DEFORMATION AND STRESS ANALYSIS OF 3D PRINTED SPANNER WRENCH

ANG CHEAU HUI

UNIVERSITI SAINS MALAYSIA

2022

DEFORMATION AND STRESS ANALYSIS OF 3D PRINTED SPANNER WRENCH

By:

ANG CHEAU HUI

(Matric no: 142564)

Supervisor:

DR. SITI SARAH KAMALUDIN

July 2022

The dissertation is submitted to

Universiti Sains Malaysia
As partial fulfilment of the requirement to graduate with honors degree in
BACHELOR OF ENGINEERING (MECHANICAL ENGINEERING)

School of Mechanical Engineering
Engineering Campus
Universiti Sains Malaysia

DECLARATION

This work has not previously been accepted in substance for any degree and is not being concurrently submitted in candidature for any degree.

Signed………………………………………………………… (ANG CHEAU HUI)

Date…………………………………………………………………..… (24/7/2022)

STATEMENT 1

This thesis is the result of my own investigations, except where otherwise stated. Other sources are acknowledged by giving explicit references. Bibliography/references are appended.

Signed………………………………………………………… (ANG CHEAU HUI)

Date…………………………………………………………………… (24/7/2022)

STATEMENT 2

I hereby give consent for my thesis, if accepted, to be available for photocopying and for interlibrary loan, and for the title and summary to be made available outside organizations.

Signed………………………………………………………… (ANG CHEAU HUI)

Date…………………………………………………………………… (24/7/2022)

ACKNOWLEDGEMENT

First and foremost, I would like to express my utmost gratitude to my FYP supervisor, Dr. Siti Sarah Kamaludin for her supervision and guidance for me throughout the whole project. The encouragement from my supervisor gave me a lot of motivation in overcoming all the challenges that I have encountered during accomplishing the research. It is a great honor to be able to complete my research under her supervision. I would like to express my gratitude to the technician from the 3D Printing Lab and Applied Mechanics Lab of the School of Mechanical Engineering, Universiti Sains Malaysia (USM), Mr. Norijas Abdul Aziz and Mr. Fakruruzi Fadzil, who provided the great assistance in my research through guiding me on how to operate the machine and sharing me the knowledge regarding his expertise throughout the research which enriched my knowledge and technical skills that will be useful in my future career. Furthermore, I would like to thank the course coordinator, Dr. Muhammad Fauzinizam Razali for providing a lot of guidance for the students in writing a good FYP thesis by organizing plenty of workshops. Next, I also would like to express my gratitude to the lecturers and librarians who gave us the talks and workshops that helps used in completing the thesis, especially the facilitator from the library, Puan Jamilah Hassan Basri for providing the knowledge of writing the thesis format and template. Last but not least, I would like to express my sincere gratitude to my family members and friends who always provide the courage in facing all the challenges when completing the research. Thanks.

TABLE OF CONTENTS

LIST OF TABLES

Page

LIST OF FIGURES

Page

LIST OF SYMBOLS

δ	Deflection
σ_{max}	Principal Stress
ε_{max}	Principal Elastic Strain
σ_{eq}	Equivalent Stress (Von Mises Stress)
ε_{eq}	Equivalent Elastic Strain (Von Mises Strain)
σ_{UTS}	Ultimate Tensile Strength
σ_{y}	Yield Strength
θ	Angle
τ	Torque
F_{max}	Maximum Force

LIST OF ABBREVIATIONS

ABS	Acrylonitrile Butadiene Styrene
ASTM	American Society for Testing and Materials
CAD	Computer-Aided-Design
COF	Coefficient of Friction
DSC	Differential Scanning Calorimetry
FDM	Fused Deposition Modeling
FEA	Finite Element Analysis
IoT	Internet of Things
IR 4.0	Industrial Revolution 4.0
PLA	Polylactic acid
SEM	Scanning Electron Microscopy
TGA	Thermal Gravimetry Analysis
UTS	Ultimate Tensile Strength
3D	Three Dimensional

DEFORMATION AND STRESS ANALYSIS OF 3D PRINTED SPANNER WRENCH

ABSTRAK

Teknologi percetakan 3D mencipta objek dengan mencetak bahan lapisan demi lapisan, dan kerana begitu orientasi pencetakan yang digunakan dalam mencipta objek sangat mempengaruhi sifat mekanikalnya. Matlamat kajian ini adalah untuk menilai tingkah laku mekanikal bahan cetakan 3D berdasarkan orientasi cetakan yang berbeza melalui ujian mekanikal seperti ujian tegangan, ujian mampatan, dan ujian lenturan. Hasilnya menunjukkan bahawa di bawah orientasi cetakan yang berbeza, bahan cetakan 3D, PLA berkelakuan seperti bahan anisotropik, di mana sifat mekanikal PLA bergantung pada arah daya yang bertindak ke atasnya. Simulasi unsur terhingga bagi setiap ujian mekanikal akan dilakukan dan keputusan yang diperoleh dibandingkan dengan keputusan daripada eksperimen makmal. Projek itu kemudiannya akan diteruskan dengan pengoptimuman dalam mereka bentuk sepana bercetak 3D menggunakan pelbagai jenis bahan cetakan 3D, seperti PLA dan ABS. Sepana bercetak 3D dioptimumkan dengan menukar dimensi sepana, seperti panjang dan ketebalan. Kemudian, simulasi FEA dilakukan pada sepana bercetak 3D pada keadaan mengetatkan atau melonggarkan bolt menggunakan nilai daya yang berbeza. Hasilnya menunjukkan bahawa sepana dengan panjang yang lebih panjang dan ketebalan yang lebih tinggi menjadikan sepana menjadi lebih keras dan menahan lebih banyak daya yang bertindak ke atasnya. Dengan membandingkan bahan yang digunakan padanya, PLA lebih sesuai digunakan berbanding ABS kerana kekuatan tegangannya yang tinggi. Untuk semua sepana yang direka bentuk, tegasan akan tertumpu pada tepi tajam berhampiran alur sepana. Reka bentuk sepana perlu dioptimumkan lagi.

DEFORMATION AND STRESS ANALYSIS OF 3D PRINTED SPANNER WRENCH

ABSTRACT

The 3D printing technology creates an object by printing the materials layer by layer, and thus the printing orientation used in creating the object highly influenced its mechanical properties. The goal of this study is to evaluate the mechanical behaviour of the 3D printing materials based on different printing orientations through mechanical testing such as tensile test, compression test, and bending test. The result shows that under different printing orientations, the 3D printing material, PLA behaves differently as the mechanical properties of PLA are dependent on the direction of force acting on it. The finite element simulation of each mechanical testing will be performed and the results obtained is compared with the results from the laboratory experiments. The project will then continue with the optimization in designing a 3D printed spanner wrench using different types of 3D printing materials, such as PLA and ABS. The 3D printed spanner wrench is optimized by changing the dimension of the spanner wrench, such as length and thickness. Then, the FEA simulation is done on the 3D printed spanner wrench at the condition of tightening or loosening the bolt using different values of forces. The result shows that the spanner with a longer length and a higher thickness makes the spanner becomes tougher and to withstand more force acting on it. By comparing the materials applied to it, PLA is more suitable to use in fabricating the spanner wrench than ABS due to its high tensile strength. For all the designed spanner wrench, the stress will concentrate at the sharp edge near the groove of the spanner wrench. The design of the spanner wrench needs to be further optimized.

CHAPTER 1

INTRODUCTION

1.1 Overview of Finite Element Analysis (FEA) and 3D Printing Technology

The finite element analysis (FEA) is a commonly used computational tools in predicting the behaviour of a components under a given conditions. Through finite element analysis, designers and engineers can simulate the behaviour of their designed products before the fabrication process to be able them to understand the performance of that design that is going to be investigated. With the aid of the finite element analysis, it can help to reduce the material waste while at the same time predicting the potential issues that might happen in the design before creating a physical prototype to minimize the risk that will happen to the model which helps a lot in cost reduction for the production line in any practical industry. The structural behaviour of the design on how the it reacts to the stress or other load conditions applied to it can be done through the simulation by subdividing the model into smaller and simple parts known as finite elements to be solved by its governed mathematical equation. Implementation of finite element analysis in the design process can ensure the working capability, safety and optimizing the product design before the fabrication process.

In this modern manufacturing, the additive manufacturing or 3D printing technology is becoming more popular due to its simple way of creating product and its ability to create a product with a complicated design by importing the product design from the computer-aided design (CAD). This 3D printing technique are often used in creating a prototype for a particular design before the product is fabricated massively to be released in the market. The common material that has always been used in 3D printers usually is Acrylonitrile Butadiene Styrene (ABS) and Polylactic acid (PLA). Unlike the subtractive manufacturing process at where the product is formed through removing the desired part from a block of the material workpiece using a cutting machine, the product formed through 3D printing will usually be created layer-by-layer deposition of material from a nozzle to a platform to form the 3D object followed the specific coordinates that is set before it is started to print which help to reduce the material waste. Because of this feature, 3D printing also allows to create a complex structure of the design. The process of creating a 3D printed product is quite customizable as the design can be easily edited through the computer-aided designs (CAD) as they can visualize how the final design of the product going to look like through CAD before they printed it out. Industries such as aerospace, automotive, biomedical and consumer products are benefitted from the 3D printing technology. The most commonly used type of 3D printer is the fused deposition modeling (FDM) printer due to its low cost, ease to use and high dimensional accuracy features.

The development of industrial revolution 4.0 (IR 4.0) has emphasized the importance of digitalization and automation in manufacturing. The availability of automation in industrial regions helps the industry in improving its production speed and quality to reach a higher stage. It allows the exchange of information in a faster way within the technologies and production line through the internet of things (IoT). Additive manufacturing has played a significant role in the way of developing industrial revolution 4.0 as it allows the product to be designed in a more flexible way before it is manufactured. The implementation of additive manufacturing enables the industry to save up their cost on exploiting the expensive tools for the traditional manufacturing at the same time it helps in reducing the material waste and the time for the post-processing process during fabricating the product. But, before any design for the product is fabricated to be used as the prototype, the design of that product can be utilized and optimized through the aid of the finite element analysis to simulate the real-world condition that might be happened in the design. Applying the finite element analysis during designing the product helps us to analyse the maximum stress that allows applying to the design before it exceeds its maximum allowable value for that design. With the aid of the computer-aided design (CAD), the design can be changed easily so that it can be optimized until it is able to meet the requirement that we want for the design. The process of discretization of the design into tiny nodes allows us to understand what going to happen to all the nodes as the stress acting on it and helps us to spot the part that going to undergo failure easily through the contour profile that formed for the results of the simulation.

In this project, we are going to design a spanner wrench that will be fabricated via a 3D printer. The 3D printed spanner will have different material compared to the original metal-made spanner and its design is optimized through finite element analysis. Unlike the metal spanner, the tensile strength of the plastic spanner is comparatively lower than the metal spanner. To ensure the plastic spanner able to apply the torque that is required to tighten or loosen the bolt or nut, there is some improvement in terms of the length and the thickness of the plastic spanner that will be done and tested under simulation for the finite element analysis of the plastic spanner. The purpose of designing a plastic spanner is to test whether the spanner fabricated through additive manufacturing like 3D printing can replace the original metal spanner in industrial or not. The design for the plastic spanner will be optimized and analysed through the application of FEA on optimizing the design. Before the spanner wrench is designed, the 3D printing material that will be going to be used in designing the plastic spanner will undergo a series of mechanical testing in order to understand the behaviour of the material and its performance before it is used in the fabrication of the 3D spanner. The result from the mechanical testing will then compare with the one that we obtained from analysing the result in the FEA simulation for the test specimen. After conducting the test, the simulation of the plastic spanner with the different dimensions will be done in order to evaluate the suitable 3D printing material and its dimension used to design the spanner wrench.

1.2 Project Objectives

The purposes to carry out this project are

i. To evaluate the mechanical behaviour of the 3D printing material that will be used in fabricating the spanner wrench through mechanical testing and simulation

ii. To implement finite element analysis as a design optimizing tool for spanner wrench using 3D printing materials

iii. To evaluate the optimized dimension and material used for the 3D printed spanner wrench

1.3 Problem Statements

Different materials exhibit different mechanical and physical properties. Understanding the behaviour of the materials under a given application and condition is important to determine the reliability of the products made from these materials. Although 3D printing technology offers a convenient way of product creation, the mechanical properties of the 3D printing material should be determined beforehand.

Evaluation of materials used for 3D printing is crucial in order to assess the behaviour of a product during simulation before the product creation process takes place.

Hand tools such as spanner wrench are very essential tools that are used in many working fields. A wrench is a tool that is used by applying torque to tighten or loosen rotary fasteners (e.g., nuts and bolts). In designing a hand tool, finite element analysis can be conducted to ensure that the component is capable to produce enough load/torque and withstand the mechanical loads and stresses that are subjected to it. The purpose of this study is to use computer-aided design (CAD) and finite element analysis (FEA) in engineering design optimization of a spanner wrench made from 3D printed material such as PLA and ABS to evaluate the effectiveness of the designed spanner wrench.

1.4 Scope of the Project

The material properties of 3D printed PLA will be determined using mechanical testing such as tensile test, compression test and flexure test. The specimen for the tensile test will be based on ASTM D638, the compression test will be based on ASTM D790, and the flexure test specimen is based on ASTM D790. The loading used for tensile, compression and flexure test is 1mm/min. A finite element simulation of these mechanical testing will be carried out using ANSYS workbench 2021 R2 using the material library available in the software under static condition.

Several designs of spanner workbench of length, 70 mm and 90 mm and thickness, 5 mm, 8 mm and 11 mm will be created using the CAD software, SOLIDWORKS. These designs will be imported into finite element analysis software, ANSYS Workbench to simulate the mechanical response of these designs. These designs will be subjected to a load of 10 N, 20 N and 30 N acting at the end of the spanner wrench and the simulation will be performed under static condition. The simulation of these design will be performed by assigning the material properties of PLA and ABS available in the ANSYS software.

In this project, there will be a total of two phases in order to accomplish this project, one is to evaluate the mechanical behaviour of the 3D printing material such as PLA through mechanical testing and simulation for the mechanical testing before this material is used in designing the 3D printed spanner wrench; the other one is to analyse the suitable material and dimension used for the fabrication of 3D printed spanner wrench through simulation only. For the first phase, the mechanical testing on the 3D printed material is done to evaluate its performance through fabricating the test specimen in different printing orientations. The result from the mechanical testing will then be compared with the result obtained through the simulation for how the specimen behaved during the mechanical test using ANSYS software. For the second phase, the spanner wrench with different lengths and thicknesses is designed using SolidWorks software. The simulation of how the spanner wrench is used to tighten or loosen the bolt is done by applying different 3D printing materials (PLA and ABS) and different forces acting on the spanner wrench in order to understand the stress and structural analysis of the plastic spanner wrench in a different situation. Only the elastic properties of the 3D printing materials are considered during the finite element analysis for all cases. The suitable 3D printing material and the dimension used to design the plastic spanner wrench are determined through this phase before it is fabricated to be used to replace the metal spanner wrench. The maximum force that can be exerted on each plastic spanner wrench with different dimensions and materials is evaluated.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

In this chapter, the findings from the literature search regarding the mechanical properties of the 3D printing materials, and the optimization of the design through finite element simulation is discussed.

2.2 Mechanical Behaviours of 3D Printing Materials under Different Conditions

There is a study related to the review on applying the 3D printing technology in the manufacturing process. [1] studied optimizing the parts that used Polylactic acid (PLA) as material through fused deposition modeling techniques. This study involved the modification of the molecular structures of the PLA when fabricating the product by doing the improvement in cross-linking of the PLA structures and the crystallinity of PLA. Based on the study, the infill parameters for the fabrication part affect the tensile strength of the parts are learnt. From the paper, the parts with higher infill density and with lines as the infill pattern have the highest tensile strength.

Based on the study from [2], research on comparing the 3D printed parts fabricated using virgin PLA and PLA/wood fibre composites as the material through mechanical testing. This study also emphasizes the change in 3D printing parameters such as the infill density, layer height, and the number of shells may also affect the performance of the sample parts during mechanical testing. Based on the study, the mechanical strength of the 3D printed parts can be improved through an increase in the number of shells or a decrease in the number of infill cavities are learnt. When comparing the material, the parts with virgin PLA as the material possess higher mechanical strength compared with PLA/wood fibres due to the chemical bonding between the PLA and wood fibres is weakened due to the randomly oriented wood fibre bundles.

A study related to the additive manufacturing to test the mechanical properties of the thermoplastic materials used, polylactic acid (PLA) and acrylonitrile butadiene styrene (ABS) during the process is conducted by [3]. In the experiment, the researchers have conducted both tensile and compression tests for both PLA and ABS specimens. There are a total of ten specimens for each thermoplastic material to analyse the ultimate tensile strength (UTS), strain, elastic modulus and, the compression yield strength for each material. Through the experiment, they found out that the tensile strength for the 3D printed samples is weaker than the injection moulding samples due to the anisotropic behaviour of the 3D printed product. Based on the result after conducting the tensile test on 10 samples for ABS and PLA materials, the average UTS for ABS was comparatively lower than the average UTS for PLA, also the elastic modulus of ABS obtained also consists of a lower value when compared with PLA. Based on the result after conducting the compression test on 10 samples for ABS and PLA materials, the average yield strength for ABS was comparatively higher than the average yield strength for PLA. Throughout the whole experiment, the researchers concluded that the tensile strength of ABS material is more consistent than PLA which varied between samples. The inter-layer bond strength properties for 3D printed samples greatly influence the consistency of the mechanical properties during the tensile test but not in the compression test.

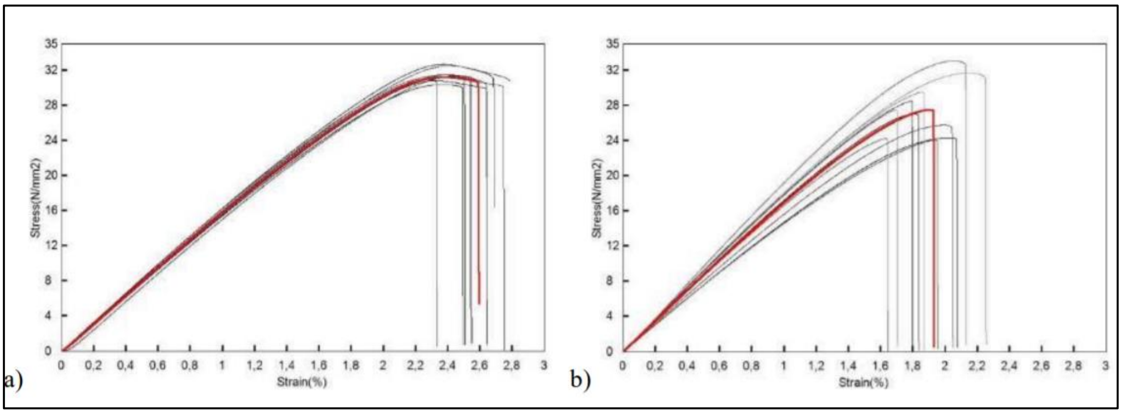


Figure 2.1: a) Stress-strain Curve for ABS and b) Stress-Strain Curve for PLA Thermoplastic Material

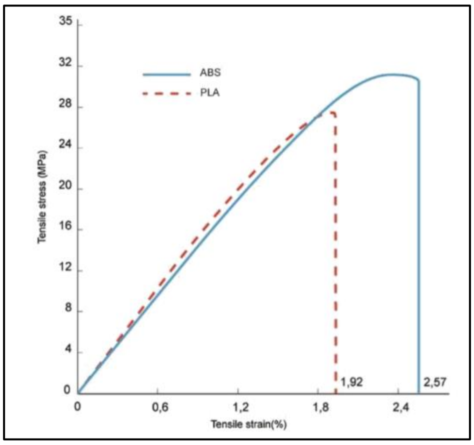


Figure 2.2: Average Tensile Strength Values for ABS and PLA Thermoplastic Material

In the study from [4] related to optimization of FDM parameters in improving the 3D printed parts using PLA and ABS, the researchers argued that the mechanical properties of the 3D printing materials greatly affected by the 3D printing parameters, such as infill density, infill speed, nozzle temperature, infill pattern and also the printing materials through mixing the original materials with other material to improve the strength of the original materials. This can be proven through series of tests, such as tensile test, compression test, bending test, differential scanning calorimetry (DSC) test, thermal imaging test, thermal gravimetric analysis (TGA) test and also scanning electron microscopy (SEM). The researchers also proposed that the result from the mechanical test cannot be compared with the computational result obtained from finite element simulation due to the presence of air gaps between the layers for the printed parts which make it cannot be considered as solid even though the infill density of that part is 100%.

The next study related to the mechanical properties of PLA material is done by [5]. In this study, the mechanical tests and simulations are conducted on the dog-bone-shaped PLA specimens which is got by cutting the PLA films fabricated from the hot-pressing process in order to understand the mechanical behaviour of PLA material. The tensile test is done under different loading angles and loading speeds (strain rate) at ambient temperature. The mechanical behaviour of PLA is depending on the strain rate.

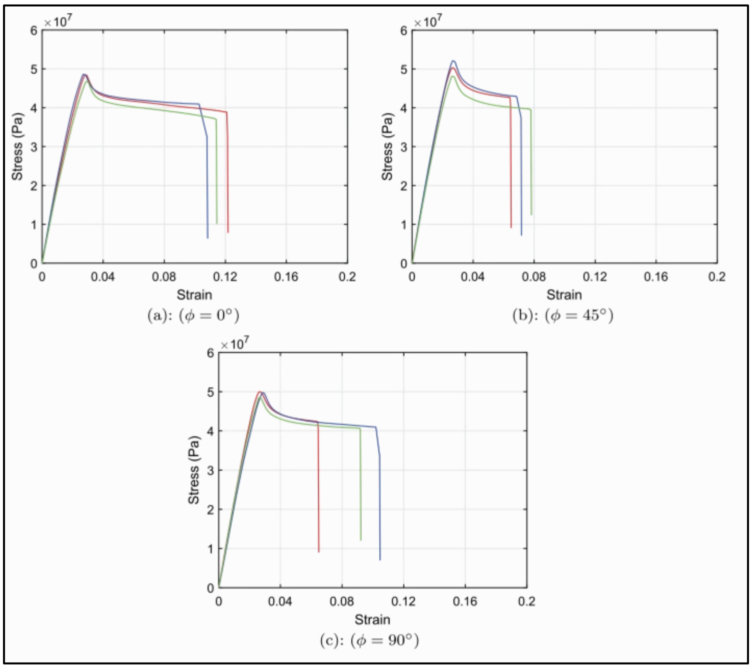


Figure 2.3: Experimental Stress-Strain Curve for PLA at Different Loading Angles (Φ= 0◦, 45◦ and 90◦) under Loading Speed of 0.006/s under Ambient Temperature

Another study that is related to tensile behaviour for ABS at different temperatures is performed by [6]. The ABS samples used in this experiment were produced through an injection moulding process. The tensile tests are conducted at different temperatures every 10 °C starting from 40 °C until 130 °C. For verification of the accuracy of the result, the researchers also did the tensile test at temperatures 75 °C and 115 °C which is below and above the glass transition temperature respectively. The result shows the stress increases gradually with the increase in temperature under the same strain due to the viscoelasticity of the material.

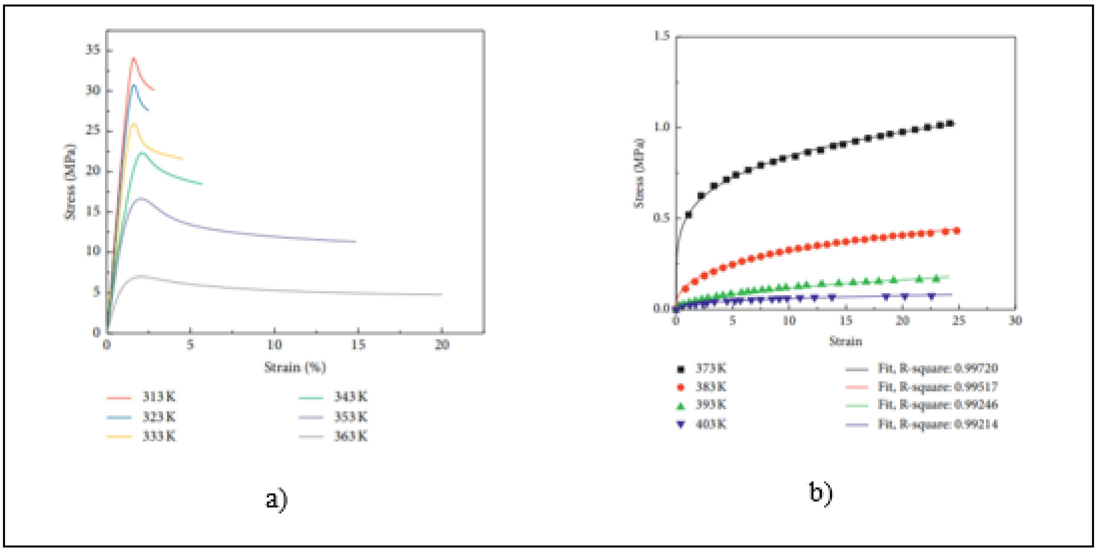


Figure 2.4: Stress-Strain Curve for ABS at Different Temperatures Below Glass Transition Temperature and b) Stress-Strain Curve for ABS at Different Temperatures Above Glass Transition Temperature

There is also a study on investigating the mechanical and also the anisotropic properties of the 3D printing materials, such as PLA and ABS conducted by [7] through changing the printing orientation and its printing layers. The test was conducted according to ASTM D638 Standard for the tensile test to observe the anisotropic properties of the 3D printing materials by using five different printing orientations. It proved that the strength of the PLA and ABS are highly dependent on the printing orientation of the material.

2.3 Optimization of the Design Using Finite Element Analysis

Various studies have been done on optimizing the design for the product through finite element analysis (FEA). Most of the researchers are more focused on reducing the weight of the original design for the product to achieve the optimal design for the product through the topology optimization process. [8] studied on applying topology optimization to the design of a bell-crank through weight reduction using finite element analysis. According to the studies, the researchers managed to reduce the weight of the modified part from the original design by 3% at the same time achieved in reducing the maximum principal strain and maximum von Mises stress for the design when the same static load conditions are applied for both the modified design part and the original part. This study demonstrated the effectiveness of FEA in design optimization to produce a lightweight product.

Besides, [9] have also done a similar design optimization using FEA by reducing the weight of the original design but the study was focused on optimizing the design of the backhoe attachment for the hydraulic excavator for the material handling equipment. Under this study, static force analysis is applied to the design in order to determine the mechanical strength of structures that worked under a specific load condition. During the formation of the geometry state, the researchers tried to reduce the weight of the arm and boom for the excavator and then compare it with the original geometry of the arm and boom under the same stress level. The result shows that the optimized model for the arm of the excavator managed to reduce its weight and also reduce the maximum stress value acting on it by comparing with the initial model of the arm. This study proved again the importance of applying FEA for the design optimization before it is fabricated.

A study related to utilizing the used of FEA method on modeling the object through applying the functional gradient material (FGM) which is also known as the heterogeneous materials that spatially vary the material property and the functional flexibility. is performed by [10]. To simplify the study process, the researchers create a cantilever beam as the model and then apply different materials to the model to evaluate the behaviours of the beam. For this study, the researchers applied hard material and soft material to the model to simulate what will be going to happen when a load is applied at the free end of the beam. The result shows that the cantilever beam with soft material experienced deflection ten times more than the one using hard material. This indicated that the Young's Modulus for the soft material is ten times lower than the hard material. In order to solve this, the researchers tried to optimize the design to achieve optimal material composition by changing the continuous material distribution into discrete material distribution to be used in the manufacturing process. The researchers believe that design can be optimized through discretization of the material distribution for the product.

Other studies related to using FEA to optimize the design of the connecting rod by changing the parameters of the design and the material used have also been done by the researchers. [11] studied changing the design parameters and the material used for the original design of the connecting rod in order to optimize the design. The thickness of the I-beam part for the connecting rod is increased while the parameters for other parts remain unchanged in this study to observe the behaviours of the model when the compressive bearing load is acting on it. Through comparing E-glass/Epoxy and Kevlar 49/Epoxy, the researchers found out that the E-glass/Epoxy consists of lower longitudinal modulus and higher density compared to Kevlar 49/Epoxy. Based on the validation through the result from FEA, the modified design for the connecting rod with a thicker I-beam helps in reducing the maximum stress value compared to the original design of the connecting rod. In terms of materials, E-glass/Epoxy gives a better result compared to the Kevlar 49/Epoxy.

CHAPTER 3

METHODOLOGY

3.1 Introduction

This chapter will give an overview of the methodology used in this project. The project starts with the evaluation of the experimental result of PLA 3D printed specimen through mechanical testing and the computational result through simulation of the testing using finite element analysis. The project will then continue with the optimization of 3D printed spanner wrench using the PLA and ABS material with applied different length and thickness to it through finite element analysis. The general flow of the project can be summarized as in Figure 3.1 and Figure 3.2.

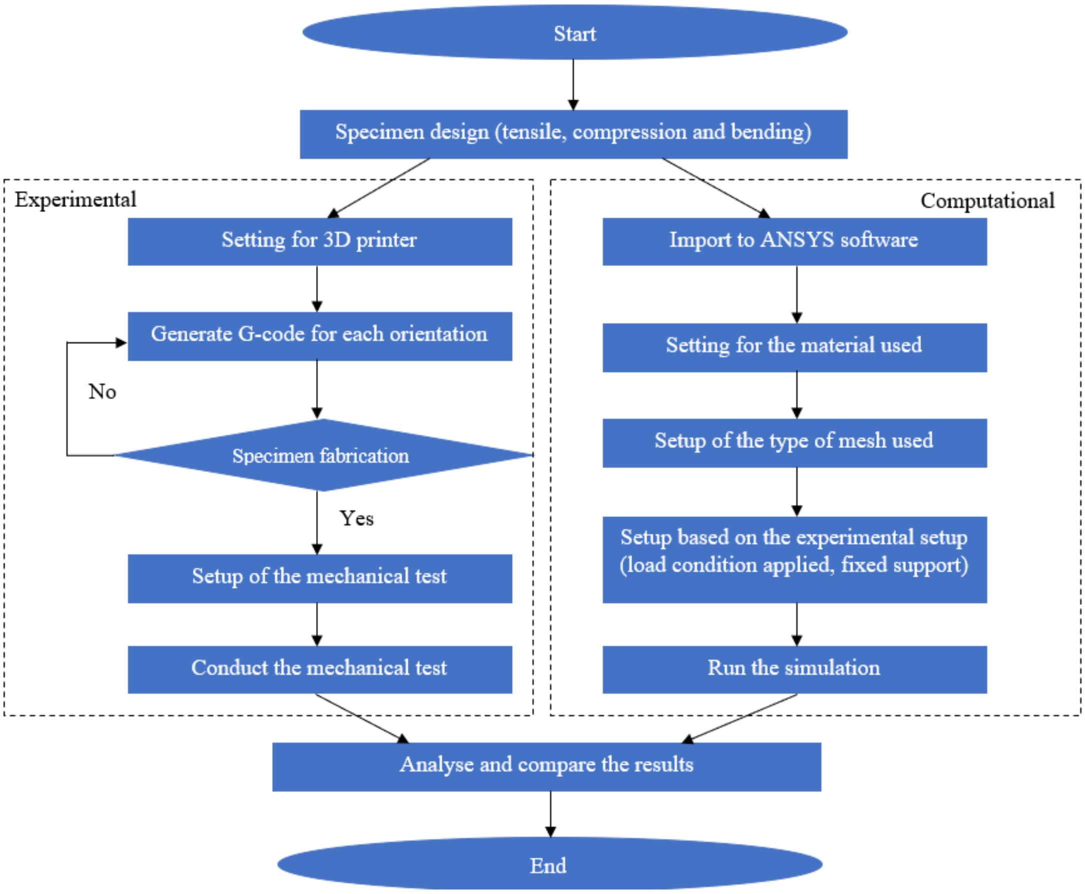


Figure 3.1: Flow chart for Mechanical Testing of 3D printed PLA specimen

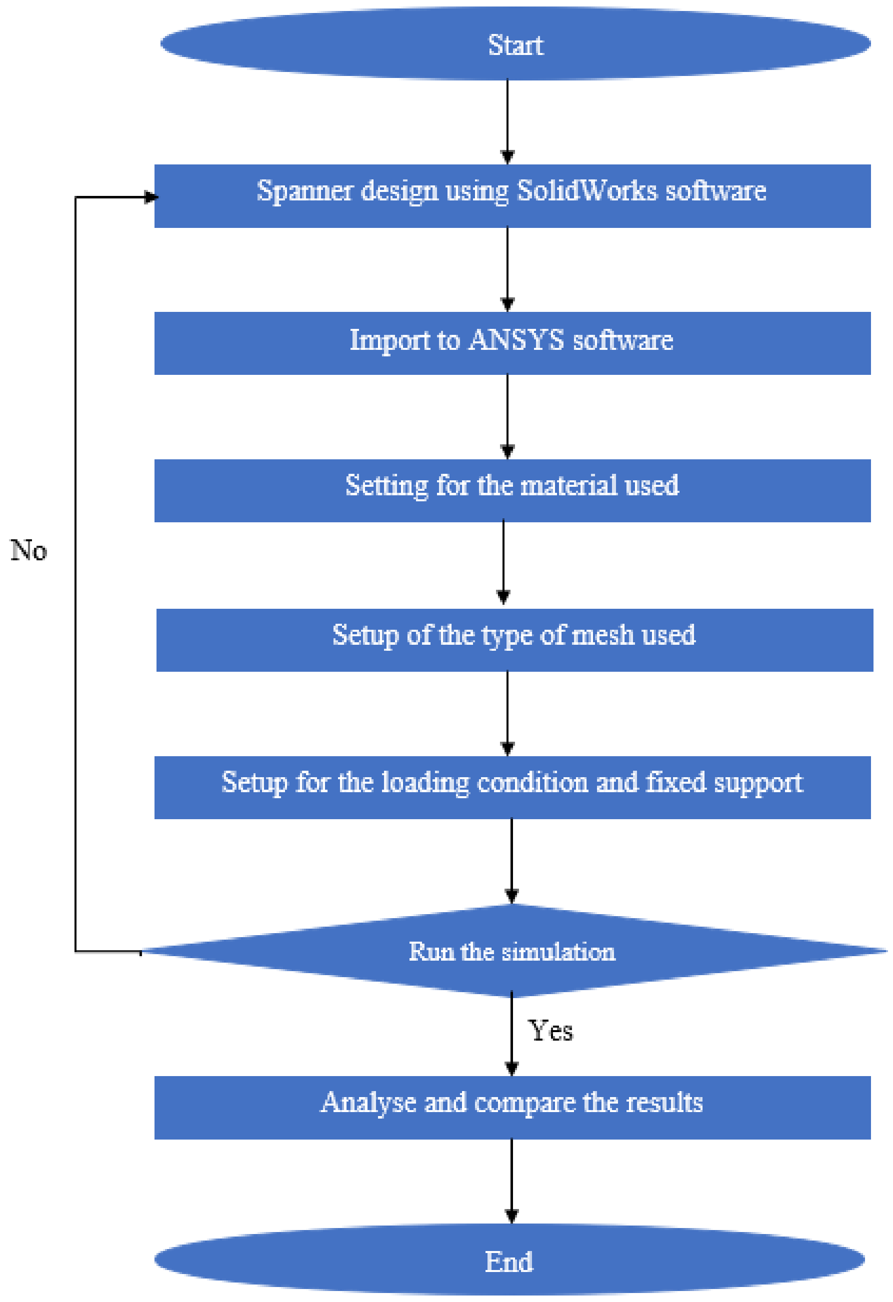


Figure 3.2: Flowchart for the Optimization of the 3D Printed Spanner Wrench Design through FEA

3.2 Material and Experimental Procedures

For the first phase of this project, the mechanical testing is done on the specimen that is fabricated using 3D printing material in order to evaluate the performance of the material before it is used to fabricate the plastic spanner wrench. The FEA simulation of the result for the mechanical testing has also been done to do the comparison with the real-life conditions.

3.2.1 3D Printed Test Specimen Preparation

For the purpose of researching the influence of the printing orientation on the mechanical properties of the 3D printing test specimens, the pure PLA filament with a diameter of 1.75 mm is used to fabricate the test specimen for each mechanical testing experiment. The information on the Rambo LanBo 3D Printed PLA filament obtained from the manufacturer is shown in Table 3.1. The printing process for each test specimen is done by using the Creality Ender 5 Plus printer, which also is a fused deposition machine as shown in Figure 3.3.

There are a total of three types of mechanical tests that will be conducted to test the mechanical behaviour of the 3D printing material: tensile test, compression test, and bending test. Each mechanical test will consist of two types of printing orientation and all the details of the printing orientation for each mechanical test are stated in Table 3.2. Three test specimens are prepared for each printing orientation for all types of mechanical tests by referring to the ASTM standard and the average value is determined to improve the reliability and accuracy of the experimental data. The Ultimaker Cura 4.13.1 software is used to adjust the parameters used during the fabrication process of a 3D printed specimen. The infill pattern used is linear with an infill density of 100% using the printing speed of 70 mm/s for all the test specimens. The appropriate set nozzle temperature is set to 215 °C according to the information of the filament data and the printing bed temperature is set to 60 °C to minimize the shrinkage of the specimen from happening through providing constant heating to the material used.

Table 3.1: Information on PLA Filament

Material	Operating temperature (°C)	Filament diameter (mm)
Pure PLA	190-220	1.75±0.02

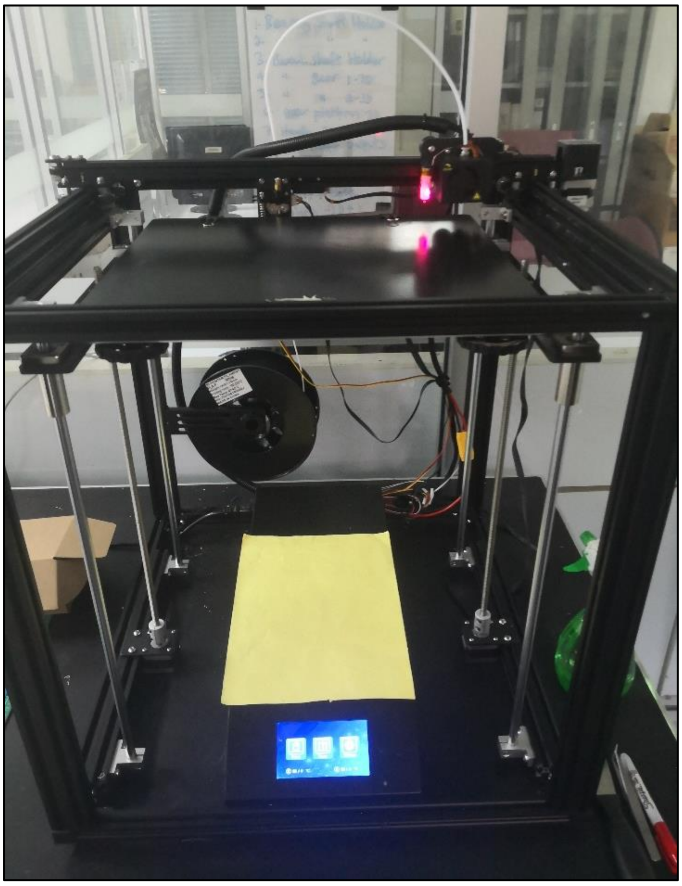
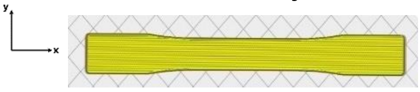
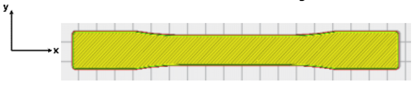
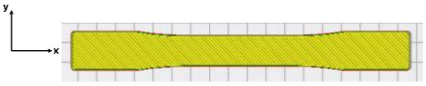
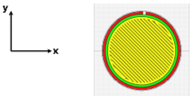
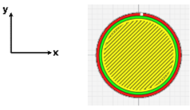
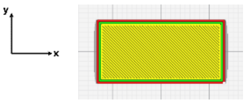
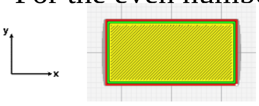
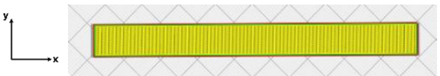
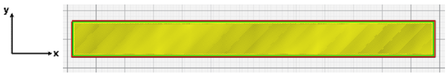
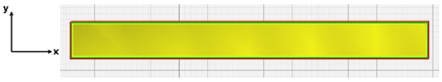


Figure 3.3: 3D Printer used for the Fabrication- Creality Ender 5 Plus

Table 3.2: Preview of the Printing Orientation of 3D Test Specimens

Test	Preview of the Printing Orientation (From the top view of the 3D printer) 1	2
Tensile (N= 20 for both)	For the odd number layer:  Orientation 0◦ from +x-axis For the even number layer:  Orientation or 90◦ from +x-axis	For the odd number layer:  Orientation 45◦ from +x-axis For the even number layer:  Orientation 135◦ from +x-axis
Compression (N= 127 for 1, N= 63 for 2,)	For the odd number layer:  Orientation 135◦ from +x-axis For the even number layer:  Orientation 45◦ from +x-axis	For the odd number layer:  Orientation 135◦ from +x-axis For the even number layer:  Orientation 45◦ from +x-axis
Bending (N= 16 for both)	For the odd number layer:  Orientation 0◦ from +x-axis For the even number layer:  Orientation 90◦ from +x-axis	For the odd number layer:  Orientation 45◦ from +x-axis For the even number layer:  Orientation 135◦ from +x-axis

Note: N means the total number of printing layers for each test specimen

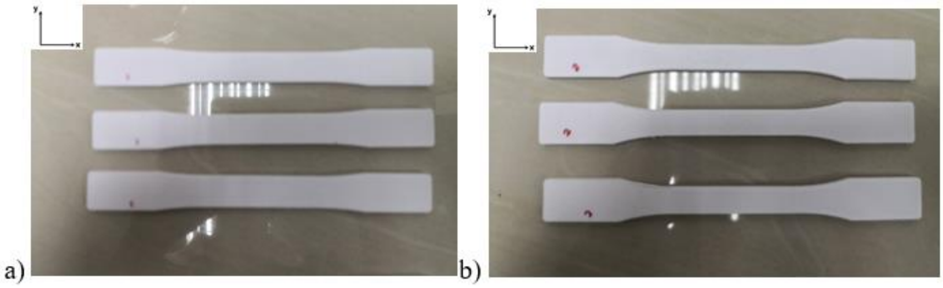


Figure 3.4: 3D Printed Tensile Test Specimen a) Orientation 1 and b) Orientation 2

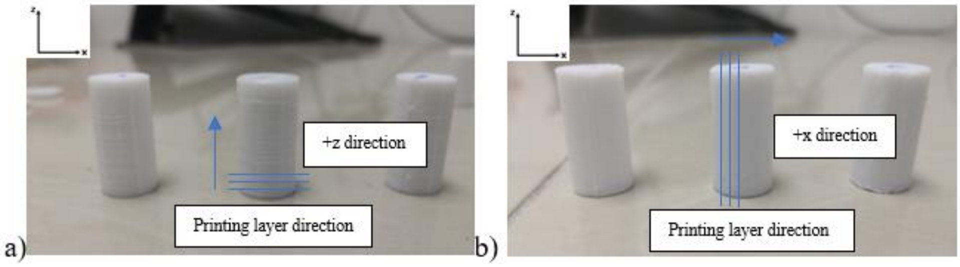


Figure 3.5: 3D Printed Compression Test Specimen a) Orientation 1 and b) Orientation 2

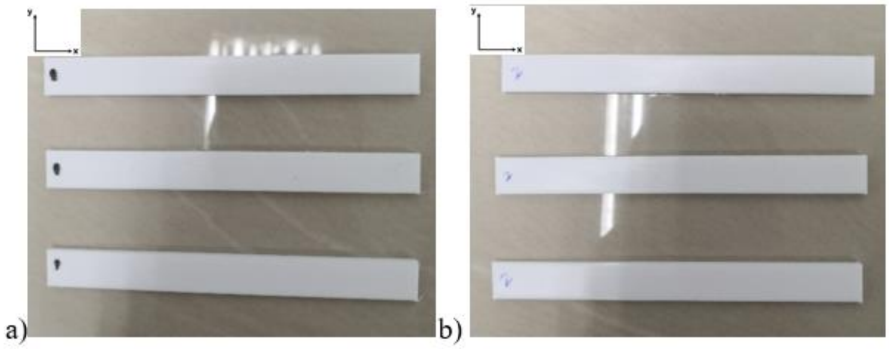


Figure 3.6: 3D Printed Bending Test Specimen a) Orientation 1 and b) Orientation 2

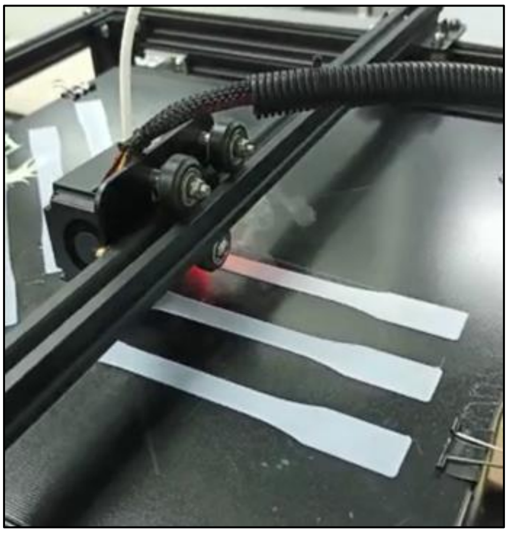


Figure 3.7: 3D Printing Process for Test Specimen

3.2.2 Experimental Setup for 3D Printed Test Specimen

All the mechanical tests are conducted under ambient temperature and pressure. There are total of two printing orientation will be compared in the test and the number of 3D printed specimens was three for each orientation. The mechanical testing was performed using Instron 3367 Universal Testing Machine (UTM) (as shown in Figure 3.8) and was used together with the Bluehill Universal software for the control of the loading method. For each mechanical testing, the specimens were subjected to the crosshead speed of 1 mm/min according to the ASTM standard. The ASTM standard used for the testing is as tabulated in Table 3.3.

Figure 3.8: Instron 3367 UTM in Applied Mechanics Lab

Table 3.3: ASTM Standard and Dimension for each Test Specimen

Test	ASTM Standard	Dimension
Tensile	D638 Type II	L180 mm × W20 mm × T4 mm
Compression	D790	D12.7 mm × T25.4mm
Bending	D790	L125 mm × W12.7 mm × T3.2mm

3.2.2.1 Tensile Test

The dog-bone shaped specimens were used in the tensile test. The specimens have a total length of 180 mm, total width of 20 mm and thickness of 4 mm according to dimension stated in ASTM D638 standard [12]. Figure 3.9 shows the dimensions of the tensile specimen. The tensile specimens were fabricated using two different printing orientations, and each orientation will have 3 specimens using PLA materials as stated in Section 3.2.1. Under the tensile test, the uniaxial tensile experiments are performed under ramp rates of 1 mm/min for the tensile test specimen based on ASTM D638 standard. Figure 3.10 shows the experimental setup of the tensile test. Both ends of the test specimen are clamped to the wedge grips and the width, thickness and gauge length of the tensile specimen are inserted into the Bluehill software that is used to program and run the UTM for the test. The test is repeated three times for the test specimen with the same orientation until the specimen is torn apart into two pieces in order to obtain the average value of its maximum tensile load, also the stress-strain curve and force-extension curve are plotted and established by the Bluehill software. The procedure is then repeated with the test specimen with another orientation as shown in Table 3.2.

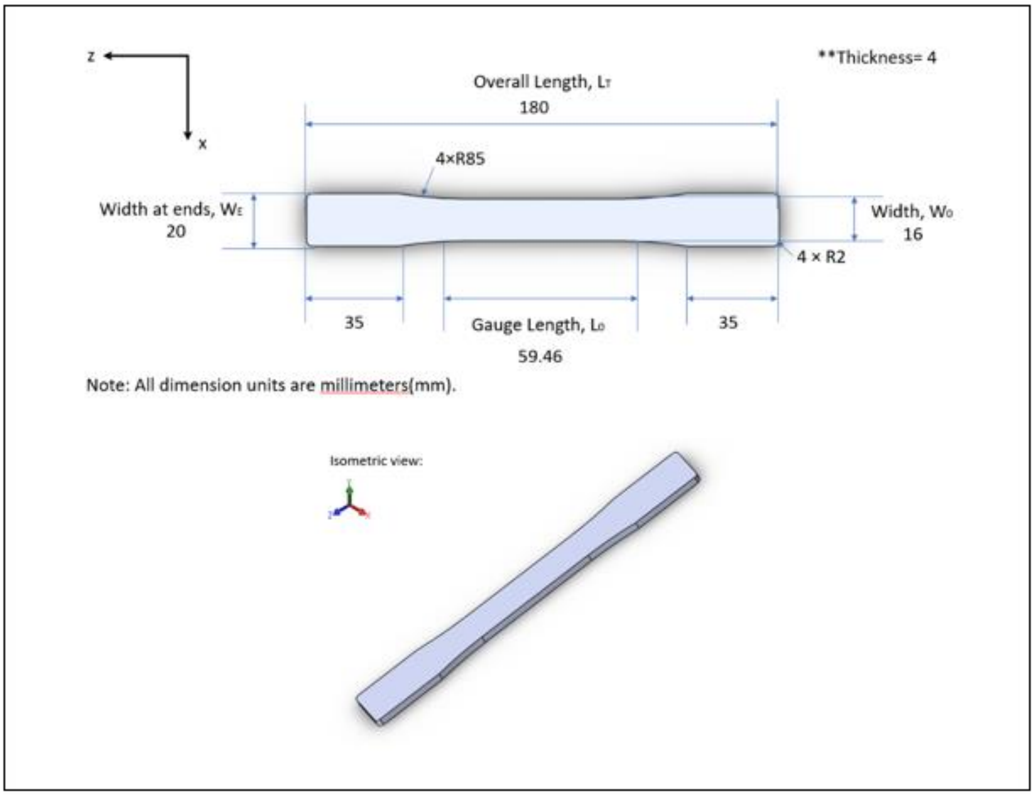


Figure 3.9: Schematic Drawing of Tensile Test Specimen

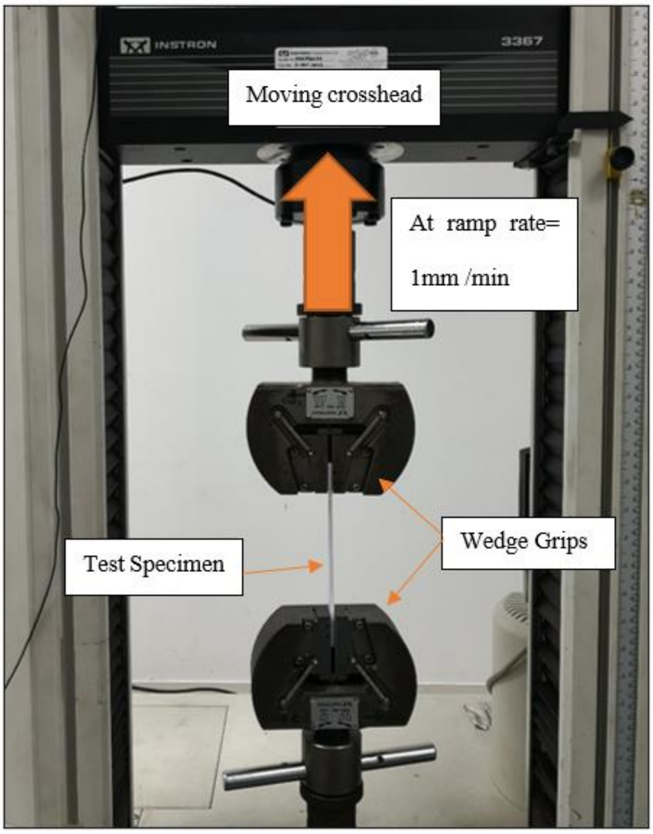


Figure 3.10: Dog-Bone-Shaped Test Specimen fixed on the UTM for Tensile Test

3.2.2.2 Compression Test

The specimen used for compression test is cylindrical specimens, with diameter of 12.7 mm and height of 25.4 mm according to dimension stated in ASTM D790 standard [13]. Figure 3.11 shows the dimensions of the compression specimen. The compression specimens were fabricated using two different printing orientations, and each orientation will have 3 specimens using PLA materials as stated in Section 3.2.1. Under the compression test, the uniaxial compression experiments are performed under ramp rates of 1 mm/min for the compression test specimen based on ASTM D790 standard. Figure 3.12 shows the experimental setup of the compression test. The compression specimen is placed at the center of the bottom platen and the top platen is adjusted to be just at the top of the compression test specimen without touching the surface of the specimen. The diameter and height of the compression test specimen are then inserted into the Bluehill software that is used to program and run the UTM for the test. The test is repeated three times for the test specimen with the same orientation until the specimen is reached its stopping criteria to obtain the average value of its maximum compression load, also the stress-strain curve and force-extension curve are plotted. The procedure is then repeated with the test specimen with another orientation as shown in Table 3.2.

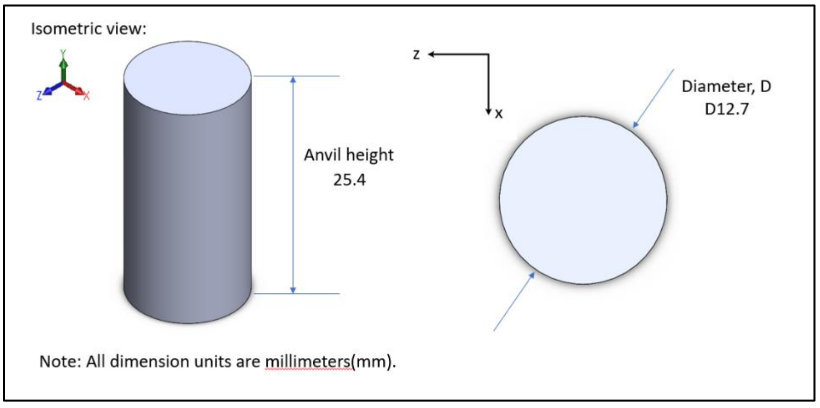


Figure 3.11: Schematic Drawing for Compression Test Specimen

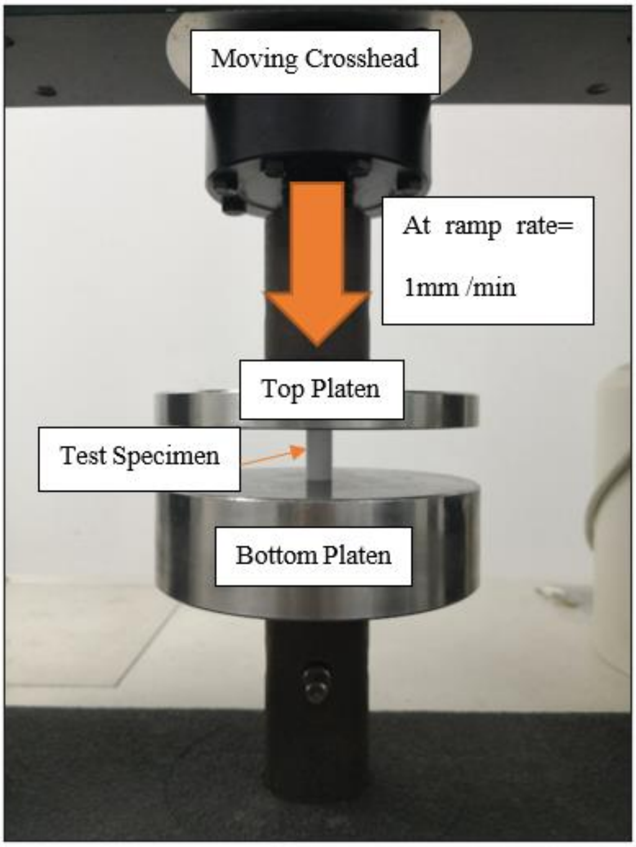


Figure 3.12: Cylindrical Specimen fixed on UTM for Compression Test

3.2.2.3 Bending Test

The rectangular bar specimens were used in the bending test. The specimens have a total length of 125 mm, width of 12.7 mm and thickness of 3.2 mm according to dimension stated in ASTM D790 standard [14]. Figure 3.13 shows the dimensions of the bending specimen. The bending specimens were fabricated using two different printing orientations, and each orientation will have 3 specimens using PLA materials as stated in Section 3.2.1. Under the bending test, the three-point bending experiments are performed under ramp rates of 1 mm/min for the bending test specimen based on ASTM D790 standard. Figure 3.14 shows the experimental setup of the bending test. The specimen rests on two cylindrical supports and the distance between the two supports was 60 mm. The loading nose is adjusted to be just at the top of the bending