Article
Journal of Systematic Innovation
June 2023 Vol.7 No.20 10-19

Characterization of additive manufactured specimens for tensile testing using finite element

Caracterización de probetas elaboradas con manufactura aditiva para el ensayo de tracción utilizando elemento finito

GONZÁLEZ-SOSA, Jesús Vicente†*, AVILA-SOLER, Enrique and ZAVALA-OSORIO, Yadira

Universidad Autónoma Metropolitana, Unidad Azcapotzalco, Departamento de Sistemas

ID 1st Author: *Jesús Vicente, González-Sosa* / **ORC ID**: 0000-0002-1325-0266, **CVU CONAHCYT ID**: 166452

ID 2st Co-author: *Enrique, Avila-Soler* / **ORC ID**: 0000-0001-8980-0925, **CVU CONAHCYT ID**: 360262

ID 3nd Co-author: *Yadira, Zavala-Osorio* / **ORC ID**: 0000-0001-5337-6624, **CVU CONAHCYT ID**: 104843

DOI: 10.35429/JSI.2023.20.7.10.19 Received March 14, 2023; Accepted June 29, 2023

Resumen

En la actualidad la manufactura aditiva (MA) y la simulación computacional son esenciales en el desarrollo tecnológico para la validación de materiales, con ello asimilar los requerimientos y propiedades de un material antes de ser utilizado para aplicaciones específicas, por lo que en este trabajo se lleva a cabo la caracterización de la probeta para los ensayos de tracción, la cual se elabora en material ABS, utilizando la manufactura aditiva por medio de la I3D (Impresión 3D) y posteriormente se aplica FEM (Método de Elemento Finito), con el propósito de obtener una base de datos con respecto a las propiedades físicas del material al ser sometido a una simulación que representa el ensayo de tracción en materiales plásticos. Con esto se busca implementar nuevas áreas de desarrollo que involucran innovación en la caracterización de materiales utilizados en la manufactura aditiva, incrementando el uso de herramientas computacionales con simuladores en ingeniería. El uso de los datos estará disponible para la comunidad académica e investigadores con la finalidad en enriquecer las aplicaciones con FEM.

Manufactura aditiva, Caracterización, Simulación computacional

Abstract

Nowadays, additive manufacturing (MA) and computational simulation are essential in the technological development for the validation of materials, thus assimilating the requirements and properties of a material before being used for specific applications, so in this work the characterization of the specimen for tensile tests is carried out, which is made in ABS material, using additive manufacturing through 3D printing and then FEM (Finite Element Method) is applied, in order to obtain a database regarding the physical properties of the material when subjected to a simulation that represents the tensile test in plastic materials. This seeks to implement new areas of development that involve innovation in the characterization of materials used in additive manufacturing, increasing the use of computational tools with engineering simulators. The use of the data will be available to the academic and research community in order to enrich FEM applications.

Additive manufacturing, Characterization, Computational simulation

Citation: GONZÁLEZ-SOSA, Jesús Vicente, AVILA-SOLER, Enrique and ZAVALA-OSORIO, Yadira. Characterization of additive manufactured specimens for tensile testing using finite element. Journal of Systematic Innovation. 2023. 7-20: 10-19

* Correspondence to Author (e-mail: jvgs@azc.uam.mx)
† Researcher contributing as first author.

Introduction

At the Universidad Autónoma Metropolitana, Unidad Azcapotzalco, researchers have applied techniques and methodologies of study with different tools to validate and analyze cases related to science and engineering in different areas, in this work we describe in a general way the use of computational tools through FEM, to characterize and evaluate the plastic materials used in MA with I3D, in order to provide information to the academic and research environment to strengthen innovation and cutting-edge technology.

In the respective to additive manufacturing can be shown to the following works of precursors that were fundamental basis to strengthen this research: (Jung et.al, 2023), mention that, in recent years, it has been shown the potential for lightweighting in load-bearing structural components could be further improved by MA technology. However, the MA process offers a large parameter space that greatly affects the part quality and its inherent mechanical properties. It was concluded that the constraints of stress and uniaxiality conditions are identified with the most influential parameters by correlation analysis. The selected design solutions are further analyzed and compared with generic transition design approaches. The most promising design features (compatible edges, rounded cross-section, column connection) are then interpreted into structural elements, leading to an innovative generic design of the load introduction region that yields promising results after a proof-of-concept study.

(Sponchiado et.al, 2023), allude that additive manufacturing by material extrusion allows combining materials in the same nozzle during the deposition process. This technology, called material coextrusion, generates an extended range of material properties, which can gradually change at the design level, ensuring a combination or further bonding/interlocking between different materials. To take advantage of the opportunities offered by these technologies, it is necessary to understand the behavior of the combined materials according to material fractions. In their work, two pairs of compatible materials, namely polylactic acid (PLA)-thermoplastic polyurethane (TPU) and acrylonitrile styrene acrylate (ASA)-TPU, were investigated by changing the material fractions in the coextrusion process.

An original model describing the distribution of the materials is proposed. Based on this, the mechanical properties were investigated by analytical and numerical approaches. The analytical model was developed under the simplified assumption that the coextruded materials are an array of rods, while the more realistic numerical model is based on homogenization theory, adopting the FEM of a representative volume element. The analytical and numerical models show similar trends and it can be assumed that the finite element model has a more realistic behavior due to the higher accuracy of the model description. In this initial work, the results show good agreement between models and experiments, providing useful tools for designers and contributing to a new branch of additive manufacturing research.

(Ringel et.al, 2023) point out that, the pressure vessel qualification process in general is subject to governmental restrictions. Therefore, introducing MA components to the market is challenging due to incomplete standardization. To increase component safety and confidence, predetermined failure points can be integrated by design using direct manufacturing methods. A predetermined failure point via a surface notch is an option to avoid dangerous part failures (e.g. explosion) and increase safety. FEM is used for investigations on the influence of surface notch geometries applied to wall structures on the structural tension camber. Evaluation of the fracture surface and plastic elongation by optical microscopy and three-dimensional surface scanning elucidate the material behavior.

Analytical and experimental approval of an integrated predetermined failure point for static overload is achieved without reducing the maximum value of burst pressure. The integration meets all theoretical structural strength requirements. All tested specimens meet expectations for static strength and failure behavior. As expected, the elongation of the component decreases by using notches in the surface to cause failure.

The results lead to a proposed guideline for the application of an integrated predetermined failure point by using a surface notch.

(Bagewadi et.al, 2023) suggest that auxetic structure MA is gaining importance in various fields ranging from structural materials to biomaterials. This work proposes a novel design of auxetic structure with regular honeycomb, i.e. hybrid auxetic structure. All the models are fabricated in extrusion based MA technique using polylactic acid (PLA) material. A comparative study was carried out on 3D printed samples that were subjected to uniaxial tensile loading. To understand the effect on functional and mechanical properties, a tool path strategy with single-pass and zigzag tool paths was proposed. Solid material properties were used FEM to compare with experimental results. Experimental results indicate that the hybrid structure requires more load than the auxetic structure. The ultimate tensile strength (UTS) of the hybrid structure was 32% higher than that of the auxetic structure. The FEM results agree well with the experimental results. The observation reported in this paper will help to frame the design rules and selection of MA, processes for applications ranging from structural protective sandwich constructions with low density cores to tissue engineering and regenerative medicine structures.

(Wang et.al, 2023) indicate that, large sizes and destructive elements hindered standard uniaxial testing for in-service parts fabricated by laser additive manufacturing (LAM). Therefore, small-scale testing techniques, such as small punch tensile testing (SPT), were widely employed to evaluate local mechanical characteristics. In this study, layer-by-layer fabrication of Hastelloy X was produced by LAM-assisted ultrasonic micro-forging (UMFT) treatment.

The microstructure and fracture of the components showed that UMFT successfully reduced LAM defects; the defect distribution characteristics of sample blocks and SPT specimens were determined by the extreme value method (EVM); the effect of defect characteristics on strength was evaluated by Pearson's correlation coefficient (PCC). The results of various error evaluations showed that the proposed models had higher accuracy and stability than classical SPT models; According to the defect distribution in LAM blocks, the estimation error of total strength of LAM Hastelloy X was within 2%.

The simulation analysis with FEM, allows to develop iterations from the computational point of view (Soratos & Masayuki, 2020), to favor the study of test specimens or final products with a theoretical-experimental validation. In this work it is carried out with the pre-set conditions for the mechanical tensile test on plastic materials.

The phenomenon of analyzing with FEM in three dimensions favors the study of the specimens (Castorena *et al*., 2011) under the mechanical stresses in the tensile test, for this study only makes use of the data when the force is exerted on the Y axis, however, it is possible to analyze the stresses at different points and areas of action under different loads, which allows a greater amount of data to validate the test.

For the FEM simulated test, the specimen is designed in flat form with the specifications of the ASTM D638 standard, which allows performing the procedure in a specific way in the application of the loads required for the tensile test, whose design was essential for the development of the methodology in the simulation (Mansilla, 2001).

One of the advantages with FEM for the present work is the use of different numerical models at the time of performing the process in the simulation (Hernández & Cárdenas, 2018), managing to satisfy different parameters considered in the specimen for the simulated tensile experimentation, taking into account that the results have linear ranges, reaching the point at which the materials do not exceed the corresponding creep limit.

In addition, FEM is a tool used in engineering to obtain approximations (Ravindranath & Sidda, 2016) and validate elements within a component subjected to both internal and external loads, depending on the case study, with the purpose of optimizing the analysis.

The estimation of mechanical behavior, as shown in this work, is based on clearly specifying the models to be simulated (De Abreu & Pertuz, 2014) and their dimensional characteristics, as well as the control parameters in the application of mechanical tests such as tensile tests, for this case study.

The following paragraphs describe the equipment used, the shape of the specimen, the loads to which they are subjected in the simulation, obtaining data to represent them in the results.

Methodology

This work describes the types of materials used in additive manufacturing, through 3D printing, with the purpose of evaluating each of these with the tensile test in plastic materials and determine the physical properties of each specimen evaluated with the Finite Element Method (FEM), through this identify possible applications in different areas of engineering.

Additive manufacturing: is a tool in the pillars of Industry 4.0, which allows innovation in specific applications in technological processes. In other words, it represents improvements compared to conventional processes, where energy savings, cost reduction and time compacting are achieved in the production of specimens or products for their commercialization.

In the industry, the use of this tool has been agreed upon to improve production processes and to analyze quality processes in the design of test specimens for use as final products. In the academic sector, in educational institutions, it is an important part of the teaching-learning methodologies in the knowledge of new innovation technologies. And for research, it is essential to have additive manufacturing at the time of carrying out analysis in case studies that would be complicated to obtain by conventional manufacturing and with additive manufacturing is achieved by reducing manufacturing times and have specimens with the required specifications for the corresponding study.

3D Printing (I3D) and equipment to be used: I3D is the application of additive manufacturing to obtain physical models with the desired parameters and specifications, the following steps should be considered to develop a prototype or functional product:

– Sketch design for the specimen.
– Modeling in three dimensions with computer aided design (CAD) software.
– Export the model in stereolithography (stl) format.
– Analyze in the 3D printer software.
– Configuration of parameters for printing.
– Final product.

I3D makes use of various materials, among the most common are: Acrylonitrile Butadiene Styrene (ABS), Polylactic Acid (PLA), Nylon, Polyethylene terephthalate (PET), Acrylonitrile Styrene Acrylate (ASA), Polystyrene (HIPS), Polyamide (PA), Polypropylene (PP), Polycarbonate (PC), which are the most common in the field of I3D. Each of these materials has specific properties and applications that will depend on the conditions to which the products obtained with these materials are subjected.

On the other hand, the equipment to be used in the elaboration of the tensile specimens, for the case study, is a Zortrax model M200 Plus machine, whose characteristics are described in Table 1 and the equipment is shown in Figure 1.

Feature	Value/unit
Working volume	200x200x180 mm
Material diameter	1.75 mm
Nozzle diameter	0.3 mm, 0.4 mm, 0.6 mm
Connectivity	Wi-Fi, Ethernet, USB
Technology	LPD (Layer Plastic Depositiom)
Resolution	90-390 microns
Tem. Maximum Extruder Tem.	290°C
Max. Max. platform temp.	105°C
AC Input	110 V
Power	320 W

Table 1 Printer characteristics
Source: https://zortrax.com/3d-printers/m200-plus/

Figure 1 3D Printer, Zortrax M200 Plus
Source: Authors

ISSN 2523-6784

An important part of this work is to consider that the equipment shown in Figure 1 is used and that, if you want to obtain specimens with other 3D printers, you must take into account the processing software, which has different parameters in each of the equipment for I3D, and the control variables of the equipment, to obtain parts with the same specifications.

The use of I3D, favors the analysis for case studies related to the characterization of materials, since it allows identifying the way in which a specimen can be processed with the different materials used in additive manufacturing (Zmindak *et al*., 2020), achieving improvement trends in the study of materials.

Physical properties of plastic materials: the materials used for the study are ABS and PLA, so the properties to be compared in the FEM study should be mentioned, Table 2 shows the data for each of these materials and the properties to be evaluated.

Material	Property	Value/unit
ABS	Tensile stress	30.46 [MPa]
	Deformation	4.52 [%]
	Hardness	69.2 Shore D
	Flexural modulus	1.08 [GPa]
PLA	Tensile stress	47.95 [MPa]
	Deformation	3.8 [%]
	Hardness	79.8 Shore D
	Flexural modulus	1. 47 [GPa]

Table 2 Printer characteristics
Source: https://www.matweb.com/index.aspx

Characterization of plastic materials: it is a comparative evaluation of the plastic materials used in this research with additive manufacturing, carrying out the comparison of these by statistical means, determining the value in the physical properties of them. The characterization is carried out with destructive testing, tensile, by means of digital tools, FEM, based on ASTM standards (Kelly *et al*., 2018).

The tensile test is established under theoretical models mentioned below, where the specific variables for the development of the test are observed (Lohdy *et al*. 2020), (Cakmak *et al*., 2022), which will be evaluated with FEM.

One of the mechanical properties is the shear modulus, which can be obtained by means of equation 1, as well as the elongation ratio, with equation 2.

$$G = E/(2(1 + v)) \quad (1)$$

$$\varepsilon_y = \Delta L/L \quad (2)$$

For which the stress and strain components are established, according to the symmetry in anisotropic materials, as shown in Table 3.

Material symmetry	No symmetry	Oriented-monoclinic	Oriented-orto
Constants ≠0	36	20	12
$\sigma_1, \sigma_2, \sigma_6$	≠0	≠0	≠0
$\sigma_3, \sigma_4, \sigma_5$	=0	=0	=0
$\varepsilon_1, \varepsilon_2, \varepsilon_3, \varepsilon_6$	≠0	≠0	≠0
$\varepsilon_4, \varepsilon_5$	≠0	=0	=0

Table 3. Stress and plane strain
Source: Design and analysis of materials Editorial Reverte

Table 3 shows the parameters to be analyzed with the FEM tool, according to the conditions in the specimen characterization, and to validate the information of the material subjected to the simulation tests.

Three-dimensional model: the model is made with a CAD modeler, whose image is shown in Figure 2.

Figure 2 Tensile specimen in the CAD modeler
Source: Authors: Authors

The model was made according to the characteristics established in the ASTM-D638 standard, taking into account each of the classifications for types I, II, III, IV and V, for the application in the characterization of the specimen.

Application of FEM: to apply the FEM tool in each of the test specimens, the parameters to be analyzed are as follows for the ABS material are shown in Table 4. Representing P (parameter), VM (Von Mises), EY (Normal stress in Y) and DS (displacement in Y).

Property	Load [kN]	Variable (Mesh in %)				
		MP 2	MP 4	MP 6	MP 8	MP10
VM [MPa]	0.05	1.42	1.23	1.19	1.17	1.15
	0.1	2.84	2.46	2.40	2.33	2.30
	0.2	5.68	4.93	4.74	4.66	4.59
	0.25	7.1	6.16	5.92	5.83	5.74
	0.5	14.19	12.32	11.85	11.66	11.48
	1	28.38	24.64	23.69	23.32	22.96
	2.5	70.96	61.62	59.24	58.3	57.4
	5	141.9	123.2	118.5	116.6	114.8
	10	283.8	246.4	236.9	232.2	229.6
	15	425.8	369.7	355.5	349.9	344.4
EY [MPa]	0.05	1.44	1.22	1.21	1.91	1.16
	0.1	2.87	2.44	2.43	2.38	2.32
	0.2	5.74	4.88	4.85	4.76	4.64
	0.25	7.17	6.10	6.06	5.96	5.80
	0.5	14.34	12.2	12.13	11.91	11.6
	1	28.69	24.41	24.25	23.82	23.19
	2.5	71.72	61.02	60.63	59.56	57.98
	5	143.4	122	121.3	119.1	116
	10	286.9	244.1	242.5	238.2	231.9
	15	430.4	366.2	363.8	357.4	347.9
DS	0.05	0.053	0.053	0.052	0.052	0.052
	0.1	0.11	0.10	0.10	0.10	0.10
	0.2	0.21	0.21	0.21	0.21	0.21
	0.25	0.26	0.26	0.26	0.26	0.26
	0.5	0.53	0.53	0.52	0.52	0.52
	1	1.06	1.05	1.04	1.04	1.04
	2.5	2.65	2.63	2.60	2.57	2.59
	5	5.30	5.27	5.19	5.19	5.18
	10	10.59	10.53	10.38	10.38	10.35
	15	15.9	15.8	15.58	15.58	15.53

Table 4 Parameter values obtained with 2-8% meshed FEM, ABS material.
Source: Authors

Table 4 identifies the data collected after the application of the tensile test in a virtual way (simulation), with the purpose of identifying the optimum values for the material, load and type of mesh for each classification in percentage of ABS.

Figure 3 shows some examples of the application of loads on the specimen and the meshing of the specimen.

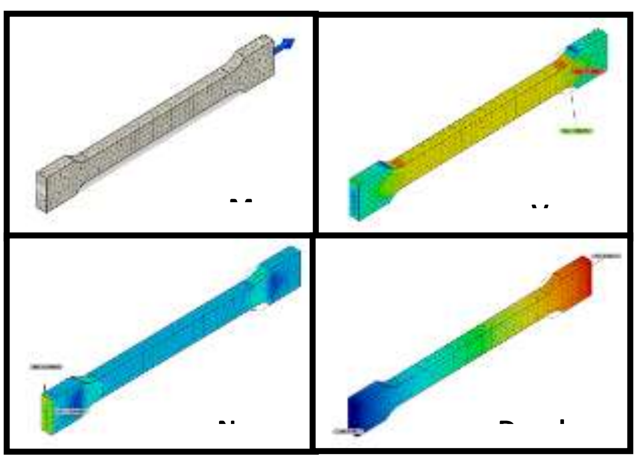


Figure 3 Application of FEM in ABS material
Source: Authors

The simulation with the FEM tool analyzes the Von Mises stress, being an anisotropic material, normal stress in Y, corresponding to the tensile test simulation and the displacement in the Y-axis. The data obtained will be analyzed and discussed in the corresponding sections.

Figure 4 shows the percentages applied to the ABS material with the similarity in the meshing.

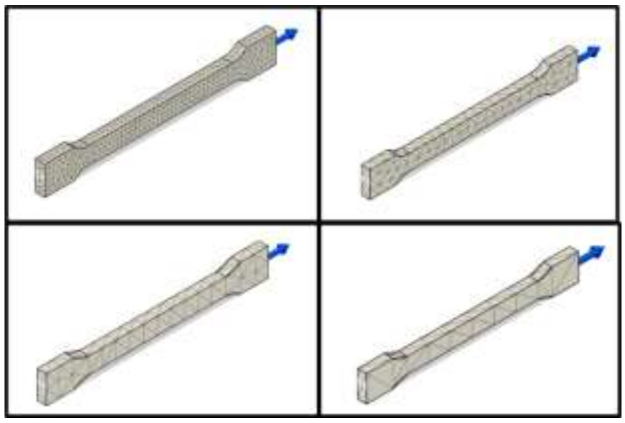


Figure 4. 2-10% mesh
Source: Authors

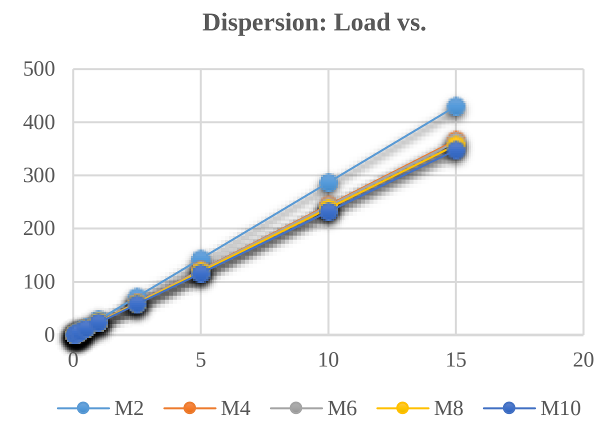
The meshing performed by means of the FEM simulator allows visualizing the percentage of filling of the specimen to carry out the characterization and verify the values of the mechanical properties according to the elements considered for the simulation (Castaño *et al*., 2020).

The following section analyzes the data obtained in the simulation and the optimization of the ABS material for its use in specific cases that meet the specifications, both of the material and of the operating conditions in the tensile tests for each of the specimens evaluated.

Results

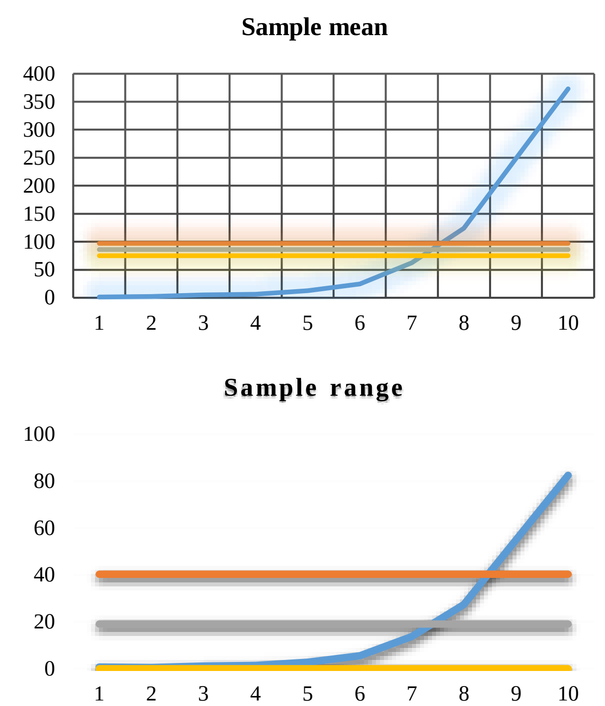
Firstly, the scatter plots corresponding to the load vs. the type of meshing are shown, validating that its behavior is linear, which was obtained with respect to each of the loads applied in the simulation by means of FEM. The graphs are shown in Figure 5.

ISSN 2523-6784



Graph 1. Load vs. 2-10% meshing graph.
Source: Authors

On the other hand, there is a graph, figure 6, with respect to the mean of the sample (M2) and the range of this sample in order to establish statistical points for the validation of future simulations with other materials with similar specifications to those of ABS.



Graph 2 Graph of mean and range for M2
Source: Authors

Next, the data is represented by means of information collected in Table 5, which indicates the stress values in Y, compared with the theoretical-experimental data, whose value corresponds to 30.46 [MPa].

PF	Load	Simulation				
	[kN]	M2	M4	M6	M8	M10
EY [MPa]	1 1.25	28.69 71.72	24.41 61.02	24.25 60.63	23.82 59.56	23.19 57.98

Table 5 Simulated Y-stress values with theoretical
Source: Authors

The purpose of Table 5 is to offer the data that represent the characterization of the specimens subjected to tensile stress, from the simulation point of view with the FEM tool and thus establish the amount of filler in a specimen manufactured with additive manufacturing with I3D.

Using the numerical interpolation method, the value of the load to which the specimen should be subjected is obtained, together with the meshing percentage and thus provide the optimization in the characterization with FEM, as shown in Table 6.

PF	Load	Simulation	
	[kN]	M2	E[%]
EY [MPa]	1 1.01 1.25	28.69 30.46 71.72	5.81 0 135.45
	Carga	M4	
EY [MPa]	1 1.04 1.25	24.41 30.46 61.02	19.86 0 100.32
	Carga	M6	
EY [MPa]	1 1.04 1.25	24.25 30.46 60.63	20.38 0 99.05
	Carga	M8	
EY [MPa]	1 1.05 1.25	23.82 30.46 59.56	21.79 0 95.53
	Carga	M10	
EY [MPa]	1 1.05 1.25	23.19 30.46 57.98	23.87 0 90.34

Table 6 Percentage of error in the evaluation
Source: Authors

With the information provided in Table 6, it is determined that M2 with a load of 1 kN corresponds to the parameter that optimizes the system for the material to be used in tensile effects. Taking into account that the load range required for this solicitation, characterization of the ABS material, is in [1-1.01] kN. If a higher resistance is required, the M10 would be used with stress in the range of [23.87-30.46]. However, with the results obtained in this study, it is possible to establish characteristic values for different applications where a material, ABS, is required with evaluated stresses and loads.

ISSN 2523-6784

On the other hand, we have the data where the amount of elements that are contemplated in the simulation to optimize the results are identified, which have the opportunity to be modified to improve in each of the simulation processes the results at the moment of executing the FEM simulation with the corresponding variables, in table 7, it is observed as mentioned.

Mesh	Nodes	No. elements
M2	6127	3490
M4	978	441
M6	454	195
M8	401	170
M10	235	94

Table 7 Variables in FEM simulation
Source: Authors

Table 7 represents the control to be compared with the meshing and the infill content in the I3D, which means that as the nodes and elements increase the percentage of infill is higher, having as equivalences M2-100% infill, M4-80%, M6-60%, M8-40% and M10-20%.

It is important to highlight that in the simulations the number of elements and thus the nodes can be modified to optimize the simulations and obtain standardized data to make comparisons between different materials.

The use of the FEM tool in the characterization for a mechanical tensile test has the advantage of speeding up the process and obtaining values of the materials being evaluated to identify the application according to the values obtained, according to the variables to be analyzed.

Conclusions

A mechanical test known as traction has been applied in a general way for materials engineering in technological innovation processes such as additive manufacturing with I3D, through a simulation tool such as FEM, obtaining favorable results to compare what has been done with theoretical experimentation and formulating new trends of analysis and characterization of engineering materials.

This leads to standardize the use of I3D with materials that meet the basic properties for its performance as a final product. In this sense, this work seeks, at all times, to provide data that other researchers and teachers of academic institutions can take advantage of for their performance in the field of engineering applications.

The use of FEM currently favors the time and space of engineering execution, taking into account that the characterization of engineering applications is developed with specific criteria in the evaluation, such as physical, chemical, optical, thermal properties, among others, with the simulation tool.

An important point with the use of FEM simulation lies in controlling the percentage of mesh, nodes and elements in the test specimen, with which the optimization of the object is achieved according to the mechanical stresses, for this case study, with respect to the tensile test.

The results showed a congruence with the behavior of the test and the simulation, appreciating at all times that the higher the mesh percentage, the more fragile and the lower the stress value in the area of action (Y).

The data obtained are used to determine the application of the specimen, with respect to the resistance of the material and the deformation it presents. This achieves the interest of the research developed in this work to strengthen the research communities in additive manufacturing with I3D.

In relation to the graphs there is a linear trend, for dispersion, with the behavior of the material when subjected to the test and also the graph that allows to increase the application of statistics to the characterization of the ABS material and the possible evaluation of other materials with this tensile test in the FEM simulator, control graphs were used to identify trends in terms of means and ranges for each of the chosen meshes and their similarity with the percentage of filler in the I3D.

The improvements to be made are directly focused on the use of the FEM software modules to optimize the greater amount of parameters and variables indicated in the simulations, in addition, to offer integrating projects in the academic and research communities to strengthen the use of innovation technologies, such as the MA with I3D, the numerical analysis to verify and validate the data obtained in each one of the case studies immersed in the applied engineering.

In the future, FEM will be applied with other materials used in the MA to enrich and increase the databases in the characterization for the tensile test and later other destructive mechanical tests. Without losing sight of the application of statistics and numerical analysis as engineering tools in the validation and optimization of engineering processes.

References

Bagewadi, SS, Sugavaneswaran, M. y Bhagchandani, RK (2023). Investigación sobre propiedades mecánicas de estructuras auxéticas híbridas de fabricación aditiva. Revista de ingeniería y rendimiento de materiales, 32 (1), 68-81. https://link.springer.com/article/10.1007/s11665-022-07074-y

Cakmak, S., Meric, B., Dal, M., Sarisac, S. & Mecit, D. (2022). Finite element análisis of the tensile strength of laminated fabric for automotive seat covers. *Textile Research Journal*. 93(1-2), pp 409-21. https://doi.org/10.1177/00405175221118838

Castaño-Urrego, C., López-Pérez, J., Forero-Rubiano, R. & Tlapanco-Rios, E. (2020). Método de comparación de resultados de modelado CAD-CAE contra probetas de ensayo destructivo. *Acta Universitaria*. 30. ISSN: 2007-9621, http://doi.org/10.15174.au.2020.2668

Castorena-González, J.H., Calderón-Guillén, J.A., Almeraya-Calderón, F., Gaona-Tiburcio, C., Almaral-Sánchez, J.L. & Martínez-Villafañe, A. (2011). Análisis con elemento finito de los esfuerzos expansivos por corrosión en las estructuras de concreto reforzado. *Ingeniería Investigación y Tecnología*, 7(1), pp 1-7. ISSN: 1405-7743. https://doi.org/10.22201/fi.25940732e.2011.12n1.001

De Abreu, M. & Pertuz, A. (2014). Estimación del ensayo de compresión con deformación plana tipo Ford utilizando el método de elementos finitos y utilizando curvas de comportamiento obtenidas en un ensayo de tracción. *Revista de la Facultad de Ingeniería U.C.V.* 29(2), pp 79-92. http://ve.scielo.org/scielo.php?script=sci_arttext&pid=S0798-40652014000200008&lng=es&tlng=es.

Hernández, S.& Cárdenasa, E. (2018). Analysis theoretical-practice of efforts and by finite elements of a tensión test. *Ingenio Magno,* 9 (1), pp 42-55. ISSN: 2145-9282. http://revistas.ustatunja.edu.co/index.php/ingeniomagno/article/view/1643

Jung, J., Meyer, G., Greiner, M. y Mittelstedt, C. (2023). Diseño de muestras de introducción de carga para la caracterización mecánica de estructuras de celosía bajo carga de tracción. Revista de Fabricación y Procesamiento de Materiales, 7 (1), 37. https://doi.org/10.3390/jmmp7010037

Kelly, J., Cyr, E. & Mohammadi, M. (2018). Finite element Analysis and Experimental Characterisation of SMC Composite Car Hood Specimens under Complex Loadings. *Journal of Composites Science.* 2(53), pp 2-24. https://doi:10.3390/jcs2030053

Lohdy, D., Ghani, A. & Muhammad, G. (2020). Tensile test simulation for polymer composites. *Proceedings of the International Conferences on Innovation in Science and Technology*. 208, pp 200-206. https://doi.org/10.2991/aer.k.211129.044

Mansilla, A., Regidor, D. & Negro, A. (2001). Dynamic tensile testing determining the stress-strain curve at different strain rate. *Revista de Metalurgía,* 37, pp 255-259. https://doi.org/10.3989/revmetalm.2001.v37.i2.475

Ravindranath, K. & Sidda, D. (2016). Elastic and elasto-plastic finite element análisis of a tensión test specimen with and without voids. *International Journal of Develoment Research*. 6(12), pp 10527-10533. https://www.journalijdr.com/sites/default/files/issue-pdf/6833.pdf

Ringel, B., Schwarz, D., Vu, HM, Meiniger, S., Schlick, G., Seidel, C. y Oechsner, M. (2023). Investigación sobre un punto predeterminado de falla para componentes cargados a presión de acero inoxidable 316L hechos por fusión de lecho de polvo láser a través de análisis de estrés y pruebas experimentales. Avances en la fabricación aditiva, 1-9. https://link.springer.com/article/10.1007/s40964-022-00388-2

Soratos, T. & Masayuki, K. (2020). An application of FEM in the determination of tensile properties for work-hardened carbon Steel by means of small punch test. *Results in Materials*. 8, pp 1-10. https://doi.org/10.1016/j.rinma.2020.100142

Sponchiado, R., Rosso, S., Dal Fabbro, P., Grigolato, L., Elsayed, H., Bernardo, E., & Savio, G. (2023). Modelado de Coextrusión de Materiales en Fabricación Aditiva de Polímeros. Materiales, 16 (2), 820. https://doi.org/10.3390/ma16020820

Wang, X., Xu, L., Zhao, L. y Han, Y. (2023). Una solución para estimar la resistencia a alta temperatura basada en las características de fabricación aditiva. Revista Internacional de Ciencias Mecánicas, 108124. https://www.sciencedirect.com/science/article/pii/S0020740323000267

Zmindák, M., Novák, P., Soukup, J., Milosavljevic, D. & Kaco, M. (2020). Finite element simulation of tensile test of composite materials manufactured by 3D printing. *IOP Conf. Series: Materials Science and Engineering.* 776, https://doi:10.1088/1757-899X/776/1/012082