Comparison of experimental result and FEM analysis of ABS and PLA on ASTM D638

Individual Project (E6H3002)

Faculty of Technology

School of Mechanical and Design Engineering

A thesis submitted in partial fulfilment of the requirements for the degree of

Bachelor of Engineering (Hons) Mechanical Engineering (International)

by

Student Name: Phone Thet Oo

Student ID: 2149426

Local Supervisor:

Dr. Thu Han Tun

Chindwin-PSB Institute

January 2022

UNIVERSITY OF PORTSMOUTH

School of Mechanical and Design Engineering

Name: Phone Thet Oo

Student ID: 2149426

Project Title: Comparison of experimental result and FEM analysis of ABS and PLA on ASTM D638

Course: BEng (Hons) Mechanical Engineering

Supervisor: Dr. Thu Han Tun

Date: 2021/22

UNIVERSITY OF PORTSMOUTH

Declaration of Originality and Approval of Research Ethics

Project Title: - Comparison of experimental result and FEM analysis of ABS and PLA on ASTM D638

Student Name: - Phone Thet Oo

HEMIS Number: - 2149426

'I certify that this is my own work, and it has not previously been submitted for any assessed qualification. I certify that School of Engineering research ethics approval has been obtained and the use of material from other sources has been properly and fully acknowledged in the text'.

Signed: - *Phone* Dated: - 2/1/2023

Statement of Originality

Title of Report: Comparison of experimental result and FEM analysis of ABS and PLA on ASTM D638

Word count: 7452

I, the undersigned, declare that this report is my own original work. Where I have taken data, ideas and/or wording from another person or published source this is explicitly acknowledged and referenced in the text.

I provide a copy of the electronic source from which this report was printed. I give my permission for this report, and the electronic source, to be used in any manner considered necessary to fulfil the requirements of the University of Portsmouth Regulations, Procedures and Codes of Practice.

■ I give permission that this report may be made available to others via the University of Portsmouth library for the purpose of research.

■ I do not give permission that this report may be made available to others via the University of Portsmouth library for the purpose of research.

Name: Phone Thet Oo

Student number: 2149426

Signature: *Phone*

Date: 2/1/2023

Table of Contents

Acknowledgements ... 12
Abstract ... 13
Chapter 1: Introduction ... 14
1.1 Project Title ... 14
1.2 Project Background ... 14
1.3 Project Objectives ... 14
1.4 Problem Statement ... 15
1.5 Project Scope ... 15
1.6 Project Gantt Chart ... 16
Chapter 2: Literature Review ... 17
Chapter 3: Methodology ... 20
3.1 Theoretical Background ... 20
3.2 Materials and Methods ... 23
3.3 Extraction of raw stress-strain data ... 25
3.4 Simulation ... 29
3.4.1 SolidWorks ... 29
3.4.1.1 3D CAD model creation ... 29
3.4.1.2 Data input ... 30
3.4.1.3 Finite element simulation ... 31
3.4.1.4 Simulated result ... 32
3.4.2 Abaqus ... 35
3.4.2.1 3D CAD model creation ... 35
3.4.2.2 Data input ... 36
3.4.2.3 Finite element simulation ... 37
3.4.2.4 Simulated result ... 39
3.4.3 Autodesk NASTRAN ... 43
3.4.3.1 3D CAD model creation ... 43
3.4.3.2 Data input ... 44
3.4.3.3 Finite element simulation ... 45
3.4.3.4 Simulated result ... 46

3.5 Experimental test 50
3.5.1 Test procedures 52
3.5.2 The first test 52
3.5.3 The second test 53
3.5.4 The third test 56
Chapter 4: Result and Discussion 58
4.1 Simulation comparison 58
4.2 Data from engineering graphs 60
4.2.1 Maximum tensile force 60
4.2.2 Maximum tensile force 61
4.2.3 ABS Vs PLA's result discussion 62
Chapter 5: Conclusion and Recommendation 64
References 65
Appendices A-H 67
Addendum 75

List of Tables

Table 1.1 Project Gantt Chart 16
Table 3.1 Data extracted from ABS and PLA stress-strain curves 28
Table 3.2 SolidWorks' simulated data comparison of ABS and PLA 32
Table 3.3 Abaqus' simulated data comparison of ABS and PLA 39
Table 3.4 Autodesk NASTRAN'S simulated data comparison of ABS and PLA 46
Table 3.5 Test specimen's related information 51
Table 4.1 Simulation comparison chart 58

List of Figures

Figure 3.1 Tensile specimen of low-carbon steel 21
Figure 3.2 Test specimen's dimensions in mm........ 23
Figure 3.3 Tensile test specimens 24
Figure 3.4 Stress-strain diagrams of ABS and PLA 24
Figure 3.5 Finding Young's modulus of ABS 25
Figure 3.6 Finding Yield stress and strain of ABS 26
Figure 3.7 Engineering and True Stress-Strain Curve of ABS 27
Figure 3.8 Engineering and True Stress-Strain Curve of PLA 28
Figure 3.9 SolidWorks' 3D CAD model 30
Figure 3.10 SolidWorks' Boundary conditions 31
Figure 3.11 SolidWorks' ABS' simulated plots 33
Figure 3.12 SolidWorks' PLA's simulated plots........ 34
Figure 3.13 Abaqus' 3D CAD model 35
Figure 3.14 True stress Vs Plastic strain of ABS........ 36
Figure 3.15 Abaqus' Tensile force's amplitude with time/frequency 38
Figure 3.16 Abaqus' Boundary conditions 38
Figure 3.17 Abaqus' Meshed conditions 38
Figure 3.18 Abaqus' Comparison of Stress-Strain Curves of ABS........ 40
Figure 3.19 Abaqus' Comparison of Stress-Strain Curves of PLA 40
Figure 3.20 Abaqus' ABS' simulated plots........ 41
Figure 3.21 Abaqus' PLA's simulated plots........ 42
Figure 3.22 Autodesk Inventor's 3D CAD model 43
Figure 3.23 Autodesk NASTRAN's ABS'input data........ 44
Figure 3.24 Autodesk NASTRAN's meshed condition........ 45
Figure 3.25 Autodesk NASTRAN's boundary conditions 46
Figure 3.26 Autodesk NASTRAN's ABS' simulated von Mises stress........ 47
Figure 3.27 Autodesk NASTRAN's ABS' simulated von Mises strain 47
Figure 3.28 Autodesk NASTRAN's ABS' simulated Displacement 48
Figure 3.29 Autodesk NASTRAN's PLA's simulated von Mises stress........ 48
Figure 3.30 Autodesk NASTRAN's PLA's simulated von Mises strain........ 49

Figure 3.31 Autodesk NASTRAN's PLA's simulated Displacement 49
Figure 3.32 Testometric's universal tensile machine 50
Figure 3.33 Experimental test's primary CAD model in mm... 51
Figure 3.34 Experimental test's secondary CAD model in mm 54
Figure 3.35 First and Second tests' results 54
Figure 3.36 Comparison of ABS' stress-strain diagram (second test).. 55
Figure 3.37 Comparison of PLA's stress-strain diagram (second test) 55
Figure 3.38 Third test's result.. 56
Figure 3.39 Comparison of PLA's stress-strain diagram (third test).. 57
Figure 3.40 Comparison of PLA+'s stress-strain diagram (third test) 57
Figure 4.1 Overall Comparison of ABS and PLA 62

List of Appendices

Appendix A: ABS'data from Baca Lopez (2020).. 67
Appendix B: Transformed true ABS'data from Baca Lopez (2020)... 68
Appendix C: PLA's data from Baca Lopez (2020) 69
Appendix D: Transformed true PLA's data from Baca Lopez (2020) 70
Appendix E: Experimental ABS' data from the second test... 71
Appendix F: Experimental PLA's data from the second test 72
Appendix G: Experimental PLA's data from the third test 73
Appendix H: Experimental PLA+'s data from the third test 74

List of Addendum

Completed Project Proposal Form 75

Completed Ethics Clearance Certificate 77

Completed Interim Report Form... 78

Completed Progress Monitoring Form 79

Completed Progress Summary Form 81

List of Achievements 84

Acknowledgements

I am grateful and would like to express my sincere gratitude to my supervisor Dr Thu Han Tun for his invaluable support in making this project possible. I am truly grateful for his genuine vision about my report writing, his tolerance of my naive mistakes, and his commitment to my experimental tests.

My sincere thanks go to all the staff of the Marine Mechanical Engineering Department, Myanmar Maritime University, Thanlyin, who helped me set up with the practical experiment and data collection and every staff member of Chindwin Technological University, who provided me information and guidelines.

The last but not the least, if it were not my parents, I could not achieve my goals of becoming a successful mechanical engineer and could not properly describe my appreciation for their devotion, support and faith in me.

Abstract

The application of thermoplastic materials such as acrylonitrile butadiene styrene (ABS) and polylactic acid (PLA), produced from Fused deposition modelling (FDM) technology is abundant in various modern industries in a way that investigating their mechanical behaviors by the usage of Finite Element Analysis (FEA) software(s) without damaging the test specimens should be promoted. This project aims to compare the simulation capability of three software(s); SolidWorks, Abaqus and Autodesk NASTRAN on the ASTM D638 Type 1 standard tensile test specimens of ABS and PLA, by comparing with the reference paper, Baca Lopez (2020) and three experimental tests.

The study found that Abaqus was the most powerful software which could predict almost exactly the same as the real tensile test's results, followed by SolidWorks and Autodesk NASTRAN. Moreover, the experimental test exhibited nearly the same result as that of the reference. This project can be a guide line for beginner and amateur design engineers, who would like to perform finite element analysis on tensile tests. Future work will consider conducting simulation with the software(s) in "non-linear dynamic (explicit)" condition.

Keywords: ABS; PLA; finite element method; SolidWorks; Abaqus; Autodesk NASTRAN; ASTM D638 Type1

Chapter 1: Introduction

1.1 Project Title

My project title is "Comparison of experimental result and FEM analysis of ABS and PLA on ASTM D638". The comparison of mechanical behaviours of each material was done on three software(s) which can perform non-linear finite element analysis and the most efficient software with the result as close as the data from the experiment and the reference paper was recommended.

1.2 Project Background

In recent years, engineers have been doing massive research on Finite Element Method in simulating the mechanical behaviours of materials under conditions nearly the same as real environments. By using certain FEA modelling software(s), engineers can save time, money and energy in investigating the mechanical properties of desired materials. Thermoplastics, such as PLA and ABS have dominated the 3D printing market for a century and it is important to know these materials' tensile properties according to ASTM standards. Herein, well-known simulation software(s) such as Abaqus, SolidWorks and Autodesk NASTRAN were used to simulate the plastics' behaviours on ASTM D638 Type 1 alongside the experimental data, the result of which can be useful for designers and manufacturers.

1.3 Project Objectives

This project's objectives can be divided into five categories;

1. To study the mechanical properties of ABS and PLA by literature review
2. To learn simulation techniques of FEA software(s)
3. To analyse the tensile test simulation on FEA software(s)
4. To evaluate the simulation results with the experiment

5. To declare the software(s)'s relative error percentage

1.4 Problem Statement

In general, this project was carried out based on the following problems;

1. In previous papers, the inputs into the FEA software(s) were assumed to be engineering stress and strain data, which are incorrect beyond the elastic region. A clear conversion from the former to the true data was lacked in those papers, and the transforming equations should be declared.
2. Abaqus is named as one of the three renowned FEA software(s), apart from ANSYS and COMSOL, but SolidWorks, which is best known for 3D modelling, is also needed to find out it can be just as capable of finite element analysis.
3. Autodesk Inventor, being the top 3D software(s), integrated with its own module, NASTRAN is claimed to be capable of the analysis, but is rarely mentioned in previous papers. So, the software needs to be found out its capability.

1.5 Project Scope

In this project, the data obtained from Abaqus©, SolidWorks © and Autodesk NASTRAN© on PLA and ABS under the ASTM D638 tensile test was simulated;

1. Literature review
   - To study the previous papers, the mechanical behaviours of PLA and ABS plastics and consider the plausible FEA software
2. 3D modelling
   - To model the 3D part, dimensions of which were referred from the ASTM standard, to be simulated in each software
3. FEA simulation

- To perform finite element analysis on the 3D model with aforementioned software(s)

4. Experiment

- A practical experimental test was also performed to compare and contrast the data from the reference paper and simulation software(s)

5. Evaluation

- The data obtained from simulation and experiments were compared to declare the error percentage of each software

1.6 Project Gantt Chart

Table SEQ Figure * ARABIC 1.1 Project Gantt Chart

	September				October				November				December			
Tasks\Week	1	2	3	4	1	2	3	4	1	2	3	4	1	2	3	4
Literature review	■	■	■	■	■											
3d modelling		■														
SolidWorks simulation						■	■	■								
Abaqus simulation								■	■	■						
Autodesk NASTRAN simulation										■	■	■				
Experimental test						■								■	■	
Evaluation													■	■		
Preparing reports														■	■	■

Chapter 2: Literature Review

The commercial use of 3D printing processes began in 1990s. On early days, the technique was initially used as rapid prototyping, but nowadays, it has become addictive manufacturing (AM) which is a powerful tool used widely in various fields; biomedical, aerospace and manufacturing (Baca Lopez, 2020) in many countries (Shahrubudin, 2019). Due to its low operational and manufacturing cost, fast processing time, good quality product and ease of control, AM has recently gained quite a renowned popularity compared to other manufacturing methods. Among the different techniques of AM, fused deposition modelling (FDM) is preferred not only by researchers and engineers, but also by commercial manufacturers and amateurs as well.

Among the various types of thermoplastic filaments used in FDM, Polylactic Acid (PLA) and Acrylonitrile Butadiene styrene (ABS) are primary materials used in various processes (Japrin, 2021). PLA is a top candidate for consumer and biomedical fields due to its superb mechanical and physical properties, good biocompatibility, sustainability, and ease of manufacture (Farah et al., 2016). The second material, ABS, is well-known for its excellent heat resistance, low thermal conductivity and toughness (Baca Lopez, 2020), primarily found in children's toy sets of LEGOS©. In addition to their mentioned properties, the plastics' high strength to weight ratio and high processibility make PLA and ABS the leading thermoplastics used in multiple applications. Therefore, it is important to understand the plastics' mechanical properties, especially tensile behaviors whether they can be suited for specific applications.

One of the solutions is Finite Element Method (FEM). Since its creation in mid-20th century, the simulation technique has been widely used in many industries (Rovalance, 2001; Magomedov, 2020). Unlike an actual uniaxial tensile, which can provide information about the plastics' mechanical properties of stress–strain relationship (Joun, 2007) by destroying the test specimen every time the test is taken, which can be very expensive for some rare materials, FEM can virtually simulate the load conditions, boundary conditions just as the real test to find out the elastic-plastic characteristics of materials (Durbacă, 2018). This can be done by breaking down a given CAD model structure into smaller interconnected sections called nodes, which possess the local domain properties. Upon receiving the external stimuli, the numerical values of these nodes are compiled to produce the global function, which, in the case of the tensile test, is the stress-strain relationship upon the input force condition (Nirbhay et al., 2014; Żur, 2019). The equation explained by Rovalance (2001) is the fundamental of every FEM simulation;

$K_{ij}u_j = f_i$, where u and f are the resultant displacements at j^{th} and external input forces at i^{th} nodes, and K is the global stiffness matrix dependent on the j^{th} displacement on the i^{th} force.

Paul (2021) suggested that due to the FE analysis' capability to help produce eco-friendly and high-quality products in the lowest possible time, more of this analysis on Addictive Manufacturing should be performed. Regarding the previous papers, Garrell et al. (2003) simulated that the stress concentration factor on a Nylon-11 matrix under the tensile test had a variation of less than 1.5% with the real data using ANSYS finite element software(s). Latifi et al. (2014) used Abaqus 6.10 EF to determine the elastic properties of rabbit vocal fold tissue using uniaxial tensile testing. Durbacă (2018) also created the composite sandwich with two glass face sheets and polymeric triangular core using Autodesk Inventor Professional and simulated on ANSYS 14.5 to identify von Mises' stress on the quasi-static tensile test. Azeez (2018) discovered that polycarbonate composite dominates those of polypropylene, high density polyethylene and polyurethane by simulating the ASTM C297/C297M-04 standard tensile test.

Żur (2019) compared and contrasted the finite element analysis data and real-life data of static bending test on PLA specimens from FDM, and the difference was only 1.7% using Siemens NX software(s) with Nastran module. Moreover, Román et al. (2020) and Hussin et al. (2020) all used various types of finite element software(s); ANSYS Workbench and Autodesk Inventor 2016, and each corresponding exhibited less than 10% variation with the respective tensile tests. Alharbi et al. (2020) simulated of uniaxial stress–strain response of 3D-printed polylactic acid by nonlinear finite element analysis using SolidWorks and obtained the error less than 7%. Japrin (2021) used COMSOL Multiphysics to simulate the mechanical performance of a generic model to reduce printing time and amount of infill by 17% by optimizing the custom infill percentage of 25% honeycomb pattern and 75% grid pattern.

However, regardless of the vast amount of FEA software(s) used, it is also required to know the capability of each and every software(s), and how they differ from one another. Magomedov (2020) compared between well-known software(s) capable of FEA analysis abilities; Abaqus and ANSYS (CAE and Mechanical respectively) have identical capabilities, but the only difference they have is the self-contained section and Unit awareness part. Inventor Nastran is also similar with the former two; the only difference of the software(s) is lack of capability of doing Acoustic analysis. SolidWorks is mainly used to make a model or assembly of separate parts and then transfer the structure to Abaqus (CAE), Ansys (Mechanical) or Inventor Nastran, but it can also perform FEM analysis but not as powerful as the formers. A surprising fact is that in the past papers, there were only a few which used SolidWorks and Autodesk NASTRAN as FEM analysis, so it is important to measure their performance in accordance with either top tiers; Abaqus or ANSYS.

Moreover, Petrik (2019) mentioned that engineering stress and strain data from the experimental results should be transformed into true values before inputting into the FEA software(s), which was rarely mentioned in the past papers. Without the conversion, there will be miscalculations in the ultimate strength position and in the plastic region as well, according to Petrik's statement. Therefore, it is also required to know whether including the true stress-strain relationship can be critical for FEM simulation.

Therefore, herein, on top of SolidWorks and Autodesk Inventor being the leading modelling software(s), this project compared and contrasted the FEM data of ASTM D638 Type 1 on PLA and ABS from the aforementioned software(s) with that of Abaqus, the experimental data and the reference paper, by taking account on the true stress-strain relationship.

Chapter 3: Methodology

This project was carried out on two well-known thermoplastics namely, Acrylonitrile Butadiene Styrene (ABS) and Polylactic Acid (PLA) by simulating their mechanical behaviors on the standard ASTM D638 Type 1 tensile test by three FEA software(s); SolidWorks, Abaqus and Autodesk Inventor Nastran and experimental tests. The required data was referred from Baca Lopez (2020), and from here and out, the term “reference paper “will be used instead of it.

3.1 Theoretical Background

This project’s main theoretical applications were mainly based on axial loading and plastic deformation concepts. As $\varepsilon = \delta / L_0$------(2) stated in Beer et al. (2015), in order to obtain the stress-strain diagram of a material, the axial stress and strain of a test specimen under tension was calculated as follows:

$$\sigma = F / A_0 \text{------(1)}$$

where σ is the axial stress, F is the applied tensile force, A_0 is the original cross-sectional area of the specimen, ε is the axial strain, δ is the resulted elongation and L_0 is the original gauge length. Elongation, δ is obtained by subtracting the new length of the specimen upon tension with the original gauge length. Upon the tensile test, the data can be obtained as the tensile force which can be converted to the stress by dividing with the original cross-sectional area, and the elongation, which can be converted to strain. The stress-strain diagram can be plotted by tensile stress in the y-axis and strain in the x-axis. Note that since the output stress are derived from the original cross-sectional area, the diagram’s data are engineering stresses and engineering strains.

This project's focused materials; ABS and PLA are ductile materials, so their stress-strain diagrams were expected to be similar to that low-carbon steel; the tensile behaviors of which were shown in Fig. 3.1.

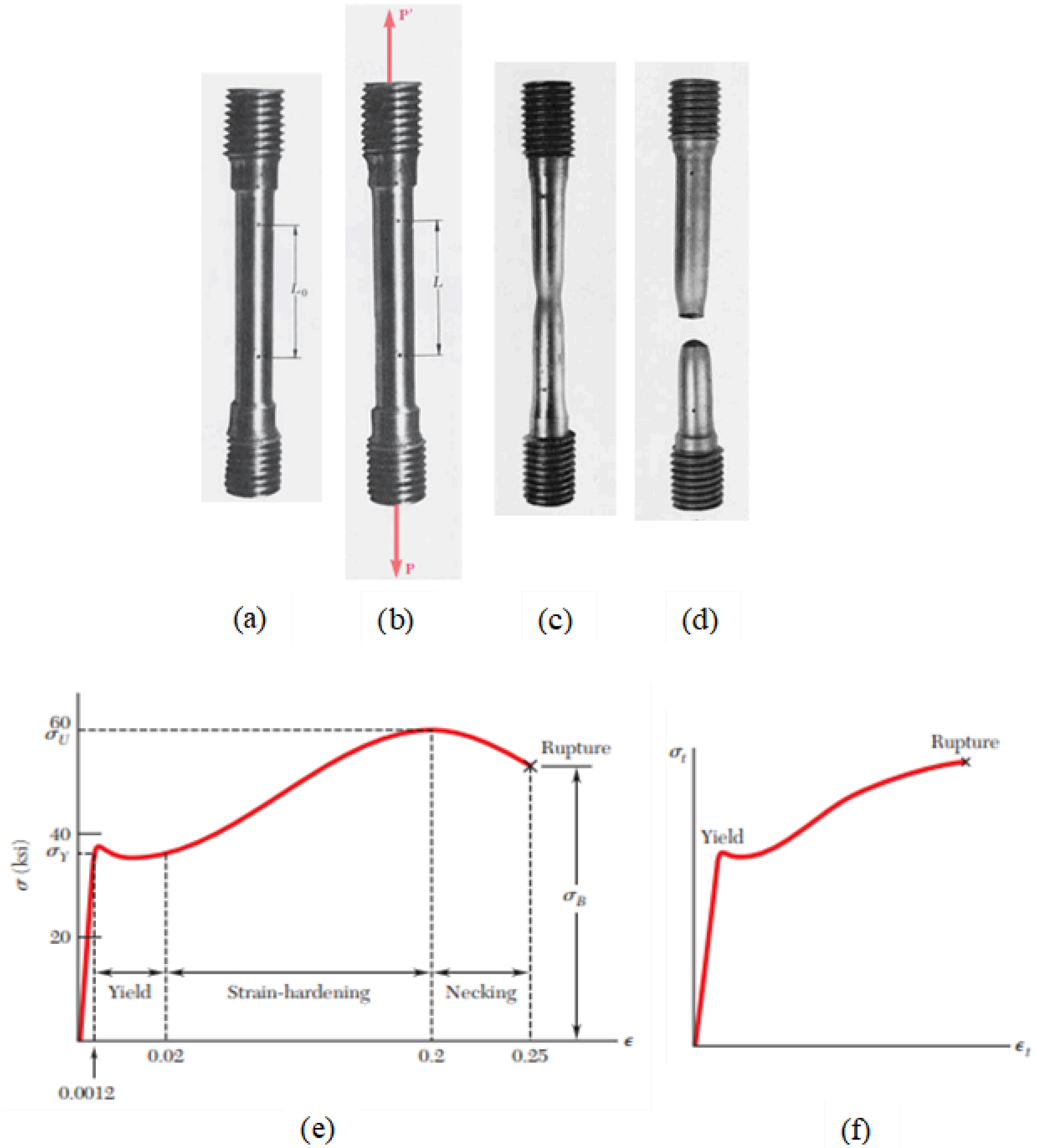


Figure 3.1 Tensile specimen of low-carbon steel

(a) Original tensile specimen (b) Specimen on tensile force (c) Specimen under necking (d) Fractured specimen (e) Engineering stress-strain diagram (f) True stress-strain diagram

Note: Data referred from Beer et al. (2015)

As seen in Fig. 3.1(e), as the respective tensile test is applied on the specimen, the tensile stress keeps increasing linearly until the yield point, the beyond of which the specimen cannot retract back to the original length. This means that the tensile stress until the yield point can be found by;

$\sigma = E\varepsilon$ where σ and ε are the elastic stress and strain, and E is the elastic modulus of a test specimen. It is also noteworthy that there are two yield points in the tensile material; the first point is the one at which the tensile stress stops increasing linearly and the second point is when the material will start to behave plastically, and the area between those points is called the yield region. Amongst two yield points, the latter point's stress is mainly used for calculation, and the yield strength's location can be found out by the 0.2% offset method for ductile materials (see Fig. 3.1(e)).

Following the yield region, there is the strain-hardening region where the specimen's stress increases in a curved pattern rather than a straight line. Here, the specimen can experience a large deformation even when a small amount of force is applied. Upon the applied force, the tensile stress keeps increasing before it reaches a peak, which is called ultimate strength. This is the maximum strength that the material can tolerate before breaking. After that, the specimen will show a decrease in the cross-sectional area (see Fig. 3.1(c)). Such phenomenon is called necking, and the area between the ultimate stress and the rupture point where the specimen breaks down completely is called the necking region (see Fig. 3.1(d)).

The necking phenomenon can be found in homogeneous materials where they undergo transverse reduction in cross-sectional area but elongate axially when applied with a uniaxial tensile force. The degree of such deformation in different materials when compared is called Poisson's ratio; the greater the value, the larger the deformation the material will be experienced. Poisson's ratio can be found out by;

$$\nu = -\frac{\text{lateral strain}}{\text{axial strain}} = -\frac{\varepsilon_x}{\varepsilon_y} = -\frac{\varepsilon_z}{\varepsilon_y}$$

where ν is Poisson' ratio, ε_y is the axial strain where the tensile force is applied, while ε_x and ε_z are the transverse strains. Since the cross-sectional area of the specimen becomes smaller, it is more accurate to deal with true values, which take account on simultaneous cross-sectional areas rather than their engineering counterparts. Although true stresses and strains are not occasionally used for daily engineering problems, they are mostly used in Finite Element simulation for accuracy. Moreover, the true stress-strain diagram accurately reflects the tensile behavior of the specimen. Due to their decreasing area values, instead of decreases in stress values beyond the ultimate point, the material's tensile stress will keep rising until the rapture (see Fig. 3.1(f)).

3.2 Materials and Methods

The simulation was carried out on two types of thermoplastics; ABS and PLA. The material properties of these plastics referred from the reference papers, Baca Lopez (2020) and Alharbi et al. (2020), will be inputted into the simulation software(s). The CAD model for simulation was according to the ASTM D638 Type 1 standard; tensile force was 5000N, strain rate was 2mms^{-1} (see Fig.3.2). Data from the reference stress-strain curve data was extracted using WebPlotDigitizer©. The distance between each point on the plotting software(s) was 5 pixels. These raw data were stored in an Excel file. The tested specimens and the experimental stress-strain data from the reference paper, Baca Lopez (2020) were shown in Fig. 3.3 and 3.4.

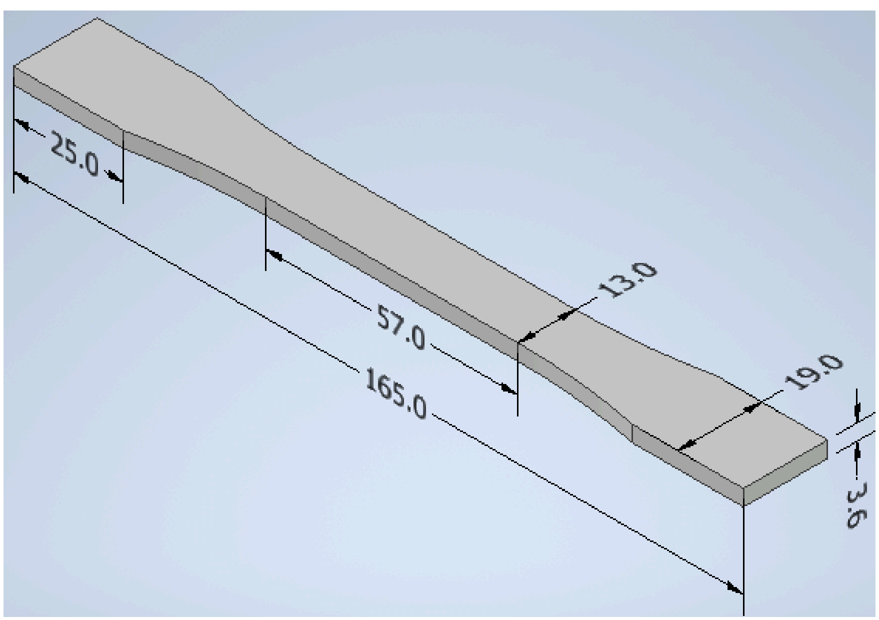


Figure 3.2 Test specimen's dimensions in mm

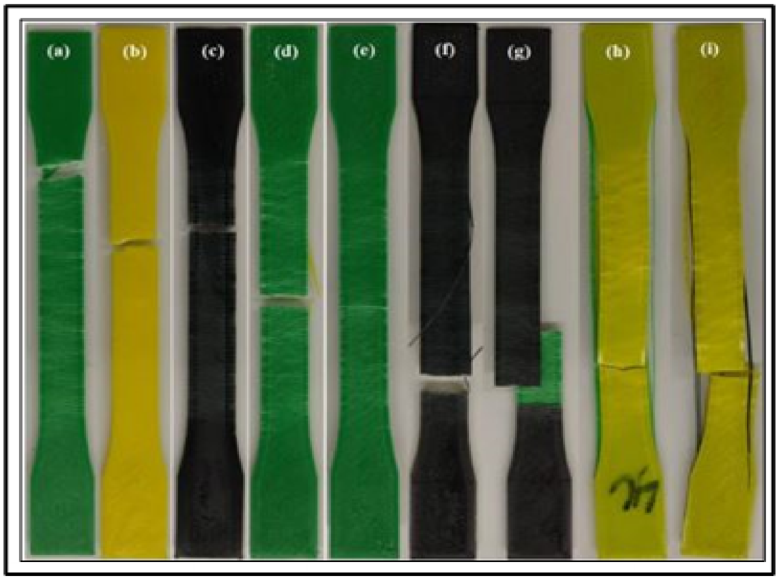


Figure 3.3 Tensile test specimens

Note: Data referred from Baca Lopez (2020)

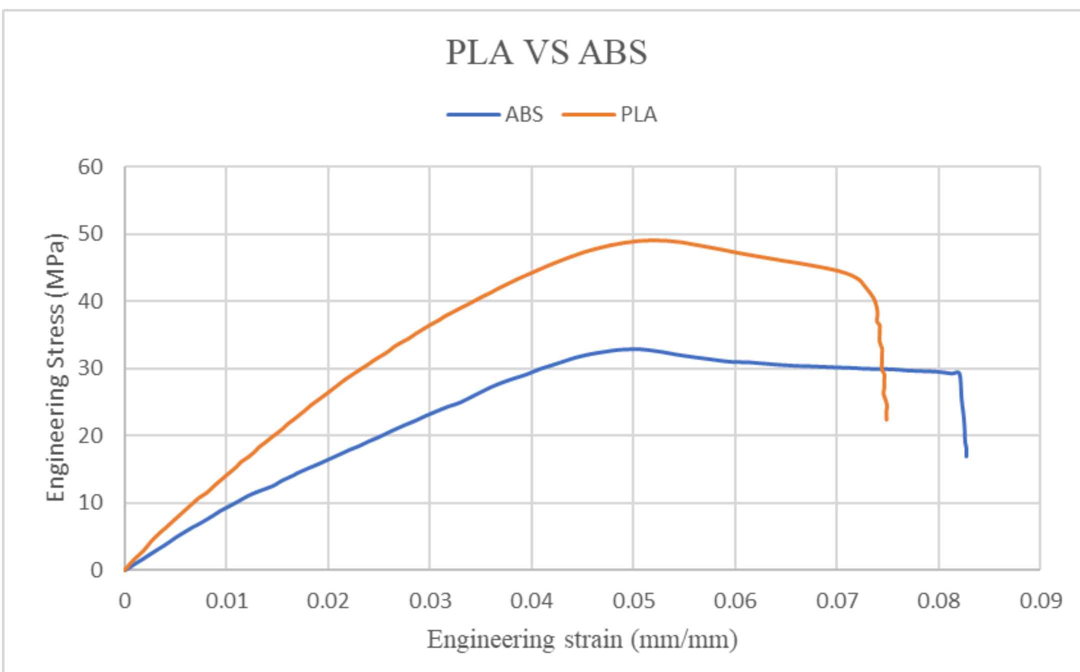


Figure 3.4 Stress-strain diagrams of ABS and PLA

Note: Data referred from Baca Lopez (2020)

3.3 Extraction of raw stress-strain data

The extraction method was universal for both plastics. The following was done on ABS. After the output stress-strain data points from the WebPlotDigitizer© were obtained and stored in the Excel file, a scatter diagram was plotted (see Fig. 3.4). Prior to finding Young's modulus, a set of first nine to eleven strains and stresses were picked to plot linear diagrams, from which the linear equations, in the form of y = mx + c, were calculated. R- squared value of linear regression for each set of points was compared, and found that up to first ten points produced the value of 0.999, which was the closest number to 1, compared to 0.998 for nine and 0.997 for eleven. The slope of this set was taken as Young's modulus, which was 967.45 MPa (see Fig. 3.5).

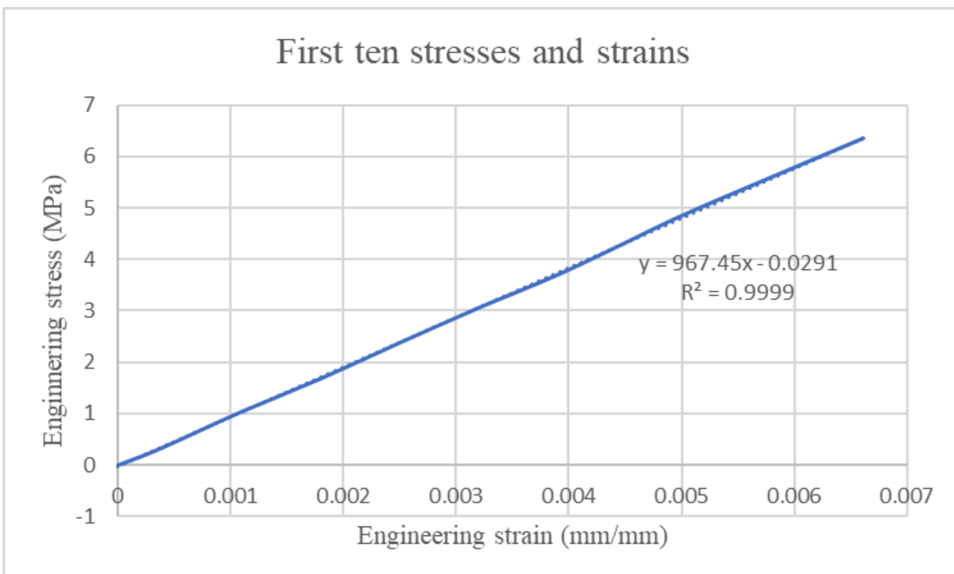


Figure 3.5 Finding Young's modulus of ABS

Following this, in order to find yield strain, the given strains were added with 0.002 as per 0.2% offset yield strength method. The yield stresses were found out by multiplying the Young's modulus with original strains, so that a line, which parallelly intersected the original curve, was obtained. For instance, for the second point ($\varepsilon_e = 0.000384$, $\sigma_e = 0.333088$), the yield values were;

$$\varepsilon_y = 0.002 + \varepsilon_e = 0.002 + 0.000384 = 0.002384$$

$$\sigma_y = E.\varepsilon_e = 967.45 \times 0.000384 = 0.371832\ MPa$$

After solving the two equations (one polynomial, one linear) of two lines, yield strain (0.01684) was obtained. By inserting that value in the original curve's equation, yield stress (14.359 MPa) was obtained (see Fig. 3.6). Note that this finding was just to find the yield position and would not be included in simulation.

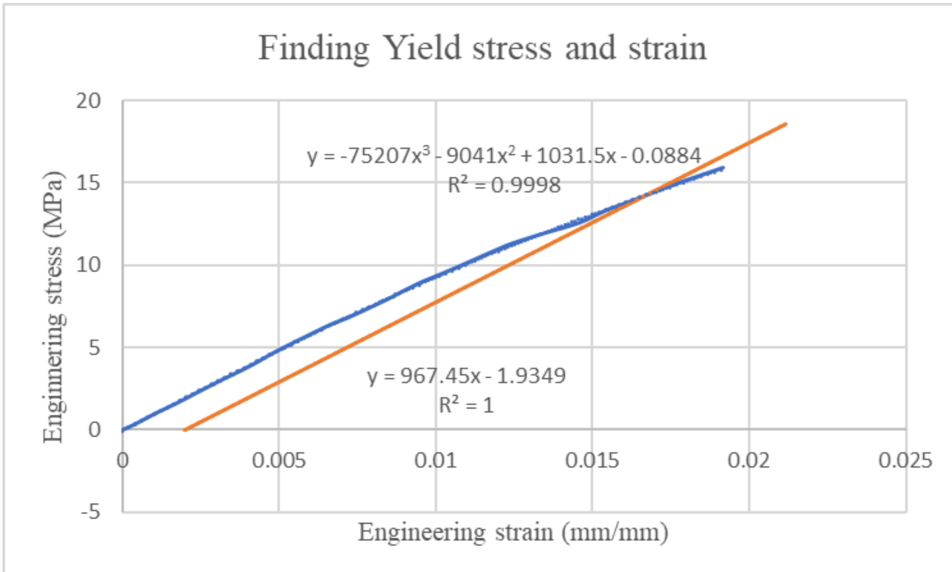


Figure 3.6 Finding Yield stress and strain of ABS

The following equations by Petrik (2019) were used to transform engineering values into true values;

where ε_t and σ_t are true values of strain and stress, while ε_e and σ_e are engineering values of strain and stress. Note that this conversion is only valid until necking, after which to the point of rupture, the equations cannot be used. For instance, the true values of the second point (ε_e = 0.000384, σ_e = 0.333088) and the ultimate point (ε_e = 0.050001, σ_e = 32.88571) of ABS can be obtained by;

$$\sigma_{f,t} = \sigma_{u,t} \cdot \varepsilon_{f,t} + \sigma_{u,t}(1 - \varepsilon_{u,t}) \text{------(5)}$$

The second point;

$$\varepsilon_t = \ln(1 + 0.000384) = 0.000384, \quad \sigma_t = 0.333088(0.000384 + 1) = 0.333216 \text{ MPa}$$

The ultimate point;

$$\varepsilon_{u,t} = \ln(1 + 0.050001) = 0.048791, \quad \sigma_{u,t} = 32.88571(0.050001 + 1) = 34.5300 \text{ MPa}$$

The locations of ultimate true and engineering stresses were marked as yellow and green dots, and were plotted in Fig. 3.7 and 3.8. Since the true stress will always increase near linearly beyond the ultimate stress, drawing a straight line to connect the rupture point is the simple method to obtain the total stress-strain curve. The strains were the same as the true strain transformation, but the stresses from the ultimate until the fracture point were calculated by;

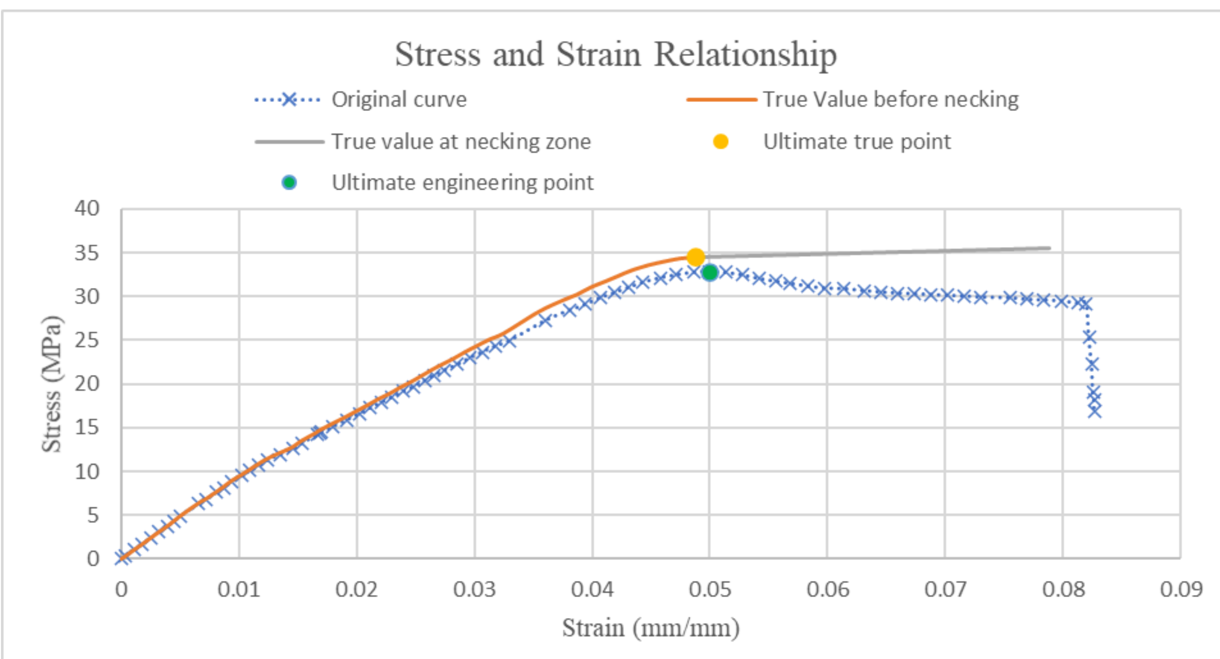
$$\varepsilon_t = \ln(1 + \varepsilon_e) \text{------(3)}$$

$$\sigma_t = \sigma_e(\varepsilon_e + 1) \text{ ------(4)}$$

where $\varepsilon_{f,t}$ and $\sigma_{f,t}$ are true values of strain and stress from ultimate to fracture at a specific location, and $\sigma_{u,t}$ and $\varepsilon_{u,t}$ is the true ultimate stress and strain. The true strains of the required points were already calculated by Eqn. (3), and substituting the ultimate point's true values in the Eqn. (5) could find out respective points. For instance, for the rupture point ($\varepsilon_{f,t}$ = 0.079404);

The same procedures were done for PLA, and the curves for PLA and ABS were plotted in Fig. 3.7 and Fig. 3.8, and the extracted data from the curves required for simulation were listed in Table 2.

$$[illegible]_{[illegible],[illegible]} = 34.5300\ [illegible]\ 0.079404 + 34.5300\ [illegible]\ 1 - 0.333216\ [illegible] = 35.5871 \text{ MPa}$$



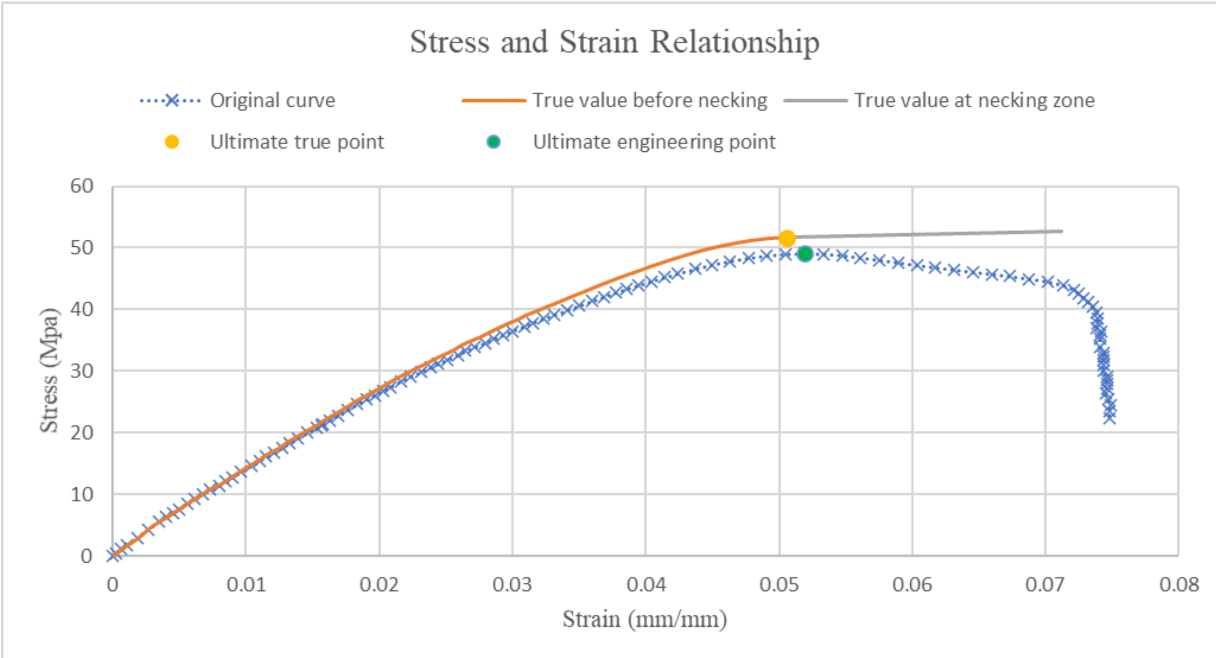


Figure 3.8 Engineering and True Stress-Strain Curve of PLA

Table 3.1 Data extracted from ABS and PLA stress-strain curves

	ABS	PLA
Density (kg/m^3)	1030	1240
Poisson ratio	0.35	0.36
Elastic Modulus (MPa)	967.45	1555
Yield Stress (MPa)	14.359	21.22575
Yield Strain (mm/mm)	0.01684	0.01565
True Yield Stress (MPa)	14.60081	21.55793
True Yield Strain (mm/mm)	0.0167	0.015529
Ultimate Stress (MPa)	32.88571	49.09914422
Ultimate Strain (mm/mm)	0.050001	0.051911718
True Ultimate Stress (MPa)	34.53003	51.64797
True Ultimate Strain (mm/mm)	0.048791	0.050609

3.4 Simulation

Every FEM software(s) followed four steps; 3D CAD model creation, data input, finite element simulation, and simulated result. This project conducted simulation on SolidWorks, Abaqus and Autodesk Inventor NASTRAN.

3.4.1 SolidWorks

It is a software, developed by Dassault Systèmes, which specializes in commercial 3D modelling, but it also offers numerical simulation of both linear and non-linear equations. This project used 2018 version, and the simulation was tested with non-linear static (force control, which showed the result when the given amount of force was reached) and (displacement control which displayed the result data within the input displacement). Moreover, it is worth knowing that Alharbi et al. (2020) performed the uniaxial tensile test using SolidWorks and mentioned that the software(s) can perform until the ultimate stress.

3.4.1.1 3D CAD model creation

The creation of 3D model was very simple and convenient as the software(s) is intended for the best human interface design. Starting with a 2D sketch with dimensions from Fig. 3.2 and then extruding to 3.6 mm thickness made the final model to be simulated. However, there were few alterations from previous papers; in order to capture the real-life scenarios of the test, instead of fixing on one end and pulling from the other, the model's surface, where the tensile machine would clamp, was sliced into sub-surfaces on both top and bottom sides. Also, the surface of the opposite end, on which the tensile traction force would act on, was also sliced. The newly found middle line's vertex was used to move a preferred displacement in the displacement control method (see Fig. 3.9). The slicing function would not affect the material's properties.

Figure 3.9 SolidWorks' 3D CAD model

3.4.1.2 Data input

Firstly, in SolidWorks' simulation, since the test was quasi-static (the load was increasing steadily but slow enough to be considered as static), non-linear simulations of both static (loads are applied gradually without changing overtime) with force control and displacement control were chosen.

The conversion of engineering stress-strain curve was needed to transform to its true value due to the fact that the former was based on the original cross-sectional area, while the latter was based on the instantaneous areas, and was appropriate for the FEM analysis, especially dealing with large strains. For that, the true values from Fig. 3.7 and 3.8 were used in the stress-strain curve and the material data from Table. 3.1 were used as input data, and von Mises plasticity failure criterion was chosen.

3.4.1.3 Finite element simulation

For the boundary conditions, sub-surfaces of one end were fixed while the others were experienced a surface tensile traction force of 5000 N. Depending on the material, the model was stretched 4.71 mm for ABS and 4.16 mm for PLA, which were the tensile elongation at break as mentioned in Baca Lopez (2020). Mesh element's parameters were referred from Alharbi et al. (2020); curvature-based 1.85mm sized mesh elements with a triangular pattern with the element size growth ratio of 1.5 (see Fig. 3.10). Two methods of force and displacement control were performed to select the optimal result.

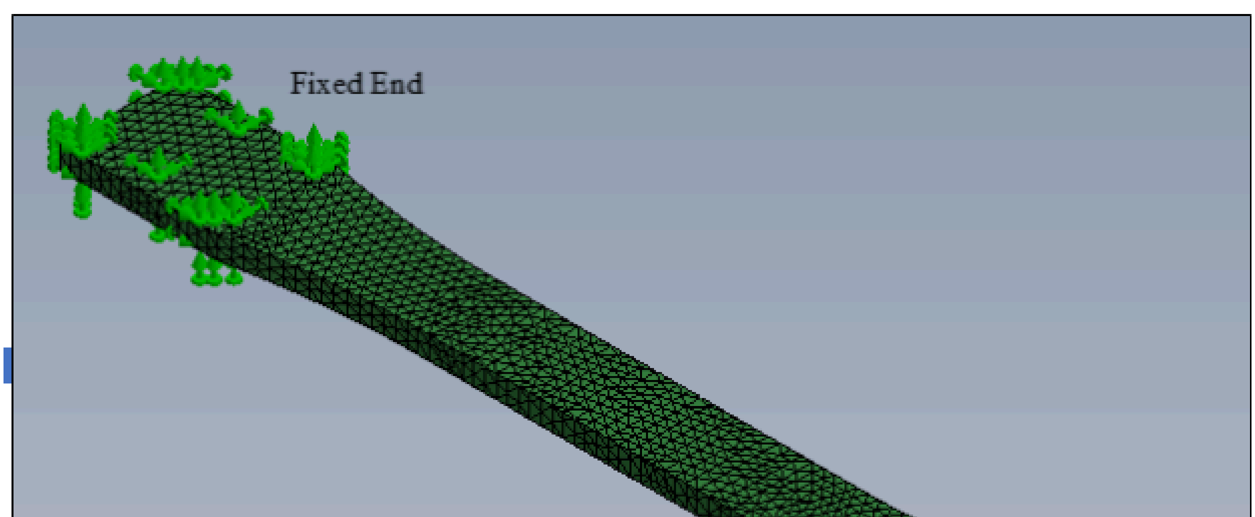


Figure 3.10 SolidWorks' Boundary conditions

3.4.1.4 Simulated result

The simulated results from non-linear static (displacement control) and non-linear static (force control) were compared in Table. 3.2. The yield stresses of all experiments displayed the same, but the differences were in ultimate stress and strain.

Table 3.2 SolidWorks' simulated data comparison of ABS and PLA

		ABS	PLA	ABS	PLA
		FEA result		Error %	
Non-linear static (Displacement control)	Ulimate stress (Mpa)	31.754	44.061	8.039	14.68977
	Ultimate strain	0.041	0.036	15.96811	28.98
Non-linear static (Force control)	Ulimate stress (Mpa)	33.296	46.301	3.574	10.35272
	Ultimate strain	0.045	0.038	7.76988	25.03452

In the above table, positive error percentages showed that the calculated true value was smaller than the FEA result. For instance, for non-linear static (force control), the error percentages of ABS can be found by;

$$[illegible] (\%) = \frac{[illegible] - [illegible]}{[illegible]} \times 100\% \text{ ------(6)}$$

By substituting ABS' ultimate true stress = 34.53003 MPa, ultimate true strain = 0.048791;

$$[illegible] (\%) = \frac{34.53003 - 33.296}{34.53003} \times 100\% = 3.574\%$$

$$[illegible] (\%) = \frac{0.048791 - 0.045}{0.048791} \times 100\% = 7.76988\%$$

It was also noteworthy that the error percentages of the ultimate stresses were lower than respective strains. By taking account on stress and strain error percentages, non-linear static (force control) method was chosen as the optimal technique as it showed relatively less error % than its displacement counterpart. The graphical results of the optimal method were shown in Fig. 3.11 and 3.12. By taking reference from Alharbi el al. (2020), it was speculated that fracture would occur at the maximum von Mises stress' locations where stress concentration was accumulated; from the fixed end, 112.8 mm for ABS and 139.6 mm for PLA.

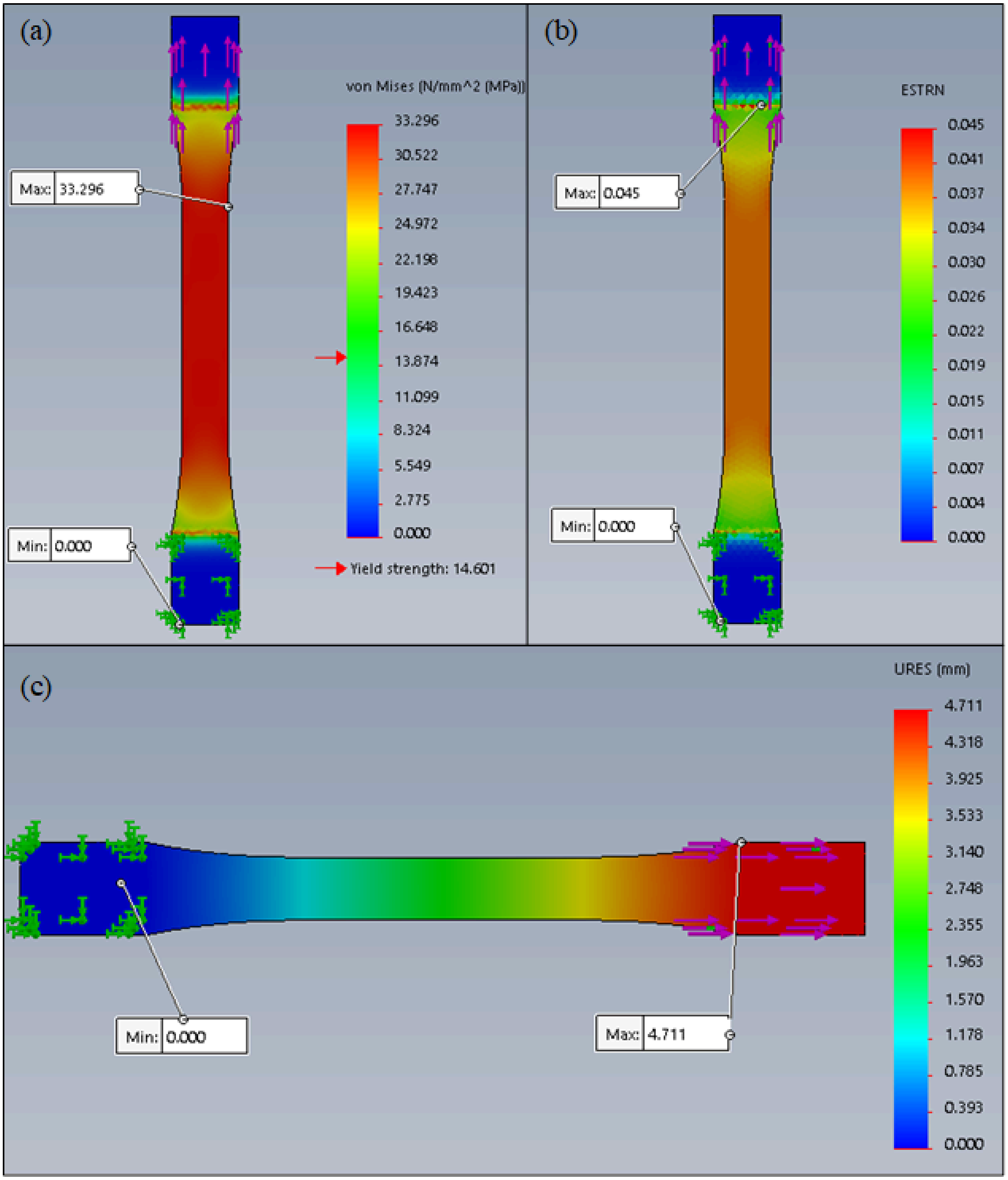


Figure 3.11 SolidWorks' ABS' simulated plots of (a) von Mises stress (b) strain (c) displacement

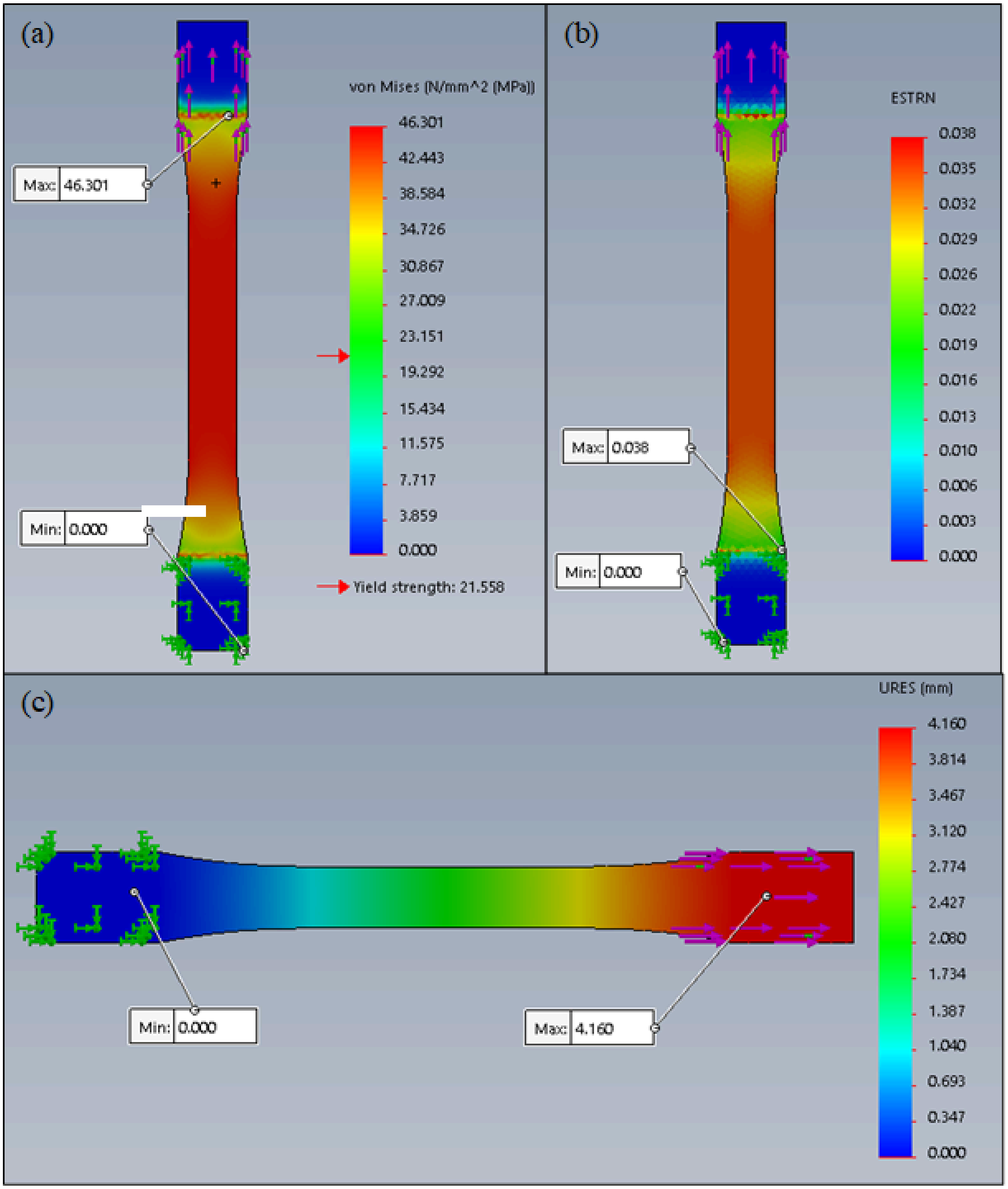


Figure 3.12 SolidWorks' PLA's simulated plots of (a) von Mises stress (b) strain (c) displacement

3.4.2 Abaqus

It is one of the FEM specialized software(s), apart from Ansys and COMSOL Multiphysics, which was used by the majority of researchers in various engineering fields. The software(s) interface is simple and user-friendly as it does display multiple tools under colorful icons and procedures can be done step by step easily by following the sequence of the module icon. While simple models can be drawn inside its interface, however, complex models from any other 3D software(s) can be inputted for further analysis due to the fact that its CAD drawing commands are not as convenient as SolidWorks or Autodesk Inventor. The current version used was 2017.

3.4.2.1 3D CAD model creation

Much like SolidWorks, the procedure started by the 2D sketch and the sketch was extruded 3.6mm for thickness. Similarly, the surfaces of the ends where one end was needed to be fixed and the other, on which the tensile force of 5000 N was subjected, were sliced. In addition, a concentrated reference point RP was marked on the top surface (see Fig. 3.13).

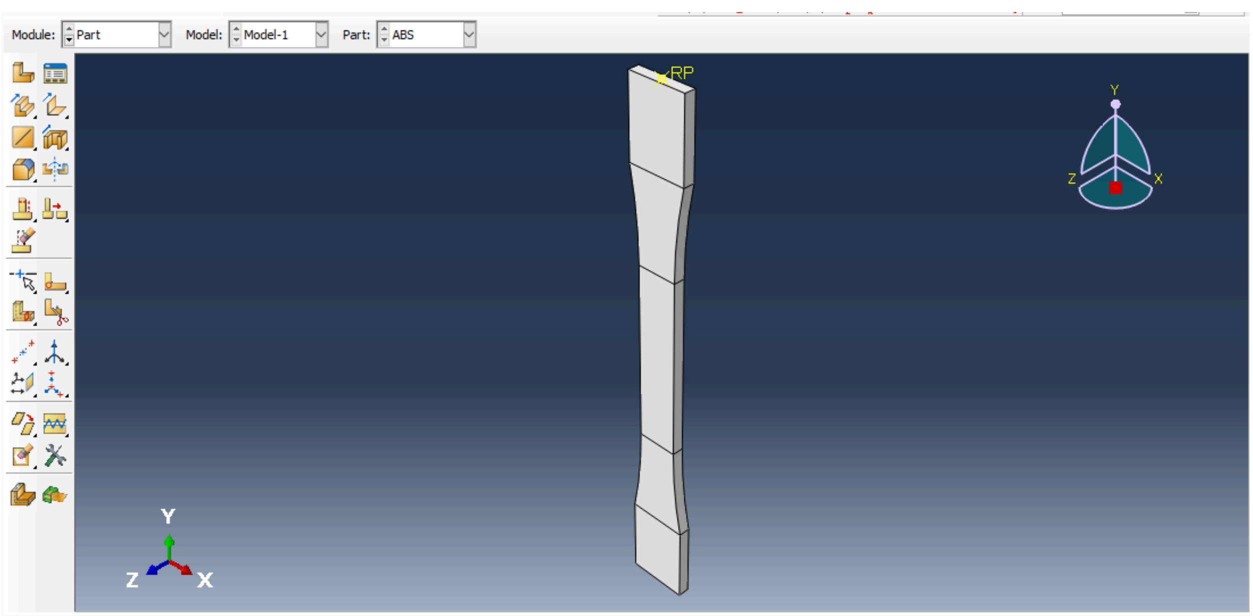


Figure 3.13 Abaqus' 3D CAD model

3.4.2.2 Data input

By changing to the subsequent module's item, which was "Property", materials' data from Table. 3, namely; density, Young's modulus and Poisson's ratio were inputted (1.03e-9 tons/mm^3, 967.45 MPa, 0.36 for ABS and 1.24e-9 tons/mm^3, 1555 MPa, 0.35 for PLA). In determining the material's plasticity, true stress values from yield point to the fracture from Fig. 3.7 and 3.8, were inputted. However, for strain values, plastic strains were needed by inserting true strain values, starting from the yield point, in the equation;

$$\varepsilon_{pl} = \varepsilon_t - (\sigma_t / E) \text{------(7)}$$

where ε_{pl} is the plastic strain, ε_t and σ_t are the true strain and stress, and E is the Young's modulus For instance, for ABS' second point after the yield (ε_t = 0.016904, σ_t = 14.7609), the plastic strain would be;

$$\varepsilon_{pl} = \varepsilon_t - \left(\frac{\sigma_t}{E}\right) = 0.016904 - \left(\frac{14.7609}{967.45}\right) = 0.001647$$

Upon data entry, the first point's strain, which was the yield point, was set to zero. True strains beyond the yield to the rupture were transformed using Eqn. (7), and respective true stresses were inputted without much change. Data required were plotted in Fig. 3.14. Afterwards, the whole CAD model was selected as a solid, homogeneous section.

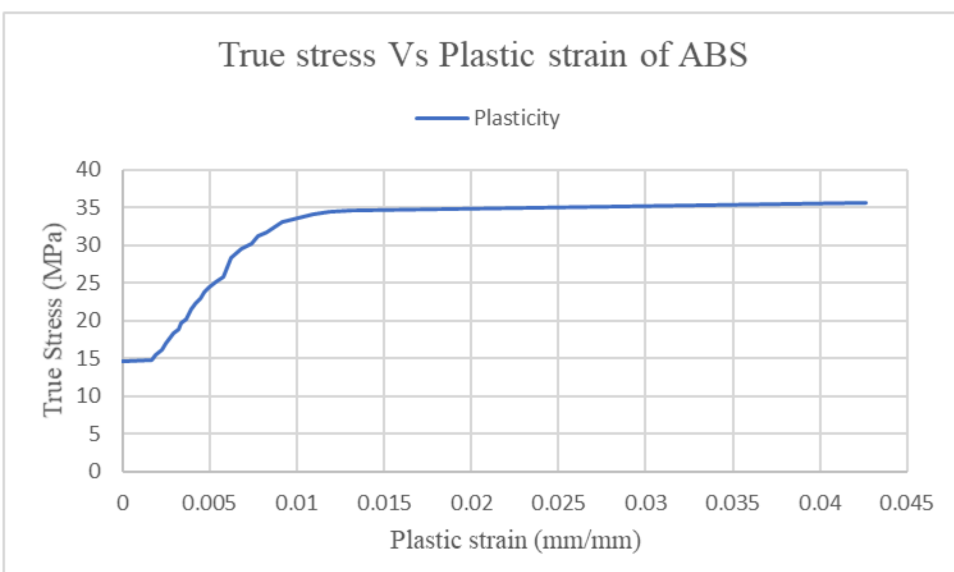
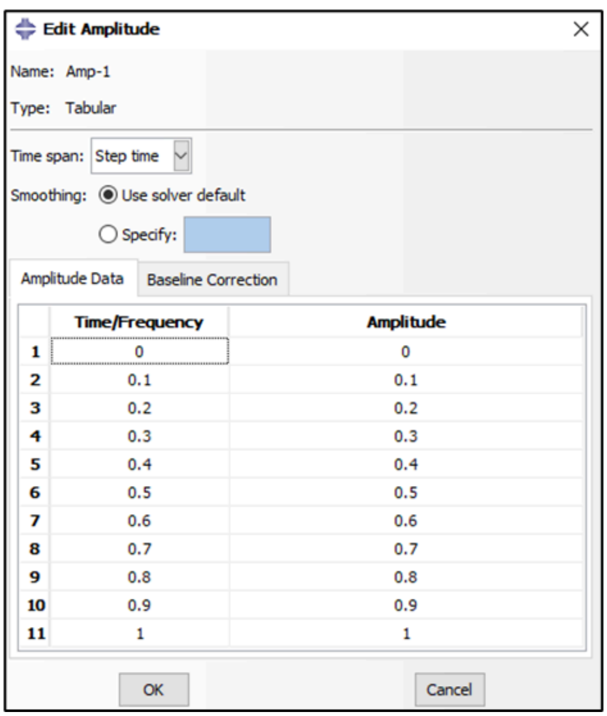


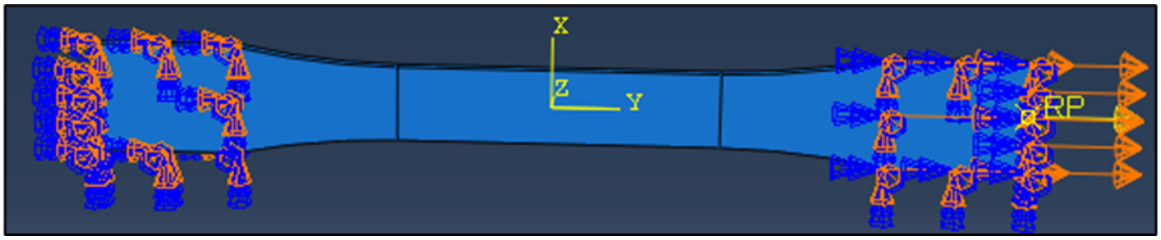
Figure 3.14 True stress Vs Plastic strain of ABS

3.4.2.3 Finite element simulation

The step-type was selected for Static (General), and the option for large strain was selected. In the field output domain, the frequency was set for evenly spaced timed intervals with 100 intervals to show more output results. Before defining boundary conditions, in the "Interface" module, the reference point RP was set as the focal point coupled with the top portion (cell), where the tensile force would start stretching.

The boundary conditions were similar to SolidWorks. The sub-surface (cell) of one end was fixed in all directions, while the other was done also the same, except the Y-direction where the model was stretched 4.71 mm for ABS and 4.16 mm for PLA. On determining the load condition, the tensile force of 5000 N was subjected on the point RP, with the amplitude steadily increasing with response to step time/ frequency (see Fig. 3.15). In Fig. 3.16, which showed the boundary conditions, the left partition was where the point RP existed and the orange tensile force arrows were pointing outwards, while the right partition was fixed firmly. Following this, the mesh condition was set with 1.85 mm sized rectangular elements (see Fig. 3.17).





3.4.2.4 Simulated result

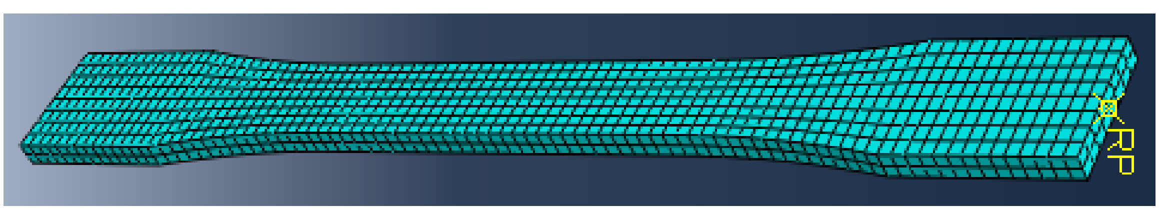
The simulated data can be extracted easily and compared with the experimental results readily,

Figure 3.15 Abaqus' Tensile force's amplitude with time/frequency

and interpreted as charts in Fig. 3.18 and 3.19. In those figures, both the simulated and true values underwent a similar experience as the true value line almost completely overlapped that of the former, but there was an obvious protuberance in the simulated line just before the yield point, which had a difference in height, about 1.2631 MPa for ABS and 1.31007 MPa for PLA. This was due to the fact that Abaqus interpreted the data set on given Young's modulus and Poisson's ratio into two yield points; the first one where the material's elastic properties ended and the other, where the plastic properties initiated. The ultimate stresses and strains were more or less the same

Figure 3.16 Abaqus' Boundary conditions

as the true values. It was also noteworthy that the simulation was solely purposed on finding the tensile mechanical



behavior so the interaction beyond the necking point would not be focused.

Figure 3.17 Abaqus' Meshed conditions

The aforementioned charts' data of ABS and PLA were compared in Table. 3.3, in which error percentages were calculated using Eqn. (4), and the graphical representations were shown in Fig. 3.20 and 3.21. The maximum von Mises stress' locations from the fixed end, where stress concentration was accumulated, were 105.3 mm for ABS and 82.5 mm for PLA.

Table 3.3 Abaqus' simulated data comparison of ABS and PLA

	ABS	PLA	ABS	PLA	ABS	PLA
	True Value		FEA result		Error %	
Yield stress (Mpa)	14.60081	21.55793	14.0131	21.5796	0.04025	-0.001
Yield strain	0.0167	0.015529	0.01449	0.014041	0.13234	-7.52212
Ulimate stress (Mpa)	34.53003	51.64797	34.5536	51.8181	-0.00068	-0.00329
Ultimate strain	0.048791	0.050609	0.049481	0.049481	-0.01414	0.02229

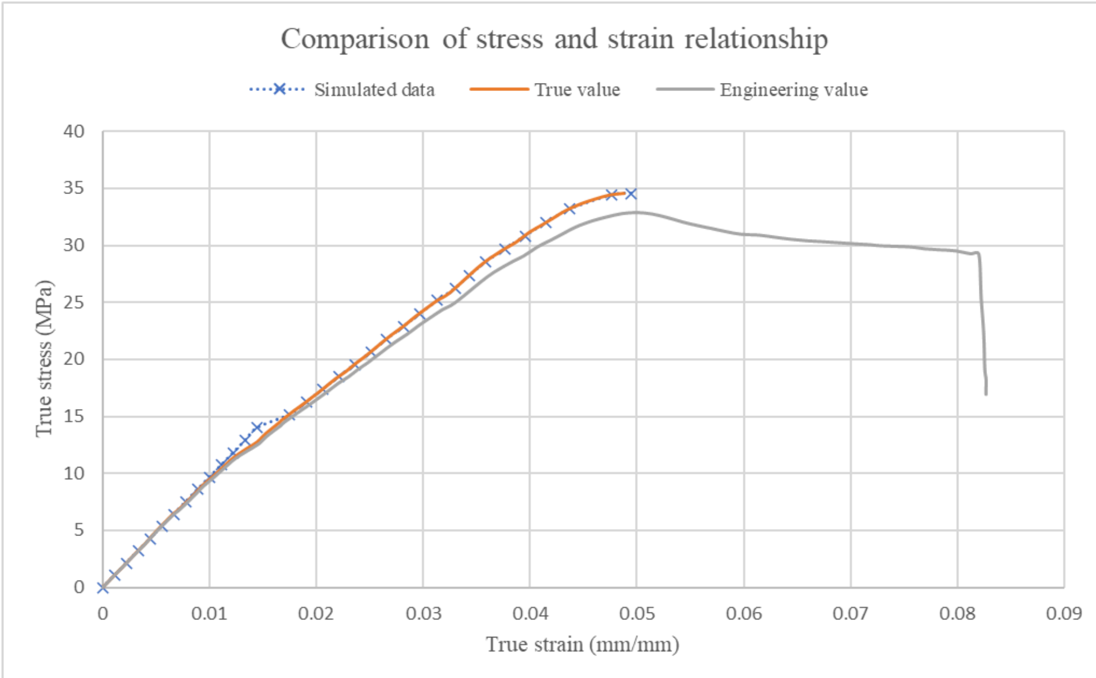


Figure 3.18 Abaqus' Comparison of Stress-Strain Curves of ABS

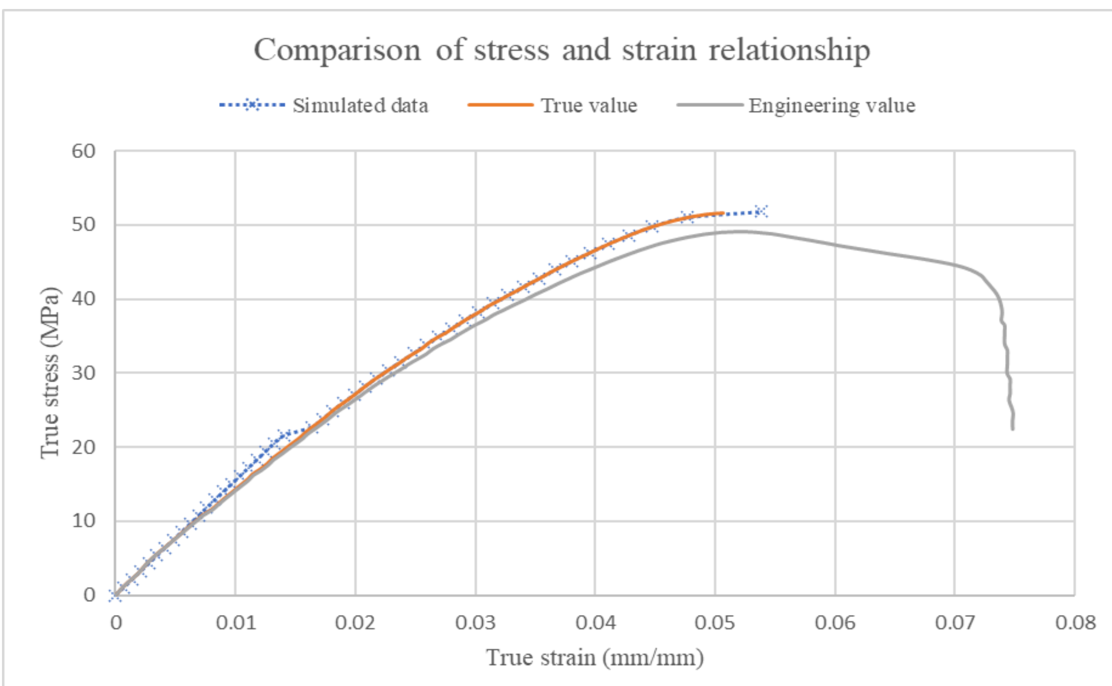


Figure 3.19 Abaqus' Comparison of Stress-Strain Curves of PLA

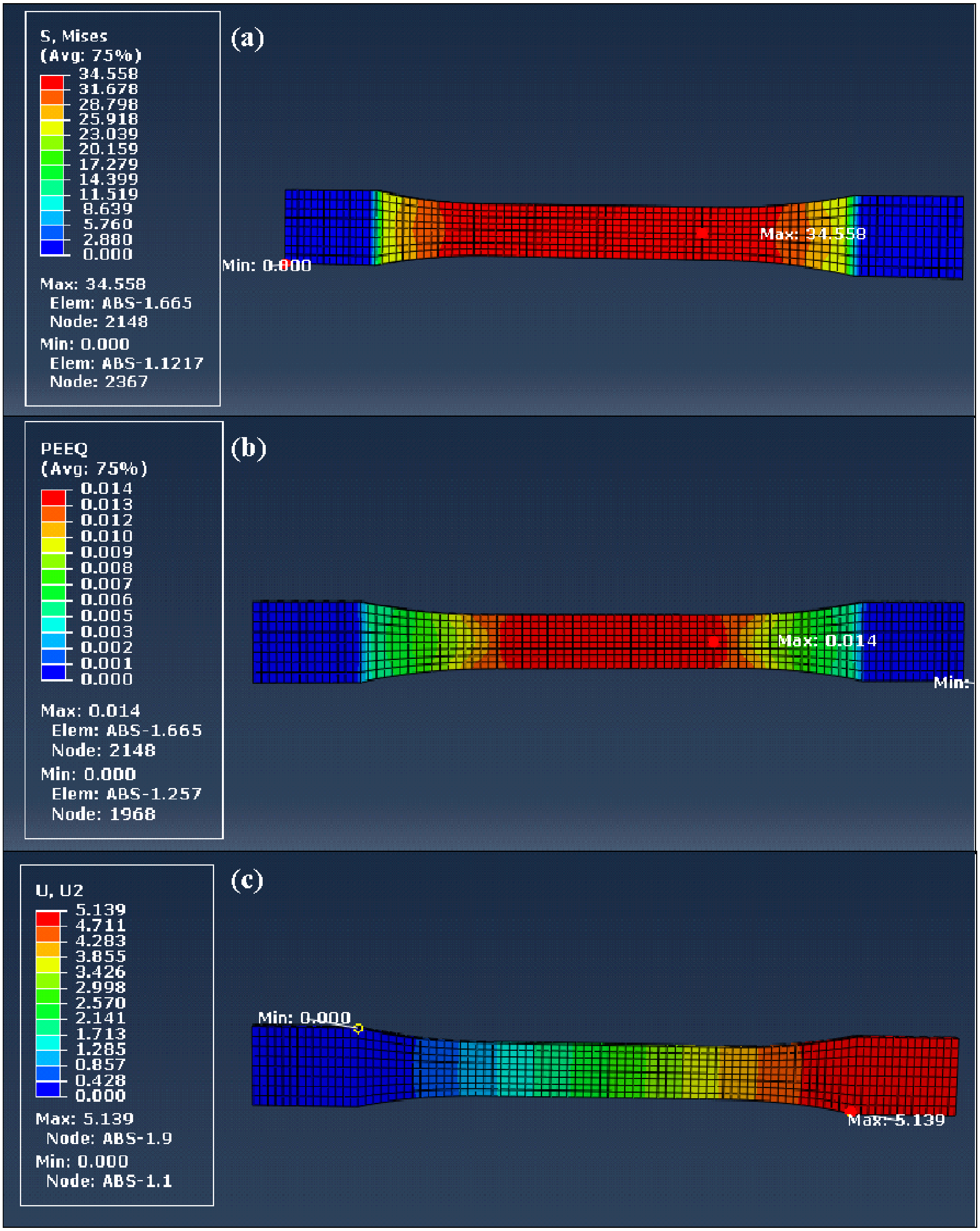
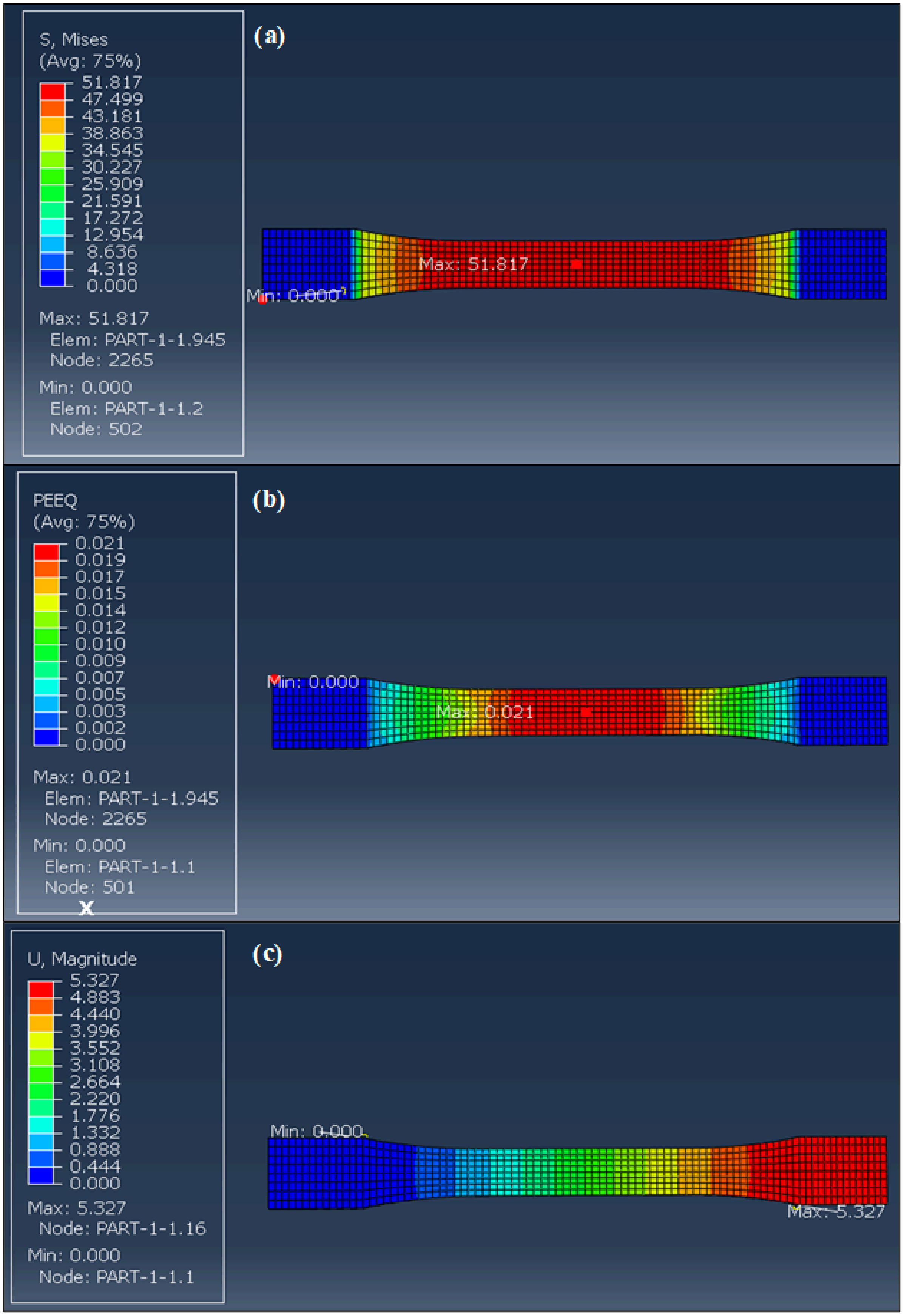


Figure 3.20 Abaqus' ABS' simulated plots of (a) von Mises stress (b) equivalent plastic strain (c) displacement

Figure 3.21 Abaqus' PLA's simulated plots of (a) von Mises stress (b) equivalent plastic strain (c) displacement

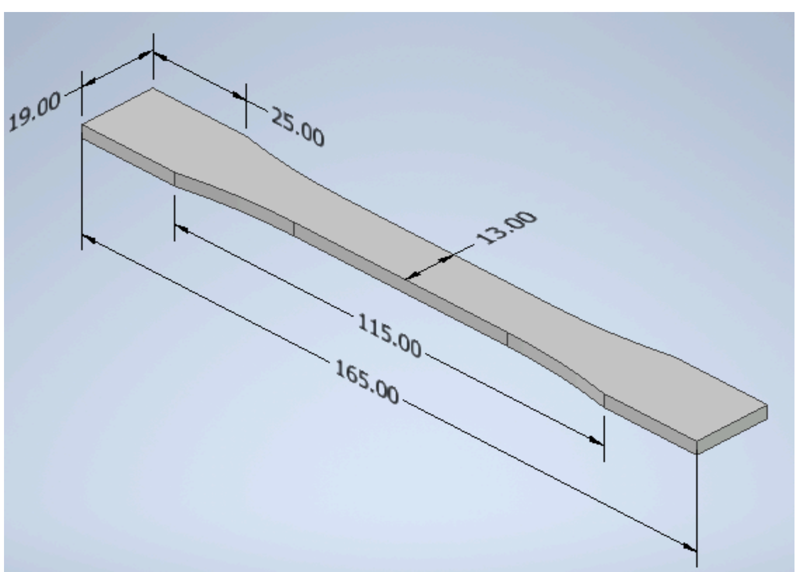

S, Mises
(Avg: 75%)
51.817
47.499
43.181
38.863
34.545
30.227
25.909
21.591
17.272
12.954
8.636
4.318
0.000
Max: 51.817
Elem: PART-1-1.945
Node: 2265
Min: 0.000
Elem: PART-1-1.2
Node: 502
(a)
Max: 51.817
Min: 0.000
PEEQ
(Avg: 75%)
0.021
0.019
0.017
0.015
0.014
0.012
0.010
0.009
0.007
0.005
0.003
0.002
0.000
Max: 0.021
Elem: PART-1-1.945
Node: 2265
Min: 0.000
Elem: PART-1-1.1
Node: 501
(b)
Min: 0.000
Max: 0.021
U, Magnitude
5.327
4.883
4.440
3.996
3.552
3.108
2.664
2.220
1.776
1.332
0.888
0.444
0.000
Max: 5.327
Node: PART-1-1.16
Min: 0.000
Node: PART-1-1.1
(c)
Min: 0.000
Max: 5.327

3.4.3 Autodesk NASTRAN

Autodesk Inventor is one of the leading 3D modelling software(s), developed by Autodesk Company. Its capabilities rival with those of SolidWorks, but the usage of user-friendly interfaces and a better learning path put it more suitable for beginner design engineers. However, for simulation, the software alone can only perform linear, static problems, so another add-in module of its own, called NASTRAN was installed to be able to perform non-linear problems. In order to perform more sophisticated conditions, one must open Autodesk Inventor, in which the NASTRAN environment can be accessed. This project used 2020 student version of Autodesk Inventor Professional with NASTRAN module installed, and the simulation was tested with the non-linear static condition.

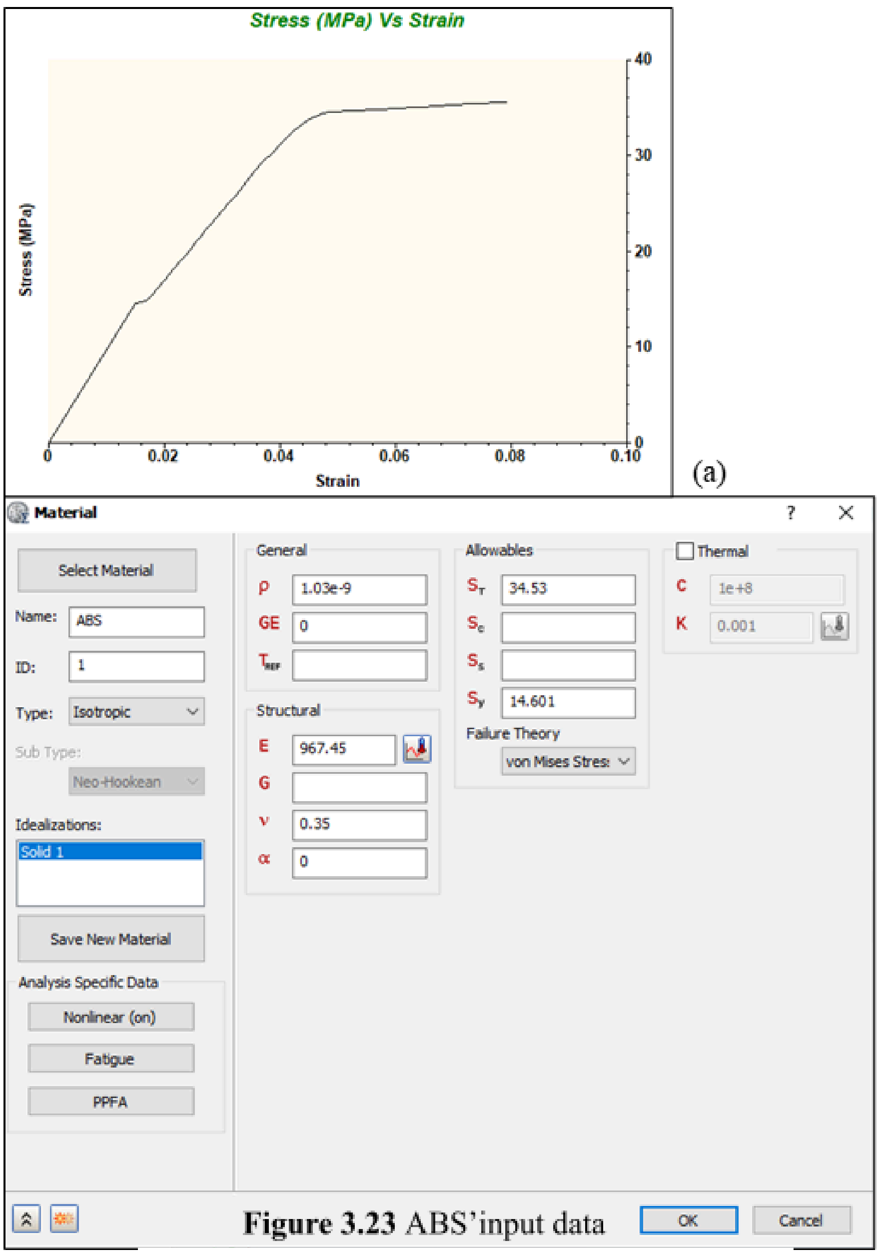
3.4.3.1 3D CAD model creation

Much like its counterparts, the procedure was started by the 2D sketch and the sketch was extruded 3.6mm for thickness. Fig. 3.22 showed the CAD model in mm. However, unlike the former software(s), the CAD model was not partitioned due to the fact that when replicating the real tensile test's conditions, the model was literally cut into partitions, which might alter its material properties.



3.4.3.2 Data input

The NASTRAN environment can be accessed through the "Environments" Panel. There, firstly, the analysis was changed to "non-linear static". The materials (ABS and PLA)'s data from Table 3.1 were inputted, and it was noteworthy that true values were used. To solve the non-linear problem, true values of strain and stress from the yield to the rupture were inputted. The procedure's data on ABS were shown in Fig. 3.23, and all was true for PLA.



(a)

(b)

Figure 3.23 ABS'input data

Figure 3.22 Autodesk Inventor's 3D CAD model

(a) Inputted data's stress-strain curve (b) Inputted data

3.4.3.3 Finite element simulation

The mesh elements were 1.85mm with the triangular pattern with angles between 20 deg and 30 deg and element growth rate of 1.5 (see Fig. 3.24). For the boundary conditions, one of the model's end was fixed in all directions while the opposite end was subjected with 5000N tensile force in the axial direction. Moreover, during the forced end, the displacement at failure (4.71 mm for ABS and 4.16mm for PLA) were enforced.

However, the prominent difference from other software(s) was that every surface of the model, except the fixed end, was needed to permit only the axial direction so that it can elongate in this direction only. Ignoring this constraint can lead to unwanted results. In Fig. 3.25, the green arrow pointing outwards and the green cylinder on the bottom right side represented the tensile force of 5000N and the displacement to reach, while the cyan arrow clusters represented a constraint that fixed the specimen in all directions except the axial, but the cluster at the opposite of the enforced surface represented the fixed condition.

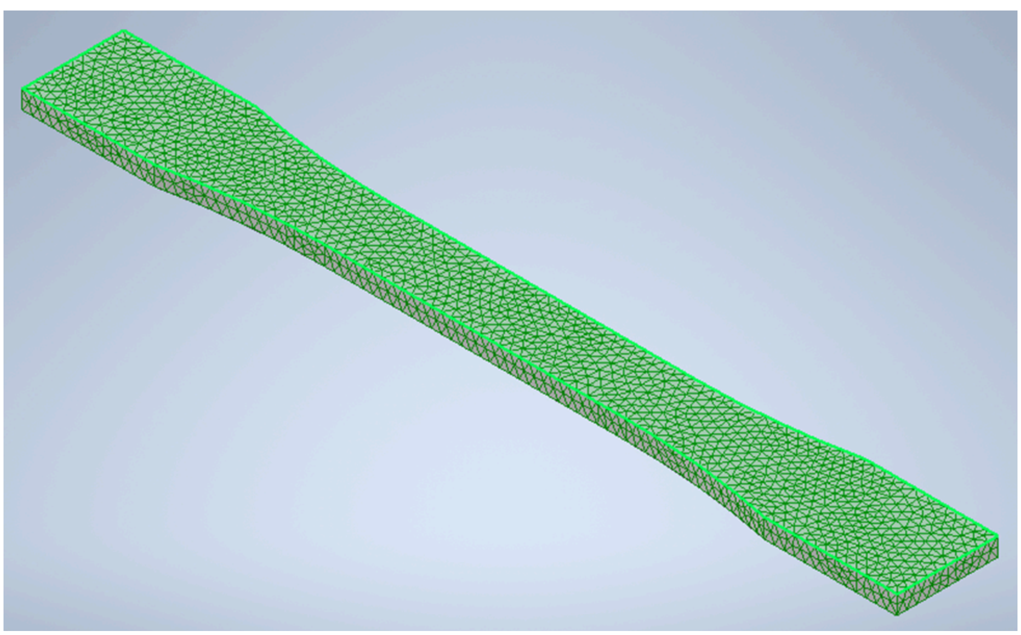


Figure 3.24 Autodesk NASTRAN's meshed condition

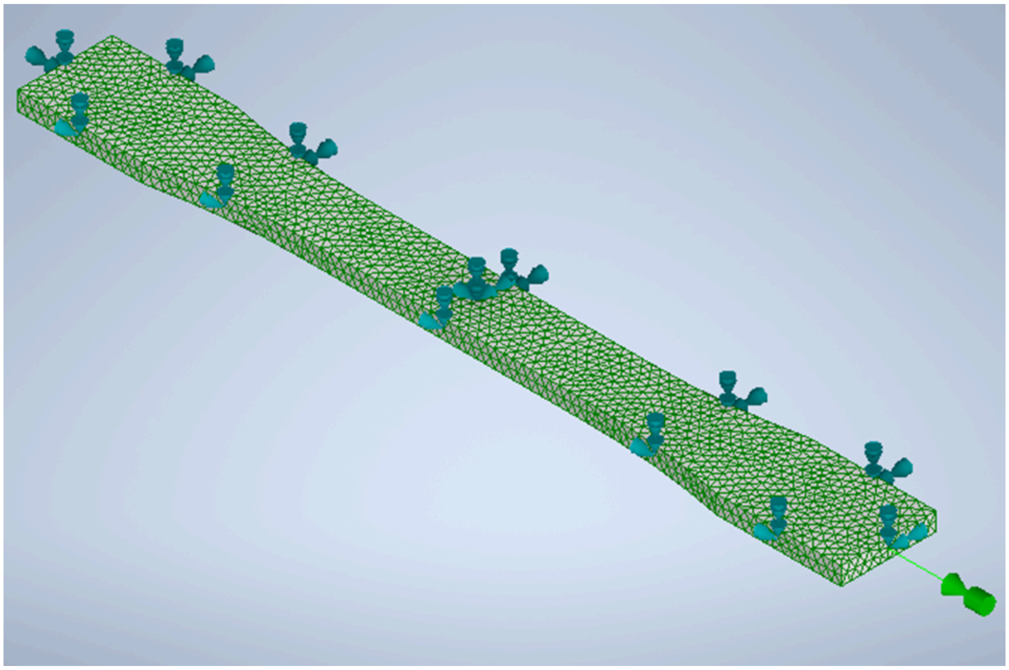


Figure 3.25 Autodesk NASTRAN's boundary conditions

3.4.3.4 Simulated result

The simulated results were interpreted as charts in Fig. 3.26-28 for ABS and 3.29-31 for PLA. The simulated results for both materials were compared with the reference from Table 3.1, and the error percentages were calculated using Eqn. (6), and shown in Table 3.4. The maximum von Mises stress' locations from the fixed end, where stress concentration was accumulated, were 115.8 mm for ABS and 50.8 mm for PLA.

Table 3.4 Autodesk NASTRAN's simulated data comparison of ABS and PLA

	ABS	PLA	ABS	PLA	ABS	PLA
	True Value		FEA result		Error %	
Ulimate stress (Mpa)	34.53003	51.64797	35.79	43.382	-3.64891	16.00444
Ultimate strain	0.048791	0.050609	0.068	0.033	-39.37	34.79421

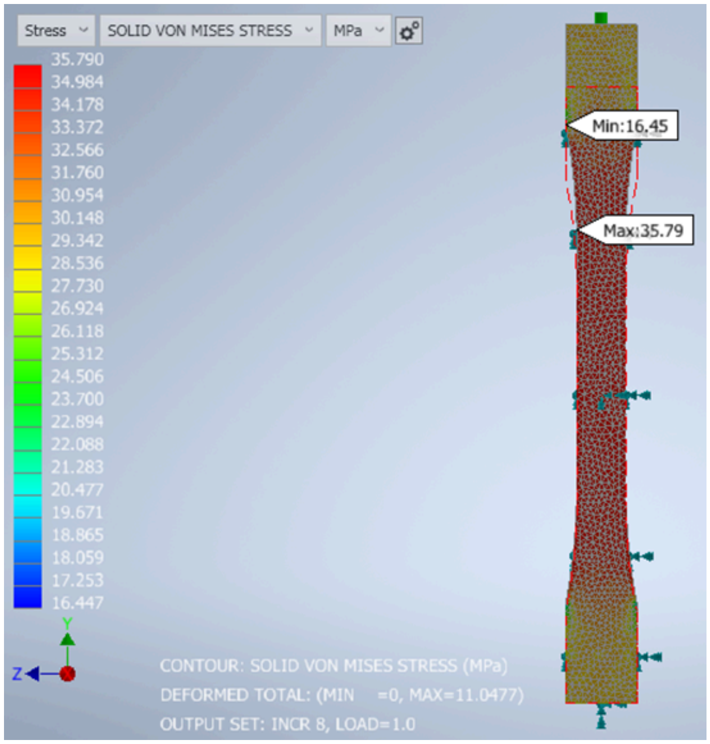


Figure 3.26 Autodesk NASTRAN's ABS' simulated von Mises stress

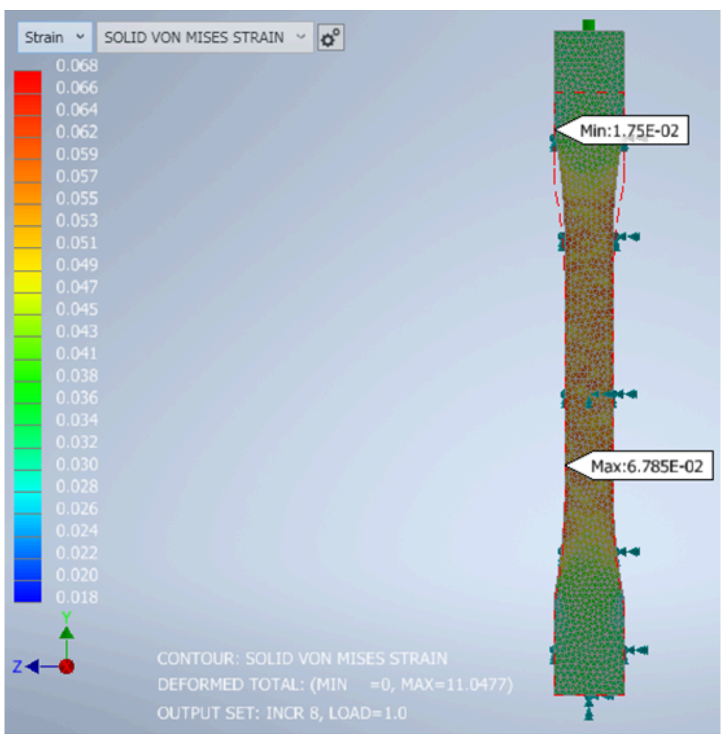


Figure 3.27 Autodesk NASTRAN's ABS' simulated von Mises strain

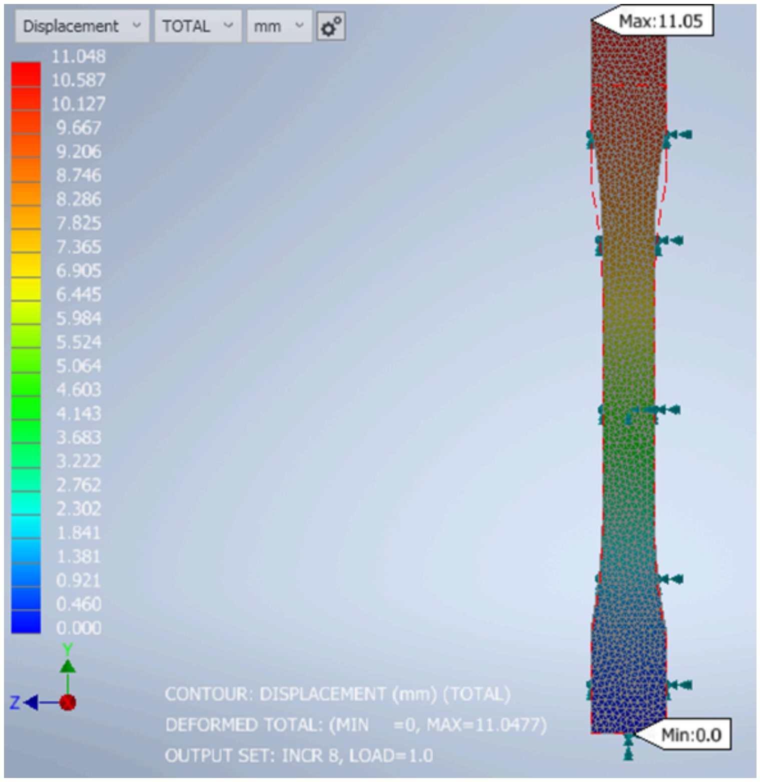


Figure 3.28 Autodesk NASTRAN's ABS' simulated Displacement

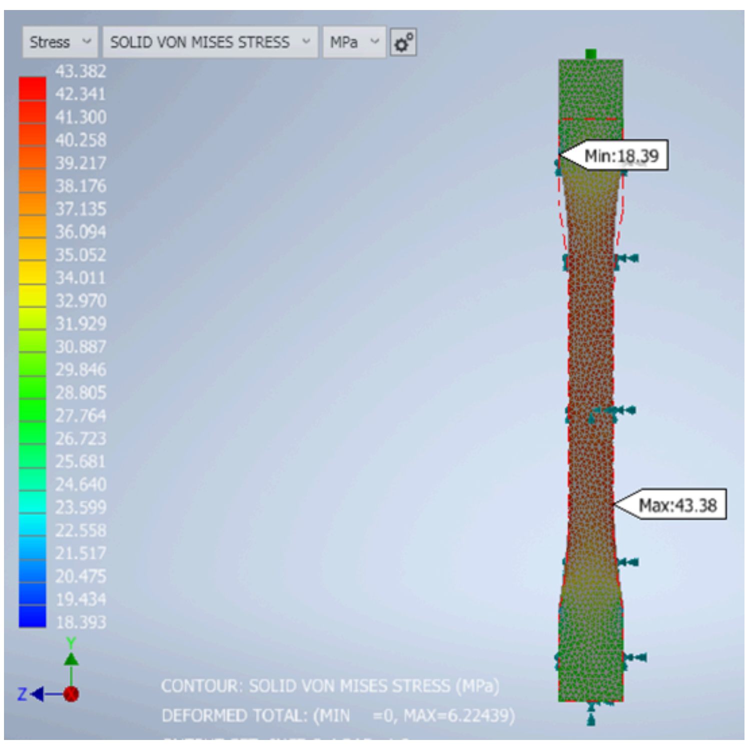


Figure 3.29 Autodesk NASTRAN's PLA's simulated von Mises stress

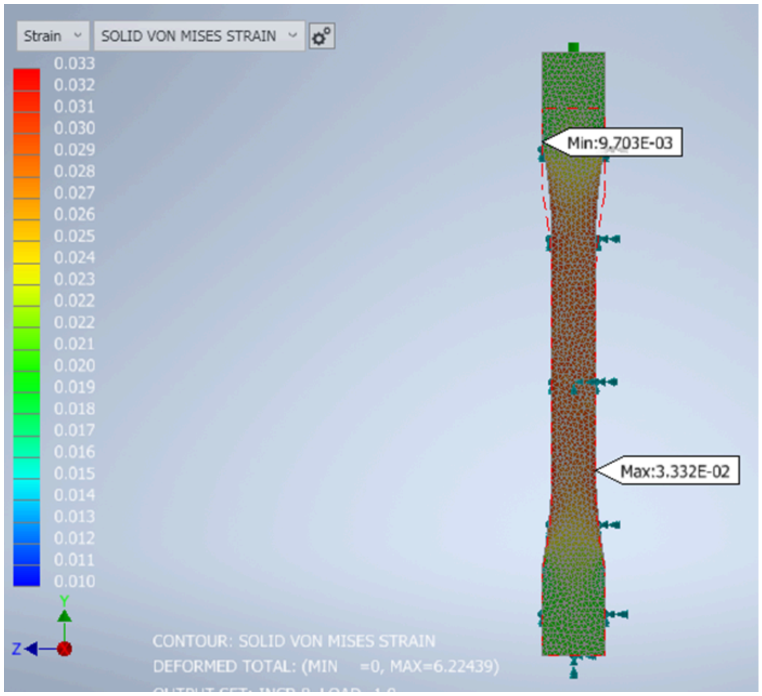


Figure 3.30 Autodesk NASTRAN's PLA's simulated von Mises strain

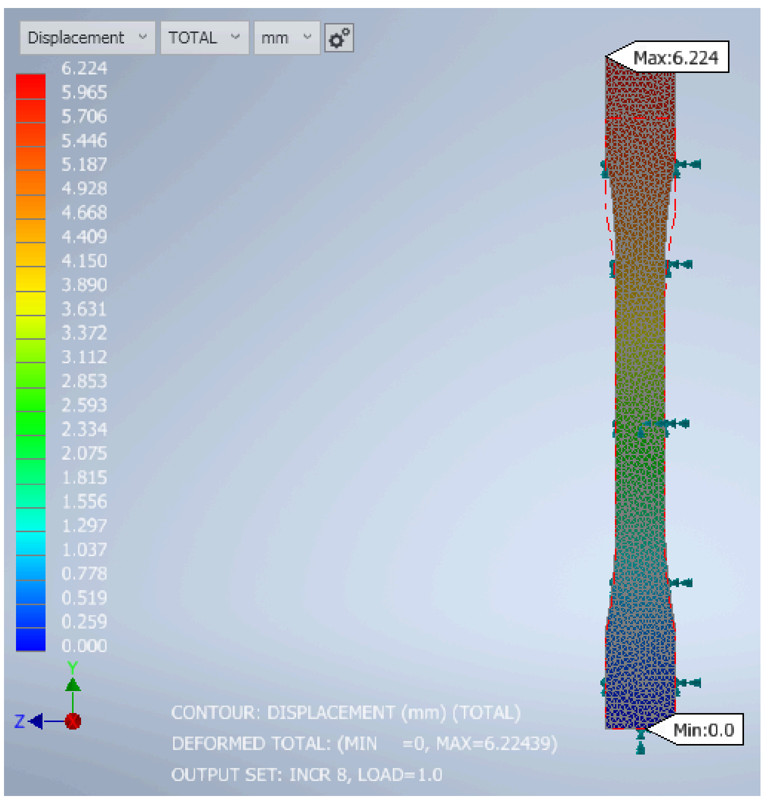


Figure 3.31 Autodesk NASTRAN's PLA's simulated Displacement

3.5 Experimental test

In addition to simulation, practical tensile tests were also performed to see the reference paper and the findings were coincided, and the tensile behaviors of locally purchased ABS and PLA. Depending on the output data, the test was performed on three occasions; one in October and the rest in December. The test was conducted using a universal tensile machine of Testometric brand with the load cell of 3 tons at Myanmar Maritime University, Thanlyin. Due to the pin type of the available tensile machine (see Fig. 3.32), the test used Fig. 3.33 as the primary specimen's dimensions, which was based on Fig. 3.2. The test specimen's specifications and the 3D printer's information were shown in Table 3.5.

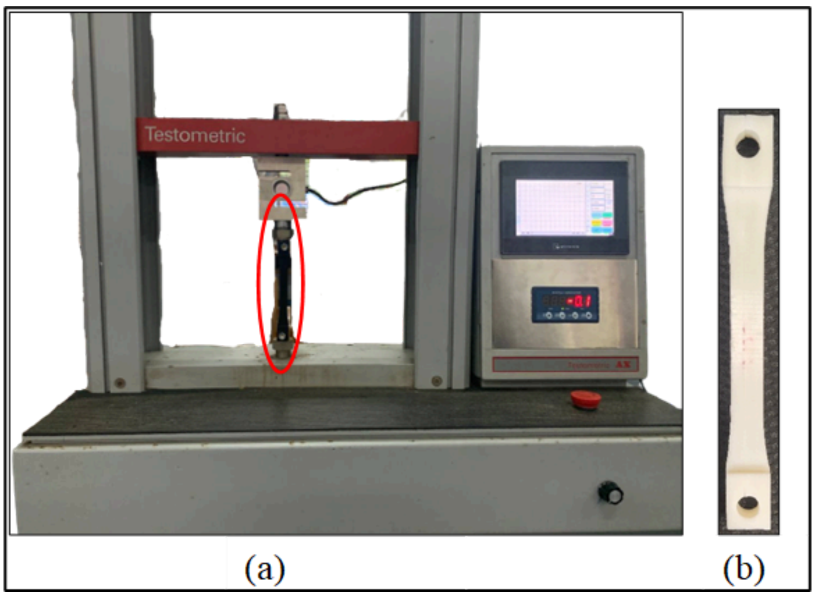


Figure 3.32 Testometric's universal tensile machine

(a) Universal Tensile testing machine with PLA (black) specimen
(b) Test specimen of ABS (white)

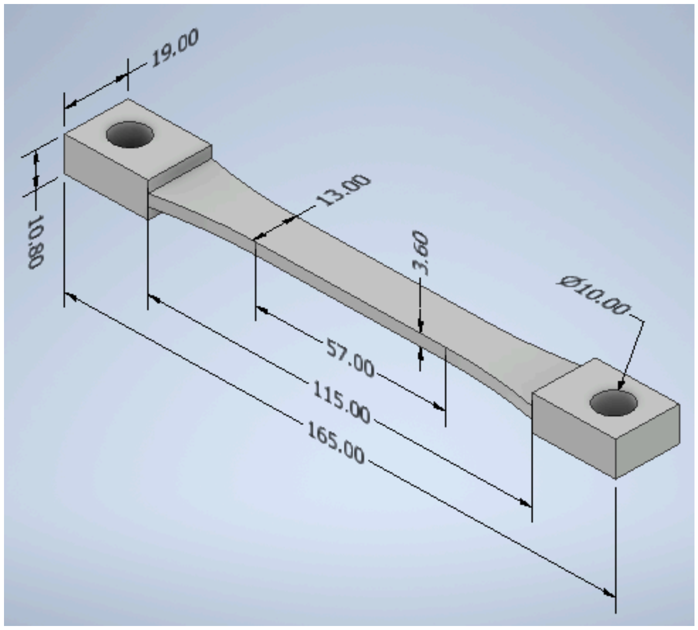


Figure 3.33 Experimental test's primary CAD model in mm

Table 3.5 Test specimen's related information

	ABS	PLA
Print Temperature	250'C	210'C
Heatbed Temperature	100'C	50'C
Filament diameter	1.75mm	1.75mm
Pattern	Rectilinear	Rectilinear
Angle	45'/-45'	45'/-45'
Color	White	Black
Filament brand	Hello3D (http://www.hello3dprint.com/)	
3D printer brand	Creality (https://creality3d.shop/)	
3D printer model	Ender 3V2 Neo	
Printing service	MKK 3D Printing service	

3.5.1 Test procedures

The general procedures were as follows;

1. The pin holes of diameter 10mm from Fig. 3.33 were connected to the 10mm steel nut of the tensile machine; the bottom pin fixed the specimen while the top was meant to stretch the specimen (see Fig. 3.32(a))
2. The specimen was adjusted to make it stable hanging on the machine
3. Due to the machine's availability, the specimen was pulled with tensile force 5000N and the crosshead speed of $7mms^{-1}$, while the rupture occurred
4. The resulted data was obtained in reaction force and elongation length. Respective stress and strain values were calculated using Eqn. (1) and (2). In this project, the original cross-sectional area would be $13\,mm$ x $3.6\,mm$ = $46.8\ mm^2$ and the gauge length would be $57\,mm$.

The whole procedure from the start to the finish took around an hour.

3.5.2 The first test

The very first experimental test was carried out on ABS and PLA's specimens of Fig. 3.33 under the supervision of Dr. Thu Han Tun on 13th of October. The rupture point was estimated to happen in between the gauge length. The procedures were according to the Section 3.5.1. However, a breakdown was occurred at the specimen's end where the tensile force was subjected, which can lead to deviations in the result (see Fig. 3.35(b)), where a red circle was drawn where the unwanted rupture occurred. So, the experiment's data were not taken account. It can be estimated that the specimen's end portions were not tolerant to the machine's tensile capability and a dimensional change at the aforementioned areas were suggested to change for the second test.

3.5.3 The second test

The second test was carried out on ABS and PLA's specimens on 9th December, after a few modifications recommended by the supervisor. The changed dimensions were shown in Fig. 3.34. In this test, the ABS specimen showed a similar resemblance to that of the reference paper, Baca Lopez (2020). However, the PLA specimen showed a completely dissimilar result. It might be due to the fact that there was a printing problem as when the tensile test was carried out, some of the specimen's filament layers were not interconnected enough, and they broke apart upon tensile. This behavior was not seen in ABS counterpart. The results were displayed in Fig 3.35.

Fig. 3.36 and 3.37 showed the stress-strain relationship of ABS and PLA of the experimental result in comparison with the result from the reference paper (see Fig. 3.4), but the curve showed a different pattern to the reference stress-strain curve. This was due to the fact that the specimen was drawn with the cross-head speed of $7mms^{-1}$ instead of 2. Therefore, the obvious representation of the elastic region was not visible enough. However, the tensile properties from the experiment of Fig. 3.35 showed a similarity with the reference. In Fig. 3.35, the ABS' highest point, which was the ultimate stress' location was (ε_e = 0.074623819, σ_e = 32.28076902) and the rupture point was (ε_e = 0.086938247, σ_e = 21.61134638), whilst those of

However, the PLA's practical test result showed different results; especially in the ultimate stress area, where its location was (ε_e = 0.061748249, σ_e = 19.20076941), which was quite different to the reference's location (ε_e = 0.051912, σ_e = 49.09914). Also, in the rupture point as the experimental one was broken down at (ε_e = 0.063395655, σ_e = 12.30442406), while that of the reference was (ε_e = 0.074807, σ_e = 22.41659). In this test, the rupture points from the fixed end were found out to be 121 mm for ABS and 119.5mm for PLA.

the reference was (ε_e = 0.050001, σ_e = 32.88571) and (ε_e = 0.082642, σ_e = 16.93277).

Due to the unexpected result, another test had to be carried out on the PLA specimen only with much care 3D printing techniques.

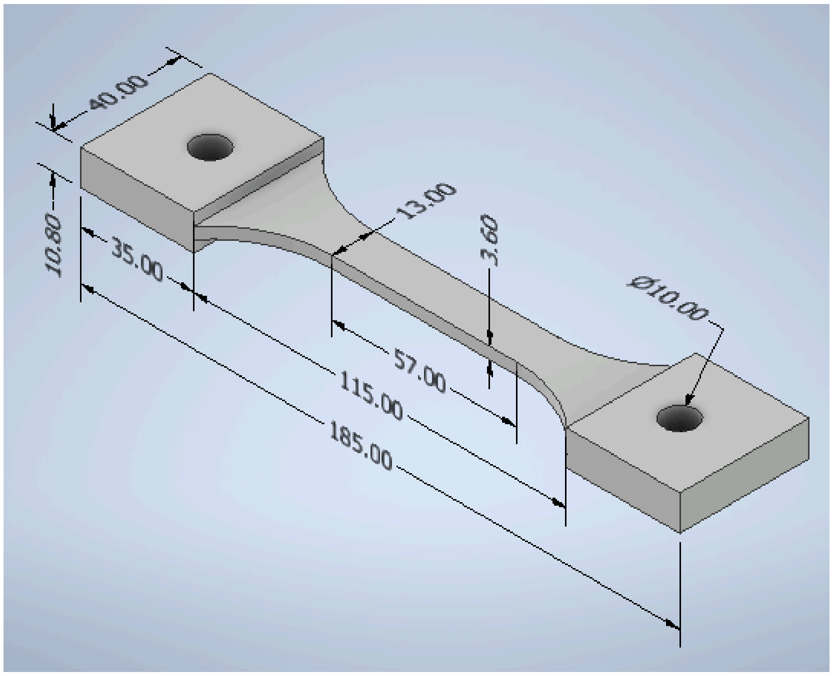


Figure 3.34 Experimental test's secondary CAD model in mm

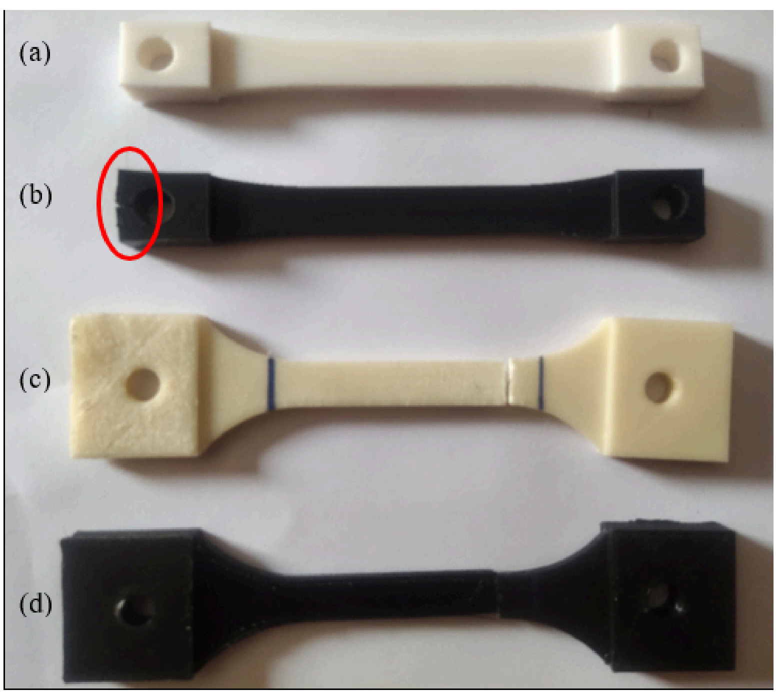


Figure 3.35 First and Second tests' results

The first test specimen of (a) ABS (b) PLA, The second test specimen of fractured (c) ABS (d) PLA

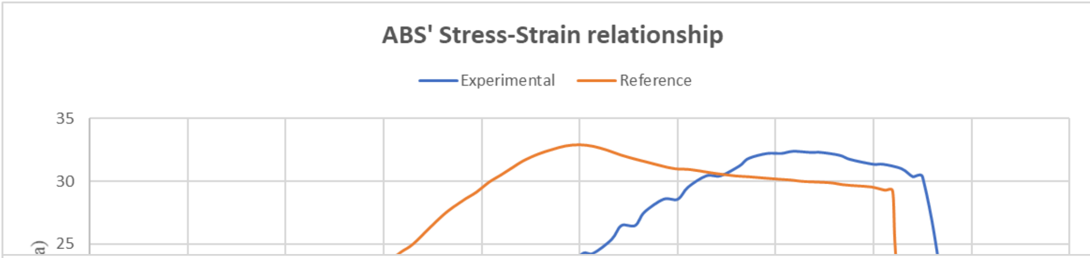


Figure 3.36 Comparison of ABS' stress-strain diagram (second test)

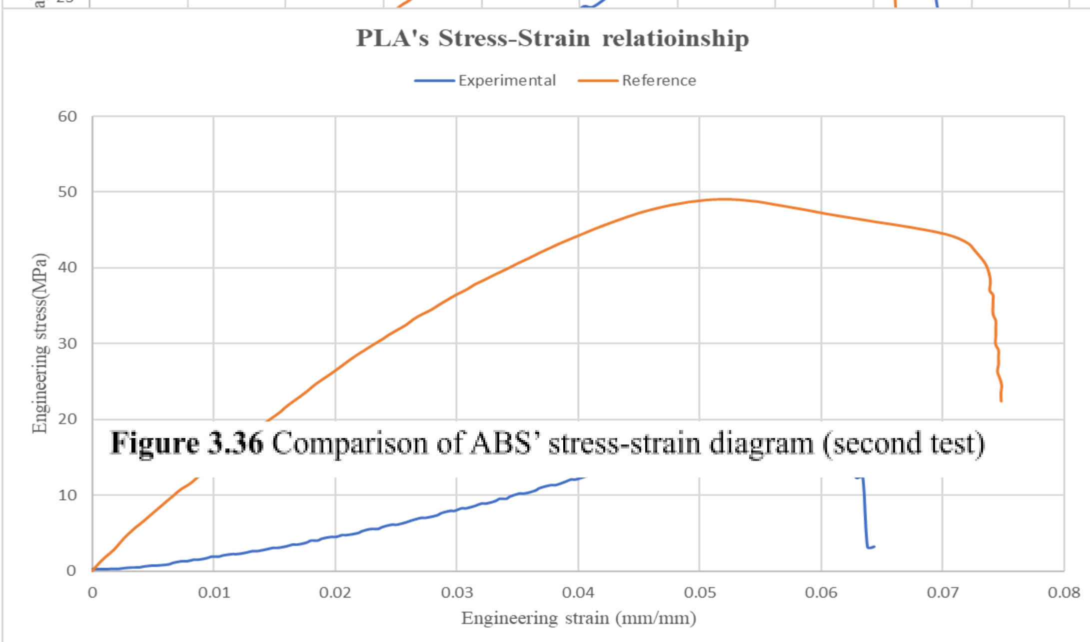


Figure 3.37 Comparison of PLA's stress-strain diagram (second test)

3.5.4 The third test

The third test was carried out on PLA and PLA+ specimens on 18th December, using the same procedures as the second test. The latter was a plasticizer-modified form of PLA, which was more ductile, flexible and stronger. In this test, both PLA specimen showed a similar resemblance to that of the reference paper but that of the reference showed higher ultimate stress but the PLA+ specimen was much more ductile than the reference sample. The tested specimens were shown in Fig. 3.38, and the results were shown in Fig. 3.39 and 3.40.

In Fig. 3.39, the PLA's ultimate stress' location was ($\varepsilon_e = 0.060198675$, $\sigma_e = 41.30646035$) and the rupture point was ($\varepsilon_e = 0.061204333$, $\sigma_e = 14.96479176$), while those of Fig. 3.40's PLA+ were ($\varepsilon_e = 0.072609366$, $\sigma_e = 34.63021172$) and ($\varepsilon_e = 0.117156547$, $\sigma_e = 15.82770877$). Both of these results were similar to those of the reference paper's ultimate stress, ($\varepsilon_e = 0.051912$, $\sigma_e =$ 49.09914) and the rupture point, ($\varepsilon_e = 0.074807$, $\sigma_e = 22.41659$).

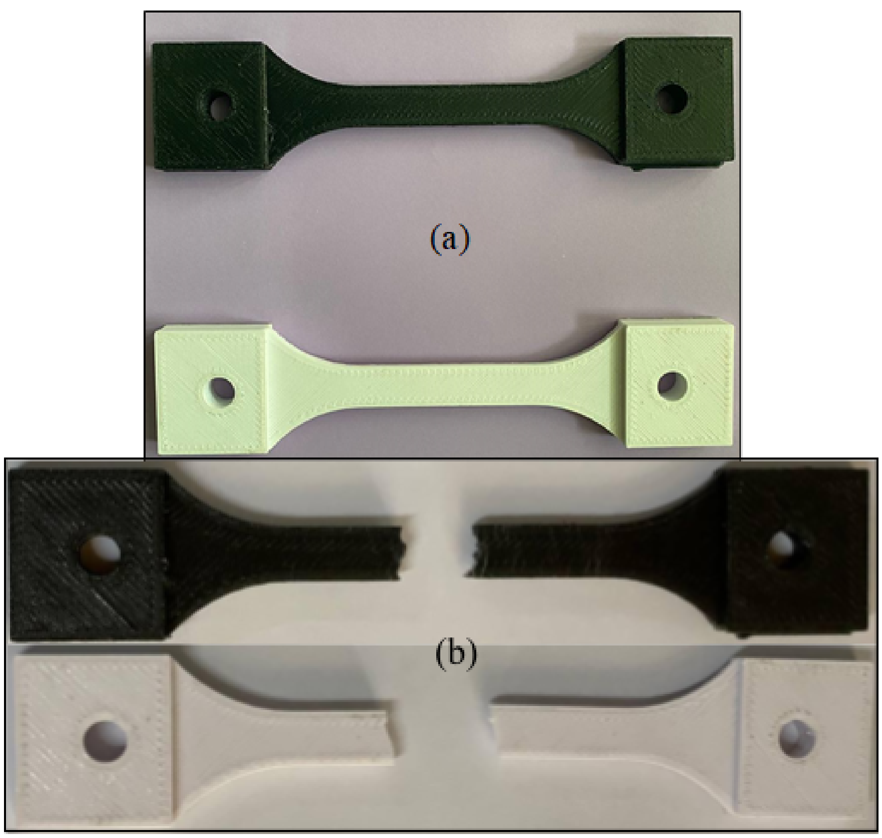


Figure 3.38 Third test's result

(a) Third test's specimen; PLA+(Black) and PLA(White), (b) Fractured specimens

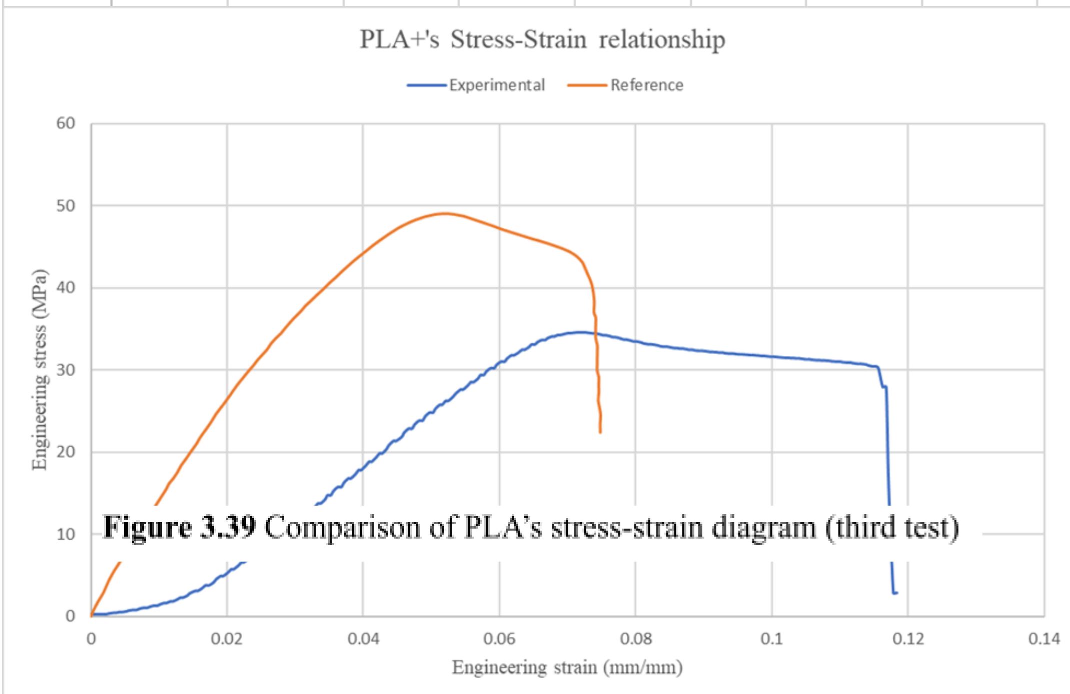


Figure 3.39 Comparison of PLA's stress-strain diagram (third test)

Figure 3.40 Comparison of PLA+'s stress-strain diagram (third test)

Chapter 4: Result and Discussion

The simulation results carried out by SolidWorks, Abaqus and Autodesk NASTRAN, were compared and contrasted with the reference paper, Baca Lopez (2020). In addition to the degree of similarity with the reference data, more criteria had been set to pick out the optimal software, especially the ease and conformance of use between SolidWorks and Autodesk NASTRAN. Moreover, several mechanical parameters had been calibrated from the graphs of the experiment and Baca Lopez (2020).

4.1 Simulation comparison

A comparison chart was drawn to compare the capabilities between the three FEM software tested. Several criteria were set to test out each software(s)' capabilities, and the grading system was set in some criteria from 1 being "very easy" to 5 being "very hard".

Table 4.1 Simulation comparison chart

	SolidWorks	Abaqus	Autodesk NASTRAN
User interface	2	3	1
CAD model creation	2	3	1
Date input difficulty	2	4	2
Boundary condition difficulty	1	5	1
FEM analysis duration	4	1	5
ABS' ultimate stress error %	3.574	-0.00068	-3.64891
PLA's ultimate stress error %	10.35272	-0.00329	16.00444
ABS' ultimate strain error %	7.76988	-0.01414	-39.37
PLA's ultimate strain error %	25.03452	0.02229	34.79421
ABS' failure location from fixed end (mm)	112.8	105.3	115.8
PLA's failure location from fixed end (mm)	139.6	82.5	50.8
Output data extraction	5	2	5

As seen from Table 4.1, Autodesk Inventor and SolidWorks, being the top modelling software(s), have the excellent user interfaces and much ease in CAD model creation; the former's interface and tool bars are more appealed to the users, which means that by using Autodesk Inventor, a beginner can create a 3D model from a scratch to web surfing a little. However, for Abaqus, one needs to take much time to learn about its interface. Similarly, in Abaqus, in order to input plasticity data, an additional and sophisticated transformation of true strains to plastic values (see Section 3.4.2.2) was needed to produce plastic behaviors, whereas the other two only required true stress-strain relationships. The same can be true for boundary conditions, such as fixed constraints, force and enforced motions and meshed conditions.

However, the duration of ABS' FEM analysis was three times shorter than the other two, which took from minimum 15 min to maximum 50 min approximately, with Autodesk NASTRAN taking the longest. It was noteworthy that the computer used was DELL's Intel(R) Core (TM) i7-6500U CPU @ 2.50GHz- 2.60 GHz with 16GB RAM and 64-bit operating system, with all three software(s) installed. In addition, Abaqus' error percentages were no more than 0.03%, which made the perfect FEM software for this project. When compared the error % of the second and the third, SolidWorks constituted 25% error, whereas the largest error percentage (34.79421%) was found in Autodesk NASTRAN. This made SolidWorks stood the second in FEM analysis, followed by Autodesk NASTRAN. Moreover, Abaqus was the only software which was easy enough to plot back the simulated stress-strain diagram (see Fig. 3.18 and 3.19).

Therefore, from the criteria, it is clear that Abaqus is the most powerful when one wants to test out finite element analysis. This can be true for single parts and assemblies which would not have complex interrelationships or constraints between them, but when it comes down to more complex parts, one should have considered SolidWorks. Although Autodesk Inventor can be a powerful 3D modelling software for beginners and amateur design engineers in comparison with the other two, its FEM capabilities are the least accurate and powerful.

4.2 Data from engineering graphs

Several mechanical parameters such as the maximum tensile force and fracture energy can be extracted for ABS, PLA and PLA+ from the second and third tests, and the reference paper (see Fig. 3.35, 3.36, 3.39 and 3.40).

4.2.1 Maximum tensile force

. The maximum tensile force was also called the tensile force, which can be calibrated by dividing the ultimate stress with the original cross-sectional area of the specimen. It was the force which required to rupture the respective specimen's material and the amount of which that material can withstand. It was noteworthy that the tensile test with the force much greater than the maximum tensile force, which, in this case, was 5000N, was carried out to find out its mechanical behaviors. The equation for the maximum tensile force was calculated by;

$F_{max} = \sigma_u . A_0$------(8) , where F_{max} is the maximum tensile force, σ_u is the ultimate stress and A_0 is the original cross-sectional area. For the calculation, the necessary specimen's dimensions were referred from Fig. 3.2 and 3.33; the cross-sectional area, A_0 = width x thickness = 13 x 3.6 = 46.8 mm^2

From the reference, the ultimate stress of ABS was 32.88571 MPa and that of PLA was

For ABS, $F_{max} = (32.88571 \times 10^6)(46.8 \times 10^{-6}) = 1539.05123$ N = 0.1569 ton-force

For PLA, $F_{max} = (49.09914 \times 10^6)(46.8 \times 10^{-6}) = 2297.83975$ N = 0.2343 ton-force

49.09914 MPa. Therefore, by using Eqn. (8), the maximum tensile forces would be;

This means that in minimum, a load cell of 0.2 ton and 0.25 ton was needed to rupture ABS and PLA specimen, and until those amounts of forces, those specimens will not break down.

Similar procedures can be down for experimental values; second test's ABS (32.28076902 MPa) and third test's PLA (41.306460 MPa) and PLA+'s (34.63021172 MPa);

The result of experimental result on ABS was similar to that of the reference with a small gap of 28.31723 N but PLA's and PLA+'s results showed big gaps of 364.69775 N and 677.14575N.

4.2.2 Fracture energy

The fracture energy is the amount of work done that needs to completely break down a m^3 of a specimen. It can be found out by the combination of total multiplication of all stress-strain values, which $A_n = \frac{\sigma_n + \sigma_{n+1}}{2} \cdot (\varepsilon_{n+1} - \varepsilon_n)$------(9) can be found out by finding the total area under the curves from the stress-strain graphs. The equation for fracture energy was calculated by;

where A_n is the specific area under the curve at n^{th} turn, σ_n and ε_n are the engineering stress and strain at n^{th} turn. After inputting all the values of stress and strain in Eqn. (9), respective areas under the curve, A_n were added altogether to obtain Fracture energy. The procedure was done for

From the reference paper, the fracture energy of ABS would be 1.95336 $\frac{MJ}{m^3}$, and that of PLA would be 2.611656 $\frac{MJ}{m^3}$. From second and third's experiments, the fracture energy of ABS would be 1.65221 $\frac{MJ}{m^3}$, and that of PLA would be 1.230924039 $\frac{MJ}{m^3}$ and PLA+'s fracture energy was 2.61923485 $\frac{MJ}{m^3}$

As seen from above, similarly, there was a slight difference between the reference and the experiments of 0.30115 $\frac{MJ}{m^3}$ between ABS specimens but there was a large gap in PLA specimen, of about 1.38073 $\frac{MJ}{m^3}$ but PLA+ showed the closest gap of 0.0075[illegible] $\frac{MJ}{m^3}$.

For ABS, $F_{max} = (32.280769 \times 10^6) \times (46.8 \times 10^{-6}) = 1510.734$ N = 0.1541 ton-force

For PLA, $F_{max} = (41.306460 \times 10^6) \times (46.8 \times 10^{-6}) = 1933.142$ N = 0.1933 ton-force

For PLA+, $F_{max} = (34.630212 \times 10^6) \times (46.8 \times 10^{-6}) = 1620.694$ N = 0.1621 ton-force both experimental and reference's data at Microsoft Excel©.

4.2.3 ABS Vs PLA's result discussion

Fig. 4.1 was plotted to compare and contrast the tensile behaviors of ABS, PLA and PLA+ of the reference paper, Baca Lopez (2020) and three experimental tests. ABS' data was taken from the second test while both PLAs' were from the third test. In the figure, both the reference and the experimental PLA showed greater ultimate strength than any other curves, which meant PLA was much more strength resistant than ABS and PLA+. However, PLA+ showed the greatest ductility, followed by ABS. Although the reference ABS' curve was more or less similar to that of the experiment, both the experimental PLA and PLA+'s highest ultimate points could not reach the near peak value of the reference, and followed a slightly different pattern.

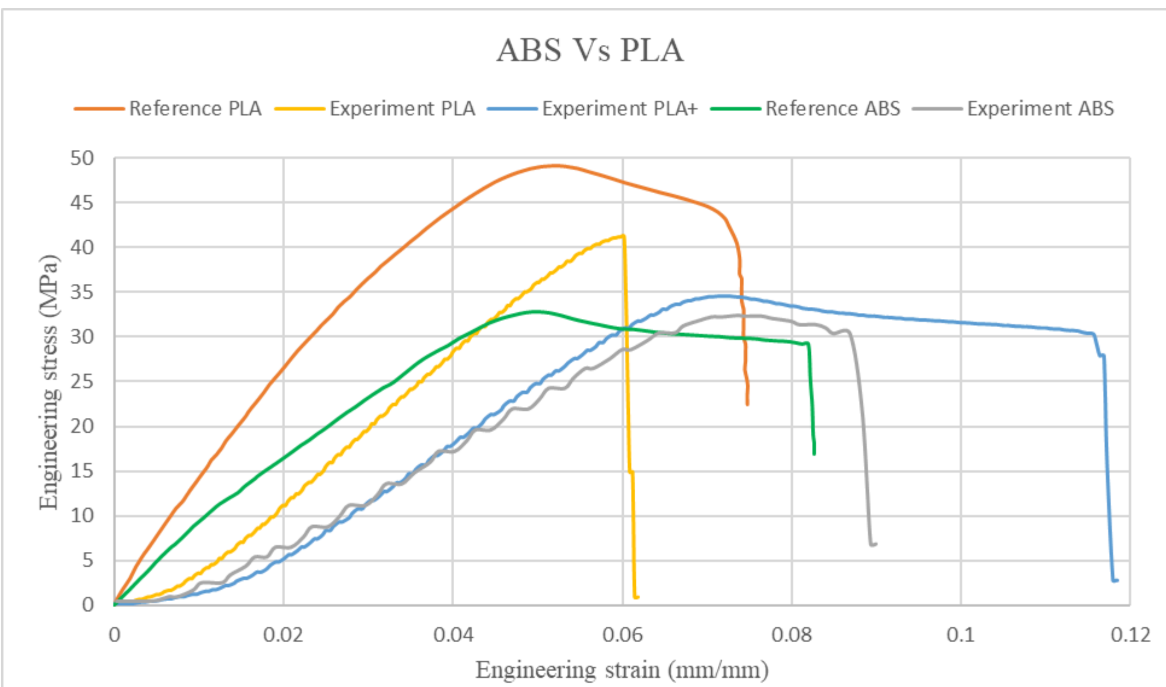


Figure 4.1 Overall Comparison of ABS and PLA

When compared with the PLA variants, PLA+ had a longer elastic limit, a longer plastic zone and a longer necking region. This meant that this modified PLA was much more flexible, tougher and ductile than its original, brittle material. Moreover, among the experiments' specimens, PLA was the most brittle specimen, while PLA+ was the most ductile. However, ABS of both the reference and the experiments showed moderate tensile properties.

Moreover, as seen from Section 4.2.1 and 4.2.2, it would take the largest tensile force to break down the specimen of PLA, followed by those of PLA+ and ABS. Generally, before fracture, ABS can tolerate around 32 N while PLA can tolerate between 40 to 50 N, and PLA+'s tolerance is about 34 N. Similarly, it would also take the largest energy per m^3 of the specimen to break down for PLA materials than ABS'.

From these findings, one can conclude that if the problem requires a high load bearing condition with high frequency, PLA+ is most suitable; if the problem requires low load bearing, the least processibility cost and the best biodegradability, PLA should be chosen, and if the problem requires the constraints of in-between, ABS should be the chosen material.

Chapter 5: Conclusion and Recommendation

This project simulated the uniaxial tensile test of ASTM D638 Type 1 Standard on ABS and PLA specimens using three 3D modelling and FEM software(s); SolidWorks, Abaqus and Autodesk NASTRAN. Also, the simulation procedures used in this project can be blueprints for beginner and amateur design engineers, who would like to perform finite element analysis on tensile tests.

The simulation of these software(s) was done in a non-linear static condition so as to imitate the real quasi-static tensile test condition. A comparison chart was drawn to compare the capabilities of the software(s) and it was found that although Abaqus cannot rival the 3D modeling abilities of SolidWorks and Autodesk Inventor, it held the first place in finite element analysis, followed by SolidWorks and then, Autodesk NASTRAN. Also, between the leading 3D modelling software(s), although Autodesk Inventor offered a more user-friendly interface and a smooth learning path for beginner engineers, its FEM capability was less powerful and inaccurate than that of SolidWorks.

Moreover, under the guidance of the supervisor, three practical tests were also performed to find out the difference between the reference paper and also to compare the mechanical behaviors of locally produced specimens with that of the reference. Maximum tensile force and fracture energy were derived from both experimental and reference's results. It was found that ABS specimen showed a similar property with that of the reference, while PLA showed a vast amount of difference.

For future recommendation, one can refine the comparison between three software(s) by conducting simulation in "non-linear dynamic (explicit)" condition. Also, the project's procedures can be stepping stones for FEM analysis on other ASTM standard tensile tests. All in all, by reading this project paper, one can clearly know that if an error-sensitive FEM analysis on a simple part or assemblies with simple constraints were needed, Abaqus is perfect to use; if a FEM analysis on assemblies with complex constraints were needed with a moderate level in 3D modelling, SolidWorks can be used, and if a beginner or amateur design engineering would like to use a FEM analysis with ease of CAD modelling, Autodesk Inventor embedded with NASTRAN module can be used.

References

1. Alharbi, M., Kong, I., & Patel, V. I. (2020). Simulation of uniaxial stress-strain response of 3D-printed polylactic acid by nonlinear finite element analysis. *Applied Adhesion Science, 8*(1), 1-10.
2. Azeez, A. T., & Mohammed, S. S. (2018). Solidworks Simulation of Mechanical Properties of Recycled Plastics/Nanocomposite Faces Sandwich Panels. *ARO- The Scientific Journal of Koya University, 6*(1), 65-70.
3. Baca Lopez, D. M., & Ahmad, R. (2020). Tensile mechanical behaviour of multi-polymer sandwich structures via deopsition modelling. *Polymers, 12*(3), 651.
4. Beer, F. P., Johnston, E. R., DeWolf, J. T., & Mazurek, D. F. (2015). *Mechanics of Materials. 7th_Edition*. New York: McGraw-Hill Education Ltd.
5. Durbacă, I., & Durbacă, A. C. (2018). THE FINITE ELEMENT ANALYSIS OF A POLYMER BASED TRIANGULAR CELL SANDWICH COMPOSITE. *International Conference on Advanced Materials and Systems (ICAMS)*, 379-384.
6. Farah, Shady, et al. (2016, December). Physical and Mechanical Properties of PLA, and Their Functions in Widespread Applications-A Comprehensive Review. *Advanced Drug Delivery Reviews, 107*, 367-92.
7. Garrell, M. G., Shih, A. J., Lara-Curzio, E., & Scattergood, R. O. (2003). Finite-element analysis of stress concentration in ASTM D 638 tension specimens. *Journal of testing and evaluation, 31*(1), 52-57.
8. Hussin, M. S., Hamat, S., Ali, S. A. S., Fozi, M. A. A., Rahim, Y. A., Dawi, M. S. I. M., & Darsin, M. (2020, October). Experimental and finite element modeling of partial infill patterns for thermoplastic polymer extrusion 3D printed material using elasto-plastic method. *AIP Conference Proceedings, 2278*(1), 020011.
9. Japrin, S. and Ibrahim, M. (2021). Fused Deposition Modeling (FDM) Custom Internal Filling Structural Mechanics Simulation. *Research Progress in Mechanical and Manufacturing Engineering, 2*(2), 334-342.
10. Joun, M., Choi, I., Eom, J., & Lee, M. (2007). Finite element analysis of tensile testing with emphasis on necking. *Computational Materials Science, 41*(1), 63-69.
11. Latifi, N., Miri, A. K., & Mongeau, L. (2014). Determination of the elastic properties of rabbit vocal fold tissue using uniaxial tensile testing and a tailored finite element model. *Journal of the mechanical behavior of biomedical materials, 39*, 366-374.
12. Magomedov, I. A., & Sebaeva, Z. S. (2020, April). Comparative study of finite element analysis software packages. *Journal of Physics: Conference Series, 1515*(3), 032073.
13. Nirbhay, M., Dixit, A., Misra, R. K., & Mali, H. S. (2014). Tensile test simulation of CFRP test specimen using finite elements. *Procedia Materials Science, 5*, 267-273.
14. Paul, S. (2021). Finite element analysis in fused deposition modeling research: A literature review. *Measurement*, 178.

15. Petrik, A., & Aroch, R. (2019, June). Usage of true stress-strain curve for FE simulation and the influencing parameters. *IOP Conference Series: Materials Science and Engineering, 566*(1), 012025.

16. Román, L. A., Ardila, J. G., Valdes, M., Castro, A., & Quintero, J. L. (2020). Numerical study of the stress concentration factor in geometries of machine elements using Ansys® and Inventor®. *Journal of Physics: Conference Series, 1448*(1), 012004.

17. Rovlance, D. (2001). *Finite element analysis.* Massachusetts Institute of Technology, Cambridge: Department of Materials Science and Engineering.

18. Shahrubudin, N., Lee, T. C., & Ramlan, R. (2019). An overview on 3D printing technology: Technological, materials, and applications. *Procedia Manufacturing, 35*, 1286-1296.

19. Żur, P., Kołodziej, A., & Baier, A. (2019). Finite element analysis of PLA 3d-printed elements and shape optimization. *European Journal of Engineering Science and Technology, 2*(1), 59-64.

Appendix [illegible] (2020)

Elongation (mm)	Reaction Force (N)	Elongation (mm)	Reaction Force (N)
0	20.60106174	3.347511778	1336.12207
0.016906811	20.60106174	3.402167599	1336.12207
0.09[illegible]	[illegible]	[illegible]	[illegible]
0.2[illegible]	[illegible]	[illegible]	1423.43103
0.287549615	44.14500046	3.6[illegible]	1423.43103
0.370343924	44.14500046	3.727936459	1459.728027
0.427417579	78.48000336	3.82403898	1488.177002
0.522899141	117.7200089	3.907145598	1506.81604
0.590444744	117.7200089	4.023062706	1506.81604
0.727617443	158.9220123	4.089842319	1514.664063
0.784181654	204.0480042	4.193915367	1510.73999
0.887585342	252.1170197	4.253557682	1510.73999
0.947227538	252.1170197	4.361912727	1499.948975
1.029754102	305.1285978	4.425123692	1484.253052
1.086586118	305.1285978	4.557989259	1466.594971
1.192755461	356.1030274	4.614480079	1466.594971
1.279400601	410.[illegible]	4.7[illegible]	1449.918091
1.3[illegible]	[illegible]	[illegible]	1419.50708
1.440781592	465.975[illegible]	4.8[illegible]	1419.50708
1.520988584	521.8920288	4.95480999	1011.411011
1.581522942	521.8920288	5.027694454	319.8059998
1.693090081	578.7900391	5.094660282	319.8059998
1.775795102	634.7070313	5.129723072	319.8059998
1.832805514	634.7070313	5.129723072	319.8059998
1.938707232	691.6050415		
2.021233797	748.5030518		
2.130079746	804.4200439		
2.184681177	804.4200439		
2.293036461	860.3370361		
2.376009226	916.2540283		
2.453014638	916.2540283		
2.536195704	971.1900[illegible]		
2.6[illegible]	[illegible]		
2.68[illegible]	[illegible]		
2.790292978	1080.084005		
2.867823601	1134.026011		
2.926752099	1134.026011		
3.035285711	1186.029053		
3.092296124	1238.022095		
3.172369242	1238.022095		
3.226346016	1288.052979		

Appendix F: Experimental PLA's data from the second test

Elongation (mm)	Reaction Force (N)	Elongation (mm)	Reaction Force (N)	Elongation (mm)	Reaction Force (N)
0	9.81000042	1.267564774	245.25	2.59138155	677.8710327
0.015345497	9.81000042	1.307980537	258.0029907	2.614310503	693.5670166
0.066422805	9.81000042	1.339742184	258.0029907	2.655663013	709.2630615
0.089530267	11.77200031	1.362626553	270.756012	2.677521467	709.2630615
0.121202655	11.77200031	1.403889894	284.4900208	2.717981815	723.9780273
0.145380735	15.69600105	1.429406285	284.4900208	2.749609709	738.6929932
0.185885713	20.60100174	1.470000505	298.2240295	2.771423578	738.6929932
0.217647314	20.60100174	1.501583576	311.9580078	2.803006649	753.40802
0.24289602	26.48700333	1.543203831	326.6730347	2.823749781	753.40802
0.283445597	32.3730011	1.563991666	326.6730347	2.865280867	768.1230469
0.305304021	32.3730011	1.615381241	341.3880005	2.88941431	781.8570557
0.35749653	39.24000168	1.638310194	356.1030273	2.939956188	795.5910034
0.380470186	49.05000305	1.678681374	369.8370056	2.963108301	795.5910034
0.420038372	58.86000443	1.699469209	369.8370056	3.005352974	808.3440552
0.445242465	58.86000443	1.73203373	385.53302	3.027166843	821.0970459
0.477048665	68.66999817	1.753535271	385.53302	3.058839321	821.0970459
0.498817861	68.66999817	1.795200109	400.2480164	3.079537868	832.8690186
0.539367437	78.48000336	1.825311184	414.9630432	3.120934963	844.6409912
0.560244441	87.30900574	1.847927928	414.9630432	3.152518272	844.6409912
0.592585981	87.30900574	1.889458895	429.6780396	3.176517725	855.4320068
0.615470409	95.15700531	1.911272645	444.3930054	3.216175318	866.2230225
0.657090604	103.0050049	1.941740632	444.3930054	3.238925934	866.2230225
0.679082811	103.0050049	1.962573051	460.0890198	3.270375252	875.052063
0.720658422	111.8340073	2.004906893	476.7660217	3.311861515	882.9000244
0.752464652	121.6440048	2.027702093	476.7660217	3.332738638	882.9000244
0.774233818	121.6440048	2.077218294	494.4240112	3.374135733	889.7669678
0.813802004	131.45401	2.099790335	511.1010132	3.405852795	894.6720581
0.846545041	141.2640076	2.151135206	528.7590332	3.428692579	894.6720581
0.869340241	141.2640076	2.17299366	528.7590332	3.46808219	897.6150513
0.910915852	151.0740051	2.213409424	546.4170532	3.50064683	898.5960083
0.940491617	161.8650055	2.245126486	563.0940552	3.519650221	898.5960083
0.963019133	161.8650055	2.266850948	563.0940552	3.561002731	875.052063
1.004550099	173.6370087	2.307222128	579.7710571	3.581969023	575.8470459
1.026274681	186.3899994	2.338983774	596.4480591	3.613552332	575.8470459
1.056831837	186.3899994	2.361912727	596.4480591	3.635187626	149.1120148
1.077619672	198.1620178	2.403443813	613.125061	3.666904688	149.1120148
1.120132089	208.9530029	2.425123692	629.802063	3.666904688	149.1120148
1.141901255	208.9530029	2.465807199	646.4790039	3.666904688	149.1120148
1.173618197	220.7250061	2.497479677	646.4790039	3.666904688	149.1120148
1.194539905	220.7250061	2.519337893	662.1750488		
1.245840192	232.4970245	2.559664488	677.8710327		

Appendix G: Experimental PLA's data from the third test

Elongation (mm)	Reaction Force (N)	Elongation (mm)	Reaction Force (N)	Elongation (mm)	Reaction Force (N)	Elongation (mm)	Reaction Force (N)
0	9.81000042	0.979301453	374.7420044	1.9726547	1028.088135	2.94910121	1607.859131
0.004104028	9.81000042	0.998706341	393.381012	1.992594957	1045.745972	2.974260569	1621.593018
0.029174287	9.81000042	1.012089014	393.381012	2.005843878	1045.745972	2.993665457	1636.308105
0.048668422	10.79100037	1.036400914	411.0390015	2.036713123	1064.38501	3.007583618	1636.308105
0.062051121	10.79100037	1.056252003	428.6970215	2.049471378	1082.043091	3.033010721	1650.041992
0.086853728	13.73400116	1.070348382	428.6970215	2.069545507	1082.043091	3.04634881	1663.776001
0.106347859	19.62000084	1.095909357	447.3360291	2.085114002	1100.682007	3.071062088	1663.776001
0.125886604	19.62000084	1.109381318	465.9750061	2.110496521	1118.340088	3.090466976	1677.510132
0.14136593	25.50600052	1.134808421	484.6140137	2.123790026	1118.340088	3.103715897	1690.263062
0.16621314	30.41100121	1.154213309	484.6140137	2.149306297	1136.979004	3.128518581	1702.035156
0.180398807	30.41100121	1.168265104	503.2530212	2.168711185	1154.636963	3.147968054	1702.035156
0.20658429	36.29700089	1.19253242	521.8920288	2.182049274	1154.636963	3.161395311	1713.807007
0.225944594	42.18300247	1.206495047	521.8920288	2.206227303	1172.295044	3.186733246	1725.579102
0.239193469	42.18300247	1.225766182	540.5310059	2.226123095	1190.934082	3.2055583	1725.579102
0.264174521	48.06900406	1.241022468	540.5310059	2.240888596	1190.934082	3.219743967	1736.370117
0.283668637	53.95500565	1.265914321	559.1700439	2.266315699	1208.592041	3.245750904	1746.180054
0.299281806	53.95500565	1.280099869	577.809021	2.279073954	1226.250122	3.265155792	1746.180054
0.324173629	60.82200241	1.312039971	596.4480591	2.304277897	1243.907959	3.279073954	1755.009033
0.336842567	67.68900299	1.325333476	596.4480591	2.323727608	1243.907959	3.303787231	1762.857056
0.369050264	74.55600739	1.350359082	615.0870361	2.337065697	1261.56604	3.32319212	1762.857056
0.382432967	74.55600739	1.370388508	634.7070313	2.361288309	1279.223999	3.336485624	1769.723999
0.406923324	84.36600494	1.385243297	634.7070313	2.381228447	1279.223999	3.361154556	1776.591064
0.426818937	95.15700531	1.410759687	653.3460693	2.395191193	1296.88208	3.373867989	1776.591064
0.441495299	95.15700531	1.423562527	671.9849854	2.42066288	1314.540039	3.393808365	1781.496094
0.467056245	106.9290009	1.449034214	691.6050415	2.438907862	1314.540039	3.407191038	1781.496094
0.480483562	118.701004	1.468439102	691.6050415	2.452870607	1332.197998	3.431324482	1784.439087
0.505821466	131.45401	1.481732607	711.2250366	2.478208542	1348.875122	3.444082499	1488.177002
0.525092542	131.45401	1.505955338	729.8640137	2.497658014	1348.875122	3.46928668	646.4790039
0.539144397	144.2070007	1.52594018	729.8640137	2.511888266	1366.533081	3.488646984	646.4790039
0.563634753	156.9600067	1.539278269	747.5220337	2.537359953	1383.210083	3.502698898	42.18300247
0.583574951	156.9600067	1.56479454	766.1610107	2.550698042	1383.210083	3.525270939	42.18300247
0.597626805	170.6940002	1.582905889	766.1610107	2.575946808	1399.886963	3.525270939	42.18300247
0.623053908	184.428009	1.598251343	784.8000488	2.594102621	1416.564087	3.525270939	42.18300247
0.635945916	184.428009	1.62372303	803.4390259	2.608109951	1416.564087		
0.65530622	199.1430054	1.636971951	803.4390259	2.63349247	1433.240967		
0.668599725	199.1430054	1.661283851	823.059082	2.64674139	1449.918091		
0.694071472	212.8769989	1.681179404	841.6980591	2.665432453	1449.918091		
0.707409561	228.572998	1.694517612	841.6980591	2.678235292	1466.594971		
0.725654662	228.572998	1.720078468	860.3370361	2.704153061	1483.272095		
0.741847694	244.2690125	1.732881308	878.9760742	2.718160391	1483.272095		
0.767319441	259.9650269	1.752241611	878.9760742	2.742918253	1498.968018		
0.780612946	259.9650269	1.766248822	897.6150513	2.762233973	1515.64502		
0.811616182	275.6610107	1.791631341	916.2540283	2.775527477	1515.64502		
0.825668037	291.3569946	1.805727839	916.2540283	2.807110786	1531.341064		
0.85109514	307.053009	1.836731076	934.8930664	2.820538044	1547.037109		
0.869786322	307.053009	1.850693703	953.5320435	2.845831394	1561.752075		
0.882410645	323.730011	1.876120806	972.1710205	2.859169483	1561.752075		
0.908417702	340.4070129	1.894410491	972.1710205	2.878574371	1577.447998		
0.921755791	340.4070129	1.909711361	990.8100586	2.891867876	1577.447998		
0.946603	358.0650024	1.935049295	1009.449036	2.916625738	1592.163086		
0.96591872	374.7420044	1.9483428	1009.449036	2.935852289	1607.859131		

Appendix H: Experimental PLA+'s data from the third test

Elongation (mm)	Reaction Force (N)	Elongation (mm)	Reaction Force (N)	Elongation (mm)	Reaction Force (N)	Elongation (mm)	Reaction Force (N)
0	9.81000042	2.008252621	637.6500244	4.014676571	1492.101074	6.037650108	1351.818115
0.006066824	9.81000042	2.029085159	659.2320557	4.052638531	1495.044067	6.074853897	1349.856079
0.052772451	9.81000042	2.067359686	680.8140259	4.072623253	1496.025146	6.09399128	1349.856079
0.081902131	9.81000042	2.09626627	680.8140259	4.108979702	1496.025146	6.123789787	1347.894043
0.101797745	10.79100037	2.117589235	703.3770752	4.138733864	1496.025146	6.143819332	1347.894043
0.129901409	10.79100037	2.155685425	725.9400635	4.162153721	1494.06311	6.181826115	1345.932007
0.149038672	13.73400116	2.175447226	725.9400635	4.200160503	1491.119995	6.202970982	1343.969971
0.18802695	17.65800095	2.213587999	747.5220337	4.22010088	1491.119995	6.241022587	1342.008057
0.207967162	17.65800095	2.240844011	770.085022	4.25806284	1486.215088	6.269928932	1342.008057
0.236963019	21.58200073	2.261631727	770.085022	4.287103653	1481.310059	6.291876793	1339.065063
0.258286119	21.58200073	2.299638748	792.6480713	4.307043552	1481.310059	6.328188419	1337.103027
0.296382219	25.50600052	2.328634501	814.2300415	4.343221664	1476.405029	6.357898235	1337.103027
0.316277832	29.43000221	2.349779129	814.2300415	4.373109818	1470.519043	6.37770462	1334.160034
0.354507744	33.35400009	2.387741327	836.7930908	4.393139362	1470.519043	6.424053192	1330.236084
0.381897658	33.35400009	2.416826487	858.375	4.430967331	1464.633057	6.445331573	1330.236084
0.402685463	38.25900269	2.438551188	858.375	4.458402157	1458.74707	6.484186172	1327.293091
0.440870762	44.14500046	2.474862814	880.9381104	4.480260372	1458.74707	6.506401539	1322.388062
0.469955832	44.14500046	2.495739937	902.5200195	4.518267155	1452.861084	6.543471336	1316.502075
0.489985287	49.05000305	2.533612967	924.1020508	4.547263145	1447.956055	6.572378159	1316.502075
0.528170586	55.91700363	2.555828094	924.1020508	4.568497181	1447.956055	6.591783047	1306.692139
0.557166457	55.91700363	2.602221489	945.684082	4.606459618	1443.051025	6.628674507	1207.611084
0.577017426	61.8030014	2.622072458	967.2659912	4.626399517	1438.146118	6.656778336	1207.611084
0.613373756	69.65100098	2.660124063	988.8480225	4.672480583	1433.240967	6.677923203	683.757019
0.63425082	69.65100098	2.688272238	988.8480225	4.691528797	1433.240967	6.716732979	122.625
0.663202047	77.49900818	2.709104776	1010.430054	4.739706516	1428.33606	6.736628532	122.625
0.683186889	77.49900818	2.748003721	1031.031006	4.759512901	1424.412109	6.74715643	122.625
0.720569193	87.30900574	2.776865721	1031.031006	4.795735359	1420.488037	6.74715643	122.625
0.749475837	98.1000061	2.797787428	1052.613037	4.825667858	1420.488037		
0.769549906	98.1000061	2.835080624	1073.214111	4.846500397	1417.545044		
0.807779789	108.8910065	2.864299297	1073.214111	4.884775162	1413.621094		
0.828121543	120.663002	2.884150505	1093.815063	4.903912067	1413.621094		
0.874381065	133.4160004	2.921354294	1114.416016	4.932729721	1410.677979		
0.894410491	133.4160004	2.940491676	1114.416016	4.953651428	1410.677979		
0.932372749	148.131012	2.970379591	1135.01709	4.991791725	1406.75415		
0.959762692	162.8460083	2.991390467	1135.01709	5.011732101	1403.811035		
0.989739954	162.8460083	3.02953124	1155.618042	5.038898945	1403.811035		
1.027746797	178.5420074	3.0566535	1175.238037	5.061962128	1400.868042		
1.049872875	195.2190094	3.095552444	1194.858032	5.100013256	1398.906006		
1.087433696	211.8960266	3.115581989	1194.858032	5.119864464	1398.906006		
1.108399868	211.8960266	3.152696609	1214.478027	5.156666756	1395.963135		
1.146540523	229.5540009	3.181781769	1234.098022	5.186599255	1393.02002		
1.174822688	247.2120056	3.202837229	1234.098022	5.206361294	1393.02002		
1.193915367	247.2120056	3.240844011	1252.737061	5.244278908	1391.058105		
1.2329036	265.8510132	3.261765718	1271.376099	5.263371468	1389.096069		
1.261721015	284.4900208	3.28902173	1271.376099	5.292233467	1389.096069		
1.281571984	284.4900208	3.309898853	1290.015015	5.312084675	1386.153076		
1.318775892	304.1100159	3.34795022	1307.673096	5.350269794	1384.19104		
1.347905636	323.730011	3.376990795	1307.673096	5.378462791	1384.19104		
1.367801189	323.730011	3.395994186	1325.331055	5.399696827	1382.229126		
1.405005097	344.3309937	3.434090137	1342.008057	5.438729763	1380.26709		
1.424186945	363.9510193	3.463130713	1342.008057	5.458669662	1380.26709		
1.454030395	363.9510193	3.484097004	1358.685059	5.495695114	1378.305054		
1.476156473	384.5520325	3.520364046	1375.362061	5.524780273	1376.343018		
1.514297128	405.1530151	3.541240931	1375.362061	5.545746326	1376.343018		
1.534415841	405.1530151	3.57987237	1390.077026	5.583753586	1374.381104		
1.581567526	425.7539978	3.608778954	1404.792114	5.611812592	1372.419067		
1.601507783	446.3550415	3.630102158	1404.792114	5.63077116	1372.419067		
1.638622522	467.9370117	3.668287516	1419.50708	5.669670105	1369.476074		
1.669045806	467.9370117	3.688183069	1432.26001	5.698710918	1367.514038		
1.690235138	488.5380249	3.717000484	1432.26001	5.71976614	1367.514038		
1.728331208	510.1200256	3.736940622	1444.031982	5.756969929	1365.552002		
1.749431252	510.1200256	3.77392149	1455.804077	5.786010742	1363.590088		
1.786768913	530.7210693	3.802872896	1455.804077	5.824061871	1361.628052		
1.815764785	552.3030396	3.822857618	1465.614014	5.843065739	1361.628052		
1.835705042	552.3030396	3.85988307	1474.443115	5.88093853	1360.647095		
1.873042822	573.8850098	3.878841877	1474.443115	5.909800529	1358.685059		
1.902127862	594.4860229	3.908819199	1482.291016	5.930767059	1358.685059		
1.923183322	594.4860229	3.928714752	1482.291016	5.967033863	1355.742065		
1.961234808	616.0679932	3.966498613	1488.177002	5.988936901	1353.780029		
1.981174946	637.6500244	3.993799448	1492.101074	6.017754555	1353.780029		

Addendum

CHINDWIN-PSB INSTITUTE

DL Individual Project Proposal/Agreement Form

STUDENT NAME: - **Phone Thet Oo** **STUDNET NUMBER: -** 2149426

COURSE: - BEng (HONS) Mechanical Engineering (International) **DATE:** 10/7/2022

PROJECT TITLE (initial working title)	Comparison of experimental result and FEM analysis of ABS and PLA on ASTM D638
BACKGROUND A brief introduction to the topic.	In recent years, engineers have been doing massive research on Finite Element Method in simulating the mechanical behaviours of materials under conditions almost exactly the same as the real environments. By using certain FEA modelling softwares, engineers can save time, money and energy in investigating the mechanical properties of desired materials. Thermoplastics, such as PLA and ABS have dominated the 3D printing market for a century and it is important to know these materials' tensile properties according to ASTM standards. Herein, well-known simulation softwares such as Abaqus©, Solidworks© and Autodesk Nastran© can simulate the plastics' behaviours on ASTM D638 Type 1, the result of which can be useful for designers and manufacturers.
AIM Describe the general aim of your project (e.g. investigate ways of catching mice).	To perform finite element analysis on ABS and PLA plastics, and compare with experimental results of ASTM D638 Type 1 to see the optimized FEM software(s) with the least error percentage in order to suggest the most efficient one.
OBJECTIVES Describe how you hope to achieve the aim of the project (e.g. literature review; feasibility study, conduct interviews, write a questionnaire, working prototype, experimental plans etc....)	1. To study the mechanical properties of ABS and PLA by literature review 2. To learn tensile test on FEA softwares 3. To analyse the tensile test simulation on FEA softwares 4. To evaluate the simulation results with the experiment 5. To suggest the best software with the least percentage
PLAN (provisional) Breakdown of the objectives in a timescale.	1. From the introductory month of July to the end of August, literature review will be done by researching previous papers regarding the simulation of the plastics. 2. Simultaneously, simulation techniques in FEA softwares; Abaqus©, Solidworks© and Autodesk Nastran© will be learnt. 3. By using the aforementioned knowledge, simulation in each software will be analysed for each month of September to November. 4. Once simulating in each software is done, evaluation of results in accordance with the experiment's will be done. The techniques, steps and methods will be recorded accordingly. 5. By the end of December, the best simulation software with the least error percentage and the best FEA results will be obtained and suggested.

ETHICS Consideration and clearance	Have you completed the fast track web review? http://ethicsreview.port.ac.uk Please add your **unique** Ethics Clearance Certificate Code No: - TETHIC-2022-103493
Health and Safety Are there any Health and Safety issues associated with any of the practical work associated with your project?	No

PROJECT SUPERVISOR NAME: Dr. Thu Han Tun **SIGNATURE**	**STUDENT NAME: Phone Thet Oo** **SIGNATURE:** ***Phone***

UNIVERSITY OF PORTSMOUTH

Certificate of Ethics Review

Project title: Comparison of experimental result and FEM analysis of ABS and PLA on ASTM D638

Name:	Phone Thet Oo	**User ID**:	2149426	**Application date**:	06/07/2022 16:09:15	**ER Number**:	TETHIC-2022-103493

You must download your referral certificate, print a copy and keep it as a record of this review.

The FEC representative(s) for the **School of Mechanical and Design Engineering** is/are **Giles Tewkesbury, Jovana Radulovic**

It is your responsibility to follow the University Code of Practice on Ethical Standards and any Department/School or professional guidelines in the conduct of your study including relevant guidelines regarding health and safety of researchers including the following:

- University Policy
- Safety on Geological Fieldwork

It is also your responsibility to follow University guidance on Data Protection Policy:

- General guidance for all data protection issues
- University Data Protection Policy

Which school/department do you belong to?: **School of Mechanical and Design Engineering**
What is your primary role at the University?: **Undergraduate Student**
What is the name of the member of staff who is responsible for supervising your project?: **Dr. Thu Han Tun**
Is the study likely to involve human subjects (observation) or participants?: No
Will financial inducements (other than reasonable expenses and compensation for time) be offered to participants?: No
Are there risks of significant damage to physical and/or ecological environmental features?: No
Are there risks of significant damage to features of historical or cultural heritage (e.g. impacts of study techniques, taking of samples)?: No
Does the project involve animals in any way?: No
Could the research outputs potentially be harmful to third parties?: No
Could your research/artefact be adapted and be misused?: No
Will your project or project deliverables be relevant to defence, the military, police or other security organisations and/or in addition, could it be used by others to threaten UK security?: No

I confirm that I have considered the implications for data collection and use, taking into consideration legal requirements (UK GDPR, Data Protection Act 2018 etc)
I confirm that I have considered the impact of this work and and taken any reasonable action to mitigate potential misuse of the project outputs
I confirm that I will act ethically and honestly throughout this project

Supervisor Review

As supervisor, I will ensure that this work will be conducted in an ethical manner in line with the University Ethics Policy.

Supervisor's signature: [signature] Date: 06/07/2022

Interim Report Form (DL)

CHINDWIN-PSB INSTITUTE

E6H3002 Individual Project
PROGRESS REPORT FORM (DL)

NAME OF STUDENT: **Phone Thet Oo**

Student Number: 2149426

NAME OF SUPERVISOR: **Dr. Thu Han Tun** **DATE:** **9-10-2022**
PROJECT TITLE: Comparison of experimental result and FEM analysis of ABS and PLA on ASTM D638

REVIEW OF PROGRESS TO DATE:

So far from the meeting of the supervisor, several tasks had been accomplished. From the filtering of various FEM software [ANSYS, COMSOL], three software; Abaqus, SOLIDWORKS and Autodesk NASTRAN, were chosen. Abaqus, being one of the top FEA software, was simulated and the latter two, being the top 3d modelling software, had been compared to find out that modelling software are just as capable as the FEA counterpart. Furthermore, about twenty papers had been reviewed, which, in turn, provided knowledge on how to efficiently simulate the results similar to the real test. After a third meeting with the supervisor, an experimental test was planned to carry out.

ACTIONS/WORK TO BE COMPLETED BEFORE END OF PROJECT PERIOD

Until the end of December, which was planned to be the end of the project, three of each software will simulate for three weeks. At the end of each software, respective optimized results will be obtained. The experimental test was planned to do on the second week of October. After the simulation, all results from the reference paper, the experiment and three software will be compared and evaluated. By December, all data will be ready and preparing on the presentation and the final report will be carried out.

TIMEFRAME/PROJECT PLAN (please refer to project handbook)

	October				November				December			
Tasks\Week	1	2	3	4	1	2	3	4	1	2	3	4
Experimental test		■										
SOLIDWORKS simulation		■	■	■								
Abaqus simulation				■	■	■						
Autodesk NASTRAN simulation						■	■	■				
Evaluation								■	■	■		
Preparing presentation										■	■	■

Signature of Supervisor:..

E6H3002 Individual Project
PROJECT MONITORING FORM

Student: Phone Thet Oo	Student Number:
Supervisor: Dr. Thu Han Tun	**2149426**
Title: Comparison of experimental result and FEM analysis of ABS and PLA on ASTM D638	
Award: BEng (HONS) Mechanical Engineering (International)	Year: 2022-2023

Note Students should consult their supervisors and keep them informed about the progress of their work.

Date and Consultation Time	Topic	Comments by Supervisor	Supervisors Sign	Student Sign
9/6/2022 21:00 – 22:00	-First meeting on Microsoft Team -Initial discussion for the project proposal	-Study the simulation method for COMSOL whether the tensile test is available for composite polymers		*Phone*
6/7/2022 21:00-22:00	-Second meeting on Microsoft Team -Discussion for the project proposal with ethic review. -Discussion about tested FEM results	-To continue on the current progress		*Phone*
30/9/2022 19:00-20:00	-Third meeting on Microsoft Team -Discussion for the project format: Introduction, Literature review, Methodology -Discussed to do the experiment	-Prepare for the specimen to be experimented		*Phone*
13/10/2022 13:00-15:00	- First experiment on ABS and PLA at Myanmar Maritime University, Thanlyin	-Need dimension changes as the fracture area was not in the expected area		*Phone*

CDC/Form/Project Monitoring Form V-1.0

Date and Consultation Time	Topic	Comments by Supervisor	Supervisors Sign	Student Sign
26/11/2022 20:00-21:30	- Fourth meeting on Microsoft Team - Discuss amendments in the research paper with the supervisor -Discuss dimension changes for the experiment's specimen	-Confirmation for specimen's dimensions and the paper's format changes and addition of "Theoretical background" in the paper		*Phone*
9/12/2022 13:00-15:00	-The second experiment at Myanmar Maritime University, Thanlyin	-Did the experiment with the changed dimensions on ABS and PLA -Got good results on ABS but bad ones on PLA - Need another test for PLA		*Phone*
11/12/2022 20:00-21:00	-Fifth meeting on Microsoft Team -Discussed amendments for the draft report	-Amend figures' captions' positions -Make changes for the draft report		*Phone*
18/12/2022 13:00-15:00	-The final experiment at Myanmar Maritime University, Thanlyin -The experiment was on PLA and PLA+	-The results were OK		*Phone*
3/1/2022 20:30-21:30	-Sixth meeting on Microsoft Team -Discussed for the final report	- Finalize the final report		*Phone*

Project Progress Summary

Week	Date	Project Progress	Feedback from Supervisor	Supervisor Signature	Student Signature
1	9/6/2022	- First meeting with the supervisor at Microsoft Team - Initial discussion for the project proposal	- Study the simulation method for COMSOL whether the tensile test is available for composite polymers		*Phone*
2	12/6/2022	- Initiate the study of COMSOL on composite materials - Finished reading Tensile mechanical behavior of multi-polymer sandwich structures via fused deposition modelling by Baca Lopez (2020)	- Continue the software(s) testing and literature review		*Phone*
3	17/6/2022	- Referring the data by Baca Lopez (2020), the proposal is to compare steps/time taken/accuracy between 1. SOLIDWORKS 2. AUTODESK INVENTOR 3. COMSOL 3.1 COMSOL live link INVENTOR 4. ANSYS 4.1 ANSYS with SOLIDWORKS 5. NASTRAN	Continue the software(s) testing		*Phone*
4	18/6/2022 To 5/7/2022	- The apparent proposed idea was to simulate the tensile behaviors of Polylactide (PLA) and Acrylonitrile Butadiene styrene (ABS), which are easy to be tested locally - Regarding my expertise, availability and capability, among the mentioned software(s), SolidWorks, Ansys and Autodesk NASTRAN were chosen and tested for the scenario of Baca Lopez (2020)'s paper	Continue the software(s) testing		*Phone*
5	6/7/2022	-Second meeting at Microsoft Team -Discussion for the project proposal with ethic review -Discussion about tested FEM results	-To continue on the current progress		*Phone*

CDC/Project/Project Progress Summary V-1

6	7/7/2022 To 31/7/2022	-Learned different FEA software including Abaqus and Autodesk Nastran -True stress-strain curves implementation in SOLIDWORKS was discovered -Read a few papers such as "Simulation of uniaxial stress-strain response of 3D-printed polylactic acid by nonlinear finite element analysis" by Alharbi (2020) and "Finite element analysis in fused deposition modeling research: A literature review" by Paul (2021)	To consider about the experiment Write the literature review.		*Phone*
7	1/9/2022 To 30/9/2022	-Restart the project by focusing on literature review. -The previous month was skipped to focus on the first semester exam. -The topics of literature include: - "Comparative study of finite element analysis software packages" by Magomedov (2020), - "Usage of true stress-strain curve for FE simulation and the influencing parameters" by Petrik (2019), -The rest of the FEA analysis of past papers -In the second week, 3D models were created using the chosen software(s)	-To continue on the current progress		*Phone*
8	30/9/2022 19:00-20:00	-Third meeting at Microsoft Team -Discussion for the project format: Introduction, Literature review, Methodology -Discussed to do the experiment	-Prepare for the specimen to be experimented		*Phone*
9	1/10/2022 To 12/10/2022	-Finished Literature review -Optimal simulation method on SolidWorks was tried and tested, which was 5000N tensile traction force on one surface and fixed at one end -Comparison between non-linear static (force control) and (displacement control) was performed -Report writing was started with "Introduction" and "Literature Review"	Should study on methodology.		*Phone*
10	13/10/2022 13:00-15:00	- First experiment on ABS and PLA	-Need dimension changes as the fracture area was not in the expected area		*Phone*
11	14/10/2022 To 31/10/2022	- SolidWorks simulation was done and all techniques were recorded in the report with error percentages - Initiate Abaqus simulation in the last week	-To continue on the current progress		*Phone*

12	1/11/2022 To 17/11/2022	-Abaqus simulation was done and all techniques were recorded in the report with error percentages -The result was much more reliable than SolidWorks -Initiate Autodesk NASTRAN simulation in the second week	-To continue on the current progress		*Phone*
13	18/11/2022 To 25/11/2022	-NASTRAN simulation was done and all techniques were recorded in the report with error percentages - The report until was tidied up and prepared ready for evaluation	Write result and discussion		*Phone*
14	26/11/2022 20:00-21:30	- Fourth meeting at Microsoft Team - Discuss amendments in the research paper with the supervisor -Discuss dimension changes for the experiment's specimen	-Confirmation for specimen's dimensions and the paper's format changes and addition of "Theoretical background" in the paper.		*Phone*
15	1/12/2022 To 8/12/2022	-The supervisor's suggestion was followed and the report was amended -Evaluation with certain criteria was performed on software(s)' performance -The performance being excellent from Abaqus followed by SolidWorks to Autodesk NASTRAN	-To continue on the current progress		*Phone*
16	9/12/2022 13:00-15:00	-The second experiment	-Did the experiment with the changed dimensions on ABS and PLA -Got good results on ABS but bad ones on PLA - Need another test for PLA		*Phone*
17	11/12/2022 20:00-21:00	-Fifth meeting -Discussed amendments for the draft report	-Amend figures' captions' positions -Make changes for the draft report		*Phone*
18	12/12/2022	-Changes required for the draft report was amended and submitted	Check the amendment.		*Phone*
19	18/12/2022 13:00-15:00	-The final experiment -The experiment was on PLA and PLA+	-The results were OK		*Phone*
20	23/12/2022	-Submitted draft report was discussed with the supervisors and amendments were suggested	-To include more calculation and comparison between ABS and PLA		*Phone*
21	3/1/2022 20:30-21:30	-Sixth meeting -Discussed for the final report	- Finalize the final report		*Phone*
22	4/1/2022	-The final report was completed	Well done.		*Phone*

CDC/Project/Project Progress Summary V-1

List of Achievements

Project Achievements

1. I designed a specimen of ASTM D638-1 myself on SolidWorks, Abaqus and Autodesk Inventor.
2. I managed to set up the tensile test using a uniaxial Testometric brand tensile testing machine.
3. I gathered simulation techniques for the mentioned FEM software(s) on the tensile test in one place.
4. I calibrated the simulation capability and error percentages of the software(s).
5. I compared the tensile behaviors of focused thermoplastic materials; Acrylonitrile Butadiene styrene (ABS), Polylactide (PLA) and PLA+.
6. I managed to find out the true stress-strain relationships of the thermoplastic materials.

Learning Achievements

1. I learned to 3D model a certain sketch using SolidWorks, Abaqus and Autodesk Inventor.
2. I learned to simulate a tensile test using the FEM modules of the mentioned software(s).
3. I learned to trace certain data points from graphs using WebPlotDigitizer©.
4. I learned to calculate true stress and strain values from extracted data using Microsoft Excel.
5. I learned how to manage time for working with the supervisor, report submission and at monthly meetings.
6. I learned the working procedures of a tensile testing machine, how to get raw data from it and how to calibrate these data to obtain desired stress-strain values.