EXPERIMENTAL AND FINITE ELEMENT EVALUATION OF BENDING FOR GALVANIZED IRON

FAKHRURRAZI B AB KARIM

Report submitted in partial fulfillment of the requirements

for the award of Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering

UNIVERSITI MALAYSIA PAHANG

JAN 2013

UNIVERSITI MALAYSIA PAHANG

FACULTY OF MECHANICAL ENGINEERING

I certify that the project entitled “Experimental and Finite Element Evaluation of Bending For Galvanized Iron” is written by Fakhrurrazi b Ab Karim. I have examined the final copy of this project and in our opinion, it is fully adequate in terms of scope and quality for the award of the degree of Bachelor of Engineering. I herewith recommend that it be accepted in partial fulfillment of the requirements for the degree of Bachelor of Mechanical Engineering.

Examiner   Signature

SUPERVISOR DECLARATION

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

Signature :………………………………………..

Name of Supervisor : JASRI BIN MOHAMAD

Position : LECTURER

Date :

STUDENT'S DECLARATION

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

Signature :……………………………………………..

Name : FAKHRURRAZI B AB KARIM

ID Number : MA08086

Date :

Dedicated to my father, Mr Hj Ab Karim Bin Hj Daud, my beloved mother, Mrs Hjh Noriah Bte Hj Md Dom , my brothers, my sisters, and last but not least to all my fellow friends.

ACKNOWLEDGEMENT

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

All of my friends at the Universiti Malaysia Pahang made it a convenient place to study. In particular, I would like to thank my lab mates, Azrulhafiq bin Mohd Azizi, Mohd Hafizu bin Hasan Basri, Mohd Jazmir bin Hassan and all of the staff of the Mechanical Engineering Department, UMP who helped me in many ways.

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

ABSTRACT

Finite element evaluation is one of the methods in predicting the springback angle in sheet metal bending. Predicting the springback is important since to produce the accuracy part geometry, the design of the die and bending tool must be accurate. This thesis aims to evaluate the reliability of finite element method by comparing the results with experimental results. The effect of parameters such as anisotropy in springback also has been studied. Abaqus software has been used to simulate the bending process and the mechanical properties provided from tensile test will be used to run the simulation. In the U-bending experiment, the die were clamped on stamping machine and the Galvanized Iron sheets then have been bent before the springback being measured with SolidWorks software. The results from the experiment and simulation is slightly different for the springback angle Θ_1 , which the simulation shows increasing the orientation will increase the springback, and for the experimental, the springback higher at 0 degree orientation angle, and lower at 45 degree. For the springback angle Θ_2 , the simulation and experimental result show that increasing the orientation angle will increase the amount of springback. Finite element method can be used to predict the springback since the results are nearly the same and percentages of error are below 10 %. It can be comprehended that the finite element method are suitable method to predict the springback angle of sheet metal bending. The finite element method is important in the future.

ABSTRAK

Kaedah analisis simulasi merupakan salah satu kaedah untuk meramal pembentukan bangkit kembali dalam pembengkokan kepingan logam. Ramalan bangkit kembali amat penting kerana untuk menghasilkan produk yang tepat, reka bentuk alat acuan dan peralatan pembengkokan mestilah tepat. Laporan ini bertujuan untuk menilai kebolehan kaedah simulasi dengan membandingkan keputusan simulasi dengan keputusan eksperimen. Kesan parameter seperti anisotropi dalam bangkit kembali juga dikaji. Perisian Abaqus telah digunakan untuk mensimulasikan proses lenturan dan sifat-sifat mekanik yang disediakan dalam ujian regangan akan digunakan untuk menjalankan simulasi. Dalam eksperimen lenturan-U, alat acuan telah dikapit pada mesin tekanan dan kepingan “ Galvanized Iron “ kemudian dibengkokan sebelum bangkit kembali diukur dengan perisian Solidworks. Hasil daripada eksperimen dan simulasi adalah sedikit berbeza untuk sudut bangkit kembali, Θ_1 yang mana simulasi menunjukkan peningkatan sudut orientasi akan meningkatkan bangkit kembali dan untuk eksperimen, bangkit kembali lebih tinggi pada sudut orientatasi 0 darjah dan lebih rendah pada 45 darjah. Bagi sudut bangkit kembali Θ_2 , simulasi dan hasil eksperimen menunjukkan bahawa peningkatan sudut orientasi akan meningkatkan jumlah bangkit kembali. Kaedah simulasi boleh digunakan untuk meramal bangkit kembali kerana corak graf adalah hamper sama dan peratusan ralat di bawah 10 % . Dapat difahami bahawa kaedah simulasi sesuai untuk meramal sudut bangkit kembali bagi pembengkokan kepingan logam. Kajian lanjut mengenai parameter yang mempengaruhi proses pembengkokan adalah penting pada masa akan datang.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

σ_2	Variance
Fmax	Bending Force
μ	Mean
ΔΘ /Θ	Spring back ratio
n	Strain hardening exponent
R	Normal anisotropic value
v	Poisson’s ratio
E	Young’s modulus
t	Sheet thickness
ρ	Neutral axis
ΔΘ	Springback angle
I	Inertia moment of cross-section per unit width
M(α)	Bending moment along the bending surface
R_n	Neutral layer radius of the sheet
K	Ultimate tensile strength
w	Die gap
t	Sheet thickness
ΔK	Spring back curvature
M	Bending moment
L	Inertia moment of cross-section

LIST OF ABBREVIATIONS

AISI	American Iron and Steel Institute
ASTM	American Society for Testing and Material
TRIP	Transformation Induced Plasticity
CNC	Computer Numerical Control
UTS	Ultimate Tensile Strength
FEA	Finite Element Analysis
CRES	Called-Corrosion-Resistant

CHAPTER 1

INTRODUCTION

1.1 PROJECT BACKGROUND

Springback commonly happens in manufacturing industries especially in the automobile and aircraft industries. It is occur when existing plasticity in sheet metal. One of the experiments to determine the process is bending process. Bending of sheet metal is one of the widely used processes in manufacturing industries especially in the automobile and aircraft industries. This bending operation is about shaping of sheet metal by straining the metal around a straight axis. A bending operation compresses the interior side of the bend and stretches the exterior side. Bending is most common operations to change the shape of a material by plastically deforming it and depends primarily on the materials type, strength, thickness and part complexity.

Springback is an issue in sheet metal forming processes. The springback is a principle problem when precise components are produced. Elastic energy stored in sheet metal in bending operation is released during unloading and the sheet metal tends to return to its initial state. Thus the dimensions and the shape of component are changed.

Basically, this project deals with experimental and finite element evaluation of bending for galvanic iron. In this project, bending analysis will take spring back as one of the part of bending analysis.

Through this project, bending analysis can be made in term of knowing about springback of galvanic iron material. Many factors affect springback such as types of material, types of bending, and thickness of material and sheet anisotropy.

1.2 PROBLEM STATEMENT

Bending is a process in which a sheet metal is plastically deformed to a curve by predicting the precision in angle. When the material has a tendency to partially return to its original shape because of the elastic recovery of the material, it is called springback. The springback is generally defined as the additional deformation of sheet metal parts after the loading is removed and the influenced factor not only by the tensile and yield strengths, but also by thickness, bend radius and bend angle**.** Springback is a phenomenon of elastic nature determined by the distribution of stress on the section of the form part. In the manufacturing industry, it is still a practical problem to predict the final geometry of the part after springback and to design the appropriate tooling in order to compensate for springback.

1.3 OBJECTIVES

1. To determine the springback angle in sheet metal bending for Galvanized Iron.
2. To determine the influence of anisotropy,R on springback.
3. To determine reliability of Finite Element Method (FEA) in sheet metal by comparing with the experimental results.

1.4 SCOPE OF WORKS

1. To study the basic understanding of springback behaviour from the past researchers (Literature Review).
2. To conduct experiments of tensile test to determine mechanical properties of Galvanized Iron.
3. To perform Finite Element Evaluation analysis of bending for Galvanized Iron.
4. To conduct experiments of sheet metal bending.
5. To analyze and compare the simulation and experimental result.

CHAPTER 2

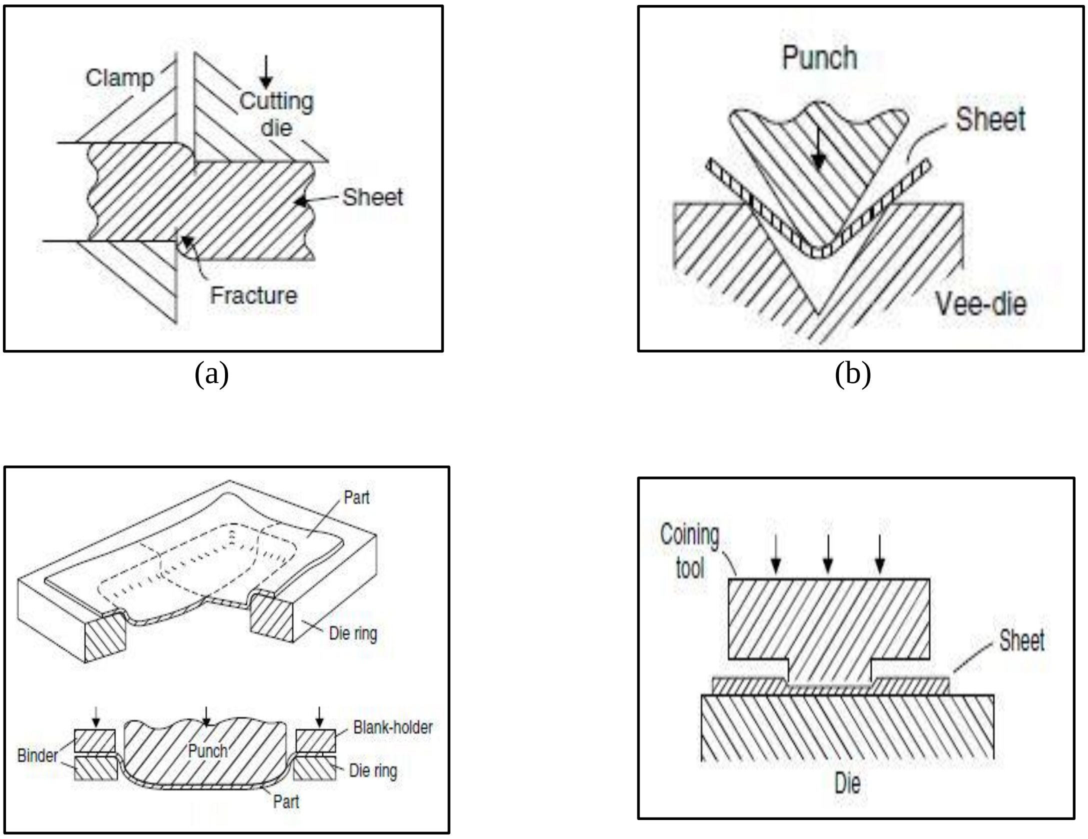
LITERATURE REVIEW

2.1 INTRODUCTION

This chapter will discuss about theory of bending, types of bending, materials and parameters involved that causing the bending. This chapter will have all necessary information from journal, book and articles that are related to this project and also about the springback study. The sources for the literature review are library books, journal from established databases such as Science Direct and Scopus, article and also newspaper article.

2.2 SHEET METAL FORMING

Sheet metal forming processes are those in which force is applied to a piece of sheet metal to modify its geometry rather than remove any material. The applied force stresses the metal beyond its yield strength, causing the material to plastically deform, but not to fail. By doing so, the sheet can be bent or stretched into a variety of complex shapes. There are a few examples of common sheet metal forming such as blanking and piercing, bending, stretching, stamping or draw die forming, coining and ironing, as shown in figure 2.1 (Z. Marciniak, 2002).



(c) (d)

Figure 2.1 : Example of sheet metal forming: (a) Cutting process, (b) Bending process, (c) Stamping or draw die, (d) Coining process.

Source : Z. Marciniak, 2002

In sheet metal forming industry, especially in sheet bending process, spring-back has a very significant role. In this process, the dimension precision is a major concern, due to the considerable elastic recovery during unloading which leads to spring-back. Also, under certain conditions, it is possible for the final bend angle to be smaller than the original angle. Such bend angle is referred to as spring-go or spring-forward. The amount of springback/spring-go is influenced by various process parameters, such as tool shape and dimension, contact friction condition, material properties, sheet anisotropy, and sheet thickness.

2.2.1 Material Properties that affect Sheet Metal Formability

a) **Ductility:** Metal used in the sheet metal work must be ductile. If we use a brittle metal it can easily undergo failure during forming. That's why metal's ductility is very important in sheet metal working.

b) **Yield strength**: Yield strength of a material used in sheet metal forming must be low. High strength metals have reduced stretch distribution characteristic, making them less stretchable and drawable than lower strength metals. Stretch distributions characteristic determine the steel ability to distribute stretch over a large surface area.

c) **Elastic modulus**: Stretch distribution affects not only stretch ability, but also elastic recovery, or spring back, and the metal's total elongation.

d) **Discontinuously Yielding**: Low carbon show a discontinuous yielding accompanied with the formation of Lǖder bands, which reduces the surface quality of the end product. In order to remove the discontinuous yield point a temper rolling can be applied.

e) **Work Hardening Rate (n)**: Work hardening rate is a very important sheet metal forming parameters. When increases material's resistance to necking also increase. The work hardening is the mechanism, which prevents local yielding and increase the uniform elongation.

f) **Anisotropy (Directionality):** Anisotropy is another factor that affects formability. Once consequence of directionality is a change in mechanical properties with direction. When forming sheet metal, practical consequences of directionality include such phenomena as excess wrinkling, puckering, ear-formation, local thinning, or actual rupture.

2.3 THEORY OF SHEET METAL BENDING

Bending can be defined as shaping materials without removing any chips around a definite axis through or without heat. Bending is the process of placing a sheet of metal over the matrix on the press bed where the sheet is bent around the tip of the punch as it enters the die. Bending dies are the setup, proper to the required piece shape, consisting of a female die and punch, and making permanent changes on steel sheet material. (Özgür Tekaslan *et. al.* 2008)

Bending along a straight line is the most common of all sheet forming processes. It can be done in various ways such as forming along the complete bend in a die, or by wiping, folding or flanging in special machines, or sliding the sheet over a radius in a die. A very large amount of sheet is roll formed where it is bent progressively under shaped rolls. Failure by splitting during a bending process is usually limited to high-strength, less ductile sheet and a more common cause of unsatisfactory bending is lack of dimensional control in terms of springback and thinning. A few among the most common applications of sheet metal parts are as automobile and aircraft panels, housings and cabinets. Customization of sheet metal parts to produce parts of varying configurations and sizes is a very common occurrence in a sheet metal fabrication scenario. (Olaf Diegel, 2002)

2.4 TYPES OF BENDING

There are several types of bending that commonly used in the industries such as air bending, bottoming, coining, V- bending, and U-bending. A bending tool must be decided depending on the shape and severity of bend.

2.4.1 Air Bending

Air bending is a bending process in which the punch touches the work piece and the work piece does not bottom in the lower cavity. As the punch is released, the work piece springs back a little and ends up with less bend than that on the. This is called spring back. In air bending, there is no need to change any equipment or dies to obtain different bending angles because the bend angles are determined by the punch stroke, as shown in figure 2.2.

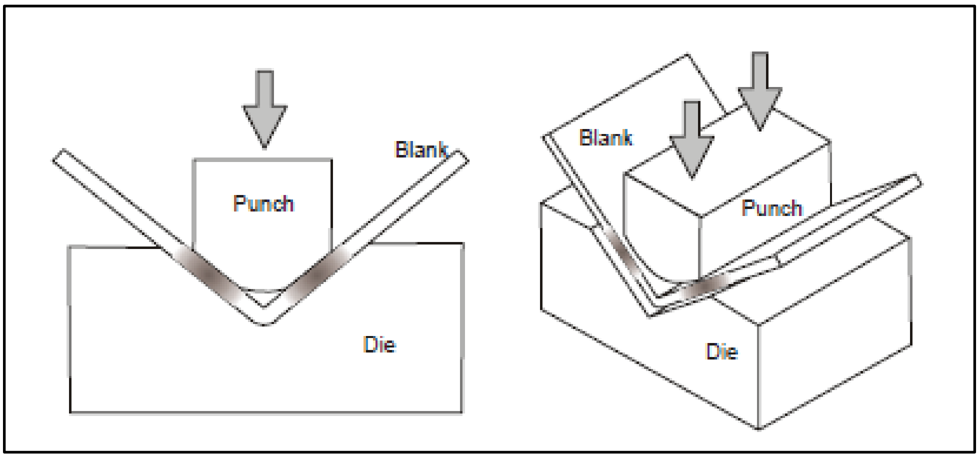


Figure 2.2: Air bending process

Source : Olaf Diegel, 2002

2.4.2 Bottoming

Bottoming is a bending process where the punch and the work piece bottom on the die. This make for a controlled angle with very little spring back. In bottom bending, spring back is reduced by setting the final position of the punch such as that the clearance between the punch and die surface is less than the blank thickness, the material yields slightly and reduces the spring back, as shown in figure 2.3.

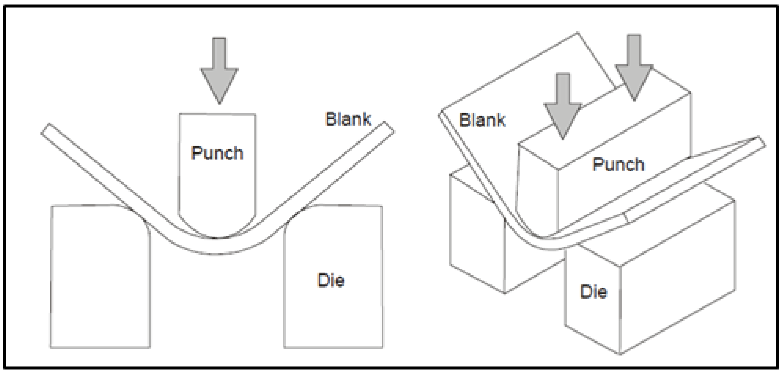


Figure 2.3: Bottoming process

Source : Olaf Diegel, 2002

2.4.3 Coining

Coining is a bending process in which the punch and the work piece bottom on the die and compressive stress is applied to the bending region to increase the amount of plastic deformation as shown in figure 2.4.

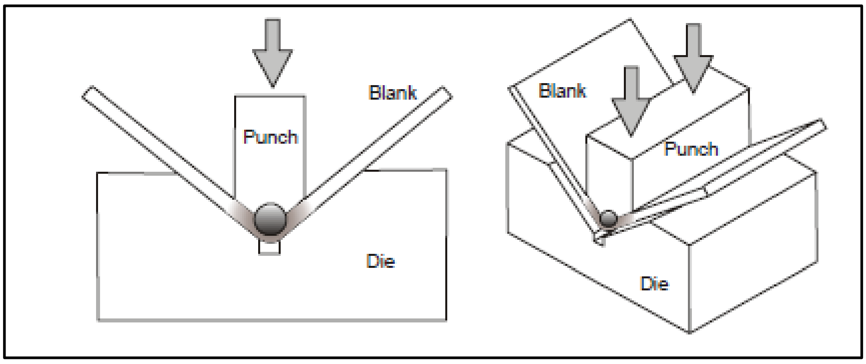


Figure 2.4: Coining process

Source : Olaf Diegel, 2002

2.4.4 V-bending

The V-die bending process is the bending of a V-shaped part in a single die. The work piece is bent between a V-shaped punch and die. The force acting on the punch causes punch displacement and then the work piece is bent.

The work piece is initially bent as an elastic deformation. With continued downward motion by the punch, plastic deformation sets in when the stresses exceed the elastic limit. This plastic deformation starts on the outer and inner surfaces directly underneath the punch as shown in figure 2.5.

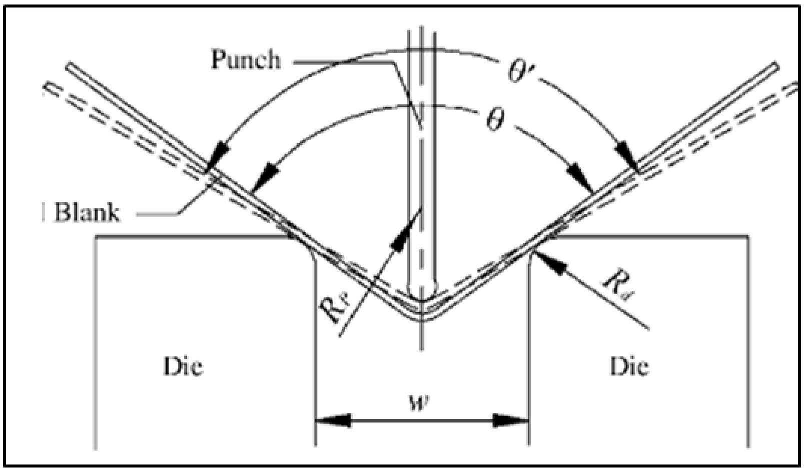


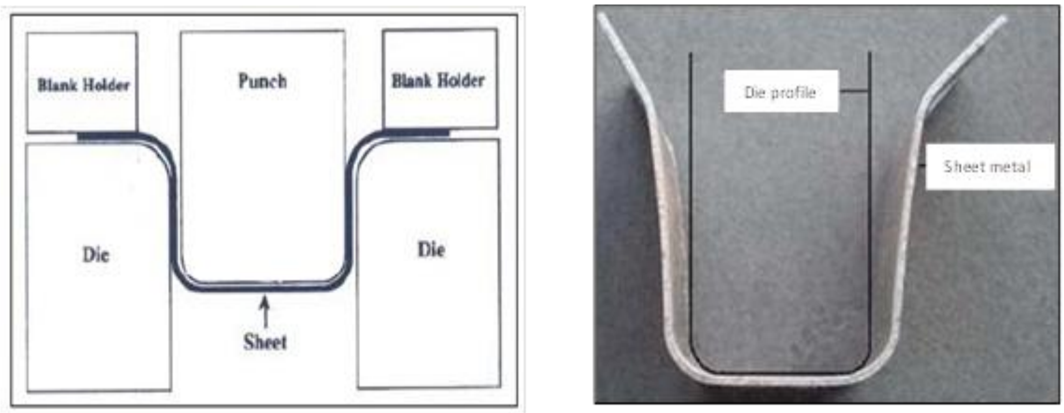
Figure 2.5: Illustration of V-die bending.

Source : V. Esat et al, 2002

2.4.5 U- bending

U – Bending is performed when two parallel bending axes are produced in the same operation. A backing pad is used to forces the sheet contacting with the punch bottom. Generally U–bending process can be divided into two steps, loading and unloading. In the loading step, the punch will completely moves down and the metal is being bent into the die as shown in figure 2.6.

During this step, the work piece undergoes elastoplastic deformation and temperature increase under frictional resistance. For the second step which is unloading step, the deformed sheet metal is ejected from the tool set and metal was experiencing the residual stress release and the temperature drop to reach a thermo-mechanical equilibrium state (M. Bakhshi-Jooybari et al, 2009).



(a) (b)

Figure 2.6: Sheet metal U-bending: (a) Schematic diagram of U–bending process, (b) Deformed specimen after unloading.

Source : M. Bakhshi-Jooybari et al, 2009.

U-bending process is often used to manufacture sheet parts like channels, beams and frames. In this process, the sheet metal usually undergoes complex deformation history such as stretch–bending, stretch–unbending and reverse bending. When the tools are removed, in addition to springback, sidewall curl often happens, which makes the prediction of springback become more difficult.

Different methods, such as analytical method, semi-analytical method and finite element method (FEM), have been applied to predict the sheet springback of U-bending, applied FEM to simulate the forming and springback process of sheet U-bending and reviewed the effects of numerical parameters, tools geometry and process parameters on the predicted accuracy of springback (D. Zhang,2007).

In U-bending process, a parameters that need to be considered are:

1. Bend Allowance – The length of the arc through the bend area.
2. Bend Angle – The included angle of the arc formed by the bending operation.
3. Bend Compensation – The amount by which the material is stretched or compressed by the bending operation. All stretch or compression is assumed to occur in the bend area.
4. Bend Lines – The straight lines on the inside and outside surfaces of the material where the flange boundary meets the bend area.
5. Inside Bend Radius – The radius of the arc on the inside surface.
6. K-factor – Defines the location of the neutral axis. It is measured as the distance from the inside of the material to the neutral axis.
7. Mold Lines – For bends of less than 180 degrees, the mold lines are the straight lines where the surfaces of the flange bounding the bend area intersect. This occurs on both the inside and outside surfaces of the bend.
8. Neutral Axis – Looking at the cross section of the bend, the neutral axis is the theoretical location at which the material is neither compressed nor stretched.
9. Set Back - For bends of less than 180 degrees, the set back is the distance from the bend lines to the mold line.

2.5 SPRING BACK IN U – BENDING

Spring back generally defined as additional deformation of sheet metal parts after the loading is removed. Springback is a major problem in sheet metal bending technique. Several bending operations done on sheet metal are V-die bending, rubber die bending and U-bending. In U-bending, the material may exhibit negative and positive springback caused by deformation as the punch completes the bending operation. The amount of springback/spring go is influenced by various process parameters, such as tool shape and dimension, contact friction condition, material properties, and sheet anisotropy, as shown in figure 2.7 (M. Bakhshi-Jooybari et al, 2009).

There are several researchers that have investigated and attempted to obtain a basic understanding of spring back behaviour. The effect of bending angle on spring back of six types of materials with different thicknesses in V-die bending has been studied by M. Bakhshi-Jooybari experimentally showed the effect of combined hot die and cold punch on reduction of spring-back of aluminium sheets.

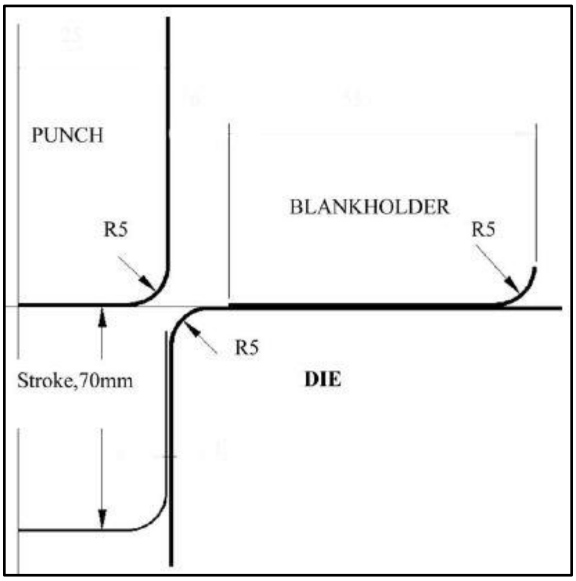


Figure 2.7 : U-bending process.

Source : M. Bakhshi-Jooybari et al, 2009.

The contact between the tool and the work piece bent during the folding operation extends over a larger area than can be occur when a U-folding or falling edge. The most studied phenomena in the process of formatting is that the springback appears to be other cases of bending and bending it induces at the walls. The relationship between the evaluation values of springback and the geometric parameters of formatting is established using a finite element simulation. In this contribution, he developed an optimization algorithm based on the coupling between the finite element code Abaqus / most standard and to minimize the springback in determining the optimal parameters of the process for each technique.

In 2005, M.Firat studied the sensitivity of springback to various factors which were introduced during the simulation of the folding process dynamically U explicitly. As example, they analysed the influence of the damping value, the number of points integration through the thickness, the size of items and the speed of the punch the accuracy and efficiency of spring back simulated as shown in figure 2.8.

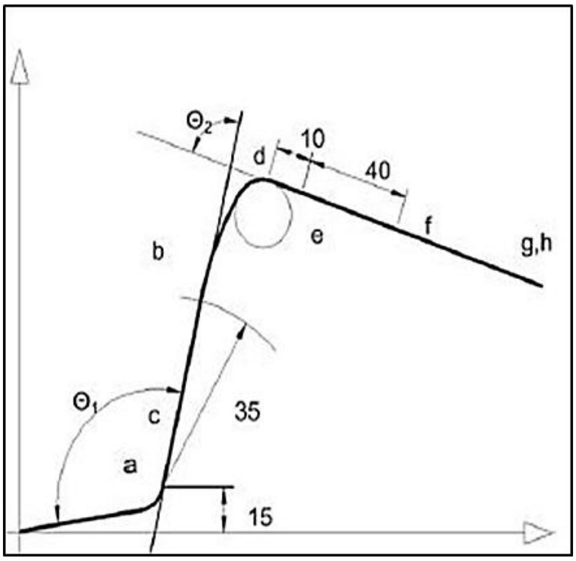


Figure 2.8 : Geometry and dimensions of the piece folded in U-bending

Source : M.Firat, 2005.

Part of the deformation is not reversible. After declining tools and relaxation of constraints, there is a springback, and it persists deformation permanent due to irreversible shift of certain crystallographic planes each compared to the other. The acceptance or refusal of a folded piece control depends greatly from its use. Therefore, the success of a bending operation is linked and material properties of the sheet and the other process parameters whose geometry is one of the tools. For this he introduced in his study of K correction factors which depend on both the nature of the material and the ratio r / e, where r represents the radius of the matrix and e the thickness of the sheet. These factors are factors K parameter as shown in figure 2.9.

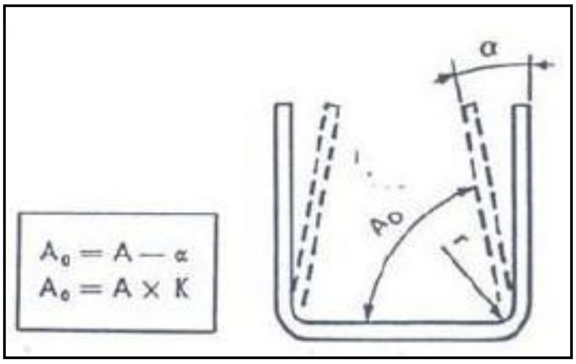


Figure 2.9: Correction springback based on r / e

Source : YU Zhong-qi, 2007

Table 2.1 : List of correction springback on different material.

Metal plates	K factor								
	r/e	1	1.2	1.6	1.8	2	2.5	3	4	8
Aluminium alloy		0.94	0.94	0.94	0.93	0.93	0.93	0.92	0.91	0.86
Ferritic steel		0.92	0.92	0.91	0.9	0.89	0.88	0.87	0.85	0.79
Mild steel		0.9	0.88	0.87	0.87	0.86	0.86	0.85	0.84	0.75

The effects of major process parameters on the elastic recovery phenomenon have not been sufficiently investigated. However, in practice the understanding of parametric characteristics of the springback amount is essential for the systematic tool design.

2.6 GALVANIZED IRON

2.6.1 Material Used

Galvanic Iron, in current use, the term refers to the coating of steel or iron with zinc. This is done to prevent rusting of the ferrous item. The value of galvanizing stems from the corrosion resistance of zinc, which, under most service conditions is considerably greater than that of iron and steel. The zinc therefore serves as a sacrificial anode, so that it cathodically protects exposed steel.

2.6.2 Material Properties

The most important criteria in selecting a material are related to the function of the part qualities such as strength, density, and stiffness and corrosion resistance. For sheet material, the ability to be shaped in a given process, often called its formability, should also be considered. Whenever there is a stress on a sheet element, there will also be some elastic strain. It is often neglected, but it can have an important effect, for example when a panel is removed from a die and the forming forces are unloaded giving rise to elastic shape changes, or springback. GI material properties has been shown in table 2.1.

Table 2.2 : Material properties

Material	Sheet thickness (mm)	Young's Modulus (MPa)	Ultimate Tensile Strength (MPa)	Strain hardening Exp (n)
Galvanized Iron	1.0	29247.754	343.7201778	0.27445

Source : ASTM, 2003.

2.6.3 The true stress–strain curve

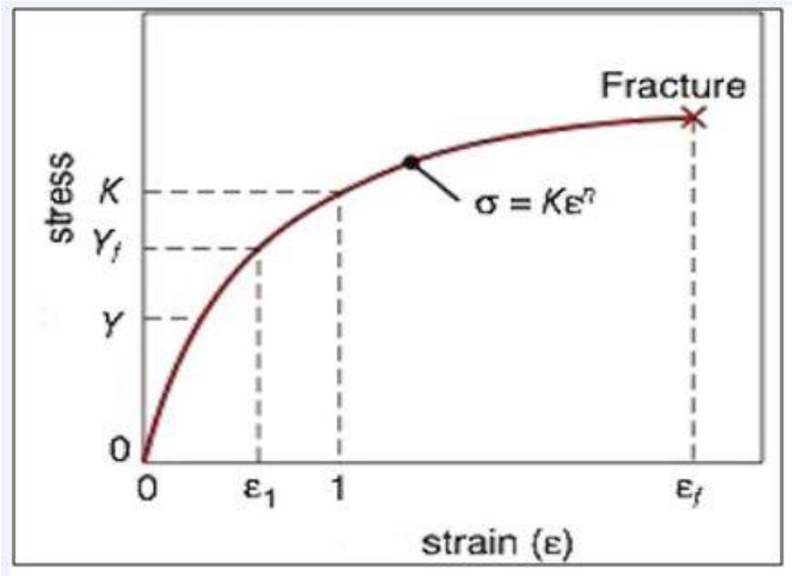


Figure 2.10: Stress-Strain Curve

Source : Michael F. Ashby, 2005.

Y : yield strength

Y_f : flow stress is the true stress to complete plastic deformation at a particular true strain ε_1.

Stress-strain curves are an extremely important graphical measure of materials mechanical properties. The yield point is that point when a material subjected to a load, tensile or compression gives and will no longer return to its original length or shape when the load is removed. The yield point is a very important concept because a part is usually useless after the material has reached that point as shown in figure 2.10 (Michael F. Ashby, 2005).

Ultimate tensile strength, often shortened to tensile strength or ultimate strength, is the maximum stress that a material can withstand while being stretched or pulled before necking, which is when the specimen's cross-section starts to significantly contract.

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

This chapter is about the method that is used to collect data in completing this study. Explanations for this chapter will be based on several elements that contains in the flow chart of the study. This methodology includes the step to design the product until the steps to develop the simulation using the finite element analysis software. This chapter also deals with procedures and parameters involved in the project.

There are two main methods used in this project which are simulation and experimental method. In the simulation method, the process to predict springback angle start from the determination of mechanical properties from tensile test. In the experimental method, the methodology of U-bending test will be discussed.

3.2 PROJECT FLOW CHART

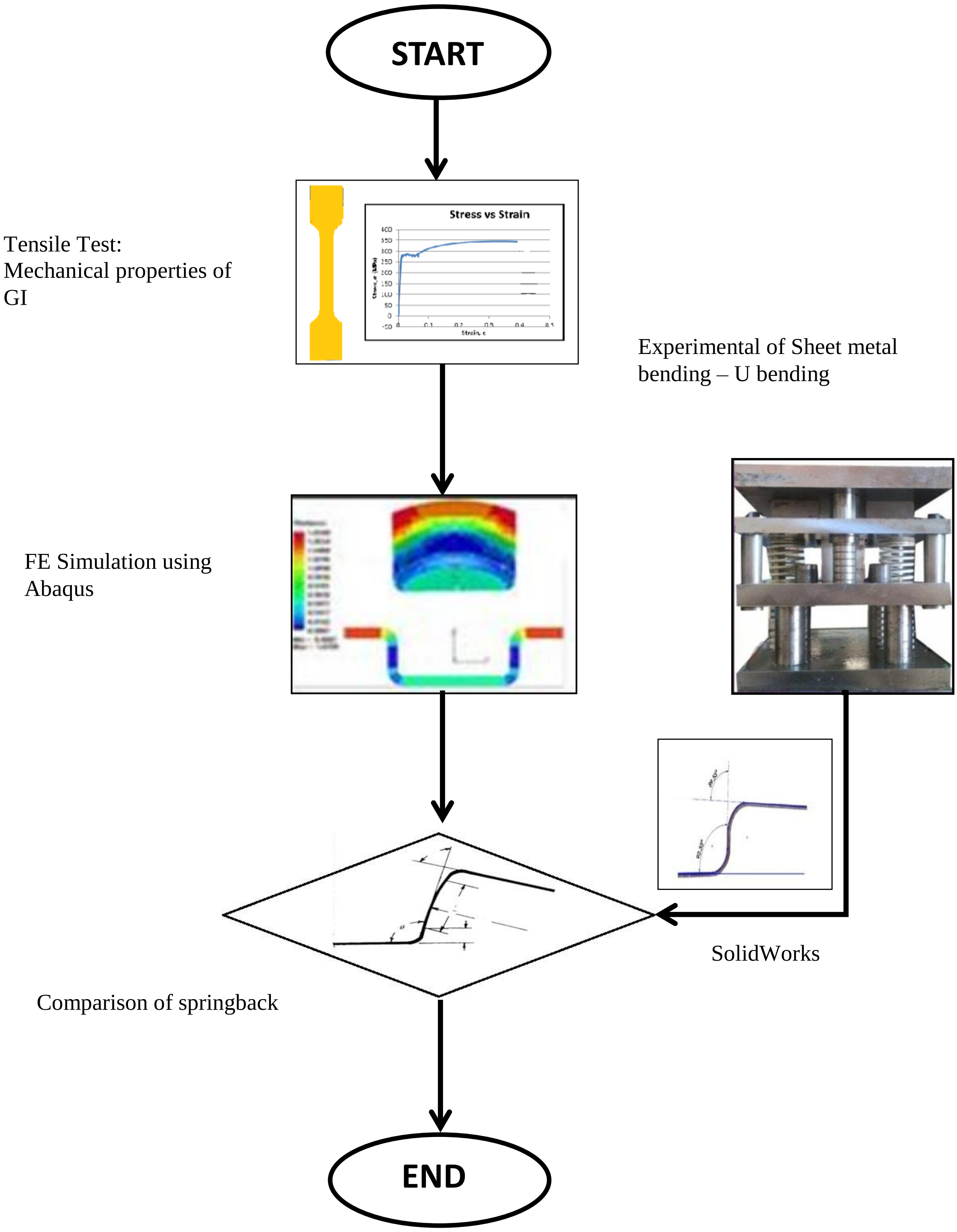


Figure 3.1 : Project flow chart

3.3 TENSILE TEST

Tensile test is one of the most commonly used tests for evaluating material properties. Material that has been used in this project is Galvanize Iron. The tensile test is used to evaluate the strength of metal alloys. In this test a Galvanize Iron sample is pulled to failure in a relatively short time at a constant rate. The tensile test is accomplished by gripping opposite ends of a test specimen within the load frame of a test machine.

During the tensile test process, force and extension data has been monitored and recorded. The tensile test provides force and extension data that can quantify several important mechanical properties of a Galvanize Iron 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 springback equation.

3.3.1 American Society for Testing and Materials (ASTM E8)

The Galvanize Iron need to be cut into a tensile test specimen which the ASTM E8 standard as the reference. Regarding to the ASTM E8 standard, the specimen of the tensile test can be divided into three types where it is plate type specimen, sheet type specimen and round type specimen. In this project, the test specimen is sheet metal type.

This specimen is used for testing metallic materials 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 3.2 and Table 3.1.

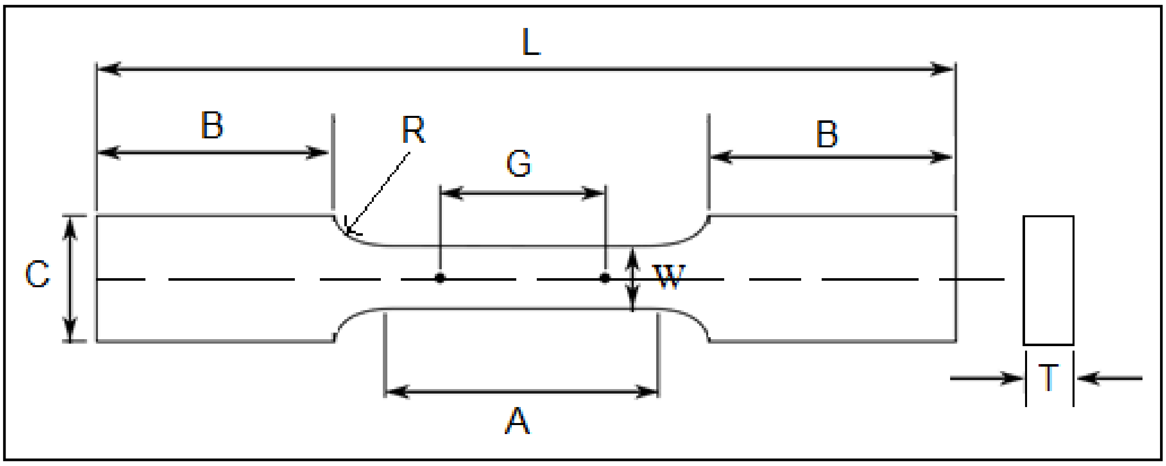


Figure 3.2 : Examples of rectangular (flat) tensile test specimen

Source: Annual Book of ASTM Standards, Vol 01.03. *et. al.*, 2003

Table 3.1 : Dimension and specification of the tensile test specimen ASTM E8

	Plate type (1.5 in. wide) mm	**Sheet type (0.5 in. wide) mm**	**Sub-size specimen (0.25 in. wide) mm**
Gage length, (G)	200 ± 0.25	50.0 ± 0.10	25.0 ± 0.08
Width ,(W)	40 + 3 - 6	12.5 ± 0.25	6.25 ± 0.05
Thickness, (T)		Thickness of Material	
Fillet radius (min.), (R)	13	13	6
Overall length (min.), (L)	450	200	100
Length of reduced section (min.),(A)	225	60	32

Source: Annual Book of ASTM Standards, Vol 01.03. *et. al.*, 2003

3.3.2 Specimen preparation

In this experiment, thickness of Galvanize Iron that been used to performed the tensile test is 1.0mm. It has been cut in the different orientation angle of 0, 45 and 90 degree and each orientation angle has been cut for 3 specimens. Table 3.2 below shown the total specimens in this project.

Table 3.2 : Total specimen preparation

Specimen Thickness (mm)	No. of Specimen			
	0°	**45°**	**90°**	**Total Specimen**
1.0	3	3	3	9

Galvanize Iron sheet metal need to be cut into specimen according to ASTM Standards, which the raw material of sheet metal need to be cut into a rectangular size of 300mm x 50mm by using LVD shearing cutting machine. 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.

Tensile specimen has been draw using the Mastercam software for generated the G-Code of the CNC milling machine. Mastercam's comprehensive set of predefined tool paths which including contour, drill, pocketing, face, peel mill, engraving, surface high speed, advanced multiaxis, and many more and enable us to cut parts efficiently and accurately. Steps of designing the tensile specimen by using Mastercam software are shown below.

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

2. 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.

3. Then, at 2D-toolpaths icon select contour as a tool paths 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.

4. 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 10 mm/min. The spindle speed is set to be 900 rpm.

5. Then, at the linking parameters icon, set up the depth to be cut to the raw material. For example, if the thickness of the material is 1mm, the suitable cutting depth is 1.5 mm.

6. After all parameter for this process has been setup in the software, the simulation has been run to check the flow of this process.

7. Finally, generate the G-code from the simulation process.

The cutting of tensile specimen will be completed by using Haas CNC Machine as shown in figure 3.3. Computer Numerical Control which refers to a computer controller that controls the movement of every axis of the machine using G and M codes instructions and drives the spindle or machine tool into a workpiece to fabricate or to remove the unwanted material from workpiece more accurately without human intervention.

Figure 3.3 : Haas CNC Milling Machine

The G-Code that has been generated from Mastercam's than be imported to the CNC milling machine. The scheme below has shown the basic processes to running the milling machine. The basic processes of how to operate the Haas CNC Milling Machine are shown below. Figure 3.4 show CNC milling machine in process and figure 3.5 show specimen done from CNC machine.

1. Powering ON the machine
2. Select and open the G-Code programme.
3. Clamping the workpiece.
4. Locating X and Y zero points of the part.
5. Loading the tool into the tool carousel.
6. Setting the tool offsets.
7. Running the programme.

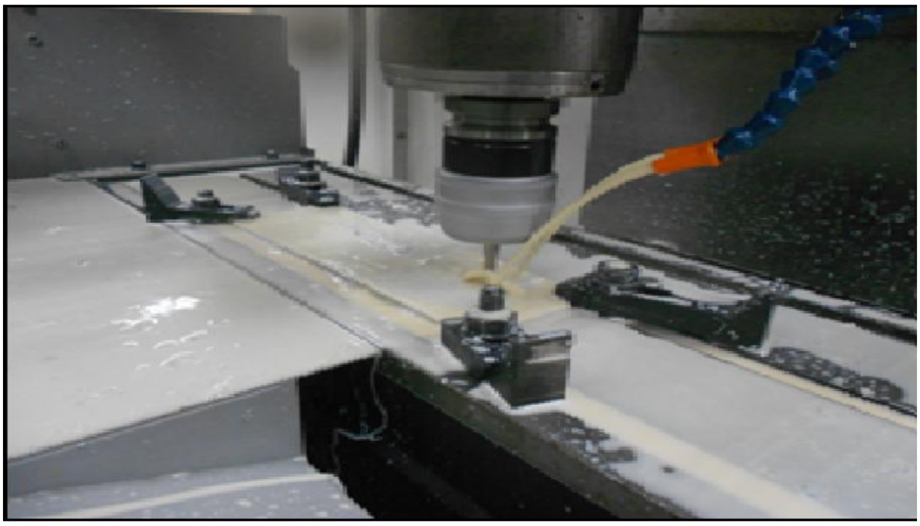


Figure 3.4 : CNC milling machine in process.

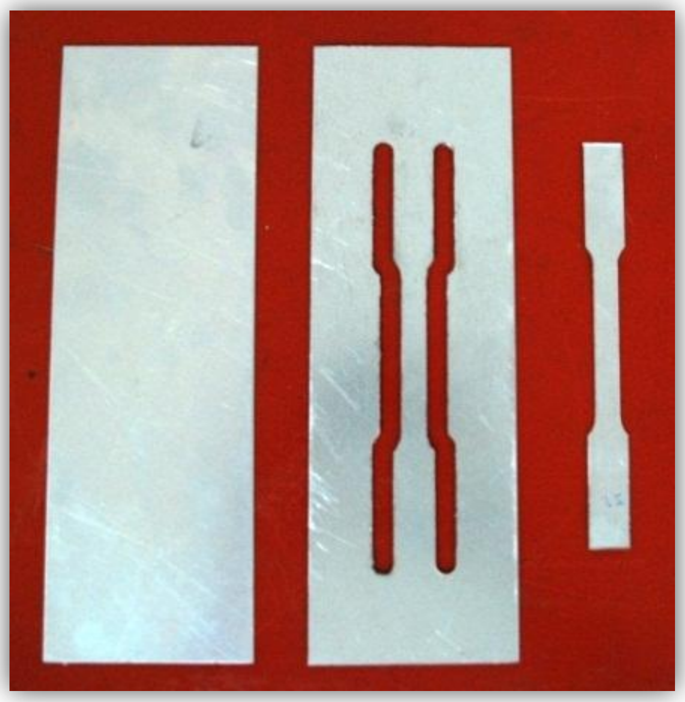


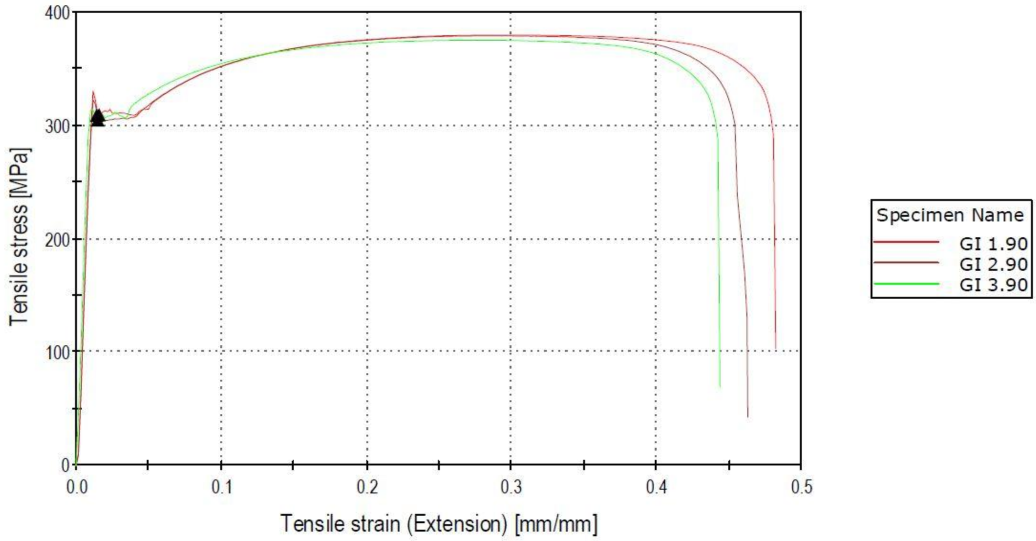
Figure 3.5 : Specimen done from CNC machine.

3.3.3 Tensile Test Experiment

After the test specimen has been properly prepared, the tensile test setup will be established to conducting the tensile test. The test specimen will be installed properly in the grips in the Instron Universal Testing Machine. A maximum load has been set to 50 kN. An experiment has been run till a Galvanize Iron broke as shown in figure 3.6.

Figure 3.6 : Specimen at break point.

After all material has been tested, the result has been show in graph and data properties like Young's Modulus, E**,** Yield Strength, Ultimate Tensile Strength (UTS) and also Poisson's Ratio, v, also can get from this test as shown in figure.3.7.



	Load at Break (Standard) [kN]	Maximum Load [kN]	Modulus (Automatic) [MPa]	Tensile strain (Extension) at Maximum Load [mm/mm]	Tensile stress at Maximum Load [MPa]	Data point at Yield (Slope threshold 0.02 %)	Tensile stress at Yield (Slope threshold 0.02 %) [MPa]	Tensile strain (Extension) at Yield (Slope threshold 0.02 %) [mm/mm]	Plastic strain ratio at r-value (Tensile strain (Extension) 0 mm/mm - Tensile strain (Extension) 0 mm/mm)
1	1.29	4.74	37,548	0.29000	379.31036	10	309.00076	0.01584	-----
2	0.54	4.73	3,514	0.28167	378.55933	9	304.61839	0.01500	-----
3	0.87	4.69	3,087	0.27500	374.80457	9	308.70326	0.01417	-----

Figure 3.7 : Stress strain graph for GI 1.0 mm thickness, 90 degree rolling direction.

Then, from the data that we get from tensile test result as shown in figure 3.7, we proceed a procedure by calculate an engineering stress to get a graph and data as shown in figure 3.8, 3.9, 3.10 and table 3.3, 3.4, 3.5 based on anisotropy of specimen.

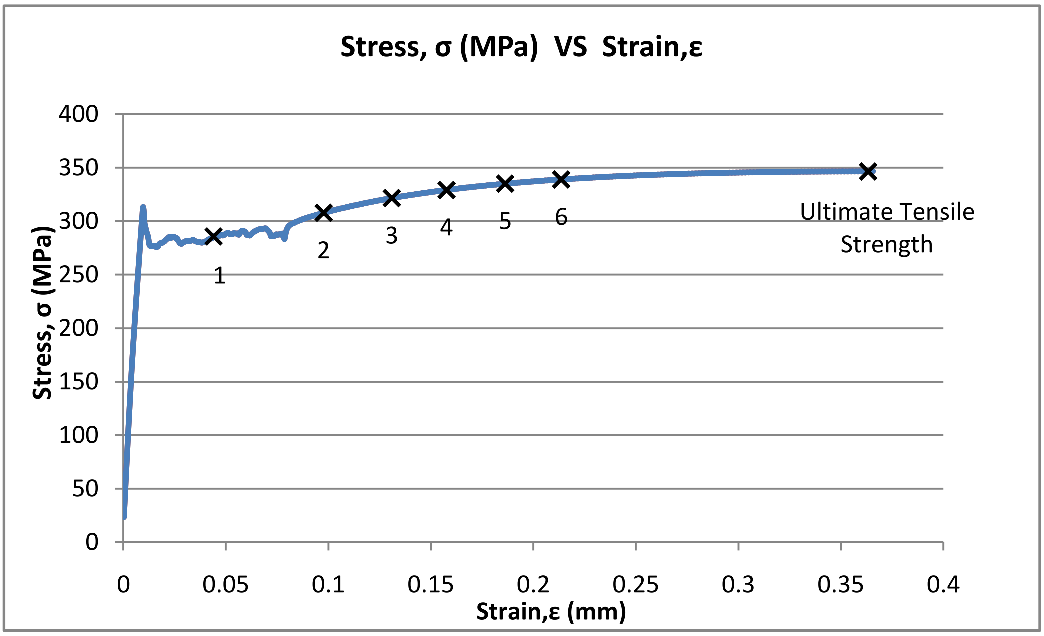


Figure 3.8 : Engineering stress – strain diagram for Galvanized Iron, anisotropy, R = 0 degree.

Young's Modulus ,E	=	32359.843 Mpa
Yield Strength	=	283.774 Mpa
Ultimate Tensile Strength (UTS)	=	346.5088Mpa
Poisson's Ratio, v	=	0.3

Table 3.3 : Selected points in stress-strain diagram for 0 degree

Point	Stress,σ (Mpa)	Strain,ε	Strain,ε at 0
1	283.774	0.043	0.00
2	307.749	0.098	0.06
3	321.577	0.131	0.09
4	329.181	0.158	0.12
5	335.073	0.186	0.14
6	339.010	0.214	0.17

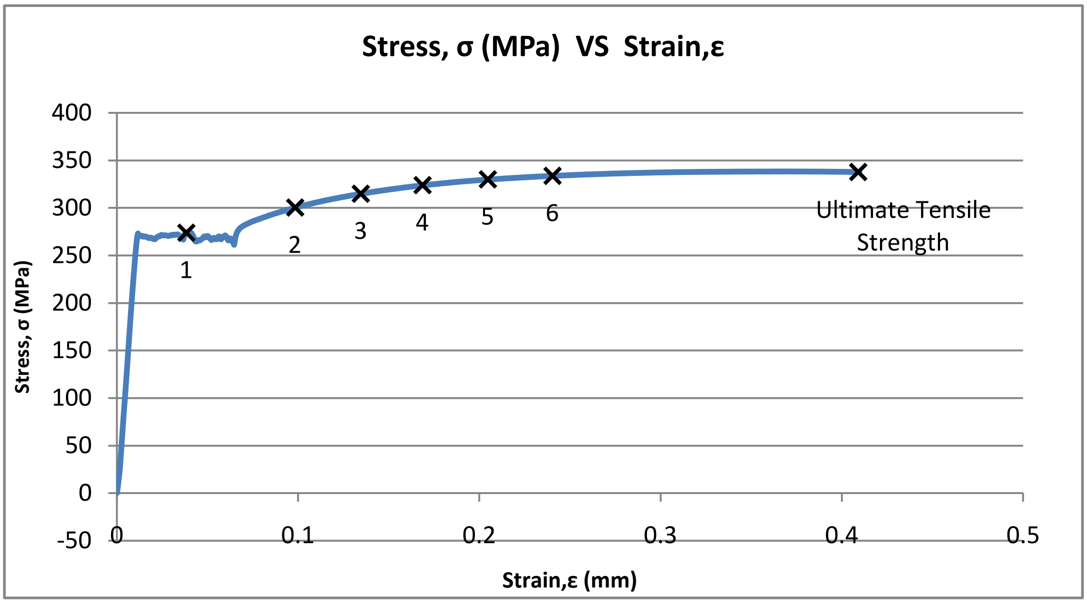


Figure 3.9 : Engineering stress – strain diagram for Galvanized Iron, anisotropy, R =45 degree.

Young's Modulus ,E	=	26381.335 Mpa
Yield Strength	=	273.887 Mpa
Ultimate Tensile Strength (UTS)	=	337.9141333Mpa
Poisson's Ratio, v	=	0.3

Table 3.4 : Selected points in stress-strain diagram for 45 degree.

Point	Stress,σ (Mpa)	Strain,ε	Strain,ε at 0
1	273.887	0.038	0.00
2	300.397	0.098	0.06
3	315.095	0.135	0.10
4	323.938	0.169	0.13
5	330.138	0.205	0.17
6	333.854	0.240	0.20

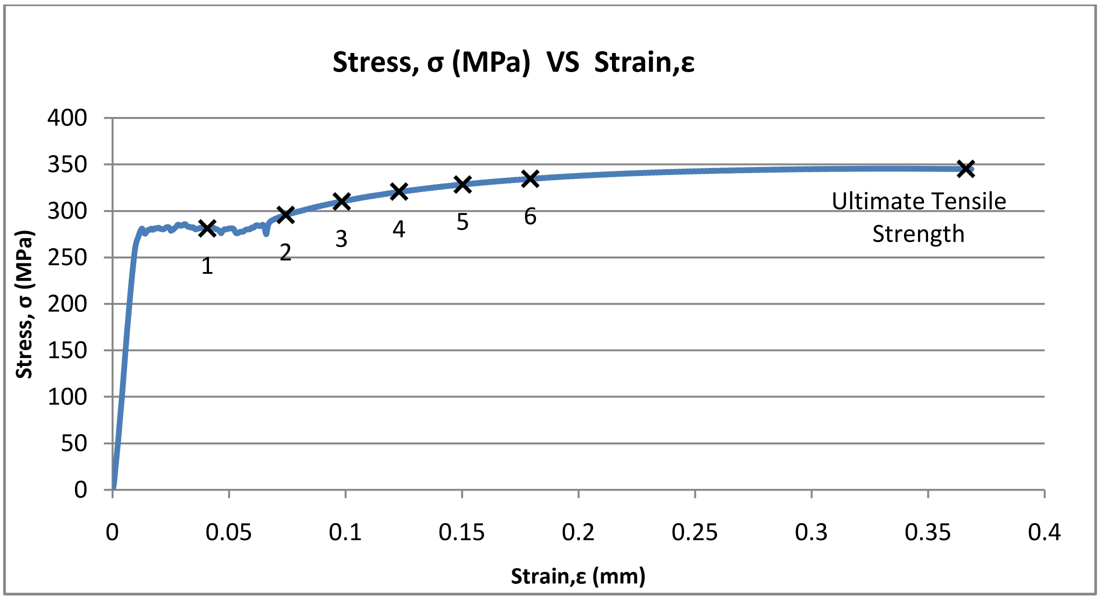


Figure 3.10 : Engineering stress – strain diagram for Galvanized Iron, anisotropy, R =90 degree.

Young's Modulus ,E	=	29002.086 Mpa
Yield Strength	=	281.3 Mpa
Ultimate Tensile Strength (UTS)	=	345.2144Mpa
Poisson's Ratio, v	=	0.3

Table 3.5 : Selected points in stress-strain diagram for 90 degree.

Point	Stress,σ (Mpa)	Strain,ε	Strain,ε at 0
1	281.300	0.041	0
2	295.724	0.074	0.03
3	310.308	0.098	0.06
4	320.714	0.123	0.08
5	328.571	0.15	0.11
6	334.588	0.179	0.14

3.4 FINITE ELEMENT

Finite element analysis is a simulation technique which evaluates the behaviour of components, equipment and structures for various loading conditions including applied 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.

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 AlgorFempro. 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.

In the finite element model, the workpiece, die, blank holder, and punch are the main components to be drawn. The punch is moving downward to bend the workpiece. Springback of the metal is then allowed to take place. Throughout the simulation, the top left and bottom left of the workpiece are constrained in the x and y directions. This is to prevent any rigid body motion of the workpiece, which will result in numerical errors during the simulation. Due to plane symmetry, only half of the process was modelled.

3.4.1 Parts Modelling.

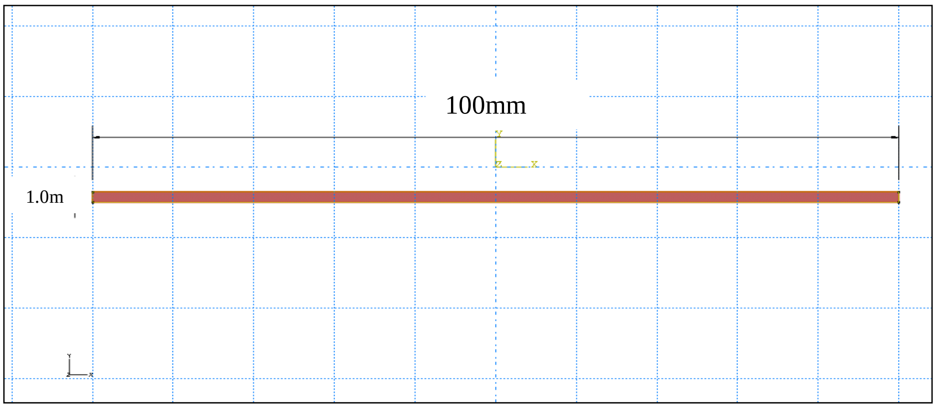


Figure 3.11 : Blank

Figure 3.11 shows a drawing of blank. The length of the blank is 100 mm while width is 1.0 mm. The sheet metal is represented by a deformable body. It is because this particle's part is moving in y and x-direction when the bending process started. The springback value will be calculated based on this part with calculating by the nodes.

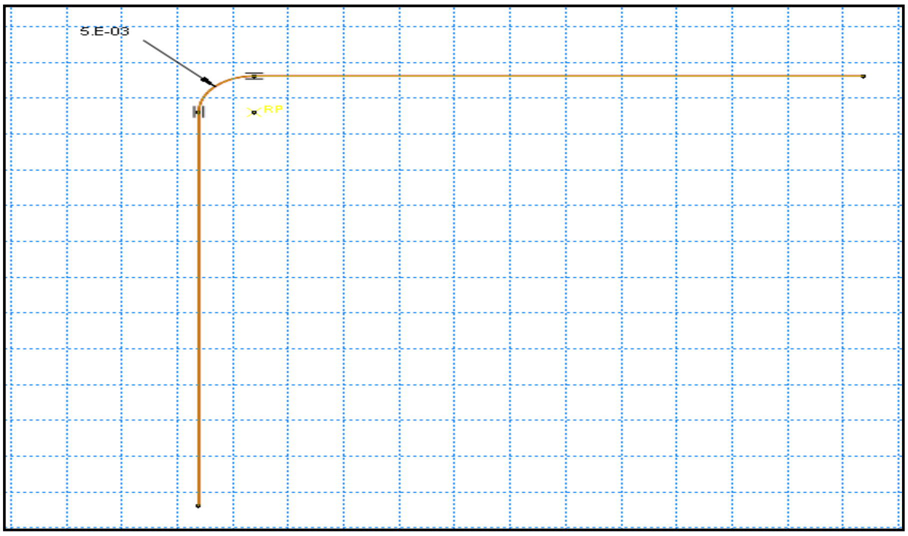


Figure 3.12 : Die

Figure 3.12 shows a drawing of die. The die is defined as rigid body. The sheet metal is placed on the die and will flank with blank holder. The workpiece will follow the shape of the die after it bending. The die radius is set by 5 mm.

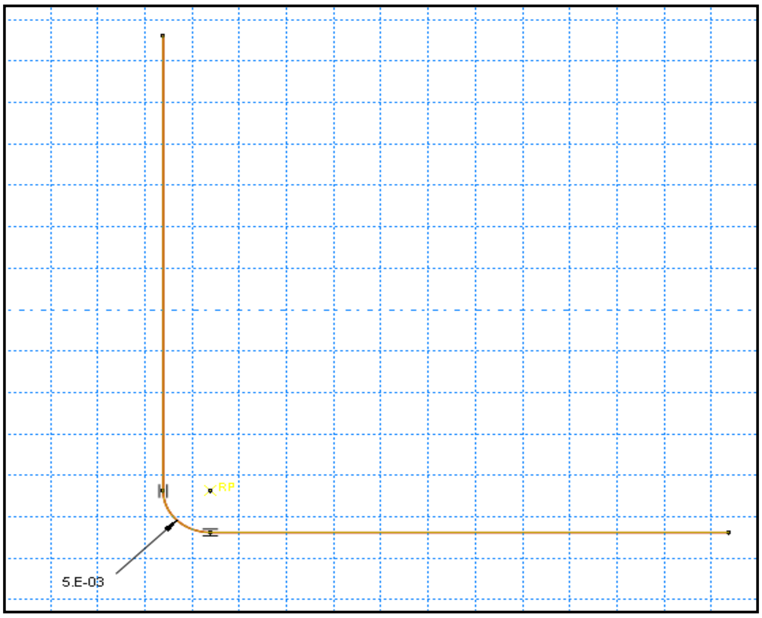


Figure 3.13 : Blank holder.

Figure 3.13 shows a drawing of blank holder. The blank holder is also defined by rigid body. The function of blank holder is to clamp the workpiece with the die and to make sure that there is no moving part when the blank is starting to bend. Contact of blank holder and blank must be declared because the blank holder will press the blank when the workpiece is bending.

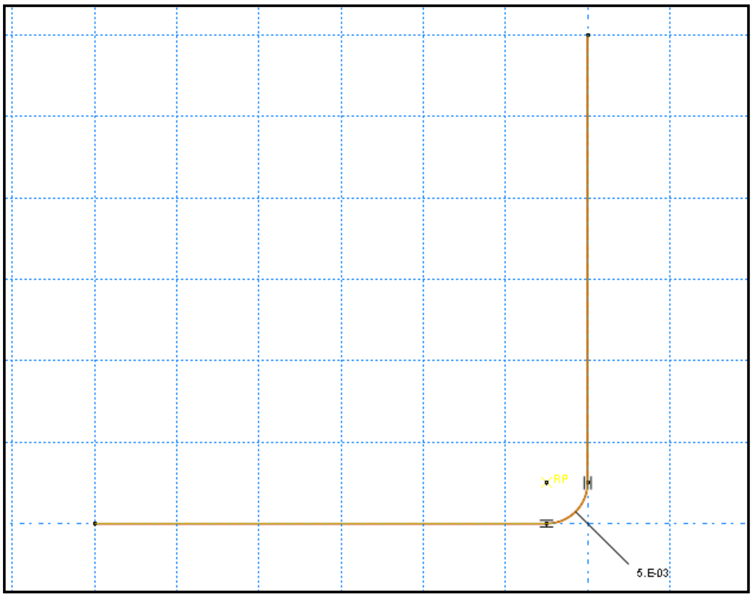


Figure 3.14 : Punch

Figure 3.14 shows a drawing of punch. Punch is modelled as a rigid body since it is functioning to punch the blank. The direction of the punch is in y-direction and it will punch blank downward follows the die shape. Contact of punch and the blank must be declared in the Abaqus software to make sure the punch will bend the blank properly.

3.4.2 Material Input.

To simulate a finite element model, the elastic and plastic properties must be declared. Elastic and plastic properties data can be obtained from stress-strain diagram of Galvanized Iron from tensile test experiment.

Elastic properties of a solid are important because they relate to various fundamental solid-state properties. Elastic properties are also linked thermodynamically to the specific heat, thermal expansion, melting point and parameter. So, it is important to know elastic constants of Galvanized Iron to put into Abaqus software. Poisson’s ratio occurs in some of the more complex stress/strain equations. It sounds complicated, but it is simply a way of saying how much the material necks down or gets thinner in the middle when it is stretched.

Plastic properties or known as plasticity is a property of solid body whereby it undergoes a permanent changing shape or size when subjected to a stress exceeding a particular value. The yield point is that point when a material subjected to a load, tensile or compression will no longer return to its original length or shape when the load is removed. Some materials break before reaching a yield point.

The initial geometry used for the simulation with finite element mesh is consists of linear quadrilateral elements. An Abaqus, software was developed to generate the process setup with finite element mesh. The value of 0.5mm mesh is used for this simulation.

The simulation of impact and contact among two or more objects of any kind has always been a challenging problem. It is one of the critical elements in successfully simulating the flanging operation. In this simulation the die and blank holder is declared as contact with the blank and is defined a friction with 0.1. The punch is also declared as contact with the blank and it is a frictionless.

When all the required data has been formed, the simulation need to be calculated to obtain the result and this can be completed by creating a job. The job then will submit the data to be analysed. The result can be analyzed based on different criteria. This can be done by modifying the convergence criteria, re-submit the analysis and the result already can be compared.

3.5 U-BENDING EXPERIMENTAL SETUP

3.5.1 Specimen preparation

The material used is Galvanized Iron which will be cut into the size of 25mm x 200mm. 1.0mm thickness has been cut in the different orientation angle of 0, 45 and 90 degree and each orientation angle has been cut for 3 specimens. Table 3.6 and Table 3.7 represent the total specimens and material properties used in this project.

Table 3.6: Total specimens used.

Specimen Thickness (mm)	No. of Specimen			
	0°	**45°**	**90°**	**Total Specimen**
1.0	3	3	3	9

Table 3.7 : Material properties

Material	Sheet thickness (mm)	Young's Modulus (MPa)	Ultimate Tensile Strength (MPa)	Strain hardening Exp (n)
Galvanized Iron	1.0	29247.754	343.7201778	0.27445

3.5.2 Die

The U-shaping stage is carried out with the experimental set-up. Three different anisotropies Galvanized Iron of 1.0 mm thickness was tested which are 0 degree, 45 degree and 90 degree which for every anisotropy there are three specimens. The die travels were stop automatically when there are no clearance between punch and specimen.

The die will be clamped on the stamping machine (hydraulic) before the specimen will be placed on the top of the blank holder and punch. When the force applied, die and blank holder will moved downward and hit the specimen while the punch stays still at that position. Figure 3.15 shown schematic and photograph for the experimental set-up for u-bending experiment.

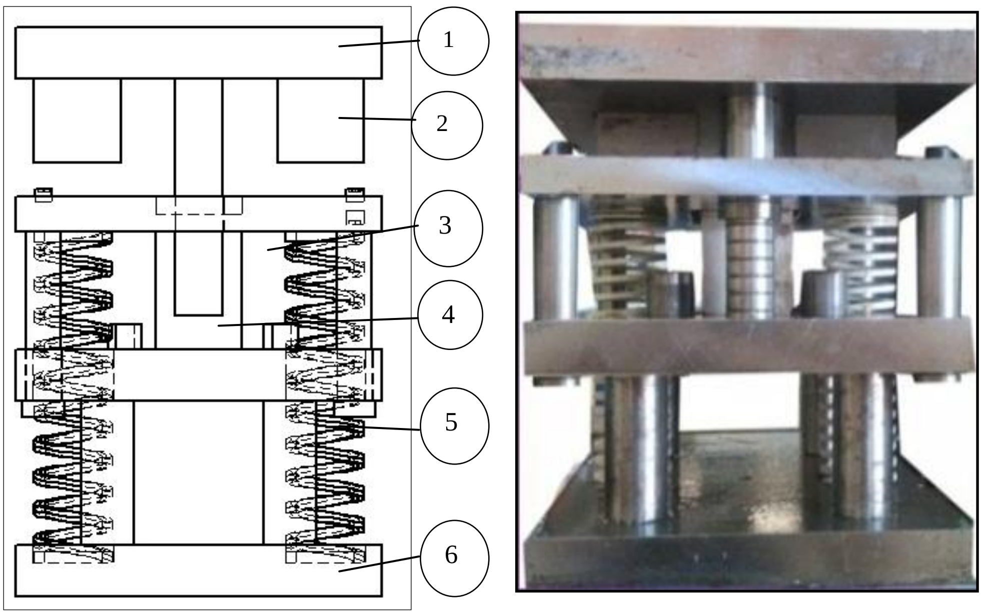
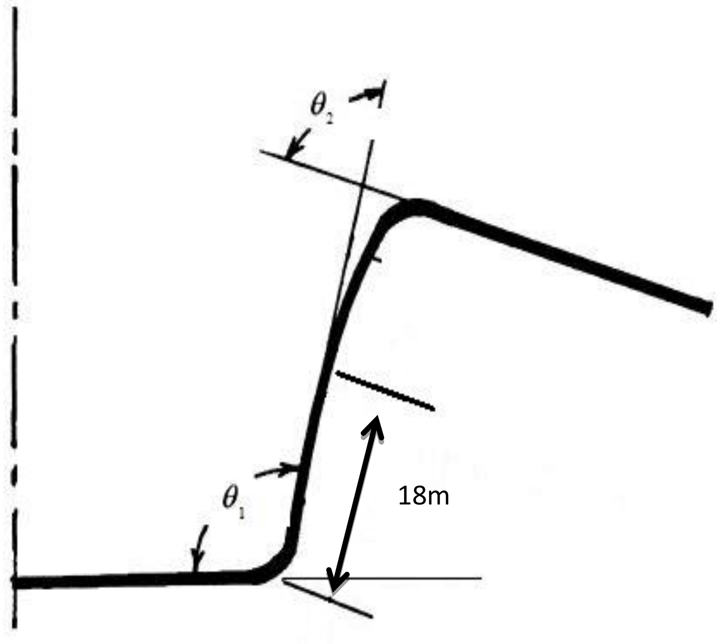


Figure 3.15 : Schematic and photograph for the experimental set-up: (1) upper part, (2) die, (3) blank holder, (4) punch, (5) spring, (6) lower part.

3.5.3 Springback measurement



parameters :

Θ_1 = Springbackangle 1 (degree)

Θ_2 = Springback angle 2 (degree)

Figure 3.16: Parameters for springback in U-bending process.

Source : M.Samuel, 2000.

There a few methods that can be used to measure springback angle and one of the method is by measure the nodes of the curve and measure the angle of the curve as stated by M.Samuel (2000). A curvature at lower part we assume as Θ_1 that is springbackangle 1 and at upper part we assume as Θ_2 that is springbackangle 2, as shown in figure 3.16.

The bending angle of the specimen has been measured using the SolidWork software. After specimen has been bending, it will be scan using Canon Inkjet MP 140 Series. The scanning profile will be save in the picture format.

The bending specimen picture then to be imported in the SolidWorks software and the angle will be measured. The steps for measuring the springback angle of the specimens are shown below and example of specimen scanning picture has been shown in figure 3.17.

1. Scan specimen using Canon Inkjet MP 140 Series printer.
2. Save the scanning profile in picture (JPEG Format).
3. Open Solidworks software and import the specimen picture using sketch tool toolbar.
4. Sketch the v-line on the specimen and measure the angle using smart dimension tool.
5. Save the result.

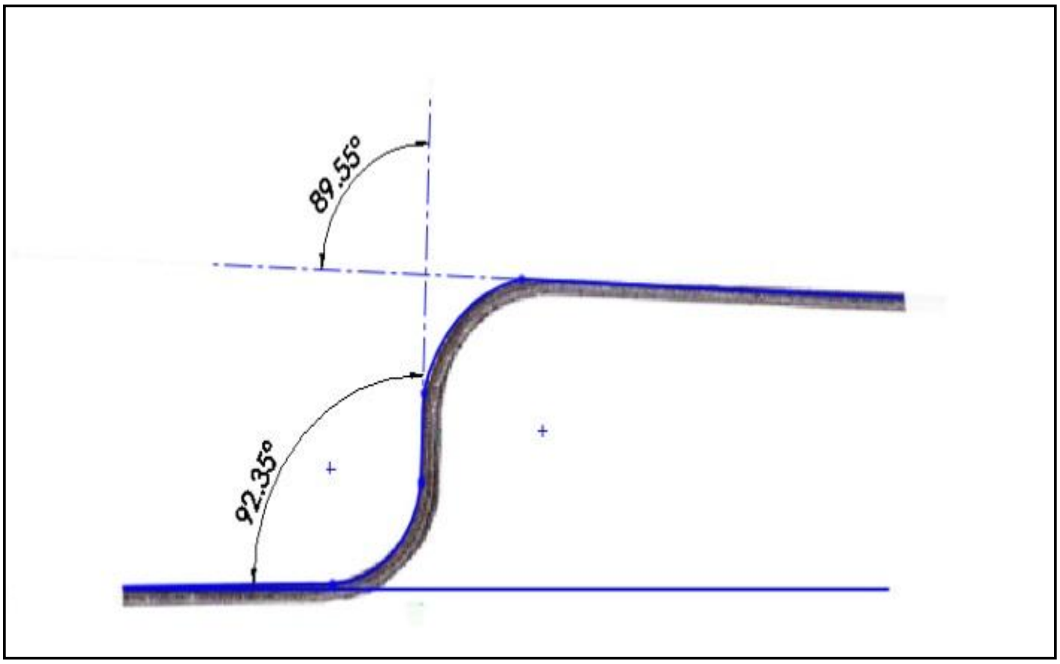


Figure 3.17 : Specimen scanning picture

CHAPTER 4

RESULT AND DISCUSSION

4.1 INTRODUCTION

This chapter is showing the results from the experimental and finite element analysis of springback of Galvanized Iron. In the experiment, the sheet metals were scan by a scanner and the angle of springback then measured by using SolidWorks 2012 software. For the simulation, Abaqus software were used and springback angle also measured by SolidWorks 2012. Tensile test results were determined and the mechanical properties will be used to run Finite Element Simulation.

4.2 TENSILE TEST RESULT

In the tensile test process, force (load) and extension data has been monitored and recorded. Force (load) and extension data that provided from this test can be use to find several important mechanical properties of a Galvanized Iron 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. But, the machine can also provided directly the mechanical properties value if the method is set up to determined the value. Table 4.1 had shown the mechanical properties value that has been selected in the tensile test method.

Table 4.1 : Mechanical Properties from Tensile Test Result

Load at Break (Standard) (KN)	Maximum Load (KN)	Modulus (Automatic) (MPa)	Strain hardening exponent at n-value (Automatic)
1.23	6.50	38,029	0.27747

Using the recorded data (load and extension), the engineering stress is found by dividing the applied load by the specimen original cross sectional area.

$$\sigma_{eng} = \frac{P}{A_0} \qquad \text{Equation 4.1}$$

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

$$\varepsilon_{eng} = \frac{\delta}{l_0} = \frac{l - l_0}{l_0} \qquad \text{Equation 4.2}$$

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

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

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

Table 4.2 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 4.2 is 10 second initial value of the data.

Table 4.2 : 10 second initial value of the data

Time(s)	Extension (mm)	Load (kN)	Engineering strain,(mm/mm)	Engineering stress(Mpa)
0	0	0.00041	0	23.31893
1	0.0835	0.1012	0.00167	36.312
2	0.16687	0.66914	0.00334	48.96533
3	0.25019	1.39566	0.005	61.71147
4	0.3335	1.98228	0.00667	74.1344
5	0.41681	2.51023	0.00834	86.72053
6	0.50019	2.9916	0.01	99.2768
7	0.58344	3.44227	0.01167	111.1803
8	0.66681	3.83883	0.01334	123.1803
9	0.75019	4.19026	0.015	134.7952
10	0.8335	4.10847	0.01667	146.2901

Based on the tensile test result above, the Stress-Strain graph can be plotted. The graph can be plotted by using the engineering stress and strain. Figure 4.1 below shown Stress-Strain graph for Galvanized Iron material having a thickness of 1.0 mm with different orientation angle (R).

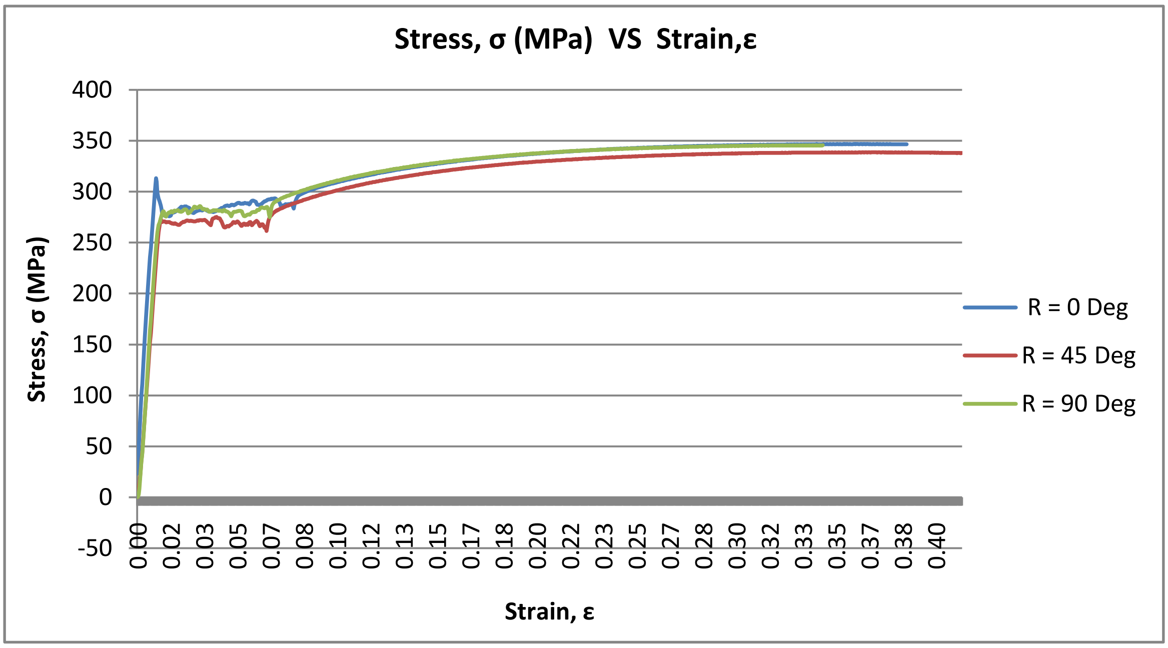


Figure 4.1: Different Orientation Angle of Stress-Strain Graph for GI 1.0mm Thickness

Another mechanical properties for Galvanized Iron such as anisotropy value and poison's ratio can be determined if the final width and length of specimens was measured. Table 4.3 shown the average width and length after the test.

Table 4.3: Final Width and Length of Specimen.

Orientation Angle	Final Width,	Final Length,
Average 0	9.29333333	23.8560
Average 45	9.12444444	21.5998
Average 90	9.01222222	23.1181
Average Result For 1.0mm Thickness	9.14333333	22.8579

Anisotropy value, R is defined to express different contractile strain ratio and is generally applied as an index of anisotropy. Due to the difficulty in measuring gage thickness changes with sufficient precision, an equivalent relationship is commonly used, based on length and width strain measurements, as shown in eq. 4.5.

$$R = \frac{\varepsilon_w}{-(\varepsilon_l+\varepsilon_w)} = \frac{\ln(w_0/w_f)}{\ln(l_f w_f/l_0 w_0)} \qquad \text{Equation 4.5}$$

Where ε_w and ε_l are true strains in width and length directions, w_0, w_f, l_0 and l_f are initial and final gage width and length, respectively. With most materials the change of R with strain ε_l is negligible. The ratio of the two normal strains (lateral and longitudinal) is a material constant called the poison's ratio, as shown in eq 4.6.

$$v = -\frac{\varepsilon_{lateral}}{\varepsilon_{longitudinal}} = -\frac{(w_f-w_0)/w_0}{(l_f-l_0)/l_0} \qquad \text{Equation 4.6}$$

Wherew_0, w_f, l_0 and l_f are initial and final gage width and length, respectively. Table 4.4 below shown the final summaries for the mechanical properties of Galvanized Iron.

Table 4.4: Mechanical Properties of Galvanized Iron

Orientation Angle	**Young's Modulus (MPa)**	**Strain Hardening Exponent, n**	**Ultimate Tensile Strength, UTS (MPa)**	**Anisotropy Value, R**	**Poisson's Ratio, v**
Average 0	32359.843	0.27747	346.7541333	0.26373	0.612382
Average 45	26381.335	0.27838	338.6325333	0.25365	0.643850
Average 90	29002.086	0.26750	345.7738667	0.24181	0.781632
Average Result For 1.0mm Thickness	29247.754	0.27445	343.7201778	0.253063	0.679288

4.3 FE SIMULATION RESULTS FOR U-BENDING

The elastically - driven change of shape of sheet metal have been simulated with ABAQUS code. Due to plane symmetry, only half of the process was modelled. The problem consists of the surface contact between steel blank strip and the tools such as the punch, die and blank holder that is a basic aspect of the stamping operations. The tools can be modelled as rigid surfaces because they are much stiffer than the blank. Figure 4.2 shows the basic arrangement of the components considered in FEM model. The blank strip is squeezed between the blank holder and the die trough a normal load applied on the blank holder while the die remains always unmoving. The contact between the punch and the blank was supposed to be frictionless whereas the contacts between respectively the blank and the die and the blank and the blank holder were supposed to have a coulomb friction law with a friction coefficient of 0.1.

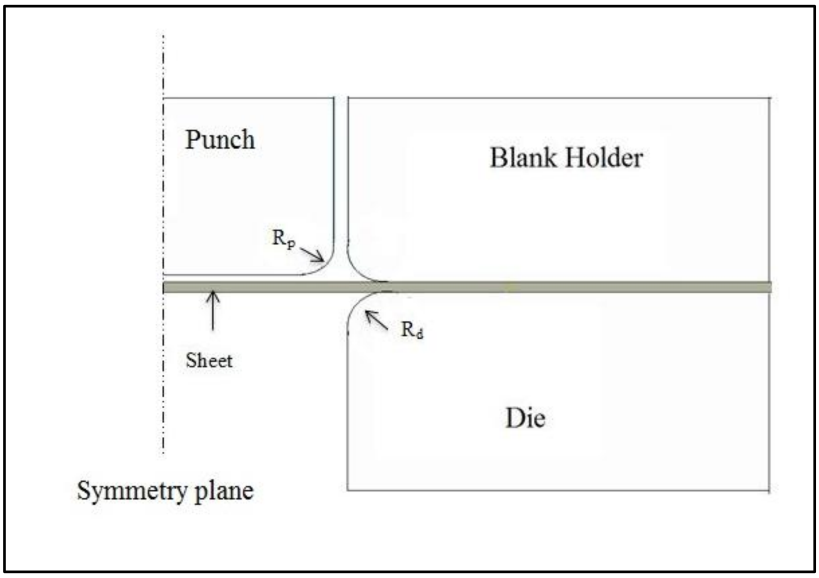


Figure 4.2 : Geometrical description of the simulation model

4.3.1 Simulation of sheet metal bending.

Figures 4.3 shows a condition when the punch is start touching the workpiece. The punch will slowly bend the workpiece in downward direction.

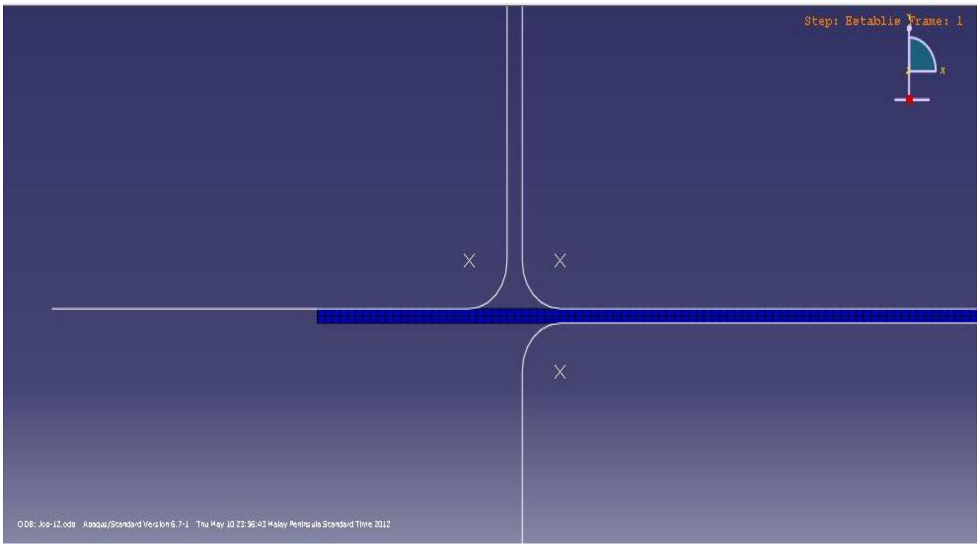


Figure 4.3 : Punch start touching the workpiece

Figure 4.4 shows a condition during bending and figure 4.5 shows a condition where the punch is moving at maximum displacement. At this condition, the sheet metal is totally bended in this simulation. Lastly, the punch will moving upward then the sheet metal will start springback. This process is showing in Figure 4.6.

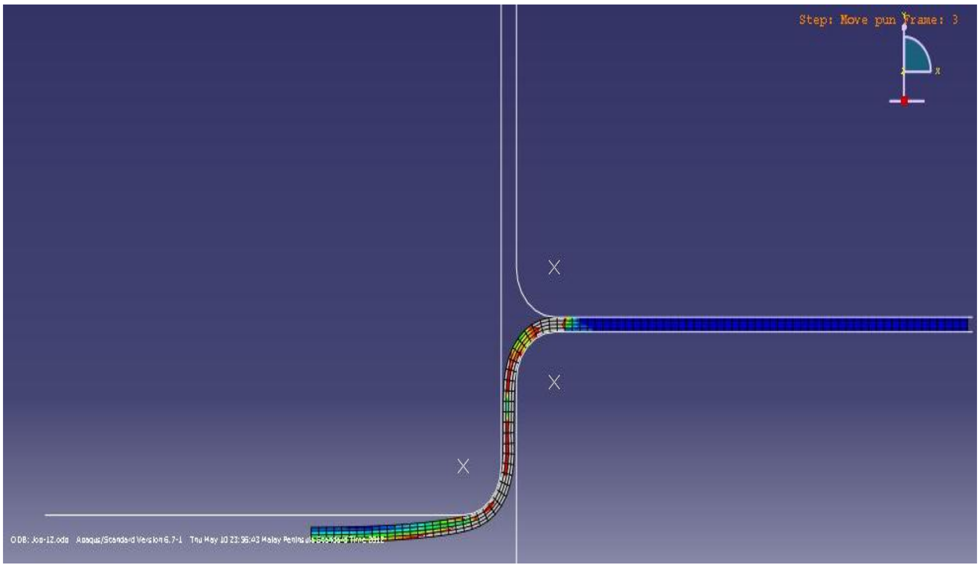


Figure 4.4 : During Bending

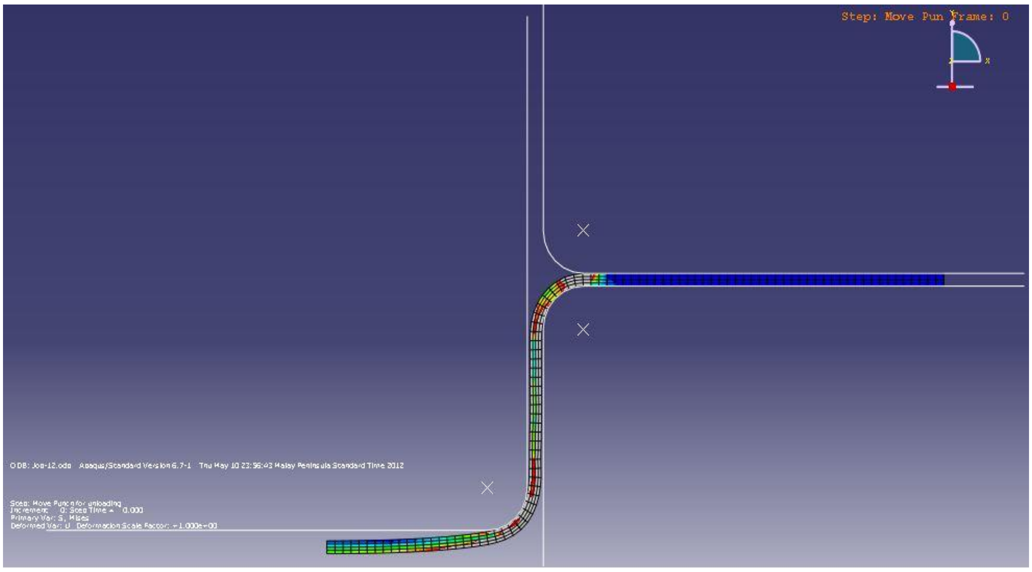


Figure 4.5 : Maximum movement of the punch

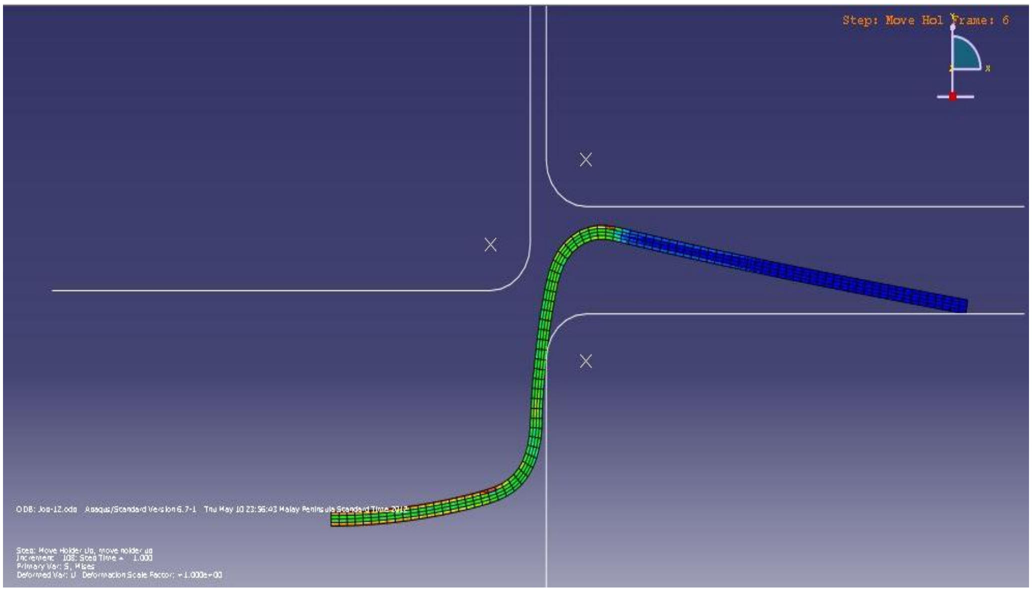
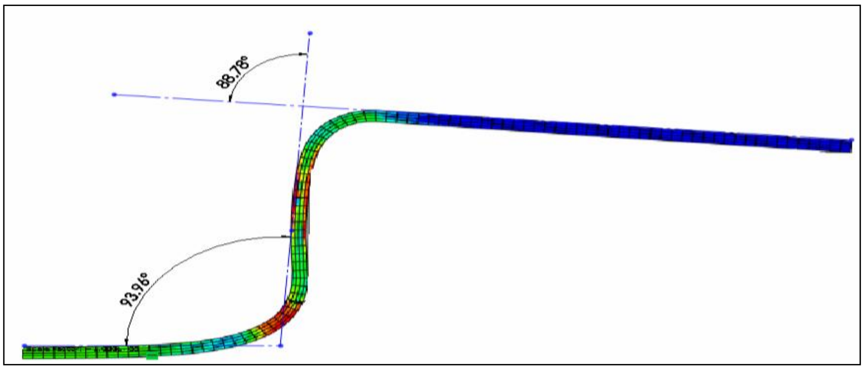


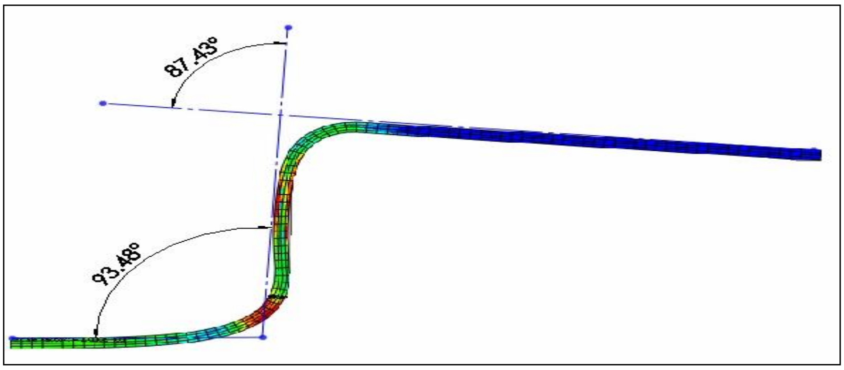
Figure 4.6 : After bending (Punch release)

4.3.2 Springback measurement from solidworks

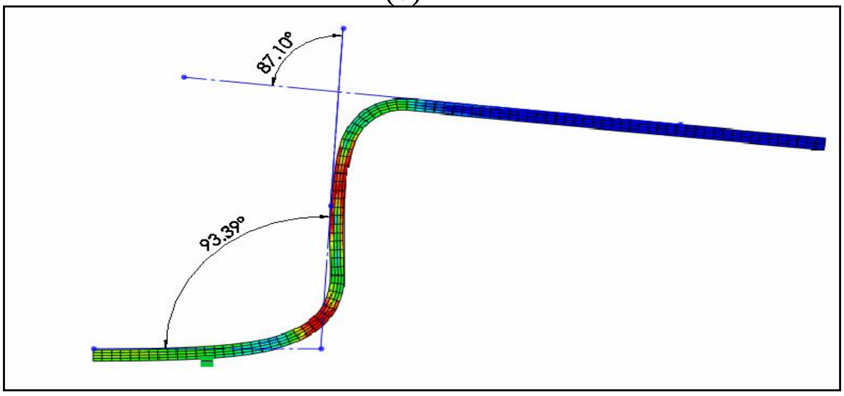
To measure a springback angle for a specimen test from U-bending using experiment technique, we scan a specimen test that done from the experiment first and import a picture to solidwork software. Then the angle for springback angle has been measured based on figure 3.16. Figure 4.7 below show a springback angle that has been measured by using solidwork for GI 1.0mm thickness with different rolling direction.



(a)



(b)



(c)

Figure 4.7 : (a) 0 degree rolling direction for Galvanized Iron 1.0mm thickness
(b) 45 degree rolling direction for Galvanized Iron 1.0mm thickness
(c) 90 degree rolling direction for Galvanized Iron 1.0mm thickness

The main objective of FEA is to predict springback for U-bending tests 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 truthfully validate materials parameters, both analyses were accomplished with similar operational conditions. This consisted of constant blank holder force of 440 KN. Table 4.5 shown the results of simulation for Galvanized Iron 1.0 mm thickness.

Table 4.5 : Simulation result of springback for Galvanized Iron.

Rolling direction	0^0	45^0	90^0
Springback angle 1, Θ_1, (^{o})	93.39	93.48	93.96
Springback angle 2, Θ_2, (^{o})	87.10	87.43	88.78

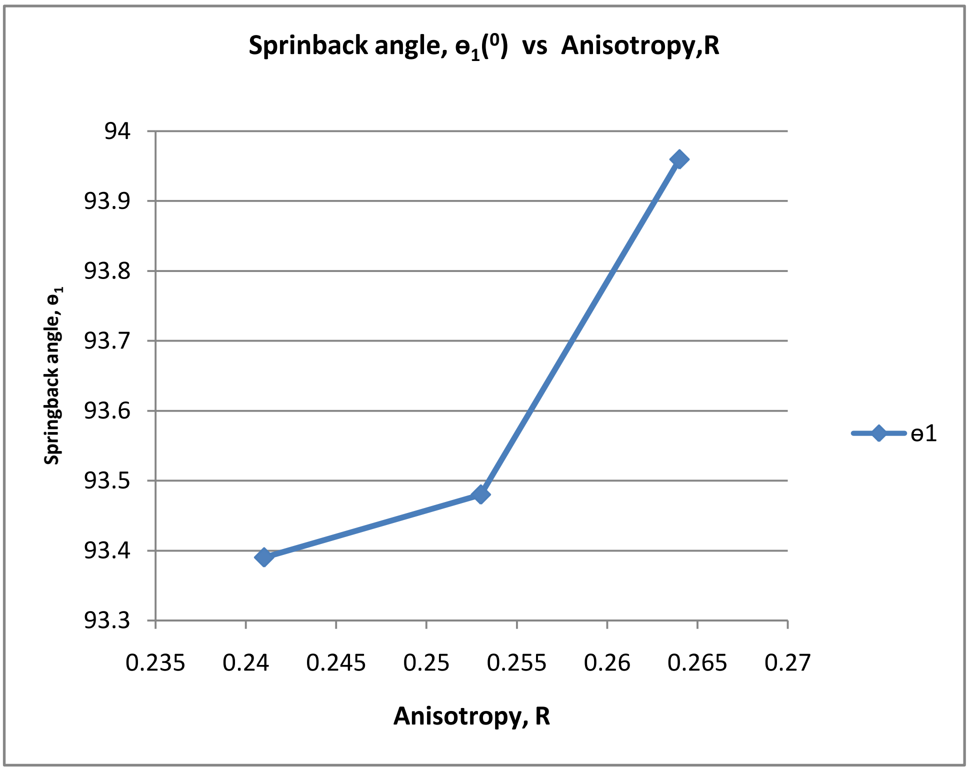


Figure 4.8: Anisotropy effect on amount of springback angle θ_1.

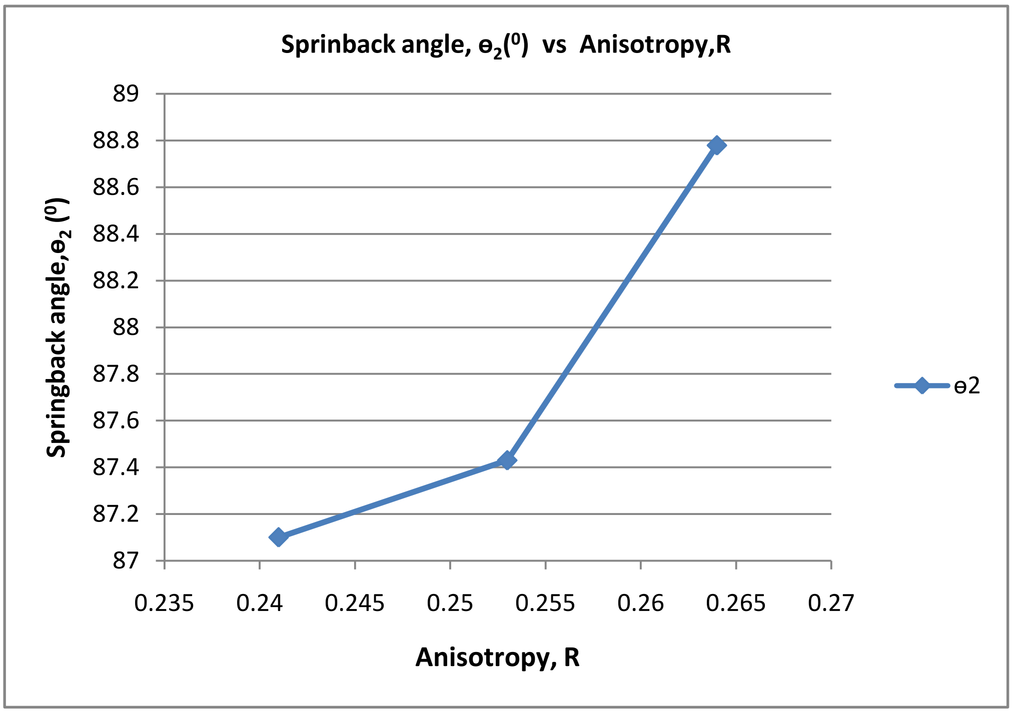


Figure 4.9: Anisotropy effect on amount of springback angle θ_2.

4.3.3 Effect of anisotropy on springback

Figure 4.8 shows that anisotropy effect on the springback , the line indicated that the values of springback angle , Θ_1 are increasing as the anisotropy value, R increasing. It is noted that, for the highest springback, sheet metal with the higher value of anisotropy is not good. Same goes to springback angle, Θ_2 , the value will increase as the anisotropy increase and conclude that springback are affected with the anisotropy value as shown in figure 4.9.

4.4 U-BENDING EXPERIMENTAL RESULTS

Based on experimental study, springback results were evaluated and these results were used in order to validate the proposed finite element calculation. After the unloading of blank holder section, springback values Θ_1 and Θ_2 were determined. Schematic descriptions of measurement position for springback are given in the Figure 4.10.

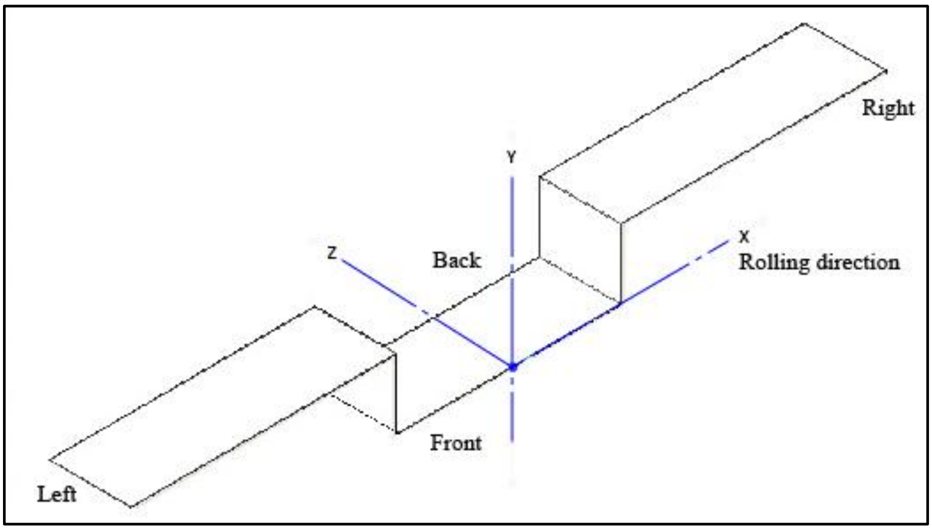


Figure 4.10 : Schematic description of measurement position for springback.

A 2D drawing of Galvanized Iron 1.0mm thickness after springback was shown on Figure 4.10. The 0 degree, 45 degree and 90 degree of Galvanized Iron specimens after removing from die also shown on Figure 4.11.

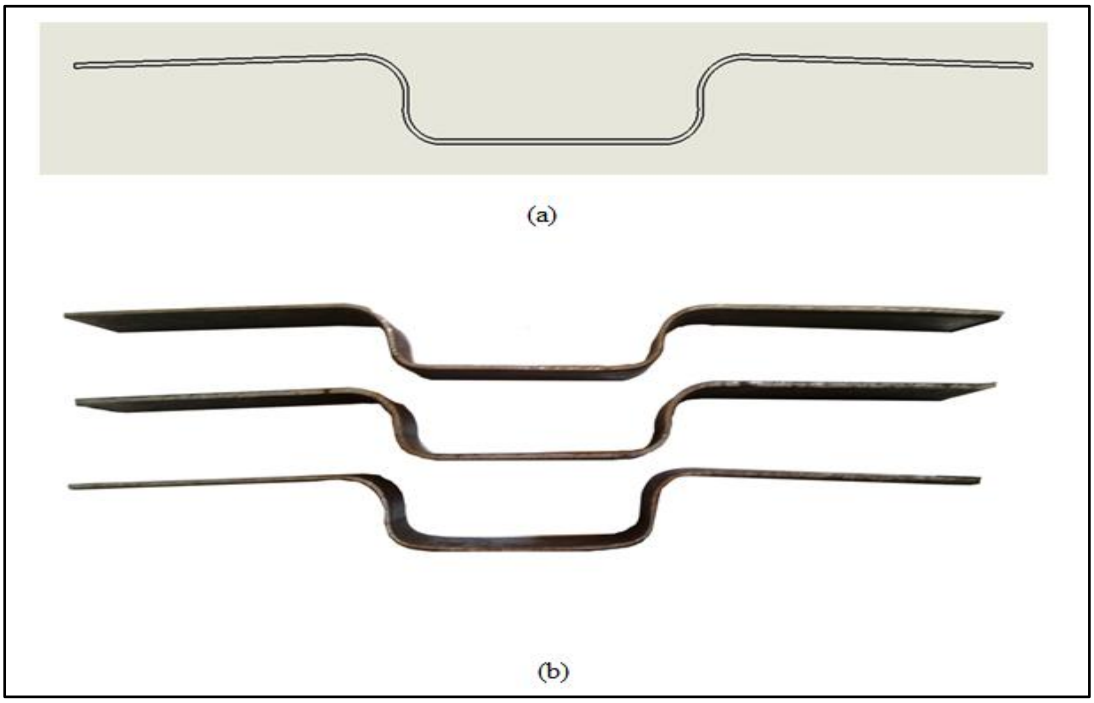


Figure 4.11 : (a) 2D drawing of Galvanized Iron after bending,
(b) Galvanized Iron specimen after bend

The calculations of springback were performed according to Figure 3.16. The specimen will be scanned by a scanner and the angle, θ_1 and θ_2 then will be measured by using Solidworks. The springback angle for each specimen then will be compared. There were nine specimens for U-bending test and the result obtained for Galvanized Iron plates for rolling direction 0 , 45 , and 90 degree were given in table 4.6.

4.4.1 Springback measurement using SolidWorks 2012

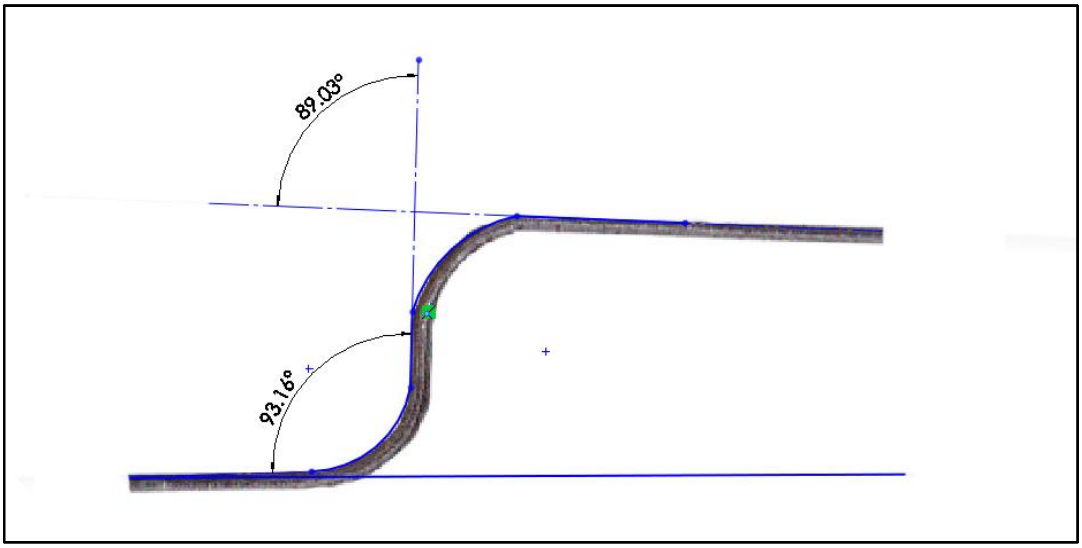


Figure 4.12 : Specimen with rolling direction 0 degree.

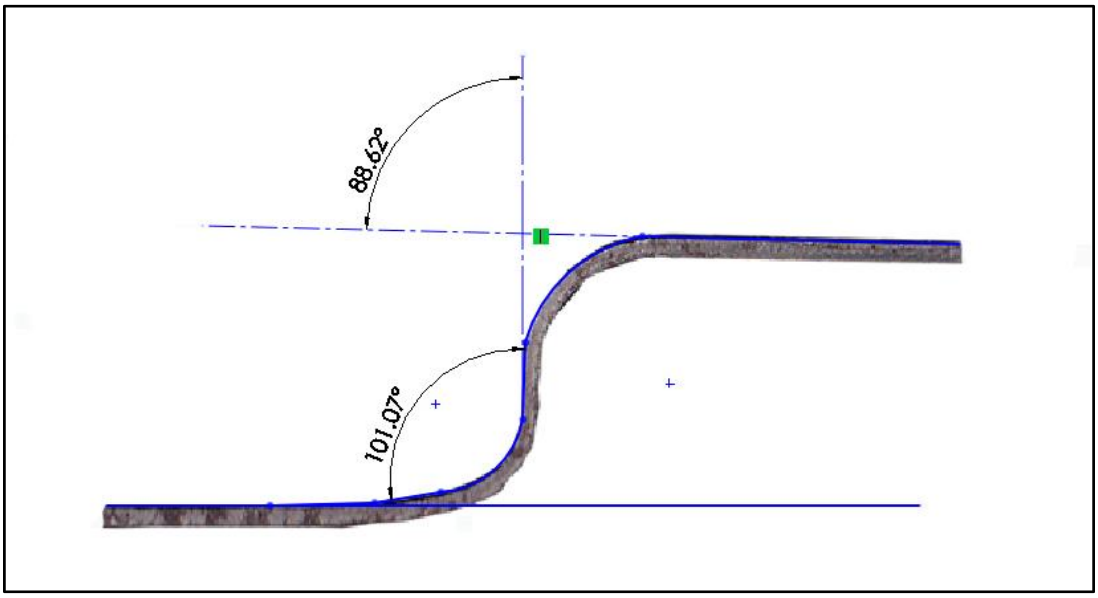


Figure 4.13 : Specimen with rolling direction 45 degree.

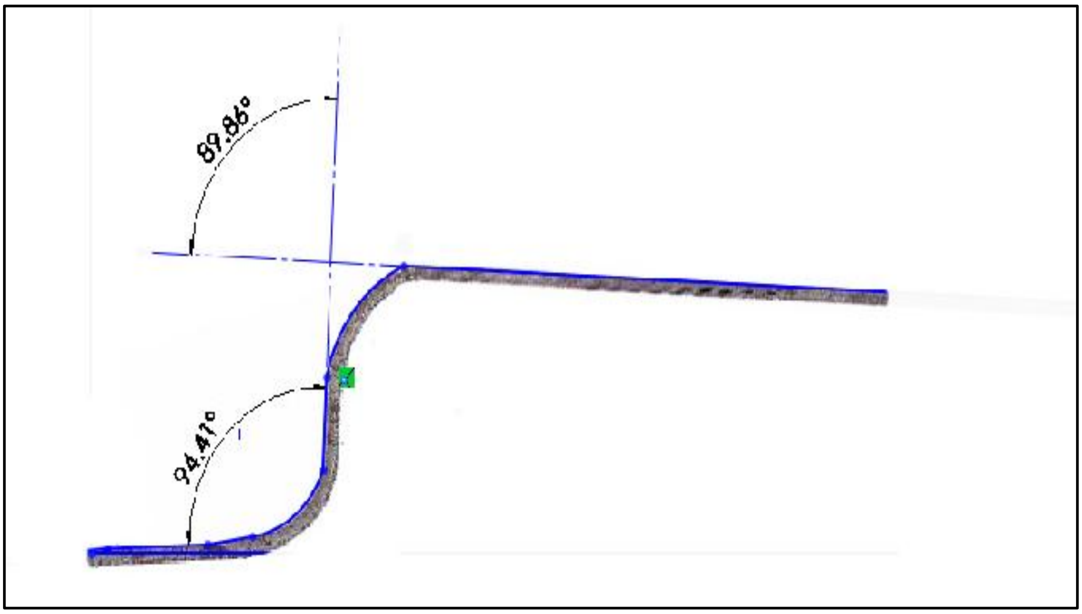


Figure 4.14 : Specimen with rolling direction 90 degree.

Table 4.6: Experimentally measured parameters of springback.

Rolling direction (Degree,)	0^0				45^0				90^0			
	1	2	3	Ave	1	2	3	Ave	1	2	3	Ave
Springback angle 1 ϴ1, (°)	92.35	90.89	93.16	**99.22**	97.13	101.07	99.46	**92.13**	93.05	94.47	94.59	**94.04**
Springback angle 2 (ϴ2),(°)	89.55	88.53	89.03	**89.04**	88.96	88.62	88.77	**88.78**	88.75	89.86	88.23	**88.95**

In this study, the springback variation in the forming of Galvanized Iron was characterized by investigating the effect of different rolling direction (anisotropy) which was 0, 45 and 90 degree. Their significance parameters on springback angles were investigated to see if they can be used to minimized springback angle. Effect of orientation angle on springback has shown in figure 4.15 and 4.16 below.

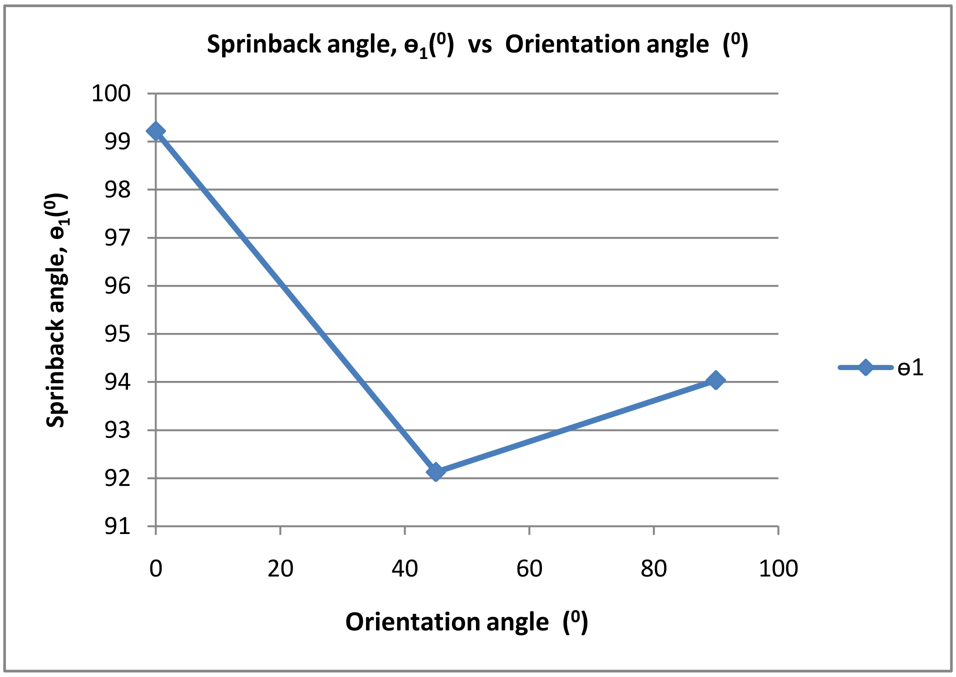


Figure 4.15: Effect of orientation angle on springback, $ɵ_1$.

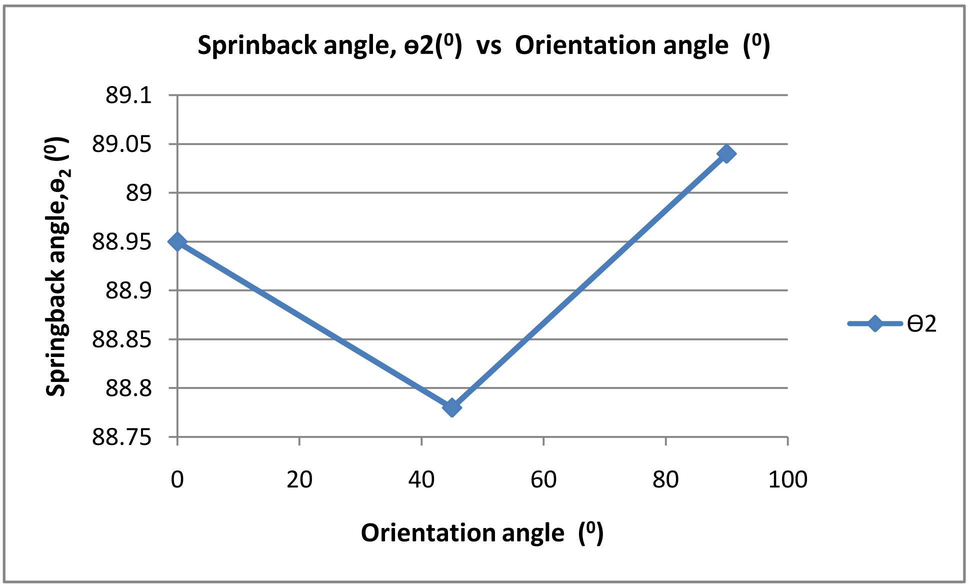


Figure 4.16 : Effect of orientation angle on springback, $ɵ_2$.

4.4.2 Effect of orientation angle on springback

Test were performed under three different orientation angle started from 0 degree, 45 degree and 90 degree. From Figure 4.15 and Figure 4.16, it can be seen that the springback angle decrease as the orientation increase, but for the orientation angle 90 degree, the springback angle is increasing for springback angle, Θ_1 and Θ_2. This is because of the experimental error occurred which, after the stamping machine bent the sheet metals for 0 and 45 degree, the machine then stopped operating. For bending sheet metals of 90 degree orientation angle, the loading force used is not the same with the previous experiment, and caused the blank holder force applied is not constant.

4.5 RESULTS COMPARISON

Table 4.7 shown the result of springback angle ,Θ_1 and springback angle , Θ_2 for both FE Simulation and experimental. Graph shown the comparison of springback angle between experimental and simulation.

Table 4.7 : Springback values for experimental and simulation.

Orientation angle	**0^0**		**45^0**		**90^0**	
	Θ1	Θ2	Θ1	Θ2	Θ1	Θ2
Experimental	99.22	89.04	92.13	88.78	94.04	88.95
FE Simulation	93.39	87.10	93.48	87.43	93.96	88.78
Percentage of error POE (%)	5.88	2.08	1.47	1.52	0.09	0.29

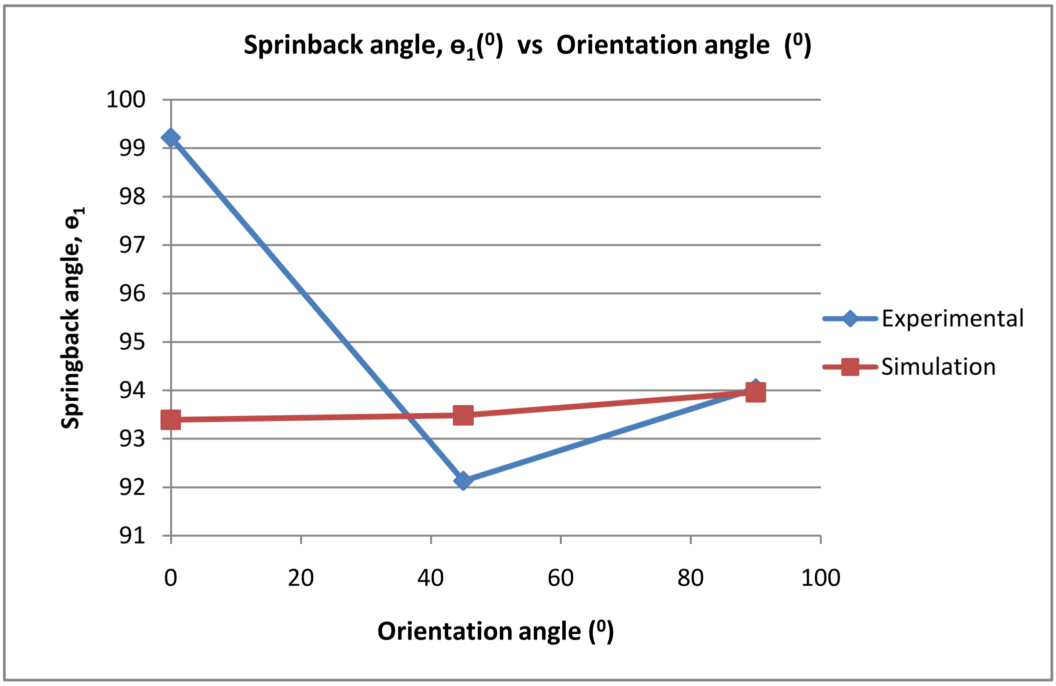


Figure 4.17 : Comparison between simulation and experimental for θ_1

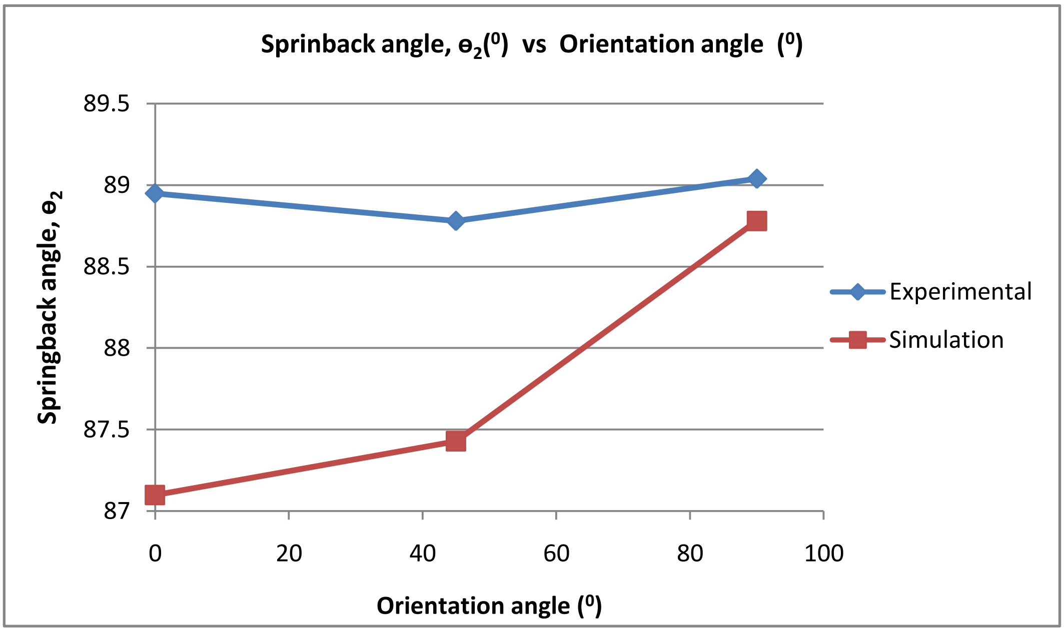


Figure 4.18 :Comparison between simulation and experimental for θ_2

Figure 4.17 and Figure 4.18 show the experimental and simulation effect of orientation angle on the springback of the Galvanized Iron material for 1.0 mm thickness. Comparing the result of the springback angle through the orientation angle, it is noted that orientation angle are strongly affected the springback angle. For springback angle from the simulation, by increasing the orientation angle, will increasing the springback value. But for experimental value, the graphs show that 45 degree has the lowest springback. This difference occurred due to the experimental that have been mentioned earlier and sheet metal bending belonging to out of plane forming process is characterized by small strain but large deformation, as well the frictional contact boundary changes during the process. This implies that the finite element analysis of sheet metal bending process is quite difficult and a lot of parameters need to be considered.

CHAPTER 5

CONCLUSION

5.1 INTRODUCTION

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

5.2 CONCLUSION

Based on the study, the following remarks are drawn :

1. The amount of springback angle are in the range of :
   1. Springback angle , θ_1 : 99.22^0 to 92.13^0. This is proven on Figure 4.17.
   2. Springback angle, θ_2 : 89.04^0 to 87.10^0 . This is proven on Figure 4.18.
2. Orientation angle are strongly affected the amount of springback which increasing the orientation angle will increasing the springback.
3. Finite Element Analysis can be used to predict springback since the pattern of the graphs compared are nearly the same and the percentage of error, (POE) are less than 10 %.

5.3 RECOMMENDATIONS

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

(i) Using a variety of materials in the experiment and simulation such as Aluminium or Stainless Steel to investigate which material that have a less springback.

(ii) Using different thickness for every material used to investigate the effect of thickness on springback.

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

(iv) Consider the blank holder force since these parameters are strongly affected the amount of springback.

REFERENCES

Z. Marciniak and Claes Magnusson (2002). Sheet metal forming. *Journal of Materials Processing Technology,* 65, 52-58.

Ozgur Tekaslan, Ulvi Seker, Ahmet Ozdemir (2008). Determining springback amount of steel sheet metal has 0.5mm thickness in bending dies. *Journal of Materials and Design,* 29, 1043-1050.

Olaf Diegel, Daw-Kwei Leu (2002). A simplified approach for evaluating bendability and springback in plastic bending of anisotropic sheet metal. *Journal of Materials Processing Technology,* 66, 9-17.

V. Esat, B. Rahmani, V. Daeezadeh, A.Gorji (2009). The study of springback of CK67 steel sheet in V-die and U-die bending processes. *Journal of Materials and Design,* 30 2410-2419.

J.R. Cho, S.J. Moon, Y.H. Moon, S.S. Kang (2003). Finite element investigation on spring-back characteristics in sheet metal U-bending. *Journal of Materials Processing Technology* 141 (2003) 109–116.

M. Bakhshi-Jooybari , B. Rahmani, V. Daeezadeh, A. Gorji (2009). The study of spring-back of CK67 steel sheet in V-die and U-die bending processes. *Journal of Materials and Design* 30 (2009) 2410–2419.

D. Zhang (2007). Investigation of spring-go phenomenon using finite element method, *Journal of Materials and Design*, 29, 1526-1532.

M. Firat (2005). U-channel forming analysis with an emphasis on springback deformation. *Journal of Materials and Design* 28 (2007) 147–154

Volkan Esat, Haluk Darendeliler, Mustafa Ilhan Gokler (2002). Finite element analysis of springback in bending of aluminium sheets. *Journal of Materials and Design*, 23, 223-229.

YU Zhong-qi and D. Lee (2007). Development of a New Model for Plane Strain Bending and Springback Analysis. *Journal of Materials Engineering and Performance*, 4291-300.

Michael F.Ashby (2005). A simplified approach for evaluating bendability and springback in plastic bending of anisotropic sheet metal. *Journal of Materials Processing Technology,* 66, 9-17.

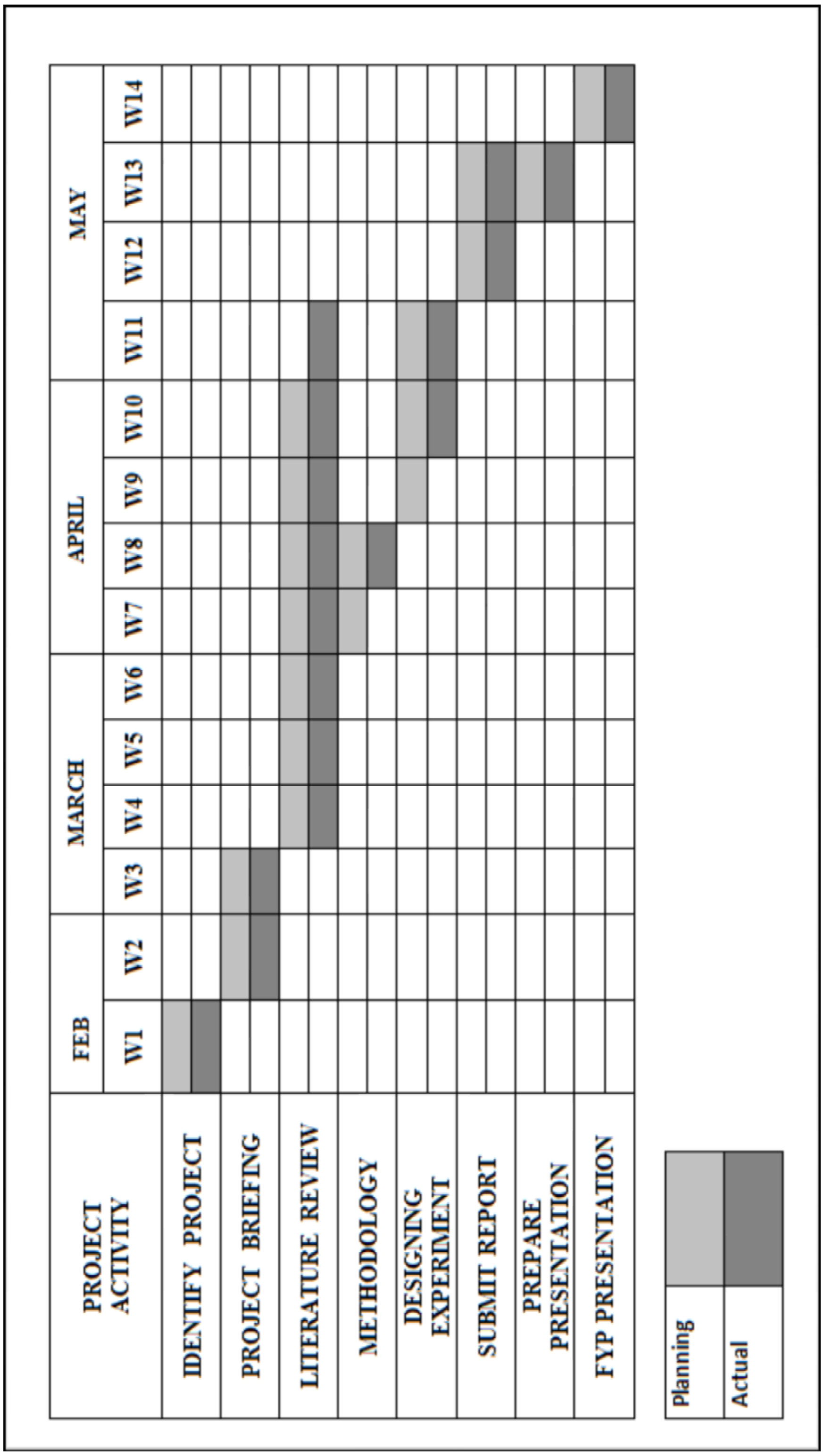
M. Samuel (2000). Experimental and numerical prediction of springback and side wall curl in U-bendings of anisotropic sheet metals. *Journal of Materials Processing Technology,* 105 (2000) 382±393

J.R. Davis (2004). *Tensile testing*. 2nd edition. USA: ASM International.

ASTM (2003). *Annual Book of ASTM Standards Vol 01.03,* USA: ASTM International

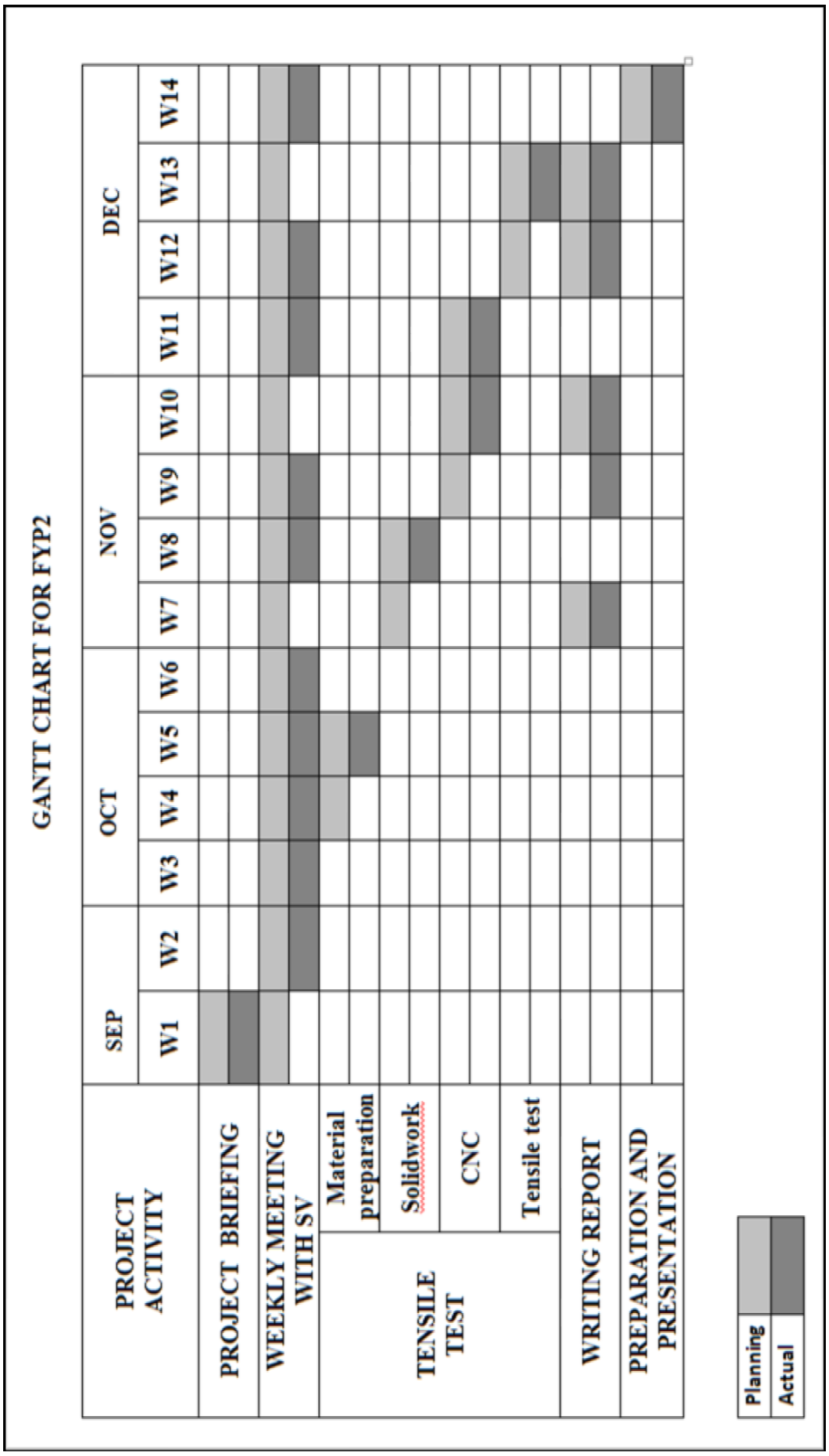
APPENDICES

APPENDIX A



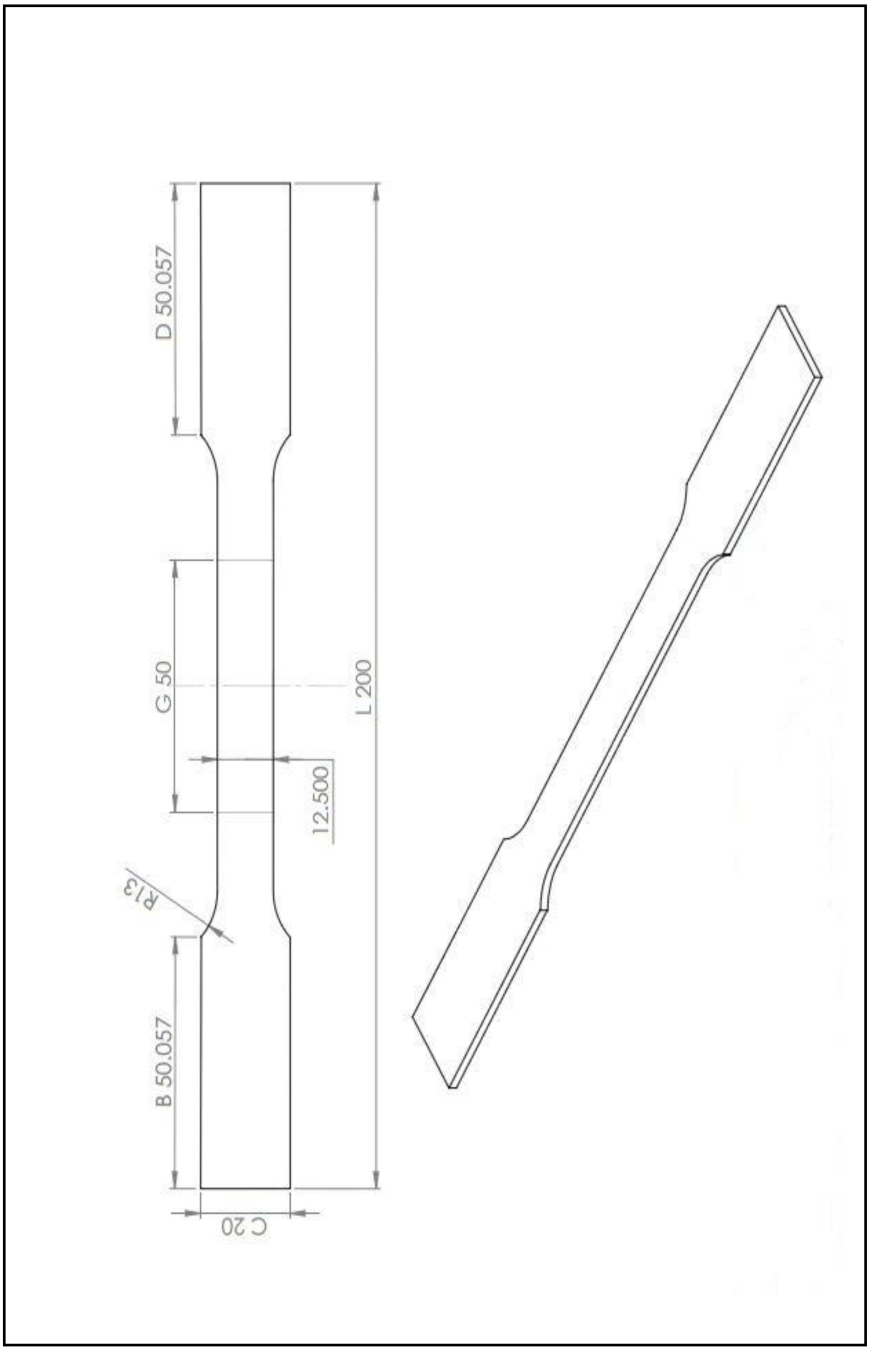
Final Year Project 1 gantt chart

APPENDIX B



Final Year Project 2 gantt chart

APPENDIX C



Tensile Specimen

APPENDIX D

```
O0000(G CODE T BONE ST 3)
(DATE=DD-MM-YY - 26-01-12 TIME=HH:MM - 11:08)
(MCX FILE - F:\T-BONE PSM\T-BONE STAINLESS STEEL.MCX-5)
(NC FILE - C:\USERS\PTMK\DESKTOP\T-BONE PSM\G CODE T BONE ST 3.NC)
(MATERIAL - STAINLESS STEEL)
( T1 |     16. FLAT ENDMILL | H1 )
N100 G21
N102 G0 G17 G40 G49 G80 G90
N104 T1 M6
N106 G0 G90 G54 X-100. Y18. A0. S900 M3
N108 G43 H1 Z25. M8
N110 Z5.
N112 G1 Z-2.5 F10.
N114 X-42.85 F40.
N116 X-40.513 Y15.692
N118 G3 X-37. Y14.25 R5.
N120 G1 X37.
N122 G3 X40.513 Y15.692 R5.
N124 G1 X42.85 Y18.
N126 X100.
N128 G0 Z25.
N130 X-100. Y-18.
N132 Z5.
N134 G1 Z-2.5 F10.
N136 X-42.85 F40.
N138 X-40.513 Y-15.692
N140 G2 X-37. Y-14.25 R5.
N142 G1 X37.
N144 G2 X40.513 Y-15.692 R5.
N146 G1 X42.85 Y-18.
N148 X100.
N150 G0 Z25.
N152 M5
N154 G91 G28 Z0. M9
N156 G28 X0. Y0. A0.
N158 M30
```

CNC Milling G-Code