PAPER • OPEN ACCESS

Numerical simulation of FDM manufactured parts by adopting approaches in composite material simulation

View the article online for updates and enhancements.

Numerical simulation of FDM manufactured parts by adopting approaches in composite material simulation

A D Akessa [*]**, A W Gebisa and H G Lemu**

Department of Mechanical and Structural Engineering and Materials Science, University of Stavanger, Norway

[*] Corresponding author: adugna.d.akessa@uis.no

Abstract. Additive manufacturing (AM) process is a promising manufacturing method that can replace conventional manufacturing methods, particularly for parts with complex geometry. It is the process of joining materials layer by layer to make object directly from 3D model data. Due to the inherent manufacturing process characteristics, such products experience material anisotropy with mechanical properties not easy to calculate analytically. On the other hand, numerical simulation is becoming increasingly important to solve complex problems such as in composite materials. However, no significant work of numerical simulation is reported for additive manufactured parts. This paper reports an approach to numerical simulation of additive manufactured parts by adopting methodologies developed for composite materials. To fill the existing gap in this area, the mechanical properties of FDM parts are experimentally studied. The experimental samples are produced from ULTEM9085 material with different printing parameters. The mechanical properties of the samples are then analyzed and numerical simulation using finite element method is done to compare the results with experimental results and verify the simulation model. The main aim of the study is to devise a numerical simulation method for additive manufactured parts by adopting existing methods for composite materials.

1. Introduction

Additive manufacturing (AM) also knows as 3D printing is a manufacturing technology that is showing a promising progress for the future manufacturing sector. To advance the technology to the next level, the mechanical properties of the materials produced by the technology should be well documented, organized and standardized. In the conventional manufacturing industry, the mechanical properties of materials are well organized and compiled on plenty of material handbooks. Furthermore, the mechanical properties of the materials can be predicted using diverse tools and methods including finite element analysis (FEA) techniques. When it comes to additive manufacturing, on the other hand, since the technology is still immature, the use of FEA for the prediction of their mechanical properties are very limited [1]. This can be attributed partly to the fact that the AM of load carrying structural parts is an emerging approach from the application of 3D printing in rapid prototyping. In addition, the AM process involves a combination of, among others, different materials, material processing energy, process parameters and fabrication (printing) methods. Thus, computational tools that are developed so far are dedicated to specific material type [2, 3] or processing technology, for example laser sintering [4 - 6] and there exists no sufficiently developed approach that can address each combination.

Fused deposition modelling (FDM), one of additive manufacturing techniques of thermoplastic materials, builds parts by heating, extruding and depositing thermoplastic materials in layer by layer fashion as illustrated in Figure 1. As a result of lack of numerical computation tools to predict the behavior of structural components by this technology, most of the reported studies focus on experimental methods [7, 8], theoretical analysis [8] and analytical/empirical methods [9, 10].
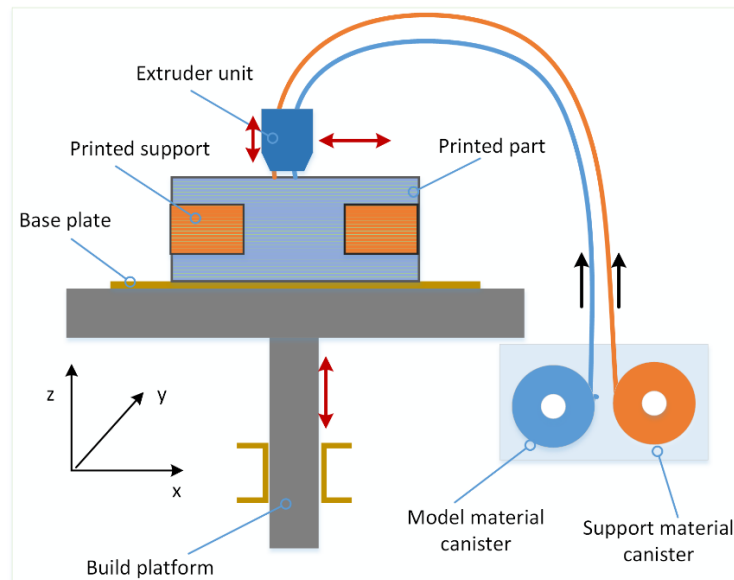
Figure 1. Illustration of fused deposition modelling (FDM) technique

The layer-wise deposition in fused deposition modelling resembles the structure of laminated composites in which the fabrication process builds by piling up layers of laminates. Laminated composites consist thin layers of different materials bonded together, such as bimetals, clad metals, plywood, formica and the like. As the FEA of composites is relatively developed and employed in different FEA software, the current research attempts to adopt FEA approach in ANSYS software that is developed for laminated composite materials to the study of the mechanical property (tensile) of ULTEM 9085 produced by fused deposition modelling and verify the approach through experimental investigation. In other words, the aim of the study reported herein is to investigate the possibility of applying the already developed FEA technique for composite material structures for the parts produced by fused deposition modelling.

The remaining part of the article is divided into 4 main sections. Section 2 explains the basic theories and mechanics of composite materials with focus on the theory that gives the rationale to use the numerical simulation approaches developed for composite laminates to additive fabricated parts. Then, the materials and methods used to conduct the study are presented in Section 3 followed by discussion of results in Section 4. Finally, the conclusions drawn from the study are summarized in Section 5.

2. Basic composite material theory and analogy with additive fabrication

2.1. Mechanics of composite materials and structures
A composite material is combination of two or more materials at macroscopic scale, which are not soluble in each other to form a third material. The combined material constituent is called the reinforcing phase and the one in which it is embedded is called the matrix [11]. Due to properties of composite structure, mechanical characterization is more complex compared to metal structures which have isotropic properties. To conduct mechanical analysis, more material properties are needed compared with monolithic materials like steel. There by, structural analysis computational and experimentally more complicated and intensive due to complexity of its nature. The properties of composite material depends on the properties of the constituents, their geometry and the distribution of the phases [12].

Lamina, or ply, is a plane (curved) layer of unidirectional fiber or woven fabric in a matrix. It is called unidirectional lamina in case of unidirectional fiber. Lamina is an orthotropic material with principal material axes in the direction of the fibers (longitudinal), normal to the fibers in the plane of the lamina (in-plane transverse) and normal to the plane of the lamina. These principal axes are designated as 1, 2 and 3 respectively as shown in Figure 2.

Laminate is a combination of two or more unidirectional laminae or plies stacked together at various orientations. The laminae can be of different thicknesses and consists of different materials, based on the orientation of the principal axes, laminates are analyzed using common coordinates (x, y, z). Lamina is a single flat layer of unidirectional fibers or woven fibers arranged in a matrix, whereas a laminate is a stack of plies of composites. Each layer can be laid at various orientations and can be made up of different material systems. However, a real structure will not consist a single lamina, which has approximately a thickness of 0.125 mm, but a laminate consisting of more than one lamina bonded together through their thickness will be required to take realistic loads [13].
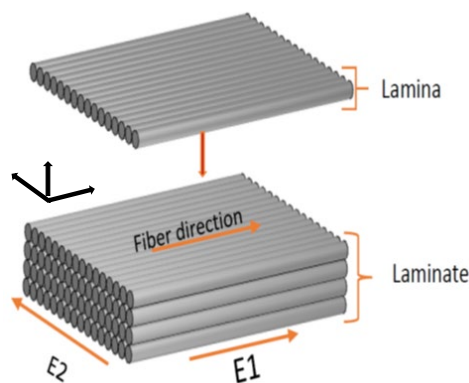
Figure 2. Illustration of composite stacking and their principal axes

2.2. Mechanical behaviour of composite materials

The mechanical behaviors of composite materials are different from conventional engineering materials. While some properties of these materials are just modifications of conventional behaviors, there are also some completely new characteristics that require new analytical and experimental approaches.

For instance, most ordinary engineering materials are both homogeneous and isotropic. For homogeneous bodies, the material properties are uniform throughout the part, i.e., the properties are independent of position in the body. Whereas isotropic bodies have material properties that are identical in all the three directions at a point in the body. Composite materials, on the other hand, are often both inhomogeneous and non-isotropic (either orthotropic or anisotropic). Thus, they have non-uniform properties throughout the body. The material properties of orthotropic bodies are different in the three mutually perpendicular directions at a point in the body i.e. the body has three mutually perpendicular planes of material property symmetry, that the properties a point in the body are orientation dependent.

Some composites like laminated safety glass have three layers, each of which is homogeneous. Composites with fibres placed at any angle are more complex. Due to the inhomogeneity nature of composite structures, they are studied from two different points of view. One is based on micromechanics point of view, where the interaction of the constituent materials is investigated at the microscopic scale to see their effects on the performance of the material. The second is micromechanics point of view, where the properties of the constituent materials are examined at the macroscopic scale assuming that the composite material is homogeneous [11].

Barbero [13] states that the single lamina of fiber reinforced composite behaves as an orthotropic material, i.e. the material has three mutually perpendicular planes of symmetry. Further, when force is applied on a composite structure, every point in the body may rotate and translate as a rigid body as well as deform to occupy a new space. However, random arrays of aligned fibers are considered as transversely isotropic having five independent elastic constants including two poison's ratio values which are caused by axial stress and transverse stress. These Poison's ratios are not independent [14].

2.3. Numerical simulation of FDM parts

Simulation in engineering is conducted to evaluate the designed performance of mechanical parts or systems using relevant computation tools. In addition, it is used in the product development process throughout the concept development, design and manufacturing phases. As a result, recent research has focused on application of simulation-driven design in order to help designers get immediate feedback on design performance by embedding simulation into the CAD environment and designers' work [15].

FDM is new and promising technology in the field of manufacturing in which one can produce complex parts from 3D model (CAD). There is limited work of numerical simulation of additive manufacturing using FDM process. Therefore, this work is done by adopting the numerical simulation methodology already developed for composite materials for FDM produced parts. This approach is adopted because the internal structure of FDM produced part is assumed analogous to the fiber layout in composite materials and as a result, classical laminate theory in composite simulation is adopted in the study. In the literature [13, 15], it is also stated that a single lamina of fiber reinforced composite behaves as an orthotropic material. In macro-mechanics, it is also assumed that the lamina, which is a building block of laminate, is macroscopically homogeneous, linear elastic, macroscopically orthotropic and initially stress free. In this simulation of FDM parts, each layer of a component is considered as a lamina and the total layers of the component is taken as a laminate.

3. Materials and methods

3.1. Sample preparation

To conduct experimental study, the samples were prepared based using ISO standard (ISO 172). The dimensions of the samples are shown in Figure 4. The models were developed in Autodesk Inventor® and saved in STL file format. Then, the STL file was loaded to a pre-processing software called Insight, which is provided with the FDM 3D printer to be used for slicing and process parameter settings. The software, then saves the data in Chromeleon Backup Archive (CBM) file format which is ready for printing.

Three printing orientations were considered: (1) along the planned loading direction (0°), (2) transverse to loading direction (90°) and (3) 45° to the loading direction. Identical geometries were used for both 0° and 90° samples (Figure 3(a)) and another geometry, shown in Figure 3(b), was used for the 45° samples. Five replicas were used for both orientations (0° and 90°) samples and four samples were tested for 45° samples. The samples were fabricated using the FDM machine, Fortus 450 mc.
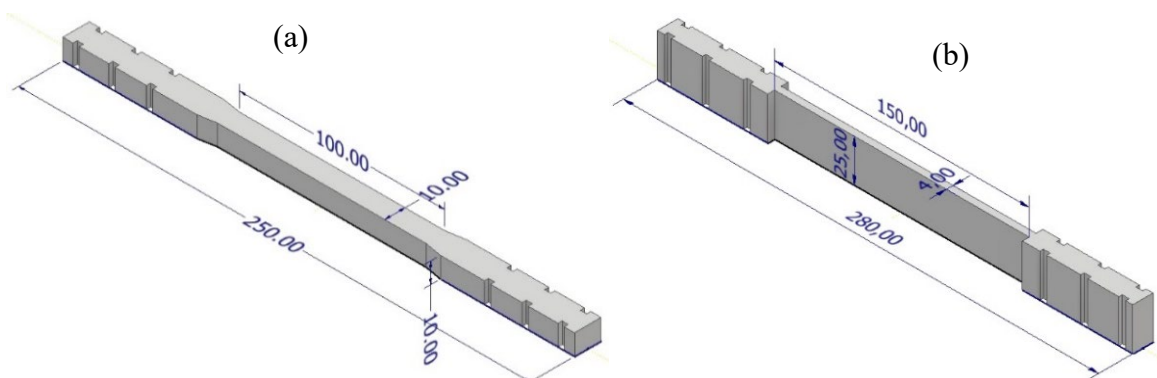

Figure 3. 3D CAD models of test samples ((a) 0º and 90º and (b) 45º (all dimensions in mm).

3.2. Experimental setup and procedure

The experiment was performed on Instron 5895 using a tensile load of 250 kN. The tensile test was performed by placing the specimen into the grips of the test machine, as illustrated in Figure 4. The grips

were tightened and then the tension load was applied at a constant recommended strain rate of 0.01 mm/mm/min. The data were then collected and analyzed.
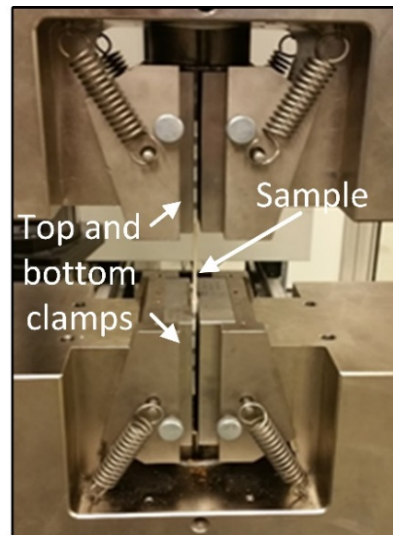
Figure 4. Experimental setup on INSTRON 5895 Universal test machine.

The tensile tests were conducted by slowly pulling the test pieces until they break. The test machine measures stress as the ratio of force applied to its original area, and the strain is the ratio of change in length to the original length. The calculated maximum stress and strain values are given in Table 1.The values of the principal elastic moduli, the shear modulus and the Poison's ratio values were obtained from experimental tests. These values for the material used (ULTEM 9085) are evaluated in elastic region and the results are given in Table 2.

Table 1. Values of maximum stress and strain for each orientation.

Raster angle	Maximum stress (MPa)	Strain 1 (mm/mm)	Strain 2 (mm/mm)
0^0	87	0.064	0.274
45^0	68	0.043	0.07
90^0	37	0.0197	0.073

Table 2. Elastic properties of ULTEM 9085 material.

Elastic modulus	Values (MPa)	Shear modulus	Values (MPa)	Poison's ratios	Values
E_{11}	2206	G_{12}	2055	ν_{12}	0.39
E_{22}	1826	G_{13}	2055	ν_{13}	0.39
E_{23}	1826	G_{23}	793	ν_{23}	0.4

3.3. Numerical simulation approach

As stated above, the elastic properties of the test samples (i.e. modulus of elasticity along principle axes, shear modulus and Poison's ratios) are obtained from experimental tests. Properties along the principal axes are assumed from specimens printed upright in Z-direction in 0° raster orientation, whose specimen is similar with those printed in 90° orientation.

The numerical analysis in the study is performed using a FEA software, ANSYS Workbench. Finite element analysis of laminated composites is done at different levels depending on the level of post-processing required. Where detailing is necessary, the stress and strain are computed at level of fiber and

matrix. In this case, it is found necessary to describe the microstructure, including the fiber shape and geometrical distribution, and the material properties of the constituents. When the composite material is a woven fabric, or the laminate is very thick, the composite should be analyzed as solid. However, most of laminate structures can be analyzed using simplified models of plates and shells. On the other hand, the composite material can be considered as a homogeneous equivalent material. In this case, its structural behavior can be analyzed using orthotropic properties. To calculate stress and strain, the actual laminate stacking sequence (LSS), the elastic properties of each lamina, as well as thickness and fiber orientation of every lamina must be given [4]. Therefore, in this study, finite element analysis is done assuming the FDM fabricated parts as linear orthotropic composite material, in which stacking sequence, elastic properties of each laminate, thickness and fiber orientation is defined from experimental tests.

As stated, test specimens for this study were printed in the following orientations; 0° where load is applied along filament orientation, 90° in which filament orientation is perpendicular to load application direction and 45° in which the filament is oriented at 45° to the applied load direction. The specimens were tested to obtain values of the nine independent constants (Table 2). Using these constants, numerical analysis was done in ANSYS workbench. The number of lamina was found by dividing specimen gauge length to layer thickness and each lamina is stacked one up on the other to form the laminate as shown in Figure 5 (a) – (c).

As stated above stiffness matrix for the orthotropic constitutive model was used to simulate the mechanical response of the same parts physically tested and data is obtained from the experimental test. In this research, three different simulations were made, one for each experimentally tested orientation. Material property data (i.e. elastic modulus, poison's ratio and shear modulus in direction 1, 2, and 3).

The models were developed using Design Modeler, through modeling each layer with thickness of 0.254 mm which is equal to layer thickness adding each layer as 'Add Frozen' for every orientation. Engineering data is defined from experimental test. ANSYS Workbench 17.0 software was used to simulate the part considering the building orientation by assuming the material is orthotropic. In ANSYS Mechanical, material is assigned for each layer as ULTEM 9085. Finally the model is meshed by setting the model properties and analysis was conducted by defining the boundary conditions according to experimental tests.
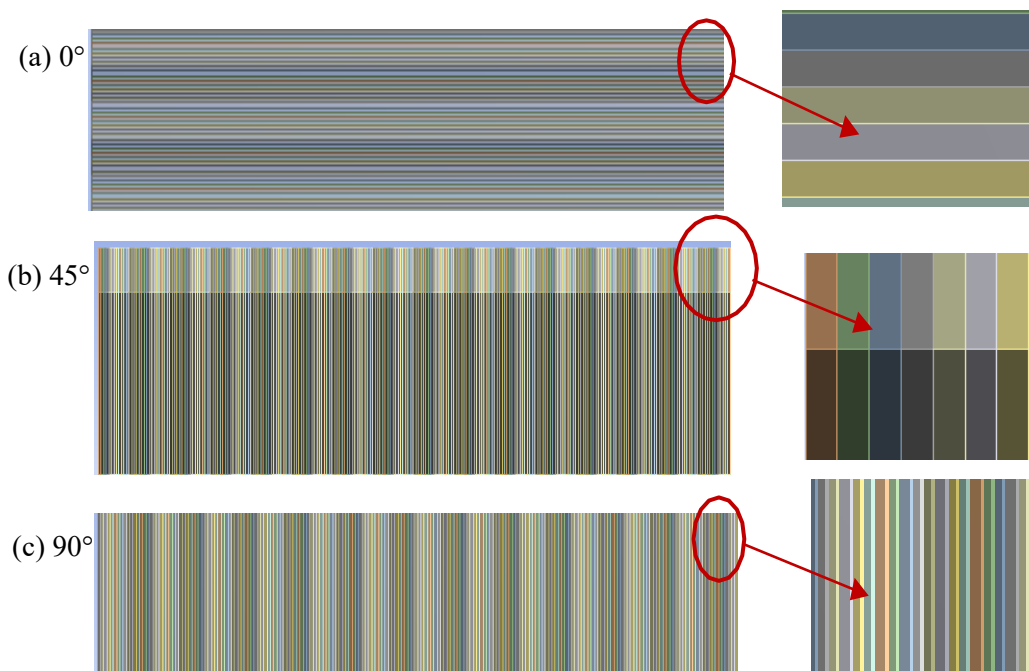

Figure 5. Illustration showing models of different orientations in ANSYS.

4. Result and Discussion

4.1. Experimental result

Tensile stress-strain graphs from experimental test for each printing orientation (i.e., 0°, 45° and 90°) are given in Figure 6(a) – (c). The maximal tensile stress obtained from the experimental results are about 78 MPa, 63 MPa and 37 MPa for 0°, 45° and 90° respectively. In the experimental test, 5 specimens were tested for each orientation, except the case in Figure 6(b), in which test 5 was observed unjustified.

In all the considered cases, the results show the same trend. As normally expected the maximum tensile stress is achieved for loading along printing direction (0°), while the lowest maximum tensile stress is obtained for loading in transverse direction (90°). The corresponding strain values at maximum tensile stress are 0,04, 0,035 and 0,018 for 0°, 45° and 90° respectively.
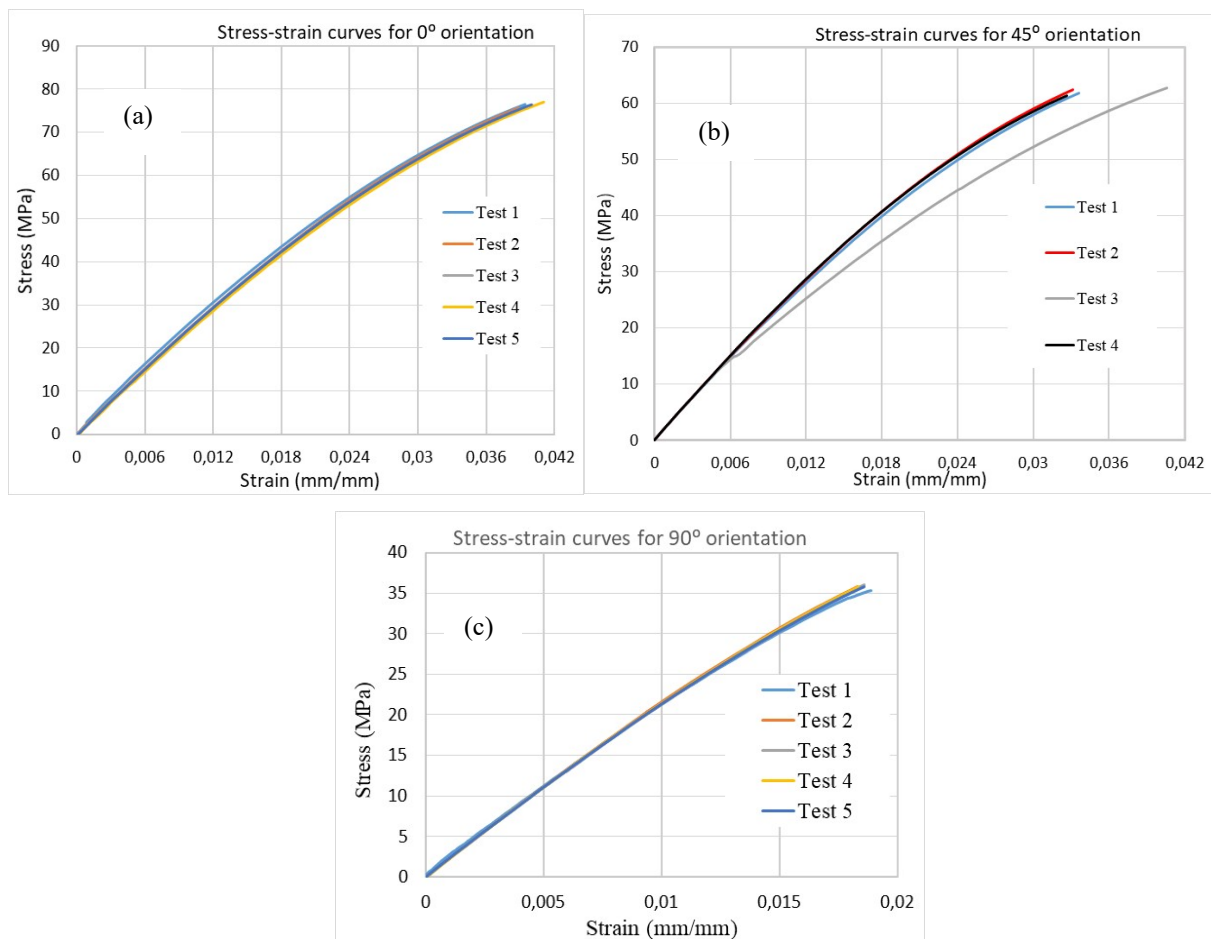

Figure 6. Tensile stress-strain curve for (a) 0° orientation (b) 45° orientation and (c) 90° orientation.

4.2. Finite element analysis results

The results of the finite element analysis using ANSYS 17.0 workbench are given in Figure 7 (a) – (c) for 0°, 45° and 90° orientations respectively. In the simulation of parts produced by 3D printing, which are identical with those produced for experimental test, orthotropic material properties are considered. The corresponding material property for FEA modeling is taken from previous study conducted by EI-Gizawy et al. [16]. The boundary conditions for the FEA are set in similar way as the experimental setup, i.e. one end of the specimen is fixed and the load is applied at the other end. Default mesh sizes are used to generate the mesh model.

The result from FEA shows maximum stress at boundary conditions due to stress concentration as a result of specimen geometry and boundary conditions. However, the result taken in the study is the average result. The result from FEA shows maximum value in 0^0 orientation where the load is applied along filament orientations and minimum in 90^0 orientation where load is applied transvers to filament orientations. In the latter case the applied load try to break the bond between filaments.
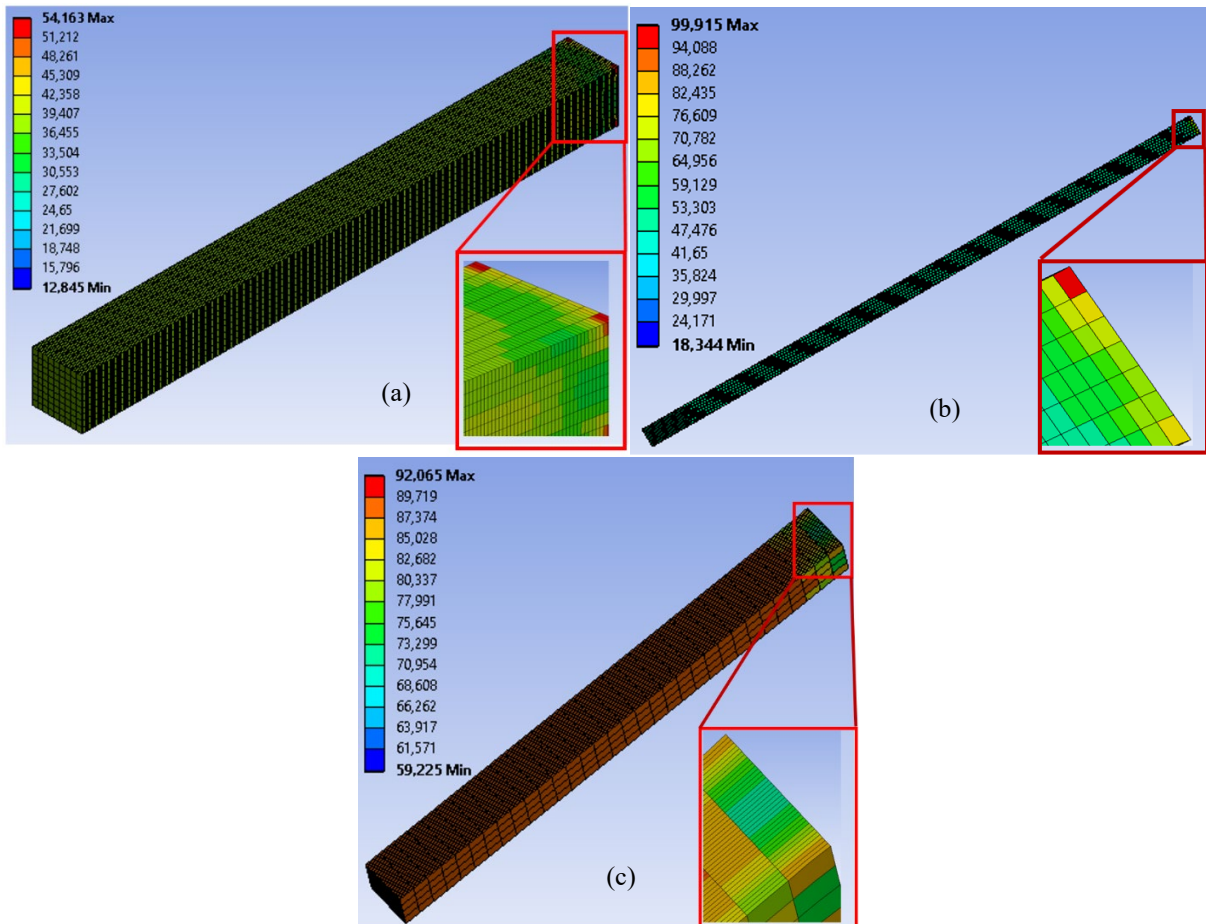
Figure 7. Illustration of stress distribution for orientations in (a) 0° (b) 45° and (c) 90° direction.

4.3. Comparison of experimental and numerical results

The results from the experiment and FEA are compared as plotted in Figure 8(a) – (c). As previously stated, the results taken from FEA are the average value and experimental results are raw data from INSTRON 5895. In the case of loading along the printing direction (0^0 orientation), the experimental and the numerical results almost overlap until the strain value reaches 0,02. For higher strain values, however, the numerical simulation (average value) proceeds linearly, while the experimental results indicate that the material gradually gives up for higher stresses. This trend is observed for the other two cases (45^0 and 90^0 orientation), though the results, in general, show that the numerical simulations underestimate the stress values for low range of strain values.
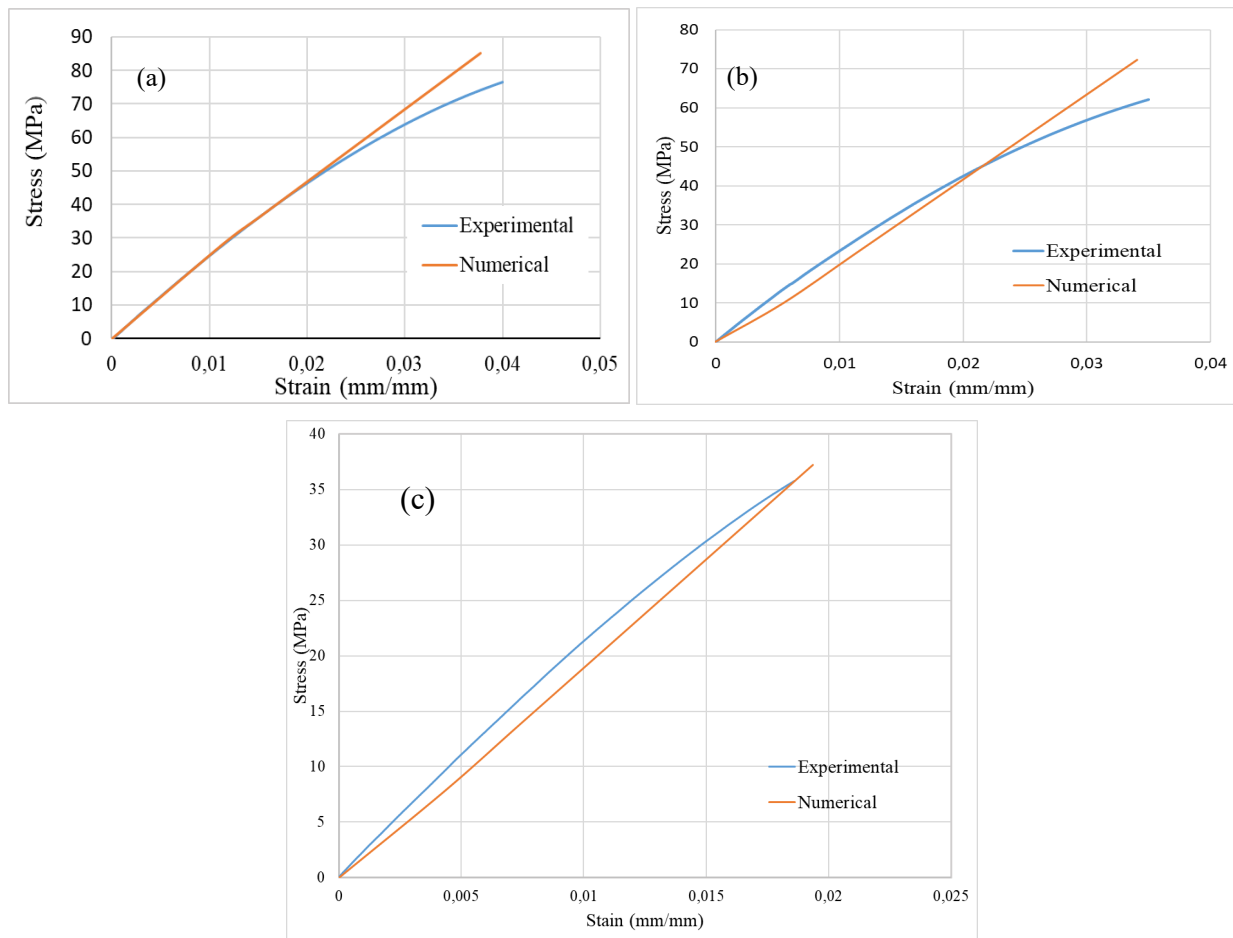
Figure 8. Comparison of results from experimental and FEA for (a) 0° (b) 45° and (c) 90° orientations.

5. Conclusion

The study of additive manufactured parts using both experimental and numerical methods are reported in this article. The numerical simulation approach is based on the approach available in ANSYS workbench for simulation and analysis of laminated composite materials. The reason to adopt this simulation technique is that the fabrication process of both cases uses layer by layer arrangement of materials that will have impact on the mechanical behavior.

In order to verify the employed numerical simulation technique, experimental tests were conducted on additive manufactured samples of the same size and material property. The results show that both results follow the same trend, though some deviations are observed for 45^0 and 90^0 printing orientations. In general, the adoption of the simulation technique of laminated composites for additive manufactured parts gives acceptable results particularly when the load acts in the printing direction. For higher strain levels than about 0,02, however, the simulation remains linearly varying and overestimates the stress value.

References

[1]   Bikas H, Stavropoulos P and Chryssolouris G 2016 Additive manufacturing methods and modelling approaches: a critical review, *Int. J. Adv. Manuf. Technol.* **83**, 389–405.
[2]   Zargarian A, Esfahanian M, Kadkhodapour J and Ziaei-Rada S 2016 Numerical simulation of the fatigue behavior of additive manufactured titanium porous lattice structures, *Mater. Sci. Eng.*: C, **60**(1), 339 – 347.
[3]   Lindgren L-E, Lundbäck A, , Fisk M and , Draxler J 2019 Modelling additive manufacturing of superalloys, *Procedia Manuf.* **35**, 252–258.

[4]   Hu D and Kovacevic R 2003 Sensing, modeling and control for laser-based additive manufacturing, *Int. J. Mach. Tools Manuf.* **43**(1), 51 – 60.

[5]   Nie P, O.A.Ojo and Lia Z 2014 Numerical modeling of microstructure evolution during laser additive manufacturing of a nickel-based superalloy, *Acta Mater*., **77**, 85 – 95.

[6]   Zhang Z, Tan Z J, Yao X X, Hu C P Ge P, Wan Z Y, Li J Y and Wu Q 2019 Numerical methods for microstructural evolutions in laser additive manufacturing, *Comput. Math. Appl.* **78**(7), 2296 – 2307.

[7]   Byberg K I, Gebisa A W and Lemu H G 2018 Mechanical properties of ULTEM 9085 material processed by fused deposition modeling, *Polymer Testing,* **72**, 335-347.

[8]   Zhao Y, Chen Y and Zhou Y 2019 Novel mechanical models of tensile strength and elastic property of FDMAM PLA materials: Experimental and theoretical analyses, *Mater. Des.* **181**, 108089

[9]   El Moumen A, Tarfaoui M and Lafdi K 2019 Additive manufacturing of polymer composites: Processing and modeling approaches, *Composites Part B*, **171**, 166 – 182.

[10]   Gebisa A W and Lemu H G 2018 Investigating effects of Fused-Deposition Modeling (FDM) processing parameters on flexural properties of ULTEM 9085 using designed experiment, *Mater.* **11**(4), 500.

[11]   Kaw A K 2006 *Mechanics of composite materials*. 2nd Ed. CRC press, NY, USA.

[12]   Daniel I M and Ishai O 2006 *Engineering mechanics of composite materials*, 2nd Ed. Oxford University Press, Oxford, UK.

[13]   Barbero, E.J., 2013. *Finite element analysis of composite materials using ANSYS*. 2nd Ed. CRC press, Taylor & Francis Group, NW, USA.

[14]   Chawla K K 1987 *Composue materials: Science and Engineering*, Springer-Verlag, NY, USA

[15]   Siemens home page: https://www.plm.automation.siemens.com/global/en/ (Last accessed: 15.09.2019)

[16]   El-Gizawy A S, Corl S and Graybill B 2011 *Process-induced properties of FDM products*. In Proc. of the ICMET, Int. Conf. on Mech. Eng. Technol. Congress and Exposition, 2011.