Assessing the influence of significative parameters used in 3D Fused Filament Deposition

C. Pisarciuc
Transilvania University of Brasov, Romania
pisarciuc.c@unitbv.ro

Abstract. The present article presents an analysis of the several parameters used at Fused Filament Deposition-FFD 3D printing process. To evaluate these parameters, a laptop / tablet support has been chosen as a subject that presents different characteristics both from the point of view of mechanical demands, to which it is subject, as well as geometry. The reason for choosing such an object to be produced by additive technology is the fact that in its geometry are encountered most of the spatial surfaces that present difficulties to be obtained by this method. In this context, the article highlights the steps that are necessary to get 3D printed objects to meet the initial design requirements. For the assessment of the degree of influence, parameters were identified and then they were evaluated to find an optimal solution. In the conclusions are presented some useful values for practical applications of this type and some considerations about two types of 3D slicers.

1. Introduction

The 3D printers that use filament are based on additive technology for depositing successive layers of material. The material to be deposited is stored on a filament spool, which ensures the continuous feeding of the extruder. The extruder is feed with the material, heats it at a certain temperature, and places it in successive layers of a certain thickness and in a certain shape following a specific path of each layer. In other words, in this process, a thermoplastic filament is extruded through an extruder and deposited into successive layers [1, 2].

The main factor differentiating printers in the additive manufacturing (AM) category is the material used to obtain the final body. This article will provide examples for a Fused Filament Deposition (FFD) printer. In some cases, in order to avoid the use of registered names, the Layer Plastic Deposition - LPD is also used to characterize the used technology. Unfortunately, this latest acronym, although it better describes technology, is also used in other areas.

2. Printer parameters and object characteristics

The first step in obtaining an object through fused filament fabrication additive technology is to configure the printer's parameters, which is done according to the printer's characteristics, the type of material used, the shape and characteristics of the final object. Configuration is done using specific software. In this case, it was used Z-Suite [3] and Slic3r [4] software. The Z-suite is printer's proprietary software and the process is much simplified. In contrast, with Slic3r the configuration is more laborious, but not complicated.
2.1. Object properties

As a subject to be 3D printed 3D by additive technology is a laptop / tablet support as shown in the figure 1.

![Figure 1. Laptop / tablet support and its dimensions.](image)

The dimensions and shape of the support were chosen to accommodate either a laptop or a tablet under static equilibrium conditions. The reason for choosing such an object to be produced by additive technology is the fact that in its geometry are encountered most of the spatial shapes that present difficulties to be obtained by this method.

Considering the destination, it is necessary that the material from which the support is built can maintain its properties at relatively high temperatures under stiffness conditions. The material that meets both conditions simultaneously is the polylactic acid PLA type [5]. PLA is the most common 3D printing material because it is easy to use and is made from renewable resources and thus, biodegradable. PLA filament is useful in a broad range of 3D printing applications, being both odourless and low-warp, and does not require a heated bed [6, 7].

Although object orientation is not a configuration parameter or a print parameter, it plays a very important role in choosing the other print parameters. Specifically, depending on the spatial geometry of each printed object, it is necessary to determine the optimal orientation of the deposited layers. The support surface of the object on the depositing tray is recommended to be as large as possible. This mode of settlement simultaneously generates an increased stability and good adhesion to printing bed. The layers that make up the inclined surfaces outside the previously deposited layer face difficulties in obtaining them with respect to the planar surfaces. For the above reason, in order to avoid aesthetic and precision defects, it is necessary for the printed object to exhibit as few inclined surfaces as compared to the previously deposited state. The optimal orientation of the support is presented in first picture from figure 1.

2.2. Dimensions of the build platform

This project will use a Zortrax M200 [8] 3D printer. The dimensions of the print bed define the maximum distances traversed by the print head along the X and Y axes and are specific to each individual printer. For mentioned printer is a 200 mm square. Considering this form, the origin can be set in the lower left corner.

2.3. Diameter of the hot end

The hot end diameter determines the maximum height of the deposited layer (on Z-axis) and its width (on XY plane). The minimum height of the deposited layer is also determined by the positioning accuracy of the extruder on the Z-axis. Opposite, the maximum height cannot be larger than the diameter of the nozzle. The width of the deposited layer correlates with the diameter of the nozzle but also depends on the extrusion speed (usually 120% of nozzle diameter) [9]. In this context, the extrusion speed is the amount of material that the print head can deposit in the time unit.
2.4. Filament diameter
Specifically, there are several diameters available for the filaments used. Usually, there are no major
differences in the results obtained depending on the diameter used but the diameter of the filament must
be correlated with the diameter of the nozzle. For this support, a 1.75 mm filament was chosen.

2.5. Printing temperature
The extrusion temperature is the temperature reached by the hot end during deposition. This temperature
is extremely important in relation to the quality of the printed object. The main influence of the extrusion
temperature is exerted on the final surface roughness of the object. As a rule, PLA materials have a melting
point in the range of 160 ~ 230 degrees Celsius and the value chosen to achieve the support is 200°C.

3. Process parameters
Configuration of print parameters is a necessary step because extrusion parameters depend on the
characteristics of the printed object. In order to be able to configure each of the extrusion parameters, it
is necessary to use a "3D slicer" software. The main role of this program is to slice the 3D model into
horizontal layers whose characteristics are determined by the chosen extrusion parameters. The next
step the program performs is to calculate the amount of extruded material and the time it takes to perform
the deposition operation, and, in the end, to transpose all this information into a G-Code file. In the end,
the file is copied on an SD card memory and loaded onto machine. This, in turn, performs the instructions
in the file just like any CNC machine.

For this article, the same two software will be used as in the case of setting up the parameters of the
3D printer. In order to compare, the same object will be sliced with the help of Z-Suite and an Open-
Source software, 3D slicer. As documented below, the Z-Suite software Z-Suite is a proprietary one,
offering less advanced options for configuring the printing parameters.

3.1. Layer height
The height of the deposited layer is the thickness of each layer, being determined by the length of the
step of the extruder on the Z-axis before depositing each layer. The height of the deposited layer has the
greatest influence on the resolution and time of obtaining the object. Depending on the desired
characteristics of the printed object, a larger thickness of the deposited layer will be chosen if the
appearance of the object is not a priority and a lower thickness otherwise. In the case of objects whose
geometry is formed by inclined surfaces, it is favourable to obtain the object by depositing a larger
number of layers resulting in a lower height of the deposited layer [10].

In the case of the printer used to make the support, the minimum height of the deposited layer is
150 μm, given the positioning accuracy of the extruder on the Z-axis. In addition, the minimum height
of the deposited layer is also related to the nozzle diameter, i.e. 20% of it [11]. For a nozzle diameter of
0.5 mm, the recommended minimum height of the deposition layer is 0.1 mm. As a rule, a maximum
height of the deposited layer is recommended with a value between 50% and 75% of the size of the
nozzle. Although objects can be obtained using a layer height greater than the recommended layer, the
effect will be a decrease in the mechanical strength of the object and an increase in the roughness of its
surfaces [11].

Considering the above, within this article, given that the main purpose of the laptop / tablet support
is to provide support, a height of the 0.375 mm deposition layer should be chosen, this being 75 % of
the deposition head diameter value. This will result in a decrease in the time required to obtain the object
in exchange for outer surfaces with a higher roughness. If the appearance is more important than other
aspects, then the suggested value for layer height is 0.15 mm.

3.2. Layer width
The width of the deposited layer is determined by the feeding speed of the filament, the speed at which
the extruder moves while depositing the filament and the diameter of the hot end diameter. The
minimum width is determined by the diameter of the hot end, since during deposition the deposited
material needs to be pressed between the hot end and the previously deposited layer to adhere thereto. Thus, in order not to affect the integrity of the printed object, a minimum width of at least 20% greater than the diameter of the hot end is recommended [12]. The upper limit of the layer width is limited by the arcing phenomenon of the layer deposited under the influence of the pressure exerted by the hot end. It is recommended to use a width up to twice the size of the deposition nozzle diameter to avoid this phenomenon [12].

The optimal width of the deposited layer depends on the characteristics that the 3D printed object must possess and the ratio between the deposited layer height and the diameter of the hot end. In addition, the optimal width differs depending on the type of layer that is deposited (top, outer, or bottom). In depositing the bottom layer, the contact surface with the deposited deposition tray increases directly in proportion to its width, thus facilitating good adhesion.

Another important factor to consider when setting the layer width is the bond between the lateral surfaces of the filaments composing the deposited layers. This bond determines the roughness of the top layer and the inclined surfaces, but also the mechanical strength of the object under the action of a force applied vertically thereto, in other words a vertical shear force.

Only the Slic3r software allows the adjustment of the size of the connecting surface between the lateral surfaces of the deposited layers by offering the possibility to configure the overlapping coefficient between them. The overlapping coefficient represents the percentage of the deposited filament width dimension that overlaps with the deposited filament on one of its sides. It is necessary to keep in mind that in order to avoid a high roughness of the horizontal surfaces it is required that the volume of the free space between the deposited filaments is not exceeded.

For the support presented, a value of 1 mm of depositing layer and 15% of overlapping coefficient will be used. By choosing this value, good adhesion will be obtained between the lateral surfaces of the filaments deposited.

3.3. Infill Percentage

In the case of 3D filament printing technology, the term "infill" refers to the density of which the internal structure of the object is composed. This parameter influences the weight of the object, the required production time, the amount of material and the mechanical strength of the object. The higher the infill percentage is, the more resistant the object becomes and the deposition time, the amount of material required and the weight increase.

The degree of infill is measured as a percentage, so a 0% infill rate describes an object that does not have the inner structure and a 100% infill rate is an object whose internal structure is entirely composed of the material. According to [13], in most cases, 5% - 15% infill is enough and 100% infill is very rarely used. If an object is produced for appearance only it can be obtained using a low infill rate, thus significantly reducing the cost and time it takes to obtain it while an object whose role is to take over forces (e.g. laptop / tablet support) will need a higher degree of infill.

Taking into account that the laptop / tablet support will be subjected to compressive force it is necessary to consider the required resistance of the object so that it can exercise its role. According to the studies carried out [14], besides the infill degree, the mechanical strength of the objects in the case of compression is directly influenced by the height of the deposited layer. Due to the orientation chosen on the tray, the area of the laptop / tablet support that is in contact with it will support all the compressive force. To determine an optimal degree of infill, it is necessary to calculate the area and determine the pressure to be exerted on it. The program used to configure 3D print parameters does not allow measurement of this area, but this value can be extracted from CAD model. The cumulated area of the contact surface that are placed horizontally on the table is about 52 cm². The average weight of a laptop is about 2.5 kg, tablets being lighter that notebooks. Using the data from [14], the base surface of the support requires a minimum resistance of about 0.005 MPa, the resistance achieved for a linear fill model even using the worst-case parameters. Thus, in order to reduce the amount of material used and the time required to obtain the object, a filling degree of 10% will be used for the support. This decision is justified by the need for support for the upper layers of inclined surfaces present in the geometry of the object.
3.4. Infill pattern

The deposition pattern is the model inside the manufactured object, thus giving it a certain structure. There are a variety of available infill patterns, each of which has certain advantages and disadvantages. Depending on the geometry of the object, in addition to providing an internal structure of the object's resistance, the infill pattern has in some cases also a supporting role for the deposited layers, thus preventing the deformation of the object during the deposition if it has edges suspended in its geometry. The same pattern can be deposited using different filling degrees.

The most used models where the object must withstand compressive forces are linear, triangular and hexagonal [15]. The linear model is the standard infill model for objects obtained through the filament printing technology. It has resistance to the compressive forces applied to each of the three axes, requiring the shortest production time due to the reduced complexity of the extruder path during its deposition. The triangular pattern has an advantage if the outer walls of the object must withstand mechanical forces, giving the object an increased stiffness. This model requires a longer lead-time than the linear model. The hexagonal model is also popular, showing resistance to the compressive forces applied to each of the three axes. Based on the same study already mentioned [14], for the same degree of infill, the linear deposition pattern of the inner layers exhibits the highest mechanical strength in the case of compression, as presented in figure 2. In conclusion, for the support will be selected a linear pattern.

3.5. Number of layers

The outer layers can be viewed as a casing of the object. In the case of the additive filament printing technology, the outer layers are the first to be deposited on each layer [15]. The outer layer deposition path is defined by the outer contour of the 3D printed object.

The most important considerations to be taken in account for the outer layers are the mechanical strength, the possibility of finishing, the quality of the inclined surfaces, the time and the material required for the production, and the relationship between the diameter of the hot end and the dimensions of the outer layers. A higher number of vertical shells results in a thicker case of the object, and thus in an increased resistance of its mechanical strength. In this way, an increase in the robustness of the object is achieved without the need for an increase in the degree of filling. As with infill, the time and material requirements increase directly in proportion to the number of outer layers deposited.

Considering that the outer layers must provide mechanical resistance to the object, it is recommended to print them at a width twice the diameter of the print nozzle. This provides the required pressure between the deposited layer and the previously deposited layer thus facilitating good adhesion between the two layers. Because the mechanical strength of the object is already guaranteed by the fact that it has an infilling structure, the number of layers chosen is justified by obtaining the inclined surfaces without the visible filling structure of the object. According to the simulations performed in the Slic3r software, the number of two outer layers is enough to cover the object filling structure on the sloping surface area.

![Figure 2. The relation between infill pattern and strength][14].
3.6. Travel speed
The travel speed is the speed of the extruder when it does not deposit material. The main benefit of a high travel speed is the decrease in the time that the object is printed. A high travel speed contributes to improving the appearance by eliminating the phenomenon of stringing of the filament during the repositioning of the extruder. The higher the repositioning distance is, the more the phenomenon of stringing is evident. Thus, by using a high repositioning velocity, the material present in the hot end is less likely to drain during repositioning. Since the support has, in the same layer, separate surfaces between them, it is necessary to use a higher travel speed to avoid the phenomenon of stringing of the filament. Theoretically, for the printer used, the maximum travel speed may reach 3.5 m/s if the repositioning distance allows the extruder to accelerate to that speed. However, a high acceleration of the extruder and a sudden stop when it has reached its destination may produce deviations from positioning accuracy due to inertia. Consequently, the maximum recommended travel speed is 250 mm/s.

3.7. Print speed
The print speed is the speed at which the deposition head (the extruder) moves while depositing the material. This speed is not limited by the mechanical structure of the printers or electric motors but by the amount of material the extruder can deposit in a unit of time. The main advantage of a high print speed is the decrease in the time required to obtain the object.

Consideration should also be given to the negative effects that a high print speed can have on a deposited object. The filament is subjected to expansion during deposition due to the high temperatures at which it is heated to reach the melting point. After being deposited, the plastic layer contracts while returning to ambient temperature. A small thickness of the deposited layer favours its rapid cooling, and because it is composed of a small amount of material, during contraction it is less prone to the arcing phenomenon. In other words, in order to avoid the arcing phenomenon of the deposited layers, they need a short cooling time before the next layer is deposited over them. Thus, the optimal deposition rate is a parameter that varies according to the horizontal plane dimensions of the object. When depositing exterior layers, it is advisable to use a lower deposition rate than the layers that make up the infilling structure because the aesthetic defects occurring in these layers will be visible.

Given that the object chosen in this project is of large size and the geometry of its layers is composed predominantly of straight lines, the probability of occurrence of the arcing phenomenon of the layers is relatively small. It is thus possible to choose a printing speed in order to decrease the required production time. Consequently, the selected print speed will be 80 mm/s.

4. Discussion and evaluation
The first step in obtaining an object through additive technology is to define its purpose and its features. Depending on this, 3D object modeling is done and a suitable printer is selected. After the object geometry is defined, the best positioning on the printing bed is identified in order to minimize defects and print time.

Specifically, for the analysed object, a PLA type material and a printer (Zortrax M200) with appropriate specifications are selected. The next step involves configuring print parameters. Choosing optimal parameters must be done in relation to the final characteristics of the object. In this article, the mechanical strength, the time to print and the amount of material used were considered. In Table 1, the influence of these parameters is presented, with a rating of 0 to 2, depending on the importance. If a criterion is not influenced by a parameter, it will get the 0 rating.

The optimal setting of the depositing parameters will consider the degree of influence of the print parameters according to those presented in Table 2. With Slic3r software can be estimated, deposition time and the quantity of material used. It is important to note that if the minimum quantity of material is to be used then the overlap coefficient is 0%, otherwise is 15%. The values for 3D printer parameters remain constant even in the optimal configuration because they are determined by the printer's technical specifications and the material used. The values for 3D printer parameters remain constant even in the optimal configuration because they are determined by the printer's technical specifications and the material used.
Table 1. 3D print parameters evaluation according to Slicer software.

| Parameter                     | Appearance | Mechanical resistance | Time | The amount of material used |
|-------------------------------|------------|-----------------------|------|-----------------------------|
| **3D printer parameters**     |            |                       |      |                             |
| Hot end diameter [mm]         | 1          | 0                     | 1    | 0                           |
| Filament diameter [mm]        | 0          | 0                     | 1    | 0                           |
| Extrusion temperature [°C]    | 1          | 1                     | 0    | 0                           |
| **3D print parameters**       |            |                       |      |                             |
| Layer height [mm]             | 2          | 1                     | 2    | 0                           |
| Layer width [mm]              | 1          | 2                     | 0    | 1                           |
| Infill degree [%]             | 1          | 2                     | 2    | 2                           |
| Infill pattern                | 0          | 2                     | 2    | 0                           |
| Number of lower layers        | 1          | 1                     | 1    | 1                           |
| Number of outer layers        | 2          | 1                     | 1    | 1                           |
| Number of upper layers        | 2          | 1                     | 1    | 1                           |
| Support structure             | 1          | 0                     | 1    | 2                           |
| Travel speed [mm/s]           | 1          | 0                     | 2    | 0                           |
| Printing speed [mm/s]         | 2          | 1                     | 2    | 0                           |

Table 2. Time and material specifications according to Slic3r software.

| Parameter                     | Appearance | Mechanical resistance | Time | The volume of used material |
|-------------------------------|------------|-----------------------|------|-----------------------------|
| **3D printer parameters**     |            |                       |      |                             |
| Hot end diameter [mm]         |            | 0.50                  |      |                             |
| Filament diameter [mm]        |            | 1.75                  |      |                             |
| Extrusion temperature [°C]    |            | 200                   |      |                             |
| **3D print parameters**       |            |                       |      |                             |
| Layer height [mm]             | 0.1        | 0.375                 | 0.375| 0.375                       |
| Layer width [mm]              | 0.6        | 1                     | 1    | 0.6                         |
| Infill degree [%]             | 10         | 100                   | 0    | 0                           |
| Infill pattern                | Linear     | Linear                | Linear| Linear                      |
| Number of lower layers        | 3          | 3                     | 1    | 1                           |
| Number of outer layers        | 3          | 2                     | 1    | 1                           |
| Number of upper layers        | 3          | 2                     | 1    | 1                           |
| Travel speed [mm/s]           | 250        | 250                   | 250  | 250                         |
| Printing speed [mm/s]         | 50         | 80                    | 100  | 100                         |
| **Time and material requirements** |           |                       |      |                             |
| Time required [h]             | 14.90      | 5.50                  | 1.20 | 1.50                        |
| Material required [cm³]       | 87.20      | 202.40                | 46.40| 29.60                       |

5. Conclusions
In order to achieve the functions for the support presented, a linear pattern of the infill structure has been selected because it is suitably resisting the compressive forces to which the support is subjected. The
number of lower layers was three to give a solid foundation to the object. A value of two superior layers has been chosen to cover the infill pattern.

The height of 0.375 mm and the width of the outer layer of 1 mm are favourable for adhesion between the layers and to the printing bed. It was preferred the option of depositing two outer layers for covering the inclined surfaces without the filling structure of the object being visible. The deposition height of 0.375 mm represents 75% of the size of the deposition head, being the maximum value for a 0.5 mm diameter nozzle. This value generates less deposition time and less roughness. The 1 mm width of the depositing layer and a 15% interference coefficient gives a good adhesion between deposited layers.

The fill rate of 10% reduces the amount of material used and the printing time. In the same time, the printing speed of 80 mm/s significantly reduces the time required for printing. Since the support has larger dimensions and the geometry of the layers is composed predominantly of straight lines, the probability of arching layers phenomenon to occur is relatively small.

The purpose of the article was not to compare side by side the two Slicer programs. The reason why the two programs were discussed was the desire to control as many parameters as possible so that the final product corresponds best with the initial requirements. For the mentioned purpose, Slic3r is the variant that allows the most modifications. This feature may, however, be a drawback of proper use by a less experienced user. However, as the data in table 2 highlight, the variations, both in the quantities used and in relation to the manufacturing time, are significant. Nevertheless, when these issues are less important, then the Z-suite software is more suited to the average user. Technological aspects are either suggested or chosen by the program in the Automatic version. This facilitates the immediate obtaining of products through additive technology. Usually, commercially available 3D printers are delivered with own software. For replicated printers is necessary to use one of the available software. Thus, this article refers to this situation and when the user wants to optimize exploitation and filament consumption.

6. References

[1] Messimer S L, Patterson A E, Muna N and Deshpande A P 2018 Characterization and Processing Behavior of Heated Aluminum-Polycarbonate Composite Build Plates for the FDM Additive Manufacturing Process J. Manuf. Mater. Process. 12 2-27

[2] Turner B N, Strong R and Gold S A 2014 A review of melt extrusion additive manufacturing processes: I. Process design and modelling Rapid Prototyp. J. 20 3 192-204

[3] Information on https://zortrax.com/software/, accessed: 10.02.2019

[4] Information on https://slic3r.org/, accessed: 10.02.2019

[5] Information on https://store.zortrax.com/materials/zortrax-m200-z-pla-pro, accessed: 10.02.2019

[6] Vasilescu M D and Groza I V 2017 Influence of technological parameters on the roughness and dimension of flat parts generated by FDM 3D printing Nonconventional Tech. Rev. 3 20-24

[7] Information on www.matterhackers.com/news/choosing-the-right-3d-print-bed-surface, accessed: 10.12.2018

[8] Information on https://zortrax.com/3d-printers/m200/, accessed: 10.12.2018

[9] Information on https://www.printitindustries.com/pages/print-speed-calculator, accessed: 10.12.2018

[10] Information on http://support.3dverkstan.se/article/30-getting-better-prints, accessed: 10.12.2018

[11] Information on https://rigid.ink/blogs/news/what-3d-printer-nozzle-size-should-i-use-the-pros-and-cons, accessed: 10.12.2018

[12] Information on http://manual.slic3r.org/advanced/flow-math, accessed: 10.12.2018

[13] Information on https://3dplatform.com/3d-printing-tech-tips-infill-percentage-and-pattern-explained/, accessed: 10.12.2018

[14] Information on What is the influence of infill, layer height and infill pattern on my 3D prints?, available at http://my3dmatter.com/influence-infill-layer-height-pattern/, accessed: 10.12.2018

[15] Information on https://www.3dhubs.com/knowledge-base/selecting-optimal-shell-and-infill-parameters-fdm-3d-printing, accessed: 10.12.2018