Journal of Experimental Systems 11[30]1-14: e51130114
Article

Optimization of the finite element meshing for the fabrication of a M3x12x0.5 screw, in 3D printing

Optimización del mallado en elemento finito para la fabricación de un tornillo M3x12x0.5, en impresión 3D

González-Sosa, Jesús Vicente*[a], Avila-Soler, Enrique[b] and Zavala-Osorio, Yadira[c]

[a] Universidad Autónoma Metropolitana • 0000-0002-1325-0266 • 166452
[b] Universidad Autónoma Metropolitana • 0000-0001-8980-0925 • 360262
[c] Universidad Autónoma Metropolitana • 0000-0001-5337-6624 • 104843

CONAHCYT classification:

Area: Engineering
Field: Engineering
Discipline: Mechanical engineer
Subdiscipline: Mechanical design

https://doi.org/10.35429/JOES.2024.11.30.1.14
Article History:
Received: January 13, 2024
Accepted: December 31, 2024

* ✉ [jvgs@azc.uam.mx]

Abstract

Meshing optimization in the finite element method (FEM) is key to designing mechanical components such as screws. This study focuses on the M3x12x0.5 screw, improving its performance and durability through the appropriate selection of meshing parameters. FEM software was used to analyze its behavior under different loads, comparing it with a 3D printed model. The results showed improvements in the accuracy of predictions about resistance, deformation and stress distribution, allowing the design to be optimized. This highlights the relevance of simulation tools in the development of mechanical components.

Objective	Methodology	Contribution
The objective of this section is the optimization of meshing in the finite element method (FEM) for designing mechanical components such as screws.	This study focuses on the M3x12x0.5 screw, improving its performance and durability through the appropriate selection of meshing parameters. FEM software was used to analyze its behavior under different loads, comparing it with a 3D printed model.	Las mejoras en la precisión de predicciones sobre resistencia, deformación y distribución de tensiones, permitiendo optimizar el diseño. Esto resalta la relevancia de las herramientas de simulación en el desarrollo de componentes mecánicos.

Additive manufacturing, characterization, computational simulation, optimize, identification

Resumen

La optimización del mallado en el método de elementos finitos (FEM) es clave para diseñar componentes mecánicos como tornillos. Este estudio se centra en el tornillo M3x12x0.5, mejorando su rendimiento y durabilidad mediante la selección adecuada de parámetros de mallado. Se usó software FEM para analizar su comportamiento bajo diferentes cargas, comparándolo con un modelo impreso en 3D. Los resultados mostraron mejoras en la precisión de predicciones sobre resistencia, deformación y distribución de tensiones, permitiendo optimizar el diseño. Esto resalta la relevancia de las herramientas de simulación en el desarrollo de componentes mecánicos.

Objetivo	Metodología	Contribución
El objetivo de este apartado es la optimización del mallado en el método de elementos finitos (FEM) para diseñar componentes mecánicos como tornillos	Este estudio se centra en el tornillo M3x12x0.5, mejorando su rendimiento y durabilidad mediante la selección adecuada de parámetros de mallado. Se usó software FEM para analizar su comportamiento bajo diferentes cargas, comparándolo con un modelo impreso en 3D.	Las mejoras en la precisión de predicciones sobre resistencia, deformación y distribución de tensiones, permitiendo optimizar el diseño. Esto resalta la relevancia de las herramientas de simulación en el desarrollo de componentes mecánicos.

Manufactura aditiva, caracterización, simulación computacional, optimización, identificación

Citation: González-Sosa, Jesús Vicente, Avila-Soler, Enrique and Zavala-Osorio, Yadira. [2024]. Optimization of the finite element meshing for the fabrication of a M3x12x0.5 screw, in 3D printing. Journal of Experimental Systems. 11[30]-1-14: e51130114.

Introduction

Currently, the use of computational tools for product validation has found room for improvement in the development, research and manufacture of specimens for their study with scientific foundations in engineering. In this regard, the Universidad Autónoma Metropolitana, Unidad Azcapotzalco, together with researchers in various engineering areas, have applied techniques such as the Finite Element Method (FEM) and additive manufacturing with 3D printing (MAI3D) to validate industrial case studies. This paper analyzes and establishes the optimization of a screw in order to improve its manufacturing and to understand the physical phenomena involved in the aforementioned specimen.

(Zhu et al. 2023) presents a highly relevant study that highlights how FEM simulators have the right conditions to make a significant contribution to science and technology. This approach takes into account the design considerations and specific parameters of each case study, allowing a better understanding of the physical phenomena affecting the specimens under analysis. In their work, specific conditions related to the case study are presented in order to facilitate the understanding and analysis of the variables involved in the performance of the screw.

Stress concentration in a FEM simulation (Figueroa-Díaz et al., 2019) allows validation of analytical results that would otherwise not be evident in the physical phenomenon. In this sense, simulations offer a valuable alternative to carry out specific studies to improve design conditions and material selection, thus facilitating the feasibility of the characterization methodology.

According to (Lee et al. (2017), simulation behaviors using the FEM are customized, which allows this tool to be efficiently adapted to each case study. This results in significant savings in execution times during the evaluation of such cases. In particular, the importance of torque is highlighted in the corresponding simulation, where a match is obtained in the analysis of a test specimen, which facilitates the identification of improvements.

The current approach (Sheng et.al, 2019) could serve as a guide to select internal fixation parameters in a more efficient manner, potentially facilitating modification and optimization of the internal fixation system. In comparison, a study using a finite element simulator (FEM) of a fastener allows a detailed analysis of the mechanical behavior and factors affecting the performance of the fastening system. This suggests that the integration of both approaches, the present method and the results obtained through FEM simulations, could result in an even more informed and accurate choice of fasteners, thus promoting significant improvements in the design and functionality of the internal fastening system.

(Hsia et al., 2016) emphasize the importance of choosing the right combination of the model in CAD to achieve accurate simulation using the evaluation variables. This choice allows to keep errors to a minimum during the process with the FEM. In this way, evaluation alternatives can be identified, which are carried out in this work under controlled conditions.

According to (Cetin et al., 2022), nowadays, most engineering applications use FEM simulation processes instead of laboratory experiments. This is due to the convenience of running simulations, which have proven to be accurate compared to the results obtained through experimentation.

According to studies by (Chandra et al., 2021), as the simulation load on threaded fasteners increases using FEM, the deformation increases continuously and linearly. It is observed that the load remains within the elastic or plastic zone, without reaching a fracture. Therefore, it is essential to perform mechanical tests to accurately identify this behavior, since the FEM only provides virtual alternatives.

In their study, (Xue et al., 2022) analyzed the optimal fastening of screws used in the medical sector using the FEM. In addition, they examined fracturing through the variables provided by this tool in the context of simulation.

ISSN: 2410-3950
RENIECYT-CONAHCYT: 1702902

González-Sosa, Jesús Vicente, Avila-Soler, Enrique and Zavala-Osorio, Yadira. [2024]. Optimization of the finite element meshing for the fabrication of a M3x12x0.5 screw, in 3D printing. Journal of Experimental Systems. 11[30]-1-14: e51130114.
https://doi.org/10.35429/JOES.2024.11.30.1.14

The use of simulators, such as the FEM, for the evaluation of mechanical elements enriches technological development and fosters innovation in the engineering field. Several researchers have identified this tool as a valuable resource for improving conditions in the case studies analyzed. The following paragraphs detail the methodology used to simulate the screw in the context of the case study, in order to obtain data to be presented in the results.

Methodology

The procedure to develop this work is given from first making the specimen with a CAD modeler, establishing the dimensions of the screw as a case study, then apply FEM to validate the results, depending on the specific variables in a screw when subjected to torque, accumulate the data, apply optimization steps in the meshing to reach the specimen that meets the mechanical stresses and finally set the parameters for the manufacture of the screw with 3D printing (I3D).

The following paragraphs describe each of the stages mentioned to identify the process in the evaluation of the case study, screw M3x12x0.5.

Specimen modeling: in this section the measurements corresponding to the case study, M3x12x0.5 bolt, are taken into consideration and the dimensional drawing is developed, then the software tools are used to obtain the product in 3 dimensions (solid). Figure 1 shows the general outline of the screw.

Box 1

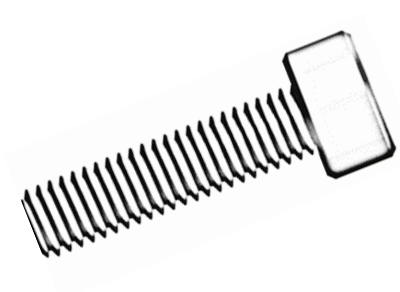

Figure 1
Sketch of specimen, screw

Source: Own elaboration

According to the conditions of the specimen, screw, shown in Figure 1, it will be analyzed from the mechanical point of view with the torsion parameters and identify the maximum loads it will withstand when manufactured with specific and general I3D materials. Figure 2 shows the drawing of the specimen generated with the CAD software, with the screw specifications.

Box 2

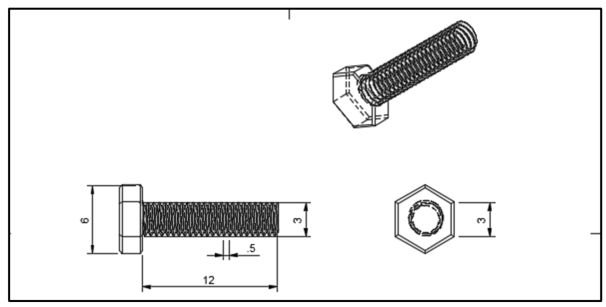

Figure 2
Plan corresponding to the screw to be evaluated

Source: Own elaboration

Figure 3 shows the 3-dimensional model with the parameters and variables defined by the data sheet in the screw classification.

Box 3

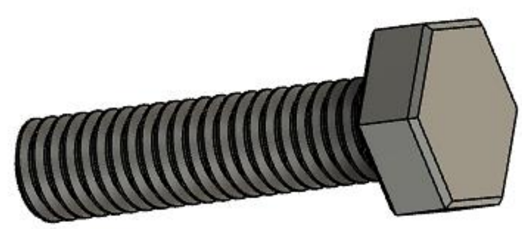

Figure 3
Screw M3x12x0.5 elaborated in CAD software

Source: Own elaboration

Figure 3 shows the screw in its three-dimensional perspective to be evaluated with the FEM tool and to obtain the best conditions for its manufacturing with MAI3D.

The materials to be used in the FEM simulations and fabrication in I3D are: Acrylonitrile Butadiene Styrene (ABS), Polylactic Acid (PLA) and NYLON 6/6, for which we have the physical properties identified in Table 1.

ISSN: 2410-3950
RENIECYT-CONAHCYT: 1702902

González-Sosa, Jesús Vicente, Avila-Soler, Enrique and Zavala-Osorio, Yadira. [2024]. Optimization of the finite element meshing for the fabrication of a M3x12x0.5 screw, in 3D printing. Journal of Experimental Systems. 11[30]-1-14: e51130114.
https://doi.org/10.35429/JOES.2024.11.30.1.14

Box 4

Table 1
Physical properties of materials

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]
NYLON	Tensile stress	33.22 [MPa]
	Deformation	9.17 [%]
	Hardness	62 Shore D
	Flexural modulus	0.78 [GPa]

Source: https://www.matweb.com/index.aspx

The variables analyzed in the FEM software are Flexural Modulus (MF), Deformation (DF) and Stress (EF), which determine the optimal conditions of the specimen for its fabrication in I3D.

FEM application: for the FEM simulation, the parameters shown in Table 2 were considered, which are fundamental for the proper development of the simulations corresponding to the described screw.

Box 5

Table 2
Parameters to be used in the simulation with FEM

Parameter	Description
Material to simulate	Plastic ABS Nylon 6/6 Softe plastic (PLA)
Load	Torque of 1600 Nmm on the screw chord.
Meshing	Parabolic 60° Minimum element size 20%

Source: Own elaboration

The materials designated in the simulation have been chosen because of their use for the manufacture of logs in I3D. After these simulations, some others can be selected from the software databases to achieve the simulation and an approach with the functional materials in the manufacturing process with additive manufacturing.

Regarding the load used in the simulation and shown in Table 2, it has been considered from the technical specifications of bolts made of plastic materials. The load application zone, torsion, is carried out on the rope, which is the weak one of a screw of these characteristics, as far as its size is concerned.

In the case of meshing, it is carried out with the characteristics mentioned in Table 2, which offers a simulation and depending on the results, this parameter will have considerable variations to achieve the optimization of the screw and the correct manufacturing of the specimen. Figure 4 below shows the first simulation for the specimen with the ABS material.

.

Box 6

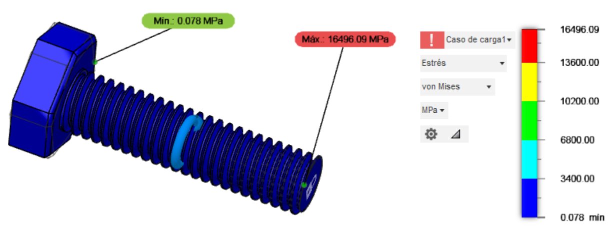

Figure 4
FEM ABS simulation

Source: Own elaboration

Figure 4 identifies the maximum and minimum value of the Von Mises load in MPa for the specimen with the material called ABS, these values are: minimum 0.078 and maximum 16496, both in MPa. It is observed, according to the color range, that the specimen has no risk of suffering any damage when the corresponding load of 1600 Nmm is applied.

Figure 5 below shows the maximum and minimum values for the nylon 6/6 material.

ISSN: 2410-3950
RENIECYT-CONAHCYT: 1702902

González-Sosa, Jesús Vicente, Avila-Soler, Enrique and Zavala-Osorio, Yadira. [2024]. Optimization of the finite element meshing for the fabrication of a M3x12x0.5 screw, in 3D printing. Journal of Experimental Systems. 11[30]-1-14: e51130114. https://doi.org/10.35429/JOES.2024.11.30.1.14

Box 7

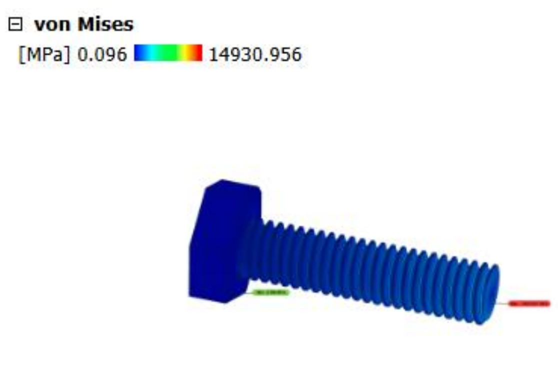

Figure 5
FEM nylon 6/6 simulation

Source: Own elaboration

For the case of the material known as Nylon 6/6, the stress is in the range of 0.096 to 14930.56 MPa, which has a decrease of 9.48% with respect to the ABS material. It is important to mention that in all the simulations the application of the load, torsion, is performed in the same environment, rope, so that the results obtained can be compared and subsequently the corresponding optimization can be carried out. Figure 6 shows the simulation of the third PLA material, which has the qualitative characteristics of being soft.

Box 8

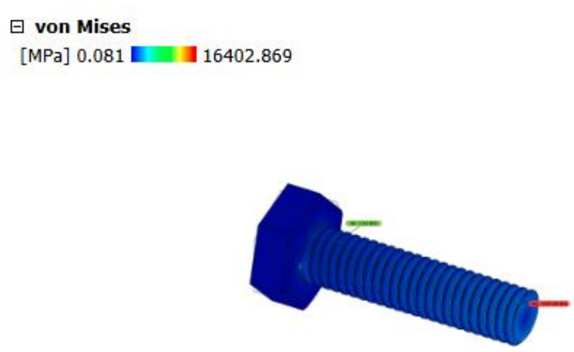

Figure 6
FEM PLA simulation

Source: Own elaboration

Figure 6 shows that the material called soft plastic has the intermediate values among the three cases, 0.081 to 16402.87 MPa, indicating that it is a material that maintains its properties congruent with the other materials.

The three materials have similar behaviors at the time of performing the FEM simulations, however, the following tables will identify the critical aspects of the simulation to identify as far as possible the application of the optimization process.

Table 3 concentrates the data obtained from the FEM simulation for the M3x12x0.5 screw, using the ABS material.

Box 9

Table 3
Simulation data obtained for the screw in ABS material

Material	Property	Value/unit
	Von Mises Stress	16496.09 [MPa]
	Principal Stress	11550.06 [MPa]
	Normal in XX	1663.35 [MPa]
	Normal in YY	2752.77 [MPa]
	Normal in ZZ	4070.01 [MPa]
	To XY cut	2661.74 [MPa]
	To YZ cut	1635.49 [MPa]
	To ZX cut	2523.50 [MPa]
ABS	Total displacement	10.86 [mm]
	X displacement	0.15 [mm]
	Y displacement	10.77 [mm]
	Z displacement	8.41 [mm]
	Total reaction force	31.19 [N]
	Force at X	1.96 [N]
	Force at Y	11.70 [N]
	Force at Z	29.48 [N]
	Deformation	13.28 [%]

Source: Authors.

The values shown in Table 3 are representative in the FEM simulation for the M3x12x0.5 bolt, which has the characteristic of withstanding a force of 31.19 N, when a torque value of 16000 Nmm is applied. This force has a variation of 0.12 to 7% with the materials selected for the screw analysis.

A characteristic for the displacement at the time of performing the FEM simulation is shown in Figure 7, for the ABS material, identifying within the range of colors that it is found with a value of 10.26 mm in the part called the head of the screw, in future simulations it is sought to reduce the value obtained.

ISSN: 2410-3950
RENIECYT-CONAHCYT: 1702902

González-Sosa, Jesús Vicente, Avila-Soler, Enrique and Zavala-Osorio, Yadira. [2024]. Optimization of the finite element meshing for the fabrication of a M3x12x0.5 screw, in 3D printing. Journal of Experimental Systems. 11[30]-1-14: e51130114.
https://doi.org/10.35429/JOES.2024.11.30.1.14

Box 10

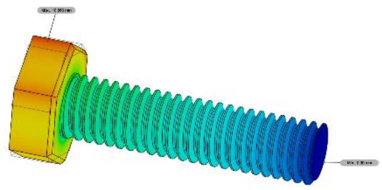

Figure 7
Displacement in the ABS material specimen

Source: Own elaboration

Table 4 below shows the simulation values applied to the screw with Nylon 6/6 material, highlighting the representative physical properties.

Box 11

Table 4
Simulation data obtained for the screw in Nylon 6/6 material

Material	Property	Value/Unit
Nylon 6/6	Von Mises Stress	14930.96 [MPa]
	Principal Stress	4612.21 [MPa]
	Normal in XX	970.58 [MPa]
	Normal in YY	2919.58 [MPa]
	Normal in ZZ	2172.08 [MPa]
	To XY cut	7059.78 [MPa]
	To YZ cut	2544.79 [MPa]
	To ZX cut	3081.28 [MPa]
	Total displacement	16.41 [mm]
	X displacement	2.64 [mm]
	Y displacement	2.57 [mm]
	Z displacement	0.72 [mm]
	Total reaction force	33.54 [N]
	Force at X	11.51 [N]
	Force at Y	10.46 [N]
	Force at Z	20.75 [N]
	Deformation	8.63 [%]

Source: Own elaboration

The percentage of comparison with respect to the ABS and Nylon 6/6 material, in the main stress property, observed in Table 4, is 60.06%, values that exceed an equilibrium in this parameter.

As in the case analyzed with the ABS material, now in Nylon 6/6, Figure 8 shows the color scale for the displacement property.

Box 12

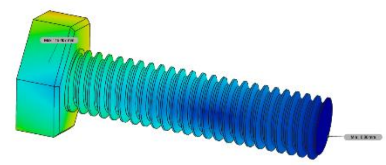

Figure 8
Displacement for Nylon 6/6 material

Source: Own elaboration

The value of the displacement is 16.41 mm, Figure 8, identifying an increase with respect to the previous material. In subsequent simulations we will analyze how to homogenize the property and thus achieve the appropriate optimization for the specimen.

Table 5 shows the effects of the FEM simulation on the PLA material under the stresses applied to each of the simulations with the described variables.

Box 13

Table 5
Data obtained from simulation for the screw in PLA material

Material	Property	Value/Unit
PLA	Von Mises Stress	16402.87 [MPa]
	Principal Stress	11598.67 [MPa]
	Normal in XX	1763.12 [MPa]
	Normal in YY	2856.72 [MPa]
	Normal in ZZ	4166.84 [MPa]
	To XY cut	2633.45 [MPa]
	To YZ cut	1640.10 [MPa]
	To ZX cut	2511.43 [MPa]
	Total displacement	26.92 [mm]
	X displacement	0.367 [mm]
	Y displacement	26.70 [mm]
	Z displacement	20.86 [mm]
	Total reaction force	31.23 [N]
	Force at X	2.06 [N]
	Force at Y	11.70 [N]
	Force at Z	29.46 [N]
	Deformation	32.83 [%]

Source: Own elaboration

For the case of deformation, according to the application of the 16000 Nmm, PLA material, named as soft material, has a higher percentage of deformation compared to ABS and Nylon 6/6, which has the value of 32.83.

Journal of Experimental Systems 11[30]1-14: e51130114
Article

Figure 9, shown below has the characteristic of observing the displacement, whose zone of action is seen in the upper part of the screw, head, where the applied torque exerts the physical effects for deformation.

Box 14

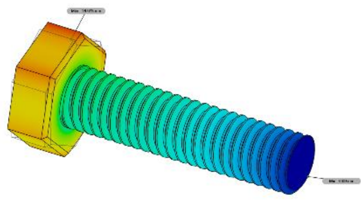

Figure 9
Displacement for PLA material

Source: Own elaboration

Each of the figures shown has the characteristic of showing the range of colors that allows to visualize the effect of the load applied during the FEM simulation on the M3x12x0.5 bolt as part of the specimen of study and analysis.

In the following section, part of the data obtained in the optimization of the screw with FEM simulation is described and observed.

Optimization of FEM in the specimen: the optimization part of the screw for its fabrication with I3D will be described in different stages, which are developed below.

First stage: decrease the number of nodes and elements obtained when performing the FEM simulation. In the data of the standard simulations the number of nodes are 16959 and elements 16167, after making the first changes in the simulation 2426 nodes and 9631 elements were obtained, for the following changes 2554 nodes, 10137 elements and in the last change of parameters 5463 nodes with 24257 elements were obtained, which is indicating that simulations 2 and 3 are the ones that decrease these two variables in the meshing for the FEM simulations. Second stage: a change is made in the format for the meshing in the screw, initially we worked with the mesh in parabolic form and the change, to optimize, was carried out with the meshing in linear form. With this variation, the data shown in Table 6 for ABS, Table 7 with Nylon and Table 8 with PLA material are obtained.

Box 15

Table 6
Simulation data obtained for the screw in ABS material optimizing the linear meshing arrangement

Material	Property	Value/Unit
PLA	Von Mises Stress	3713.26 [MPa]
Linear	Principal Stress	1689.89 [MPa]
Meshing	Normal in XX	522.56 [MPa]
	Normal in YY	818.13 [MPa]
	Normal in ZZ	634.43 [MPa]
	To XY cut	1212.17 [MPa]
	To YZ cut	460.17 [MPa]
	To ZX cut	651.38 [MPa]
	Total displacement	9.17 [mm]
	X displacement	0.13 [mm]
	Y displacement	8.89 [mm]
	Z displacement	6.88 [mm]
	Total reaction force	69.78 [N]
	Force at X	6.41 [N]
	Force at Y	35.35 [N]
	Force at Z	59.27 [N]
	Deformation	2.71 [%]

Source: Own elaboration

The use of tables to analyze the improvement in the evaluation parameters highlights the relevance of the simulator in this case study.

Box 16

Table 7
Simulation data obtained for the screw in Nylon 6/6 material optimizing the linear meshing arrangement.

Material	Property	Value/Unit
PLA	Von Mises Stress	3772.34 [MPa]
Linear	Principal Stress	1668.72 [MPa]
Meshing	Normal in XX	401.96 [MPa]
	Normal in YY	827.58 [MPa]
	Normal in ZZ	631.60 [MPa]
	To XY cut	1262.75 [MPa]
	To YZ cut	464.33
	To ZX cut	[MPa]
	Total displacement	691.53
	X displacement	[MPa]
	Y displacement	6.93 [mm]
	Z displacement	0.10 [mm]
	Total reaction force	6.71 [mm]
	Force at X	5.19 [mm]
	Force at Y	70.03 [N]
	Force at Z	5.38 [N]
	Deformation	34.52 [N]
		57.64 [N]
		2.07 [%]

Source: Authors.

Each of the materials evaluated offers characteristics that give value to the FEM analysis.

ISSN: 2410-3950
RENIECYT-CONAHCYT: 1702902

González-Sosa, Jesús Vicente, Avila-Soler, Enrique and Zavala-Osorio, Yadira. [2024]. Optimization of the finite element meshing for the fabrication of a M3x12x0.5 screw, in 3D printing. Journal of Experimental Systems. 11[30]-1-14: e51130114.
https://doi.org/10.35429/JOES.2024.11.30.1.14

Box 17

Table 8

Simulation data obtained for the screw in PLA material optimizing the linear arrangement of the mesh

Material	Property	Value/Unit
PLA Linear Meshing	Von Mises Stress	3683.70 [MPa]
	Principal Stress	1705.85 [MPa]
	Normal in XX	584.63 [MPa]
	Normal in YY	812.35 [MPa]
	Normal in ZZ	642.24 [MPa]
	To XY cut	1187.95 [MPa]
	To YZ cut	457.87 [MPa]
	To ZX cut	631.95 [MPa]
	Total displacement	22.65 [mm]
	X displacement	0.33 [mm]
	Y displacement	21.94 [mm]
	Z displacement	16.97 [mm]
	Total reaction force	69.64 [N]
	Force at X	6.93 [N]
	Force at Y	35.81 [N]
	Force at Z	60.18 [N]
	Deformation	6.66 [%]

Source: Own elaboration

Third stage: the variables to be modified and generate another optimization is the minimum size of elements at 10% with linear arrangement in the mesh, the data are shown in table 9 ABS material, table 10 nylon material and table 11 PLA material, each table with the corresponding physical properties.

Box 18

Table 9

Simulation data obtained for the screw in ABS material by optimizing the size of elements in the mesh

Material	Property	Value/Unit
ABS Minimum element size at 10% Linear meshing	Von Mises Stress	3942.40 [MPa]
	Principal Stress	2048.54 [MPa]
	Normal in XX	160.28 [MPa]
	Normal in YY	764.14 [MPa]
	Normal in ZZ	874.34 [MPa]
	To XY cut	1147.46 [MPa]
	To YZ cut	420.68 [MPa]
	To ZX cut	725.29 [MPa]
	Total displacement	9.12 [mm]
	X displacement	0.12 [mm]
	Y displacement	8.79 [mm]
	Z displacement	6.82 [mm]
	Total reaction force	83.49 [N]
	Force at X	7.50 [N]
	Force at Y	26.64 [N]
	Force at Z	49.95 [N]
	Deformation	2.92 [%]

Source: Own elaboration

In this phase, modifications have been taken into account to re-evaluate each of the materials, applying the corresponding loads, in order to identify significant improvements in simulations of this type.

Box 19

Table 10

Simulation data obtained for the screw in Nylon 6/6 material by optimizing the size of elements in the mesh

Material	Property	Value/Unit
Nylon 6/6 Minimum element size at 10% Linear meshing	Von Mises Stress	4040.32 [MPa]
	Principal Stress	2047.39 [MPa]
	Normal in XX	109.25 [MPa]
	Normal in YY	774.61 [MPa]
	Normal in ZZ	865.84 [MPa]
	To XY cut	1176.75 [MPa]
	To YZ cut	425.70 [MPa]
	To ZX cut	737.10 [MPa]
	Total displacement	6.89 [mm]
	X displacement	0.09 [mm]
	Y displacement	6.64 [mm]
	Z displacement	5.14 [mm]
	Total reaction force	82.58 [N]
	Force at X	6.58 [N]
	Force at Y	26.66 [N]
	Force at Z	49.34 [N]
	Deformation	2.25 [%]

Source: Own elaboration

Box 20

Table 11

Data obtained from simulation for the screw in PLA material by optimizing the size of elements in the mesh

Material	Property	Value/Unit
PLA Minimum element size at 10% Linear meshing	Von Mises Stress	3893.97 [MPa]
	Principal Stress	2048.14 [Mpa]
	Normal in XX	188.19 [Mpa]
	Normal in YY	758.15 [Mpa]
	Normal in ZZ	877.32 [Mpa]
	To XY cut	1133.08 [Mpa]
	To YZ cut	418.07 [Mpa]
	To ZX cut	719.44 [Mpa]
	Total displacement	22.52 [mm]
	X displacement	0.29 [mm]
	Y displacement	21.70 [mm]
	Z displacement	16.84 [mm]
	Total reaction force	83.98 [N]
	Force at X	7.98 [N]
	Force at Y	26.64 [N]
	Force at Z	50.32 [N]
	Deformation	7.14 [%]

Source: Own elaboration

ISSN: 2410-3950
RENIECYT-CONAHCYT: 1702902

González-Sosa, Jesús Vicente, Avila-Soler, Enrique and Zavala-Osorio, Yadira. [2024]. Optimization of the finite element meshing for the fabrication of a M3x12x0.5 screw, in 3D printing. Journal of Experimental Systems. 11[30]-1-14: e51130114.
https://doi.org/10.35429/JOES.2024.11.30.1.14

Fourth stage: modify the angle of rotation in the mesh to optimize the test specimens and compare them with the other FEM simulations obtained. For this purpose, the data are shown in tables 12, 13 and 14, with the same materials as in the previous stages.

Box 21

Table 12
Simulation data obtained for 30° angle in ABS material by optimizing the size of elements in the mesh.

Material	Property	Value/Unit
ABS Pivoting angle 30° Linear meshing	Von Mises Stress	2421.23 [MPa]
	Principal Stress	1436.17 [MPa]
	Normal in XX	129.15 [MPa]
	Normal in YY	607.76 [MPa]
	Normal in ZZ	865.55 [MPa]
	To XY cut	1103.79 [MPa]
	To YZ cut	106.02 [MPa]
	To ZX cut	645.79 [MPa]
	Total displacement	9.03 [mm]
	X displacement	0.05 [mm]
	Y displacement	8.83 [mm]
	Z displacement	7.11 [mm]
	Total reaction force	80.56 [N]
	Force at X	3.85 [N]
	Force at Y	24.47 [N]
	Force at Z	56.96 [N]
	Deformation	1.88 [%]

Source: Own elaboration

As a result of adjusting the parameters in the case study simulations, new opportunities in the field of applied engineering are investigated and evaluated, using screws as a case of analysis.

Box 22

Table 13
Simulation data obtained for 30° angle in Nylon material by optimizing the size of elements in the mesh.

Material	Property	Value/Unit
Nylon 6/6 Pivoting angle 30° Linear meshing	Von Mises Stress	2458.48 [MPa]
	Principal Stress	1429.20 [MPa]
	Normal in XX	107.61 [MPa]
	Normal in YY	564.98 [MPa]
	Normal in ZZ	864.68 [MPa]
	To XY cut	1122.10 [MPa]
	To YZ cut	410.53 [MPa]
	To ZX cut	651.34 [MPa]
	Total displacement	6.78 [mm]
	X displacement	0.04 [mm]
	Y displacement	6.63 [mm]
	Z displacement	5.34 [mm]
	Total reaction force	80.62 [N]
	Force at X	3.20 [N]
	Force at Y	24.43 [N]
	Force at Z	55.72 [N]
	Deformation	1.43 [%]

Source: Own elaboration

Some data may be synthesized to facilitate a statistical analysis in the results section, which provides additional support for enriching studies using simulators such as the one used in this work, which is FEM.

Box 23

Table 14
Simulation data obtained for 30° angle in PLA material by optimizing the size of elements in the mesh.

Material	Property	Value/Unit
PLA Pivoting angle 30° Linear meshing	Von Mises Stress	2402.74 [MPa]
	Principal Stress	1440.10 [MPa]
	Normal in XX	146.51 [MPa]
	Normal in YY	629.53 [MPa]
	Normal in ZZ	867.14 [MPa]
	To XY cut	1094.74 [MPa]
	To YZ cut	403.51 [MPa]
	To ZX cut	642.91 [MPa]
	Total displacement	22.36 [mm]
	X displacement	0.13 [mm]
	Y displacement	21.86 [mm]
	Z displacement	17.60 [mm]
	Total reaction force	80.53 [N]
	Force at X	4.19 [N]
	Force at Y	24.89 [N]
	Force at Z	57.62 [N]
	Deformation	4.64 [%]

Source: Own elaboration

The next section will employ a statistical tool using graphs to support screw production optimization in collaboration with the FEM.

Results

The scatter plots corresponding to some of the data obtained from the tables showing the results of the FEM simulations are presented. Table 15 contains the data to be used to graphically represent the results when performing the simulation on the screw using FEM.

Box 24

Table 15
Representative values for the graphs

Material	EVM [MPa]	DZ [mm]	FR [N]	DF [%]
ABS	16496.1	10.86	31.19	13.28
	3713.3	9.17	69.78	2.71
	3942.4	9.12	83.49	2.92
	2421.2	9.03	80.56	1.88
Nylon 6/6	14931.0	16.41	33.54	8.63
	3772.3	6.93	70.03	2.07
	4040.3	6.89	82.58	2.25
	2458.5	6.78	80.62	1.43
PLA	16402.9	26.92	31.23	32.83
	3683.7	22.65	69.64	6.66
	3894.0	22.52	83.98	7.14
	2402.7	22.36	80.53	4.64

Source: Own elaboration

ISSN: 2410-3950
RENIECYT-CONAHCYT: 1702902

González-Sosa, Jesús Vicente, Avila-Soler, Enrique and Zavala-Osorio, Yadira. [2024]. Optimization of the finite element meshing for the fabrication of a M3x12x0.5 screw, in 3D printing. Journal of Experimental Systems. 11[30]-1-14: e51130114. https://doi.org/10.35429/JOES.2024.11.30.1.14

The following Figure 10 shows the dispersion of the measurements for each material, as well as the variations in certain parameters.

Box 25

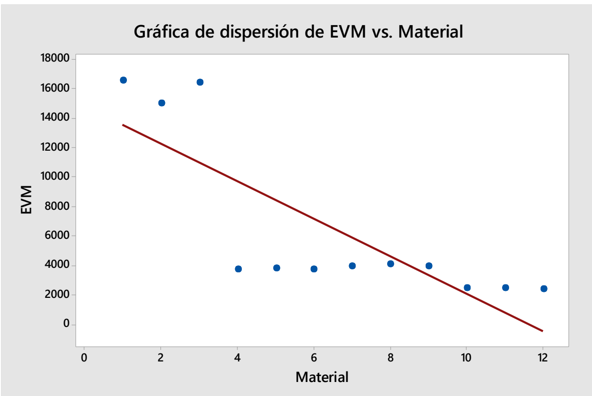

Figure 11
Scatter plot for the variable Von Mises Stress (VMS)

Source: Own elaboration

Figure 11
Source: Own elaboration

Figure 10 shows that specimens 8, 9 and 10 received the highest acceptance in the simulation, as indicated by the dispersion fit.

Box 26

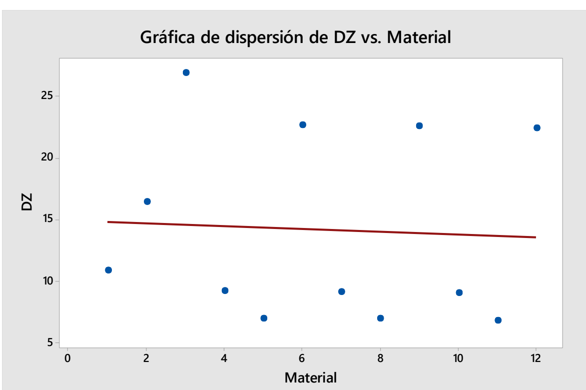

Figure 12
Scatter plot for the variable Displacement (DZ)

Source: Own elaboration

In Figure 11, specimen number 2 also shows the best behavior, since it presents the smallest displacement.

Figure 12 shows the behavior of the material in relation to the reaction force at the time of the simulation.

Box 27

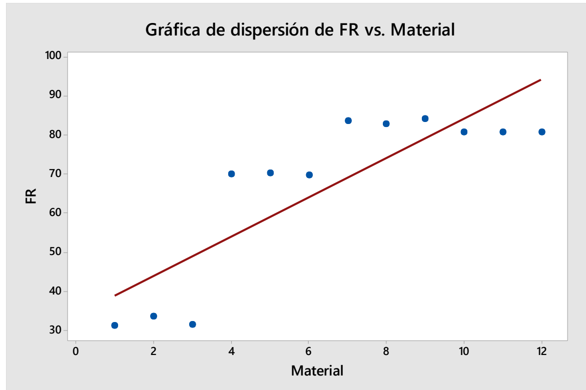

Figure 13
Scatter plot for the variable Reaction Effort (RF)

Source: Own elaboration

The specimens showing the best performance in relation to reaction force during the simulation with FEM are numbers 6, 9 and 10, as observed in the scatter in Figure 13.

Finally, the deformation variable experienced by the bolt when applying a torque of 1600 Nmm is considered.

Box 28

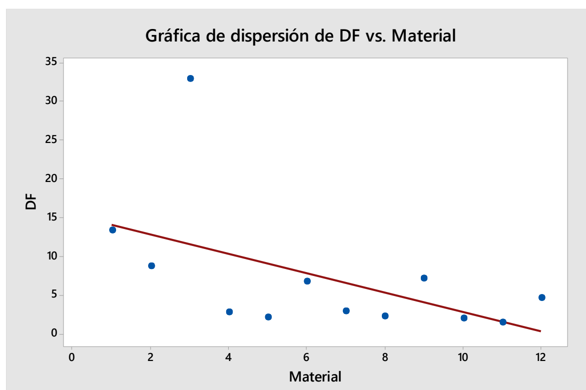

Figure 14
Scatter plot for the variable deformation
Source: Own elaboration

From Figure 14 it can be observed that there is a greater number of specimens that satisfactorily comply with an acceptable percentage in the dispersion for a second evaluation, when related to other variables. In this sense, specimens 1, 6, 10 and 11 meet the objective established for dispersion.

ISSN: 2410-3950
RENIECYT-CONAHCYT: 1702902

From a statistical perspective, the specimens that meet the requirements are presented in Table 16. This table also includes the satisfactory results obtained from the simulation with the finite element method (FEM). This information allows us to identify the best option for the evaluation of the screw.

Box 29

Table 16
Optimal simulation and statistical data

		EVM	DZ	FR	DF
1	FEM				
	Estadístico				👍
2	FEM				
	Estadístico		👍		
3	FEM				
	Estadístico				
4	FEM				
	Estadístico				
5	FEM				
	Estadístico				
6	FEM				
	Estadístico			👍	👍
7	FEM			👍	
	Estadístico				
8	FEM			👍	
	Estadístico	👍			
9	FEM			👍	
	Estadístico	👍		👍	
10	FEM	👍	👍		👍
	Estadístico	👍		👍	👍
11	FEM	👍	👍		👍
	Estadístico				👍
12	FEM	👍	👍		👍
	Estadístico				

Source: Own elaboration

According to what is shown in Table 16, the specimens that present the best results in both evaluations, both in the FEM and statistical analysis, are numbers 10 and 11, which correspond to the ABS and Nylon 6/6 material, with a twist angle of 30° and arranged in a linear fashion.

The use of the FEM tool on a screw has the advantage of optimizing and accelerating the evaluation process in order to improve the manufacturing conditions of the screw. In addition, it allows to obtain comparative values in the simulation that are validated, to a certain extent, by statistical tools.

Discussion

In the discussion of recent studies, the growing importance of advanced methodologies in the simulation and fabrication of materials is emphasized. Tian et al. (2024) highlight that generating high-fidelity models is fundamental for achieving precise finite element simulations in composite materials. Their innovative meshing methodology, which utilizes micro-computed tomography (μCT) and neural networks, represents a significant advancement in automation and accuracy in modeling, potentially reducing preparation times and increasing reproducibility of models. This approach not only enhances mesh quality but also expands the possibilities for analyzing complex structures.

On the other hand, Jemal et al. (2024) investigate perforated metals and their optimization in bending processes, addressing the impact of variables such as bend angle and thickness on material behavior. Their findings suggest that perforation geometry has a considerable effect on elastic recovery, highlighting the importance of considering these factors in the design of metal components. This research contributes not only to understanding the behavior of aluminum alloys but also offers a framework for future optimizations in manufacturing processes.

In the healthcare sector, Om et al. (2024) underscore the use of Additive Manufacturing techniques for creating orthopedic braces. Their focus on determining optimal dimensions demonstrates how the integration of simulations and data analysis can enhance the functionality and comfort of medical devices. This study is a clear example of how modern engineering can adapt to the specific needs of patients, paving the way for personalized solutions in medical treatment.

Su et al. (2024) present an innovative approach using the adaptive scaled boundary finite element method (SBFE) to optimize structures that must withstand dynamic loads. The use of advanced adaptive meshing techniques and their ability to handle variable volumes reinforce the need to integrate dynamic simulation in structural design. This approach promises improved efficiency in optimization processes, allowing for more robust and adaptable designs under changing conditions.

ISSN: 2410-3950
RENIECYT-CONAHCYT: 1702902

Finally, Ho et al. (2024) explore the combination of topology optimization and additive manufacturing in creating continuous carbon fiber composite structures. The significant increases in stiffness achieved with different resins testify to how research in materials and manufacturing techniques can lead to drastic improvements in structural performance. The analysis of microstructures also provides crucial insights into material behavior, which can inform future design iterations.

Collectively, these studies reflect a trend toward the integration of advanced techniques in design and manufacturing, underscoring the importance of innovation in engineering to address contemporary challenges across various industries. The synergy between precise simulations, process optimization, and customization in manufacturing promises to revolutionize how materials and components are designed and produced in the future.

Conclusions

The manufacture of an M3x12x0.3 screw by 3D printing represents an innovative alternative in the design and production of mechanical components. This analysis focuses on the evaluation of bolt performance under specific loads using FEM. Through computer simulations, it is possible to determine how the material and dimensions of the screw affect its mechanical behavior, thus providing a deeper understanding of its functionality and limitations. The integration of scatter plots to visualize Von Mises stress, displacement, strain and reaction stress allows discerning relevant patterns and trends in the structural analysis of the bolt.

In the Von Mises stress evaluation, the results obtained illustrate how the bolt behaves under applied loading conditions. The scatter plots show that, even if the bolt is designed to withstand certain applied loads, there is a critical threshold beyond which the material begins to yield. This type of analysis is crucial as it allows to accurately identify the safe operating range of the screw, avoiding potential failures that could compromise the integrity of the systems in which it is integrated. Interpretation of these graphs helps to understand the importance of careful design with respect to the direction the head should take for screw production.

Displacement and deformation are other key parameters analyzed in this research. By examining the results obtained, it is evident that the screw presents an elastic behavior within a specific range of loads, which suggests that 3D printing of the screw allows a structure with certain recovery capabilities under transient loads. However, it was observed that the displacement increases significantly with increases in load, indicating a possible limitation in applications where extreme stiffness is required. Scatter plots clearly reflect this relationship, facilitating design adjustments to improve performance.

On the other hand, the reaction stress is a fundamental aspect in the evaluation of the load capacity of the bolt. Through simulation, it was possible to identify the critical areas where the greatest stresses are concentrated, which points to potential points of failure. This phenomenon not only highlights the importance of structural analysis in screws manufactured by 3D printing, but also suggests the possibility of optimizing the design by modifying the geometry or choosing more appropriate materials. The visualization of this information becomes a valuable tool for engineers seeking to innovate in the use of bolts in various applications.

Finally, the analysis and evaluation of the M3x12x0.3 screw manufactured with 3D printing highlights the relevance of employing tools such as FEM and scatter plots to make informed mechanical design decisions. This study not only provides an in-depth understanding of the mechanical behavior of the screw, but also highlights the potential of 3D printing in the manufacture of high-precision components. As we continue to advance 3D printing technology, new opportunities will open up to optimize designs, improve materials and, consequently, raise engineering quality and performance standards. The conclusion of this analysis serves as a starting point for future studies that delve deeper into the interplay between materials, design and 3D printing in the creation of advanced mechanical components.

Annexes

Tables and adequate sources.

ISSN: 2410-3950
RENIECYT-CONAHCYT: 1702902

Declarations

Conflict of interest

The authors declare no interest conflict. They have no known competing financial interests or personal relationships that could have appeared to influence the article reported in this article.

Author contribution

González-Sosa, Jesús Vicente: Contributed to the project idea, research method and technique.

Avila-Soler, Enrique: Contributed to the project idea, research method and technique.

Zavala-Osorio, Yadira: Contributed to the project idea, research method and technique.

Availability of data and materials

The data obtained was derived from modeling and measurement data with metrological instruments.

Funding

This article did not receive financial support from any institution.

Acknowledgements

Nothing.

Abbreviations

FEM	Finite Element Method
MAI3D I3D	manufacturing with 3D printing 3D printing
ABS	Butadiene Styrene
PLA	Polylactic Acid
MF	Flexural Modulus
DF	Deformation
EF	Stress

References

Antecedents

Çetin, A., & Bircan, D. A. [2021]. 3D pull-out finite element simulation of the pedicle screw-trabecular bone interface at strain rates. Proceedings of the Institution of Mechanical Engineers, Part H: Journal of Engineering in Medicine, 236(1), 134–144.

Chandra, G., & Pandey, A. [2021]. Design and analysis of biodegradable buttress threaded screws for fracture fixation in orthopedics: a finite element analysis. Biomedical Physics & Engineering Express, 7(4), 045010.

Hsia, S.-Y., Chou, Y.-T., & Lu, G.-F. [2016]. Analysis of Sheet Metal Tapping Screw Fabrication Using a Finite Element Method. Applied Sciences, 6(10), 300.

Figueroa Díaz, R. A., Balvantín García, A. de J., Diosdado de la Peña, J. Á., Cruz Alcantar, P., Murillo Verduzco, I., & Pérez Olivas, P. A. [2019]. Stress analysis in a screw conveyor axis under a specific fault condition. Ingeniería Investigación y Tecnología, 20(3), 1–11.

Lee, S. H., Hong, M. H., & Lee, K. B. [2017). Finite Element Analysis of Screw-Tightening Torque Applied to Custom and Conventional Abutment. Global Journal of Health Science, 9(9), 165.

Sheng, W., Ji, A., Fang, R., He, G., & Chen, C. [2019). Finite Element- and Design of Experiment-Derived Optimization of Screw Configurations and a Locking Plate for Internal Fixation System. Computational and Mathematical Methods in Medicine, 2019, 1–15.

Xue, H., Zhang, Z., Liu, M., Lin, Z., Endo, Y., Liu, G., Mi, B., Zhou, W., & Liu, G. [2022]. Finite element analysis of different fixation methods of screws on absorbable plate for rib fractures. Frontiers in Bioengineering and Biotechnology, 10.

Zhu, Y., Babazadeh-Naseri, A., Dunbar, N. J., Brake, M. R. W., Zandiyeh, P., Li, G., Leardini, A., Spazzoli, B., & Fregly, B. J. [2023]. Finite element analysis of screw fixation durability under multiple boundary and loading conditions for a custom pelvic implant. Medical Engineering & Physics, 111, 103930.

Discussions

Ho, T. N. T., Nguyen, S. H., Le, V. T., & Hoang, T. D. [2024]. Coupling design and fabrication of continuous carbon fiber-reinforced composite structures using two-material topology optimization and additive manufacturing. The International Journal of Advanced Manufacturing Technology, 130(9), 4277-4293.

Jemal, A., Salau, A. O., & Wondimu, A. [2024]. Finite element method–based multi-objective optimization of press-brake bending of sheet metal. The International Journal of Advanced Manufacturing Technology, 130(9), 4263-4275.

Om, M. S., Yang, W. C., Choe, C. M., Kim, U. H., Ri, W. S., & Sok, S. H. [2024]. A reasonable approach to determine structural dimension of FDM fabricated orthopedic orthosis using finite element simulation and simple additive weighting method. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 09544062241282496

Su, R., Zhang, X., Tangaramvong, S., & Song, C. [2024]. Adaptive scaled boundary finite element method for two/three-dimensional structural topology optimization based on dynamic responses. Computer Methods in Applied Mechanics and Engineering, 425, 116966.

Tian, X., Zhang, H., Qu, Z., & Ai, S. [2024]. An efficient finite element mesh generation methodology based on μCT images of multi layer woven composites. Composites Part A: Applied Science and Manufacturing, 184, 108255.