

ANALYSIS AND EVALUATION OF THE MAIN PARAMETERS FOR THE 3D PRINTING PROCESS

Cristian Pisarciuc [1]
[1]Transilvania University of Brasov, Romania, pisarciuc.c@unitbv.ro

ABSTRACT: The purpose of this article is to perform a systematic analysis and to present the main parameters for the 3D printing process. To evaluate these parameters, as convincingly as possible, a 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. 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 identify an optimal solution. In the conclusions are presented some useful values for practical applications of this type.
KEY WORDS: additive manufacturing; Fused Filament Deposition, fabrication parts, 3D printing, process

1. INTRODUCTION

A 3D printer is, above all, a numerically controlled machine. Major difference compared to conventional ones, is the fact that the additive technology is used to obtain objects. 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 takes over 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 this process, a thermoplastic filament is extruded through an extruder and deposited into successive layers [1, 2]. To achieve the designed shape and to obtain the desired precision and quality, the extruder generation moves are essential.

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.

The component that makes it possible to deposit successive layers is the extruder. Essentially, it is fed with the plastic filament (cold end), melts the plastic, eliminates it through a nozzle (hot end) and then puts it on the print bed. The cold end is the area through which the filament enters the extruder, its role being to guide the filament to the hot end. From a structural point of view, there are included a motor and two gears (hobbed and idler). Although constructively this assembly is simple, a stepper motor acting on the hobbed and idler assembly between which the filament circulates. Its operation affects the flow of material being deposited.

2. CONFIGURATION OF PRINTER PARAMETERS

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 made using specific software. In this case it was used an open source software, Slicer.

2.1 Object properties

The object chosen to be obtained by the 3D printing method is a laptop / tablet support as shown in the figure1.

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 surfaces 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 [3]. 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 [4, 5].

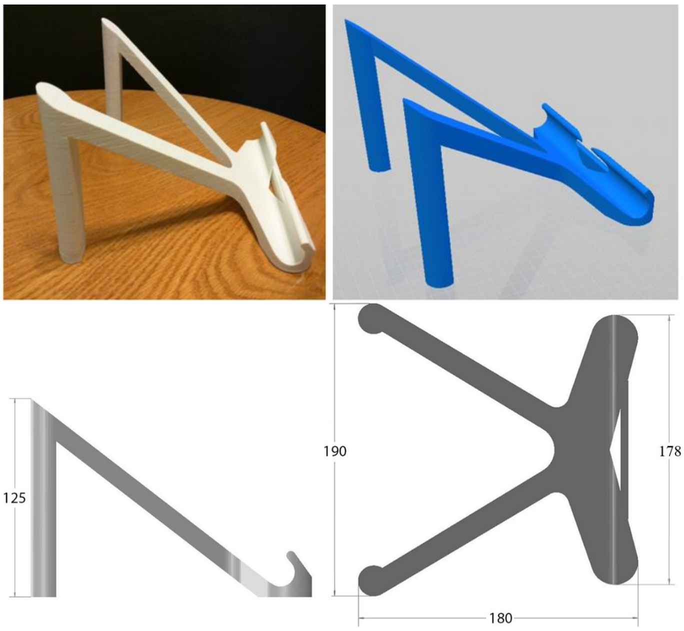


Figure 1. Laptop / tablet support and its dimensions

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.

Because the objects result from the deposition of successive layers, the practical aspects to be taken into account are listed below.

The support surface of the object on the depositing tray is recommended to be as large as possible. This mode of settlement simultaneously generate 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 distance of repositioning of the extruder, while it does not deposit the material, is determined by the necessity of repositioning it between the surfaces of the same layer that are not connected to each other. The longer the repositioning distance is the longer the time it takes to get the object. Since the number of repositioning moves of the extruder increases with the number of separate surfaces in the same layer, the optimal orientation of the object is where the layers are laid continuously or so that only a few separate surfaces result.Due to the above, the optimal orientation of the support is shown in figure 2.

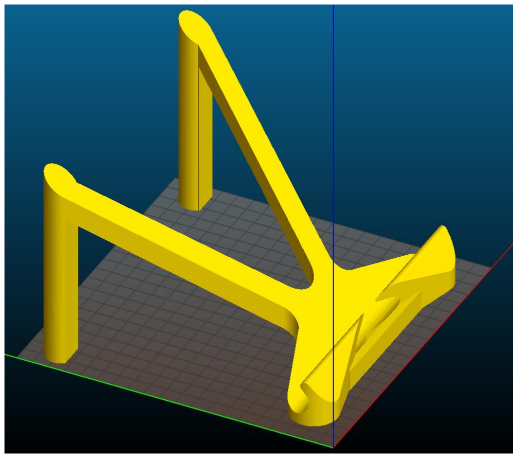


Figure 2. Optimal orientation of the support

2.2 Firmware configuration

Firmware is the software package that allows communication between the 3D printer hardware. This project will use a Zortrax M200 [6] model 3D printer. Thus, the type of firmware used will be the one dedicated to such a printer and this model should be chosen within the Slicer program.

2.3 Dimensions of the build platform

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.

2.4 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) [7]. In this context, the extrusion speed is the amount of material that the print head can deposit in the time unit.

2.5 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.

2.6 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. Depending on the melting point of the material used, a certain temperature must be set. 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. The value chosen to achieve the support is 200^{o} C.

3. CONFIGURATION OF 3D PRINT 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. For this article, the same program will be used as in the case of setting up the parameters of the 3D printer, namely the 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. The file is sent to the 3D printer, and this, in turn, performs the instructions in the file just like any CNC machine.

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 [8].

The time of obtaining the object is directly proportional to the thickness of the deposited layer. A small thickness of the layer translates into the fact that the extruder will go through the same route several times than in the case of a thicker layer, resulting in a higher production time.

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 [9]. For a nozzle diameter of 0.5 mm, the recommended minimum height of the deposition layer is 0.1 mm. The value of the upper limit comes from the need for pressure exerted on the layer deposited by the extruder. 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 [9].

Taking into account 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 than with the use of a lower height of the deposition layer, as presented in figure 3.

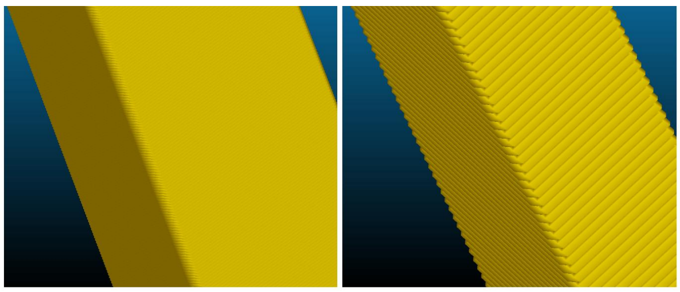


Figure 3. The appearance of the object surface based on the height of the deposition layer

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. By means of an algorithm, the program used to configure the print parameters modifies the extruder filament feed rate according to the deposition rate and the diameter of the hot end diameter to achieve the chosen width of the deposited layer.

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 [10]. 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 [10].

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.

The slicer program used allows adjusting 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 must be taken into account that, depending on the properties of the printing material, its viscosity varies, and in order to obtain the same bonding surface between the lateral surfaces of the deposited filament, the optimal overlapping coefficient will be different between different types of materials. It is necessary to keep in mind that in order to avoid a high roughness of the horizontal surfaces it is necessary 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. Different percentages of infill rate are obtained by adjusting the scale to which the fill pattern is deposited, as presented in figure 4. According to [11], in most cases, 5% - 15% infill is sufficient and 100% infill is very rarely used.

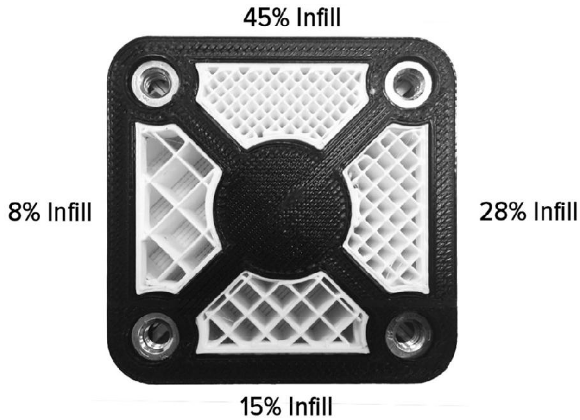


Figure 4. Infill Percentage [11]

If an object is produced for appearance only (eg, figurines) 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 take into account the required resistance of the object so that it can exercise its role. According to the studies carried out [12], 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. Although the program used to configure 3D print parameters does not allow measurement of this area, most CAD modeling programs provide this option.

The laptop / tablet support has a contact surface area that is approximately 52 cm^2 (Figure 5) and the average weight of a laptop is about 2.5 kg. Using the data from [12], 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.

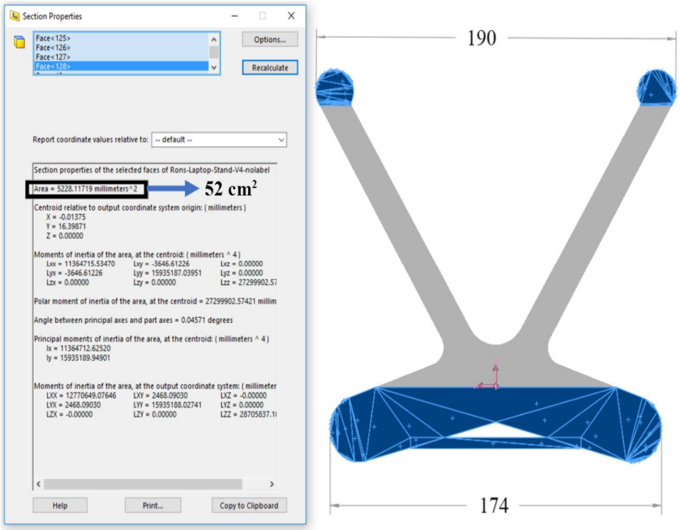


Figure 5. Area of the contact surface with the tray

3.4 Infill pattern

The deposition pattern is the route the extruder enters while depositing material inside the manufactured object, thus giving it a certain structure. The same pattern can be deposited using different filling degrees (Figure 4). There are a variety of available infill patterns, each of which has certain advantages and disadvantages. Depending on the geometry of the object chosen, 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 most used models where the object has to withstand compressive forces are linear, triangular and hexagonal (honeycomb) (Figure 6) [13].

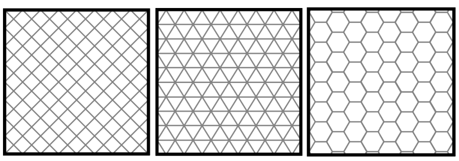


Figure 6. Infill geometry [13]

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 have to 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 [12], for the same degree of infill, the linear deposition pattern of the inner layers exhibits the highest mechanical strength in the case of compression.

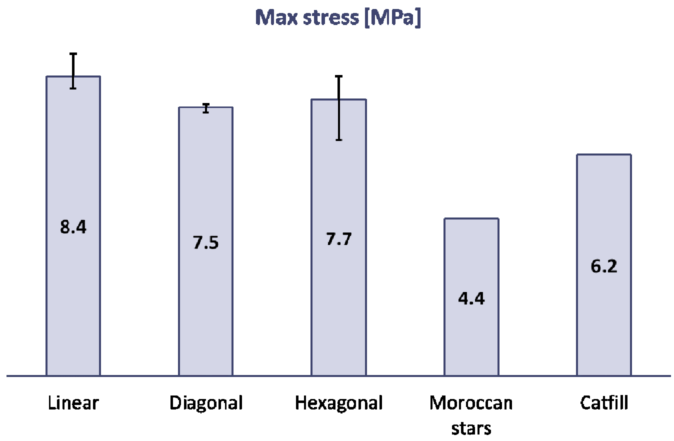


Figure 7. Infill pattern strength [12]

In conclusion, for the support will be selected a linear pattern as documented in figure 8.

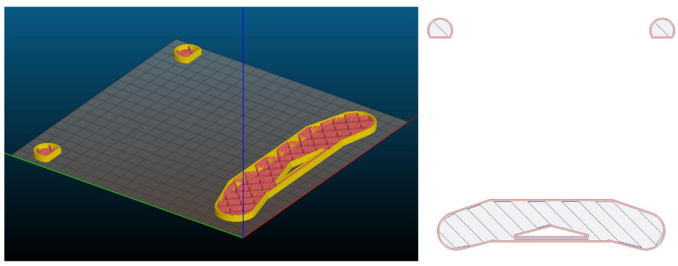


Figure 8. Area of the contact surface with the tray

3.5 Shells

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 (Figure 9) [13]. The outer layer deposition path is defined by the outer contour of the 3D printed object.

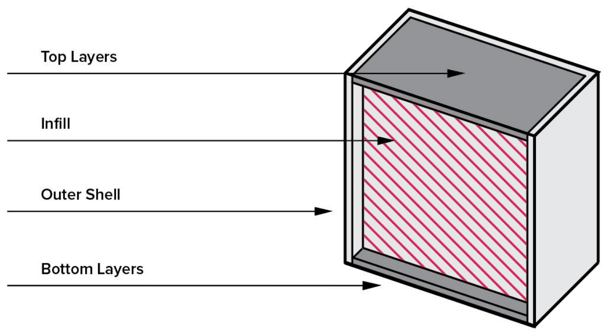


Figure 9. Sections of an FFD print [10]

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 larger number of vertical shells results in a thicker case of the object (Figure 10) 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. Most slicer programs allow the thickness of the object's case to be adjusted in the points where it is most likely to yield under an applied force. As with infill, the time and material requirements increase directly in proportion to the number of outer layers deposited.

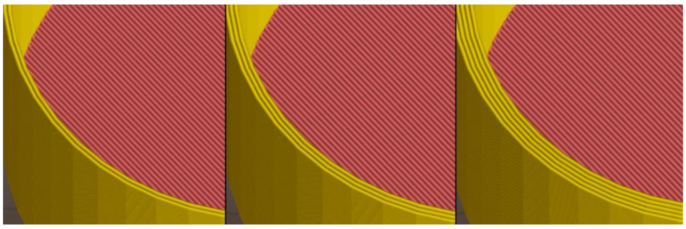


Figure 10. Different outer shells (2, 3, 5) [14]

Taking into account 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 giving it a uniform appearance, as presented in figure 11.

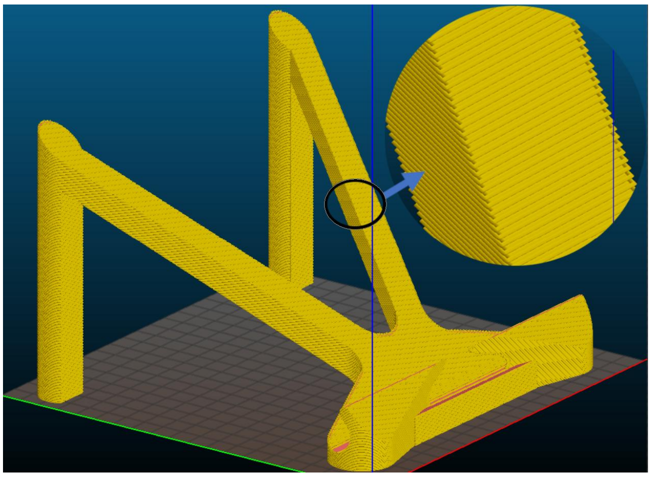
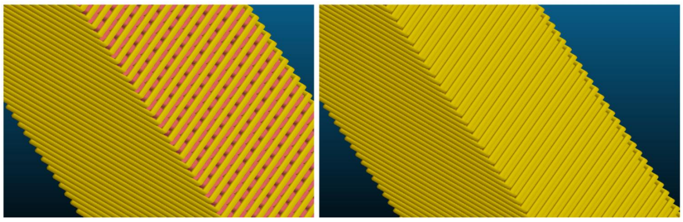


Figure 11. Structure of the sloping surface

According to the simulations performed in the Slicer program, the number of two outer layers is enough to cover the object filling structure on the sloping surface area as it is visible in figure 12.



a. One outer shell b. Two outer shells

Figure 12. The influence of the number of outer layers on the appearance of the object

3.6 Travel speed

The travel speed is the speed at which the extruder moves when it does not deposit material. In the deposition process, the main benefits of a high travel speed are the decrease in the time that the object is printed and the improvement in its final appearance. A high travel speed contributes to improving the appearance by eliminating the phenomenon of stringing of the filament during the repositioning of the extruder. This phenomenon is often caused by a high speed and high extrusion temperature. 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 due to the shorter time it takes to perform.

Since the support shown 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, in the case of 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. Considering the printer used, the maximum recommended travel speed for this one 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 particular parameter, it will get the 0 rating.

Table 1. Assessing the degree of influence of the parameters

	Parameter evaluation according to			
	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

The optimal setting of the depositing parameters will take into account the degree of influence of the print parameters according to those presented in Table 2.With Slicer software can be estimated, depending on the parameters chosen, 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 due to the fact that they are determined by the printer's technical specifications and the material used.

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 number 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.

Table 2. Time and material requirements according to the chosen criteria

	Time and material requirements			
	Appearance	Mechanical resistance	Time	The amount of material used
3D printer parameters				
Hot end diameter [mm]	0.5			
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]	14.9	5.5	1.2	1.5
Material required [cm^3]	87.2	202.4	46.4	29.6

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.

6. REFERENCES

1. Messimer, S.L., Patterson, A.E, Muna, N., Deshpande, A.P., Characterization and Processing Behavior of Heated Aluminum-Polycarbonate Composite Build Plates for the FDM Additive Manufacturing Process, *Journal of Manufacturing and Materials Processing*, vol. 12, pp. 2-27, doi:10.3390/jmmp2010012, (2018).
2. Turner, B.N.; Strong, R.; Gold, S.A., A review of melt extrusion additive manufacturing processes: I. Process design and modelling. *Rapid Prototyping Journal*, Vol. 20, Issue 3, pp.192-204, https://doi.org/10.1108/RPJ-01-2013-0012, (2014).
3. *** - https://store.zortrax.com/materials/zortrax-m200-z-pla-pro, access (2018).
4. Vasilescu. M.D., Groza, I.V., Influence of technological parameters on the roughness and dimension of flat parts generated by FDM 3D printing, *Nonconventional Technologies Review*, vol. 3, (2017).
5. *** - https://www.matterhackers.com/news/choosing-the-right-3d-print-bed-surface, access (2018).
6. *** - https://zortrax.com/3d-printers/m200/, access (2018).
7. *** - https://www.printitindustries.com/pages/print-speed-calculator, access (2018).
8. *** - http://support.3dverkstan.se/article/30-getting-better-prints, access (2018).
9. *** - https://rigid.ink/blogs/news/what-3d-printer-nozzle-size-should-i-use-the-pros-and-cons, access (2018).
10. *** - http://manual.slic3r.org/advanced/flow-math, access (2018).
11. *** - https://3dplatform.com/3d-printing-tech-tips-infill-percentage-and-pattern-explained/, access (2018).
12. ***- 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/, access (2018).
13. *** - https://www.3dhubs.com/knowledge-base/selecting-optimal-shell-and-infill-parameters-fdm-3d-printing, access (2018).
14. *** - https://www.prusaprinters.org/what-are-perimeters-good-for/, access (2018).