BULETINUL INSTITUTULUI POLITEHNIC DIN IAŞI
Publicat de
Universitatea Tehnică „Gheorghe Asachi” din Iaşi
Volumul 70 (74), Numărul 4, 2024
Secţia
CONSTRUCŢII DE MAŞINI
DOI:10.2478/bipcm-2024-0020

BRIEF OVERVIEW OVER FUSED FILAMENT FABRICATION INFILL PARAMETERS

BY

ALEXANDRU-IONUȚ IRIMIA*, VASILE ERMOLAI, RĂZVAN STAVARACHE and GHEORGHE NAGÎȚ

“Gheorghe Asachi” Technical University of Iași,
Faculty of Machine Manufacturing and Industrial Management

Received: January 28, 2025
Accepted for publication: February 16, 2025

Abstract. Fused Filament Fabrication (FFF) is an additive process manufacturing based on the deposition of thermoplastic material on a heated plate to produce a three-dimensional part. FFF manufacturing technology creates parts with complex geometries by overlapping material layers. FFF technology is used in various fields to produce physical mock-ups and functional parts. This article focuses on the infill process parameters available in the UltiMaker Cura software. Based on the Ishikawa method, seven parameters directly related to infill were identified. These parameters present other subcategories of parameters, resulting in a total of 41 parameters which can be found in the following slicer’s parameter sections: quality, infill, material, speed, special models, supports, and experimental. These process parameters influence many characteristics of the resulting parts, such as the aesthetic, material consumption, which further influences the production costs and the manufacturing time.

Keywords: Additive manufacturing, 3D printing, Process optimization, Printing accuracy, Slicer.

---

*Corresponding author; *e-mail*: alexandru-ionut.irimia@academic.tuiasi.ro

1. Introduction

Fused Filament Fabrication (FFF) is an extrusion-based technology which uses thermoplastic polymers to additively build parts. Because is an additive technology, parts' geometrical complexity does not significantly influencing the manufacturing cost. Depending on the material layer locus, its structure can vary. However, regardless of its locus the material layer can be made out of walls (Kristiawan *et al*., 2021) or perimeters (Mishra *et al*., 2017), solid fill (Akhoundi and Behravesh, 2019) or solid layers and an internal fill (Milde and Morovič, 2016) or structure (Milde and Morovič, 2016). Each of these building components/ elements is chosen depending on the part's requirements, and their parameterizing is made in the printer's software, which is known as a slicing tool or slicer. In this study, the research will focus only on the infill slicing parameters for the UltiMaker Cura software. Even if the internal structure of the printed part is hidden, it's a significantly influencing factor over the part's mass, center of mass, inertia, mechanical strength, and manufacturing time. For a better understanding of the influence of the infill variable over the resulting part's properties, each infill parameter was analyzed functionality-based. This way a total of 41 parameters were detailed and discussed.

FFF manufacturing technology produces parts with complex geometries by overlapping layers of extruded thermoplastic material, the process consisting of depositing layers of material in a semi-melted state (Patel *et al*., 2022). The entire process of additive manufacturing starts from a CAD model that is built using any 3D modeling platform.

Traditionally, industries used subtractive manufacturing to build products, whereby the parts are manufactured from a solid block of material. In contrast, additive manufacturing uses a layer-by-layer technique (Fig. 1), which allows more complex parts to be made (Su and Al'Aref, 2018).

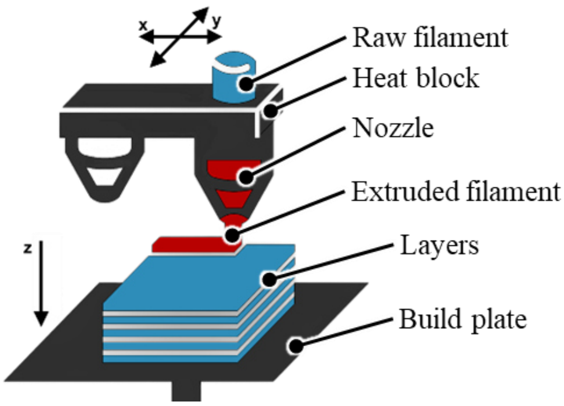

Fig. 1 – FFF part building principle (Adapted after Su and Al'Aref, 2018).

Once the 3D model is finalized, the file is saved in STL format and sent to the slicing software, the G-code is generated and the part is realized by the printer (Tanveer *et al*., 2022). The material is extruded and placed on the build table in a semi-melted state, the newly deposited material merges with the previously deposited material. After the deposition of a full layer, the build platform moves down or up, depending on the type of printer used, along the z-axis equal to the filament height (layer thickness), and the next layer is deposited on top of it (Dudescu and Racz, 2017; Prabhakar *et al*., 2021).

The 3D printer makes a part in the following steps (Fig. 2):

- the first layers of material, the base layers, are deposited;
- the external and internal walls of the part are deposited;
- the infill pattern is printed;
- the top layers of material are deposited.

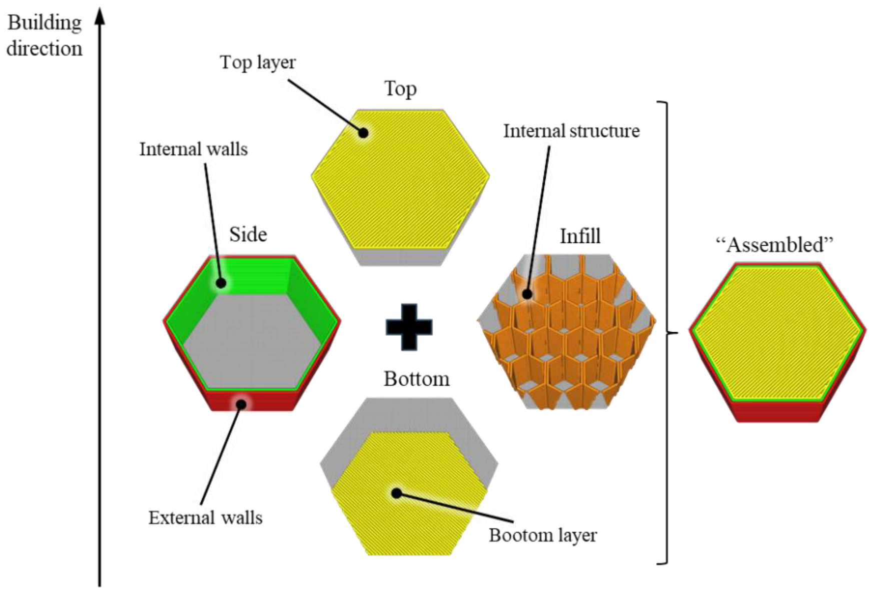

Fig. 2 – Stages of the additive construction process.

Infill density is the amount of filament used for the inside of the 3D prints and is a percentage of the total volume of the object. The density of the printed part leads to increased manufacturing time but also increases material consumption (Prabhakar *et al*., 2021). Internal structure infill density range from 0 to 100% (Fig. 3).

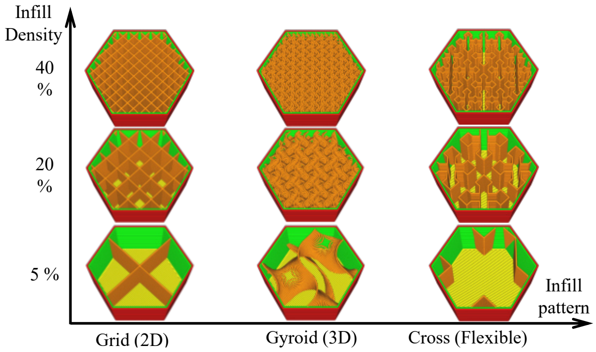

Fig. 3 – Comparison of different infill patterns and density.

2. Infill factors

2.1. Influencing factors

The Ishikawa method, also known as the cause-effect diagram, was used to identify the process parameters used in FFF technology. This method led to the identification of process parameters. The parameters of interest in this article are those of infill. Following the application of the Ishikawa method (Fig. 4), a number of seven parameters, marked with red in the Fig. 4, directly related to infill were identified: walls, infill, material, speed, support, special models and experimental. These parameters present other subcategories of parameters, resulting in a total of 41 parameters.

The parameters studied in this article have been taken from the UltiMaker Cura software.

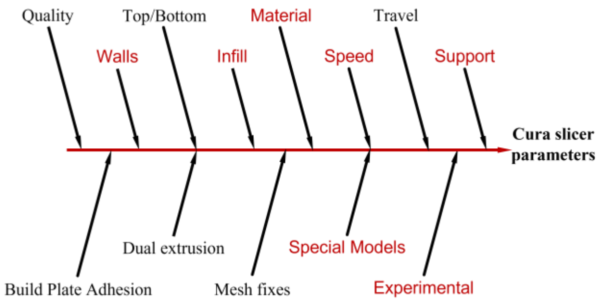

Fig. 4 – Cura slicer parameters based on Ishikawa diagram.

2.2. Infill parameters

Based on the cause-effect diagram, the parameters of interest were extracted from Table 1

Table 1
Infill parameters available in Cura Slicer (Cura Print settings)

Parameter tab	No.	Parameter	Parameter level
Quality	1	Infill Line Width	(0.8-2.5)*nozzle size (Ermolai and Irimia, 2021)
Infill	2	Infill Extruder	Extruder no.
	3	Infill Density	0-100%
	4	Infill Pattern	16
	5	Connect Infill Lines	Yes/ No
	6	Connect Infill Polygons	Yes/ No
	7	Infill Line Directions	0-180°
	8	Infill X Offset	Value (mm)
	9	Infill Y Offset	Value (mm)
	10	Randomize Infill Start	Yes/ No
	11	Infill Line Multiplier	No.
	12	Extra Infill Wall Count	No.
	13	Cubic Subdivision Shell	Value (mm)
	14	Infill Overlap	Value (mm)
	15	Infill Wipe Distance	Value (mm)
	16	Infill Layer Thickness	Value (mm)
	17	Gradual Infill Steps	No.
	18	Infill Before Walls	Yes/ No
	19	Minimum Infill Area	Value (mm^2)
	20	Infill Support	Yes/ No
	21	Infill overhang angle	0-90°
	22	Skin edge support thickness	Value (mm)
	23	Infill Overhang Angle	Degrees (°)
	24	Skin Edge Support Layers	No.
	25	Lightning Infill Support Angle	Value (°)
	26	Lightning Infill Overhang Angle	Value (°)
	27	Lightning Infill Prune Angle	Value (°)
	28	Lightning Infill Straightening Angle	Value (°)
	29	Tile size	Value (mm)
Material	30	Infill Flow	1-100 %
Speed	31	Infill Speed	Value (mm/s)
	32	Infill Acceleration	Value mm/s^2
	33	Infill Jerk	Value mm/s
Support	34	Support Infill Line Directions	0-360°
	35	Support Infill Layer Thickness	Value (mm)
	36	Gradual Support Infill Steps	No.
	37	Gradual Support Infill Steps Height	Value mm
Experimental	38	Infill Travel Optimization	Yes/ No
	39	Cross 3D Pocket Size	Value mm
	40	Cross infill Density Image	0-100 %
Special Models	41	Infill Mesh	Volume modifier

2.3. Process parameters detailing

In the following, the infill parameters available in the UltiMaker Cura software, the following parameters are presented (Cura Print settings):

- **Quality**
  - Infill Line Width – The width of every line of infill being draw. The width of a line can be different from the nozzle size simply by extruding more or less material than needed.
- **Infill**
  - Infill Extruder – Enables the selection of the extruder for the infill deposition.
  - Infill Density – The setting configures the density of the volume inside the print from 0 to 100% (Fig. 5).

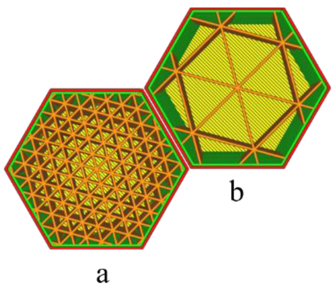

Fig. 5 – Infill Density – (a) high density, (b) low density.

- Connect Infill Lines – This setting connects the endpoints of the infill patterns, where the infill meets the inner wall or skin, using a line that follows the edge of the infill area (Fig. 6).

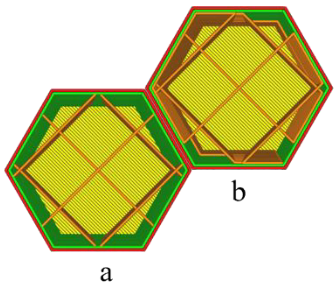

Fig. 6 – Connect Infill Lines – (a) simple inner wall, (b) connected endpoints.

- Infill Pattern – The infill pattern defines a structure that is used to the volume of the object (Fig. 7, Fig. 8).

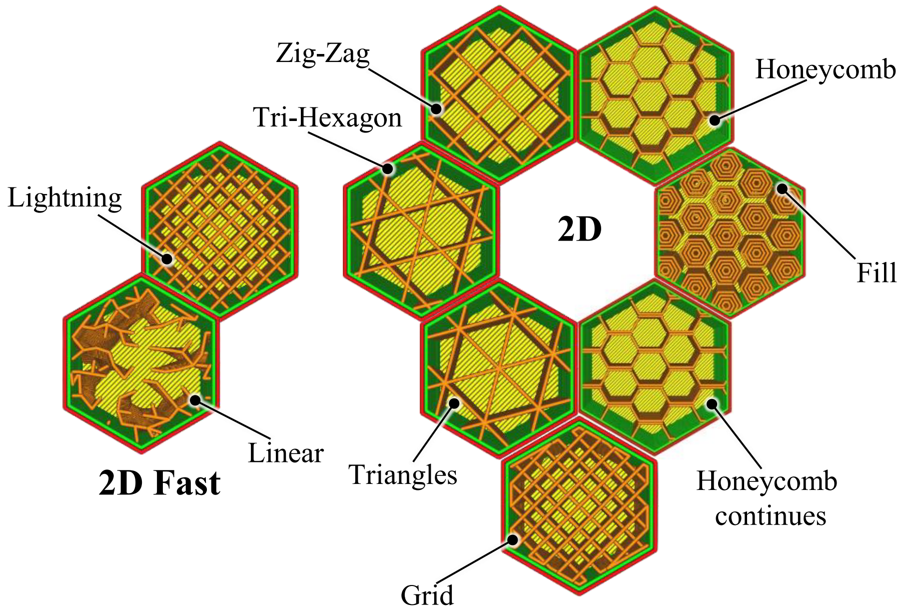

Fig. 7 – 2D Infill Patterns.

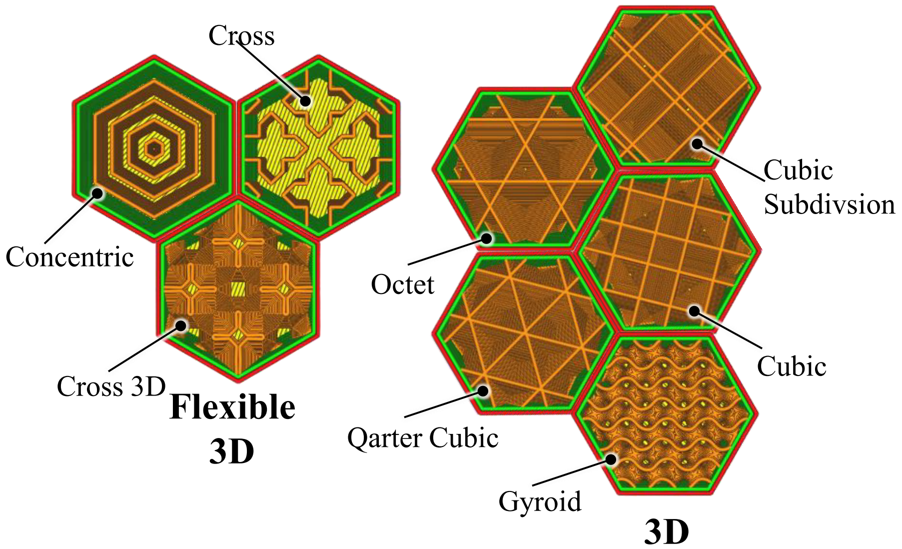

Fig. 8 – Infill Pattern 3D.

- Connect Infill Polygons – The connect infill polygons are used to connect the infill lines.
- Infill Line Directions – This parameter refers to the angle of infill lines (Fig. 9).

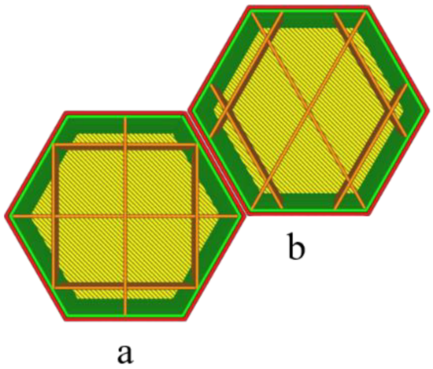

Fig. 9 – Infill Line Directions – (a) normal infill, (b) rotated infill.

- Infill X, Y Offset – This parameter allows shifting the center of the pattern (Fig. 10).

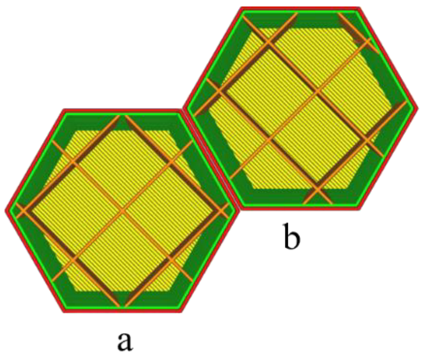

Fig. 10 – Infill X, Y Offset – (a) centered infill, (b) shifted infill.

- Randomize Infill Start – This parameter is used by software to randomly start each layer of infill in a different location.
- Infill Line Multiplier – It converts each infill line into multiple lines that do not cross over each other (Fig. 11).

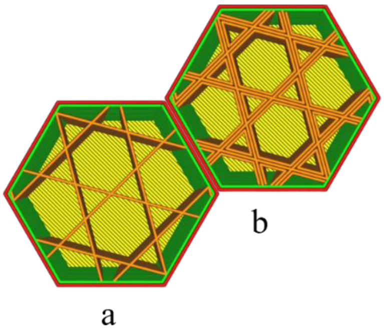

Fig. 11 – Infill Line Multiplier – (a) one line infill, (b) multiple line infill.

- Extra Infill Wall Count – Help to prevent the top layers from gaps by having those extra walls for your top layers to print onto when there are gaps in the infill (Fig. 12).

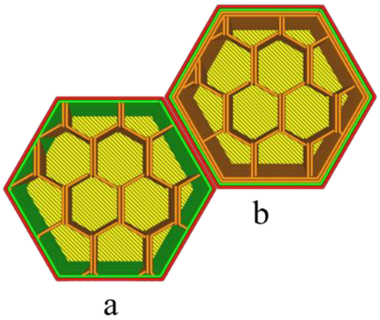

Fig. 12 – Extra Infill Wall Count – (a) standard walls, (b) extra walls.

- Cubic Subdivision Shell – Makes the cubic subdivision infill pattern start reducing infill a bit more towards the inside.
- Infill Overlap – It is the overlap of the infill into the wall, the bond between infill and walls.
- Infill Wipe Distance – The infill wipe distance determines the distance which the printer will travel over the print to clear any excess material from the nozzle before moving on to the next layer.
- Infill Layer Thickness – Infill Layer Thickness determines the thickness of the layers used in the infill part.

- Gradual Infill Steps – This parameter allows to reduce the amount of infill used on the part by decreasing the infill percentage in the lower layers – at the bottom of the model.
- Infill Before Walls – If this parameter is on, the printer start to print the infill before the walls of a 3D print.
- Minimum Infill Area – This parameter enables the minimum area required for infill to be printed in a specific area of the print.
- Infill Support – When is enabled, this treats infill as support. Infill will then only generate where it is needed to support the top surface.
- Infill overhang angle – When using infill support, this parameter determines the minimum overhang angle of the surface that needs to be supported.
- Skin edge support thickness – This setting adds an extra line through the infill to support the edge of the skin, so that ot sags a little bit less.
- Infill Overhang Angle – This adjusts the thickness of the extra infill that is added to support the edges of a 3D model that has curved surfaces.
- Skin Edge Support Layers – The Skin Edge Support Layers add an extra line through the infill to support the edge of the skin so that it sags a little bit less.
- Lightning Infill Support Angle – The Lightning Infill pattern is only intended to support the print from the inside.
- Lightning Infill Overhang Angle – Lighting infill will only support the top side of the model from the inside where it overhangs.
- Lightning Infill Prune Angle – The Lightning infill pattern produces a tree-like structure on the inside of the print which starts small, but branches out to reach all the parts of the top of the print that need to be supported from the inside.
- Lightning Infill Straightening Angle – This parameter determines how steeply the Lightning infill pattern may overhang internally.
- Tile size – Affects the infill density of the object.

- **Material**
  - Infill Flow – This parameter adjusts the flow rate for the infill only.
- **Speed**
  - Infill Speed – The speed at which the infill material is printed can be configured separately from the normal print speed.
  - Infill Acceleration – Controls how fast the nozzle accelerates in different directions while printing the infill.
  - Infill Jerk – Determines the speed at which the nozzle can go through corners while printing infill.
- **Support**
  - Support Infill Line Directions – With this parameters, the orientation of the support lines can be customized.
  - Support Infill Layer Thickness – Configures how think the layers will be that the support is printed with.
  - Gradual Support Infill Steps – Gradual support reduces the amount of support material used, by reducing the support density in the lower layers.
  - Gradual Support Infill Steps Height – When is used gradual support, the density of the support gets periodically reduced from top to bottom in several steps.
- **Experimental**
  - Infill Travel Optimization – If this parameter is enabled the travel time for print it’s reduced.
  - Cross 3D Pocket Size – The Cross 3D Pocket Size causes the pattern to avoid crossing in order to avoid being too rigid in certain places.
  - Cross infill Density Image – For this parameter is posible to provide an image that specifies the density in various places across the infill.
- **Special Models**
  - Infill Mesh – This mesh modifies the infill or other meshes with which it overlaps.

3. Conclusions

This study highlight the importance of infill parameters in Fused Filament Fabrication (FFF), directly influencing production time and material consumption. The infill level, ranging from 0% to 100%, should be chosen based on application requirements: a high density increases mechanical strength but also material usage, while a lower density speeds up manufacturing but may weaken the structure of the printed part. The type of infill structure that can be chosen can be either 2D or 3D. If rapid prototyping is desired, it can be obtained with a fast 2D model infill.

Infill structures can be either 2D or 3D, each with its own benefits. 2D patterns are ideal for fast prototyping, whereas 3D structures provide better distribution of mechanical stresses, making them suitable for applications requiring both strength and material efficiency.

To analyze the impact of infill parameters, the Ishikawa method was applied, leading to the identification of 41 relevant parameters categorized into quality, infill, material, and speed. These parameters are crucial for optimizing the printing process, allowing users to adjust settings according to the specific needs of their printed parts.

Choosing the right infill parameters depends on the purpose of the printed object. For models that do not require high mechanical strength, a low infill percentage is sufficient, whereas for parts that must withstand greater loads, denser infill structures are necessary. The study suggests that future research should focus on advanced optimization methods for infill settings to achieve the best balance between material usage, printing time, and mechanical performance.

A detailed understanding of these parameters offers significant opportunities for improving the FFF process, enhancing the quality of printed parts, and increasing manufacturing efficiency. Selecting the right settings can reduce costs and improve component performance, making this a promising field for future research.

REFERENCES

Akhoundi B., Behravesh A.H., *Effect of filling pattern on the tensile and flexural mechanical properties of FDM 3D printed products*, Experimental Mechanics, 59, 883-897 (2019).

Dudescu C., Racz L., *Effects of raster orientation, infill rate, and infill pattern on the mechanical properties of 3D printed materials*, Acta Universitatis Cibiniensis, Technical Series, 69(1), 23-30 (2017).

Ermolai V., Irimia A.I., *Influence of nozzle parameters in 3D printing under the manufacturing time*, Bul. Inst. Polit. Iași, Machine constructions Section, 67(3), 63-72 (2021).

Kristiawan R.B., Imaduddin F., Ariawan D., Ubaidillah Arifin Z., *A review on the fused deposition modeling (FDM) 3D printing: Filament processing, materials, and printing parameters*, Open Engineering, 11(1), 639-649 (2021).

Milde J., Morovič L., *The influence of internal structures in fused deposition modeling method on dimensional accuracy of components*, Research Papers Faculty of Materials Science and Technology Slovak University of Technology, 24 (38), 73-80 (2016).

Mishra S.B., Malik R., Mahapatra S., *Effect of external perimeter on flexural strength of FDM build parts*, Arabian Journal for Science and Engineering, 42, 4587-4595 (2017).

Patel R., Desai C., Kushwah S., Mangrola M.H., *A review article on FDM process parameters in 3D printing for composite materials*, Materials Today: Proceedings, 60, 2162-2166 (2022).

Prabhakar M.M., Saravanan A.K., Lenin A.H., Mayandi K., Ramalingam P.S., *A short review on 3D printing methods, process parameters and materials*, Materials Today: Proceedings, 45, 6108-6114 (2021).

Su A., Al'Aref S.J., *History of 3D printing*, In 3D printing applications in cardiovascular medicine (pp. 1-10), Academic Press (2018).

Tanveer M.Q., Mishra G., Mishra S., Sharma R., *Effect of infill pattern and infill density on mechanical behaviour of FDM 3D printed Parts-a current review*, Materials today: proceedings, 62, 100-108 (2022).

*** Cura Print settings https://support.UltiMaker .com/s/topic/0TO5b000000Q4w3GAC/cura-print-settings, accessed on data 02.01.2025.

SCURTĂ PREZENTARE A PARAMETRILOR DE UMPLERE ÎN FABRICAȚIA CU FILAMENT FUZIBIL

(Rezumat)

Fabricația prin Filament Fuzibil (FFF) este un proces de fabricație aditiv bazat pe extrudarea unui material termoplastic pe o placă încălzită pentru producerea de piese tridimensionale. Tehnologia de fabricație FFF permite obținerea pieselor cu geometrii complexe prin suprapunerea straturilor de material. Această tehnologie este utilizată în diverse domenii pentru realizarea atât a prototipurilor fizice, cât și a pieselor funcționale. Acest articol se concentrează asupra parametrilor procesului de umplere disponibili în software-ul UltiMaker Cura. Pe baza metodei Ishikawa, au fost identificați șapte parametri direct corelați cu procesul de umplere. Acești parametri conțin și alte subcategorii, rezultând un total de 41 de parametri, care pot fi găsiți în următoarele secțiuni ale setărilor software-ului de secționare: calitate, umplere, material, viteză, modele speciale, suporturi și experimental. Acești parametri influențează multiple caracteristici ale pieselor fabricate, precum estetica, consumul de material, ceea ce afectează în mod direct costurile de producție și timpul de fabricație.