EXPERIMENTAL AND FINITE ELEMENT EVALUATION OF BENDING FOR STAINLESS STEEL

MOHAMAD JAZMIR HASSAN BIN MOHAMED

Report submitted in fulfilment of
The requirements for the award 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 and in my opinion, this project is adequate 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 OF MECHANICAL ENGINEERING
Date	:	

Signature	:	
Name of Second Reviwer	:	DR. AHMAD SYAHRIZAN SULAIMAN
Position	:	LECTURER OF MECHANICAL ENGINEERING
Date	:	

STUDENT'S DECLARATION

I hereby declare that the work in this project 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 of Student	:	MOHAMAD JAZMIR HASSAN BIN MOHAMED
ID Number	:	STUDENT OF MECHANICAL ENGINEERING
Date	:	

Dedicated to my father, Mr. Mohamed bin Muda, my beloved mother, Mrs. Jawahir bt Che Omar, my brothers Mohamad Jazril Yusof bin Mohamed,Mohamad Jazlan Omar bin Mohamed and Mohamad Jazman Ismail bin Mohamed, my sister Nurul Arina Akmal binti Mohamed and last but not list to all my beloved fellow friends.

ACKNOWLEDGEMENTS

I am grateful and would like to express my sincere gratitude to my supervisor Mr. Jasri bin Mohamad for his germinal ideas, invaluable guidance, continuous encouragement and constant support in making this research 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 graduate program to these concluding moments. I am truly grateful for his progressive vision about my training in science, his tolerance of my naive mistakes, and his commitment to my future career.

My sincere thanks go to all my lab mates and members of the staff of the Mechanical Engineering Department, UMP and TATIUC, who helped me in many ways and made my stay at UMP pleasant and unforgettable.

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

ABSTRACT

The finite element method (FEM) (its practical application often known as finite element analysis (FEA)) is a numerical technique for finding approximate solutions of partial differential equations (PDE) as well as integral equations. The solution approach is based either on eliminating the differential equation completely (steady state problems), or rendering the PDE into an approximating system of ordinary differential equations, which are then numerically integrated using standard techniques such as Euler's method, Runge-Kutta.

In solving partial differential equations, the primary challenge is to create an equation that approximates the equation to be studied, but is numerically stable, meaning that errors in the input and intermediate calculations do not accumulate and cause the resulting output to be meaningless. There are many ways of doing this, all with advantages and disadvantages.

The finite element method is a good choice for solving partial differential equations over complicated domains (like cars and oil pipelines), when the domain changes (as during a solid state reaction with a moving boundary), when the desired precision varies over the entire domain, or when the solution lacks smoothness. For instance, in a frontal crash simulation it is possible to increase prediction accuracy in "important" areas like the front of the car and reduce it in its rear (thus reducing cost of the simulation). Another example would be in Numerical weather prediction, where it is more important to have accurate predictions over developing highly nonlinear phenomena (such as tropical cyclones in the atmosphere, or eddies in the ocean) rather than relatively calm areas.

ABSTRAK

Kaedah unsur terhingga (FEM) (permohonan praktikal yang sering dikenali sebagai analisis unsur terhingga (FEA)) adalah satu teknik berangka untuk mencari penyelesaian hampir persamaan pembezaan separa (PDE) serta persamaan kamiran. Pendekatan penyelesaian adalah berdasarkan sama ada untuk menghapuskan persamaan pembezaan sepenuhnya (masalah keadaan mantap), atau menyebabkan PDE ke dalam sistem yang hampir persamaan pembezaan biasa, yang kemudiannya berangka bersepadu menggunakan teknik standard seperti kaedah, Euler Kaedah Runge-Kutta.

Dalam menyelesaikan persamaan pembezaan separa, cabaran utama adalah untuk mewujudkan satu persamaan yang menghampiri persamaan untuk dikaji, tetapi berangka stabil, yang bermaksud bahawa kesilapan dalam pengiraan input dan pertengahan tidak terkumpul dan menyebabkan output yang terhasil menjadi sia-sia. Terdapat banyak cara untuk berbuat demikian, semua dengan kelebihan dan kekurangan.

Kaedah unsur terhingga merupakan pilihan yang baik untuk menyelesaikan persamaan pembezaan separa atas domain rumit (seperti kereta dan saluran paip minyak), apabila perubahan domain (seperti semasa tindak balas keadaan pepejal dengan sempadan yang bergerak), apabila ketepatan yang dikehendaki berubah seluruh domain , atau apabila penyelesaian kekurangan kelancaran. Sebagai contoh, dalam simulasi kemalangan hadapan, ia adalah mungkin untuk meningkatkan ketepatan ramalan di kawasan "penting" seperti bahagian hadapan kereta dan mengurangkan di belakang (sekali gus mengurangkan kos simulasi). Contoh yang lain akan berada dalam ramalan cuaca berangka, di mana ia adalah lebih penting untuk mempunyai ramalan yang tepat lebih membangun fenomena yang sangat tak linear (seperti siklon tropika di atmosfera, atau pusaran di lautan) daripada kawasan yang agak tenang.

TABLE OF CONTENTS

TABLE OF CONTENTS 1

LIST OF FIGURES 4

LIST OF TABLES 5

LIST OF SYMBOLS 6

LIST OF ABBREAVIATION 7

CHAPTER 1 8

INTRODUCTION 8

1.1 Introduction 8

1.2 Project Background 8

1.3 Objectives 9

1.4 Scope of Works 9

CHAPTER 2 10

LITERATURE REVIEWS 10

2.1 Introduction 10

2.2 Sheet Metal Bending Process 10

2.3 Theory of Bending 13

2.4 Type of Bending 14

2.4.1 V-Bending 14

2.4.2 U-bending 17

2.5 Improvement of Springback 17

2.6 Previous Works 19

CHAPTER 3 21

METHODOLOGY 21

3.1 Introduction 21

3.2 Flow Chart 21

3.3 Specimen Dimension ... 22

3.4 Test Specimen Preparation ... 22

3.4.1 Cutting Raw Materials ... 23

3.4.2 Cutting Tensile Specimen ... 23

3.5 Tensile Test Experiment ... 27

3.6 Finite Element ... 29

3.7 U-Bending Experiment ... 31

3.7.1 Introduction ... 31

3.7.2 Objective ... 31

3.7.3 Modification Die ... 31

3.7.4 Process ... 32

CHAPTER 4 ... 35

RESULTS AND DISCUSSIONS ... 35

4.1 Introduction ... 35

4.2 Tensile Test Result ... 35

4.3 Experimental Results ... 41

4.3.1 Springback Measurement ... 41

4.3.2 Angle Results ... 42

4.4 Finite Element Results ... 43

4.5 Comparison Results ... 44

CHAPTER 5 ... 45

CONCLUSION ... 45

5.1 Conclusion ... 45

5.2 Summary of study ... 45

5.3 Recommendation ... 46

REFERENCES ... 46

APPENDIX A ... 48

(Die Drawing) ... 48

APPENDIX B ... 55

(Guntt Chart) 55

APPENDIX C.......... 57

(Specimen Dimension) 57

APPENDIX D 58

(G-Code).......... 58

APPENDIX E.......... 59

(PROJECT PROGRESS) 59

LIST OF FIGURES

Figure 2.1 Maxform press brake machine 12
Figure 2.2Bending sample 12
Figure 2.3 Bending process description 13
Figure 2.4 Coining Bending 15
Figure 2.5 Bottoming Bending 16
Figure 2.6 Air Bending 17
Figure 3.1 Methodology 21
Figure 3.2 E8M standard specimen 22
Figure 3.3 LVD shearing machine 23
Figure 3.4 Design Test Specimen using Mastercam 24
Figure 3.5 CNC Milling Process Simulation using Mastercam 25
Figure 3.6 Haas CNC Milling Machine 25
Figure 3.7 Basic Processes to Running The Milling Machine 26
Figure 3.8 (a) Milling Process (b) Semi-Finish Specimen 27
Figure 3.9 Specimen griped on machine 27
Figure 3.10 Tensile Experiment Procedure 28
Figure 3.11 Finished specimen 29
Figure 3.12 Steps in Abaqus 30
Figure 3.13 Angles in Finite Element 31
Figure 3.14 Die Drawing 32
Figure 3.15 Sheet metal placed 33
Figure 3.16 Bending angle 33
Figure 4.1 Different Angle of Stress-Strain Graph for Stainless Steel 1.5mm Thickness 37
Figure 4.2 Linear Stress-Strain Graph for 1.5mm Thickness 38
Figure 4.3 Selected point at region 40
Figure 4.4 Measurements parameters for the springback 41
Figure 4.5 2D-draw bending springback 42
Figure 4.6 Angles of 0 degree springback 42
Figure 4.7 Abaqus Simulation 43
Figure 4.8 Simulation measurement 43
Figure 4.9 Theta 1 comparison angles 44

LIST OF TABLES

Table 2.1 Summary of journal 20
Table 3.1 Number of specimen 23
Table 4.1 Mechanical Properties from Tensile Test Result 35
Table 4.2 Tensile Test Result for Stainless Steel 1.5mm Thickness 36
Table 4.3 Final Width and Length of Specimen 38
Table 4.4 Mechanical Properties of Stainless Steel AISI 304 39
Table 4.5 Plasticity data for FE 40
Table 4.6 Theta 1 and 2 42
Table 4.7 Angle of Finite Element 43
Table 4.8 Percentage error FE and Experimental 44

LIST OF SYMBOLS

w_o	Original width
w	Final width
lo	Original Length
l	Final length
F	Force
A_o	Original specimen area
Δl	Length difference
l_o	Original length
$\Delta\theta$	Springback Angle
θ	Punch angle(original angle)
R	Normal Anisotropy Value
n	Strain Hardening Exponent
v	Poisson Ratio
t	Thickness of sheet metal
E	Young Modulus
ρ	Neutral axis

LIST OF ABBREAVIATION

FEA	Finite Element Analysis
UTS	Ultimate Tensile Strength
CNC	Computer Numerical Control
ASTM	American Society of Testing Material

CHAPTER 1

INTRODUCTION

1.1 Introduction

Spring-back commonly happens in real world. It is occur when existing plasticity in sheet metal. One of the experiments to determine the process is bending process. 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. Spring-back is an issue in sheet metal forming processes. The spring-back 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.

1.2 Project Background

Bending is a process in which a sheet metal is plastically deformed to a curve by predicting the precision in angle which is a major concern in low carbon steel bending process. 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. 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

i. To determine selected material modeling accuracy in predicting springback using Finite Element Analysis (FEA).
ii. To determine FEA technology in sheet stainless steel forming or bending.

1.4 Scope of Works

i. Develop 2D modeling of U-bending in Finite Element Analysis (FEA) software.
ii. Develop 2D modeling in Finite Element Analysis (FEA) software for springback prediction. (Proposed: Patran, Algor, Abaqus, etc.)
iii. To conduct experiment of U-bending and simulation FEA
iv. To compare the simulation results with experimental results.

CHAPTER 2

LITERATURE REVIEWS

2.1 Introduction

Planning is very important initial step for making decision to start the project. Literature review is a guard line to find substances for projects. In other hands, they are a lot of researches doing the experimental and finite element analysis. In this chapter, ten journal have been analyzes for the project.

2.2 Sheet Metal Bending Process

Bending process is an important process in the sheet metal forming in many industries. The main problem of the bending process is spring-back phenomenon after removing the punch. This research aims to reduce spring-back value of sheet metal in U bending process. The conventional U bending and corner setting technique were designed for experiments. The corner setting technique has reduced the thickness in bending area to 5, 10, 15 and 20 percent of the original sheet thickness. Clearance between punch and die of both processes was equal to same thickness. The spring-back value was investigated by commercial program code Finite Element Analysis (FEA) 2D which able to analyze the stress and force in bending area.

Bending is a process by which metal can be deformed by plastically deforming the material and changing its shape. The material is stressed beyond the yield strength but below the ultimate tensile strength. The surface area of the material does not change much. Bending usually refers to deformation about one axis.

Bending is a manufacturing process that produces a V-shape, U-shape, or channel shape along a straight axis in ductile materials, most commonly sheet metal. The behavior of the material during the bend process is reflected in the stress/strain curve. In this, a distinction is made between the elastic region and the plastic region. Within the elastic region, the material returns to its original state as soon as the force is lifted. Past a certain critical value of the forces exerted, the material will show plastic deformation. This is referred to as the yield zone. A permanent change occurs in the structure of the sheet material. When bending sheet material, under the influence of the bend force applied, strain occurs on the outer side of the bend while material is upset on the inside. In other words, there is a transition from a tensile stress to a compressive strain over the cross-section of the sheet, by which the highest values are achieved on the two surfaces. Plastic deformation therefore first occurs on the outer sides of the bend sheet, and is the greatest there during the entire bend cycle. When creating a plastic deformation, the fact must be taken into consideration that the bend force may not be so large that the tensile stress on the outside becomes so great that the ultimate stress is exceeded and cracking occurs. Naturally, thicker sheets are the most sensitive to this phenomenon. This explains why the minimum values for the bend radius to be applied for air bending and bottoming are expressed as a function of the thickness. Here only elastic deformation occurs, even with high bend forces, which is why the sheet always springs back to some degree after the bend force is lifted. This springback in thin material up to approximately 2 mm / .078” can be limited to approximately 1° by selecting the right bending parameters. With thicker sheets, especially stainless steel, this will soon increase to even 3°. Corrections can be made to compensate for the springback by modifying the bend angle of the punch.(Tekaslan et al. 2004)

Figure 2.1 Maxform press brake machine

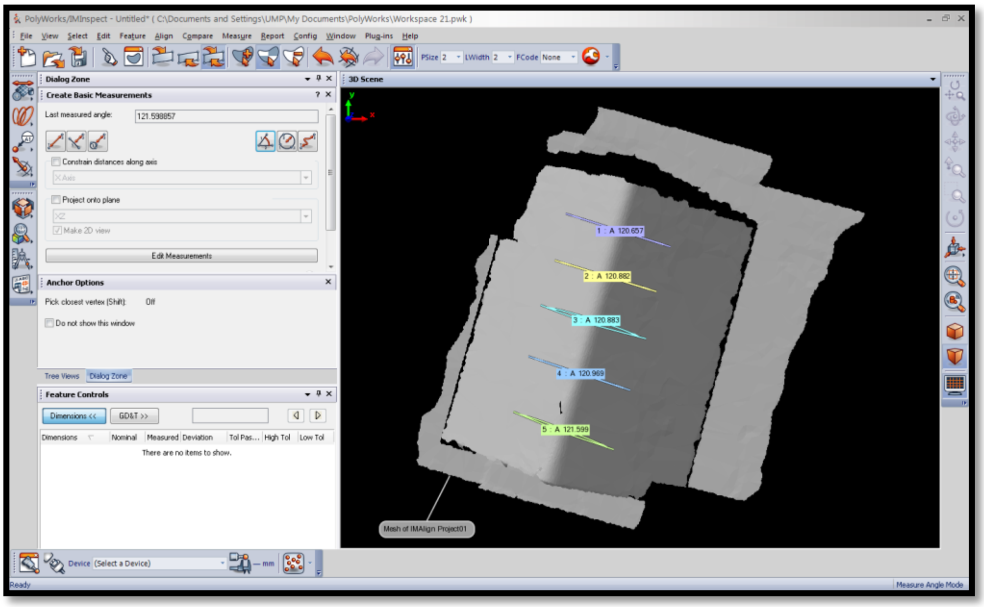


Figure 2.2 Bending sample

2.3 Theory of Bending

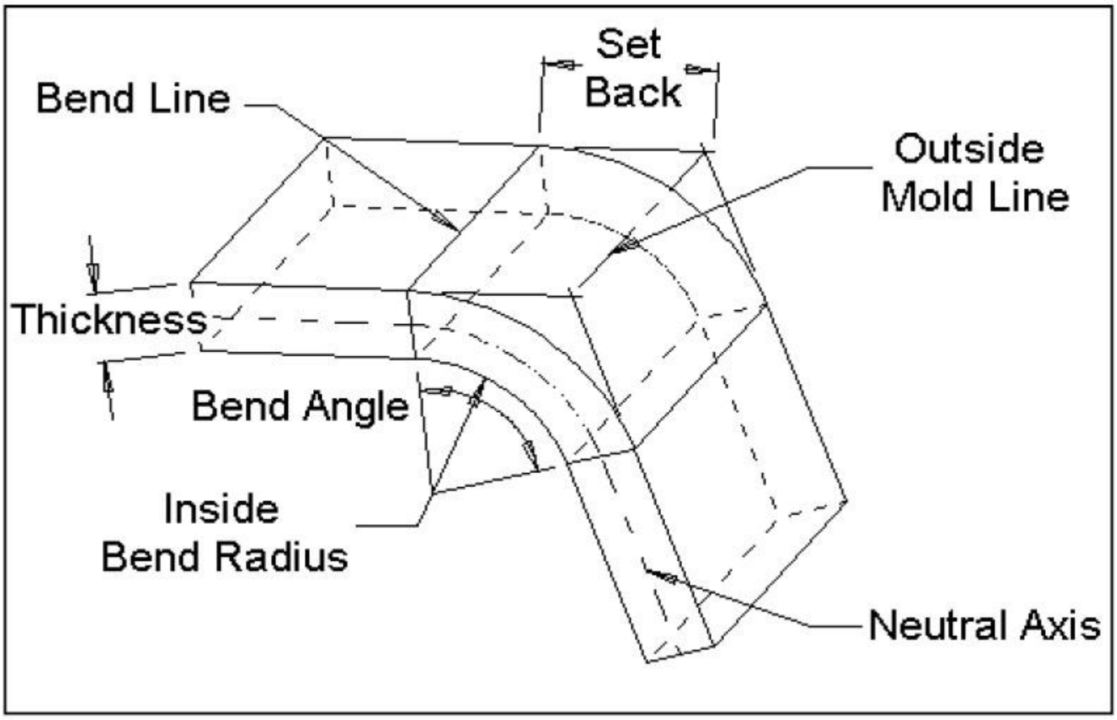


Figure 2.3 Bending process description

Source: SME, 2010.

Bend Allowance – The length of the arc through the bend area at the neutral axis.

Bend Angle – The included angle of the arc formed by the bending operation,

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.

Bend Lines – The straight lines on the inside and outside surfaces of the material where the flange boundary meets the bend area.

Inside Bend Radius – The radius of the arc on the inside surface of the bend area.

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 divided by the material thickness.

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.

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.

Set Back - For bends of less than 180 degrees, the set back is the distance from the bend lines to the mold line.

2.4 Type of Bending

There are two basic types of bending; V-bending and U-bending. Each is defined by the relationship of the end tool position to the thickness of the material. For V-bending consist coining, bottoming and air-bending.

2.4.1 V-Bending

A bending operation performed by compressing the sheet metal between a matching V-shaped punch and die.(SME, 2010). Happen when the punch touch the workpiece and the workpiece, is bottoming in the lower cavity. As the punch is released, the workpiece ends up with less bend than that on the punch with greater included angle. During V-bending, the punch slides down, coming to a contact with the unsupported sheet metal. By progressing further down, it forces the material to follow along, until finally bottoming on the V-shape of the die. As may be observed, at the beginning of this process, the sheet is unsupported but as the operational cycle nears its end, the bent-up part becomes totally support while retained within the space between the punch and die. (Gowri et al. 2008)

2.4.1.1 Coining

Coining is a bending process in which the punch and the sheet on the die. Here, in the process compressive stress is applied to the bending region to increase the amount of plastic deformation. This reduces the amount of springback. (Diegal, 2002)

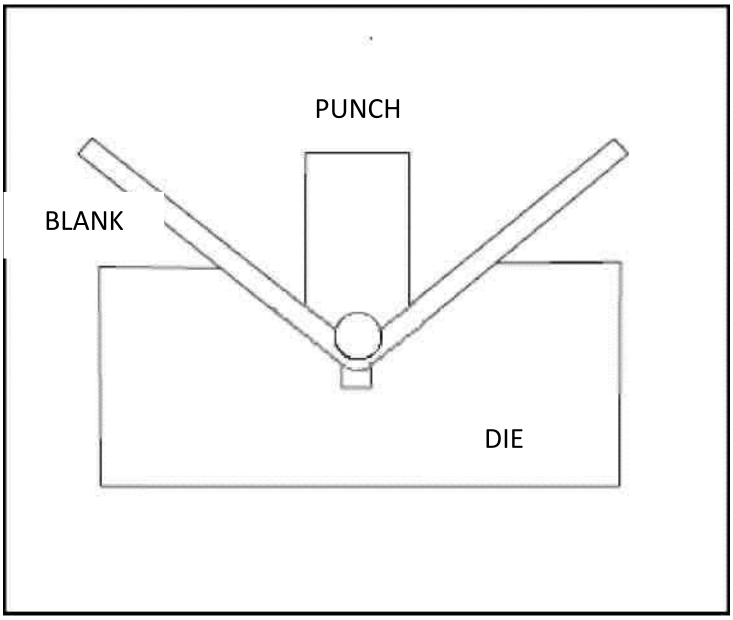


Figure 2.4 Coining Bending

Source: Diegal, 2002

2.4.1.2 Bottoming

Bottoming is a bending process where the punch and the work piece bottom on the die. This makes for a controlled angle with very little springback. The tonnage required on this type of press is more than air bending. The inner radius of the work piece should be a minimum of 1 material thickness. In bottom bending, springback is reduced by setting the final position of the punch such that the clearance between the punch and die surface is less than the blank thickness. As a result, the material yields slightly and reduces the springback. (Diegal, 2002)

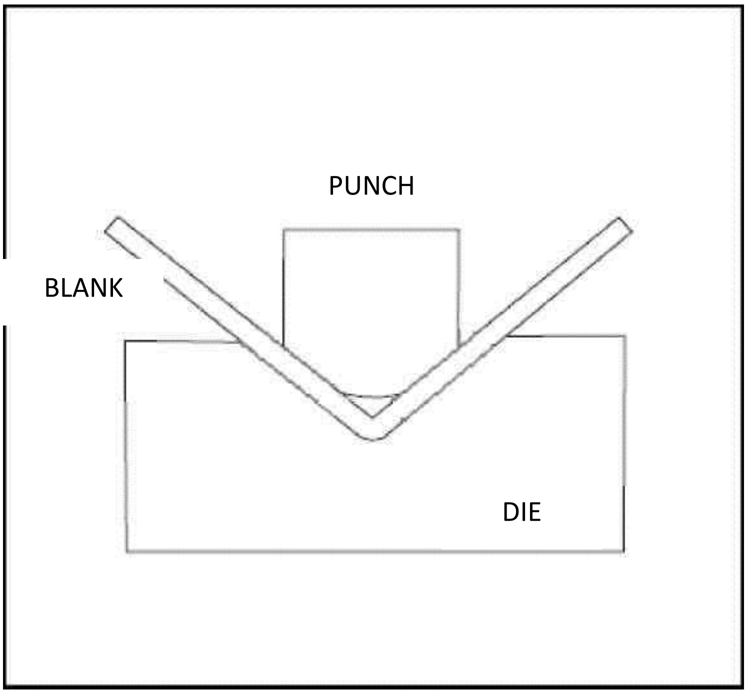


Figure 2.5 Bottoming Bending

Source: Diegal, 2002

2.4.1.3 Air Bending

Air bending is a bending process in which punch moving to 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 less bend than that on punch with greater included angle which is called springback.

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. The forces required to form the parts are relatively small, but accurate control of the punch stroke is necessary to obtain the desired bend angle. (Diegal, 2002)

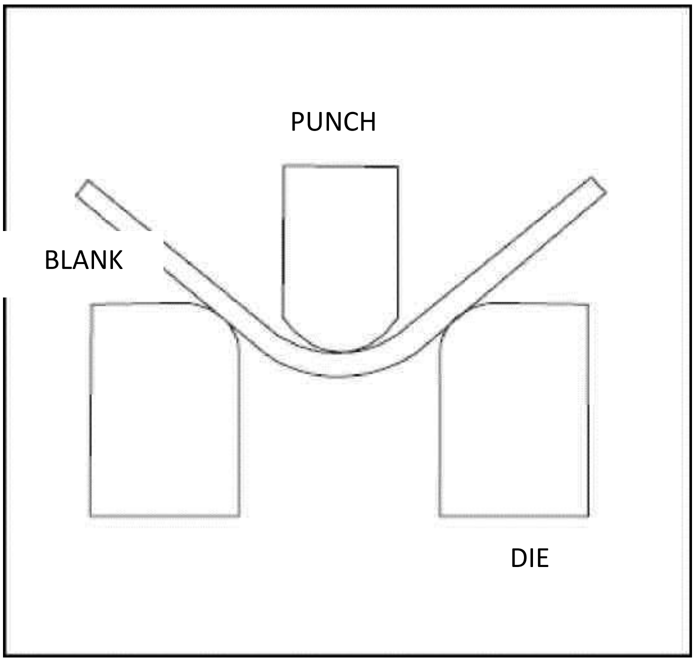


Figure 2.6 Air Bending

Source: Diegal, 2002

2.4.2 U-bending

U-bending process is a process of 2 parallel of axes to form U shaped of workpiece. Due to the symmetry of the tooling, half of the geometry is generated along with the applied symmetry boundary conditions. Furthermore, only the tooling profile is considered in the computations based on the rigid die assumption. The material properties of the strip are as follows; elastic modulus is 207 GPa, Poisson's constant is given as 0.3. The friction between the sheet and tooling is given as 0.144 (Nakamachi,1993).

2.5 Improvement of Springback

Finite element simulation of sheet metal forming is a well-established tool which is used in industrial practice to evaluate geometrical defects caused by elastic springback. Springback can be defined as an elastically-driven change of shape of the deformed part upon removal of external loads. This phenomenon results in a deviation of the real product geometry from that defined in the design phase and can cause significant problems during assembly. To keep the product development time and manufacturing costs low, finite element analysis aims to provide reliable information necessary for the modification of tool and product geometry. Therefore, the accuracy of information obtained in a numerical simulation of springback is essential for the product designers and die makers.

Various aspects of a finite element analysis must be carefully considered to prepare an adequate numerical model. An experimental and simulation study was carried out to reach a better understanding of the sensitivity of springback to various physical and numerical parameters. It was demonstrated that a large modelling error can be encountered if an inappropriate material model, contact description, element type and method of unloading is used in the numerical analysis. It was additionally emphasised that the accuracy of springback prediction can be significantly affected by purely numerical factors, such as mass scaling and iterative equation solvers. Artificial adaptations of the model or a low accuracy when solving the global system of equations may lead to an unrealistic change of the part's shape upon unloading.

In simulations of sheet metal forming, using very coarse meshes to describe the blank and tools can reduce the total computation time, but it can also reduce the accuracy of the stress state predicted at the end of the deformation. Recommendations for an appropriate discretisation of the blank and tools have been developed and validated in this thesis. It was demonstrated that the suggested mesh densities minimize the negative effects caused by a poor discretisation and lead to a more accurate springback prediction.

A simple stretch bending model was used to demonstrate the limitations of the tradition numerical rules used for the through-thickness integration in shell elements. If a sheet material deforms elasto-plastically, these rules require a large number of through-thickness integration points to ensure an accurate springback prediction. To overcome this problem, an adaptive scheme was developed to integrate through the thickness of the shell elements. This scheme uses a limited number of points which can be relocated during the simulation to provide a more accurate prediction of stress resultants at the end of forming. The adaptive scheme relies on several algorithms that: locate the interfaces which separate the elastic and plastic regions of the material, redistribute the integration points such that a point coincides with the elastic-plastic transition; update state variables of the relocated points and perform the actual integration.

The developed adaptive scheme was extended to make it suitable for simulations of realistic deep drawing problems. An advanced algorithm that locates elastic-plastic interfaces was developed. The algorithm is independent of the yield function used in the numerical analysis and can determine the through-thickness location of the elastic-plastic transitions that appear when a material is cyclically bent under tension. Based on this algorithm, a generally applicable adaptive integration scheme was formulated and implemented in the implicit finite element code DiekA. Simulations of several numerical examples were used to evaluate the performance of the advanced adaptive scheme in various deformation regimes. It was demonstrated that the adaptive scheme can guarantee the same level of accuracy of a traditional scheme with a significantly lower number of integration points, hereby decreasing the computation time. Besides, it was shown that the accuracy of adaptive integration does not depend on material properties nor geometrical parameters of a problem.(Burchitz, 2008)

2.6 Previous Works

In industrial of bending process, there are same project was doing by researches to examine the spring back of metal. They are doing with difference of factors on metal that effect of springback ratio. Tables below shows the summary of bending.

Table 2.1 Summary of journal

AUTHOR	MATERIAL	METHOD	SUMMARY
YU Zhong-qi.Al.2006	Aluminium	Finite Element Simulation	~ U-shaped profile bent part decreases with the increasing of the side pressure, hereas the springback of curvature increases
Özgür et. Al. 2007	Stainess Steel	Experimental (V-bending)	~ Holding the punch load decreases the spring back value ~ Increase in thickness of the material increases the spring back value and spring back value increases in case of an increase in bending angle
S.W. Lee a, D.Y. Yang b, et. al. 1998	Mild Steel	Finite Element Simulation	~ The optimal combination of the five factors has been found from comparison of the computed results with the average experimental results
Dongye Fei Nilsson et. Al. 2006	Cold Rolled TRIP Steel (CMnSi TRIP Steel)	Experimental & numerical studies (V-Bending)	~ The spring-back angles increase with the decreasing of blank thickness while the increasing of die gap
Annika Nilsson et. al .1997	~ Stainless Steel-(AISI304) ~ Aluminium-(AA6019-T4) ~ Hot Rolled Steel-(St06), (S12) ~ Low Carbon Deep Drawing Steel	Experimental & Finite-element simulation (V-bending)	~ The FEM results correspond well with the experimental result for the materials tested, the deviation being roughly half a degree ~ The error in spring-back is estimated to be half a degree due to variation in the material and the accuracy of the method

According from table, most of journalist done the finite element analysis for project. From YU Zhong results, simulation of the Rotary Stretch Bending (RSB) of the U-shaped profile LY 12M (Material) was carried out. The comparison of the simulation with the experiment proves that the present bending simulation model is reliable. Parametric analysis shows that the side pressure and the stretching force significantly influence the dimension precision of the RSB part of the U-shaped profile LY 12M. For the two main factors above, both of the springback of curvature and the section distortion decrease with the increasing of the stretching force, whereas the two cannot decrease simultaneously with the increasing of the side pressure.

CHAPTER 3

METHODOLOGY

3.1 Introduction

Discussion in methodology is to clear out the steps taken on project. Every step is shown starting with raw material Stainless Steel until the end. Core experiment is Finite Element Method and U-bending experiment.

3.2 Flow Chart

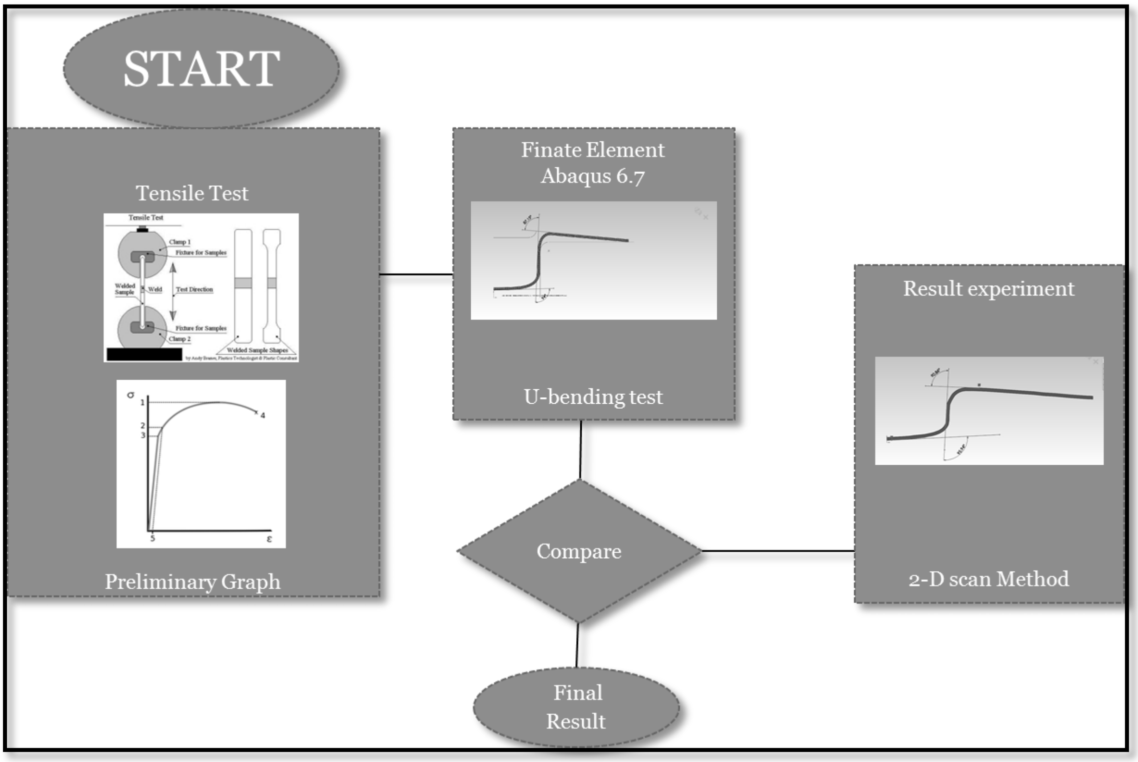


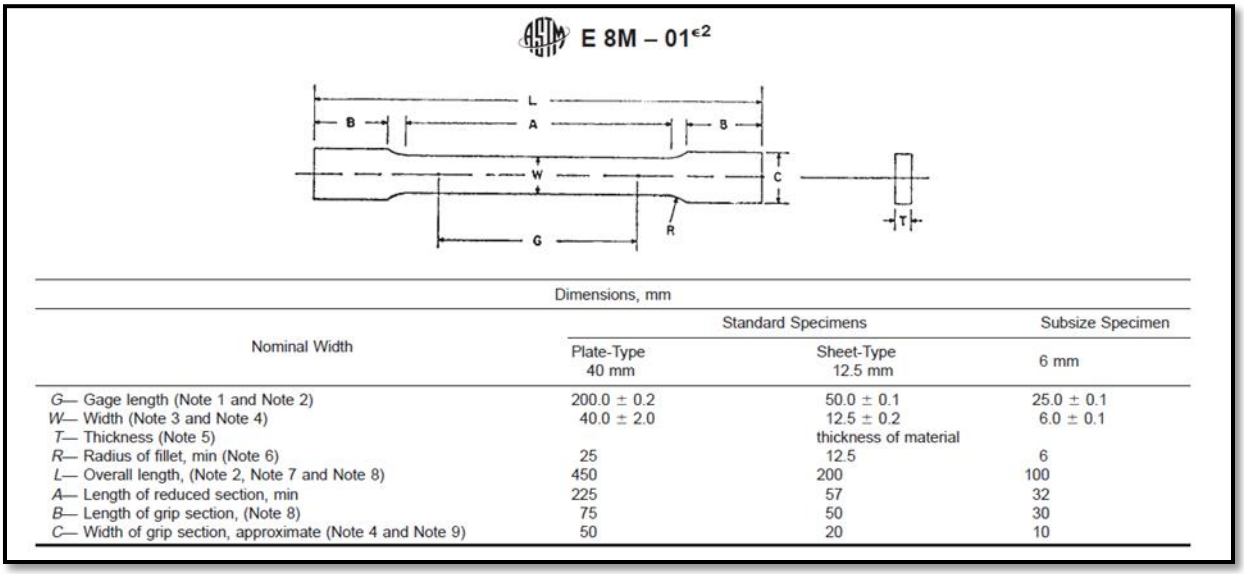
Figure 3.1 Methodology

3.3 Specimen Dimension

Material that brought into this project as an analyze specimen is stainless steel. The selected thickness of this stainless steel sheet is 1.5mm and 2mm. There are 10 specimens taken in this experiment that is reliable by referring other researcher that made this kind of project by taking at least 6 specimens. Besides, for getting the material properties of stainless steel, we referring standard ASTM E-8M specimen for tensile test.

Design of specimen are using in tensile test 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.

The result from the tensile test are been use in finite element analysis to evaluate the springback angle during the simulation of bending test.



Dimensions, mm

Nominal Width	Standard Specimens		Subsize Specimen
	Plate-Type 40 mm	Sheet-Type 12.5 mm	6 mm
G— Gage length (Note 1 and Note 2)	200.0 ± 0.2	50.0 ± 0.1	25.0 ± 0.1
W— Width (Note 3 and Note 4)	40.0 ± 2.0	12.5 ± 0.2	6.0 ± 0.1
T— Thickness (Note 5)		thickness of material	
R— Radius of fillet, min (Note 6)	25	12.5	6
L— Overall length, (Note 2, Note 7 and Note 8)	450	200	100
A— Length of reduced section, min	225	57	32
B— Length of grip section, (Note 8)	75	50	30
C— Width of grip section, approximate (Note 4 and Note 9)	50	20	10

Figure 3.2 E8M standard specimen

3.4 Test Specimen Preparation

In this test, two different thickness of stainless steel AISI 304 has been use to performed the tensile test. The thickness is 1.5mm. Each 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 3.1 below shown the total specimens in this project.

Table 3.1 Number of specimen

Specimen Thickness (mm)	Bil. of Specimen			
	0°	45°	90°	Total Specimen
1.5	3	3	3	9

3.4.1 Cutting Raw Materials

Stainless steel AISI 304 sheet metal need to be cut into specimen according to ASTM Standards, which the raw material of stainless steel AISI 304 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 0o, 45o and 90o, which varying with the direction of rolling.

Figure 3.3 LVD shearing machine

3.4.2 Cutting Tensile Specimen

The material then need to be cut into a tensile test specimen base on ASTM Standard as the reference from the 300mm x 50mm sheet size. Regarding to the ASTM 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.

3.4.2.1 Design Tensile Specimen using Mastercam's

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. The scheme 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.

Figure 3.4 Design Test Specimen using Mastercam

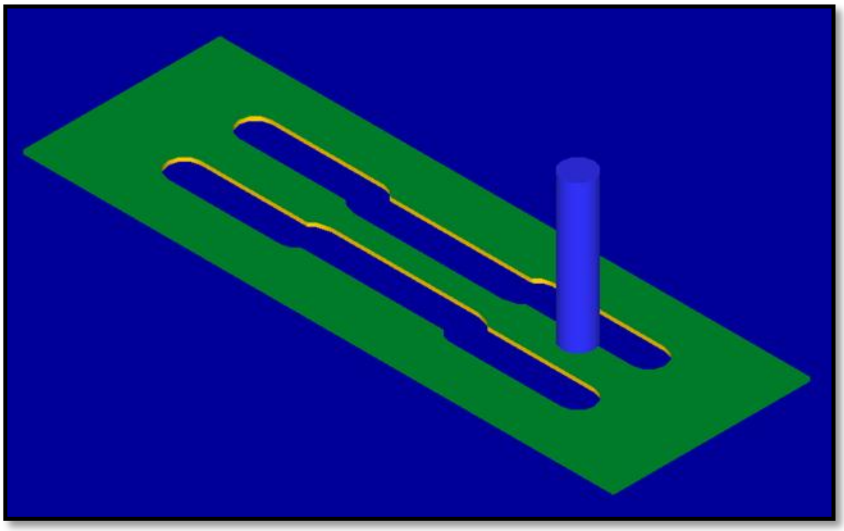


Figure 3.5 CNC Milling Process Simulation using Mastercam

3.4.2.2 Cutting Tensile Specimen using CNC Milling Machine

The cutting of tensile specimen will be completed by using Haas CNC Machine. 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 3.6 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.

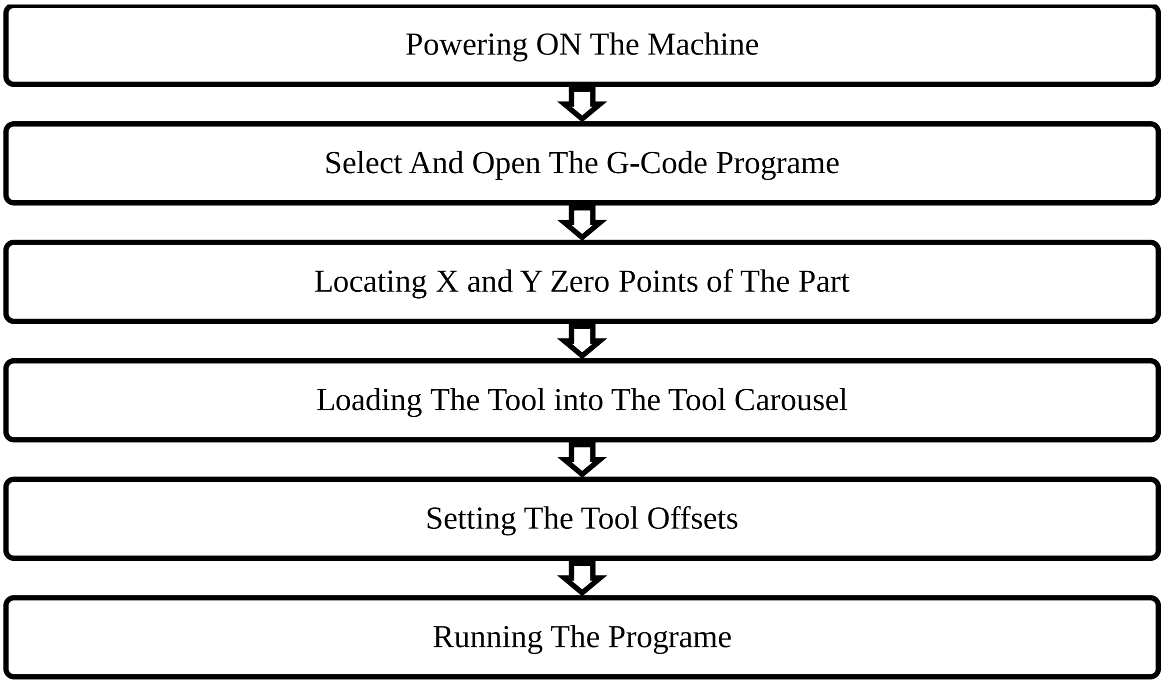
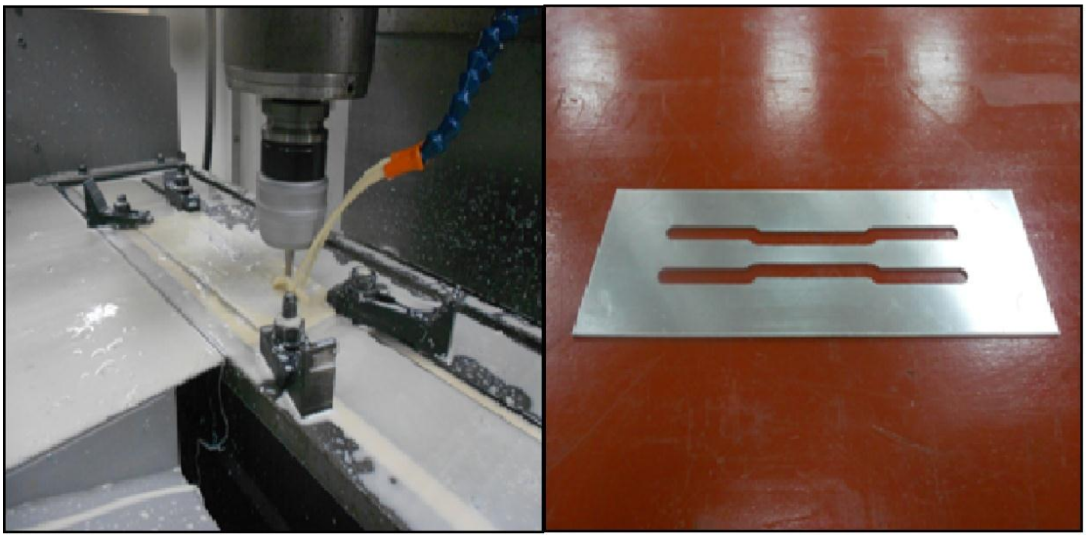


Figure 3.7 Basic Processes to Running The Milling Machine

The HAAS milling machine will be on when the green POWER ON button in the upper left of the control is press. After that, closed the front doors and press the POWER UP/RESTART button to set up the zero available axes. Then press the LIST PROG key and this will shown a list of any programs presently in the control panel. Import the G-Code programme from USB and press MEMORY and F2 key to select the programme. After the programme was open, clamp the workpiece on the machine and use an edge finder or indicator to located the part zero point with the jog handle. Press the OFFSET key and PAGE UP until the work coordinate page appears. Then use the cursor arrows to get to G54 X and push the PART ZERO SET key to store the X-axis value as offset. The curson will automatically move to the G54 Y location. Push again the PART ZERO SET key to set the G54 Y. Usually the Z-axis value will not have to be set and should be zero.



(a) **(b)**

Figure 3.8 (a) Milling Process (b) Semi-Finish Specimen

3.5 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.

Figure 3.9 Specimen griped on machine

A check can also be run to ensure that the test will run at the proper testing speed and temperature. The test is following the ASTM E8M standard which is the standard tensile test procedure. The test is the begun by initiating force application. The data of material properties will be automatically transfer and view in the tensile software in the computer. The scheme below has shown the tensile experiment 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.

Figure 3.10 Tensile Experiment Procedure

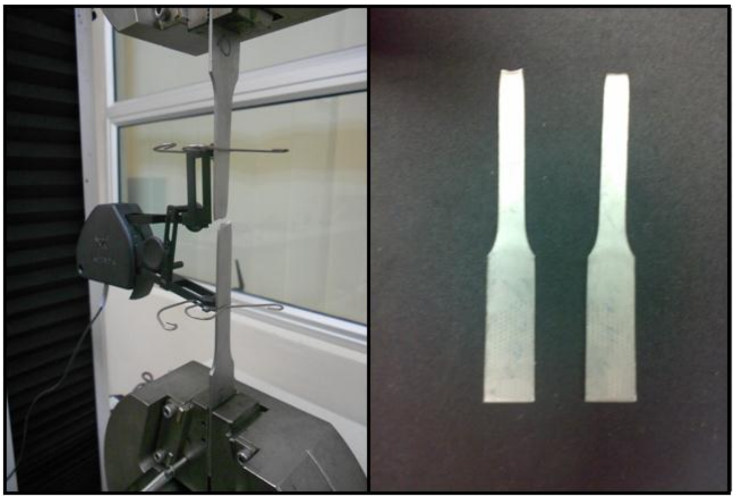


Figure 3.11 Finished specimen

3.6 Finite Element

Plasticity data from tensile test are used to conduct Abaqus 6.7 Software.

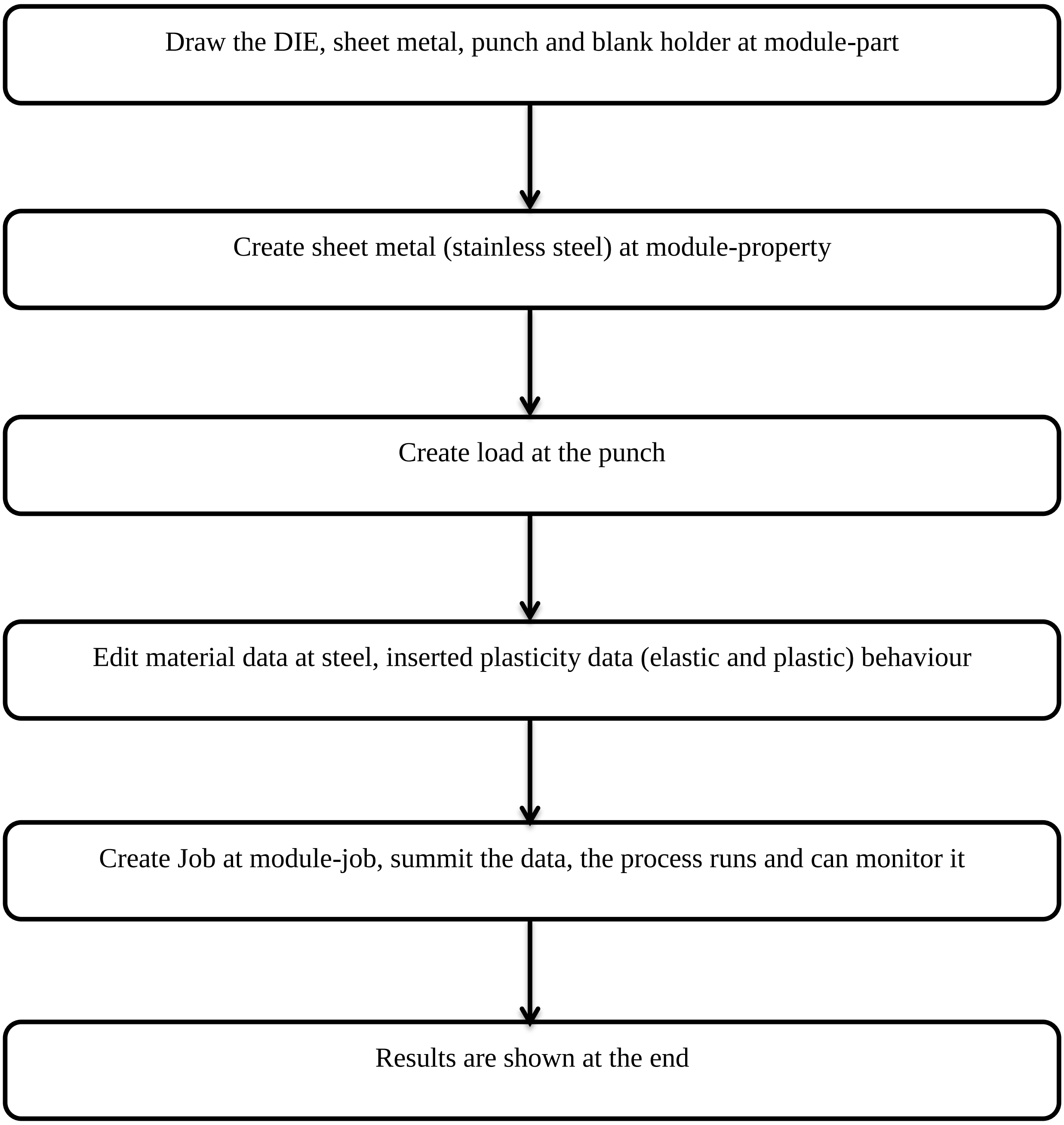


Figure 3.12 Steps in Abaqus

If error occur, the part would be failure due to incorrect data. Calculation and data must in standart SI units.

The result is saved to PNG format, the angles are determined using Solidwork software.

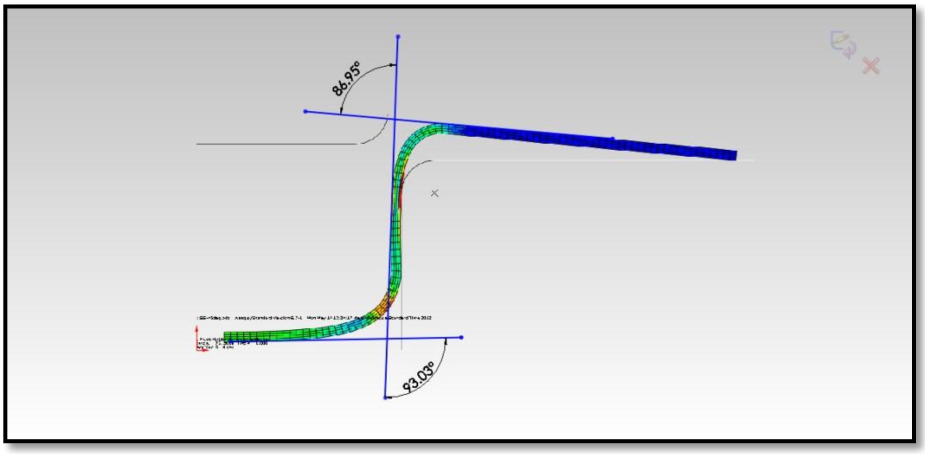


Figure 3.13 Angles in Finite Element

3.7 U-Bending Experiment

3.7.1 Introduction

The experiment of bending is operated to find springback or spring-go of sheet metal (Stainless Steel AISI 304). Todays, the industry prefered to used Finite Element to find reability of springback. Springback is a common phenomenon in sheet metal forming processes that is the elastically-driven change of shape of a part after forming.

3.7.2 Objective

Accurate prediction of springback and sidewall curls is essential for the design of tools mainly used in sheet stamping operations. These deformation sets generate more complex stress-strain states in the sheet resulting in the formation of sidewall curl after the sheet is allowed to unload. The results validate the finite element approach as a trustworthy tool for predicting the springback parameters for a given set of stamping conditions and material properties.

3.7.3 Modification Die

This project have difficulties where are the machine cannot operate with already Die because of higher force from gas-spring Die. In order to operate project, I used to modified the Die to suit with FKM press machine.

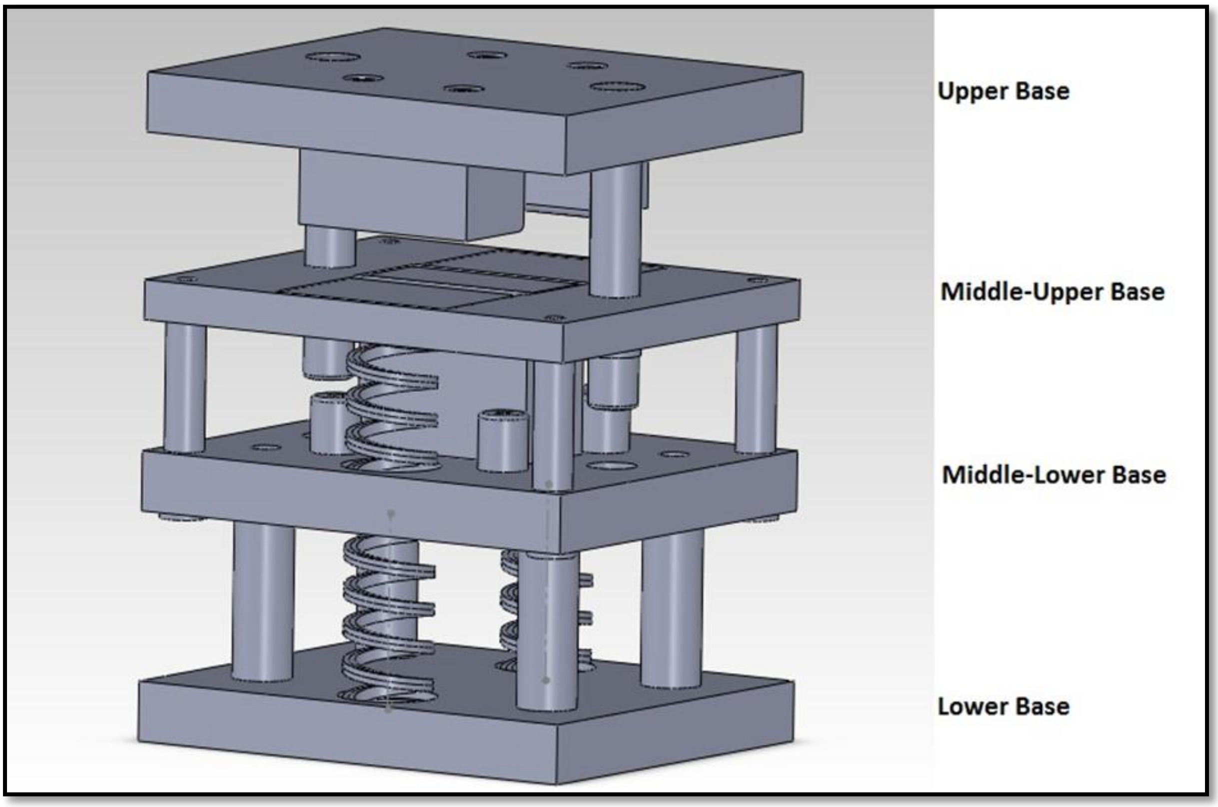


Figure 3.14 Die Drawing

The die is devided to 4 parts which is upper, middle-upper, middle-lower, and lower base. Each part is hooked each other. When upper is pressed downward, the sheet metal is placed at platform and being pressed by punch from bottom.

3.7.4 Process

Stainless Steel sheet metal 20mm X 200mm are prepared, placed at workstation at middle-upper base of Die.

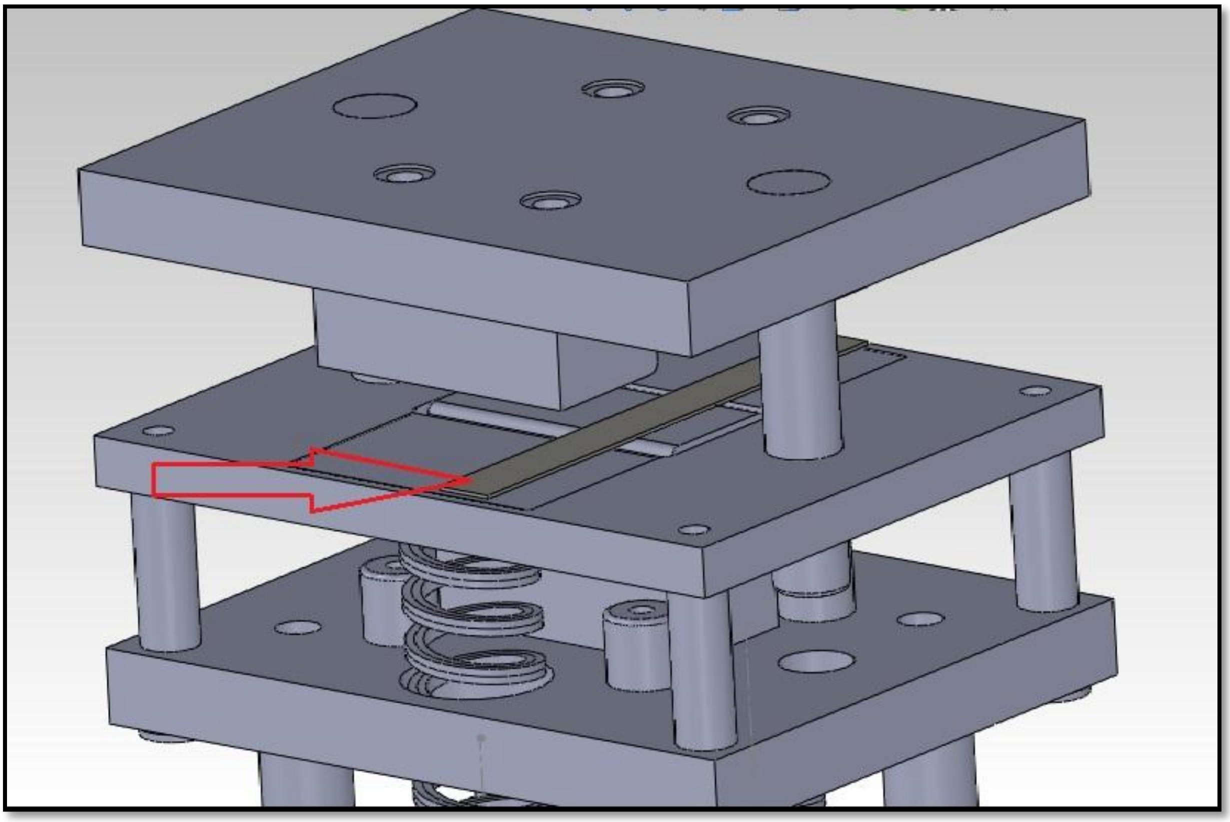


Figure 3.15 Sheet metal placed

Punch is pressed to bend the sheet metal. Simultinously, the springback occur after punch is released.

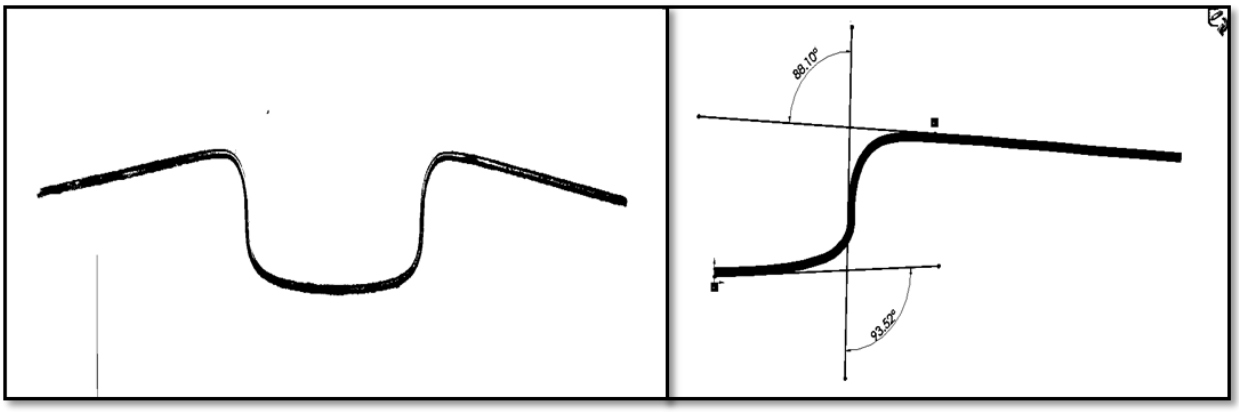


Figure 3.16 Bending angle

The measurement is defined using Solidwork 2011. The sheet metal is scanned and save as PNG format.

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. This force, in conjunction with the friction between the blank and blank holder and the blank and die, controls how the blank material is drawn into the die during the forming process. 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 0.144. Although, the experimental is used small dimension of sheet metal, the friction is not influeance the result and considered frictionless.

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 Introduction

The results validate the finite element with experimental approach as a trustworthy tool for predicting the springback parameters for a given set of stamping conditions and material properties. The results for experimental and finite element are been analyzed to compare springback.

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 Stainless Steel 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)

3.96	13.16	1154	0.47418

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

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

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

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

The true stress and true strain of stainless steel AISI 304 can be found by using the engineering stress and strain value with the equation below:

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

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

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 below is 10 second initial value of the data.

Table 4.2 Tensile Test Result for Stainless Steel 1.5mm Thickness

Time (s)	**Extension (mm)**	**Load (kN)**	**Eng. Stress (MPa)**	**True Stress (MPa)**	**Eng. Strain (mm/mm)**	**True Strain (mm/mm)**
0	0	1.43190	76.3680000	76.3680000	0	0
1	0.08362	2.71182	144.6304000	144.8722799	0.0016724	0.001671003
2	0.16700	3.80415	202.8880000	203.5656459	0.0033400	0.003334435
3	0.25025	4.65057	248.0304000	249.2717922	0.0050050	0.004992517

4	0.33369	5.18077	276.3077333	278.1517559	0.0066738	0.006651629
5	0.41700	5.39963	287.9802667	290.3820221	0.0083400	0.008305414
6	0.50025	5.52776	294.8138667	297.7634794	0.0100050	0.009955281
7	0.58369	5.62730	300.1226667	303.6262387	0.0116738	0.011606187
8	0.66700	5.71148	304.6122667	308.6757943	0.0133400	0.013251806
9	0.75031	5.78725	308.6533333	313.2850470	0.0150062	0.014894721
10	0.83362	5.85621	312.3312000	317.5385107	0.0166724	0.016534941

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 Stainless Steel material having a thickness of 1.5mm with different orientation angle (R).

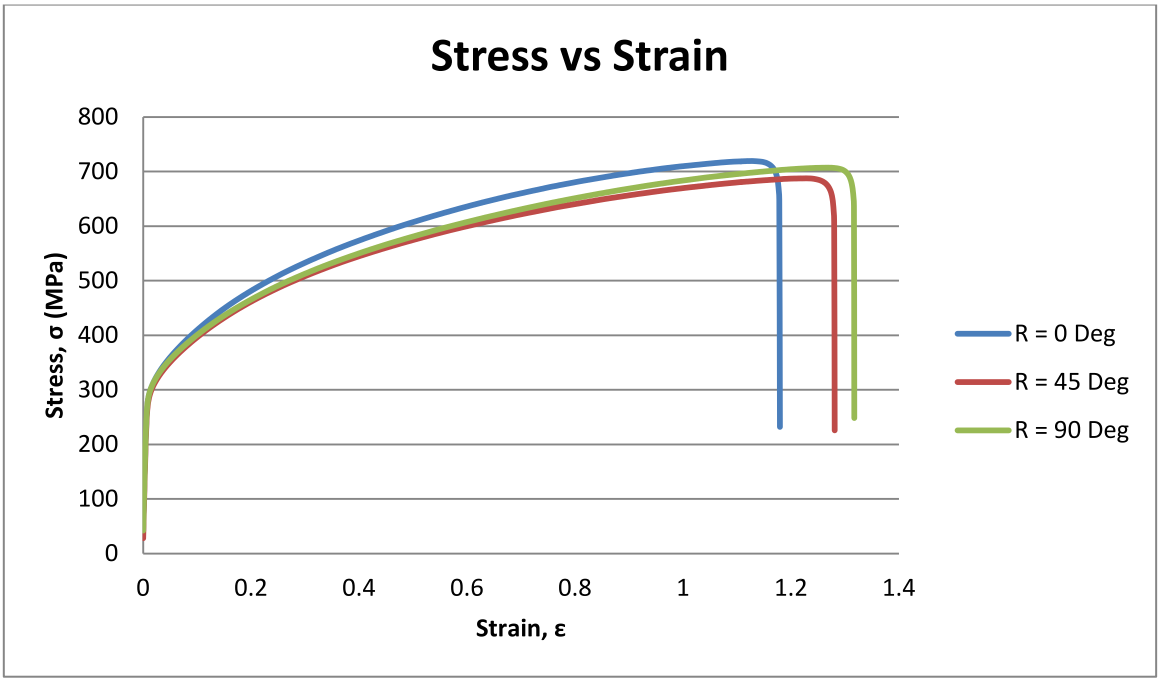


Figure 4.1 Different Angle of Stress-Strain Graph for Stainless Steel 1.5mm Thickness

Based on the Stress-Strain graph for stainless steel 1.5mm thickness, 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, the 0 degree orientation angle specimen has a higher UTS value followed by 90 degree and 45 degree orientation angle specimens. At small strain values (the elastic region), the relationship between stress and strain in nearly linear. Within this 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 stress about 250 MPa. From Table 4.2 select the initial data of engineering stress less than 250 MPa and plot the stress-strain again to produce the linear stress-strain graph. Graph 4.2 below shown the linear stress-strain graph.

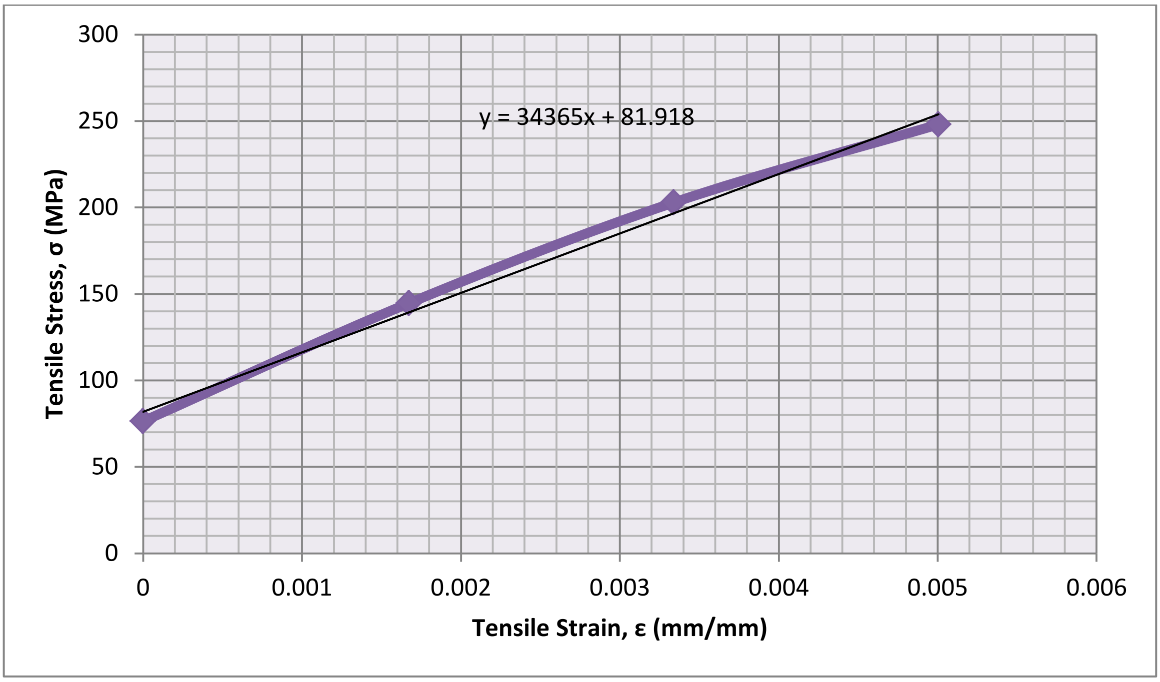


Figure 4.2 Linear Stress-Strain Graph for 1.5mm Thickness

The linear stress-strain graph above shown the slope of the curve is about 34365 MPa. The linear stress-strain graph has been plotted for every each of specimens and the average value will produce accurate young's modulus value. Another mechanical properties for stainless steel 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	8.566666667	108.8057667
Average 45	8.326666667	113.7223667
Average 90	8.493333333	113.3891000

Average Result For 1.5mm Thickness 8.462222222 111.9724111

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:

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

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.

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

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 stainless steel AISI 304.

Table 4.4 Mechanical Properties of Stainless Steel AISI 304

Orientation Angle	Young's Modulus (GPa)	Strain Hardening Exponent, n	Ultimate Tensile Strength, UTS (MPa)	Anisotropy Value, R	Poisson's Ratio, v
Average 0	200	0.494603333	712.9808000	0.954668958	0.267547457
Average 45	200	0.499413333	694.1545000	0.979193519	0.261969764
Average 90	200	0.503130000	692.3155667	0.897142772	0.252830008

Average Result For 1.5mm Thickness	**200**	**0.499048889**	**699.8169556**	**0.943668416**	**0.260782410**

According to table 4.4 Mechanical Properties, consideration for Young's Modulus is 200 GPa due to standard test method for stainless steel AISI 304.

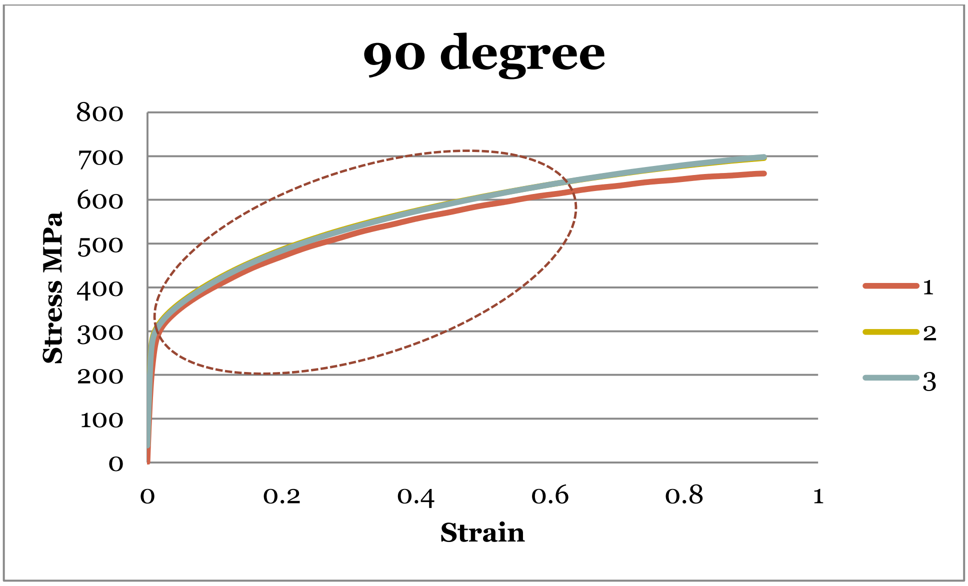


Figure 4.3 Selected point at region

The region in figure 4.3 shown where the 6 points are selected to inserted in Finite Element Analysis software(Abaqus 6.7).

Table 4.5 Plasticity data for FE

	0 deg			45 deg			90 deg		
Average point	Stress pa	strain	Strain at 0	Stress Mpa	strain	Strain at 0	Stress Mpa	strain	Strain at 0
1	429580800	0.1111115	0	420948800	0.1111157	0	422318933	0.1111145	0
2	552317333	0.300002	0.1888905	534773333	0.3083375	0.1972218	533778666	0.3000042	0.1888897
3	604280000	0.425002	0.3138905	576779733	0.4236165	0.3125008	581750933	0.4236145	0.3125
4	648120000	0.5638905	0.452779	616022933	0.5611155	0.4499998	624860266	0.562504	0.4513895
5	681392000	0.7013915	0.59028	647384533	0.7013937	0.590278	659708266	0.7013937	0.5902792
6	707555733	0.861111333	0.7499998	673981333	0.8666707	0.755555	690641066	0.8666717	0.7555572
Strain Hardening Exponent, n	0.494603333			0.499413333			0.50313		
Anisotropy Value, R	0.954668958			0.979193519			0.897142772		
Poisson's Ratio, v	0.267547457			0.261969764			0.252830008		
Young's Modulus, E (GPa)	200			200			200		

4.3 Experimental Results

4.3.1 Springback Measurement

Based on experimental study, springback results were evaluated and these results were used in order to validate the proposed finite element calculation. After unloading of the hat sections, the sidewall curls (ρ, the radius of a circle through a, b and c) and springback values (θ1, angle between the ox and the [ab] segment and θ2, angle between the [ab] and [ef] segments) of the parts were determined.

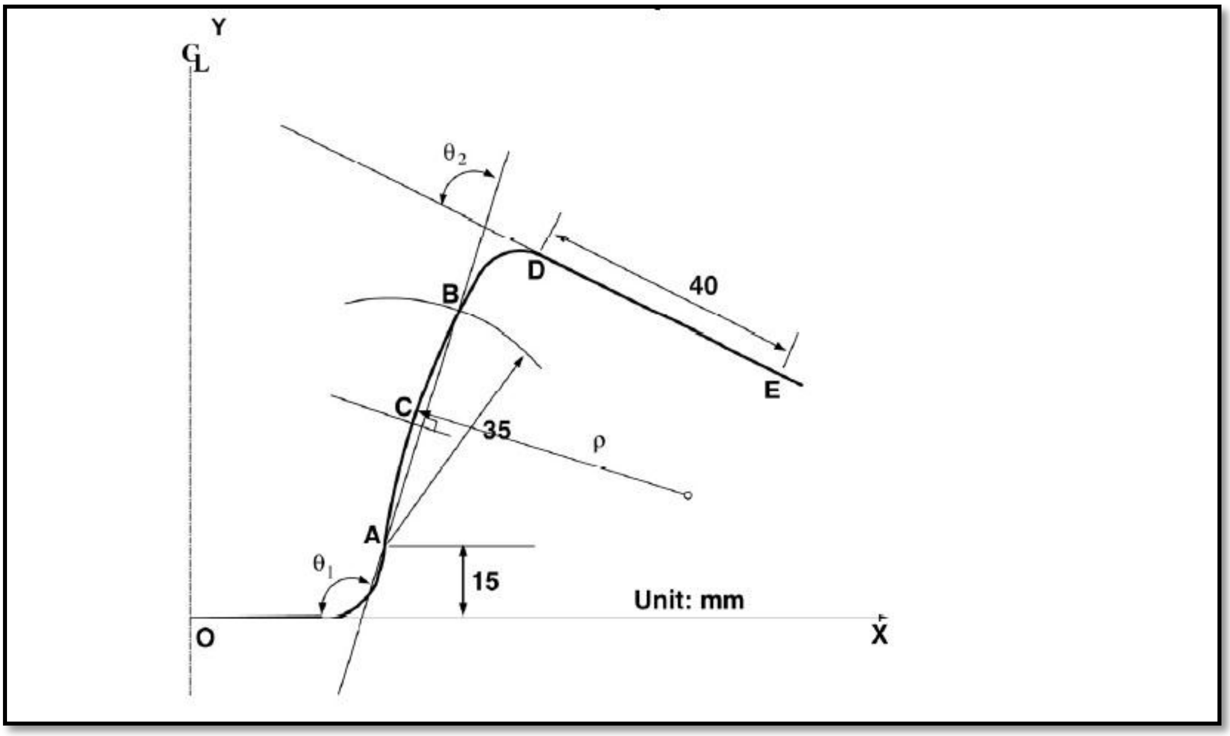


Figure 4.4 Measurements parameters for the springback

Source: Gang Liu , Zhongqin Lin, Youxia Bao

4.3.2 Angle Results

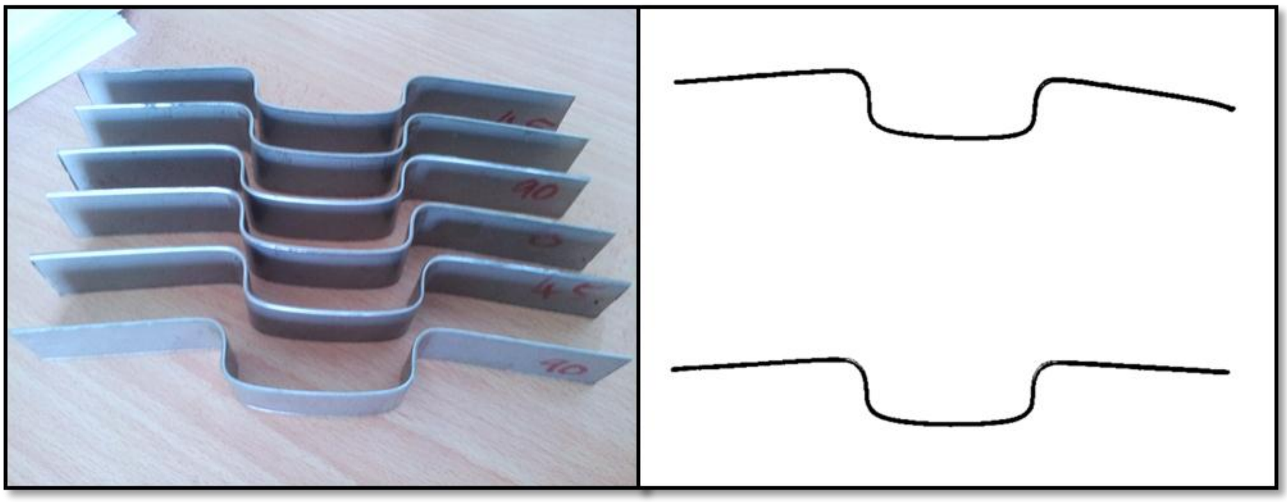


Figure 4.5 2D-draw bending springback

Figure 4.5 above shows specimens after bending, two pieces from each oreintation is bended. The angles of theta one and two is measured using software SolidWork 2001 X64 Edition. The accuracy using the software is 95%.

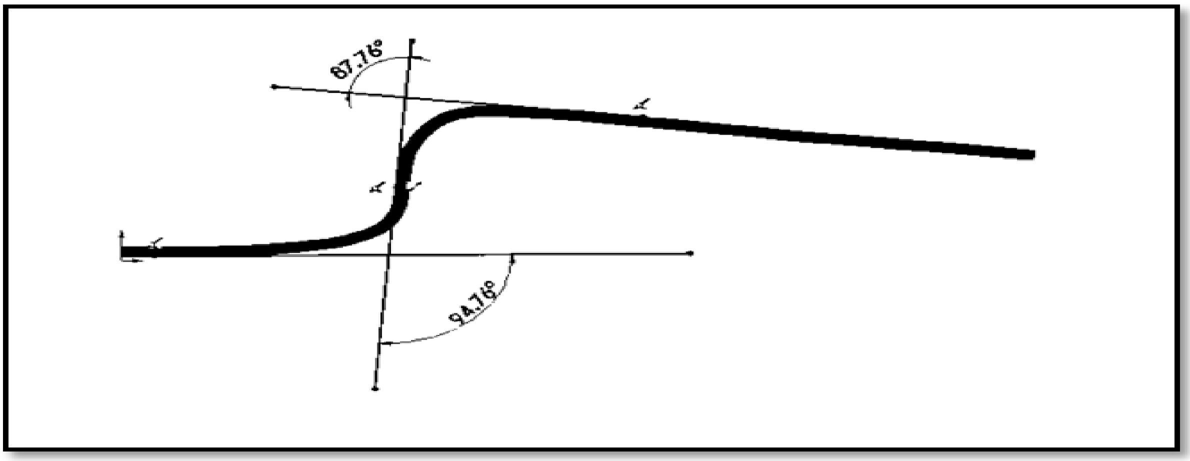


Figure 4.6 Angles of 0 degree springback

The specimen angles are determined at one side.

Table 4.6 Theta 1 and 2

	0 Degree		45 Degree		90 Degree	
	1	2	1	2	1	2
θ1	87.96	87.76	89.95	89.52	88.10	90.86
θ2	95.68	94.76	94.30	96.20	93.52	95.94

4.4 Finite Element Results

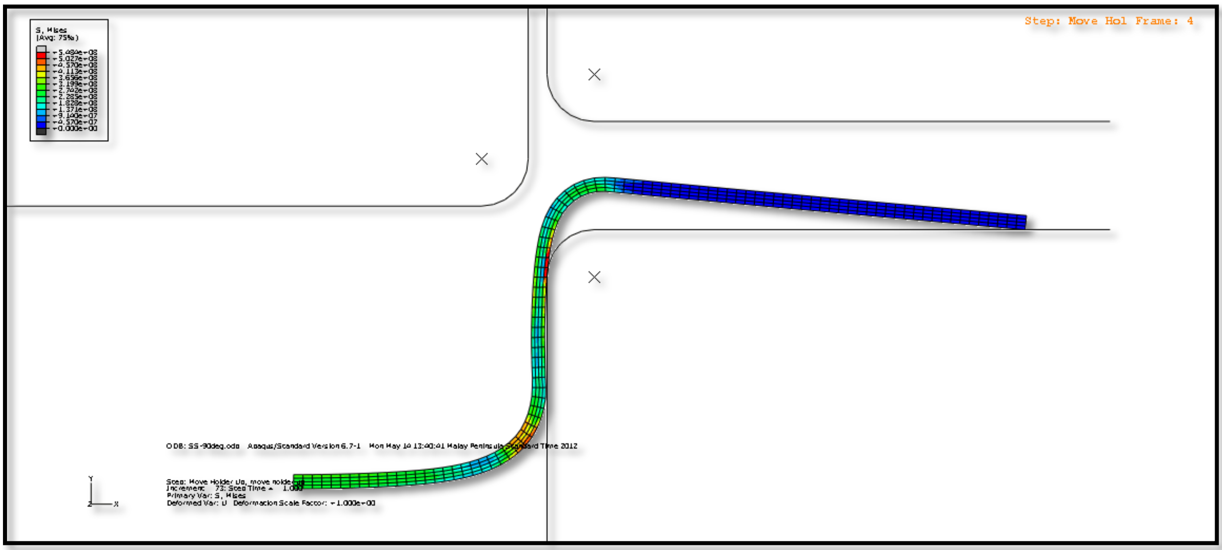


Figure 4.7 Abaqus Simulation

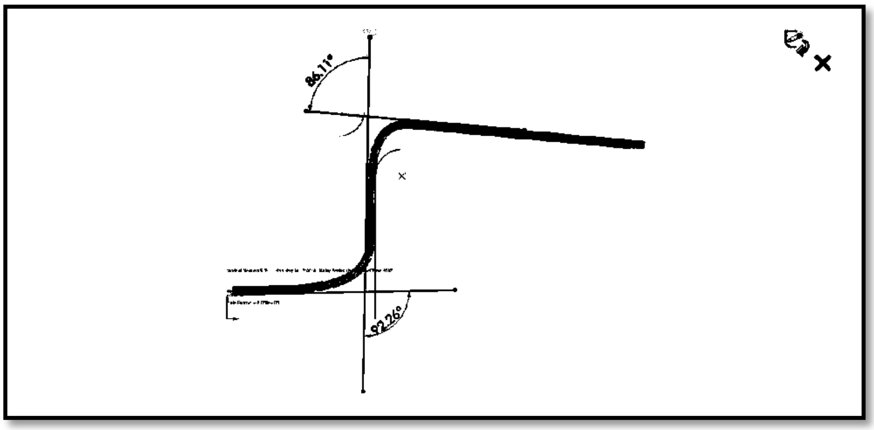


Figure 4.8 Simulation measurement

Table 4.7 Angle of Finite Element

	0 Degree	45 Degree	90 Degree
	1	2	3
$\theta 1$	**87.15**	**86.95**	**86.11**
$\theta 2$	**94**	**93.03**	**92.26**

Table 4.7 shows the average angles of each orientation. The results are compared with experimental.

4.5 Comparison Results

Table 4.8 Percentage error FE and Experimental

	0 DEGREE		45 DEGREE		90 DEGREE	
	THETA 1	THETA 2	THETA 1	THETA 2	THETA 1	THETA 2
EXPERIMENTAL	87.96	95.68	89.95	95.98	88.1	93.52
FINITE ELEMENT	87.15	94	86.95	93.03	86.11	92.26
PERCENTAGE ERROR (%)	**0.92**	**1.76**	**3.33**	**3.07**	**2.2**	**1.35**

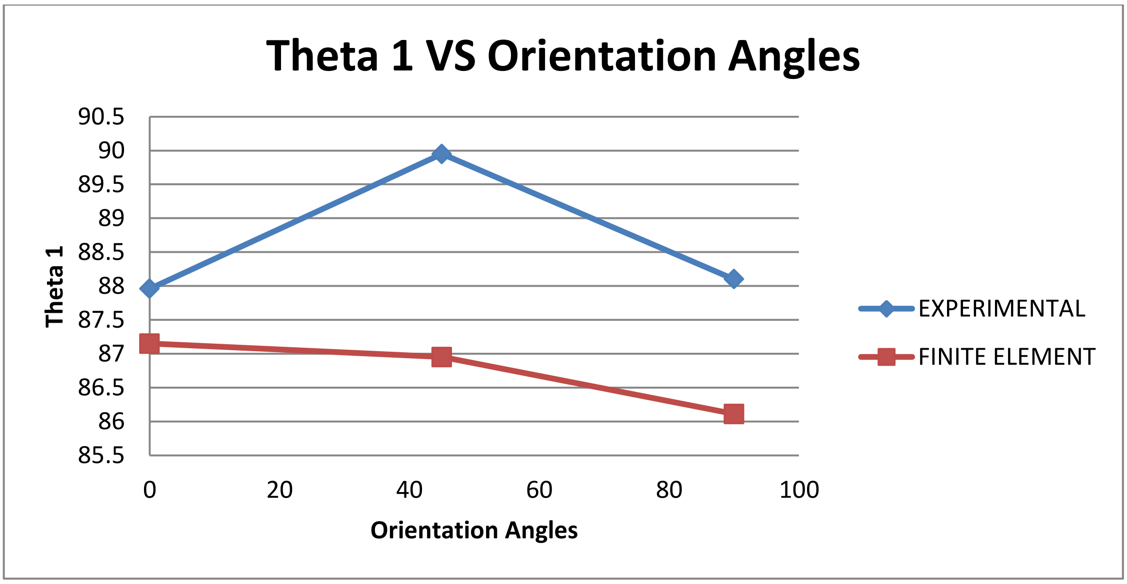


Figure 4.9 Theta 1 comparison angles

From table 4.8, the theta 1&2 for 0 degree, 45 degree, 90 degree are differences due to effect of anisotropy.

From figure 4.9, the pattern FE and Experimental are same. So, FE or Experimental are accurately method for designer to use.

Results are not perfectly match due to negligible friction of material.

CHAPTER 5

CONCLUSION

5.1 Conclusion

The conclusion made to conclude the objectives of project, this thesis concludes that:

i. The effects of parameter the anisotropy from rolling direction for stainless steel with thickness 1.5mm on the amount of springback are presented. The results represented the springback increases with higher of orientation. The results observed by two method which are finite element and experimental method. Both of these methods represented same result. The result also were good agreement with the literature review.

ii. FEM can use to simulate the springback for stainless steel in error less than 5%. The percentage error between 1-3% are small amount, and the FEM can simulate angles of the material in industries with maximum springback theta 1 is 3.89 degree. It was found the 0 degree orientation provides a suitable condition in U-bending process. So, Experimental and FEM is reliability for accuracy of springback.

5.2 Summary of study

This experimental study is basically an application of Finite Element to forming bending. Starting from tensile test result, plasticity result inserted to FEM. Bending are formed but differences angles due to variety factor occurs. Factors effects on springback are anisotropy, hardness exponent, thickness and geometric, temperature and bending method.

5.3 Recommendation

For future experimental works on this topic, in order to improve the study, the following recommendations being suggested.

i. The tools of CNC machines should be suitable with materials for reducing costs.
ii. FKM Press machine should calibrate before operating.
iii. Due to get accurate angles, using CMM machine best method to determine angles.
iv. Sheet metal must not in twisted condition.

REFERENCES

American Society of Mechanical Engineers (ASME, 1995).

Engineering Properties of Steels, Philip D. Harvey, editor, American Society for Metals, Metals Park, OH, (1982).

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

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

Behrouzi, A., Shakeri, M., and Dariani, B.M. 2008. Inverse Analysis of Springback in Sheet Metal Forming by Finite Element Method. Proceedings of the International Conference on Engineering Optimization Burchitz, I.A. 2008. Improvement of springback prediction in sheet metal forming.

Ph.D. Thesis. University of Twente, Netherlands. Diegal, O. 2002. The fine-art of Sheet Metal

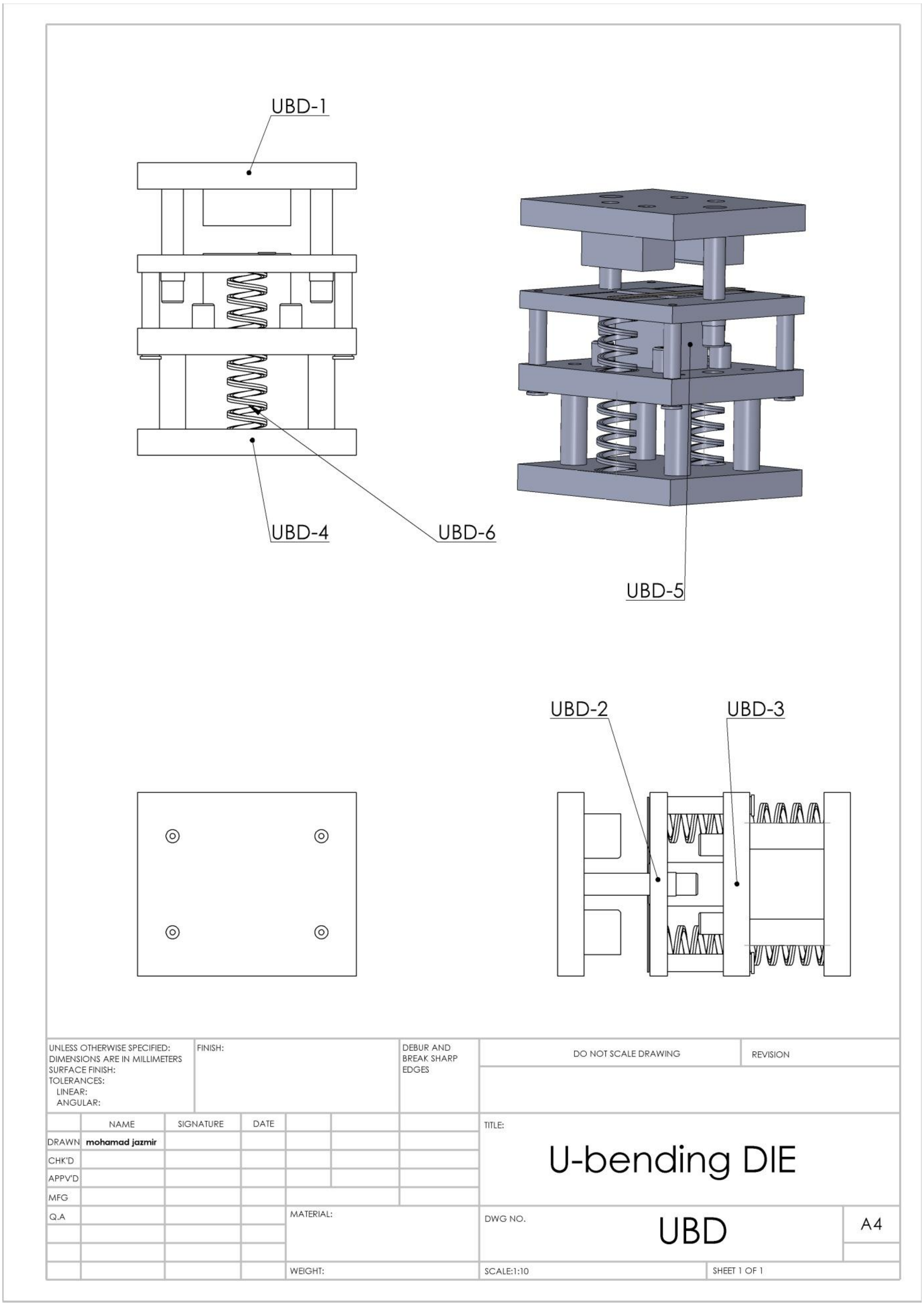
Bending. Slide. Bend Work Gowri, S., Hariharan, P., and Babu, A.S. 2008. Manufacturing Technology-I. India:Pearson Education. Huang, Y.M. and Leu, D.K 1996. Effect of Process Variables on V-Bending.

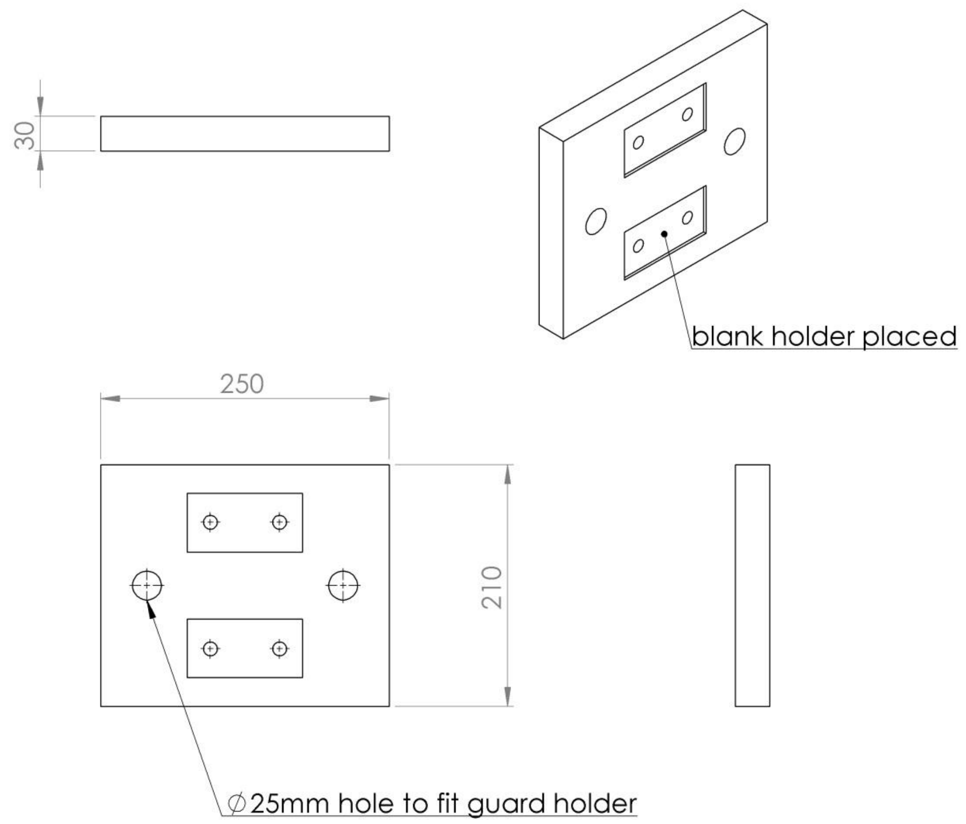
Journal of Process of Sheet Metal. 97(20): 632-650 John C.B.J 2006. Structural Finite Element. Slide. UCSB LLC. 2009. Electronic source: Metal Suppliers Online. http://www.metalsuppliersonline.com/ research/property/metals/809.asp (30 November 2010) Nargundkar, N. 2005. Bend allowance and springback in air bending. The Fabricator.

13 September. Scribd. 2010. Electronic source: How to calculate bend allowance. http://www.scribd.com/doc/29350996/Sheet-Metal-Bend-Allowance (30 November 2010) Society of Automotive Engineers (SAE, 2010) Society of Manufacturing Engineer (SME, 2010) Tekaslan, O., Seker, U., Ozdemer, A. 2004.

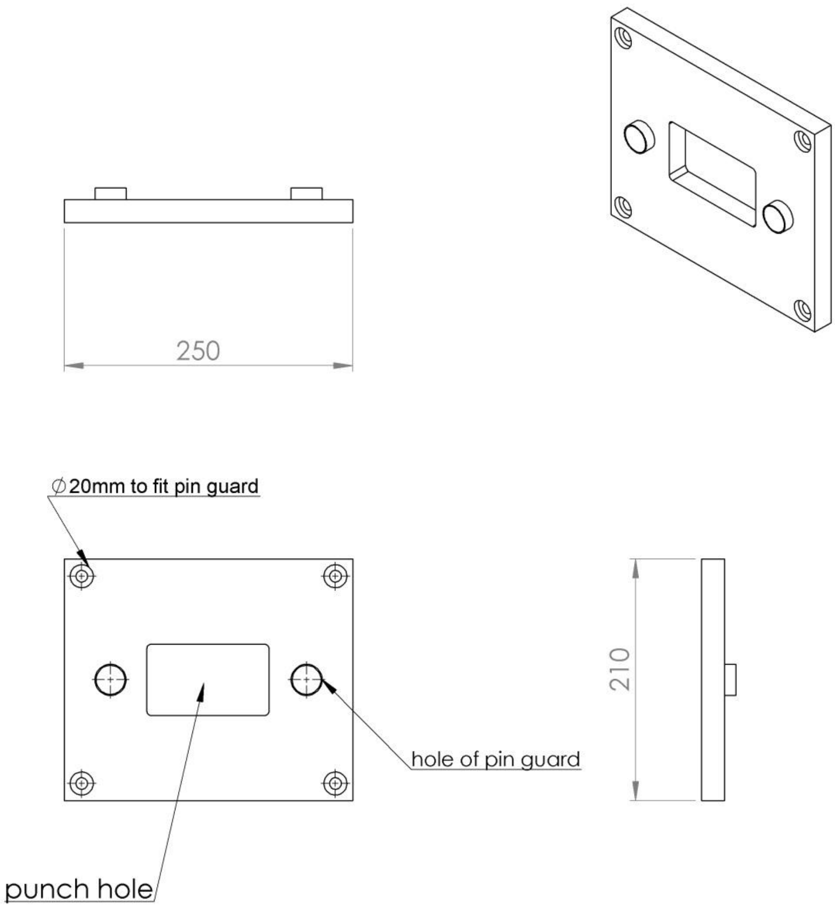
APPENDIX A

(Die Drawing)

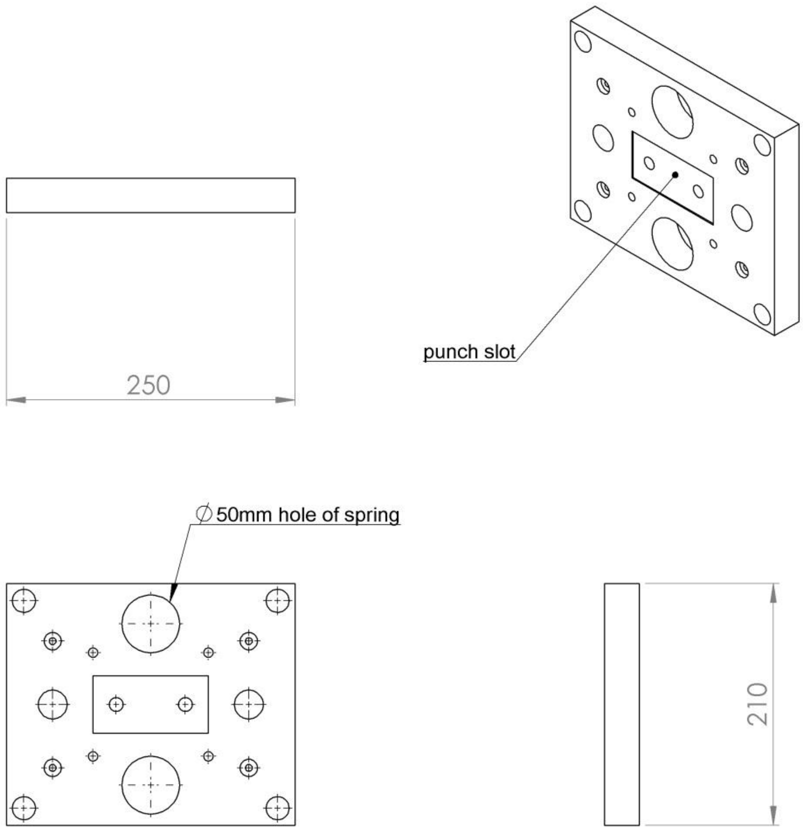




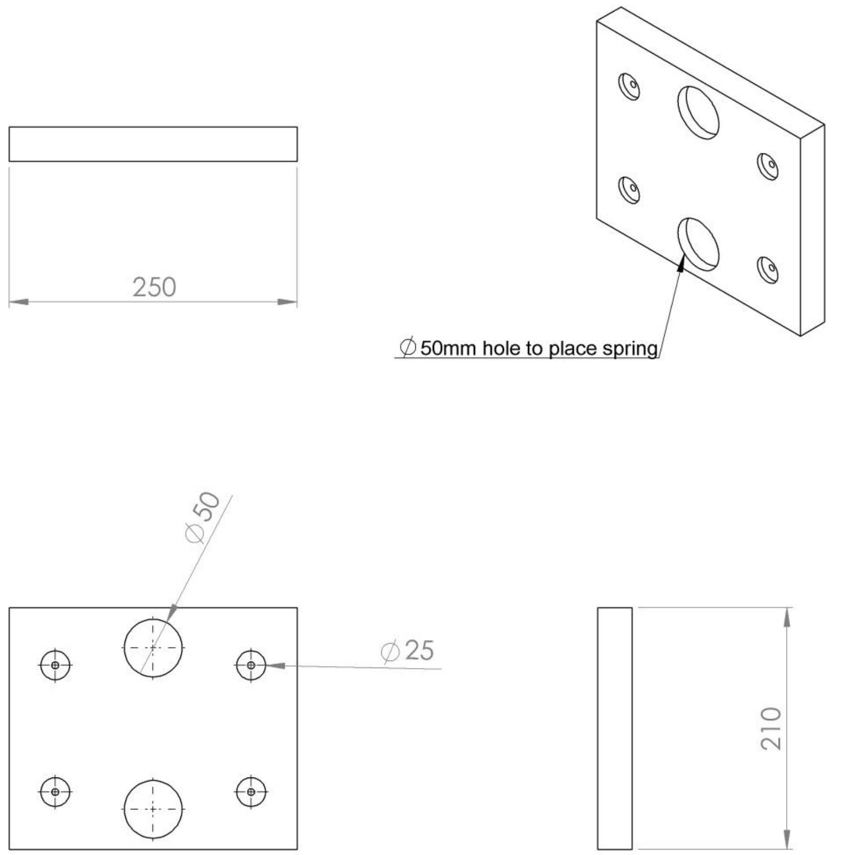
UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN MILLIMETERS SURFACE FINISH: TOLERANCES: LINEAR: ANGULAR:	FINISH:	DEBUR AND BREAK SHARP EDGES	DO NOT SCALE DRAWING	REVISION	
	NAME	SIGNATURE	DATE	TITLE: U-bending DIE	
DRAWN	jazmir				
CHK'D					
APPV'D					
MFG					
Q.A			MATERIAL:	DWG NO. UBD-1	A4
			WEIGHT:	SCALE:1:5	SHEET 1 OF 1



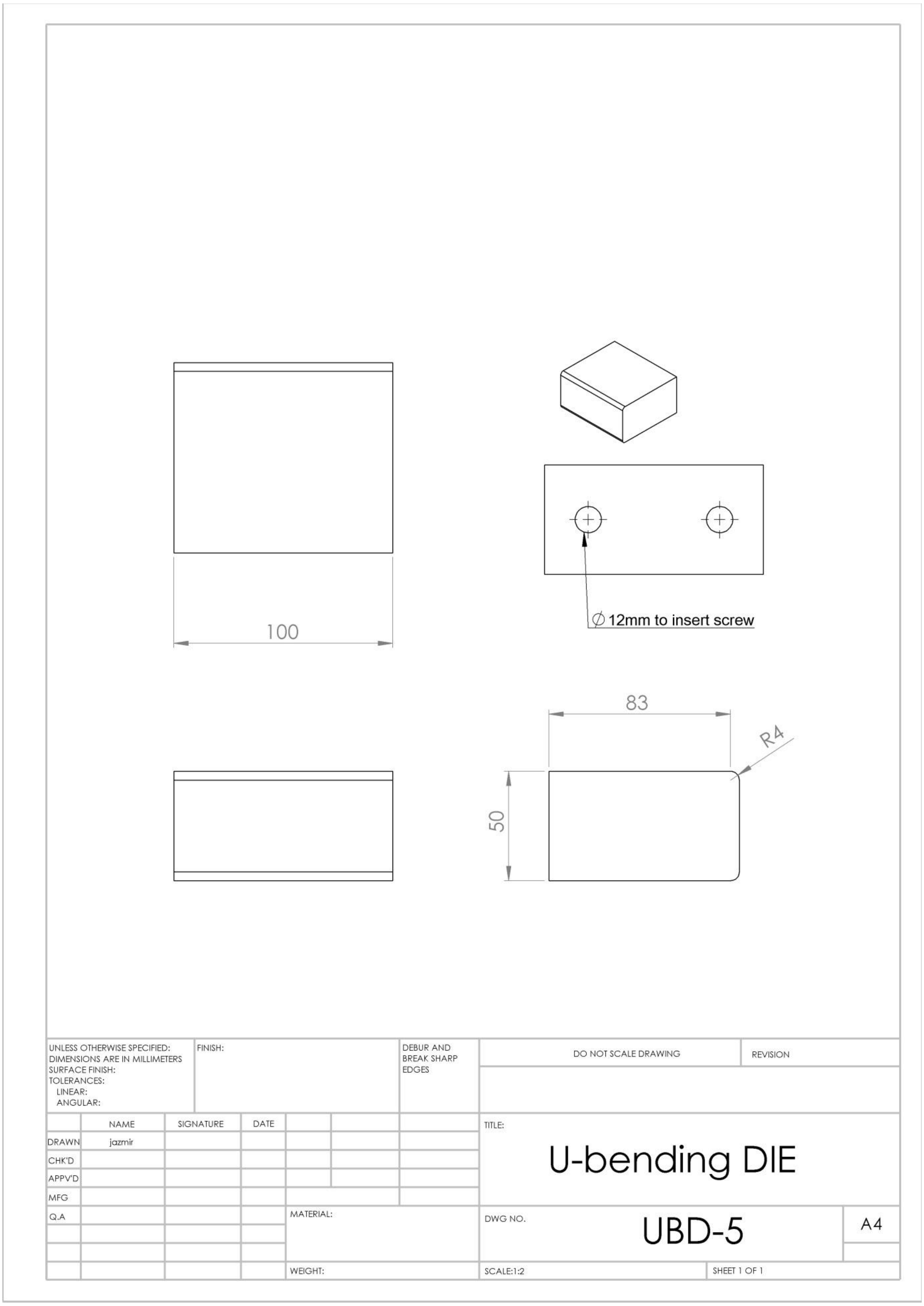
UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN MILLIMETERS SURFACE FINISH: TOLERANCES: LINEAR: ANGULAR:			FINISH:			DEBUR AND BREAK SHARP EDGES	DO NOT SCALE DRAWING	REVISION
	NAME	SIGNATURE	DATE				TITLE: U-bending DIE	
DRAWN	jazmir							
CHK'D								
APPV'D								
MFG								
Q.A				MATERIAL:			DWG NO. UBD-2	A4
				WEIGHT:			SCALE:1:5	SHEET 1 OF 1

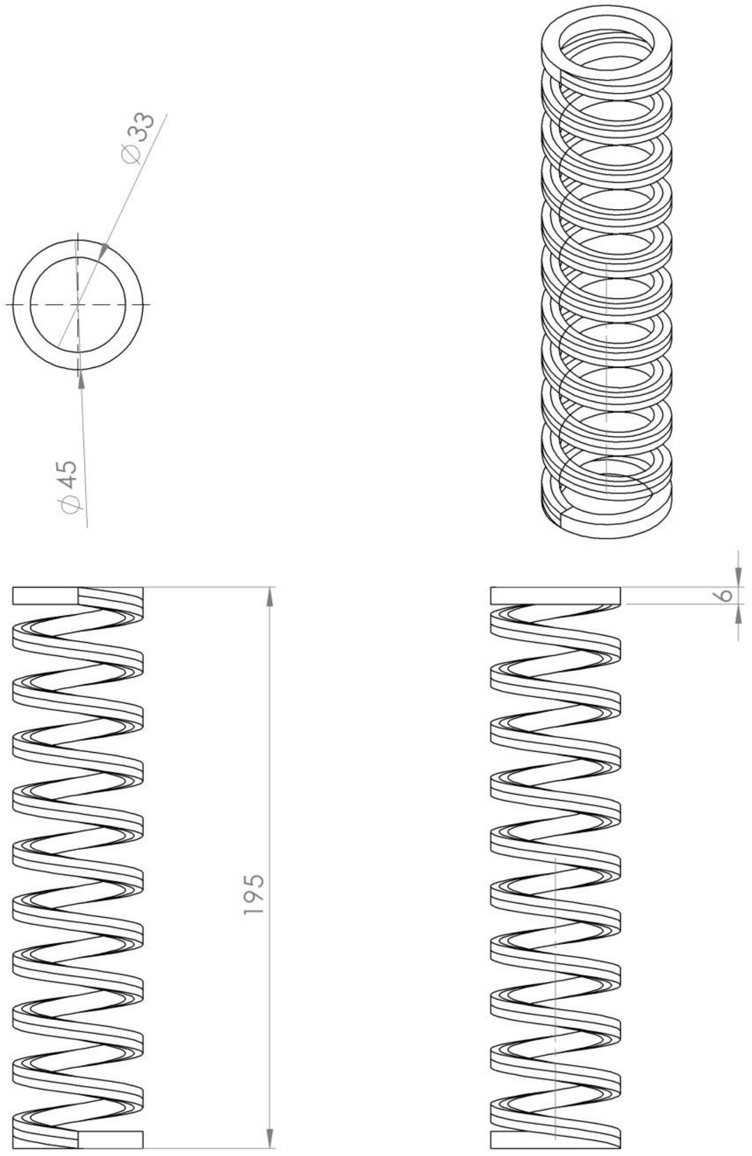


UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN MILLIMETERS SURFACE FINISH: TOLERANCES: LINEAR: ANGULAR:			FINISH:		DEBUR AND BREAK SHARP EDGES	DO NOT SCALE DRAWING	REVISION
	NAME	SIGNATURE	DATE			TITLE: U-bending DIE	
DRAWN	jazmir						
CHK'D							
APPV'D							
MFG							
Q.A				MATERIAL:		DWG NO. UBD-3	A4
				WEIGHT:		SCALE:1:5	SHEET 1 OF 1



UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN MILLIMETERS SURFACE FINISH: TOLERANCES: LINEAR: ANGULAR:	FINISH:		DEBUR AND BREAK SHARP EDGES	DO NOT SCALE DRAWING	REVISION
	NAME	SIGNATURE	DATE	TITLE:	
DRAWN	jazmir			U-bending DIE	
CHK'D					
APPV'D					
MFG					
Q.A			MATERIAL:	DWG NO. UBD-4	A4
			WEIGHT:	SCALE:1:5	SHEET 1 OF 1

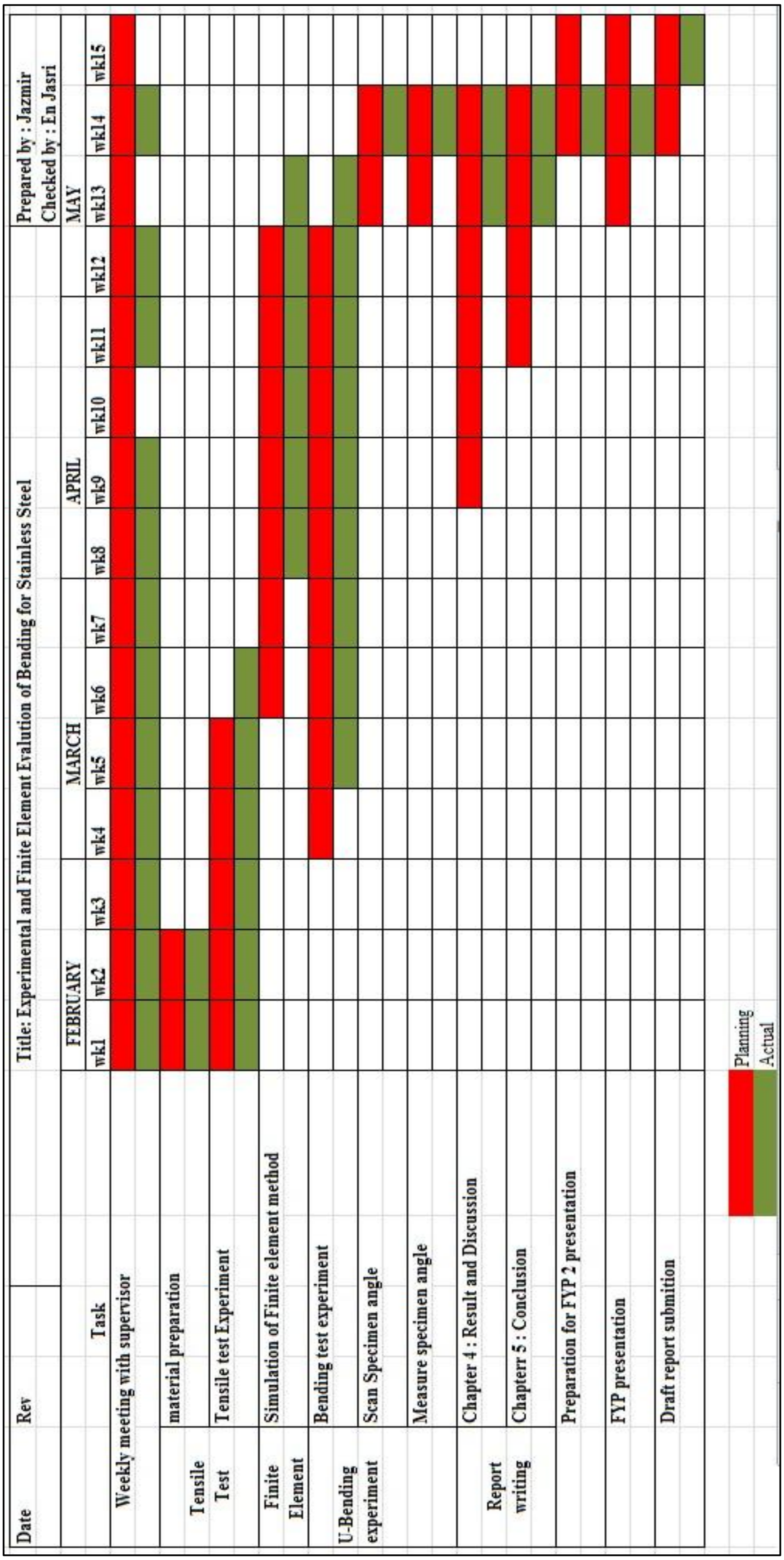

100
∅ 12mm to insert screw
83
R4
50
UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MILLIMETERS
SURFACE FINISH:
TOLERANCES:
LINEAR:
ANGULAR:
FINISH:
DEBUR AND BREAK SHARP EDGES
DO NOT SCALE DRAWING
REVISION
NAME
SIGNATURE
DATE
DRAWN
jazmir
CHK'D
APPV'D
MFG
Q.A
MATERIAL:
WEIGHT:
TITLE:
U-bending DIE
DWG NO.
UBD-5
A4
SCALE:1:2
SHEET 1 OF 1

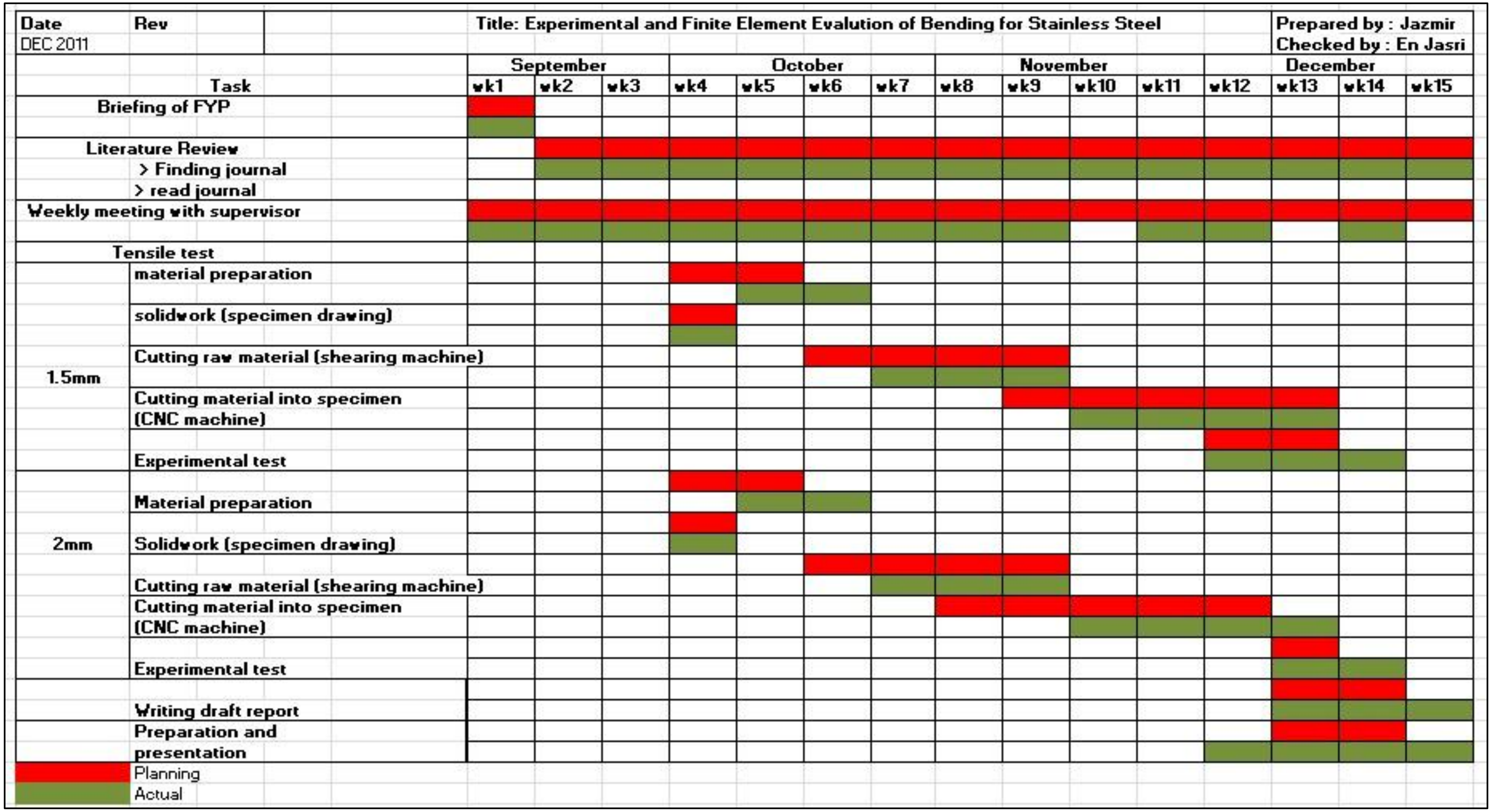


UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN MILLIMETERS SURFACE FINISH: TOLERANCES: LINEAR: ANGULAR:		FINISH:		DEBUR AND BREAK SHARP EDGES	DO NOT SCALE DRAWING	REVISION	
	NAME	SIGNATURE	DATE		TITLE: U-bending DIE		
DRAWN	jazmir						
CHK'D							
APPV'D							
MFG							
Q.A				MATERIAL:	DWG NO. UBD-6		A4
				WEIGHT:	SCALE:1:2	SHEET 1 OF 1	

APPENDIX B

(Guntt Chart)

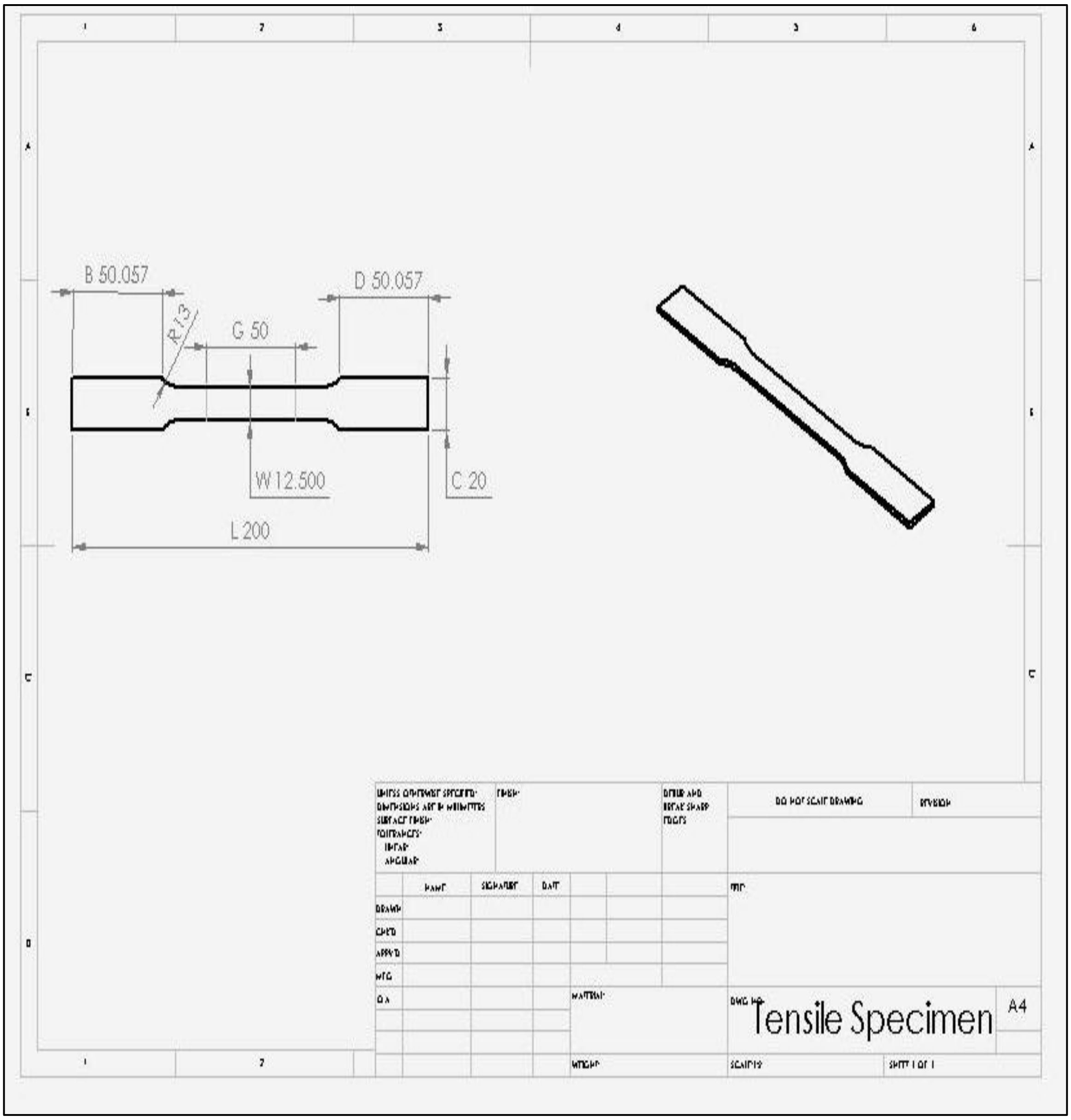




Date
DEC 2011
Rev
Title: Experimental and Finite Element Evalution of Bending for Stainless Steel
Prepared by : Jazmir
Checked by : En Jasri
Task
September
October
November
December
wk1
wk2
wk3
wk4
wk5
wk6
wk7
wk8
wk9
wk10
wk11
wk12
wk13
wk14
wk15
Briefing of FYP
Literature Review
> Finding journal
> read journal
Weekly meeting with supervisor
Tensile test
1.5mm
material preparation
solidwork (specimen drawing)
Cutting raw material (shearing machine)
Cutting material into specimen (CNC machine)
Experimental test
2mm
Material preparation
Solidwork (specimen drawing)
Cutting raw material (shearing machine)
Cutting material into specimen (CNC machine)
Experimental test
Writing draft report
Preparation and presentation
Planning
Actual

APPENDIX C

(Specimen Dimension)



APPENDIX D

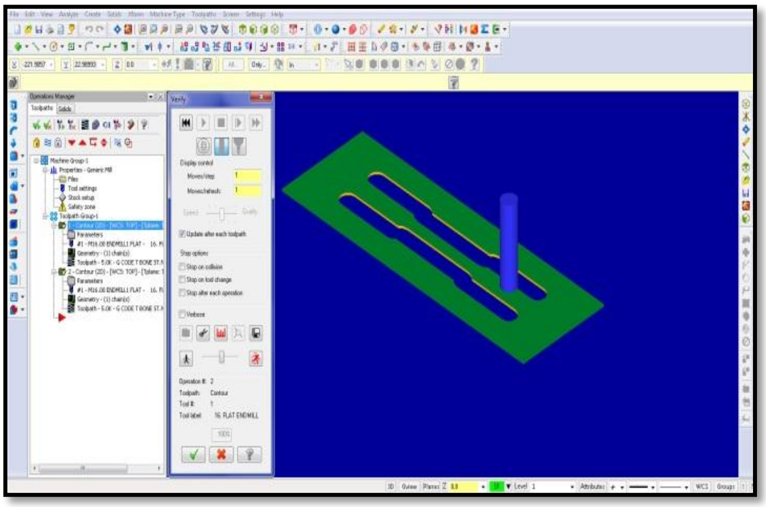
(G-Code)

```
%
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
```

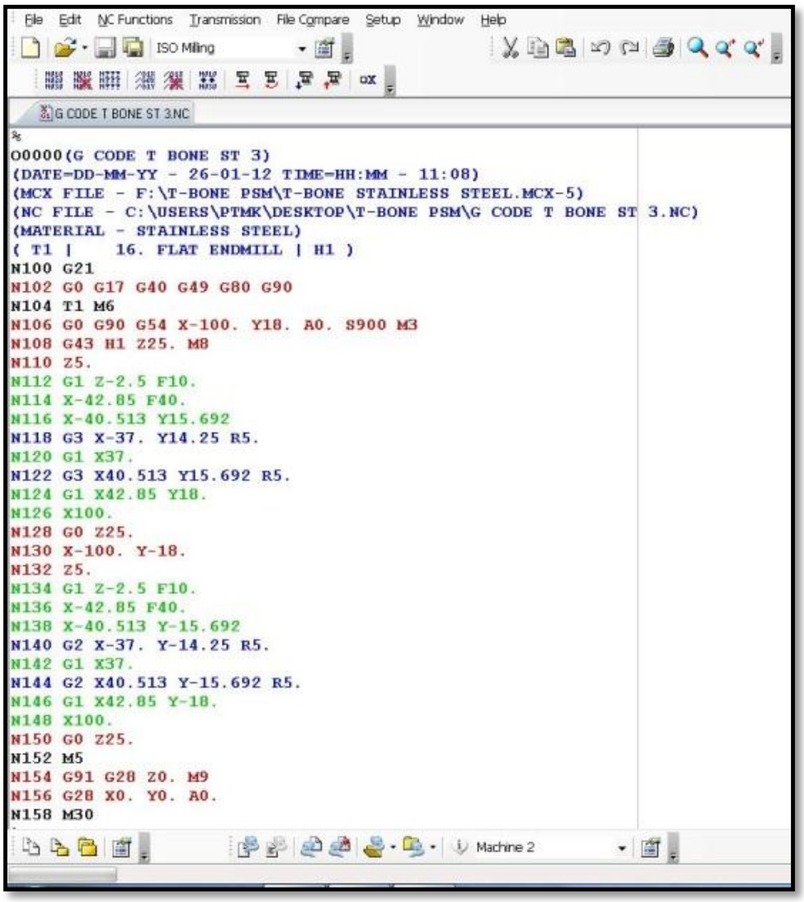
APPENDIX E

(PROJECT PROGRESS)

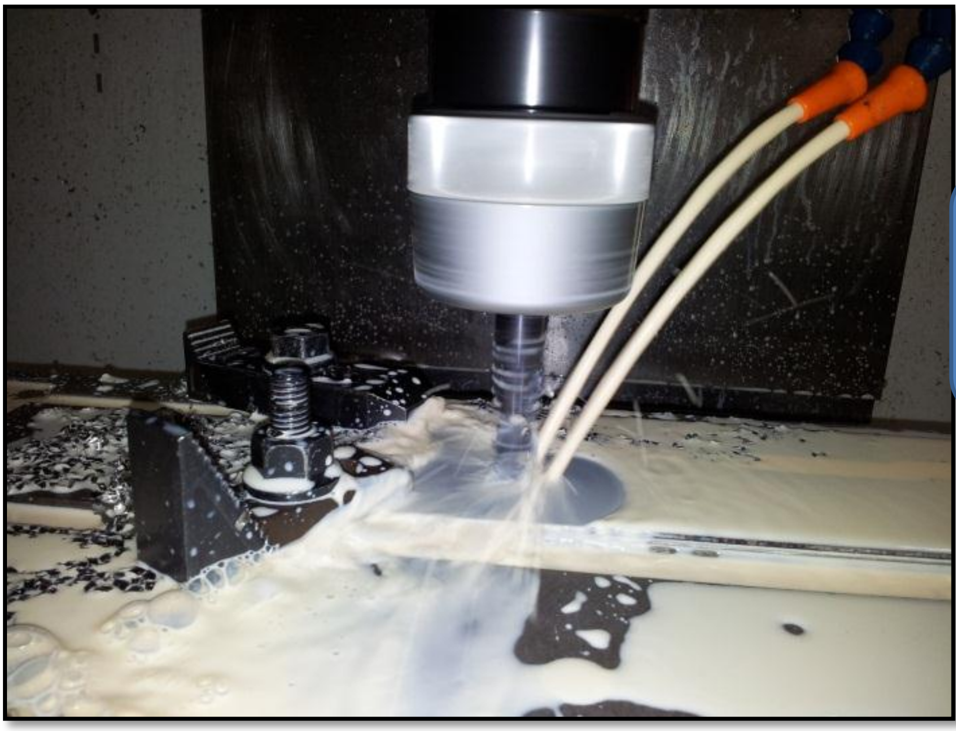
MasterCam



G-Code

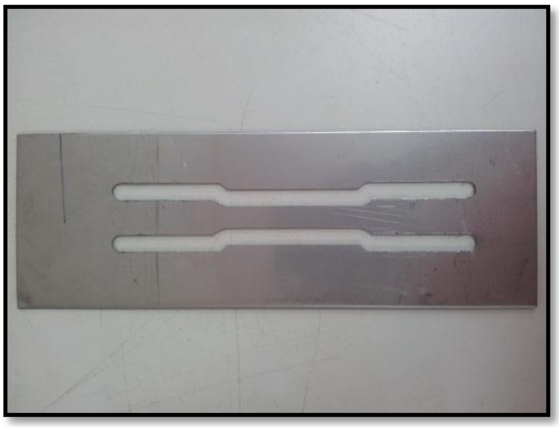


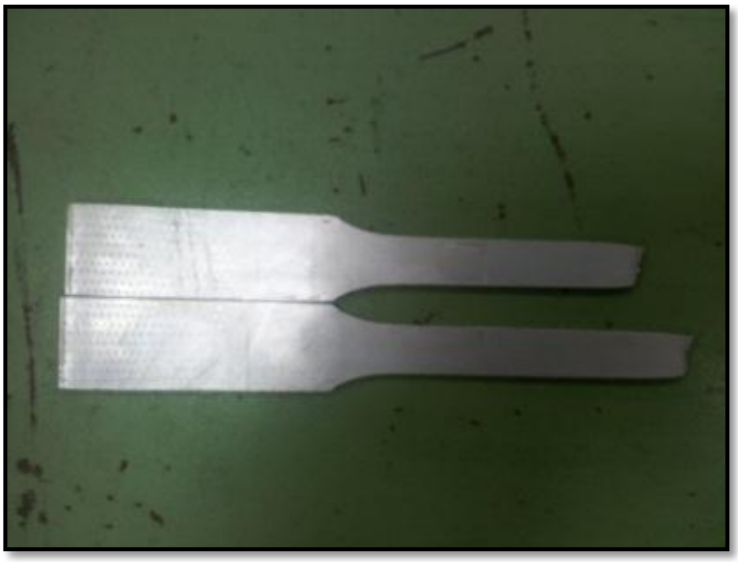
CNC-Machining



12mm Carbide
Endmill tools

Tensile Test





Finite Element