Finite element modelling and evaluation of mechanical properties of wood-PLA parts manufactured through Fused Filament Fabrication

J Durlan1, B Adrover1*, M D Zandi1, J Lluma2, R Jerez3 and J A Travieso1

1 Universitat Politècnica de Catalunya, Department of Mechanical Engineering, Barcelona 08019, Spain
2 Universitat Politècnica de Catalunya, Department of Material Science and Engineering, Barcelona 08019, Spain
3 Universitat de Vic Universitat Central de Catalunya, Department of Science and Engineering, Vic, Spain

*Corresponding author: barbara.adrover@upc.edu

Abstract: Fused Filament Fabrication (FFF) is one of the most common additive manufacturing (AM) technologies that has arisen interesting from industry in a wide range of applications. In this paper, the mechanical properties of FFF 3D printed wood-PLA (Timberfill®) parts are investigated both experimentally and computationally to predict the mechanical characteristics of this material. Firstly, experimental tensile test is carried out to achieve the properties of the material. Secondly, the obtained parameters (Young’s modulus, Poisson’s ratio and yielding stress) will be used as input data in the ANSYS software to simulate a 4-point bending test. Finally, in order to validate the obtained model, the simulation results are compared to an experimental flexural test results indicating the correspondence between them. The main result of this work is an appropriate model to predict the behaviour of a 3D-printed piece formed by an internal structure with certain characteristics suitable for the manufacturing process and surrounded by a skin, which is subjected to certain external load.

Keywords: FEM, Bending, Wood-PLA, FFF, Composite.

1. Introduction

Additive manufacturing processes (AM) are a group of technologies very useful nowadays. They can be used to manufacture a wide range of usable components with complex geometries of various materials. One of the most popular technique in this field is the fused filament fabrication (FFF) which generates three-dimensional shapes by depositing the extruded wires of thermoplastic filament. FFF is a comprehensive process with a number of different parameters that can be influential on the mechanical properties and qualities of the productions. Since the combination of these parameters and finding the optimal set of them with appropriate levels is difficult to understand [1,2], recently a large number of studies have investigated the effects of these parameters on the mechanical properties of different materials [3-6].

Kazem Fayazbakhsh et al. [7], have studied the directional properties of FFF 3D printed polylactic acid (PLA) specimens. Through this work the tensile strength, Young’s modulus, and failure strain of specimens along and transverse to the printing direction were evaluated. The results showed that for
specimens without defects, the FFF process introduces anisotropic behavior on them. The pieces printed along the loading direction showed a tensile strength of 57.7 MPa and those printed transverse to the loading direction, a tensile strength of 30.8 MPa. Quasi-isotropic stacking has 20% higher failure strain compared to the case where all extrudates are along the loading direction. Another research [8], has considered the effect of the layer thickness and raster angle on mechanical properties of 3D printed samples. Tensile test results along with statistical analyses of the data clearly suggest that specimens with 0.2 mm layer thickness are stronger than specimens with 0.4 mm layer thickness and that layer thickness and raster orientation both have a significant effect on the mechanical properties of material.

In [9] the mechanical properties of the 3D printed parts of a wide range of commercial materials have been investigated. The study clearly demonstrated that the tensile strength of a 3D printed specimen depends largely on their mass, for all materials. Very recently, the influence of four printing parameters (layer height, nozzle diameter, fill density, and printing velocity) in three various levels on the flexural properties of wood-PLA composite specimens manufactured through FFF have been considered [10]. This study resulted that the layer height is the most influential parameter on flexural strength, followed by nozzle diameter and infill density, whereas the printing velocity has no significant influence. An optimal parameter set that maximizes the material’s flexural strength is found by combining a 0.2 mm layer height, 0.7 mm nozzle diameter, 75% fill density, and 35 mm/s velocity. The highest flexural resistance achieved experimentally is 47.26 MPa.

Furthermore, as the finite element method (FEM) advances into more practical and industrial methods in the manufacturing process area, in-depth understanding of printing parameters, a large number of studies have verified the simulation models based on this method. The study [11], combined the use of FFF with finite element analysis (FEA) to enhance the understanding of certain manufacturing parameters (i.e. material, infill density, infill pattern, and outer vertical shell) in the design process of a lumbar fusion cage. Three FFF materials with distinct mechanical properties namely polycarbonate (PC), acrylonitrile butadiene styrene (ABS), and PLA were tested. Three infill densities (i.e. 25%, 50%, 75%) were investigated along with two different infill patterns (i.e. rectangular and honeycomb). Compressive modulus and compressive yield strength values obtained from standard mechanical analysis were used as input for FEA to assess numerically the mechanical performance of a lumbar fusion cage under physiological static loading. The results of FEA analysis indicated that both PC and ABS can be adopted to fabricate a porous cage with a 50% infill density and a honeycomb infill pattern, without the need of a vertical outer solid shell. In the paper [12], a novel approach of voxelization modelling-based Finite Element simulation and process parameter optimization for FFF is presented. In the approach, firstly, a general meshing method based on voxelization modelling and automatic voxel element sorting is developed. Then, FE based simulation for the FFF process is conducted by combining the ANSYS Parametric Design Language (APDL) with the element birth and death technique. During the simulation, the influence of key process parameters on the temperature field, including scanning speed, molding chamber temperature and nozzle temperature, are analyzed in detail. Results show that among the process parameters, the molding chamber temperature have the most significant effect on the warping deformation of the FFF parts. The optimal parameters for the FFF process with ABS under the analyzed conditions are 50 mm/s for the scanning speed, 80 °C for the molding chamber temperature of, and 180 °C for the nozzle temperature, respectively. Courter, B. et al. [13], have simulated the fused deposition modeling (FDM) process by the commercial finite element software package Abaqus. A Mobius arm part is used to illustrate the simulation procedure and a sequentially coupled thermo-mechanical analysis is performed. The heat transfer analysis calculates the temperature history which is mapped onto and used to predict residual stress and potential part distortion in the subsequent structural analysis. Independent tool path events are characterized using event series data such as time, location and bead cross-sectional area. Abaqus solves for dependent events such as progressive element activation, local material orientation, and evolution of cooling surfaces, temperature profile, stress and distortion. The results show that finite element simulations effectively capture the interaction between tool path and thermomechanical physical processes and their impact on the final state of the printed part.
The purpose of current study is to distribute the mechanical characteristics of an innovative wood-PLA composite material (Timberfill) for designers and manufacturers on which they can make better engineering decisions by having all required information available. In order to do so, this paper aims to compare the results obtained through an experimental tensile test considering the optimal parameters resulted in a previous work [14] with the results that can be obtained by means of a finite element method simulation. Thus, this research includes three sections: I) the experimental tensile test according to ASTM D-638 standard test method is carried out on the specimens manufactured by FFF using the optimal printing parameters found in previous research under different orientations, II) the experimental results are used as input data for Finite Element method (FEM) simulation to verify the flexural test analysis, III) the obtained model is compared with a 4-point bending test results to evaluate their coincidences.

The model developed in this work considers that the specimens are formed by two different parts: the skin and the filler. Those parts are considered as two different materials in the model, in order to study their influence in the final properties of the whole piece. This is the novelty of this model, if it is compared to those found in the bibliography.

2. Materials and methods
Timberfill "Champagne" filament of 2.85 mm diameter developed and manufactured by company Filamentum Ltd. located in the Czech Republic was used to manufacture the specimens for this work. This is a new commercial composite material of biodegradable PLA reinforced with wood fibers in a range of 5% to 10% ratio.

2.1. Preliminary experiment: Tensile test
Firstly, a tensile test was developed. The specimens are manufactured according to ASTM D-638 standard test method for tension of plastics and composites utilizing dog-bone shape as shown in figure 1, with 7 mm thickness. Through this study, elastic modulus (Young’s modulus), elastic limit (Yield stress), tensile strength and maximum elongation at break are measured to characterize the mechanical behavior of Timberfill specimens.

![Figure 1. Dimension and geometrical shape of tensile sample.](image)

The sample was designed by SolidWorks software with the standard shape and dimension, then it was exported as a STL format which can be read and interpreted by the printing parameterization software. In this work the Simplify 3D software was applied to slice the drawing and generate the G-Code of the sample. To select the printing parameters, the information was derived from the previous experimental study [14] using the optimal combination of printing parameters that has shown the most influence on the tensile properties (table 1).

| Nozzle diameter | Layer height | Fill density | Infill pattern | Printing velocity |
|----------------|-------------|--------------|---------------|------------------|
| 0.5 mm         | 0.4 mm      | 75 %         | Honeycomb     | 40 mm/min        |
The orthotropic behavior of the specimens manufactured is determined mainly for the selected material and the parameters related with the infill geometry, which are the fill density and the infill pattern detailed in table 1. Therefore, the same printing condition has been repeated to print the specimens in three different building orientation (0º-X, 0º-Y, and 0º-Z), printed only with inside filling configuration without perimeter shell as shown in figure 2(a). Besides, the perimeter (skin) of the samples were printed separately with three layers in 0.4 mm height (figure 2(b)). It should be mentioned that for each printing condition, 5 specimens were manufactured and tested to guarantee the repeatability of the results. Thus, a total of 20 specimens were printed.

![Figure 2](image)

**Figure 2.** (a) Different building orientations infill samples. (b) Outlier or skin.

The experimental test method, equipment and the results analysis process has been executed following the methodology previously developed in [14]. During these processes, the tensile properties such as Young’s modulus ($E$), yield strength ($R_{p0.2}$), maximum strength ($\sigma_{\text{max}}$) and maximum deformation ($\varepsilon$) are defined for all samples.

The average results of five repetition tests with the standard deviations of each printing filling conditions and the skin are indicated in the table 2. All data required to describe material’s properties have been obtained experimentally. This information includes the $E_j$ (Young’s modulus in three-orthogonal directions), $\nu_{ij}$ (Poisson ratio) for introducing orthotropic structure which is attributed to linear behavior of material.

| # | $E$ (GPa) | $R_{p0.2}$ (MPa) | $\sigma_{\text{max}}$ (MPa) | $\varepsilon$ |
|---|---|---|---|---|
| 0º-X | 0.637 ± 0.025 | 4.256 ± 0.286 | 4.671 ± 0.476 | 1.359 ± 0.097 |
| 0º-Y | 0.520 ± 0.059 | 2.451 ± 0.227 | 2.467 ± 0.378 | 1.112 ± 0.074 |
| 0º-Z | 0.588 ± 0.027 | 3.051 ± 0.082 | 3.305 ± 0.164 | 1.006 ± 0.177 |
| Skin | 1.653 ± 0.039 | 12.657 ± 0.436 | 13.286 ± 0.436 | 1.570 ± 0.291 |

The Poisson’s ratio as 0.5 for samples printed in 0º-X, 0º-Z and skin and 0.3 for those printed in 0º-Y have been also determined.

3. **Simulation**

In order to simulate a four-point bending test, the mechanical behavior of this innovative material are also investigated applying ANSYS Workbench which is based on (FEM). The values obtained from the tensile experimentation are prepared and exported into the ANSYS software as engineering data to define the material.
3.1. Modelling process

Once the tensile tests have been carried out, the Young's modulus in all 3 directions are obtained. The orthotropy of the material can be specified with the corresponding calculation of the shear modules in the three directions, the Young’s modulus and the hypothesis of the Poisson's ratio values.

In the next step the geometrical model of the sample is created according to the ASTM D6272 standard [15]. This test method covers the procedure to determine the flexural properties of unreinforced and reinforced plastics, including high-modulus composites and electrical insulating materials in different forms. Hence its adequacy for the purposes of these works with a composite material. In this case the configuration has been modeled in three-layered structure: the bottom and top layers as skin, and the central layer as infill so that they have been considered to be integrated, all of them are considered solid blocks in order to reduce the computational cost so no infill is designed for the simulation process. Nevertheless, this aspect is no important to get proper results. Whereas the applied test method has geometrical and enforcement symmetry, therefore using a vertical slide restriction for the central plane of the part can correctly represent the evolution of tensions (figure 3(a)). Also, the sample is divided into different segments for two reasons: 1) to make structured and regular mesh, 2) to define accurate contact between the spans and specimen (figure 3(b)).

Figure 3. The finite element model corresponding to the experimental setup. (a) Schema of the piece symmetry considered. (b) Sample divided into different segments.

Figure 4. Boundary conditions defined on the sample.

A higher order 3D element composed of 20 nodes named SOLID186 has been used. Each node has three degrees of freedom: translations in the nodal x, y and z directions. Since this type of element exhibits quadratic displacement behavior, it is more accurate in estimating displacement than other elements with linear shape functions. It should be mentioned that frictionless has been considered as type contact between force span and specimen. To avoid lateral displacement in the center face of the sample, constrain is determined in X and Z directions, but it is free to move in Y direction (vertically).
and the supports span are limited at the bottom area in all directions. In order to apply uniform displacement into the force span, a reference point is created and all degree of freedom of this body is assigned to this point. All boundary conditions are detailed in figure 4.

The assembly has been meshed as indicated in figure 5 to obtain different elements for both the skin and the filler, and the supports that are sufficient to get plausible results. The analysis has been started by moving down the upper roller (force span) to simulate the force direction at which the experiments were performed at the same displacement rate while the other roller (support span) remains fixed.

![Mesh diagram of modelling assembly](image1)

**Figure 5.** Mesh diagram of modelling assembly.

### 3.2. Simulation results

Once the simulation process is finished, the mechanical properties such as maximum principal stress, maximum principal elastic strain, total deformation, directional deformation in y axis and force reaction versus time are achieved. Stress distribution extracted from element analysis of equivalent stress is indicated layer by layer in figure 6. As it was expected, the maximum stress is distributed at the outliers of the sample between the force spans, which the samples during the experimental test have experienced the similar behavior. The maximum values obtained are shown in the table 3.

**Table 3.** Properties’ maximum values obtained by simulation.

| Property                  | Value     |
|---------------------------|-----------|
| Maximum Principal Stress  | 32.315 MPa|
| Total deflection          | 9.71 mm   |

![Equivalent maximum principal stress diagram](image2)

**Figure 6.** Equivalent maximum principal stress diagram.
4. Result’s discussion
To validate the obtained model, the Force-Deflection curve is compared to those obtained experimentally in two different 4-points bending test. These tests were done by [10] and are published in a previous work. The conditions used in the modeling process and in the experiments are the same: material and manufacturing parameters, considering the solid as a block. Figure 7 shows the curves obtained by the model and the experiments. This graph validates the experimental results evidencing the simulation by the coincidence of the deformation trajectory.

![Figure 7. Load - deflection curves comparison of experiments and simulation.](image)

5. Conclusions
In this research, a model of simulation has been validated through a theoretical analysis of the mechanical behavior of FFF manufactured wood-PLA and comparing the theoretical results with the experimental ones.

Based on the results obtained, a noticeable differentiation is observed on two areas with clearly different properties: on the one hand, the skin of the piece with isotropic characteristics, on the other hand, the infill mainly orthotropic.

Using this hypothesis, a model has been designed and evaluated by means of computational simulations using FEM method (Finite Element Method) analysis. This type of modeling provides great advantages in the field of material characterization, caused by its simplicity in design and low computational cost. Properties such as maximum principal stress or deformations that are very useful to predict the behavior of the material that can be obtained simply without a deep knowledge of computational simulations. The results obtained show the power of this analysis due to the coincidence with the experimental values carried out.

Acknowledgements
Authors wishing to acknowledge the financial support from the Centre de cooperació pel desenvolupament (CCD) from the Universitat Politècnica de Catalunya by the grant 2020-A029.

References
[1] Chacón J M, Caminero M A, García-Plaza E and Núñez P J 2017 Additive manufacturing of PLA structures using fused deposition modelling: Effect of process parameters on mechanical properties and their optimal selection Materials & Design 124 pp 143-157
[2] Mohamed O A, Masood S H and Bhowmik J L 2015 Optimization of fused deposition modeling process parameters: a review of current research and future prospects Advances in Manufacturing 3 (1) pp 42-53
[3] Gomez-Gras G, Jerez-Mesa R, Travieso-Rodriguez J A and Lluma-Fuentes J 2018 Fatigue performance of fused filament fabrication PLA specimens Materials & Design 140 pp 278-
8

[4] Domingo-Espin M, Travieso-Rodriguez J A, Jerez-Mesa R and Lluma-Fuentes J 2018 Fatigue performance of ABS specimens obtained by fused filament fabrication Materials 11 (12) p 2521

[5] Travieso-Rodriguez J A, Jerez-Mesa R, Llumà J, Traver-Ramos O, Gomez-Gras G and Roa Rovira J J 2019 Mechanical properties of 3D-printing polylactic acid parts subjected to bending stress and fatigue testing Materials 12 (23) p 3859

[6] McLouth T D, Severino J V, Adams P M, Patel D N and Zaldivar R J 2017 The impact of print orientation and raster pattern on fracture toughness in additively manufactured ABS Additive Manufacturing 18 pp 103-109

[7] Fayazbakhsh K, Movahedi M and Kalman J 2019 The impact of defects on tensile properties of 3D printed parts manufactured by fused filament fabrication Materials Today Communications 18 pp 140-148

[8] Tanikella N G, Wittbrodt B and Pearce J M 2017 Tensile strength of commercial polymer materials for fused filament fabrication 3D printing Additive Manufacturing 15 pp 40-47

[9] Rankouhi B, Javadpour S, Delfanian F and Letcher T 2016 Failure analysis and mechanical characterization of 3D printed ABS with respect to layer thickness and orientation Journal of Failure Analysis and Prevention 16 (3) pp 467-481

[10] Zandi M D, Jerez-Mesa R, Lluma-Fuentes J, Roa J J and Travieso-Rodriguez J A 2020 Experimental analysis of manufacturing parameters’ effect on the flexural properties of wood-PLA composite parts built through FFF The International Journal of Advanced Manufacturing Technology 106 (9) pp 3985-3998

[11] Provaggi E, Capelli C, Rahmani B, Burriesci G and Kalaskar D M 2019 3D printing assisted finite element analysis for optimising the manufacturing parameters of a lumbar fusion cage Materials & Design 163 p 107540

[12] Zhou Y, Lu H, Wang G, Wang J and Li W 2020 Voxelization modelling based finite element simulation and process parameter optimization for Fused Filament Fabrication Materials & Design 187 p 108409

[13] Courter B, Savane V, Bi J, Dev S and Hansen C J 2017 Finite element simulation of the fused deposition modelling process NAFEMS World Congress (Stockholm) pp 11-14

[14] Zandi M D, Jerez-Mesa R, Lluma-Fuentes J, Jorba-Peiro J and Travieso-Rodriguez J A 2020 Study of the manufacturing process effects of fused filament fabrication and injection molding on tensile properties of composite PLA-wood parts The International Journal of Advanced Manufacturing Technology 108 pp 1725-1735

[15] ASTM D6272-02 2002 Standard Test Method for Flexural Properties of Unreinforced and Reinforced Plastics and Electrical Insulating Materials by Four-Point Bending