direction, Figure 2.2.

In practice, it is quite difficult to precisely measure small changes of the sheet thickness. Since plastic deformation conserves volume, i.e. $\mathcal{E}_1^p$ $+\mathcal{E}_2^p$, $+$ $\mathcal{E}_3^p$ $=$ 0**.** It is convenient and more accurate to calculate *r-value* alternatively:

$$r_\psi = - \frac{\mathcal{E}_2^p}{\mathcal{E}_1^p + \mathcal{E}_2^p} \tag{2.2}$$

While $\mathcal{E}_1^p$ is the plastic strain in the longitudinal direction.

The plastic strain ratio has a profound effect on sheet formability. It is a measure of the resistance to thinning or localized deformation, which usually precedes failure in sheet metal forming. Therefore, high *r-value* is desirable to achieve good formability. Usually, *r-value* varies with orientation in the sheet plane, and it is also a function of plastic strain. An averaged *r-value* is defined as:

$$r = \frac{r_0 + 2r_{45} + r_{90}}{4} \tag{2.3}$$

Which is closely correlates to the deep drawability of sheet metals. Here, r_0 , r_{45} and r_{90} designate the *r-value* measured from samples cut at 0°, 45°and 90° from the sheet rolling direction. On the other hand, the in-plane variation of the *r-value* can be evaluated by:

$$\Delta r = \frac{r_0 - 2r_{45} + r_{90}}{2} \tag{2.4}$$

Δr is a measure of the planar anisotropy, and it relates to the earing profile in deep- drawn products.

2.3 Tensile Test

The tensile test is widely used for the measurement of basic mechanical properties of metals, plastics, and composite materials. Mechanical properties are an important measure in order to produce the good products qualities, and the tensile test is just one way to certify the products. In tensile test, the materials properties that are always define are elastic deformation properties (Young's modulus and Poisson's ratio), yield strength, ultimate tensile strength, ductile properties (elongation and reduction in area) and strain-hardening characteristics.

2.3.1 Tensile Test Preparation

Based on the ASTM standard, there are three types of the specimens for the tensile test specimens where it is plate, sheet and round type specimen. In this project, sheet metal type was being performed to get the material properties. This is because in the finite element and experiment U-bending, the specimens of sheet metal will be use in order to get the springback angle.

By using ASTM standard E-8M, for the standard sheet type test specimen is shown in Figure 2.1. This specimen is used for cover the tension testing of metallic materials in any form at room temperature, specifically, the methods of determination of yield strength, yield point elongation, tensile strength, elongation and reduction area in the form of sheet, plate, flat wire, strip, band and hoop ranging in nominal thickness from 0.13 to 19mm. The detail dimension and specification of the tensile test specimen is shown in the Figure 2.3 and Table 2.2.

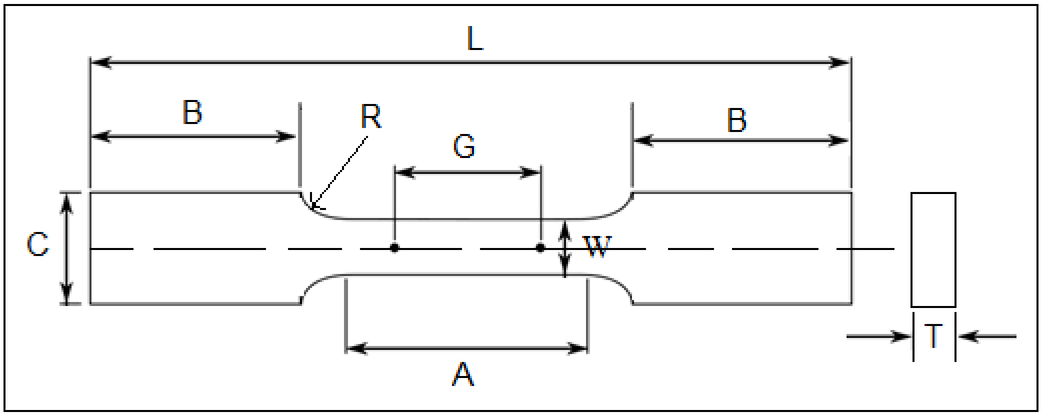


Figure 2.3: Rectangular (flat) tensile test specimen

Source: Annual Book of ASTM Standards, Vol 01.03. (2003)

Table 2.2: Dimension and Specification of the Tensile Test Specimen ASTM E-8M

	Plate type (1.5 in. wide) mm	Sheet type (0.5 in. wide) mm	Sub-size specimen (0.25 in. wide) mm
Gage length	200 ± 0.25	50.0 ± 0.10	25.0 ± 0.08
Width	40 + 3 - 6	12.5 ± 0.25	6.25 ± 0.05
Thickness		Thickness of Material	
Fillet radius (min.)	13	13	6
Overall length (min.)	450	200	100
Length of reduced section (min.)	225	60	32
Length of grip section (min.)	75	50	32
Width of grip section (approx.)	50	20	10

Source: Annual Book of ASTM Standards, Vol 01.03. (2003)

2.3.2 Stress Strain Curve

The Stress-strain curve was created from the plotting the value of the applied forces (stress) with the elongation the specimen (strain). The figure 2.4 was show the graph stress-strain by plotting the raw data from the tensile test experiment.

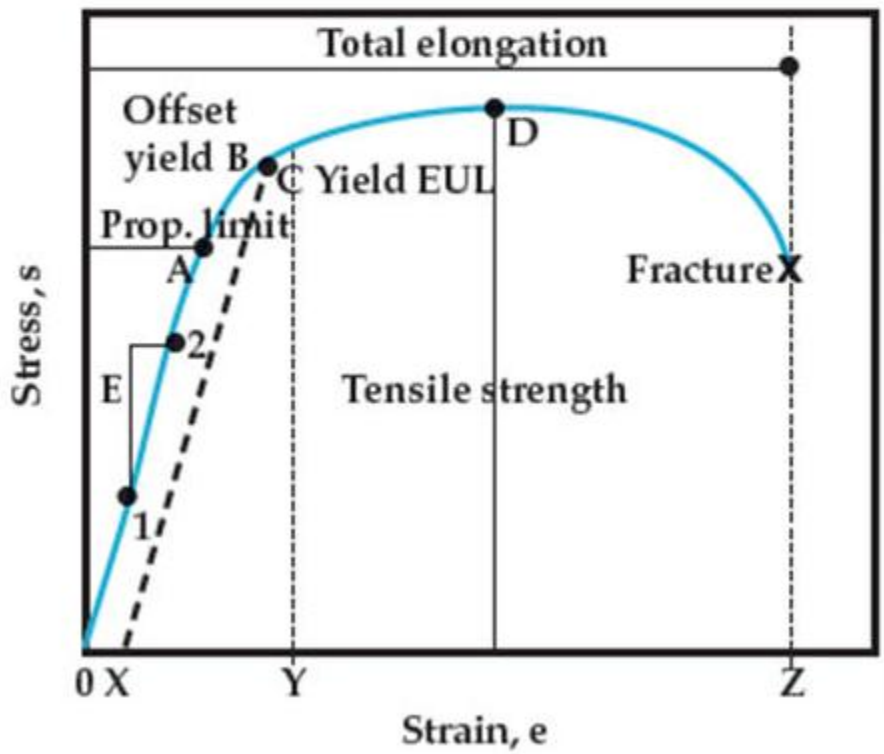


Figure 2.4: Stress Strain Curve

Source: Marc Venet (2010)

2.3.3 Yield Strength

The yield strength is the stress where the material which had been a tension compression load was gives longer return to its original length or shape when the load is removed. Some materials break before reaching yield strength, for example some of glassfilled nylons or die cast aluminum. Michael F. Ashby (2005) had suggested taking a piece of wire and slightly bending it in order to further visualize this property. It will return to its original shape. The point at which it stays bent is the yield point. The yield point is a very important concept because a part is usually useless after the material has reached that point. With most materials there is a gradual transition from elastic to plastic behavior, and the exact point at which plastic deformation begins to occur is hard to determine. Therefore, various criteria for the initiation of yielding are used depending on the sensitivity of the strain measurements and the intended use of the data. For most engineering design and specification applications, the yield strength is used. The yield strength is defined as the stress required producing a small, amount of plastic deformation. The offset yield strength is the stress corresponding to the intersection of the stress-strain curve and a line parallel to the elastic part of the curve offset by a specified strain (in the US the offset is typically 0.2% for metals and 2% for plastics).

2.3.4 Ultimate tensile strength

The ultimate tensile strength (UTS) is the maximum engineering stress level reached in a tension test. The strength of a material is its ability to withstand external forces without breaking. In brittle materials, the UTS were located at the end of the linear-elastic portion of the stress-strain curve or it was close to the elastic limit. In ductile materials, the UTS will be well outside of the elastic portion into the plastic portion of the stress-strain curve.

Based on figure 2.4, the UTS is the highest point where the line is momentarily flat. Since the UTS are based on the engineering stress, it is often not the same as the breaking strength. In ductile materials strain hardening occurs and the stress will continue to increase until fracture occurs, but the engineering stress-strain curve may show a decline in the stress level before fracture occurs. This is the result of engineering stress being based on the original cross-section area and not accounting for the necking that commonly occurs in the test specimen. The UTS may not be completely representative of the highest level of stress that a material can support, but the value is not typically used in the design of components anyway. For ductile metals the current design practice is to use the yield strength for sizing static components. However, since the UTS is easy to determine and quite reproducible, it is useful for the purposes of specifying a material and for quality control purposes. On the other hand, for brittle materials the design of a component may be based on the tensile strength of the material. However, it is depend on other factor such as the preparation of the specimen, the presence or otherwise of the surface defect, and the temperature of the test environment and material according to Michael F. Ashby (2005).

2.4 Bending Test

Springback is a very complicated behavior and not easy to predict through the mathematical models. Therefore, some experiments for understanding springback behavior were conducted and some mathematical models were developed. Various experimental techniques and procedures have been developed to study and characterize springback of sheet metals. The most popular and commonly used techniques are cylindrical bending, U-bending, V-bending and flanging. These methods are attractive because the level of springback is large and it can easily be measured. In this project, the U-bending techniques will be used to investigate the springback phenomena. Mohamed Faraj Alfaidi (2009) have stated that sensitivity of springback to basic parameters, such as R/t ratio (tool radius to sheet thickness), geometric parameters of the tools, mechanical properties of sheet material and friction parameters is usually studied by means of these techniques.

2.4.1 U-Bending Process

The u-bending process was requires at least a blank, blank holder, punch, and die. The blank is a piece of sheet metal, typically a rectangle shape, which is pre-cut from stock material and will be formed into the part. The blank is clamped down by the blank holder over the die, which has a cavity in the external shape of the part. A tool called a punch moves downward into the blank and draws, or stretches, the material into the die cavity. The movement of the punch is usually hydraulically powered to apply enough force to the blank. Know that U-shaped part is one of the representative parts in sheet metal forming.

Side wall curl is one of the main defects found in the forming of a U-shaped part based on Zhanga (2007).

By introducing a large blank-holder force (BHF) into the forming process is useful to remove side wall curl. According to Liu (2002) and Samuel (2000), they found that as the BHF increased, flow resistance of the material will also increase, the stress distribution through the thickness of side wall may be turned to tensile stress over the whole section. Accordingly, springback directions of both sides become consistent, which is conducive to decreasing shape distortion based on journal Zhanga (2007).

Before this, there are many researchers have investigated and attempted to obtain a basic understanding of springback behavior by using U-Bending techniques. Gomes C (2005) have done the simulation of material models based on various anisotropic models and compare his result with the experimental outcome to show the variation of spring-back with orientation of anisotropic sheet in U-die bending process. Papeleux and Ponthot (2002) have investigated the effect of blank holder force and friction coefficient on spring-back in U-die bending. M. Bakhshi-Jooybari (2009) had done the experimental and numerical studies of the effects on significant parameters including sheet thickness, sheets anisotropy and punch tip radius on spring-back/ spring-go in V-bending and U-bending processes of CK67 (DIN 17222) steel sheet have been conducted.

Based on journal Bhadpiroon Sresomroeng (2011), U-bending experimental has been conducted to verify the analytical model. The initial rectangular blanks of 210 mm in length and 50 mm in width were bended into hat-shaped parts. Experiments were carried out using the U-bending die shown in Figure 2.5 on a 1500-kN hydraulic press machine.

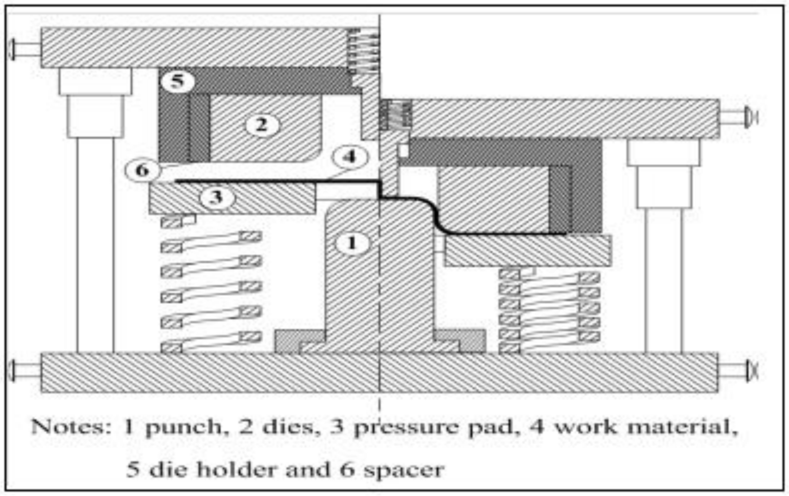


Figure 2.5: U-die Bending Process

Source: Bhadpiroon Sresomroeng *et. al* 2011

2.5 Finite Element Simulation

Finite element analysis is a simulation technique which evaluates the behavior of components, equipments and structures for various loading conditions including applied forces, pressures and temperatures. According to Hakim S. Sultan (2004), he was say that the analysis of whole structure is obtained by simultaneously analysis the individual finite elements, having due regard to their individual positions within the mesh and being totally dependent upon the assistance of an automatic computer. Aljibori (2009) was decide by using the finite element analysis method, simulation of design can helps to predict errors and modification can be done at early stage before the parts was fabricated and tested. Subsequently the labour cost and time lost can be reduced. When we are use finite element simulation, there are two majors step that we must follow. M.Firat (2005) have give this two step. First, a forming analysis is conducted, including the blank and tooling, in order to determine the sheet metal deformation during the stamping process and, second, the sheet metal springback deformations following the removal of the stamping tooling are computed using the forming stress distribution and the deformed geometry along with thickness distribution.

Wang (1988) Finite element analysis is now widely used in the sheet metal forming process to predict draw-in, strain distribution and optimal blank holder forces. M. Bakhshi-Jooybari (2009) done the simulation of springback was perform by using the FE code, ABAQUS 6.4. were material properties obtained from the tensile test will put into the software and the hill's anisotropy parameters were implemented into the code to model the sheet anisotropy. According to M.Tisza (2004), incremental FEM simulation are suitable for optimize the process and die design, and to make the necessary changes and even optimize the process parameters to ensure feasible processes and acceptable product quality. Besides that, he also makes assumption that the finite element simulation is powerful enough to predict all the forming defects and provide optimum stamping tools and conditions, the prototype tools may be completely eliminated from the design and manufacturing procedure, and the number of trial and modification can be significantly reduced.

Based on M.Kadkhodayan (2008) influence of different hardening models in simulating the U-bending process by utilizing the finite element code, ABAQUS 6.5 has been investigated. The simulation of the U-bending process is modeled in the finite element program ABAQUS/Explicit, while the springback analysis is simulated in ABAQUS/Standard as it would take a long time to obtain a quasistatic solution of springback analysis in ABAQUS/Explicit. V.Esat (2001) Analyses were carried out using the commercially available finite-element software MARC/MENTAT with two different aluminum wrought alloys, 2014-T6 and 6061-T6, were used.

According to Mehmet Firat (2008), the FE analyses of Numisheet'93 U-channel forming process with blank geometry is a sheet metal strip of size 300mm x 39 mm x 0.78 mm. Due to the symmetry conditions, only half of the tooling and the blank are include in the simulations along with appropriate boundary conditions. The figure 2.6 was showed the drawing of the simulation part. In this journal also tell about the materials properties were used in this simulation such as the elastic modulus and Poisson's ratio of the blank are 207 Gpa and 0.3 respectively.

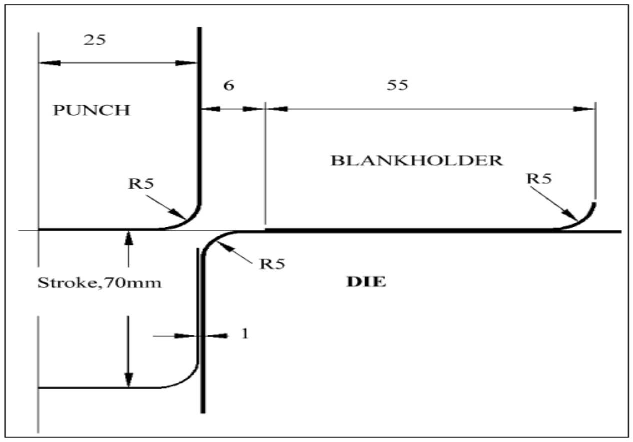


Figure 2.6: Symmetry of the FE Simulation Set Up for U-Bending

Source**:** Mehmet Firat *et. al* 2008

2.6 Measuring Angle

There is much journal use this method in order to find the springback parameter of θ1 and θ2. Following the procedure provided in figure 2.7 for measuring angles of U-bending parts, the angle θ1 had defined between the bottom of the profile and sidewall and for the angle θ2 had defined between sidewall and restraint profile under the blankholder.

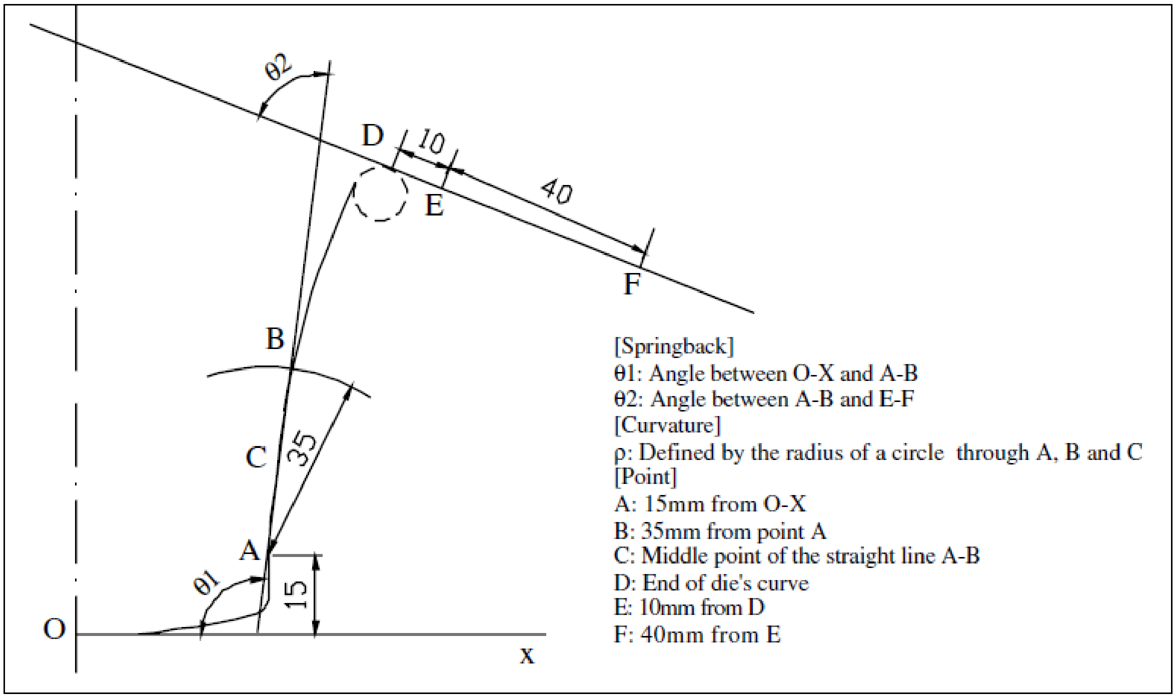


Figure 2.7: Measuring Angle of Springback

Source: Mehmet Firat *et. al* 2008

CHAPTER 3

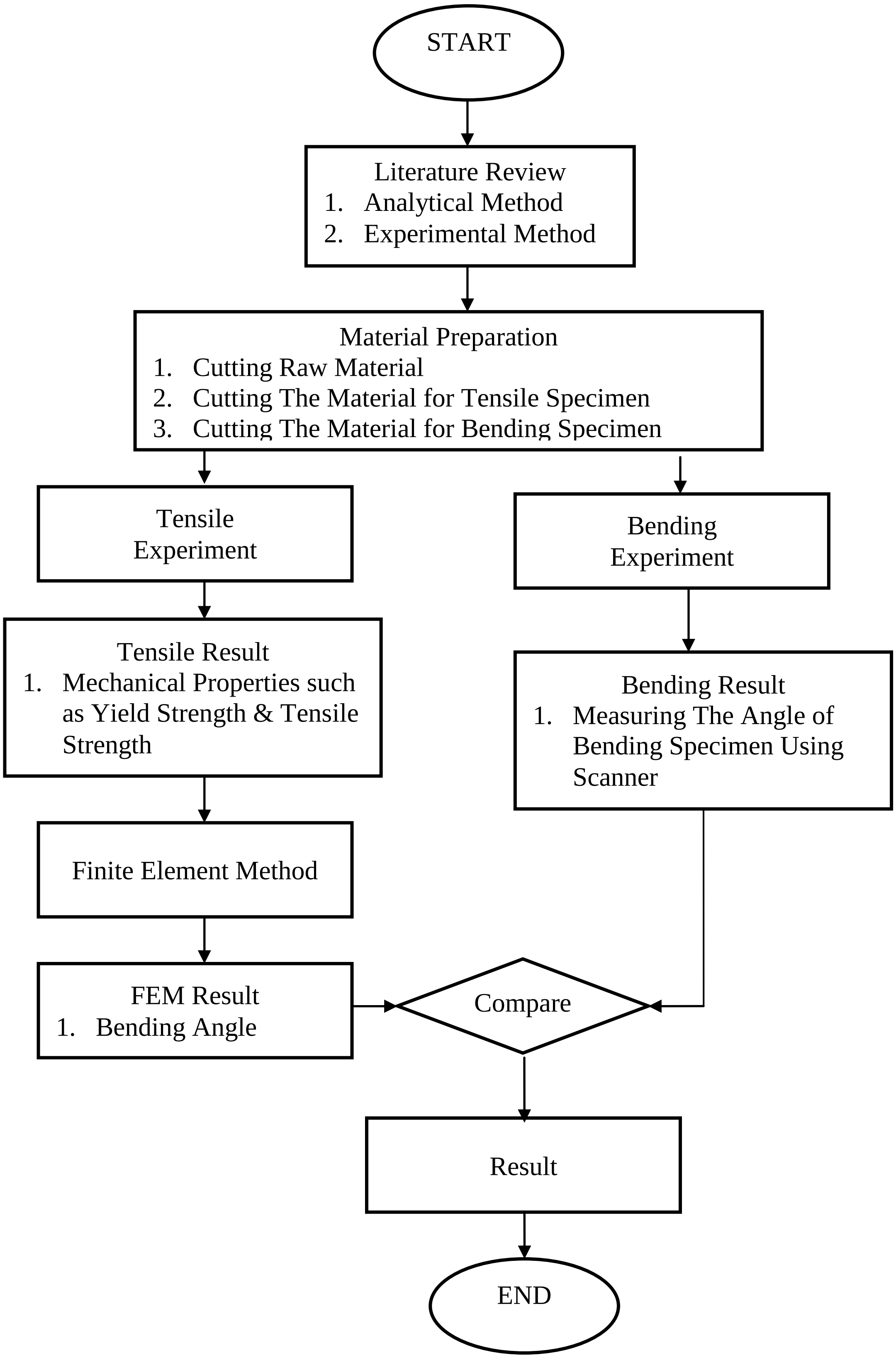
METHODOLOGY

3.1 Introduction

The methodology was conducted in this chapter based on literature review from the others researchers have done that related to our project. It was discussed about the process flow chart that is required to guide in order to make sure the result outcomes are accurate. This chapter also will explain about the procedures and parameters include in bending test. All the method tensile test, experimental and simulation of the U-bending model will be stated.

The flow chart will be discussed step by step following the literature. This flow chart will be guide of our project to measured springback parameters. This project had conducted two methods, experimental and simulation. Besides that, tensile test was performed to get the material properties as the input in finite element method. For the experimental, we use the U-die bending to analysis the springback. For the simulation, we are using Abaqus 6.7 software to analysis the angle of springback. Result of this two analysis will compared whether simulation finite element was reliability to use in industry. The flow chart of this project had showed in figure 3.1.

3.2 Project Flow Chart



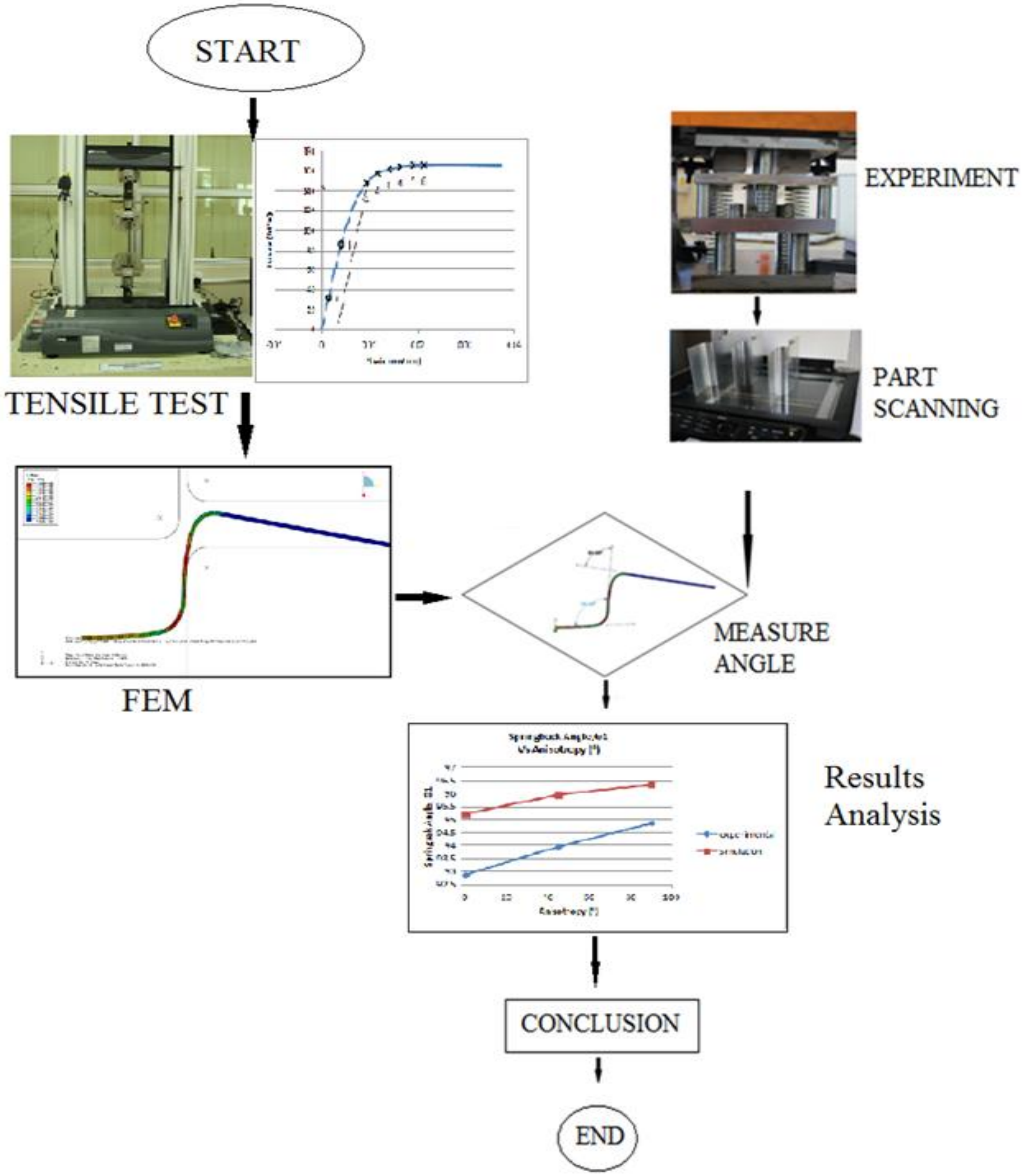


Figure 3.1: Project Flow Chart

3.3 Tensile Test

Tensile test is used to evaluate the strength of metals and alloys. In this test an aluminium in shape of dog bone is pulled to failure in a relatively short time at a constant rate. Before testing, two small punch marks are identified along the specimen's length. The ability of a material to resist breaking under tensile stress is one of the most important and widely measured properties of materials used in structural applications.

Using the recorded data, the nominal or engineering stress is found by dividing the applied load P by the specimen's original cross sectional area, A_0 .

$$\sigma = \frac{P}{A_0} \tag{3.1}$$

The nominal or engineering strain is found by dividing the change in the specimen's gage length, δ ($\delta = L - L_0$) by the specimen's original gage length, L_0

$$\varepsilon = \frac{\delta}{L_0} \tag{3.2}$$

From the data of a tension test, it is possible to compute various values of the stress and corresponding strain in the specimen and then plot the results. The resulting curve is called the stress-strain diagram. The yield strength, ultimate tensile strength, breaking strength and elastic or Young's modulus of a material can all be determined from this curve.

The valid result from the tensile test will perform engineering applications same as actual experimental. Tensile properties frequently are included in material specifications to ensure quality. Tensile properties often are measured during development of new materials and processes, so that different materials and processes can be compared. Finally, tensile properties often are used to predict the behavior of a material under forms of loading other than uniaxial tension. In tensile test, the strength of a material often is the primary concern. The strength of interest may be measured in terms of either the stress necessary to cause appreciable plastic deformation or the maximum stress that the material can withstand. Tensile test also give the material's ductility, which is a measure of how much it can be deformed before it fractures.

3.3.1 Objectives

This chapter provides an explanation of associated with tensile test. These include ASTM Standard, material preparation, tensile specimen and test machine, stress-strain curves including discussion of elastic versus plastic deformation, yield points, and ductility.

3.3.2 Test Specimen Preparation

In this test, the Aluminium with the thickness 1mm has been used to get the material properties. Specimens have been cut in the different orientation angle of 0, 45 and 90 degree and each orientation angle has been cut for 3 specimens. Table 3.1 has shown the total specimens in this project.

Table 3.1: Total Specimens

Specimen Thickness (mm)	Bil. of Specimen			
	0°	45°	90°	Total Specimen
1.0	3	3	3	9

3.3.3 Cutting Raw Material

Aluminium sheet metal needs to be cut into specimen according to ASTM Standards. There are many of ASTM standard can be used in design the specimen of tensile test. One of the standard were used for designed specimen is ASTM Standard 8E-M. Design of specimen use in tensile test was following by this ASTM standard to get the accurate value of material properties like modulus of elasticity, elastic limit, elongation, proportional limit, reduction in area, tensile strength, yield point, yield strength and other tensile properties. In this project, the standard sheet-type (12.5mm) are used for the tensile test specimen. The result from the tensile test will be use in finite element analysis to evaluate the springback angle during the simulation of bending test. Follow this Standard, sheet metal need to be cut into a rectangular size of 300mm x 50mm by using LVD shearing cutting machine in figure 3.2. The material then need to be cut in the direction of 0^o, 45^o and 90^o, which varying with the direction of rolling.

Figure 3.2: LVD Shearing Machine

3.3.4 Design Tensile Specimen using Mastercam's

By using Mastercam's, tensile test specimen has been draw to generate the G-Code of the simulation cutting. This G-Code is important as the input for start run the CNC milling machine. There are many parameters need to be change which including contour, drill, speed, thickness, face, peel mill, advanced multiaxis, and many more and enable us to cut parts efficiently and accurately. The scheme 3.1 below shown the tensile specimen design processes.

First step, the specimen of tensile test should be drawing in this Mastercam software.

⇩

Then the type of material for specimen should be specifying in the machine group properties toolbar. After that, at the machine group properties toolbar again, set up the raw material size that need to be cut at stock setup.

⇩

Then, at 2D-toolpaths icon select contour as a toolpaths type. After that, select 16 flat endmill as the tool for this process. 16 flat endmill is a 16mm diameter cutting tool for endmill cutting process type.

⇩

The next step is setup the feed rate, spindle speed and plunge rate for this process. The feed rate has been set about 40mm/min while the plunge rate is about 10mm/min. The spindle speed is set to be 900rpm.

⇩

Then, at the linking parameters icon, set up the depth to be cut to the raw material. Forexample, if the thickness of the material is 2mm, the suitable cutting depth is 2.5mm.

⇩

After all parameter for this process has been setup in the software, the simulation has been run to check the flow of this process. Finally, ganerate the G-code from the simulation process.

Scheme 3.1: Design Test Specimen using Mastercam's Process

3.3.5 CNC Milling Machine Process

The process of cutting tensile specimen has been performing by using Haas CNC Machine in figure. The G-Code that has been generated from Mastercam has been imported to the CNC milling machine. The scheme 3.2 below has shown the basic processes to running the milling machine and the figure 3.3 have show the milling process and outcome part from this process.

Powering ON The Machine

↓

Select And Open The G-Code Programe

↓

Locating X and Y Zero Points of The Part

↓

Loading The Tool into The Tool Carousel

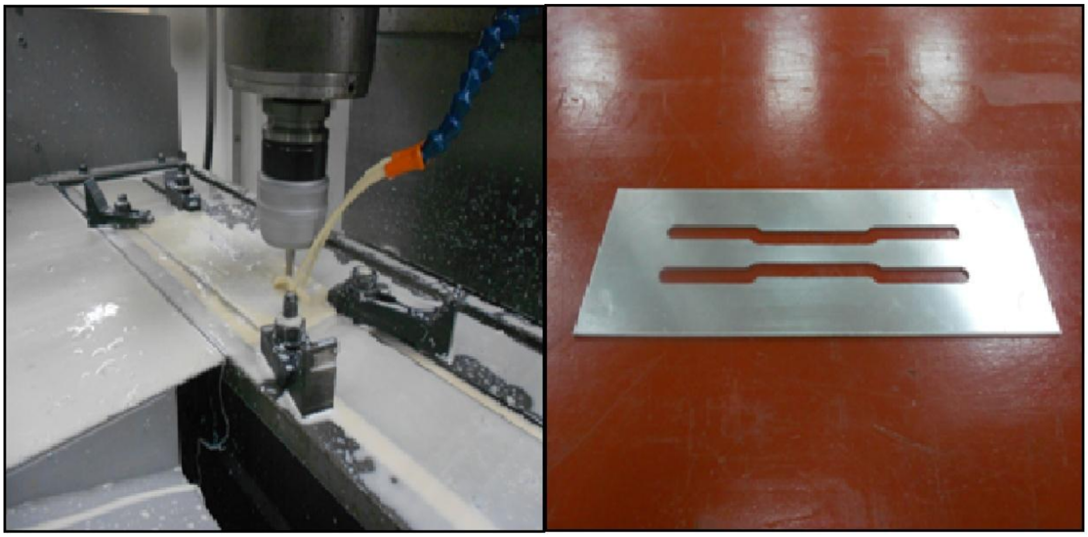
↓

Setting The Tool Offsets

↓

Running The Programe

Scheme 3.2: Basic Processes to Running The Milling Machine



(a) (b)

Figure 3.3: Tensile Specimens preparation; (a) Milling Process (b) Semi-Finish Specimen

3.3.6 Tensile Test Experiment

After the test specimen has been properly prepared, the tensile test setup will be established to conduct the tensile test. Tensile test have be done using the Machine INSTRON. The capacity of this machine are 50KN (11,250 lb). The force that available for this machine is 50KN.

There are several potential problems that must be watched during the test, including specimen misalignment and worn grips. The physical alignment of the two points of attachment of the specimen is important, because any off-center loading will exert bending loads on the specimen. This is critical in testing of brittle materials, and may cause problems even for ductile materials. Alignment will be affected by the testing-machine load frame, any grips and fixtures used, and the specimen itself.

Worn grips may contribute to off-center loading. Uneven tooth marks across the width of the specimen tab are an indication of trouble in wedge grips. Split-collar grips may also cause off-center loading. Uneven wear of grips and mismatching of split-shell insert pairs are potential problem areas. The scheme 3.3 has showed the flow of the tensile test procedure.

Open BLUEHILL Software in the computer.

Load the specimen into the lower grip and upper grip. Make sure the specimen is straight.

Click METHOD icon and set the specimen parameter (lenght, thickness, width) and crosshead speed (5mm/min).

Click TEST icon and enter the file name. Choose a test method that has been created .

Select the data that should be provided from the test and setting the Stress vs Strain graph in the TEST RESULT and save the method.

Reset the load and strain and click START TEST to start the tensile test.

After the test finished, click to the ULTILITIES in the main screen to view the result and data and save the result.

Finally, measure the final lenght and width of the specimen and record it in the table.

Scheme 3.3: Tensile Experiment Procedure

3.4 U-bending Experiment

Springback is a very complicated behavior and not easy to predict through the mathematical models. Therefore, some experiments for understanding springback behavior were conducted and some simulation models were developed. Various experimental techniques and procedures have been developed to study and characterize springback of sheet metals. The most popular and commonly used techniques are cylindrical bending, U-bending, V-bending and flanging. These methods are attractive because the level of springback is large and it can easily be measured. In this project, the U-bending techniques will use to investigate the springback occur. The experiments were performed using press machine with U-bending die. The aluminium sheet specimens from three different orientations were bent and the specimen has been scanned to measure the spring-back angles. The result of this experiment has been compared to the simulation method.

3.4.1 Objectives

The objectives of this experiment of U-bending are:

I. To determine the spring-back angle in sheet metal of aluminium.
II. To determine effect of different orientation angles.

3.4.2 Specimen Preparation

In this experiment, 1 mm thickness of Aluminium has been use to performed the U-bending test. The sheet metal has been cut in the different orientation angle of 0, 45 and 90 degree. For each orientation, three specimens are being tested. Tables 3.2 show the total specimens in this project and table 3.3 shows the material properties outcome from the tensile test.

Table 3.2: Amount of total specimen

Specimen Thickness (mm)	**Bil. of Specimen**			
	0°	**45°**	**90°**	**Total Specimen**
1	3	3	3	9

Table 3.3 : Material properties

Material	Sheet thickness (mm)	Young’s Modulus (MPa)	Ultimate Tensile Strength (MPa)	Strain hardening Exp (n)
Aluminium	1	21544.56	170	0.094

The material was cut into a rectangular size of 200mm x 100mm by using LVD shearing cutting machine. It was cut following the direction of 0^o, 45^o and 90^o, which varying with the direction of rolling. The figure 3.4 have shown the LVD Shearing machine and the specimen have been cut into it dimension.

Figure 3.4: LVD Shearing Machine and Specimen

3.4.3 Experiment Process

In this project, we consider a whole U-bending process into loading and unloading. In the loading step, a sheet metal is being bend into the die until the punch moves down completely, so that its shape is formed closely to the die shape. During this step, the workpiece undergoes elastroplastic deformation. Next, the deformed sheet metal is ejected from the tool set during the unloading step, while experiencing the residual stress release. The figure 3.5 has show the drawing of the U-bending die. At the left is the real die and at the left is the drawing die with the solidworks. The other view of die will put in APPENDIX E.

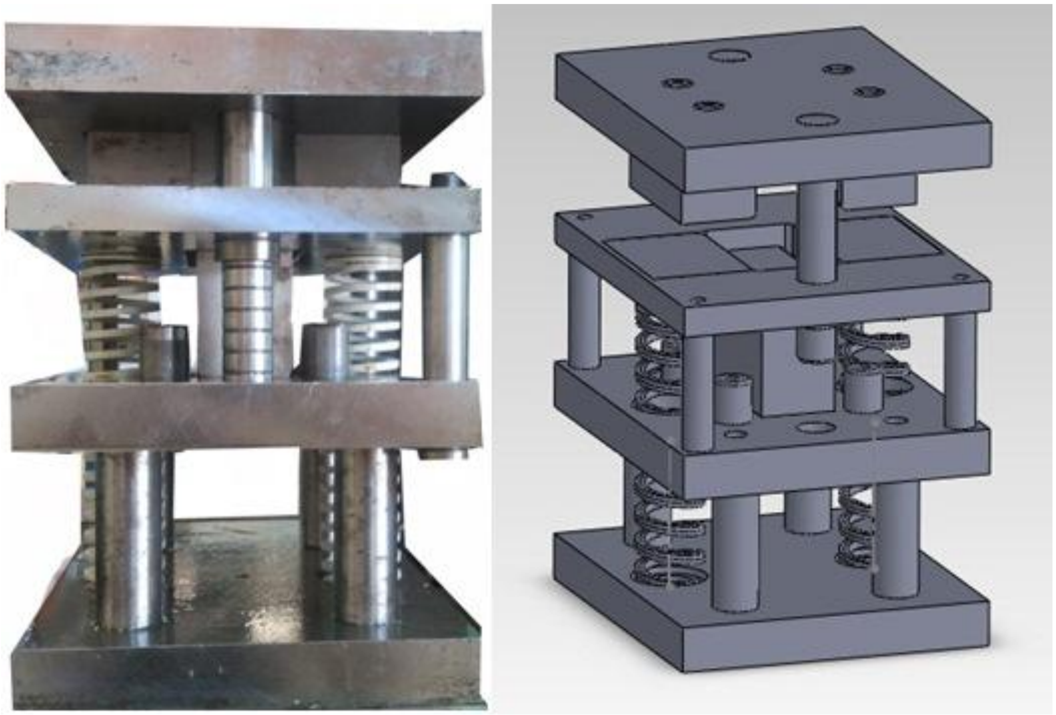


Figure 3.5: U-bending Die

3.4.4 Bending Machine

The SIGMA press machine (Figure 3.6) with a capacity of 1000KN maximum force was used to compress die and to perform the bending test in these experiments.

Figure 3.6: Press Machine Using for U-Bending Experiment

The specification of the machine is shown in table 3.4 below.

Table 3.4: Specification of Press Machine

Power Required	5 Hp
No. of Operations	50 - 60 Per Hour
Vibrator Motor	50 Hz
Hydraulic Pump Motor	5 Hp,415v, 3phases
Travel Motor	5 Hp, 415v, 3phases
Machine Weight	1100kg (Approx.)
Vibration	50/60 Hz
Operating Pressure	140kg/Cm2

3.4.5 Bending Process

The bending test procedure will be setup to conduct the bending test after the test specimen has been prepared. The specimen will be placed properly on the die at the press bending machine. The figures 3.7 show the setup of the die at the press machine and show the place for the specimen. The schemes 3.4 show the flow of the procedure to run this experiment.

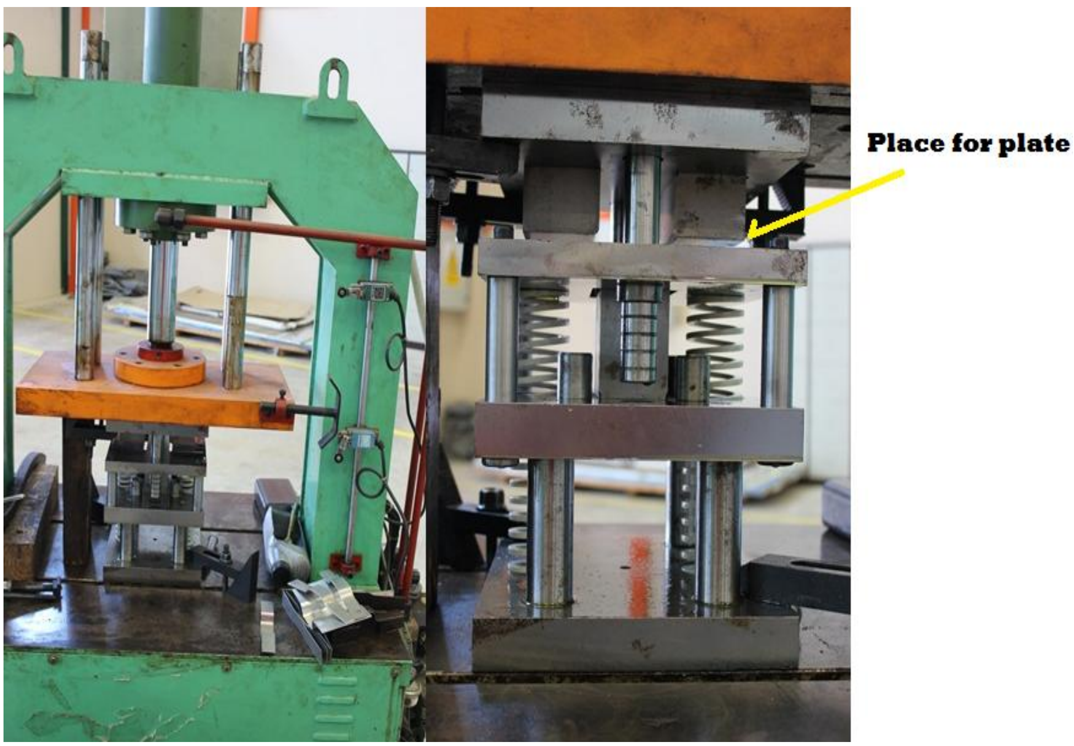


Figure 3.7: Die Setup in the Machine for U-Bending Experiment

Powering ON The Machine

⇩

Press button UP achieve maximum distance

⇩

locating the specimen on die

⇩

Press the button DOWN

⇩

Press button UP to comeout the specimen

Scheme 3.4: Flow of U-bending Process

Once the Press Machine have been ON, the UP button have be press to up the die to get the maximum force during the compression process. This button will be press until achieve the maximum distance. The specimen to be testing must be located on die correctly. The fault to locate the specimen will influence the spring back result. Press the DOWN button to give force for compress the die with the specimen. To come out the specimen those have been bending, press the UP button. The figure 3.8 below had shown the result from the bending process.

Figure 3.8: U-bending Parts Produced from Experiment

3.5 Finite Element Method

Finite element analysis is a simulation technique which evaluates the behavior of components, equipments and structures for various loading conditions including application of forces, pressures and temperatures. Finite element analysis is a computerized method for predicting how a real object will react to forces, heat, vibration, etc. by mesh of simpler interlocking structures, the simpler structures or finite elements being amenable to mathematical analysis. The analysis of whole structure is obtained by simultaneously analysis the individual finite elements, having due regard to their individual positions within the mesh and being totally dependent upon the assistance of an automatic computer. Numerical modeling of metal forming processes has now gained the industrial stage, and it became possible to simulate metal deformation and to calculate stress and strain states for complex processes. By using the finite element analysis method, simulation of design can helps to predict errors and modification can be done at early stage before the parts was fabricated and tested. Subsequently the labor cost and time lost can be reduced. Therefore finite element methods will gradually replacing manual trial-and-error design iteration with sophisticated numerical simulations in future. It can be seen that the increasing demands of using finite element method in manufacturing process (especially in pre-processing analysis stage) will greatly enhance the efficiency and saving of time and manpower.

In order to evaluate the performance of the proposed constitutive model in the deformation analysis of sheet metals, the forming analysis of U-channel benchmark is performed using ABAQUS 6.7. The main reasons for the selection of the channel forming are that the tooling geometry is designed specifically for springback benchmark of typical isotropic metals. In addition, the standard for the measurement of final U-channel geometry after forming process is well defined, and experimental results are available in the literature.

To predict the springback of sheet metal aluminium for the U-bending, the simulation of sheet metal forming are be done by using software ABAQUS. Based on logically, the sheet metal will form follow the shape of die. However, after loads are removed from the sheet metal, the final result is not typically. This case happens because of the behavior of this material that has plastic elasticity to change the final shape of material. The process elastically-driven change in the blank is term of springback.

In this finite element simulation, only half of the process was modeled in 2-D direction followed previous work by Mehmet Firat, (2008). The basic aspect in order to drawing the stamping operations consists of the surface contact between aluminium part and the tools such as the punch, die and blank holder. The tools can be modeled as rigid surfaces because they are much stiffer than the blank. Figure 3.9 shows the basic arrangement of the assembly part with the punch, die and blankholder that considered in FEM model.

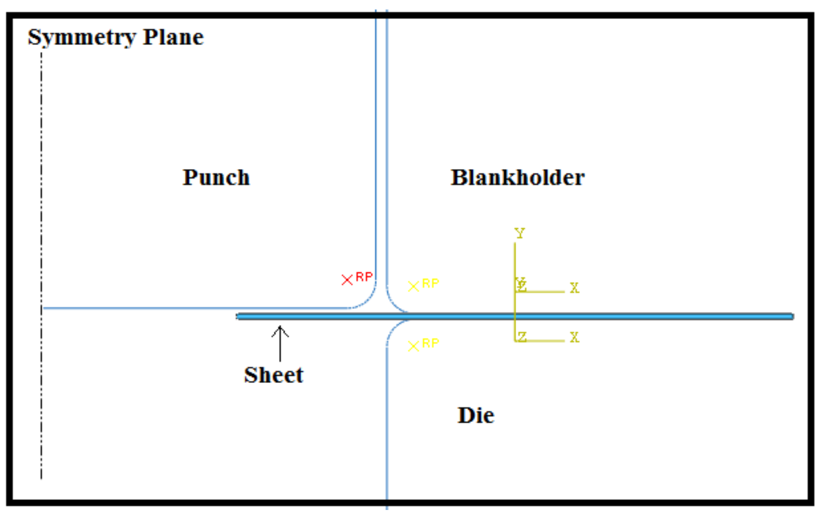


Figure 3.9: Geometrical Description of the FE Simulation Model

3.5.1 Objectives of FEM

The objectives of the Finite Element Method are:

I. To determine the effect of anisotropy in deformation of springback.
II. To determine the angle of springback of Aluminium.

3.5.2 Material Properties

From the tensile test result, the material properties of aluminium are determined. Materials properties of the aluminium were define from the curve stress-strain**.** The measured value of Young's modulus for aluminium is around 20 GPA in room temperature which is defined from slope of curve stress-strain. The ratio of the two normal strains (lateral and longitudinal) is a material constant called the poison's ratio.

$$v = -\frac{\varepsilon_{lateral}}{\varepsilon_{longitudinal}} = -\frac{(w_f - w_0)/w_0}{(l_f - l_0)/l_0} \tag{3.3}$$

Where w_0, w_f, l_0 and l_f are initial gage, final gage, width, and length.

Plastic properties or known as plasticity is a property of a solid body whereby it undergoes a permanent change in shape or size when subjected to a stress exceeding a particular value. In this simulation, data of plasticity contain of plastic strain and plastic stress for low aluminium. The data is shown in Table 3.5.

Table 3.5: Plastic properties for FE Material Input Data

	Plastic Strain, ε	**Yield Stress,** σ
1	0	148
2	0.002	157
3	0.005	162
4	0.007	164
5	0.009	165

3.5.3 Material Modeling

The study of this project consists of Young's modulus, E and Poisson's ratio, v. Therefore, a simple material modeling for describing the elasticity characteristic of an isotropic material is assuming linear behavior. Contact treatment forms an integral part of many large-deformation solutions. Accurate modeling of contact interfaces between bodies is crucial to the prediction capability of the finite element simulations. In this project, the simulations concentrate on friction. Assume the friction and friction less on the modeling contact in the simulation. Friction between die and plate is assumed to be 0.1 due to the plate need a small value of friction to determine elastic and plastic properties so that the plate has a tension and compress of its boundary. Tool movement is defined as small sliding which is default constant Coulomb friction.

3.5.4 Finite Element Process

In the initial process, the flat blank was placed between the die and blankholder. The punch was pushed the blank into the die's cavity. The part was slide into the die and deformed based the shape of die. After the punch is moving at the maximum displacement, it was removed from the die, the elastically-driven was changed the final shape of product. The figure 3.10 below was shown the movement of FE simulation process of this project.

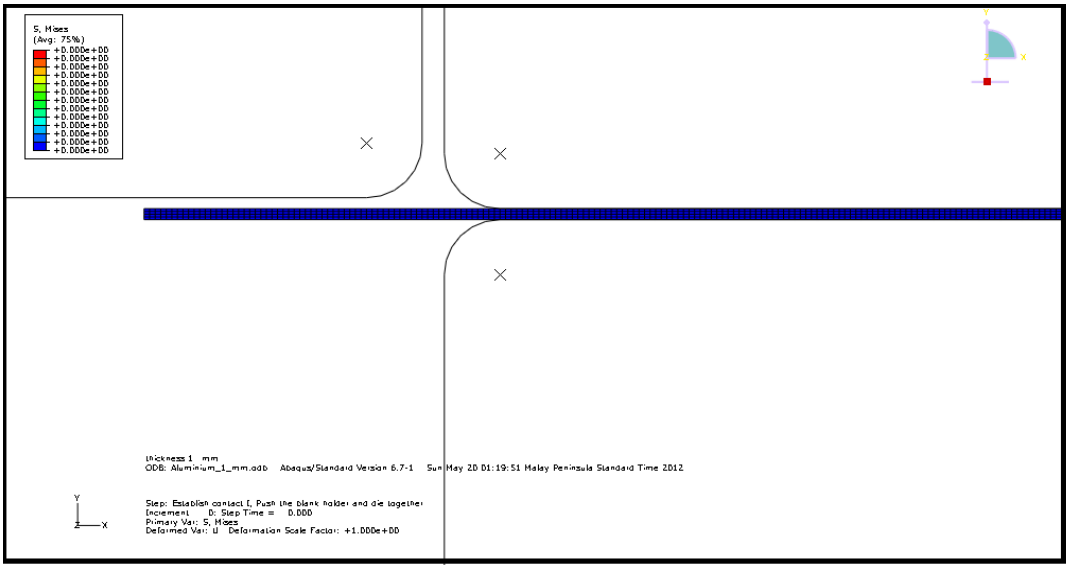


Figure 3.10 (a) : Initial Position before Loading.

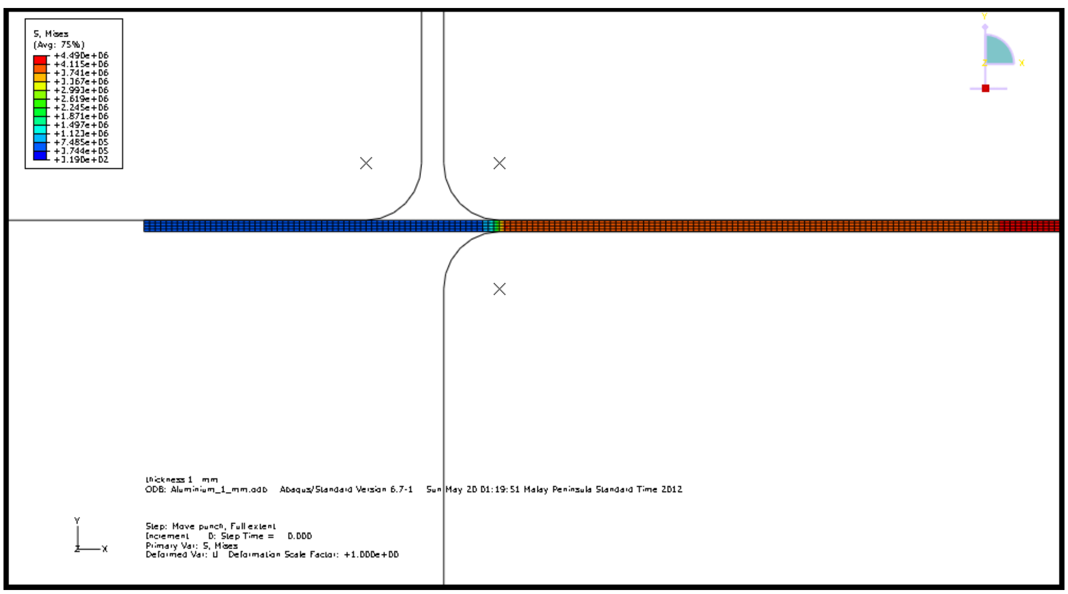


Figure 3.10 (b) : Initial Contact; Punch Touch the Sheet Metal

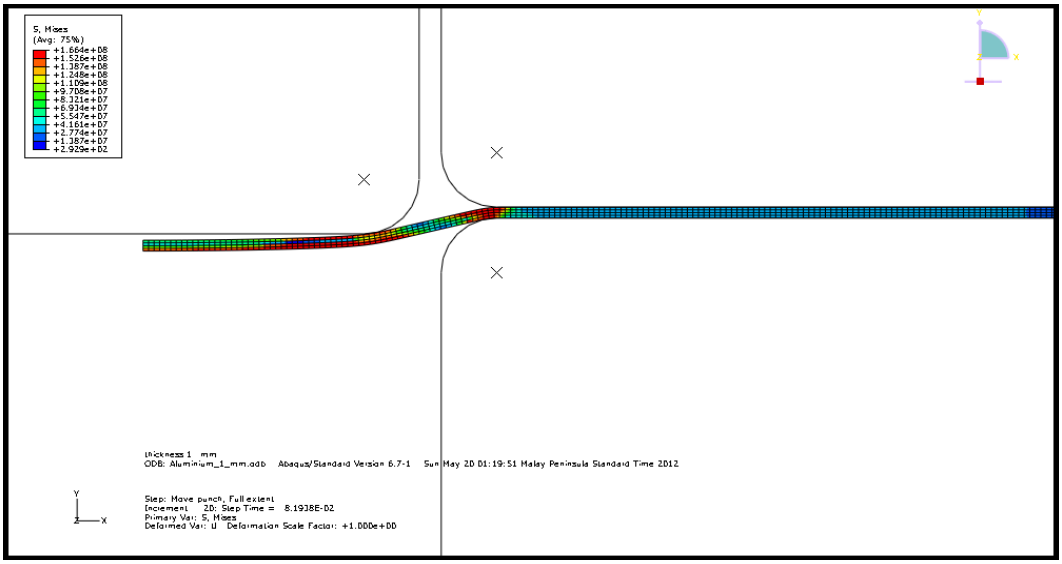


Figure 3.10 (c) : Punch start push the blank into die.

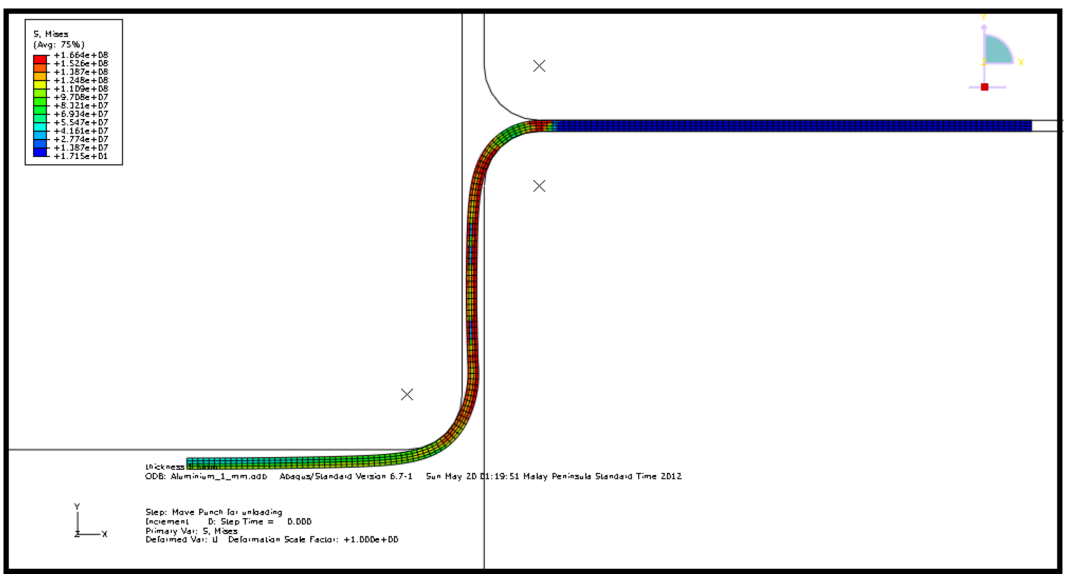


Figure 3.10 (d) : Sheet Metal are Punch in Maximum Die Cavity.

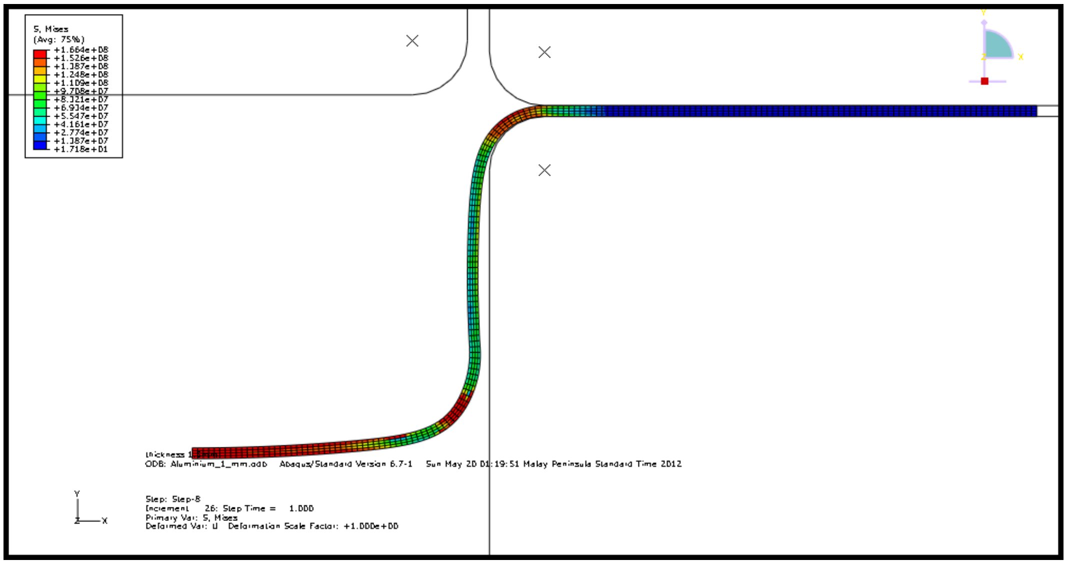


Figure 3.10 (e) : Punch are remove from sheet metal

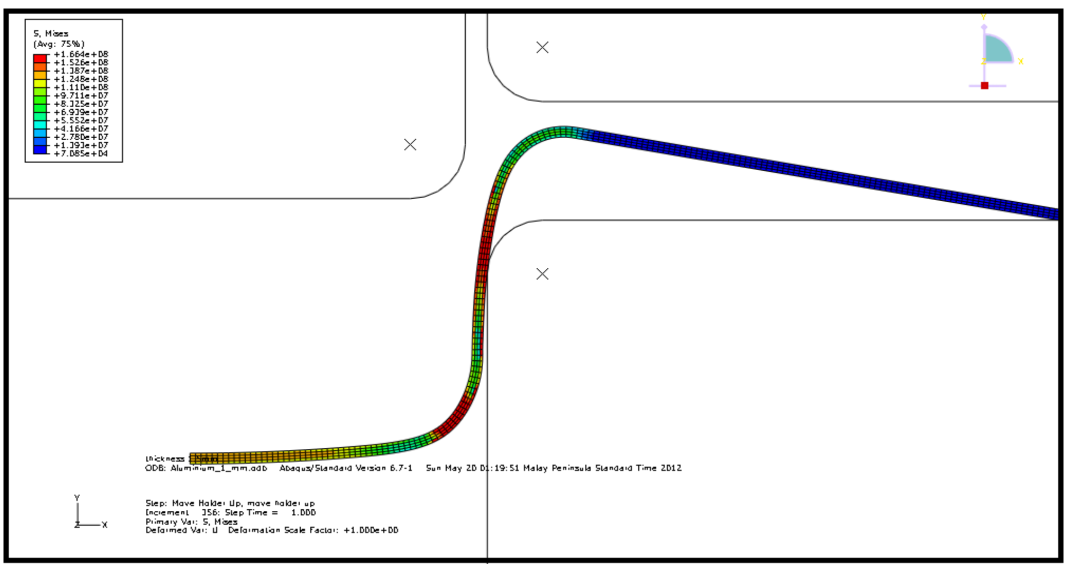


Figure 3.10 (f) : Blankholder Remove From the Sheet Metal and Springback Phenomena of Aluminium.

3.6 Angular Measurement

Defining the angle of specimen is important to study about springback. Those, we need follow the guideline to find the exact angle. The figure 2.7 in previous chapter had been discussed the guideline used in this experiment to find the springback angle of specimen.

3.6.1 U-bending Experiment

After the bending process, specimen from that process will be measure it angle of bend. The purpose of this measured are to determine the spring back or spring go. The bending angle of the specimen has been measured using the SolidWork software. After specimen has been bending, it will be scan using Canon MP 145 Series. The scanning profile will be save in the JPEG picture format and will be edit in Adobe photoshop 7.0 to get the clear image and will be save in adobe format. The bending specimen picture then to be imported in the SolidWorks software and the angle will be measured by sketching the line follow the specimen. After that, the dimension between two lines had been finding to get it angle. Schemes 3.5 show the procedure in measuring specimen angle and the figure 3.11 show the measuring angle in solidworks. The others specimen measuring angle will put in Appendix C.

Scan specimen using Canon MP 145 Series printer

⇩

Save the scanning profile in picture (JPEG Format)

⇩

Open Solidworks software and import the specimen picture using skecth tool toolbar

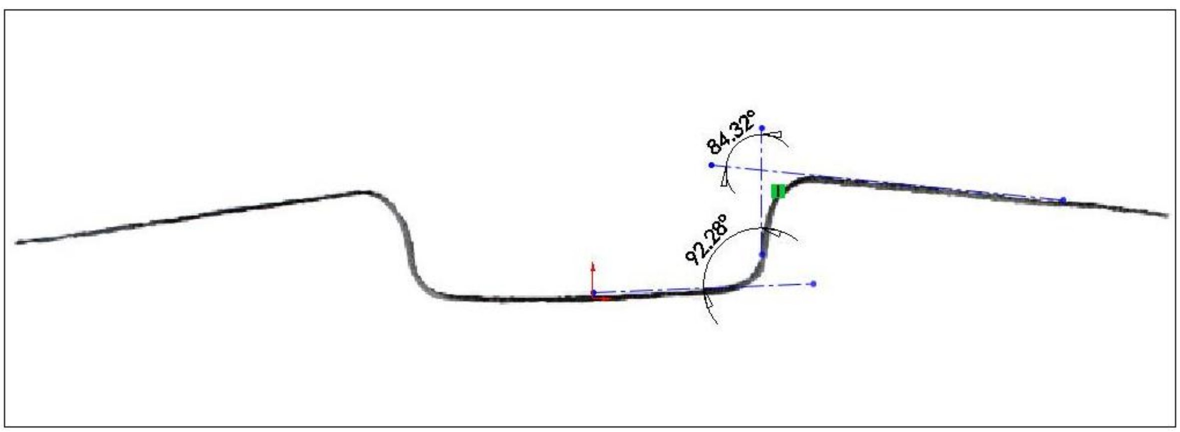
⇩

Skecth the straight line on the specimen and measure the angle using smart dimension tool

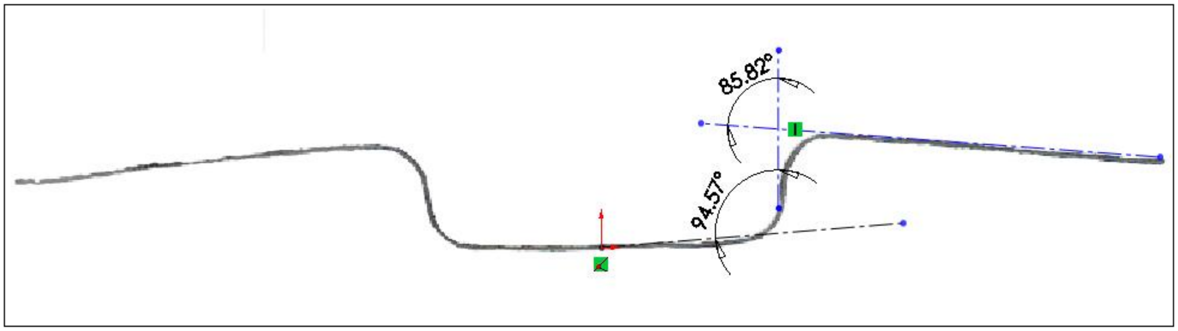
⇩

Save the result

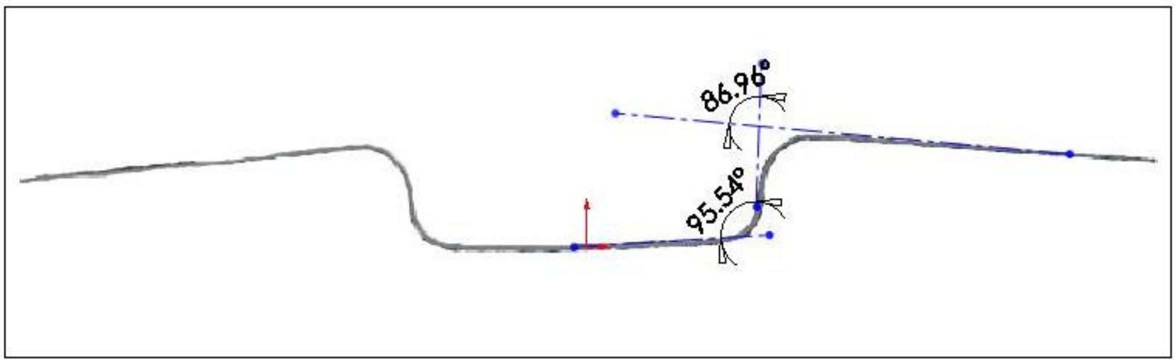
Scheme 3.5: Procedure to Measuring specimen angle



(a)



(b)

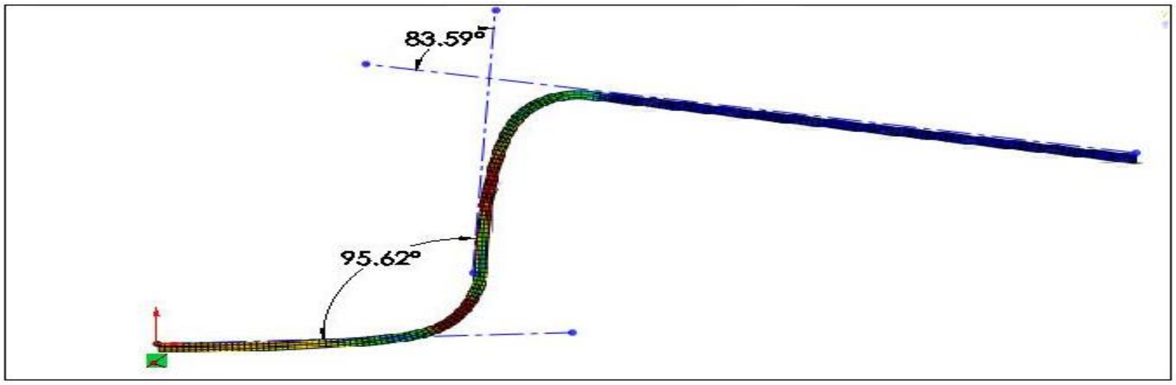


(c)

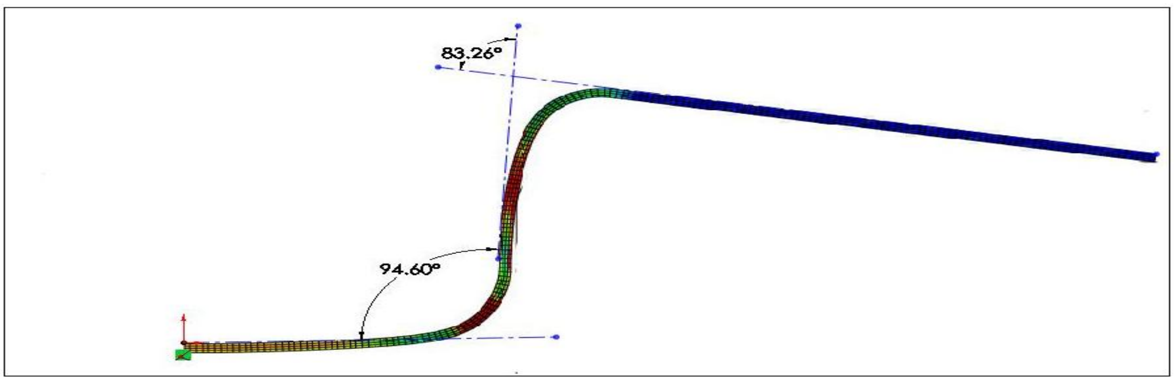
Figure 3.11: Measuring Angle of u-bending experiment parts in Solidworks ; a) 0 degree b) 45 degree c) 90 degree

3.6.2 Finite Element Method

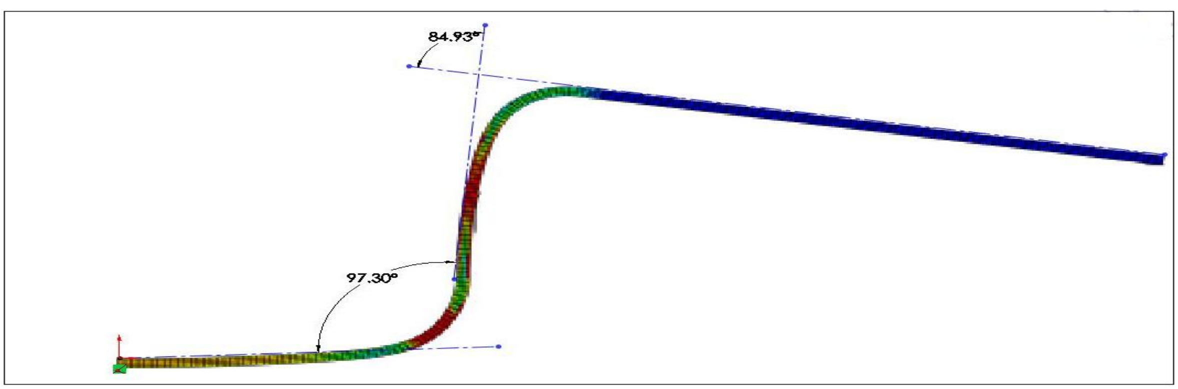
The bending specimen from the simulation using ABAQUS had saved in picture format and then was be imported in the SolidWorks software. The angle of specimen had been measured by sketch the line follow the blank shape. After that, the dimension between two lines had been finding to get it angle. The figure 3.12 was show step to define of angle in the solidworks for finite element method.



(a)



(b)



(c)

Figure 3.12: Measuring Angle of FE Simulation parts in Solidworks ; a) 0 degree b) 45 degree c) 90 degree

CHAPTER 4

RESULTS AND DISCUSSION

4.1 Introduction

This chapter will discuss about the finding result from the simulation and the experiment. The both result will be compare in term of springback angle. The validity will be discussed.

4.2 Tensile Test

Tensile test was performing to evaluate material properties of specimens. During the tensile test experiment, force and extension data has been monitored and recorded. The tensile test raw data like force and extension give several important mechanical properties of an Aluminium such as elastic deformation properties (Young’s modulus and Poisson’s ratio), yield strength, ultimate tensile strength, ductile properties (elongation and reduction in area) and strain-hardening characteristics. The mechanical properties that provided from tensile test will be use in the analytical method in solving the spring-back equation.

4.2.1 Material Properties

In the tensile test process, raw data like force (load) and extension data has been recorded. From the data, the graph of stress-strain was being plotted with some calculation of the engineering stress, engineering strain, true stress, and true strain. The engineering stress is found by dividing the applied load by the specimen original cross sectional area.

$$\sigma_{eng} = \frac{P}{A_0} \tag{4.1}$$

The engineering strain is found by dividing the change in the specimen gage length by the specimen original gage length.

$$\varepsilon_{eng} = \frac{\delta}{l_0} = \frac{l - l_0}{l_0} \tag{4.2}$$

The true stress and true strain of stainless Aluminium can be found by using the engineering stress and strain value with the equation below:

$$\sigma_{true} = \sigma_{eng}\left(1 + \varepsilon_{eng}\right) \tag{4.3}$$

$$\varepsilon_{true} = \ln\left(1 + \varepsilon_{eng}\right) \tag{4.4}$$

Table 4.1 below shown the value of engineering stress, true stress, engineering strain and true strain that can be find by using force (load) and extension data from this test. The data in the table below is initial value of the data in 10second and based on the tensile test data in table, the Stress-Strain graph can be plotted. The graph can be plotted by using the engineering stress and engineering strain. The graph in figure 1 was show stress-strain curve with different orientation.

Table 4.1: Tensile Test Result for Aluminium 1.0mm Thickness

Time (s)	Extension (mm)	Load (kN)	Eng Stress (Mpa)	True Stress (Mpa)	Eng Strain (mm/mm)	True Strain (mm/mm)
0	0.00000	0.105	8.472	8.472	0.000000	0.000000
1	0.01662	0.199	15.998	16.004	0.000332	0.000332
2	0.03337	0.295	23.604	23.619	0.000667	0.000667
3	0.04981	0.389	31.129	31.160	0.000996	0.000996
4	0.06687	0.484	38.764	38.817	0.001337	0.001337
5	0.08356	0.574	45.999	46.076	0.001671	0.001670
6	0.10000	0.664	53.160	53.267	0.002000	0.001998
7	0.11669	0.752	60.172	60.312	0.002334	0.002331
8	0.13312	0.837	66.972	67.150	0.002662	0.002659
9	0.15025	0.923	73.870	74.092	0.003005	0.003000
10	0.16694	1.004	80.339	80.607	0.003339	0.003333

.

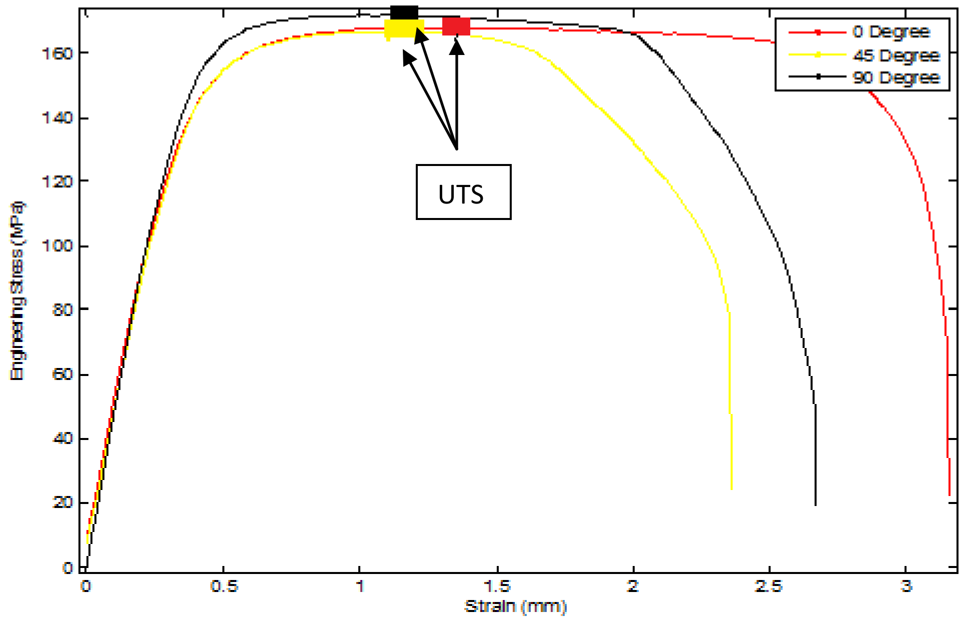


Figure 4.1: Stress Strain Curve with Different Orientation

Based on the figure 4.1, the proportional limit stress, σ_{pl} is seem to be a very close value for three different orientation angles. After it passes the proportional limit stress point, the test specimen is continuously extent until it reached the maximum stress normally known as ultimate tensile strength (UTS) before it suddenly break. From the graph, 90 degree orientation angle specimen has higher UTS value followed by 0 degree and 45 degree orientation angle specimens. At the elastic region, the slope of the stress-strain curve is defined as the elastic modulus or young's modulus. The slope of the curve is measured before the stress value reached the proportional limit. The parameters that include from the curve stress strain and tensile test result are:

i) Yield Strength

Yield strength, or the yield point, is defined in engineering as the amount of stress that a material can undergo before moving from elastic deformation into plastic deformation. Offset yield strength is determined from a stress-strain diagram. It is the stress corresponding to the intersection of the stress-strain curve, and a line parallel to its straight line portion offset by a specified strain. Offset for metals is usually specified as 0.2%.

ii) Young's Modulus

The Young Modulus, E is a material property that describes its stiffness and is therefore one of the most important properties in engineering design. Young's modulus can be used to predict the elongation or compression of an object as long as the stress is less than the yield strength of the material. In this project, the value of young's modulus had be calculated manually because of the extensometer at INSTRON machine not working as well. Young's modulus can be expressed as:

$$E = \frac{\text{stress}}{\text{strain}} = \frac{(\frac{F}{A})}{(\frac{dL}{L})} \tag{4.5}$$

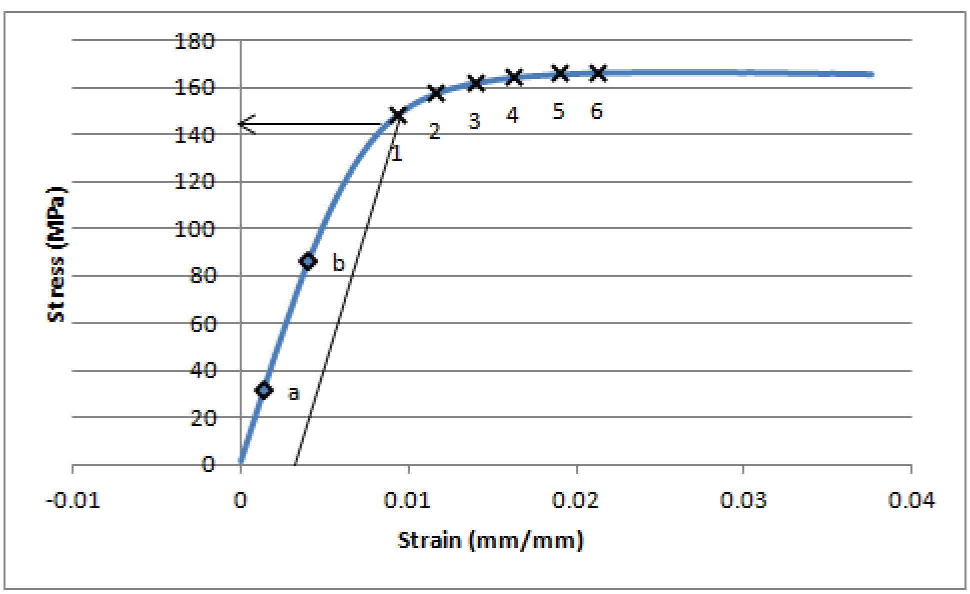


Figure 4.2: Stress-Strain Curve

Based on the graph, the arrow was show the yield strength of this curve where it was determined by using offset method. The slope of the graph was determined firstly and then with the same slope draw at 0.002 mm, the point intersection with the slope is determined. The value at Y-axis are represented the yield strength of this curve. In order to determine the young's modulus of the material, the slope of the curve must be calculated. Two points at elastic deformation was chosen; the first point is represent as 'a' and the second point is 'b'. The calculation of determined young's modulus was show in figure 4.3.

Young's Modulus = Slope of the Graph

Point a = 0.0013312, 31.4776

Point b = 0.0040012, 85.9544

$$\text{Slope} = \frac{y2-y1}{x2-x1}$$

$$= \frac{85.9544 - 31.4776}{0.004-0.0013} = 20403.3 \text{ Mpa}$$

Figure 4.3: Calculation of Young's Modulus

Besides, from stress-strain curve the points of plastic deformation also can be determined. Point 1, point 2 and the others are the points that were are chosen as the input to insert in the finite element method for the others specimen will show in Appendix D.

4.2.2 Summary

The results from the tensile test are combining and were put in a table 4.2 as the summary for tensile test result. All of the material properties such as Young's modulus, UTS, and Poisson ratio can determined in this table.

Table 4.2: Tensile Test Summary Result

Orientation Angle	**Final Width, (mm)**	**Final Length, (mm)**	**Young's Modulus (MPa)**	**Ultimate Tensile Strength, UTS (MPa)**	**Poisson's Ratio, v**
0	12.23	53.14	20403.3	166.897	0.343949045
0	12.21	53.23	20752.7	167.972	0.359133127
0	12.28	52.85	20648.7	168.968	0.30877193
Average 0	**12.24**	**53.0733**	**20601.57**	**167.946**	**0.338394794**
45	12.32	52.24	20392.7	166.751	0.321428571
45	12.35	52.12	20448.8	166.791	0.283018868
45	12.31	52.53	20324.6	165.026	0.300395257
Average 45	**12.32666667**	**52.2967**	**20388.7**	**166.189**	**0.301886792**
90	12.33	51.84	16696.15	190.332	0.369565217
90	12.35	51.57	22259.1	171.919	0.382165605
90	12.32	52.33	21921.5	165.363	0.309012876
Average 90	**12.33333333**	**51.9133**	**20292.25**	**175.871**	**0.348432056**
Average Result For 1mm Thickness	***12.3***	***52.4278***	***20427.5***	***170***	***0.329571214***

Based on the table 4.2, the average range of young's modulus is between 20.6 GPa and 20.29 GPa. These results might be valid because all the test procedure based ASTM E-8M. The average range of Poisson ratio is between 0.30 and 0.34 is quite same with the previous finding by Aljibori (2009).

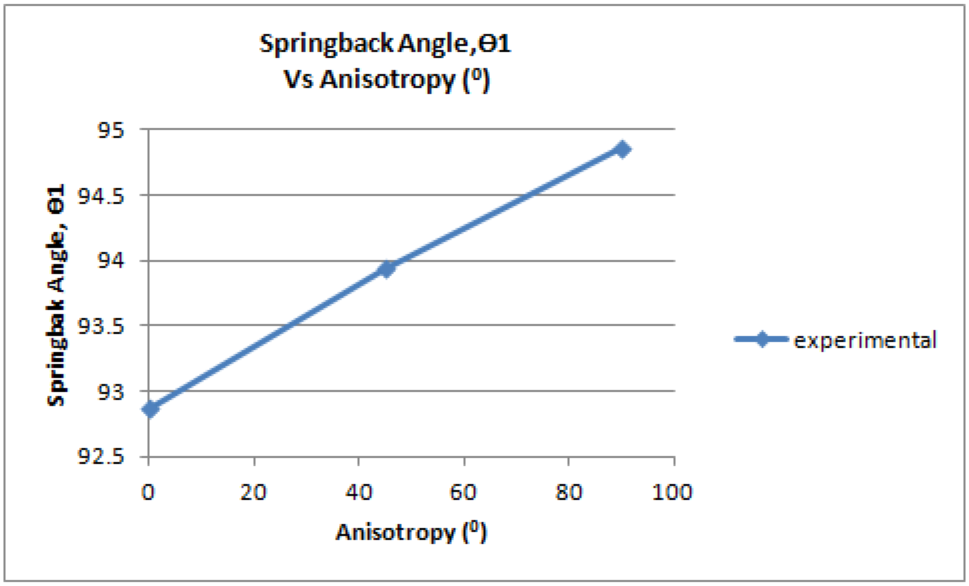
4.3 U-Bending Experiment

In the U-bending test, the measured of specimen angle is very important to defined the spring back of the part. The specimen angle of aluminium material usually is greater than die angle after the bending process. This is because the elastic recovery characteristic of the material. Thus, the specimen tries to spring-back and the bent part slightly opens out. The results of this experiment are presented in Table 4.3.

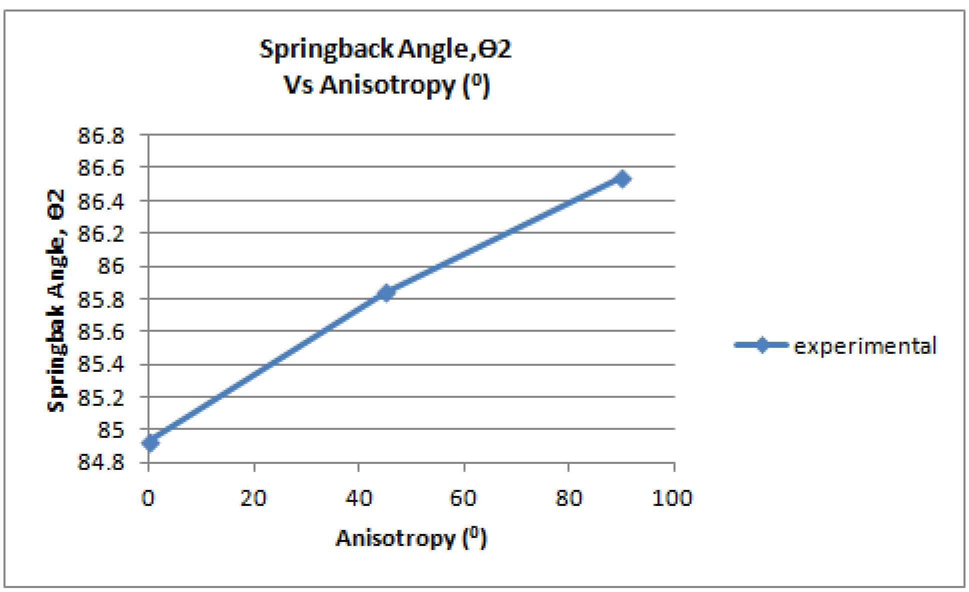
Table 4.3: Experimental Angle of $\boldsymbol{\theta_1}$ and $\boldsymbol{\theta_2}$

Thickness (mm)	Specimen Angle	Orientation Angle (Degree)	Bending Angle (Degree) Sample			Average Angle (Degree)
		0	92.28	92.60	93.73	92.87
1mm	$\boldsymbol{\theta_1}$	**45**	94.57	93.11	94.13	93.94
		90	95.54	94.34	94.69	94.86
		0	84.32	86.13	84.33	84.93
	$\boldsymbol{\theta_2}$	**45**	85.82	85.87	85.82	85.84
		90	86.96	85.97	86.69	86.54

Based on this result, the average of springback angle was calculated and the graph of springback versus anisotropy is plotting. The figure 4.4 was show the graph of springback versus anisotropy.



(a)



(b)

Figure 4.4: Experimental Springback versus Anisotropy (a) for θ_1 (b) for θ_2

The graph in figure 4.4(a) had shown the experimental results of springback versus Anisotropy for θ_1. Based on the graph, when the anisotropy value was increased, the springback angle also increased. This finding was good agreement with the previous study by M. Bakhshi-jooybari (2009). The lower springback occurred at the 0 degree orientation, which is the angle of springback only 92.87°. The higher springback occured at 90 degree orientation, which is have the average angle of springback 94.86°. At the middle, for orientation 45 degree the average deformation angle of springback is 93.94°. These phenomena also happen in figure 4.4(b) that shows the springback versus anisotropy for θ_2. The higher angle deformation of springback is at 90 degree which is 86.54. For the 45 degree orientation, the average angle is 85.84° and the lower springback occur at 0 degree orientation which is only 84.93°.

One of the specimens value has been plotted by using the Solidworks to get the length value follows the shape of the specimen from the origin. The results of the plotting graph have been show in figure 4.5.

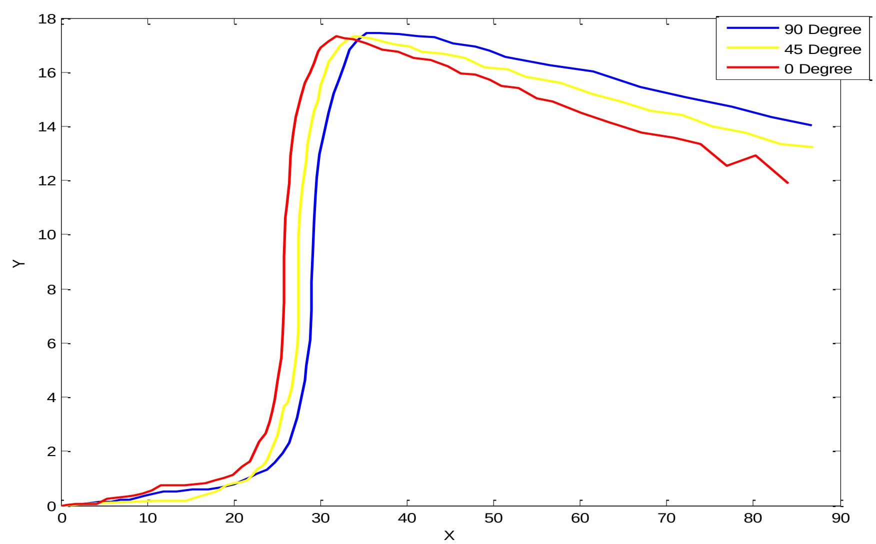


Figure 4.5: Graph of U-bending shape.

Based on the graph figure 4.4 and figure 4.5, both graphs shows the effect of sheet anisotropy on the springback by using U-bending Experimental for the Aluminium with the thickness 1mm. It was show that the specimen 90 degree orientation has more springback effect than the others orientation but the difference between specimens is small and the value almost same. As it shown in both of the figure, increasing the bending direction to the rolling direction resulted in an increase of springback because the grain structure of the sheet metal was different at the different orientation. Figure 4.4(a) and figure 4.4(b) show the effect of this parameter on springback for both angles whether θ_1 and θ_2 is similar. Accordingly, it concluded that the bending of the sheet at 0° orientation was suitable condition for springback reduction in U-die bending process.

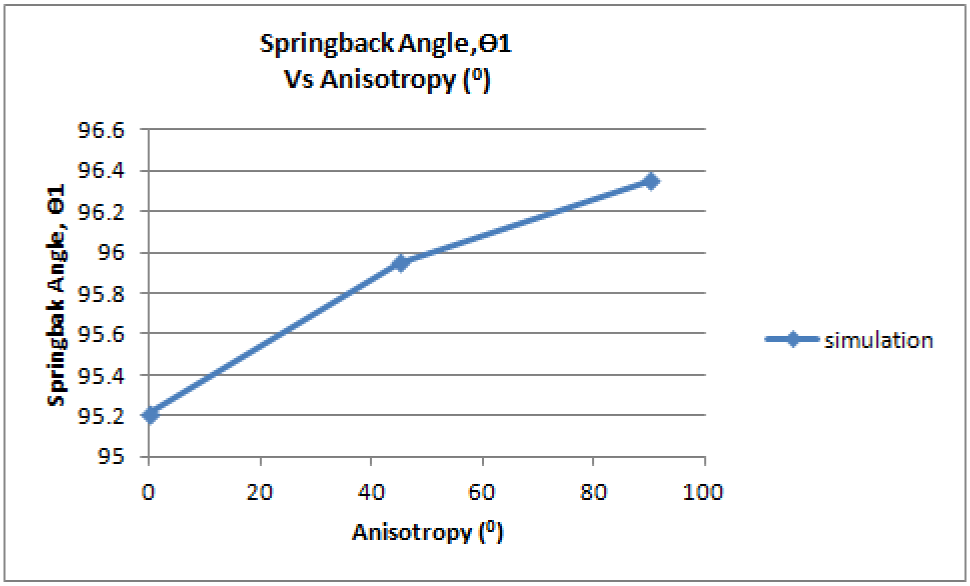
4.4 Finite Element Method

The proposed of the model FEA is to measure springback for U-bending simulation test then the results obtained will be validated with experimental values. A 2D numerical analysis of the U-shaped bending process was carried out for comparison with experimental results. In order to validate materials parameters, both analyses were accomplished with similar operational conditions. This consisted of constant blank holder force of 100 kN. Table 4.4 shown the results of simulation for Angle of Θ_1 and Θ_2.

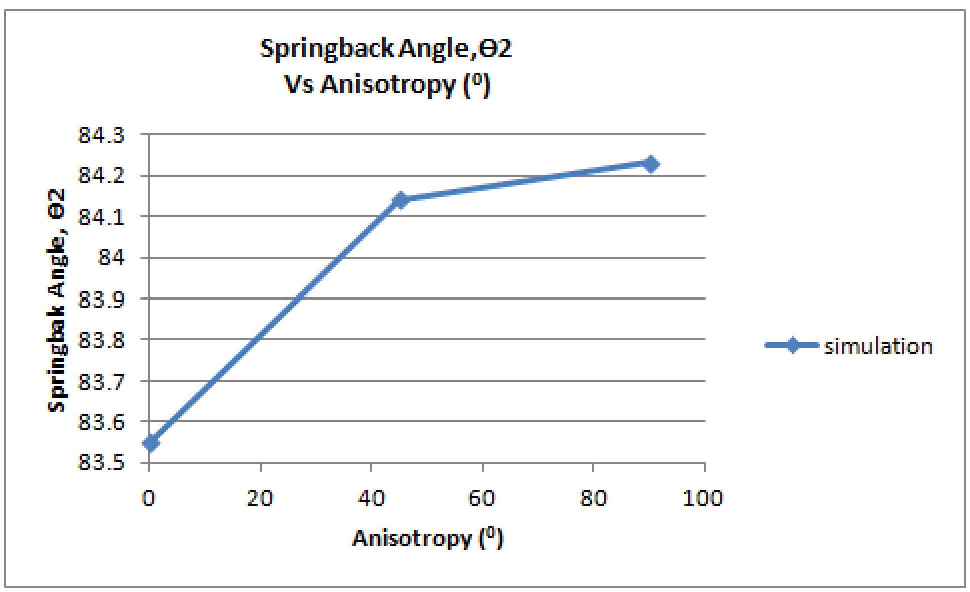
Table 4.4: Finite Element Angle of Θ_1 and Θ_2

Thickness (mm)	**Specimen Angle**	**Orientation Angle (Degree)**	**Bending Angle (Degree)**			**Average Angle (Degree)**
			Sample			
1mm	$\boldsymbol{\theta_1}$	**0**	95.62	94.49	95.52	95.21
		45	94.60	96.50	96.76	95.95
		90	97.30	94.84	96.90	96.35
	$\boldsymbol{\theta_2}$	**0**	83.59	83.55	83.52	83.55
		45	83.26	84.20	84.96	84.14
		90	84.93	83.27	84.50	84.23

Based on this result, the average of springback angle was calculated and the graph of springback versus anisotropy is plotting. The figure 4.6 was show the graph of Finite Element springback versus anisotropy for Θ_1 and Θ_2.



(a)



(b)

Figure 4.6: Finite Element Springback versus Anisotropy (a) for θ_1 (b) for θ_2

The graph in figure 4.6(a) was shown the results simulation of springback versus Anisotropy for θ_1. Based on the graph, the springback angle increased when the anisotropy value was increased. This finding was good agreement with the previous study by M. Bakhshi-jooybari (2009). The lower springback are happen at the 0 degree orientation, which is the angle of springback only 95.21°. The higher of springback occur at 90 degree orientation which is having the average angle of springback 96.35°. At the middle, for orientation 45 degree the average deformation angle of springback is 95.95°. These phenomena also happen in figure 4.6(b) that shows the springback versus anisotropy for θ_2. The higher angle deformation of springback is at 90 degree which is 84.23°. For the 45 degree orientation, the average angle is 84.14° and the lower springback occur at 0 degree orientation which is only 83.55°.

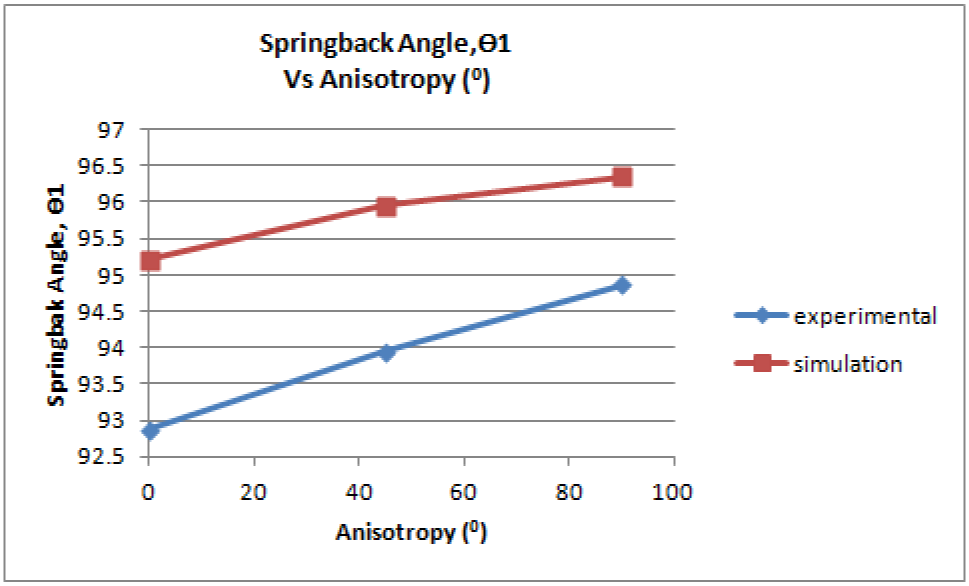
Both of graph resulted the anisotropy are influence the angle of springback occur. Decreases of the rolling direction, will reduce the angle of springback occur in bending process. In order to investigate the effect of parameter in deformation of springback, the anisotropy value must be considered. It was concluded that the bending of the sheet at orientation 0° was suitable condition for springback reduction in U-die bending process.

4.5 Comparison Experimental and Finite Element Method

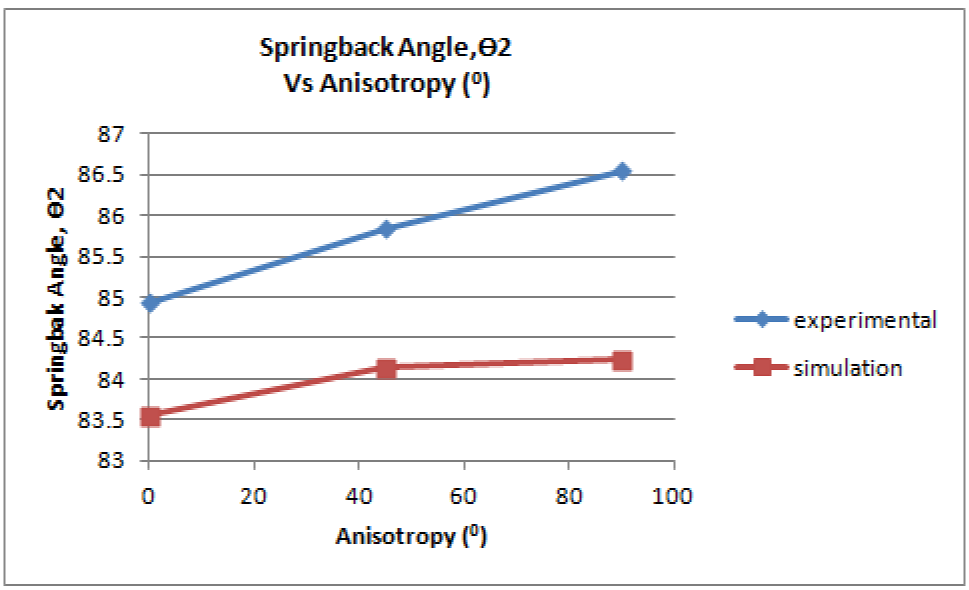
Table 4.5 was shown the result of springback angle, Θ_1 and Θ_2 for both FE Simulation and experimental.

Table 4.5: Springback values for Experimental and Finite Element.

Anisotropy	**0^0**		**45^0**		**90^0**	
	Θ1	Θ2	Θ1	Θ2	Θ1	Θ2
Experimental	92.87	84.93	93.94	85.84	94.86	86.54
FE Simulation	95.21	83.55	95.95	84.14	96.35	84.23
Percentage of Error	2.52	1.62	2.14	1.98	1.57	2.67



(a)



(b)

Figure 4.7(b): Comparison Finite Element with Experimental result a) for θ_1 b) for θ_2

The graph in figure 4.7(a) was shown the comparison results FE simulation with experimental of springback versus Anisotropy for θ_1. Based on the graph, both method shows the springback angle increased when the anisotropy value was increase. The graph presented the same pattern but with the different value. The percentage of these differences was calculated to define the percentage of error (POE) for both methods as shown in table 4.5. For the 0 degree of orientation, the POE is 2.52 % and the others are 2.14 % for 45 degree and 1.57% for 90 degree.

For the graph in figure 4.7(b), it was shown the comparison results simulation with experimental of springback versus Anisotropy for θ_2. Same as in figure 4.7(a), this graph resulted both method presented the increased of anisotropy value are increased the angle of springback. The percentage of error for 0 degree is 1.62%. For the others are 1.98% for 45 degree and 2.67% for the 90 degree of orientation.

Both of graph from the figure 4.7(a) and figure 4.7(b) show the comparison of the results simulation with experimental of springback versus Anisotropy for θ_1 and θ_2. The percentage of error between experiment and simulation is reasonable. The graphs are resulted the increasing of anisotropy are increased the angle of springback for both method. According the graph, the similarity of the pattern graph was representing the validity of the simulation in order to replace the experimental method to designing a die. As the percentage of error is lower than 5% for each orientation, the simulation method is may be reliable to use in industry as the one of method to designing.

CHAPTER 5

CONCLUSION

5.1 Introduction

This chapter summarized conclusion and recommendations for the overall objectives of the project based on comparison of experiment and finite element analysis.

5.2 Conclusions

The conclusion made to answer the objectives in this project. This thesis concludes that:

I. The effects of anisotropy parameter which is from the rolling direction of the aluminium with the thickness 1mm on the amount of springback are presented. The results represented the springback increases when the anisotropy increased. The results observed by two method which are finite element method and experimental method. Both of these methods represented same result. The results also were good agreement with the literature review.

II. The bending angles to the rolling direction are influence the springback occurs which is the higher of the anisotropy proved the higher of spring back of aluminium. Based on the experiment result, at the 0 degree orientation have resulted the lowest of springback which is 92.87° for θ_1 and 84.93° for θ_2. It was found the 0° orientation provides a suitable condition in U-die bending process.

III. According to the results obtain from the experiment and simulation, different spring-back values from both tasks is quite similar. The higher percentage of error is in θ_2 at 90° rolling direction which is 2.67% and it still lowest than 5% percentage of error. Those Finite element analyses can be used in order to predict the spring-back on u-bending.

The objective of this project had been achieved since the value of spring-back is good agreement from the previous research and both experimental and simulation also shown almost the same value. The finding from this project is the finite element method might be use in order to predict the springback angle of die designing.

5.3 Recommendations

In order to improve the study of spring-back prediction of Aluminium on U-die bending process for future study, there is some aspect should be taken into consideration:

a) Make the comparison deformation of springback with different material such as mild steel and stainless steel.

b) Study the others parameter that will affect the deformation of springback such as blank holder force, punch tip radius, and the thickness.

c) Study the meshing effect to predict the spring-back by using a different mesh in simulation and choose the result that has a nearest value with the experiment

d) Study the effect of the temperature with same thickness. This because the temperature is one of the parameter will influence the strength of the material. It will change the grain structure of the material.

e) Use 3D model in Finite Element Analysis in order to determine springback angle because 3D model is more effective.

REFERENCES

Boljanovic, Vukota, 2004. Sheet Metal Forming Processes and Die Design.

Gau J.A, 1999. Study of the Influence of the Bauschinger Effect on the Springback in the two-dimensional Sheet Metal Forming.

Gomes C, Onipede O, Lovell M, 2005. Investigation of Springback in High Strength Anisotropic.

Hakim S. Sultan, 2004. Experimental Investigation and Finite Element Analysis of Composite Conical Structures.

Hakim S. Sultan Aljibori, Abdel Magid Hamouda, 2009. Finite Element Analysis of Sheet Metal Forming Process.

Jianfeng Wang, B.S, M.S, 2004. Principles of The Draw-Bend Springback Test.

Liu, G., Z.Q. Lin, W.L Xu and Y.X. Bao, 2002. Variable blankholder force in U-shaped part forming for eliminating springback error.

M. Bakhshi-Jooybari, B.Rahmani, V. Daeezadeh, A. Gorji, 2008. The Study of Springback of CK67 Steel Sheet in V-die and U-die Bending Processes.

M.Firat, 2005. U-Channel Forming Analysis With an Emphasis on Springback Deformation.

M.Kadkhodayan, I. Zafarparandeh, 2008. U-Bending Analysis With An Emphasis on Influence of Hardening Models.

M.Tisza, 2004. Numerical Modelling and Simulation in Sheet Metal Forming.

Mehmet Firat, Bilgin Kaftanoglu, Orhan Eser, 2006. Sheet Metal Forming Analysis With an Emphasis on The Springback Deformation.

Micari F. Forcellese A, Fratini L, Gabrielli F, Alberti N, 1997. Springback Evaluation in Fully 3-D Sheet Metal Forming Processes.

Michael F. Ashby and David R.H Jones, 2005. Engineering of Materials 1.

Mohamed Faraj Alfaidi, Li Xiaoxing, 2009. Determination of Springback in Sheet Metal Forming.

Papeleux L, Ponthot JP, 2002. Finite Element Simulation of Springback in Sheet Metal Forming.

Samuel, M., 2000. Experimental and numerical prediction of springback and side wall curl in U-bendings of anisotropic sheet metals, J Mater Process Technol., 105: 382-393.

Vial C, Caddlell.R.M, Hosford.W.F, 1993. Yield Loci of Anisotropic Sheet Metals

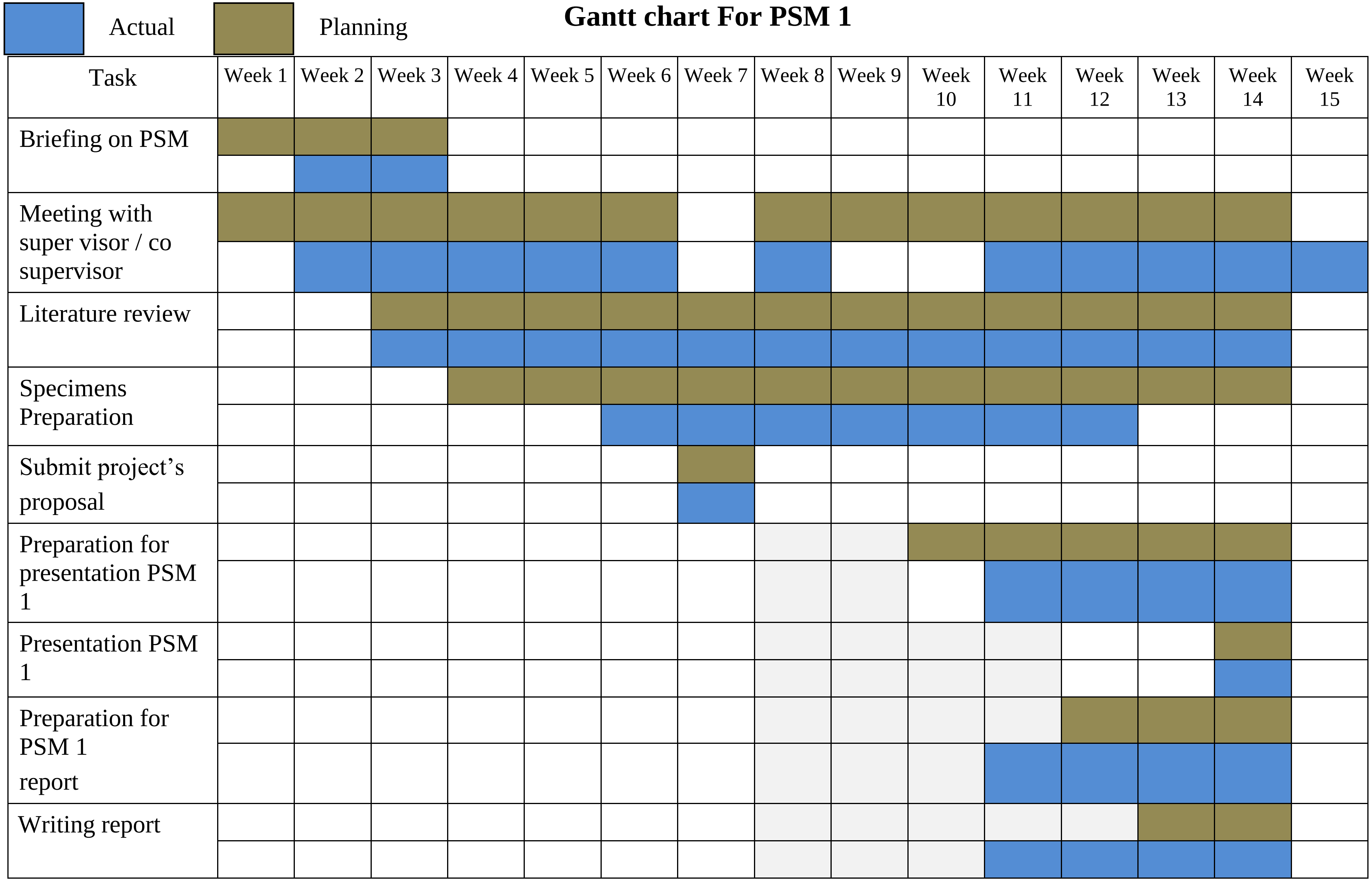
Volkan Esat, Haluk Darendeliler, Mustafa Ilhan Gokler, 2001. Finite Element Analysis of Springback in Bending of Aluminium Sheets

Wang, N. and Somaratna. N., 1989. Numerical Simulation of Industrial Sheet Forming Processes.

Zhanga, D.J., Z.S. Cui, X.Y. Ruan and Y.Q. Li, 2007. An analytical model for predicting springback and side wall curl of sheet after U-bending, Comp Mater Science, 38(4): 707-715.

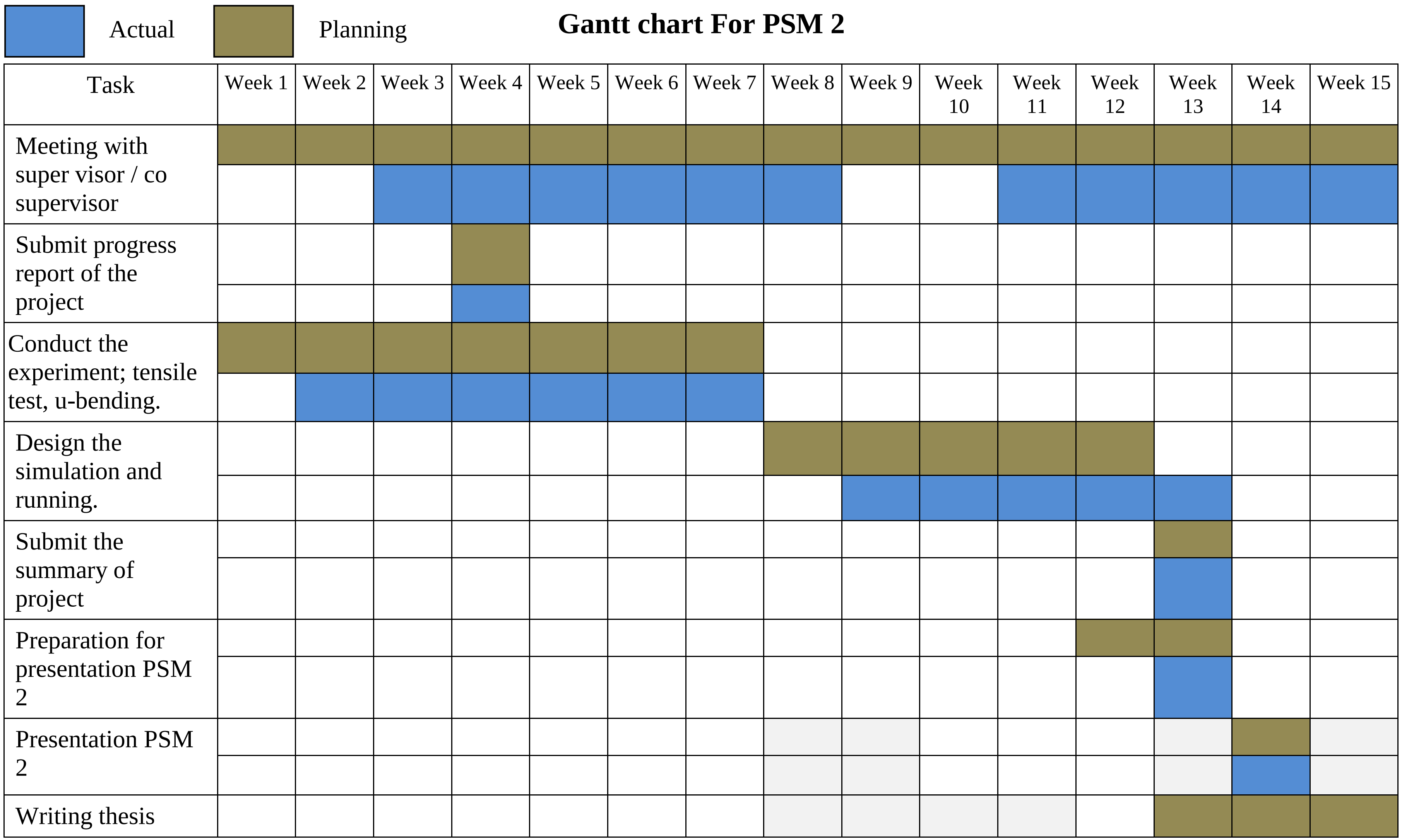
APPENDICES A

Gantt chart For PSM 1



APPENDICES B

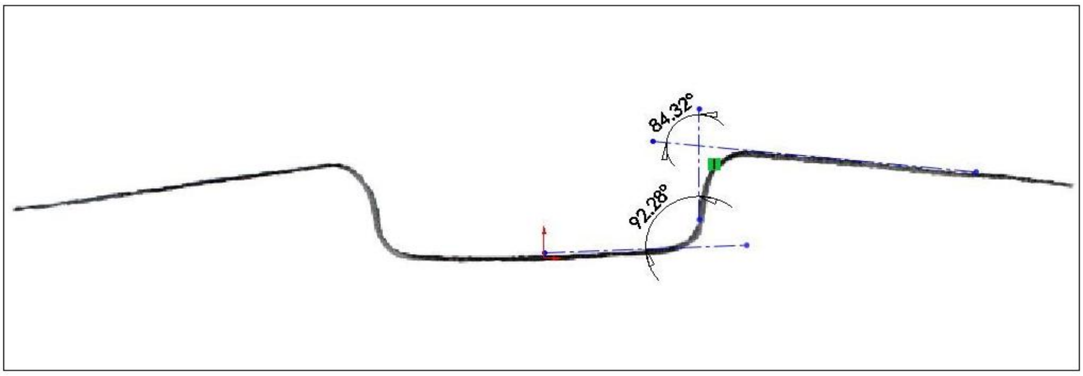
Gantt chart For PSM 2



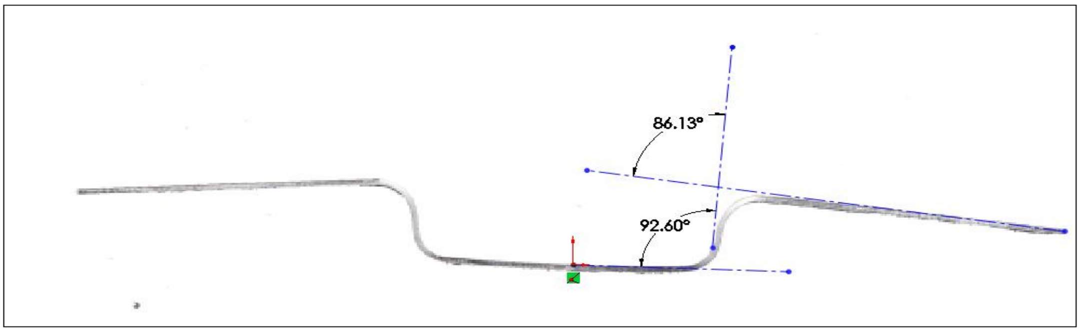
APPENDIX C

1. Measuring Angle of Experimental Parts

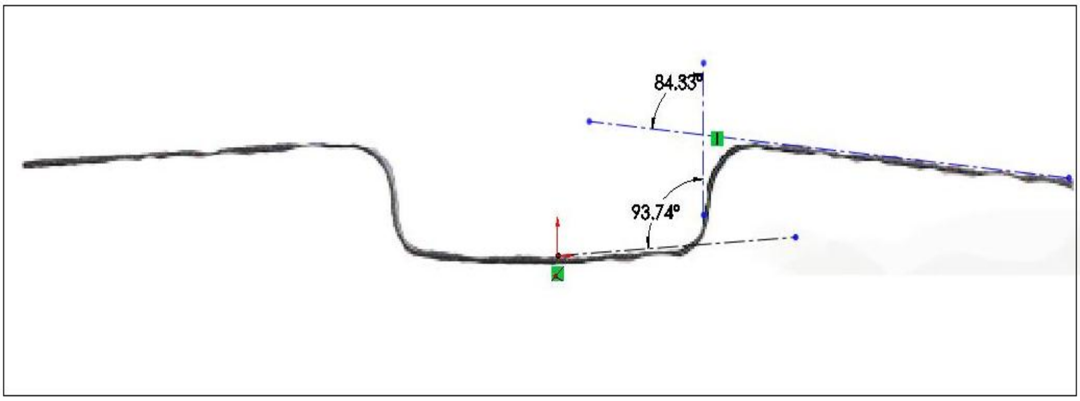
i. 0 degree



Specimen 1

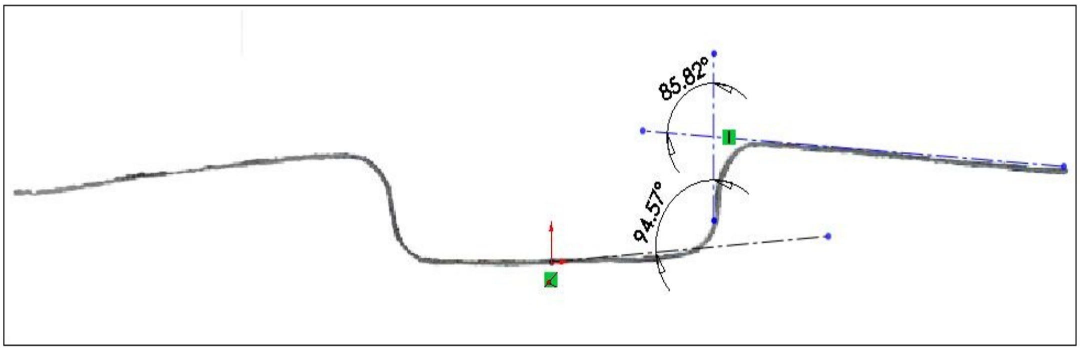


Specimen 2

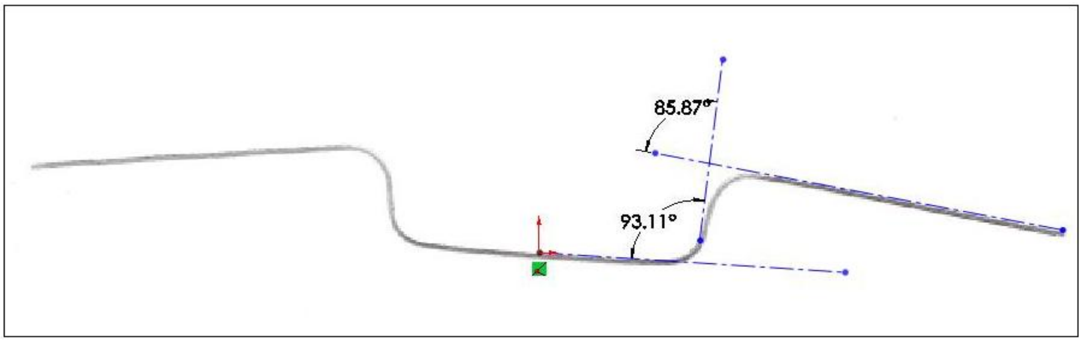


Specimen 3

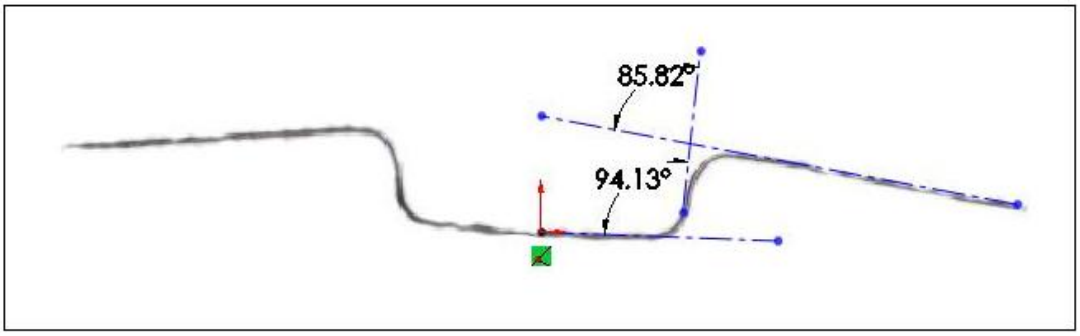
ii. 45 degree



Specimen 1

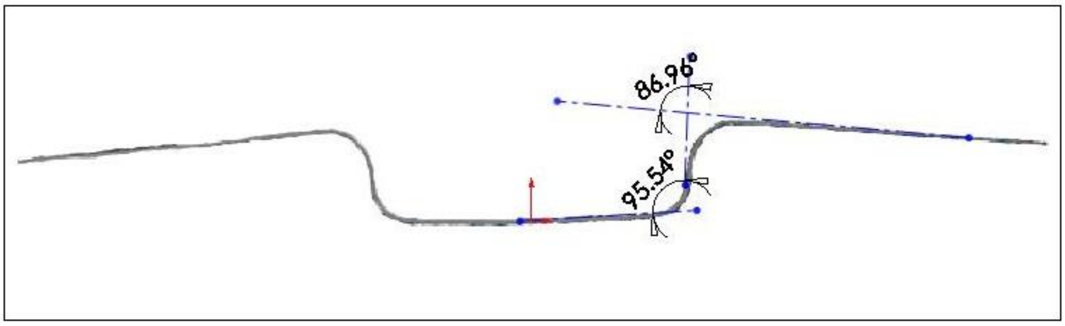


Specimen 2

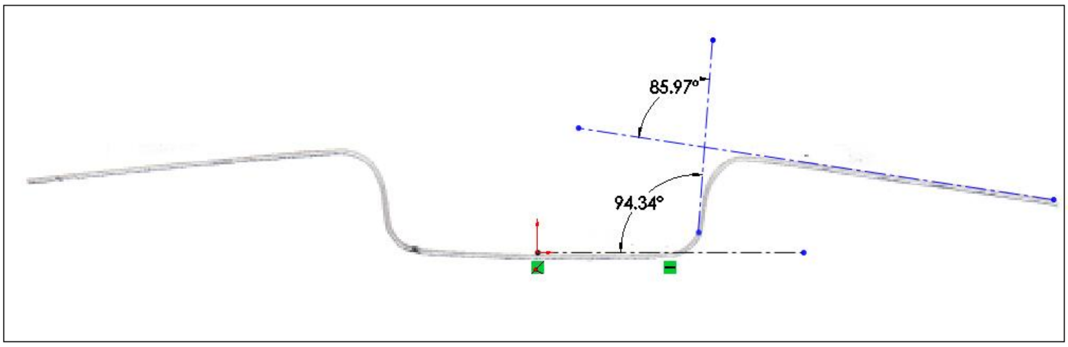


Specimen 3

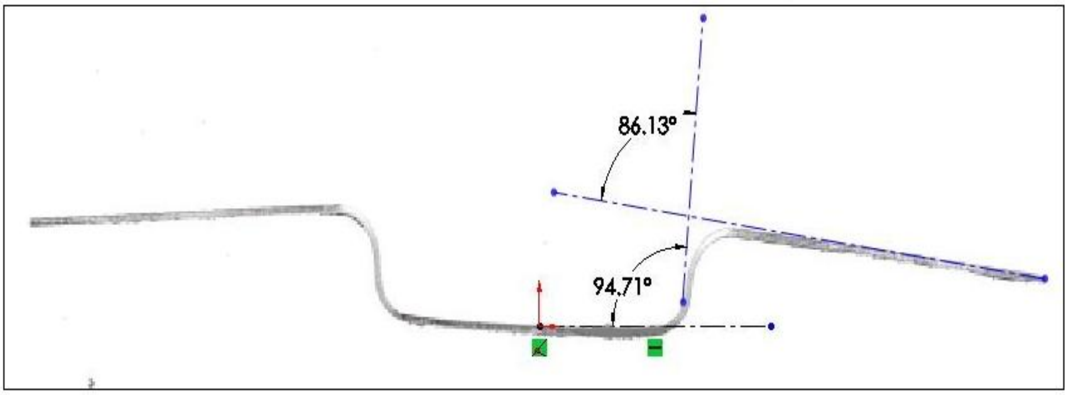
iii. 90 degree



Specimen 1



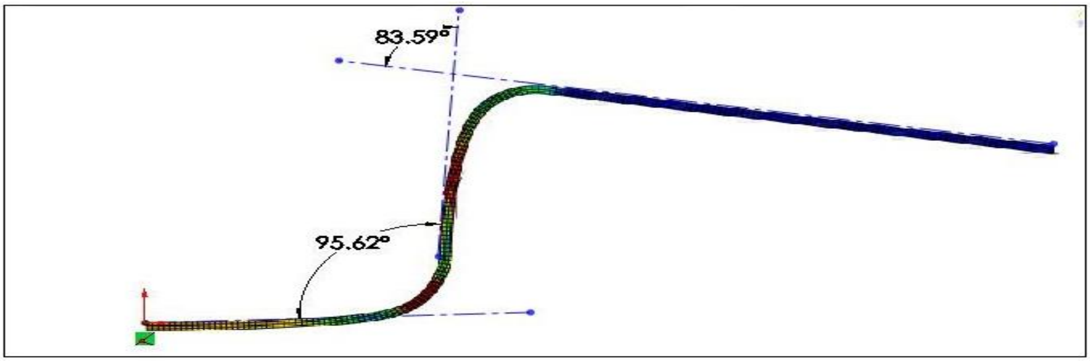
Specimen 2



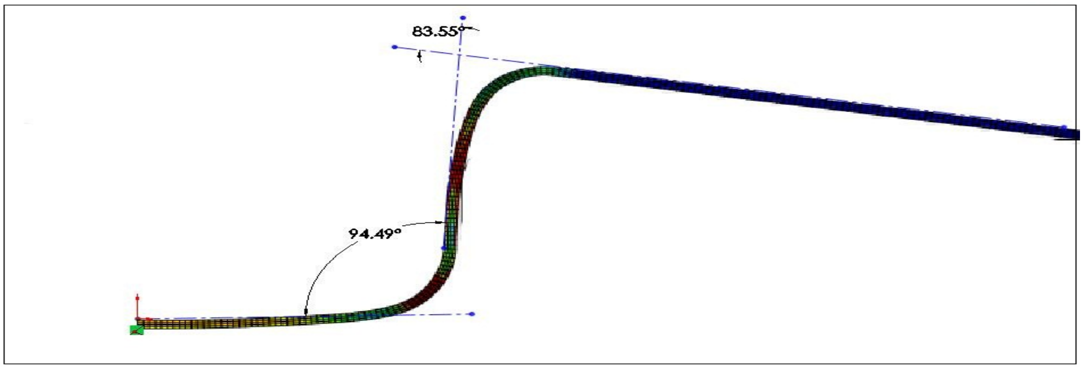
Specimen 3

2. Measuring Angle of Finite Element Method

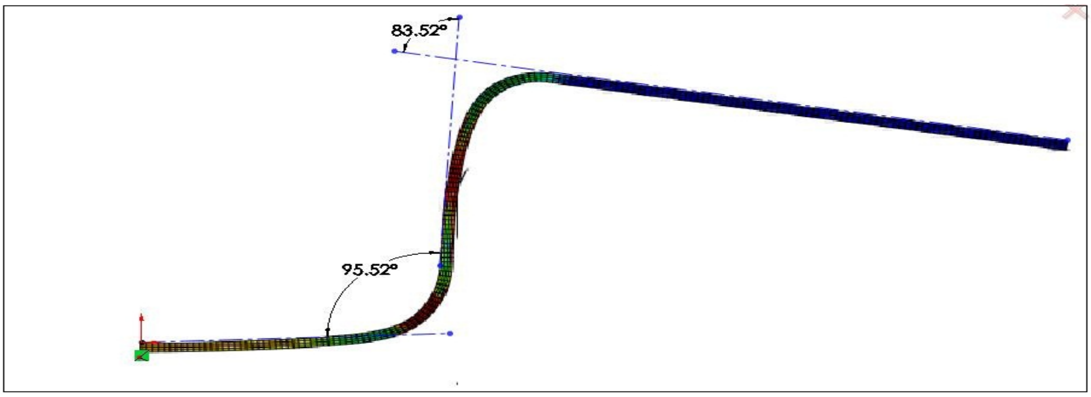
i. 0 degree



Specimen 1

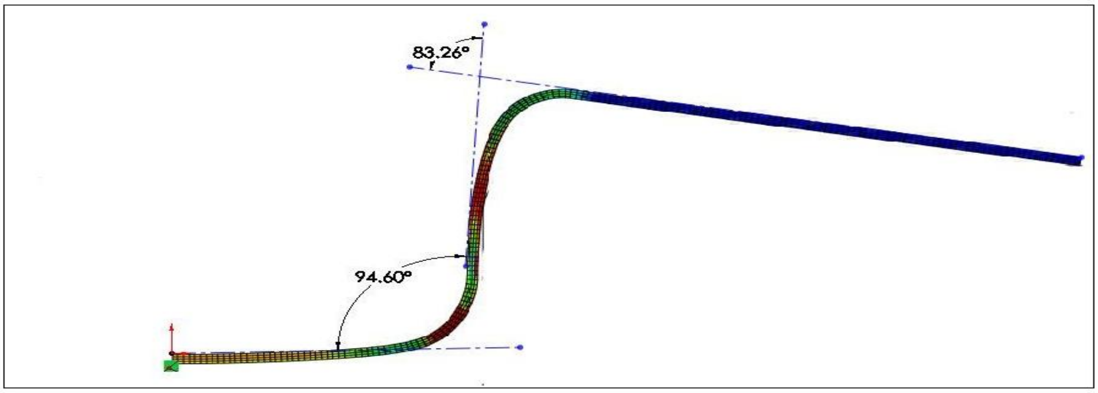


Specimen 2

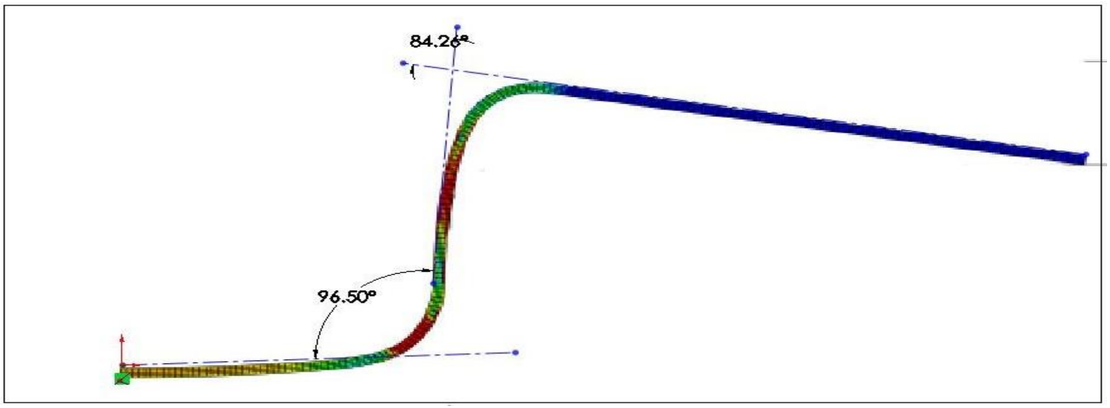


Specimen 3

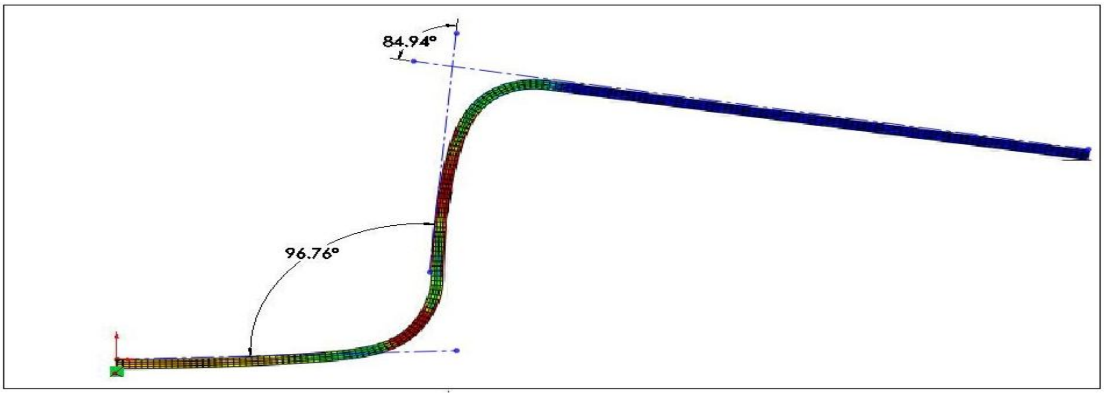
ii. 45 degree



Specimen 1

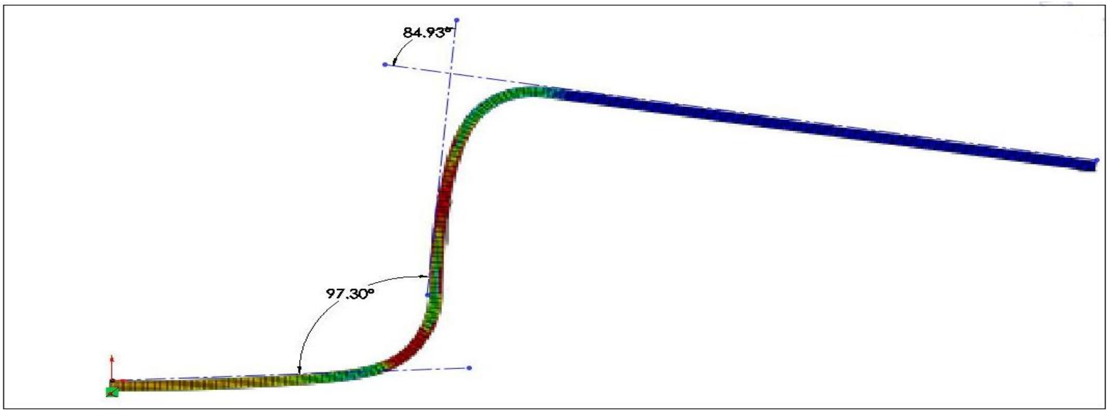


Specimen 2

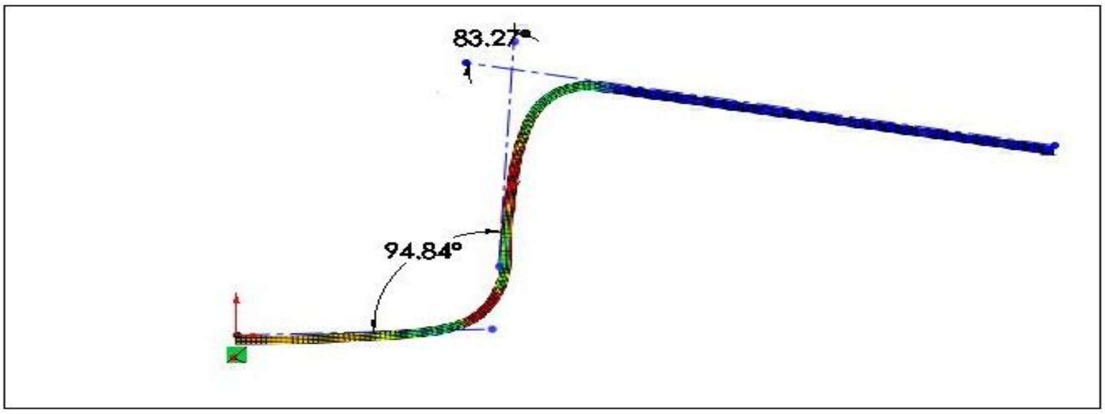


Specimen 3

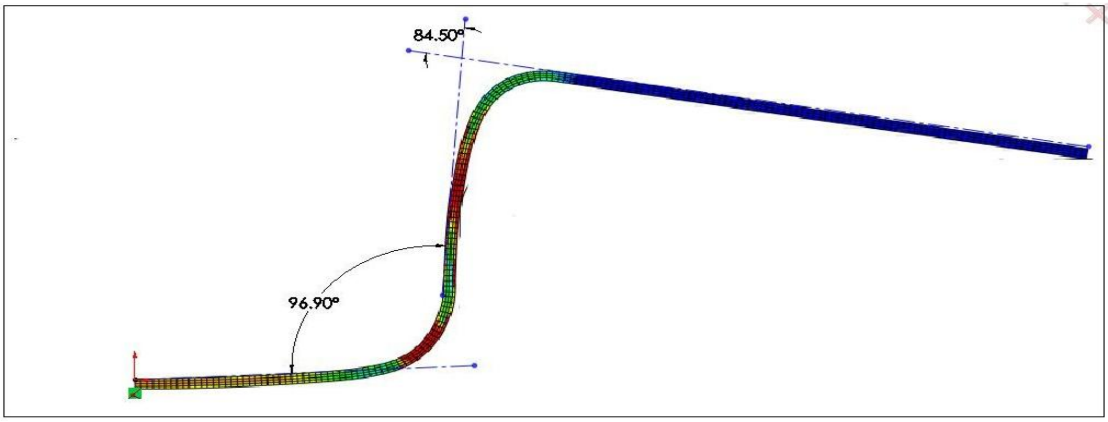
iii. 90 degree



Specimen 1



Specimen 2



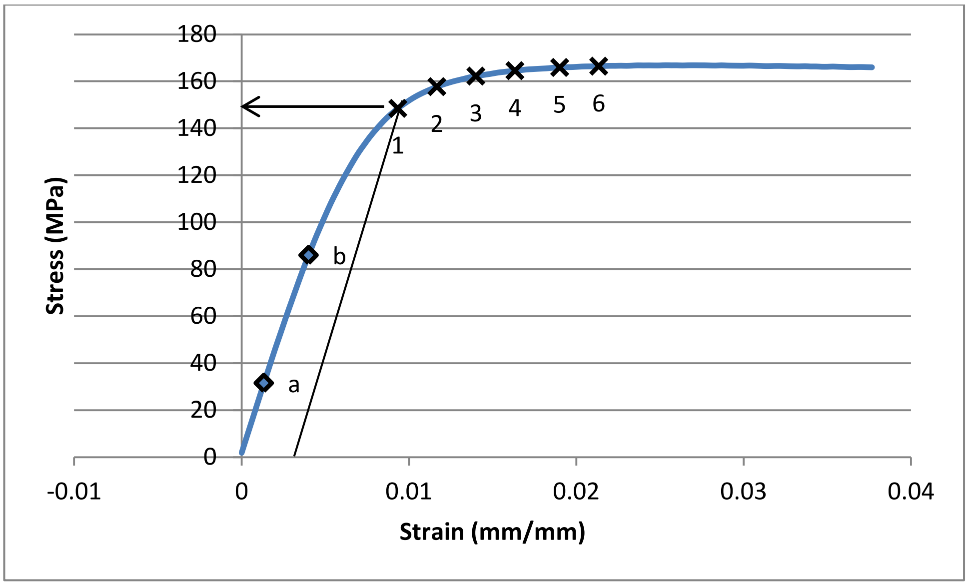
Specimen 3

APPENDIX D

1. Material Properties from Curve Stress-Strain Tensile Test.

i. 0 degree

- Specimen 1



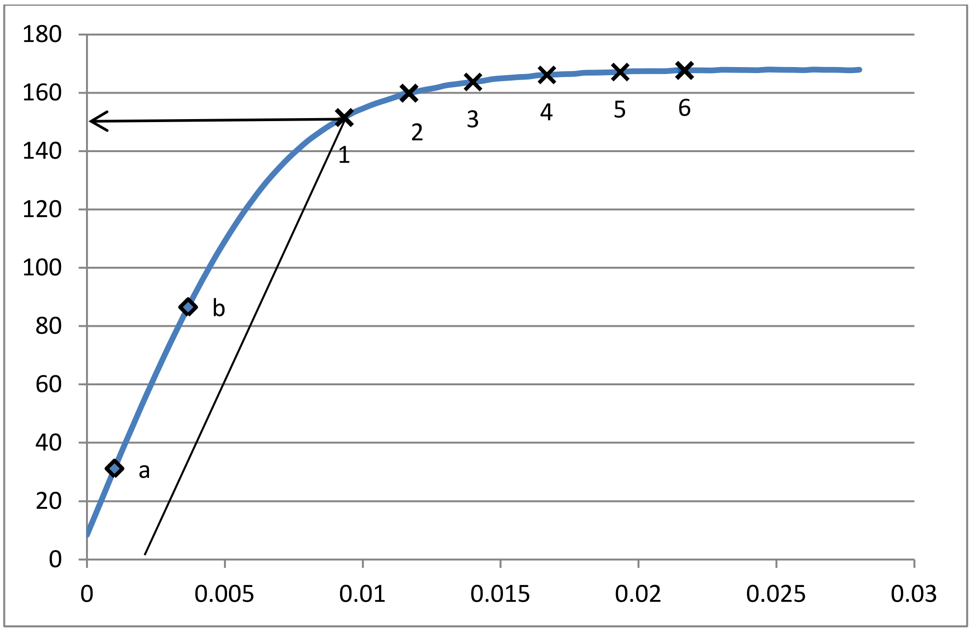
Plasticity Stress strain =

Points	Strain	Stress
1	0	148.5088
2	0.002333	157.736
3	0.004669	162.224
4	0.006999	164.516
5	0.009669	165.908
6	0.011999	166.4448

Point a= (0.0013312, 31.4776)
Point b=(0.0040012, 85.9544)

Young's Modulus=20403.2958 MPa
Yield Strength =148.5088 MPa

- Specimen 2



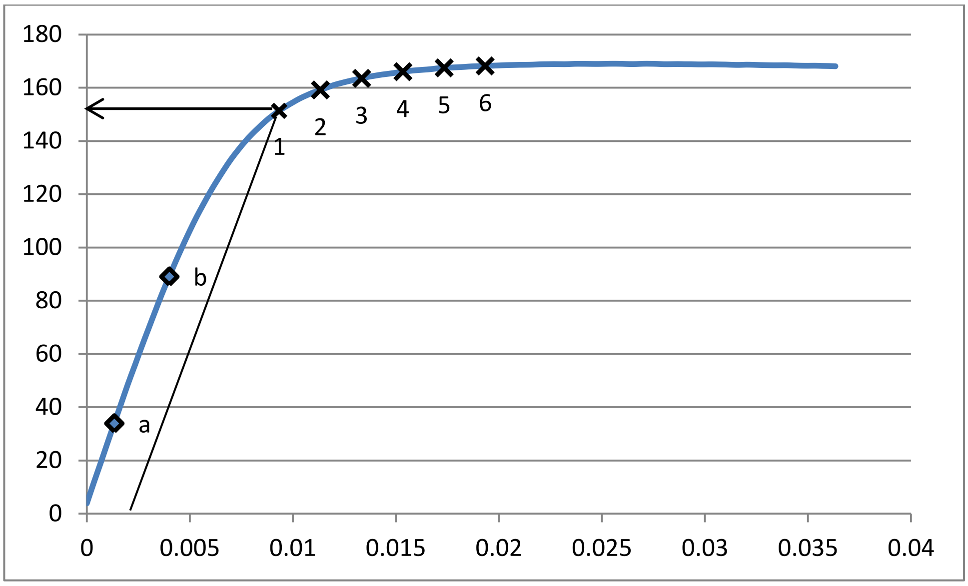
Plasticity Stress-Strain=

Point	Strain	Stress
1	0	151.536
2	0.002341	159.832
3	0.004671	163.736
4	0.007342	166.1328
5	0.01	167.088
6	0.012342	167.6568

Point a= (0.009962, 31.1296)
Point b= (0.0036662, 86.5392)

Young's Modulus= 20752.6592 MPa
Yield Strength= 151.536 Mpa

- Specimen 3



Plasticity Stress-Strain

Point	Strain	Stress
1	0	151.2936
2	0.001999	159.1512
3	0.003999	163.4888
4	0.006	165.9712
5	0.008004	167.4
6	0.010001	168.1816

Point a= (0.0013312, 33.9192)

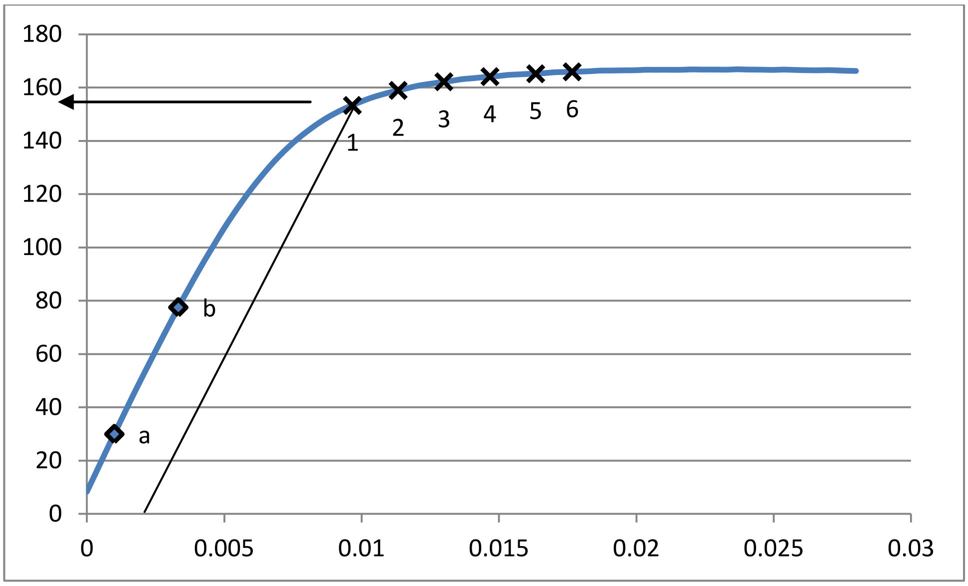
Point b=(0.0040024, 89.076)

Young’s Modulus= 20648.6972 MPa

Yield Strength= 151.2936 MPa

ii. 45 degree

- Specimen 1



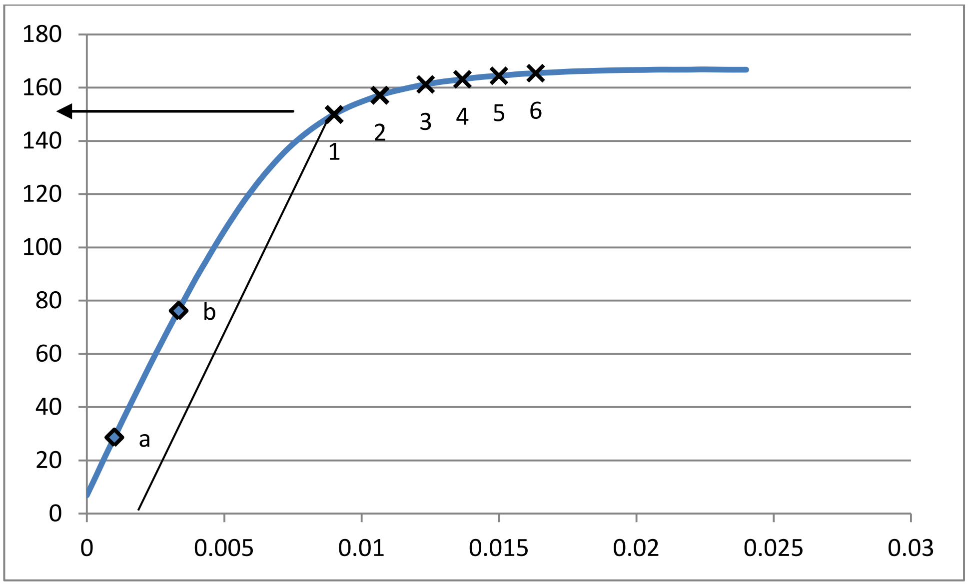
Plasticity Stress-Strain

Points	Strain	Stress
1	0	153.3296
2	0.001666	158.912
3	0.003333	162.1184
4	0.005	164.0312
5	0.006666	165.1984
6	0.007999	165.8968

Point a= (0.0009974, 29.9312)
Point b= (0.00333, 77.4992)

Young's Modulus= 20392.6949 MPa
Yield Strength= 153.3296 MPa

- Specimen 2



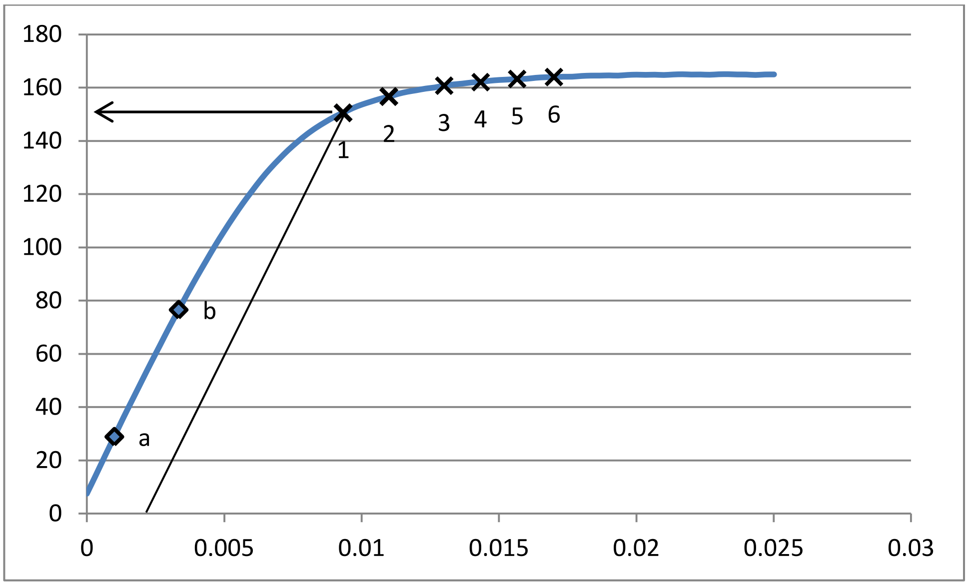
Plasticity Stress-Strain

Point	Strain	Stress
1	0	149.9304
2	0.009774	157.1144
3	0.011434	161.176
4	0.012767	163.1256
5	0.0141	164.4504
6	0.015434	165.3544

Point a= (0.0010038, 29.9312)
Point b= (0.00333, 77.4992)

Young's Modulus= 20448.7554 MPa
Yield Strength= 149.9304 MPa

- Specimen 3



Plasticity Stress-Strain=

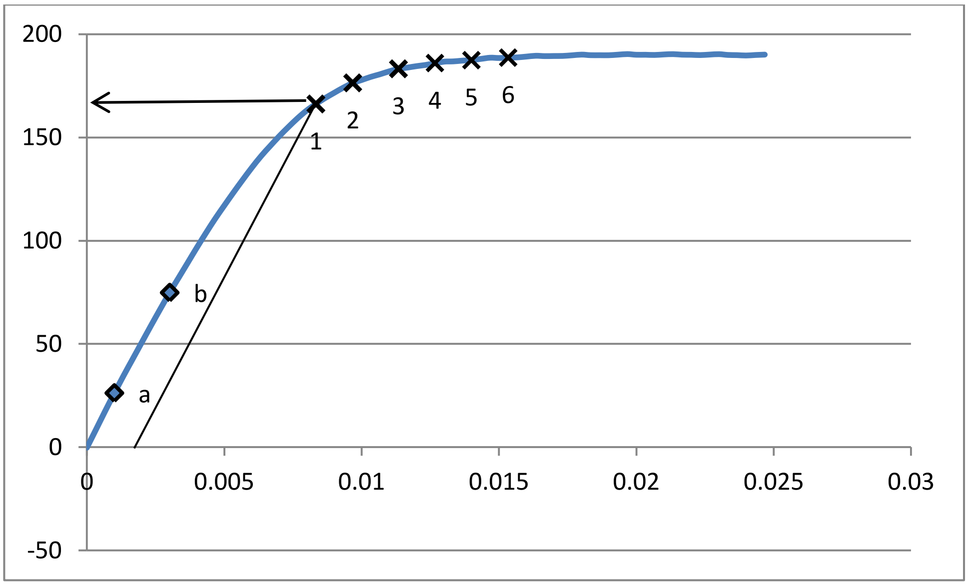
Point	Strain	Stress
1	0	150.5464
2	0.001668	156.6032
3	0.003674	160.7296
4	0.005	162.0944
5	0.006338	163.2376
6	0.00767	163.9088

Point a= (0.000995, 28.8704)
Point b= (0.0033388, 76.5072)

Young's Modulus= 20324.6011 MPa
Yield Strength= 150.5464 MPa

iii. 90 degree

- Specimen 1



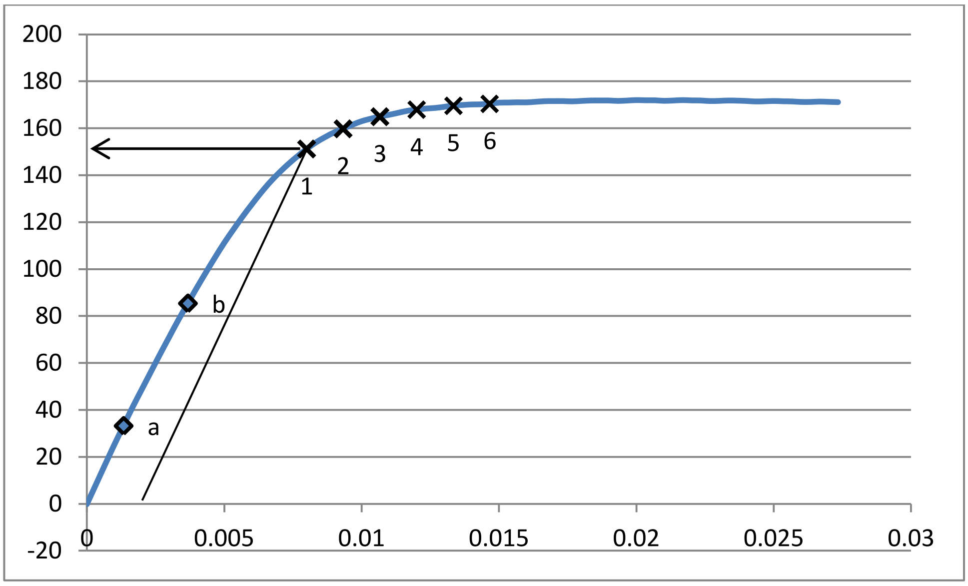
Plasticity Stress-Strain=

Point	Strain	Stress
1	0	166.1632
2	0.001345	176.336
3	0.003009	183.192
4	0.004332	185.9168
5	0.005659	187.4592
6	0.006997	188.6424

Point a= (0.001, 26.1816)
Point b= (0.003412, 74.8006)

Young's Modulus = 16696.1539 MPa
Yield Strength = 166.1632 MPa

- Specimen 2



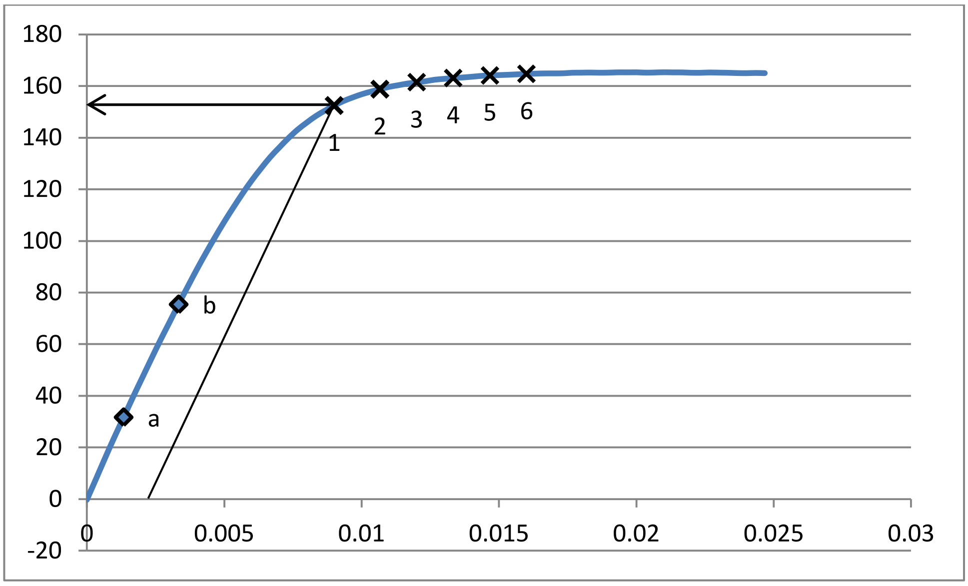
Plasticity Stress-Strain=

Point	Strain	Stress
1	0	151.1016
2	0.001331	159.7744
3	0.002673	164.9784
4	0.004009	167.86
5	0.005341	169.5968
6	0.006666	170.4368

Point a = (0.0013312, 33.2168)
Point b = (0.0036721, 85.3232)

Young's Modulus= 22259.1311 MPa
Yield Strength= 151.1016 MPa

- Specimen 3



Plasticity Stress-Strain=

Point	Strain	Stress
1	0	152.436
2	0.001668	158.7872
3	0.003001	161.4384
4	0.004336	163.132
5	0.005671	164.1432
6	0.007004	164.7192

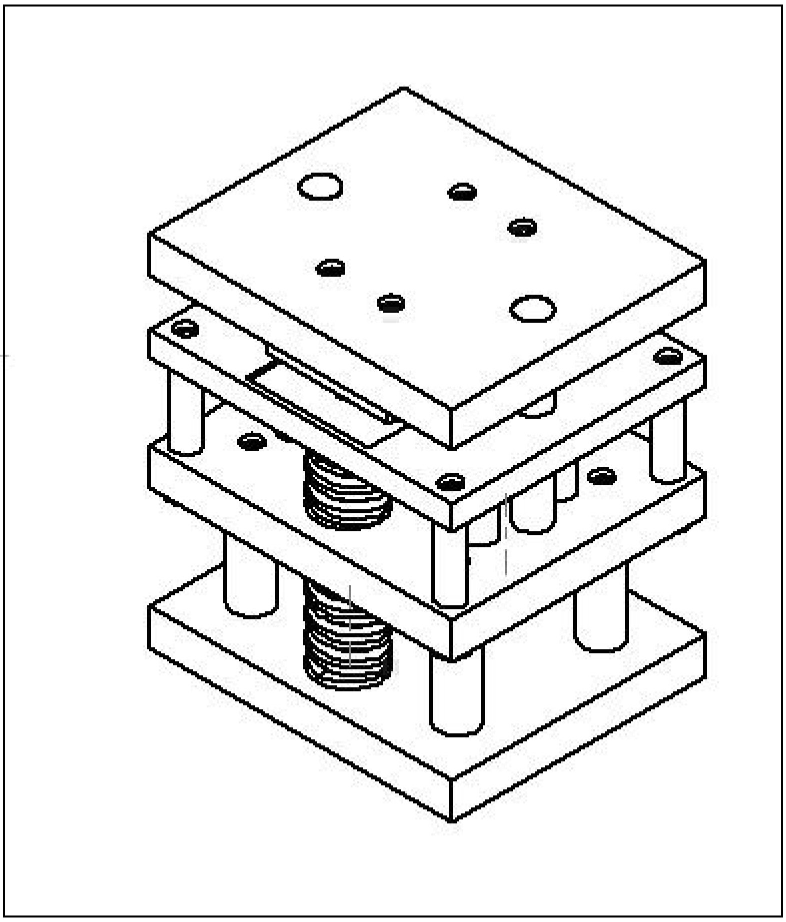
Point a= (0.0013362, 31.6552)
Point b= (0.0033338, 75.4456)

Young's Modulus= 21921.5058 MPa
Yield Strength= 152.436 MPa

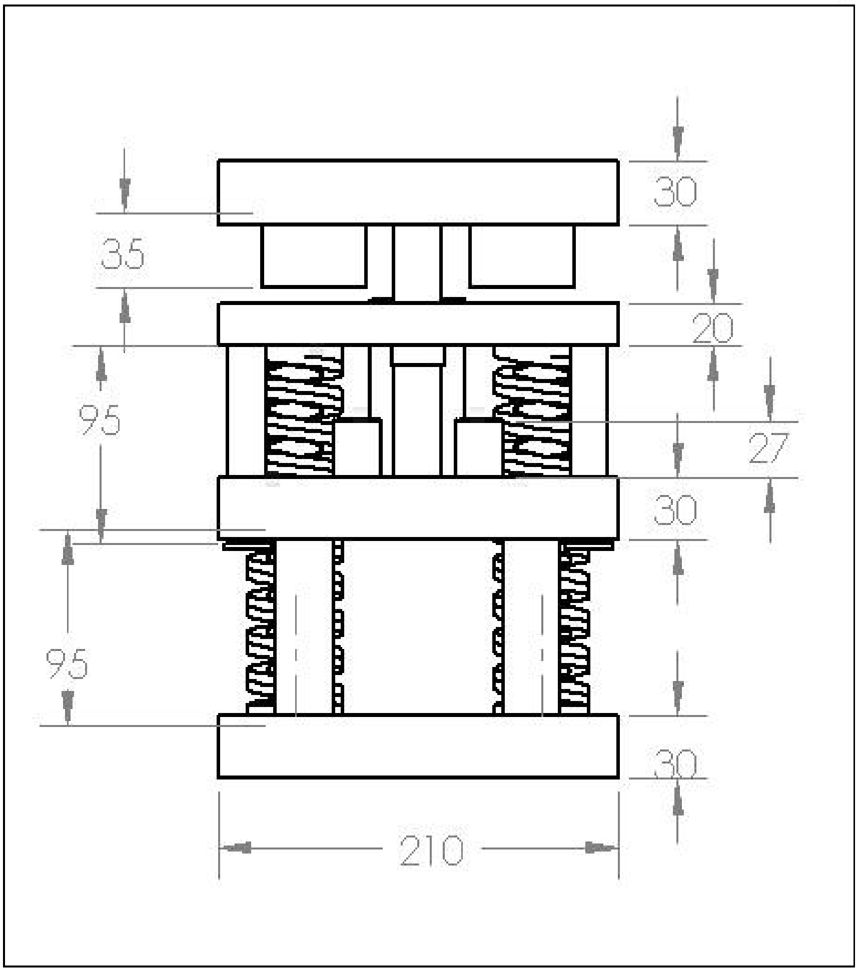
APPENDIX E

1. U-Die Bending experiment view from solidworks.

i) Isometric



ii) Right side



iii) Front Side