INNOVATIVE MECHANICAL ENGINEERING **ISSN 2812-9229 (Online)**
University of Niš, Faculty of Mechanical Engineering
VOL. 3, NO 1, 2024, PP. 58 - 68

Original scientific paper *

LATTICE OPTIMIZATION OF ADDITIVELY MANUFACTURED PARTS: A CASE STUDY

Nikola Korunović[1], Jovan Aranđelović[1], Rajko Turudija[1]

[1]University of Niš, Faculty of Mechanical Engineering

Abstract. *Additive manufacturing (AM) provides increased design flexibility compared to traditional manufacturing methods. However, this advantage is frequently diminished by the reduced mechanical performance. This is especially true for parts produced by the most widespread AM technology, fused filament fabrication (FFF). The internal structure of an FFF-produced part (the infill) is typically defined in a slicer, which is the software used to prepare G-codes for FFF machines. This process relies on predefined patterns without considering the mechanical performance of the part. However, various methods can be used to optimize the mechanical performance of an FFF printed part, such as topology optimization (TO) or generative design. We present a case study in which a subtype of TO, lattice optimization, was used to optimize the infill of a wall bracket. Thereby, the optimization process was based on the finite element method (FEM). In this way, it was possible to improve the mechanical properties of the bracket while keeping its mass at the same level, which confirms the advantage of lattice-optimization-based infill generation over standard infill generation in slicers.*

Key words: *Additive Manufacturing (AM), Fused Filament Fabrication (FFF), Lattice Optimization, Finite Element Method (FEM), Finite Element Analysis (FEA)*

1. INTRODUCTION

Fused filament fabrication (FFF) is the most used additive manufacturing (AM) technology in home, office, industrial, and research settings [1]. AM offers many advantages over traditional manufacturing methods, including increased part complexity, instant assembly generation, part consolidation, mass customization, design freedom, lightweighting, and on-demand manufacturing [2]. However, parts produced using AM, especially FFF, often have poor mechanical performance [3]. This can be attributed to various factors, such as the mechanical properties of filaments and inadequate print settings. When it comes to the influence of print settings on the mechanical properties of

*Received: March 29, 2024 / Accepted May 13, 2024.**
Corresponding author: Nikola Korunović
Faculty of Mechanical Engineering, University of Niš
E-mail: nikola.korunovic@masfak.ni.ac.rs

printed parts, most research focuses on manufacturing parameters like layer height, nozzle speed, and printed line direction.

Typically, 3D geometrical models of parts printed by FFF are created in CAD software, translated into polygonal models, and saved in the STL format. These polygonal models are then imported into a slicer program, which generates the G-code based on the part geometry and selected print settings. The slicer creates infill structures using libraries of pattern shapes without considering the mechanical performance of the parts. As a result, users of slicer software typically have to define infill properties using an intuitive approach. While the slicer may suggest a pattern type based on the application of the printed parts (e.g. line patterns for quick prints or cubic patterns for functional parts), there are currently no slicer software options that offer reliable optimization of infill structures based on the desired mechanical performance of the parts.

The mechanical performance of an AM-produced part may be checked through structural analysis, which mostly relies on finite element analysis (FEA). However, FEA is just a tool that may be used in a trial-and-error approach to design optimization. To achieve the desired goal directly, FEA needs to be combined with other methods, such as generative design or topology optimization (TO) [5], which refines existing designs. TO is considered the most general type of structural optimization as it affects both the dimensions and shape of the structure [6]. It aims to achieve optimum material distribution for a given design while adhering to specific constraints such as functionality, weight, and structural integrity [7]. TO focuses on removing the least loaded sections of the structure to reduce its mass. When TO is based on a finite element (FE) model, these sections represent individual finite elements.

Lattice optimization is a subtype of TO that defines the optimal shape of internal lattice structures, i.e., the optimal lattice density distribution (Fig. 1). It can be effectively applied to optimize the internal structures of AM parts. By doing so, it achieves material savings while preserving the necessary mechanical properties of the component.

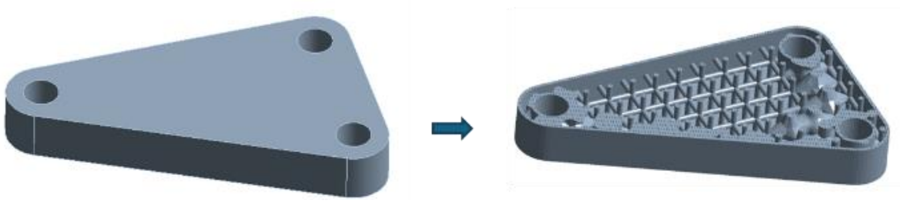

Fig. 1 Lattice optimization

Several studies have utilized this approach to determine the density of lattice structures within parts. Pan et al. conducted a comprehensive research review of lattice structures, covering various design procedures, lattice structure performance, optimization, and production. According to Pan et al., lattice structures can be categorized into two groups: uniform lattice structures (porous structure patterns of the same size and distribution) and non-uniform lattice structures (different topological shapes or geometric sizes). The review suggests that non-uniform lattice structures outperform uniform ones, as they offer more customization options [8]. Birosz and Andó presented a method for optimizing infill size based on 2D FEA (the FEA of a cross-section of a 3D printed part). In this approach, the unit cell size is scaled to reduce the maximum stress in the structure, allowing for different cell sizes. The results obtained using scaled unit cells were compared to those obtained using equally sized unit cells throughout the whole model. All models with scaled unit cells showed a significant improvement by having a lower maximum stress than the original one [9]. Gopsill et al. developed a five-stage methodology for infill optimization: model creation, identification of cross-section perimeter and mesh size, classical infill design, FEA-influenced infill design, and G-code generation. This methodology was tested on beams and brackets, showing a 3.5-fold increase in loading capacity and improved mechanical test results consistency [10]. Yu et al. developed a 2D FEA-based methodology using different stress constraints for failure criteria in lattice infill optimization: von Mises stress for the solid outer shell and Tsai-Hill stress for the porous lattice infill structure [11]. Wu et al. created a novel methodology for generating infill for additively manufactured parts inspired by the porous structures of trabecular bone. Parts containing bone-like porous infill structures showed a 27.5% decrease in maximal stress compared to parts with traditional infill structures [12]. There seems to be a lack of research comparing the mechanical behavior of parts containing lattice-optimized infill structures to those containing standard infill structures, using 3D FE models.

This paper presents a case study that uses FEA to compare the mechanical behavior of a wall bracket with a solid internal structure (100 percent infill) to the mechanical behavior of brackets with two different infill structures: one using a classic infill structure (uniform pattern type typically used in slicers [8]) and the other using lattice optimization. The non-solid infill parts were designed to have nearly the same mass. The results showed that both infill creation methods significantly reduced part mass compared to full part creation while maintaining acceptable values of the safety factor. Additionally, the study confirmed that the lattice-optimization-based method is superior to the slicer-based method, resulting in a design with greater stiffness and a larger safety factor.

2. Finite element model and analysis of the part with solid internal structure

The initial stage of the study involved performing FEA on an L-bracket with a solid internal structure. This was done to establish a reference for comparing results and to set a baseline for lattice optimization.

A CAD model of an L-bracket was created in SolidWorks and translated into ANSYS (Fig. 2). An isotropic linear model was used to model its material properties, which matched the typical properties of FFF produced PLA, as shown in Table 1 [13].

Table 1 FFF printed PLA material properties

Property	Values
Density	1.24 g/cm^3
Young’s Modulus	2388.8 MPa
Poisson's Ratio	0.35
Tensile Yield Strenght	30.77 MPa

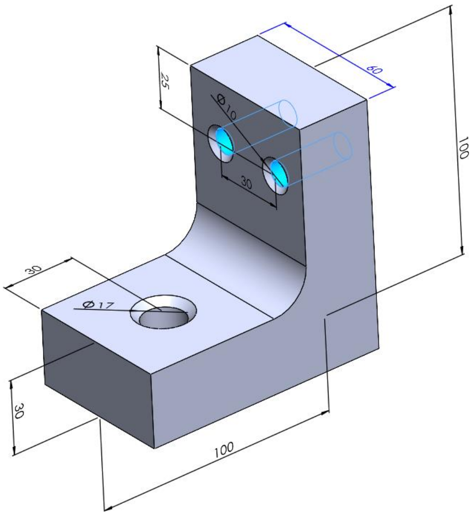

Fig. 2 CAD model of L-bracket with dimensions (mm)

Based on the CAD model, an FE model was created. It was meshed with 77,412 quadratic tetrahedron elements containing 121,779 nodes. Frictionless supports were applied to the back surface of the bracket, as well as to the chamfers of the two holes used for fixing the bracket. A load of 250 N was applied to the chamfer surfaces of the central hole, acting in the vertical direction (Fig. 3).

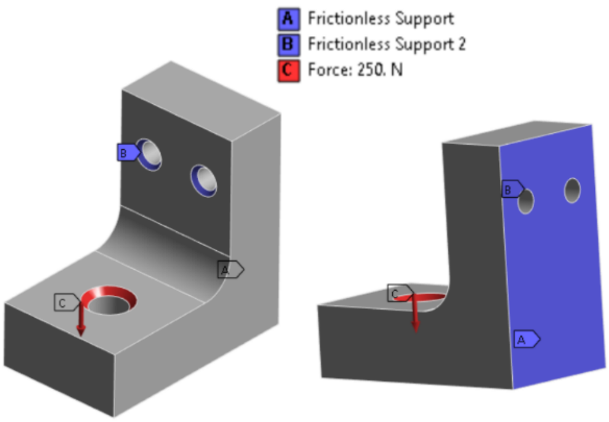

Fig. 3 Loads and boundary conditions imposed on the FE model of L-bracket

Von Mises stress field and displacement field obtained by static structural analysis of the defined model are shown in Fig. 4. The achieved safety factor of 8.85 suggests that the part was overdesigned, meaning it did not require a solid design to fulfill its intended purpose.

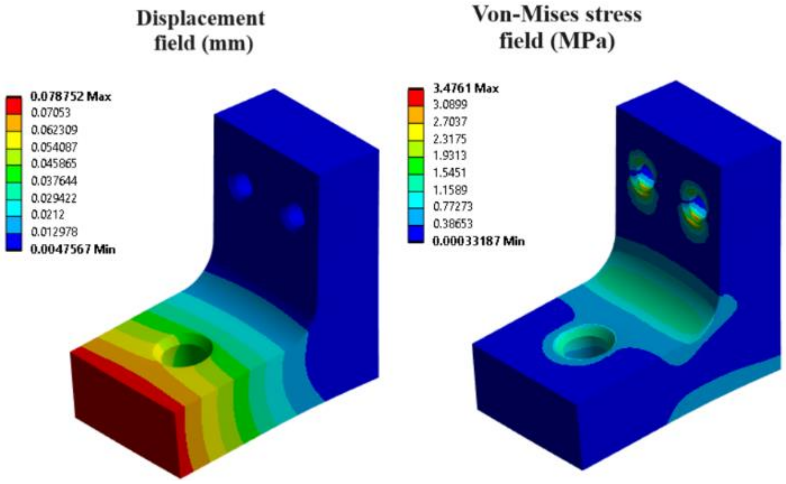

Fig. 4 Result of FEA of L-bracket with solid internal structure

3. Lattice optimization-based model and analysis

The process of lattice optimization using ANSYS [14] and SpaceClaim is illustrated in Fig. 5.

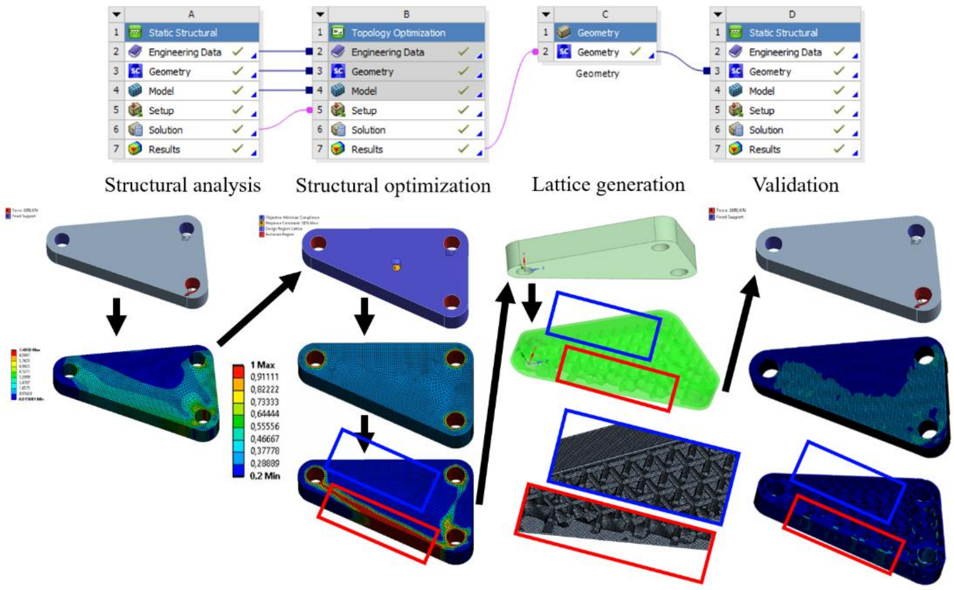

Fig. 5 Lattice optimization workflow.

After conducting structural analysis, the "lattice density" is computed in ANSYS across the entire optimization area. This data is then imported into SpaceClaim, where the lattice geometry is generated based on the density results. A lattice density of 1 represents solid infill, while smaller density values result in less dense lattice structures. Users can define minimum and maximum values of lattice density to exert control over the lattice creation process. The finalized geometry is then sent back to ANSYS for validation, involving another round of structural analysis to assess the stress state of the optimized lattice structure. In the example shown in Fig. 5, the minimum value of lattice density was set to 0.2 and the maximum value was set to 1. As a result, sections with a calculated lattice density of 1 (red) obtained solid infill, while sections with a calculated lattice density of 0.2 (blue) obtained the rarest lattice structure.

Based on the FE model of the L-bracket with solid internal structure, a structural optimization model was set up in ANSYS. The entire body, except for the surfaces where the boundary conditions were applied in the previous FEA, was designated as the optimization region. The regular cubic shape (Fig. 6) was selected as the pattern for infill creation. To control the properties of the lattice structure, a minimum density of 0.2 and a maximum density of 1 were set. The minimum value was selected to prevent the generation of excessively thin lattices, while the maximum value was chosen to allow for some parts of the model interior to be generated as solid.

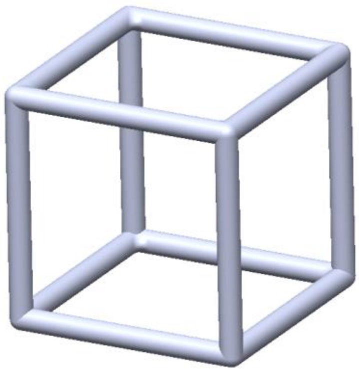

Fig. 6 Regular cube lattice shape

Due to the 30 mm thickness of the L-bracket profile, a lattice cell size of 15 mm was chosen to create two rows of lattice cells across the thickness. Two optimization goals were established, with equal importance: reducing compliance (increasing stiffness) and minimizing stress. Additionally, a constraint was implemented to maintain 40% of the original mass of the solid bracket. The initial outcome of the lattice optimization process produced a lattice density distribution depicted in Fig.7.

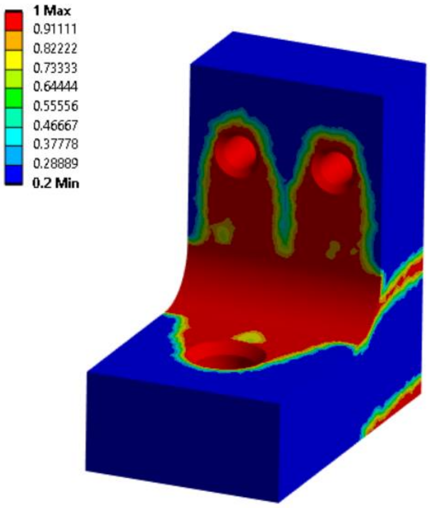

Fig. 7 The first result of structural optimization – lattice density distribution

According to the lattice density results, SpaceClaim was utilized to design the lattice structures within the original solid model of the bracket. In FFF printing, it is generally recommended that part walls should be comprised of two to four printed lines. Here the middle value, equal to three lines, was chosen. As a result, a 1.2 mm thick shell feature was implemented to create the model's outer shell, ensuring it could be printed on an FFF printer with a 0.4 mm nozzle. This approach yielded a CAD model with an optimized infill structure, as depicted in Fig. 8.

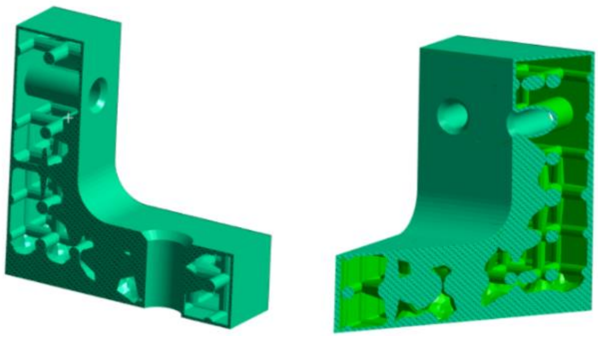

Fig. 8 Cross-sections of the optimized bracket with lattice infill

To validate the mechanical behavior of the optimized bracket with lattice infill, a structural analysis was conducted. Due to the optimized model consisting of facets, it was necessary to define the loads and boundary conditions using sets of finite element facets rather than the originally used faces of the CAD model. The finite element mesh was created using the patch-independent method, which involves creating a volume mesh of tetrahedrons first and then projecting it onto the model faces. The minimum element size was set to 1 mm and the maximum element size to 2 mm. This resulted in a mesh containing 1,309,212 finite elements and 1,985,483 nodes. Displacements were fixed on specific element facets, as frictionless supports could not be assigned to the facets of finite elements. For the results to be comparable, the displacement constraint was set to prevent the movement in the direction normal to the selected facets. Additionally, a force identical to the one used in the FEA of the solid part was applied to the corresponding FE facets (see Fig. 9 for details.)

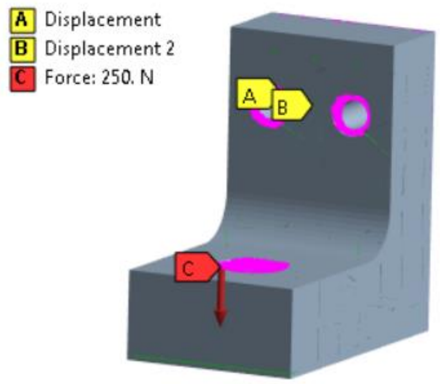

Fig. 9 Optimized L-bracket boundary conditions

Based on the defined FE model, a static structural analysis was performed to assess the mechanical behavior of the optimized part. The Von Mises stress field and displacement field obtained during this process are illustrated in Fig. 10. In this instance, the safety factor reached a minimum value of 3.88.

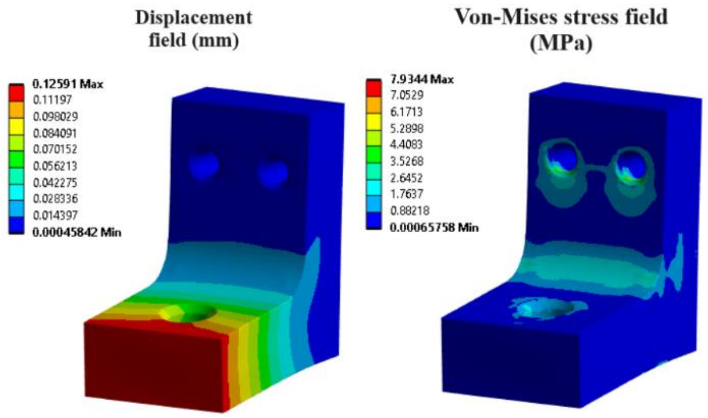

Fig. 10 Result of FEA of optimized L-bracket

4. SLICER-BASED MODEL AND ANALYSIS

The term "classical infill" refers to a repeating geometric pattern made up of uniform lattices with equal-sized unit cells, as defined by Pan et al. [8]. This type of infill can take on various shapes such as grid, honeycomb, and cubic. Currently, it is not feasible to directly extract a CAD model from a slicer that accurately represents the geometry of printed parts and is compatible with FEA. While PrusaSlicer has taken a step in this direction by enabling the export of an FFF printer's toolpath as a complex 3D polygonal model, these models are still not suitable for FEA.

To obtain the third case for comparison of mechanical properties, a solid CAD model of L-bracket with a "classical" infill was created in Space Claim to replicate the infill that would be created in a slicer. The infill density was set to 40%, resulting in a model that had the same mass as the one with the optimized lattice infill. The cross-section of this model is shown in Fig. 11. Based on the CAD model, an FE model was created with the same mesh settings, loads, and boundary conditions as previously described. The results of the corresponding FEA can be seen in Fig. 12. The minimum value of the safety factor obtained for this model was 2.

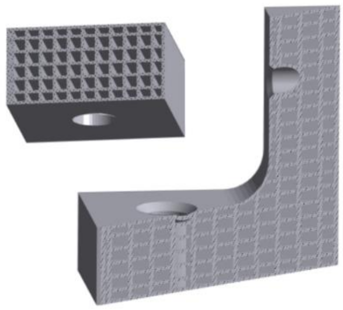

Fig. 11 Cross-sections of the bracket with classical infill

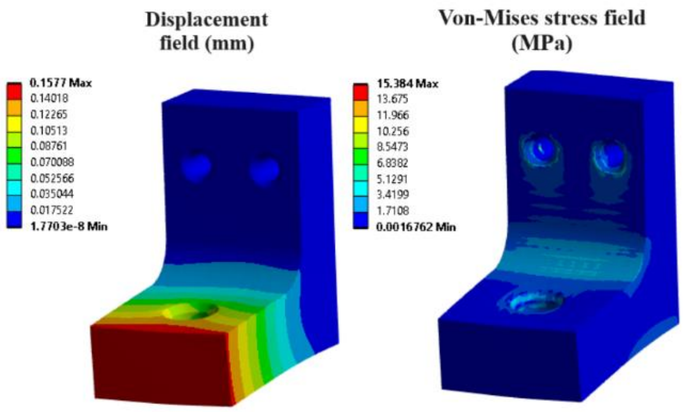

Fig. 12 Result of FEA of L-bracket with classical infill

5. Comparison of Results and Discussion

The results of FEA of the 3 different bracket structures are presented in Table 2.

Table 2 Results of FEA

Type	Mass	Min. safety factor	Max. stress	Max. displacement
Solid model	368 g	8.85	3.48 MPa	0.078 mm
Optimized lattice infill	176 g	3.88	7.93 MPa	0.126 mm
Classic infill	178 g	2	15.38 MPa	0.158 mm

According to the FEA results, the bracket with a classical infill of 40 % density had the lowest safety factor. Despite being similar in mass to the bracket with classical infill, the bracket with optimized infill had a safety factor that was 1.94 times higher. The bracket with solid infill had the highest safety factor and was the stiffest, but it also had a mass 2.06 times larger than the other two brackets. This would require twice as much material and time to produce, making it impractical for production using FFF technology. The production times needed to 3D print each of the bracket types were determined using Ultimaker Cura slicer software and are presented in Table 3. The Creality Ender-3 V2 was set as the target 3D printer, with a default 0.2 mm profile and 100% infill. It is interesting to note that the bracket with a classical infill structure can be printed faster than the bracket with an optimized infill with a time difference of almost 4 hours.

Table 3 Bracket production time

Type	Production time
Solid model	45 h 9 min
Optimized lattice infill	27 h 34 min
Classic infill	23 h 44 min

6. Conclusion

Based on the findings of the study, it is evident that lattice optimization represents a highly effective method for designing FFF-produced parts. It can be used to retain the desired mechanical performance of parts while reducing their mass. At the same time, an optimized design decreases printing time and production costs. However, a notable limitation of the methodology used in this study is the lack of a bidirectional link between FFF slicer software and FEA software, requiring significant time and effort for optimization compared to traditional methods.

Future research may include conducting mechanical tests on the brackets and comparing the results with FEA outcomes to examine the impact of additive manufacturing process parameters on the mechanical behavior of lattice-optimized parts.

Acknowledgement: *This research was financially supported by the Ministry of Science, Technological Development and Innovation of the Republic of Serbia (Contract No. 451-03-65/2024-03).*

REFERENCES

1. https://www.sculpteo.com/en/ebooks/state-of-3d-printing-report-2022/, "The State of 3D Printing Report: 2022 by Sculpteo." (last access: 05.02.2024)
2. Redwood, B., Schöffer, F. Garret, B., 2017, The 3D Printing Handbook, 3D Hubs, 304 p.
3. Caminero, M.Á., Chacón, J.M., García-Plaza, E., Núñez, P.J., Reverte, J.M., Becar, J.P., 2019, Additive Manufacturing of PLA-Based Composites Using Fused Filament Fabrication: Effect of Graphene Nanoplatelet Reinforcement on Mechanical Properties, Dimensional Accuracy and Texture, Polymers, vol. 11 no. 5, 799 p.
4. Ntintakis, I., Stavroulakis, G.E., 2020, Progress and recent trends in generative design, MATEC Web of Conferences, 318 p.
5. Bendsøe, M.P., Sigmund, O., 2003, Topology Optimization - Theory, Methods, and Applications, Springer Verlag
6. Christensen, P.W., Klarbring, A., 2009, An introduction to structural optimization, Springer, 211 p.
7. Zhang, W., Zhu, J., Gao, T., 2016, Topology Optimization in Engineering Structure Design, Elsevier, 1–294 p.
8. Pan, C., Han, Y., Lu, J., 2020, Design and optimization of lattice structures: A review, Applied Sciences, vol. 10 no. 18, 1–36 p.
9. Birosz, M.T., Andó, M., 2023, Simplified local infill size optimization for FDM printed PLA parts, Scientific Reports, vol. 13 no. 1, 1–8 p.
10. Gopsill, J.A., Shindler, J., Hicks, B.J., 2018, Using finite element analysis to influence the infill design of fused deposition modelled parts, Progress in Additive Manufacturing, vol. 3 no. 3, 145–163 p.
11. Yu, H., Huang, J., Zou, B., Shao, W., Liu, J., 2020, Stress-constrained shell-lattice infill structural optimisation for additive manufacturing, Virtual Phys Prototyp, vol. 15 no. 1, 35–48 p.
12. Wu, J., Aage, N., Westermann, R., Sigmund, O., 2018, Infill Optimization for Additive Manufacturing-Approaching Bone-Like Porous Structures, IEEE Trans Vis Comput Graph, vol. 24 no. 2, 1127–1140 p.
13. Pastor-Artigues, M.M., Roure-Fernández, F., Ayneto-Gubert, X., Bonada-Bo, J., Pérez-Guindal, E., Buj-Corral, I., 2020, Elastic asymmetry of PLA material in FDM-printed parts: Considerations concerning experimental characterisation for use in numerical simulations, Materials, vol. 13 no. 1
14. Ansys, I., 2023, Reference Guide - Release 2023R1. ANSYS Inc, Canonsburg