- Design for additive processes, rules to follow -

Report *Introduction to additive manufacturing*

Project Number	GE 7
Project Title	Design for additive processes, rules to follow
Date of presentation	17/11/2025
Supervising Professor	Christophe Moser, Jürgen Brugger, Eric Boillat
Student name 1	Elias Pajot
Student name 2	Andres Gilles Bettinger
Student name 3	Paul Cheron
Student name 4	Tristan Jemmely

Table des matières

1 Abstract

Today, the field of additive manufacturing is extremely active, as it allows for a new look at quick prototyping. FDM (Fused Deposition Modeling) gained a lot of popularity in commercial and personal uses due to its ease of learning and use; however, there are some rules that must be followed when this process is used.

First, the right materials must be chosen, and the structure of the part has to be capable of resisting the various mechanical constraints the part will be subject to when it will be in use. The printers have some limitations that must be taken into account during the design, and a bad design might lead the entire print to fail during or after the manufacturing.

FDM printing requires consideration to the printing parameters, material deformation and pre-print preparation. All of these consideration have an impact on the resulting quality of the printed part, and as such require careful consideration depending on the situation. It is possible to reduce the environmental impact of FDM in various ways (energy, materials, quantity of materials, etc.). Finally, there are several solutions for reducing FDM printing time.

2 Introduction

This document presents multiple rules to follow concerning the manufacturing processes by addition of matter (additive manufacturing). The Fused Deposition Modeling (FDM), also called Fused Filament Fabrication (FFF), will be the central point of this report, and other additive manufacturing processes will only be mentioned.

The report will be divided into four sections:

- Material-and-structure oriented design
- Geometrical challenges
- Considerations related to the FDM printing process
- Optimization of manufacturing time, sustainability and eco-design

The choice in materials plays a fundamental role in the additive manufacturing field. Whether it is about plastics, resins or metals, the final properties of the part depend heavily on the material used. Each one can be used with specific processes and have their own constraints. For the FDM printing, a particularly important element is the filling structure: multiple variations exist, and it is crucial that we select the correct one depending on the constraints and use of the part.

Then, we will analyze the physical challenges and other risks that one might encounter when designing the geometry of a part that will be printed. Whether we need an overhang, a bridge, or any other support-less suspended structure, we will see that the combination of gravity and a hot, molten plastic filament do not work well together.

The section relating to the FDM printing process will focus on best practices, aiming for the best result. Specifically, the printing parameters, part orientation, and the drying process. This section will also examine possible complications during printing, such as warping and dimensional accuracy.

A section on sustainability will also be included, referring to various ways of reducing the environmental impact of additive manufacturing, particularly for FDM. This will involve studying energy optimization, reducing the quantity of material used and choosing the best materials to use.

Time is a precious resource for any business. Time to market is constantly decreasing. It is therefore a good idea to look at ways of reducing manufacturing time. To do this, it will be necessary to understand the various parameters that influence FDM printing time, such as fill rate, nozzle speed, nozzle diameter, etc.

3 Material- and structure-oriented design

Rapid prototyping now makes it possible to design and test functional parts quickly. However, achieving this required overcoming many challenges, particularly those related to the materials used.

Indeed, the plastics traditionally used for injection molding did not meet the requirements of 3D printing, mainly due to differences in mechanical, thermal, and viscosity constraints. These materials, designed to be injected under high pressure into closed molds, do not adapt well to a layer-by-layer additive process.

Therefore, new formulations specifically adapted to 3D printing had to be developed. These materials include additives that improve interlayer adhesion, have lower melting temperatures, and offer greater dimensional stability during printing. Among them, PLA, PET-G, and ABS quickly became reference materials for additive manufacturing.

In this work, after an overview of the main additive manufacturing processes [34] and their associated materials, we will focus more particularly on the usage rules and characteristics of the three most commonly used thermoplastics today: PLA, PET-G, and ABS.

3.1 Material

Within the family of material extrusion processes, the most widespread technology is FDM (Fused Deposition Modeling), also known as FFF (Fused Filament Fabrication). Thermoplastics such as PLA, ABS, PET-G, Nylon (PA), TPU, or PC are melted and extruded through a heated nozzle to deposit material layer by layer. Some filament spools are reinforced with carbon or glass fibers, which improves rigidity, mechanical strength, and sometimes the dimensional stability of printed parts.

The metallic variant of FDM is called BMD (Bound Metal Deposition). In this process, metal powders (stainless steel, copper, Inconel, titanium, etc.) are bound in a polymer matrix and deposited in the same way as plastic filaments. However, the resulting parts then require debinding (removal of the polymer binder) and sintering steps to obtain a dense and functional metal part.

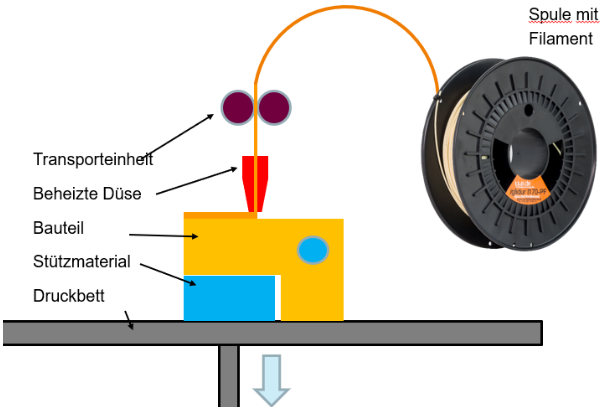


Figure 1: FDM [9]

In the family of photopolymerization processes, we distinguish SLA (Stereolithography) and DLP (Digital Light Processing). These two technologies rely on the polymerization of a photosensitive resin, activated by a laser (SLA) or a projector (DLP).

The resins, varying in color, transparency, and texture, include standard, rigid, biocompatible, and flexible versions.

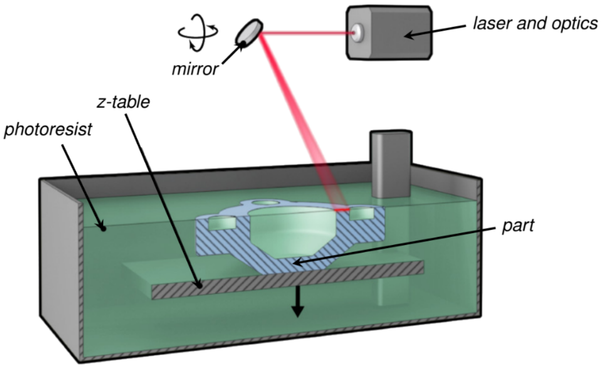


Figure 2: SLA [7]

Additive manufacturing also enables the production of metallic parts with increased geometric complexity, particularly through the Powder Bed Fusion (PBF) process, which includes technologies such as SLM (Selective Laser Melting) and EBM (Electron Beam Melting). Printable metals include stainless steel, aluminum, titanium, cobalt-chrome, and nickel-based superalloys.

Titanium is especially valued for its mechanical strength and biocompatibility, making it an ideal material for medical prostheses. Other metals, such as aluminum, stainless steel, or superalloys, are widely used in demanding sectors such as aerospace (for their lightness or heat resistance) and space applications.

This technology also applies to the printing of polymer powders via the SLS (Selective Laser Sintering) process. A key advantage of these methods is that parts generally do not require supports, as they are held in place by the unfused powder within the build chamber.

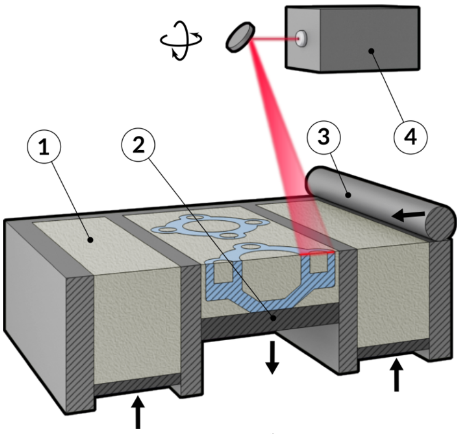


Figure 3: SLM [8]

Other methods exist, such as Binder Jetting (Fig. 4), which works on a similar principle to powder bed printing, but instead of solidifying the powder using a laser or electron beam, a liquid binder is deposited drop by drop to bind the particles together.

For metal printing, there is also DED (Directed Energy Deposition) (Fig. 5), where a metal wire or powder is fed and melted directly by a laser or electron beam, thus allowing material to be deposited layer by layer.

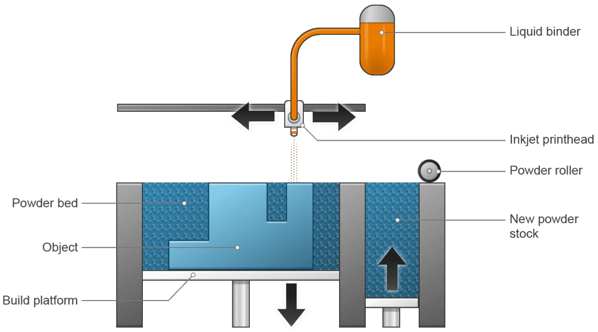


Figure 4: Binder Jetting [33]

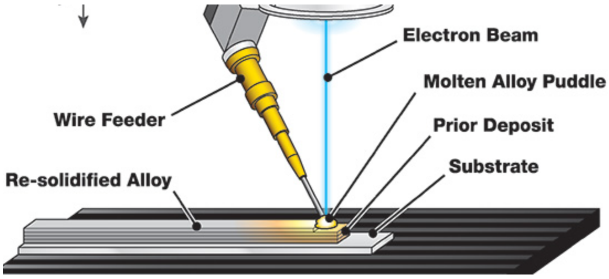


Figure 5: DED (*Directed Energy Deposition*) [31]

3.1.1 PLA ABS PETG

Now that the different possible methods and their associated materials have been reviewed, we will focus on the three most commonly used plastics in 3D printing. The report will also revolve around these three materials.

Below is a figure showing the various rules/parameters to consider when printing parts using FDM. The choice of material depends on its properties, and the diagram (Fig. 6) clearly illustrates the parameters to take into account:

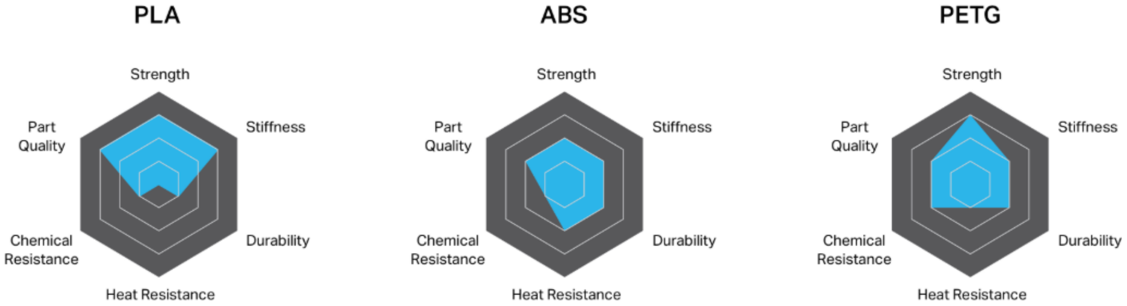


Figure 6: Comparison of PLA, ABS, and PETG [20]

The table below indicates the values of the different parameters shown in the figure above.

Material	Strength (MPa)	Stiffness (GPa)	Heat Resistance (°C)
PLA	37–70	3.5–4.0	50–60
ABS	30–50	2.0–2.7	90–100
PETG	50–75	2.0–2.5	70–85

Table 1: Comparison of filament properties [25]

3.2 3D printing basics

With the material now selected, the 3D printer settings must be properly configured. The parameters listed in the table below are taken from the PrusaSlicer software [32] and correspond to the standard baseline settings used in this study.

Parameter	**PLA (Generic)**	**ABS (Generic)**	**PETG (Generic)**
Filament-dependent settings			
Nozzle temperature (°C)	220-230	240–250	250–260
Bed temperature (°C)	50–60	100–110	85–90
Printing settings			
Layer height (mm)	0.1–0.3		
Vertical walls	refer to section 4.2.1		
Infill density – rapid prototyping (%)	15-20		
Infill density – functional part (%)	60-80		
Infill pattern	refer to section 3.3.1		
Support required	Depending on geometry		
Support type	Grid / Adjust / Organic		
Perimeter speed (mm/s)	170		
Infill speed (mm/s)	200		

Table 2: Typical 3D printing parameters for PLA, ABS, and PETG (generic values).

3.3 Infill structure

One of the major advantages of additive manufacturing lies in the ability to choose the internal structure of parts, unlike traditional machining, which offers very few options in this regard. There is a wide variety of infill structures, each offering specific performance depending on the type of stress applied. In this report, the information is drawn from the scientific study [27], focused on the behavior of parts under compressive stress. We will focus here only on the final results, the goal being to establish a simple and effective rule for quickly selecting the most suitable infill structure.

3.3.1 Infill structure for 3D printing

Cylindrical specimens, printed in Polylactic Acid (PLA) using the Fused Deposition Modeling (FDM) process and compliant with ASTM D695[39], were used to test the 14 selected infill structures (Fig. 7).

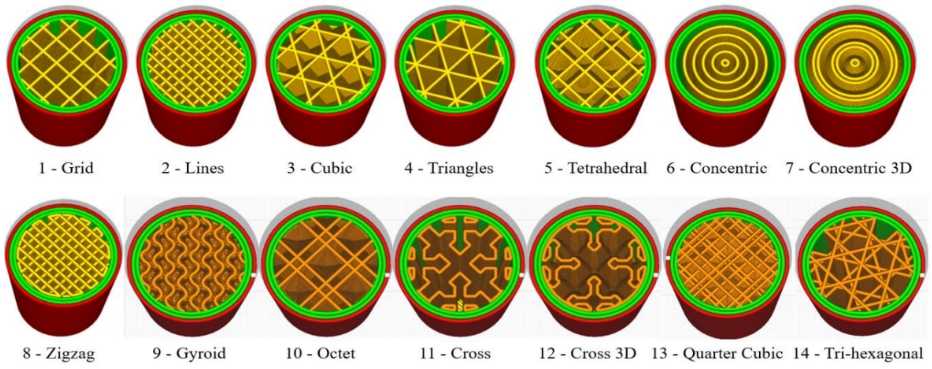


Figure 7: Internal infill structures used [27]

Figure (Fig. 8) shows the strength of each specimen relative to its weight. This graph thus makes it possible to identify the most suitable infill structure based on the desired mechanical requirements. A significant increase in strength is also observed for parts with a high infill rate, a point that will be discussed in more detail later in the report.

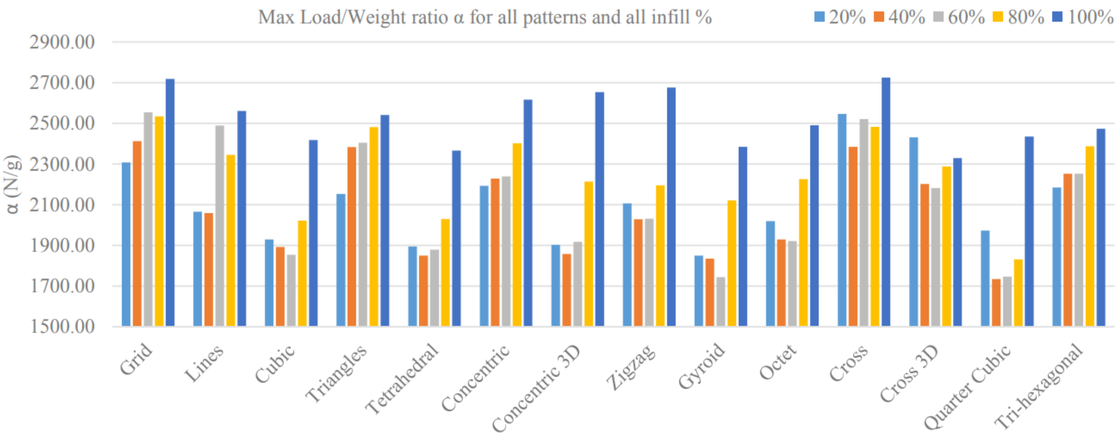


Figure 8: Max load/Weight ratio for all patterns and infill [27]

The experimental results show that compressive strength varies considerably depending on the infill structure used. The results show that 2D infill structures — such as Grid, Cross, Lines, Triangles, Concentric, Tri-hexagonal, and Zigzag — offer the best strength-to-weight ratios under compressive loads. Their efficiency can be explained by the fact that material is deposited in a single direction, allowing them to better withstand oriented stress, such as axial compression. Conversely, 3D structures use more material to distribute stresses isotropically in all three directions. They are therefore less efficient under pure compression but become more suitable when a part is subjected to multiaxial or complex loads such as torsion or bending. The study thus highlights that there is no universally optimal pattern: the choice of infill depends directly on the type of stress and the intended function of the part. If we were to provide a guideline for rapid prototyping, the following would emerge:

Type of stress	**Recommended internal structures**
Compression	Grid, Cross, Triangle, Tri-hexagonal
Torsion	Cubic, Concentric 3D, Cross 3D
Flexion / Bending	Cubic, Concentric 3D, Cross 3D
Traction / Tension	Cubic, Concentric 3D, Cross 3D

Table 3: Examples of internal structures according to stress type

3.3.2 Infill structure for powder-based processes

Powder-based processes also have internal structures, called lattices. These, similarly to FDM printing, allow the parts to be lightened while maintaining their physical properties. We will not go into detail about these structures, but an illustrative example is shown in Figure 9.

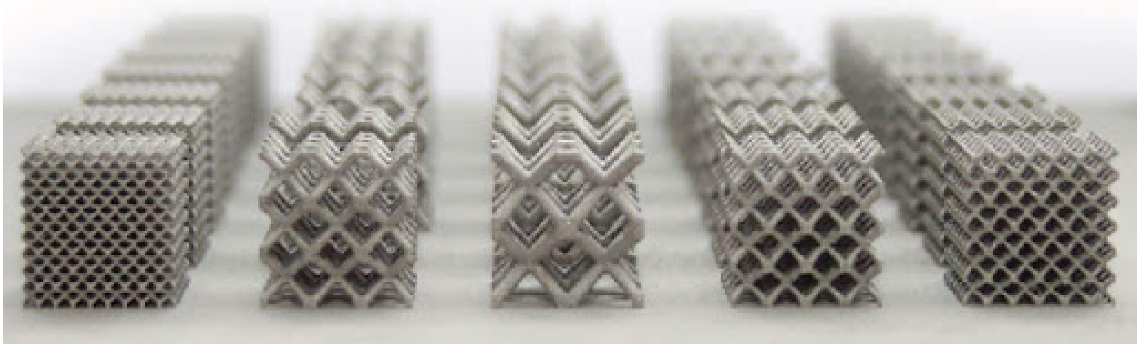


Figure 9: Lattice example [28]

4 Geometrical challenges

Additive manufacturing processes are governed by many fundamental geometrical rules. These rules are not dictated by some regulations, but by physics themselves and must be taken into account during the design of a part. If not respected, the part may present weaknesses that can alter the integrity and safety of the complete structure, or even make the whole part collapse, either during printing or when in use. These rules also remind us that we cannot always print any geometry we want if we need a certain level of precision.

When designing a part that will be printed layer by layer, we must take into account multiple factors such as gravity, that may alter the quality of the final product. With a focus on FDM with the materials studied beforehand (PLA, PETG, and ABS), we will see some specifications that can prevent a part from having defects, such as an unstable wall, or can even prevent the complete destruction of the part during printing.

In this section, we will first analyze the design rules for suspended shapes such as overhangs, cantilevers, or bridges. Then, we will specify the minimal design geometries for walls, holes, and internal channels.

4.1 Suspended geometries

The working principle of FDM is very simple: a layer-by-layer process deposits the $n+1$th layer on the n th. However, some geometries such as bridges or overhangs require the most recent layers to be placed over areas where no filament was placed. As supports can add up to 40% weight (Fig. 10.2) to a suspended structure [29] and greatly reduce the surface quality, we will analyze these geometries without any.

The challenge with these geometries will be to avoid dropping, which happens when the filament is not solidified enough to withstand its own gravity or the weight of the layer deposited above it, and starts dropping/sagging. This results in a very poor surface quality (Fig. 10.1) and maybe a drop in mechanical properties.

4.1.1 Overhangs

Overhangs are needed on the underside of some slopes [13], where we need the $n+1$th layer to cover a little more area than the previous layer. This extra distance must be very short, as we can not deposit a layer on nothing and expect it to magically stay up in the air. The hot material, still molten, will get pulled down by gravity and will completely fall off the structure if the angle is too extreme.

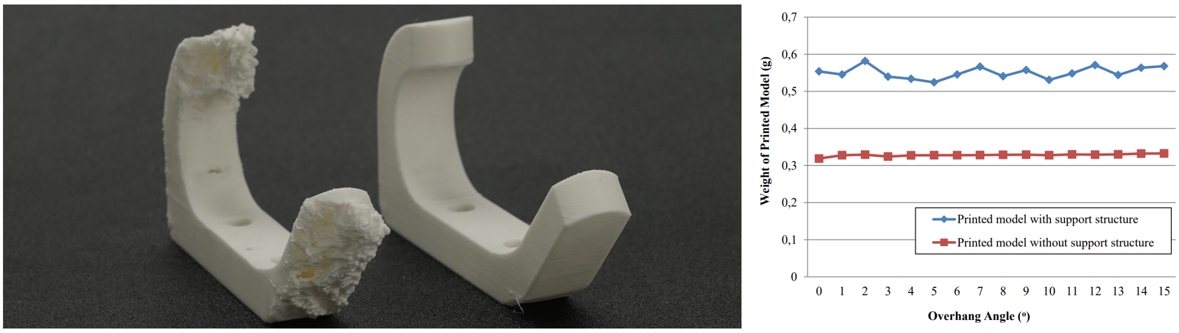


Figure 10: Supports can help preventing the collapse of a part during printing, but waste a lot of materials [4] [29].

The general guideline for overhangs states that **any design with a slope under** 45° **requires supports**. This rule tends to underestimate the capabilities of the most recent printers, that can achieve better slopes with the right settings. The standard values used in many printers can be found in 3.2, but modifying them using some of the tips below will improve the underside surface (Fig. 10.1) quality of small-sloped overhangs, while avoiding as much as possible the need for supports [4]:

1. **Lower nozzle temperature.**

   The main problem when printing overhangs is the molten state of the material coming out of the nozzle. Reducing its temperature means the filament will come out colder, and will solidify quicker, reducing dropping. Here is a list of recommended nozzle temperatures for diverse materials to improve overhangs quality:

   - PLA: 190-220°C instead of 220-230°C
   - PETG: 220-240°C instead of 240-250°C
   - ABS: 230-250°C instead of 250-260°C

2. **Increased cooling.**

   A better cooling also means that the filament will solidify quicker, leaving it molten for a smaller amount of time and reducing dropping (Fig. 11). That being said, we must be careful not to use a fan too strong as the layers might start to delaminate and not stick well to the underneath ones.

3. **Reduced printing speed.**

   Once again, the goal is to help the filament cool faster. By reducing the printing speed (it is recommended to reduce it 10mm/s at a time to observe the printer's capacity), the filament has more time to cool before even being deposited on the previous layer, and has the time to solidify before getting the next layer deposited on it. This solution is very effective as it has the same benefits as the two previous ones, without the risks of not sticking to the previous layers or delaminating. Printing multiple parts at once is also a very good solution, as it gives each layer more time to cool down.

   It is important to note that in some cases, like bridges for example, increasing the printing speed might be a good idea as it can stretch the filament and add tension, which may help the part keep its shape.

Figure 11: The same part printed with (top) and without effective cooling (bottom) shows a great difference in print quality [13].

4.1.2 Bridges

Printing overhangs already represent a hard challenge, but they have the advantage of, at least partially, depositing the new layer on top of the previous one. The objective when designing a bridge is to get the first layers to adhere to each other, without letting any strand of filament let loose. On the other hand, what we want to avoid when "bridging" is getting loose or sagging filament (Fig. 12.2), layers that do not complete the bridge, or even the complete absence of the bridge in the final result.

The main solutions to avoid a bad bridging are exactly the same as for the overhangs: the more cooling there is, the better. We can also improve the quality of the bridge by tuning the extrusion multiplier (Fig. 12.1): if this multiplier is too high, the printer might push too much filament out and push on the structure, helping the sagging of the bridge.

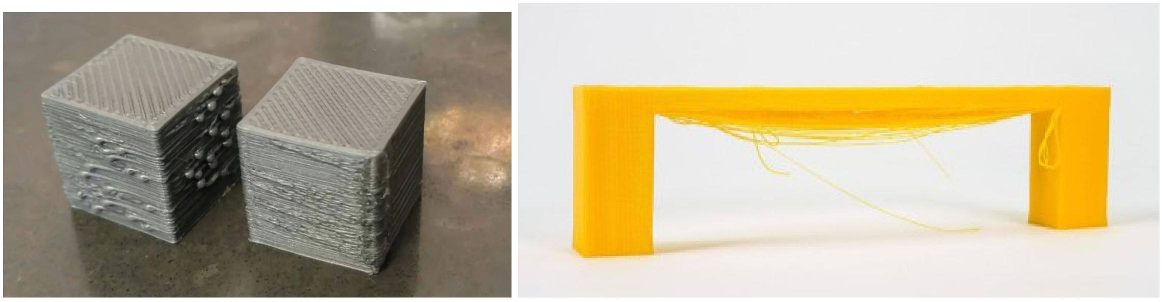


Figure 12: Extreme case of an extrusion multiplier too big [16] and a sagging bridge [30].

Finally, it is highly recommended to keep the suspended part of bridges under 50-100 millimeters long [35]. Using the tips provided may help the structure not fall apart when the bridge gets longer, but this 50-100 millimeter limit is an excellent starting point. Cantilevers are an even worse version (in terms of FDM geometry challenges) than bridges, and there exists no real solution to build them apart from using supports.

4.2 Minimal design geometries

4.2.1 Minimum wall thickness

When designing a part, we want to reduce the quantity of filament used. This way, we can reduce the cost of the print, as well as the printing time. To achieve this objective, one of the most common techniques used is the partial filling of the structure, and the reduction of the wall thickness. However, we cannot reduce this last parameter too much, as it may result in a great weakening of the structure, which can lead the structure to collapse (Fig. 13) or diminish its strength, as well as reducing the printing precision.

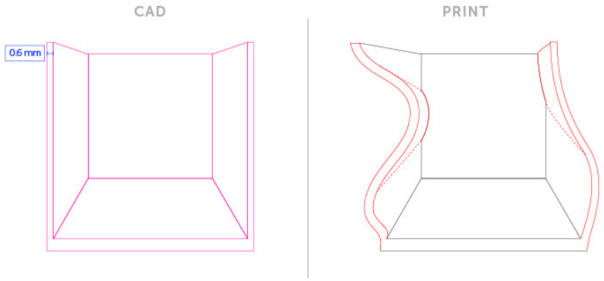


Figure 13: A thin-walled design might collapse during the print if the wall cannot resist its own weight [5].

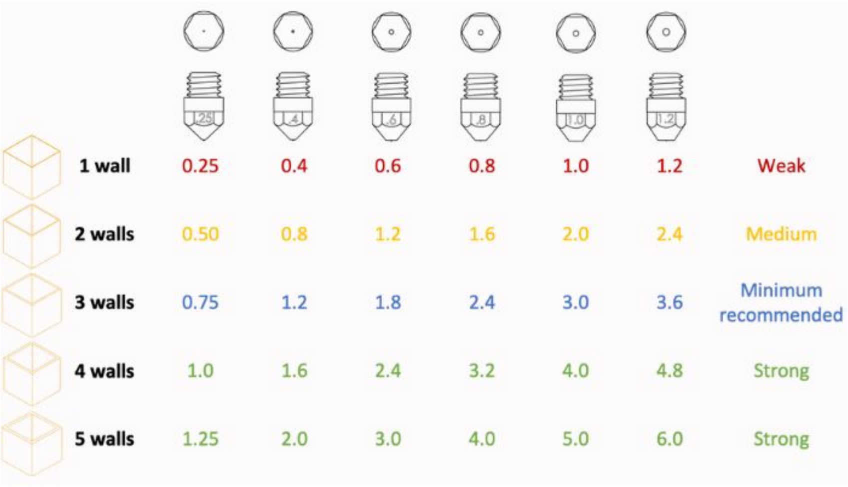
Determining an optimal minimum wall thickness will be a complex task, and might require some prototyping [5]. That being said, we should keep in mind that this thickness is mostly tied to the material, rather than to the printing process, and we can sort the recommended and absolute minimal thicknesses by material this way (Tab. 4):

Minimum wall thickness recommendation per material						
Material	PLA	ABS	Nylon	Verowhite	Transparent	ABS-like
Recommended minimum thickness value [mm]	1.5	1.5	1.5	1.0	1.0	1.0
Absolute minimum thickness value [mm]	0.8	0.8	0.8	0.6	0.6	0.6

Table 4: The minimum wall thickness depends heavily on the material used for printing the part [5].

In practice, even though the absolute limit with PLA is around 0.8 millimeters, most designs do not actually need such thin geometry and should use at least 1.5 millimeters. For rubber-like materials like TPU, it is recommended to use a higher wall thickness, around 2.0 millimeters. However, some parts that use these materials need to be flexible and must not have a wall thickness too high, as it will make the part bulky.

The other way of determining the minimal wall thickness for a design is directly linked to the hardware specifications of the printer itself, especially the nozzle diameter [26]. We can choose the wall thickness as a multiple of the nozzle diameter. It is strongly recommended to use a thickness of at least 3 times the diameter.



1 wall	0.25	0.4	0.6	0.8	1.0	1.2	Weak
2 walls	0.50	0.8	1.2	1.6	2.0	2.4	Medium
3 walls	0.75	1.2	1.8	2.4	3.0	3.6	Minimum recommended
4 walls	1.0	1.6	2.4	3.2	4.0	4.8	Strong
5 walls	1.25	2.0	3.0	4.0	5.0	6.0	Strong

Figure 14: The minimum wall thickness can be calculated as a multiple of the printer's nozzle diameter [26].

Most nozzles have a diameter of 0.4 mm, and we should only use the 0.8 mm wall thickness, corresponding to 2 times the diameter, in extreme cases. A good reflex to have is to keep the thickness-nozzle diameter multiple as a whole number (Fig. 14): using a decimal multiplier does not make much sense and will force the printer to overlap itself, slowing down the whole printing process.

4.2.2 Holes, channels and gaps

Most, if not all 3D printed parts are designed in a way that allows them to be attached to a larger structure, or to allow some kind of flow through them. To achieve these objectives, the structures can contain holes, gaps, or channels. Avoiding the use of supports will once again be crucial, whether that is because accessing them for removal can be physically impossible - as in a deep or curved channel for example - or because we want to ensure a proper surface quality. In order to avoid any issue with the printing process of a structure containing these specifications, we can apply some of these recommendations:

1. **Holes:**

   The holes in 3D printed structures can easily present a bad surface quality and a bad degree of precision, due to the suspended nature of their top half.

   If a part needs a hole to have a precise diameter, it is recommended to design a bigger hole than necessary and use a joint that we will fit inside said hole, which will always show more precision in the inside diameter. This also applies to any kind of threaded hole.

2. **Channels:**

   Internal channels (Fig. 15.2) might be used to transfer a flow through the part. If the user cannot access the entirety of the channels inside structures in order to remove any eventual supports, the part must be designed in a way that avoids them. This can for example mean using as much as possible non-round channels, to ensure that their top half does not risk dropping.

3. **Gaps:**

   In most designs, there is no recommendation on the gap size between two walls of the structure. However, if the build needs to have a good, clean surface quality, or if it might need some sanding, keeping an opening of at least 1 millimeter proved its worth in practice.

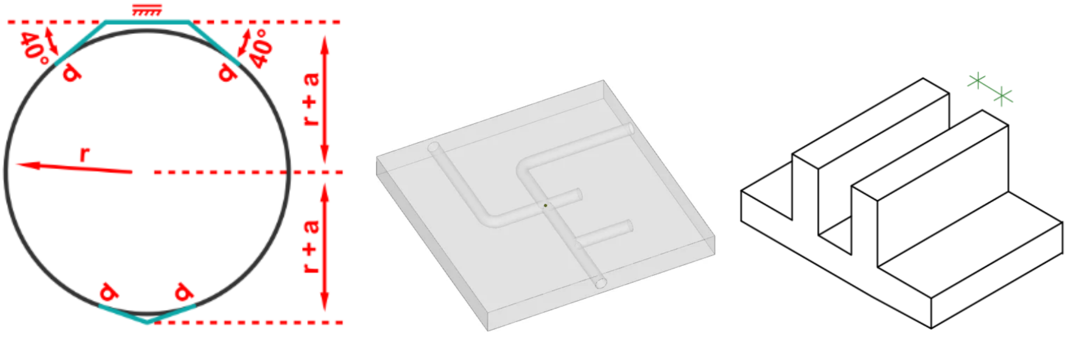


Figure 15: Very small holes, channels and gaps [37] must respect some minimal design rules to achieve a good level of precision [36].

4.2.3 Text, logos and characters

Adding text on a part, either by embossing or engraving, should not represent any threat to the structural integrity of the structure, even if the printing process does not work exactly as intended. That being said, there are still some recommendations to ensure that these details are visible and in good shape.

- For details on a vertical surface (Fig. 16.1):
  1. **Embossing**: The line width should be at least twice the filament width for visibility. The extrusion is recommended to not go further than two layers, to avoid sagging.

2. **Engraving**: There is absolutely no minimal limit on the width of the engraving details, but the maximal limit will be determined by the theory in section 4.1. There is no limit for the depth of the engraved surface on a vertical surface either.

- For details on a horizontal surface (Fig. 16.2):
  1. **Embossing**: The width of the details will be linked to their height, as we want to avoid the collapse of the structure. The height of the embossings has no minimal limit, but it should almost always be over 2 millimeters, to ensure that it is visible, especially on single-colored parts.
  2. **Engraving**: As these details cannot ever put the structure at risk during or after the printing process, there is no recommendation for their size.

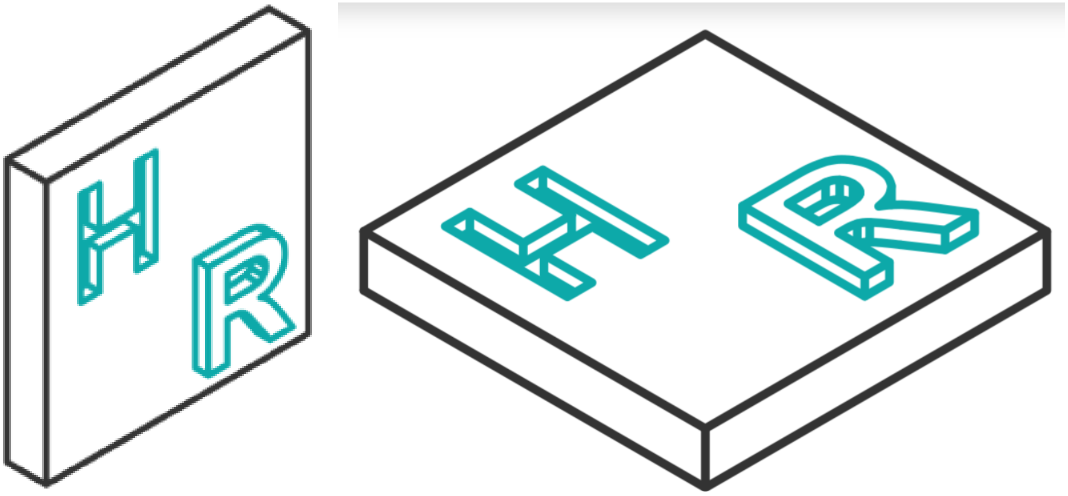


Figure 16: Even on small details of the structure [36], the designs must watch out for dropping issues.

5 Considerations related to the FDM printing process

To achieve a consistent high quality part using the FDM printing process requires extensive testing. This is due to many factors that affect the final result, such as ambient temperature, humidity, etc. Every material has different properties and weakness, such as exposure to humidity, heat or UV-radiation. This section will be discussing various considerations to be had when using FDM printing.

As one of the most popular printing processes for hobbyists and professional prototyping alike, FDM printing has multiple requirements for an optimal part. This results from the use of thermoplastic extrusion through nozzles, a relatively large process when compared to other processes such as SLA, SLS or other laser based additive manufacturing. This has a range of impacts, notably the sensibility to cooling time. Due to this large extruded filament, the layers are also a weak point and thus need to be taken into consideration.

5.1 Orientation of the part

FDM printing machines fabricate parts by following a computer provided path, which results in strands on the part. These strands have a significant influence on the resulting strength of the printed part. This is true both horizontally around the z-axis, known as raster angle and vertically.

The strands inline (0°) with the force are much stronger because the main failure point is the "weld" between the strands. This results in a anisotropic part which loses up to 50% of-axis (90°) due to this effect.[22] The importance of the raster orientation of printing can as such not be neglected, especially in higher force application. This can be compensated for in much the same way as in composites fabrication, by using multiple layers in different raster directions. The following picture shows the failure direction in dog-bone samples. It can be noticed that the failure point follows the welds between layers, illustrating the weakness.

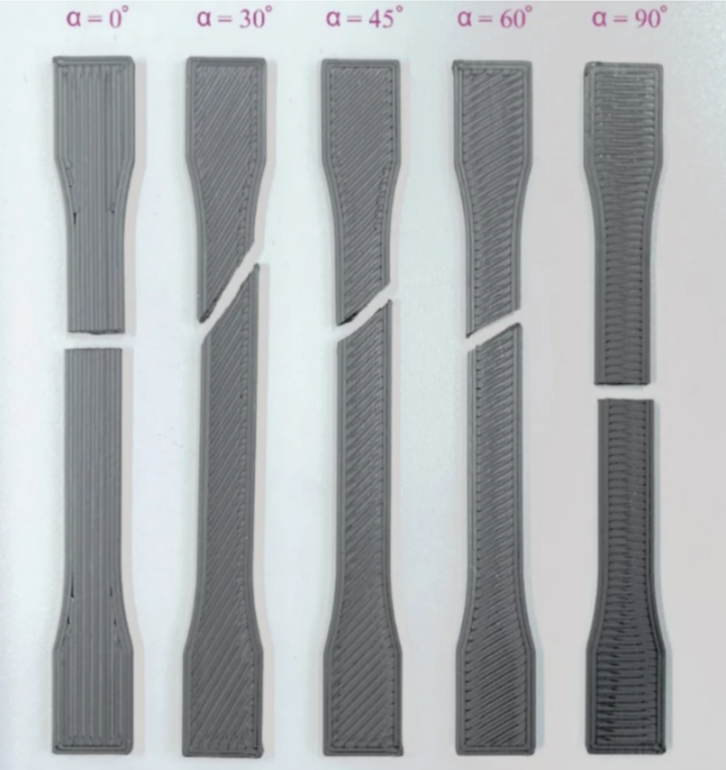
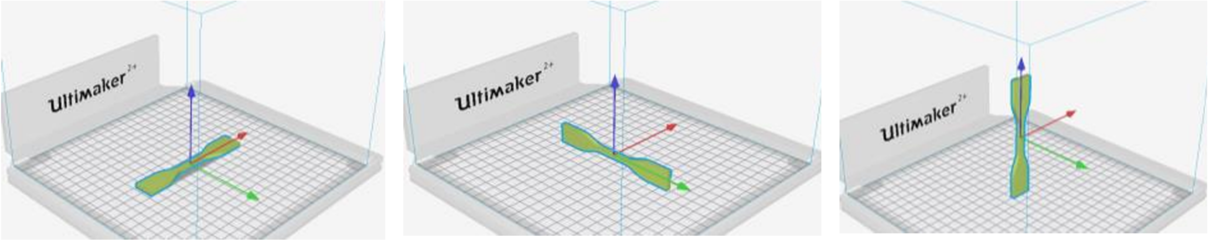


Figure 17: Effect of raster direction on failure angle [22].

There is also the vertical component, or printing direction to take into account for the same anisotropic reasons as mentioned above. By moving up a layer, the extruded filament produces another layer line, leaving each deposited filament strand up to 4 welds. Each angle represents a different orientation in 3D space, as shown by the following figure.



(a) (b) (c)

Figure 18: Different printing angle: (a) XY-axis (b) YZ-axis (C) ZX-axis [23].

After tensile testing, [23], it was found that the YZ-axis is the highest tensile strength. This is due to it having more printed layers parallel to the force. The same could be said for the XY-axis, but the lower vertical layer thickness that characterizes the YZ-axis produces a stronger part. The ZX-axis is much weaker, due to perpendicular direction of the layers, as discussed above. The following figure represents the tensile characterization of each printing direction.

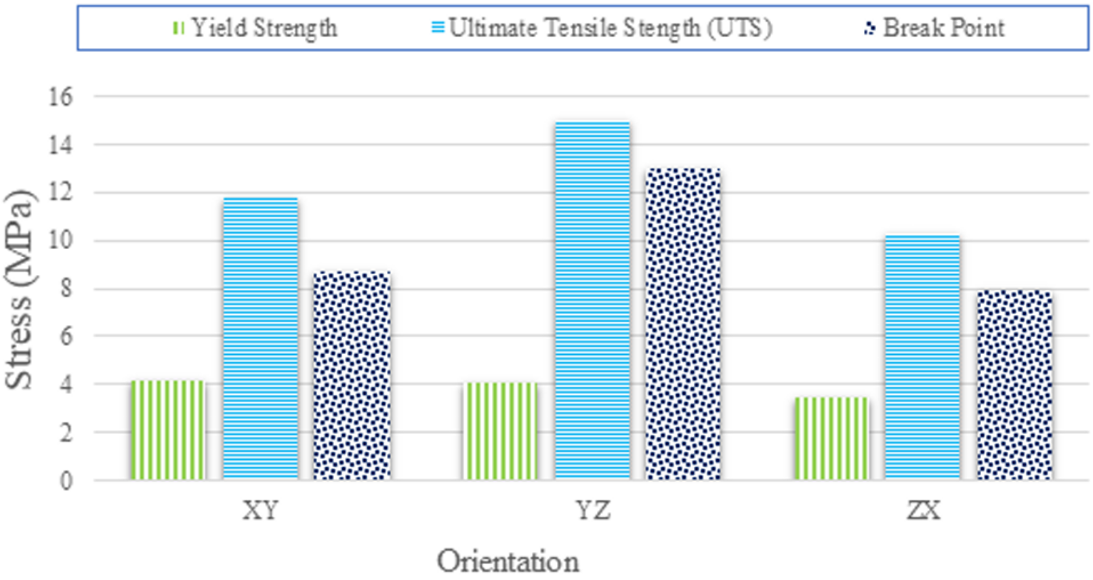


Figure 19: Tensile strength for each direction [23].

5.2 Printing speed and temperature

Printing speed is a parameter expressed in mm/sec and represents the speed at which the nozzle extrudes and layers filament on the build bed. Usually, a faster printing speed correspond to a quicker print time, reducing production cost while lowering the final quality of the piece due to the faster speed lowering stability. This causes causes small internal defects and external visual blemishes. Depending on the material, this may also have an effect of the final strength of the part, lowering the mechanical properties.

The temperature at which the filament is extruded depends on the material used, as for example PETG is printed at a higher temperature than PLA. The temperature parameter also has a important influence on the final quality of the part, as it dictates the fluidity and viscosity of the polymer while bonding to previous layers.

To illustrate how variable these parameters are, a comparison of PLA and PETG shows that the reaction to changing printing speed and temperature is not equivalent. For both filament, an increase in respective printing temperature corresponds to a lower viscosity. This increases the inter-layer fusion and porosity decreases which results in improved tensile properties. However, both materials react in opposite way to the printing speed. PLA shows higher tensile properties when printed at higher speeds, as the heat dissipates more quickly and lower the filling rate, improving bonding. On the opposite, PETG shows better properties when printed at a lower speed, which correspond the a slower heat dissipation thus lowering porosity [15]. The following graphs show these results.

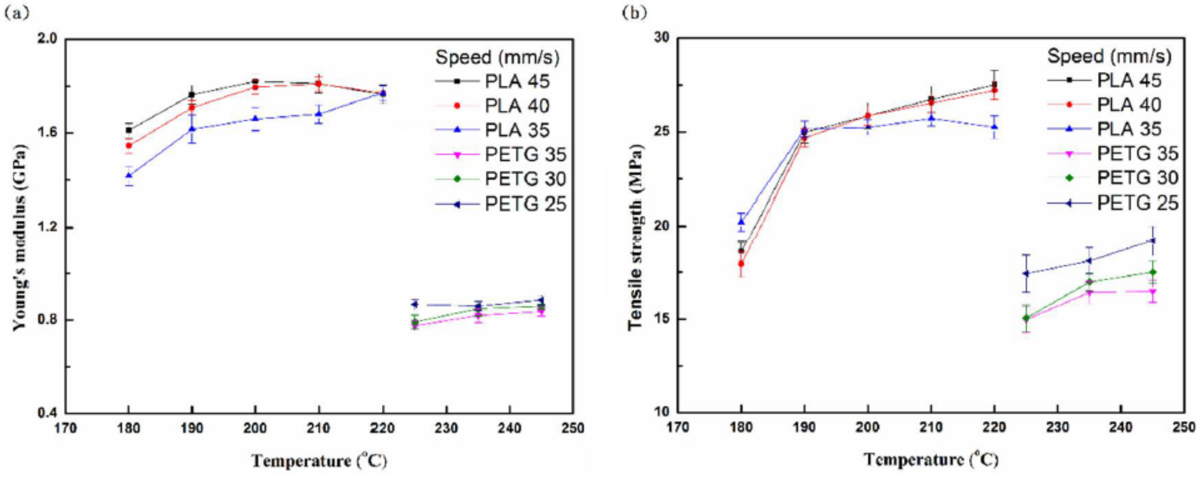


Figure 20: Effect of printing speed and temperature on material properties [15].

5.3 Warping

Warping deformation caused by thermal contraction while cooling is a recurring difficulty with the use of FDM printing. There is no one-size-fits all solution to this challenge, however it is possible to reduce this effect by carefully choosing the printing parameters[2]. This is done either by experimentation for new materials, although manufacturers give recommended parameters as a good starting point, commonly presented in the manufacturers proprietary slicing software. These parameters include all of the aforementioned considerations.

This procedure can be applied, as demonstrated by [17], whose research focused on ABS. This article found that the contact area of the part with the hot bed plays an important role in limiting the overall warping. For small parts the deformation is not significant, the warping becoming increasingly of an issue until reaching a point after which the relatively large contact patch provided sufficient adherence to limit warping. They also noted that the overall parameters such as nozzle and bed temperature are factors in the overall deformation.

5.4 Dimensional accuracy

The dimensional accuracy of FDM printing depends on a variety of factors, ranging from the machine precision to the specific filament batch used. There is also the importance of printing parameters, explored previously in this section. All of these aspects combine to define the overall precision and resolution of the final part. As such, the final CAD to part fidelity is lower than traditional machining, or other additive manufacturing techniques such as SLA printing.

5.4.1 Machine accuracy

In general, the final precision of a part will depend on the tools used, as they directly impact the final part. For FDM printing, the nozzle is the biggest factor relating to part resolution. Compared to a laser working on the micro scale, a very precise FDM nozzle only has a 0.1mm diameter. This means that no feature smaller than this can be printed, limiting the dimensional accuracy. The motors or servos and micro controller used by the machine also determine the final accuracy in the same manner as CAM. Loss of precision can occur due to backlash or belt tension, depending on the exact configuration used by the printer. This means that there must be calibration steps to achieve maximum precision, while still taking into account the error margin linked to specific components. [11]

5.4.2 Printing parameters

To achieve a high relative accuracy, not only the machine precision but crucially, the control of key parameters. These parameters are: printing speed and layer height. Printing speed directly affects the extrusion flow of the filament and dictates the total cooling time before the next layer is applied. A higher speed correlates to a higher dimensional deviation for PLA, according to [1]. Lowering the layer height improves dimensional accuracy, as this reduces the cross-sectional area of each layer, making the part more uniform, limiting distortion due to varying mechanical properties.

5.5 Drying the filament

Every filament has a tendency to absorb humidity and becoming damp. This results in the moisture vaporizing during the extrusion process and causing multiple issues to the molten filament [6]:

- Physical expansion
- Increased fluidity
- Formation of air holes

These defects in the extruded filament can impact the final part in a range of ways:

- Stringing
- Lack of materials/holes
- Rough surface
- Reduced strength

The severity of the resulting issues depend on the total moisture absorbed by the filament. It can as such go from minimal visual damage to lowering the tensile profile of the final part. This has major implications in commercial use, as ensuring quality is an important aspect. The faulty parts can result in looking similarly to the following image:

Figure 21: Effect of humidity of part quality [6].

To prevent these issues, it is recommended to dry the filament before printing. This is done according to the manufactures recommendation, which indicates the temperature and the drying time. These parameters vary a lot depending on the material and even within the same material class, differences exist. For example, basic PLA requires around 8 hours of drying at 50°C in a forced air oven, compared to 8 hours at 75-85°C for ABS. [6]

To protect the filament after the drying process and during the printing, using a desiccant such as silica gel in a hermetic box is recommended. This box should only leave a small hole for the filament to pass through, assuring the best print quality. The following image shows such a box:

Figure 22: Protecting the filament from humidity [6].

Applying these steps assures a higher part quality and a more repeatable print. This consistency is most important for commercial production, where large part numbers are to be produced.

To conclude this section, the performance and reliability of FDM-printed parts is strongly linked to process parameters. Orientation has the greatest influence on mechanical strength due to the anisotropic nature of layer bonding, while printing temperature and speed determine inter layer adhesion and surface quality. Warping and dimensional deviations arise primarily from thermal contraction and machine limitations but can be mitigated through proper calibration and optimized settings. Finally, maintaining filament dryness is critical to ensure consistent extrusion and prevent structural defects. Considering these parameters allows for reproducible, high-quality prints for both hobbyist and industrial application.

6 Optimization of manufacturing time, sustainability and eco-design

6.1 Sustainability and eco-design

Eco-design and sustainability are issues that are becoming increasingly important. It is therefore worth considering the environmental aspects of additive manufacturing. We will mainly discuss the FDM process.

6.1.1 Energy

The manufacturing of a part using FDM involves three main phases:

- Heating : this includes the nozzle and sometimes the tray and build chamber.
- Printing : the phase during which the part is actually generated
- Finishing : cooling of hot components by fans

Figure 23 illustrates these three phases and their respective powers over time.

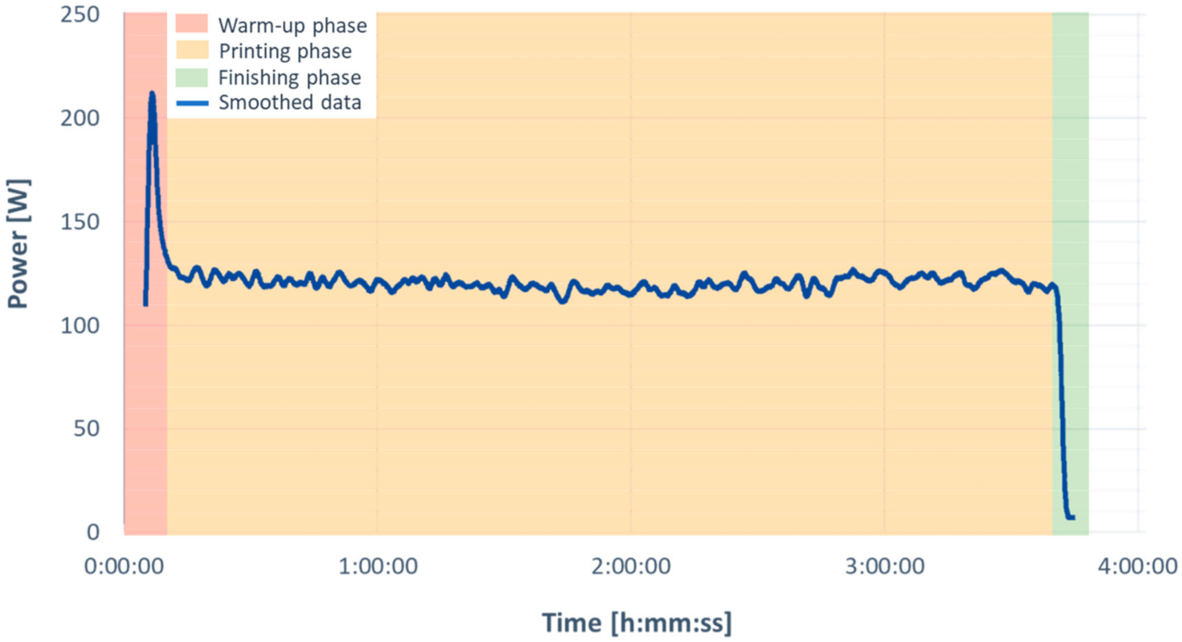


Figure 23: Smoothed power profile of a MEX 3D printer, edited from [3] et [14]

In general, heating is the phase that consumes the most power, but only lasts a few minutes. The printing phase is then the most power-intensive. It is also the longest of all (usually several hours) and the one that consumes the most energy. Finally, the finishing phase only lasts a few minutes and does not consume much energy. The second step is therefore the one on which we must focus most.

To reduce energy consumption during printing, it is possible to add : [12]

- a hot-end insulation : - 33.8–30.63 %
- a printer enclosures - 15.65–18.16 %
- a heated bed insulation : - 5-10 %

By combining these three optimizations, it is possible to reduce average energy consumption by 29-38%.

It is possible to reduce consumption during the heating phase by printing several parts in a single print job. In absolute terms, the amount of energy required will not change, but it will decrease when compared to the number of parts.

An effective way to reduce the overall energy consumption of an FDM printer is to use less material. The energy consumption required is 50–55 kWh/kg for PLA, 70–80 kWh/kg for PETG, and 60–70 kWh/kg for ABS [24].

6.1.2 Matter quantity

It is possible to reduce this material consumption by adjusting the filling rate. Figure 24 shows the amount of material used based on the fill rate to print a cube with a 20 mm edge.

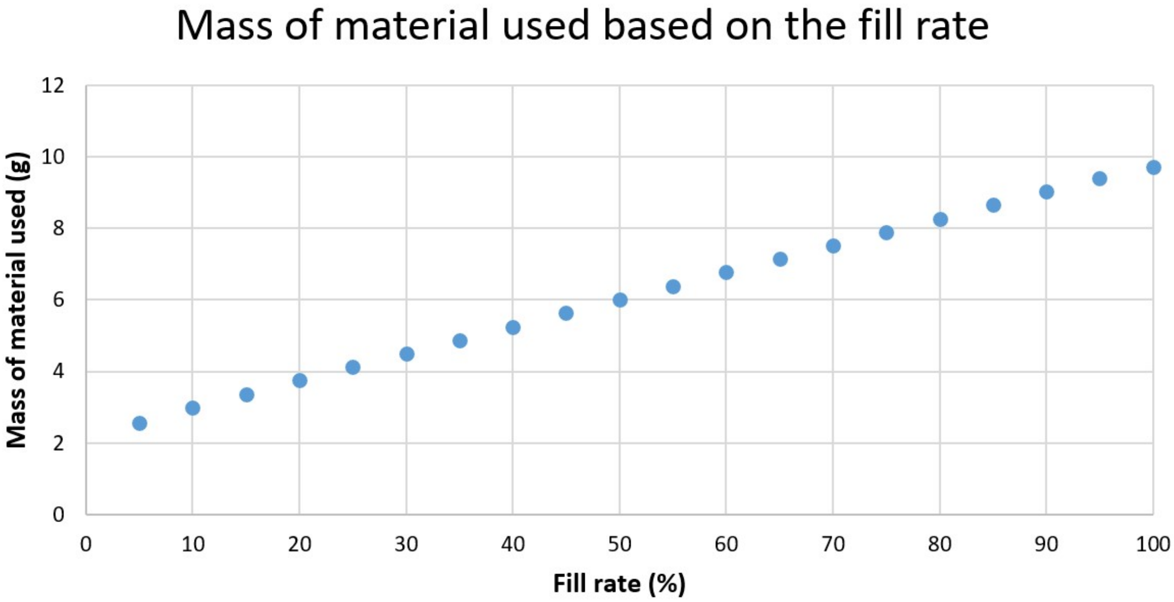


Figure 24: Mass of material used to print a cube with 20-millimeter edges using a nozzle with a diameter of 0.25 mm, depending on the fill rate, according to PrusaSlicer

To reduce the amount of material consumed, it is possible to reduce the amount of support used [12]. To do this, it is possible to reduce the thickness of the wire supports and change the orientation of the workpiece. Figure 28 illustrates the influence of orientation on the amount of support used. The mass of material used is 4.97 grams without support and 5.63, 6.03, and 10.73 grams for the three images, respectively. It should be noted that although the second orientation will not produce optimal print quality, it requires more than five times less support than the third (see subsection 4.1).

6.1.3 Materials

The environmental impact depends on the materials used. The polymers used for FDM are generally thermoplastics (PLA, ABS, PEEK, etc.). These materials are polymers that connect to each other by weak bonds. Above a certain temperature, they melt, allowing them to be recycled.
Sometimes, in more specific FDM applications, such as the use of flexible materials, it is possible to use

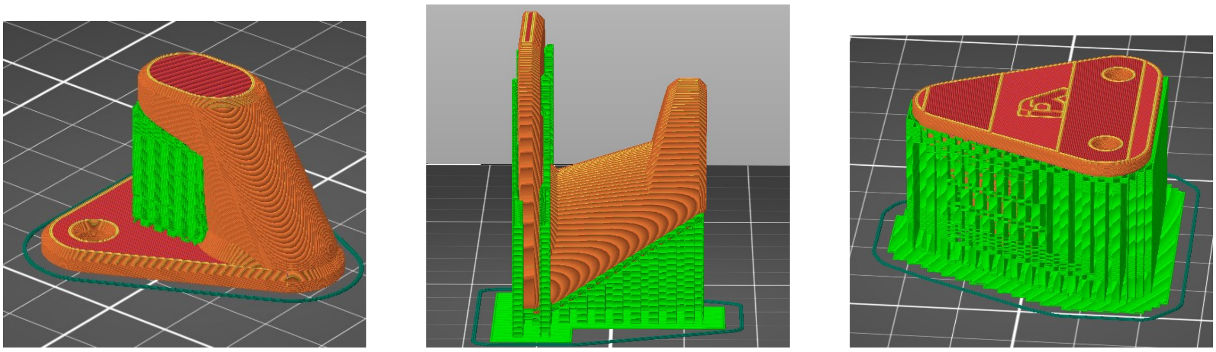
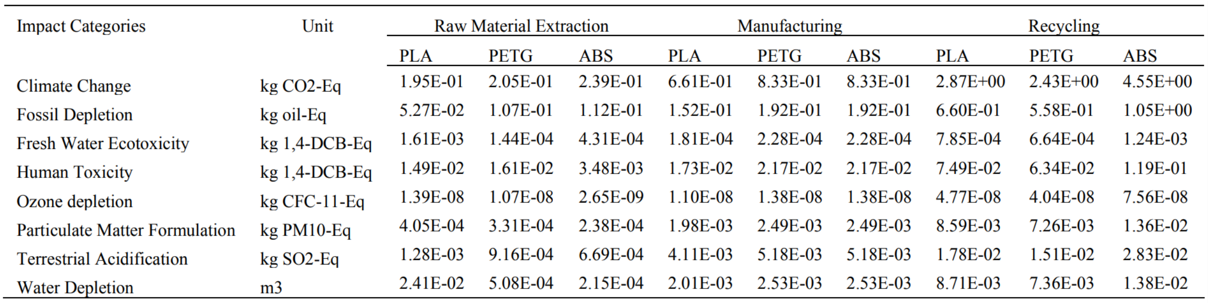


Figure 25: Good orientation

Figure 26: Poorer orientation

Figure 27: Bad orientation

Figure 28: Illustration of the effect of different orientations on the amount of support used



Impact Categories	Unit	Raw Material Extraction			Manufacturing			Recycling		
		PLA	PETG	ABS	PLA	PETG	ABS	PLA	PETG	ABS
Climate Change	kg CO2-Eq	1.95E-01	2.05E-01	2.39E-01	6.61E-01	8.33E-01	8.33E-01	2.87E+00	2.43E+00	4.55E+00
Fossil Depletion	kg oil-Eq	5.27E-02	1.07E-01	1.12E-01	1.52E-01	1.92E-01	1.92E-01	6.60E-01	5.58E-01	1.05E+00
Fresh Water Ecotoxicity	kg 1,4-DCB-Eq	1.61E-03	1.44E-04	4.31E-04	1.81E-04	2.28E-04	2.28E-04	7.85E-04	6.64E-04	1.24E-03
Human Toxicity	kg 1,4-DCB-Eq	1.49E-02	1.61E-02	3.48E-03	1.73E-02	2.17E-02	2.17E-02	7.49E-02	6.34E-02	1.19E-01
Ozone depletion	kg CFC-11-Eq	1.39E-08	1.07E-08	2.65E-09	1.10E-08	1.38E-08	1.38E-08	4.77E-08	4.04E-08	7.56E-08
Particulate Matter Formulation	kg PM10-Eq	4.05E-04	3.31E-04	2.38E-04	1.98E-03	2.49E-03	2.49E-03	8.59E-03	7.26E-03	1.36E-02
Terrestrial Acidification	kg SO2-Eq	1.28E-03	9.16E-04	6.69E-04	4.11E-03	5.18E-03	5.18E-03	1.78E-02	1.51E-02	2.83E-02
Water Depletion	m3	2.41E-02	5.08E-04	2.15E-04	2.01E-03	2.53E-03	2.53E-03	8.71E-03	7.36E-03	1.38E-02

Figure 29: Environmental impacts of PLA, ABS, and PETG [18] [3]

thermoplastic elastomers (TPE) such as TPE-s, TPE-U, TPE-E, etc. These materials combine the properties of elastomers and thermoplastics [38]. In particular, they offer the advantage over conventional elastomers of being meltable and therefore recyclable.

Processes using resin (thermoset or elastomer before curing), such as stereolithography, are not recyclable and should not be preferred except for specific applications.

Let's focus on three classic 3D printing materials: PLA, PETG, and ABS. Comparing their environmental impacts is complex and depends on the criteria chosen (climate change, freshwater ecotoxicity, human toxicity, ozone depletion, etc.) and the life cycle stage (raw material extraction, manufacturing, or recycling). Recycling is the phase of the life cycle that generally has the greatest impact on the environment. ABS has the worst overall environmental impact, mainly due to its high specific heat capacity (Figure 29), followed by PLA. PETG is generally the least polluting of the three. It is therefore the preferred choice, even though it is slightly more expensive. (24-34 $/kg for PETG versus 20-30 $/kg for PLA and ABS [19]).

6.2 Optimization of manufacturing time

Product development and time-to-market are becoming increasingly shorter and more competitive. In this sense, it is beneficial to reduce manufacturing time. This chapter will discuss ways to reduce FDM manufacturing time.

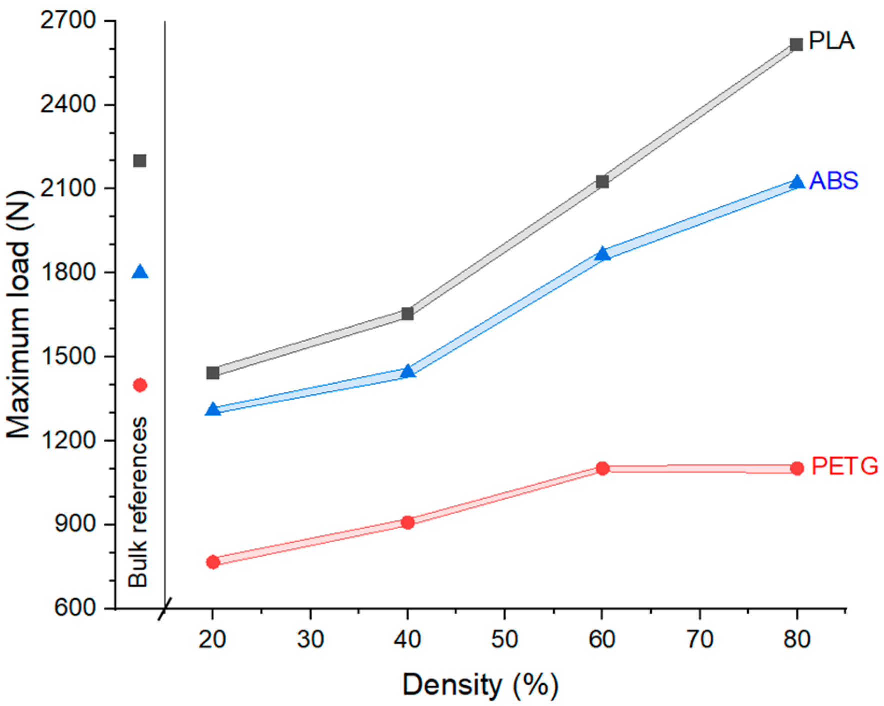


Figure 30: Effect of the fill rate on the samples' mechanical strength [21]

6.2.1 fill rate

The first interesting idea is to reduce the fill rate; if the print nozzle deposits less material, it will perform its task more quickly. However, it is important to be aware that reducing the fill rate will slightly alter the mechanical properties of the part, as shown in Figure 30. Figure 31 allows you to visualize the effect of the fill rate on printing time.

6.2.2 Nozzle speed

Another simple way to reduce printing time is to increase the nozzle speed. The maximum printing speed ranges from 50 mm/s for the slowest printers to 1000 mm/s for the fastest (such as the FLSUN T1). It is also possible to slow down the speed. Printing time will generally decrease as printing speed increases, but heating and cooling times will not decrease. Increasing the speed generally results in a decrease in quality (precision) and mechanical strength (Figure 32). It is also worth noting that increasing the printing speed generally involves increasing the nozzle temperature. This is because as the flow of material through the nozzle increases, the heat transfer must also increase, requiring a rise in the nozzle temperature.

6.2.3 Nozzle diameter and layer thickness

Increasing the nozzle diameter also reduces printing time. Increasing the thickness of each layer also reduces printing time. In principle, it is recommended that [10] :

$$layer\ height < 0.75 \cdot diameter\ of\ the\ nozzle \tag{1}$$

Ideally :

$$layer\ height = 0.5 \cdot diameter\ of\ the\ nozzle \tag{2}$$

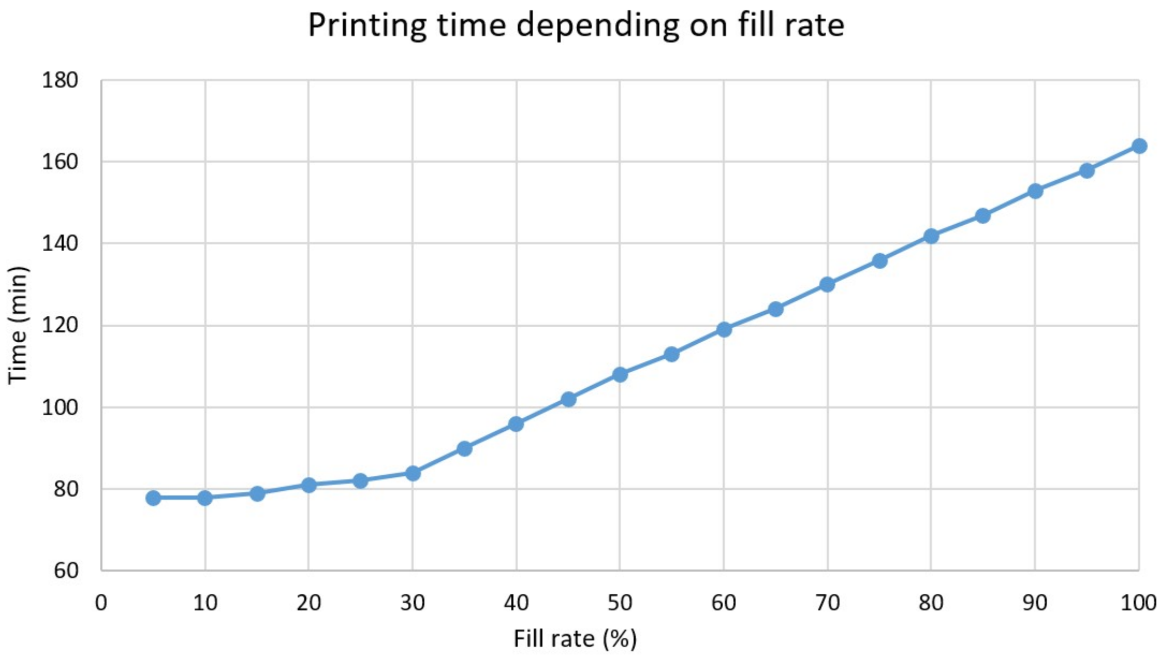


Figure 31: Printing time for a cube with 20 mm edges using the Prusa i3 MK3 model with a 0.25 mm diameter nozzle, depending on the fill rate according to PrusaSlicer

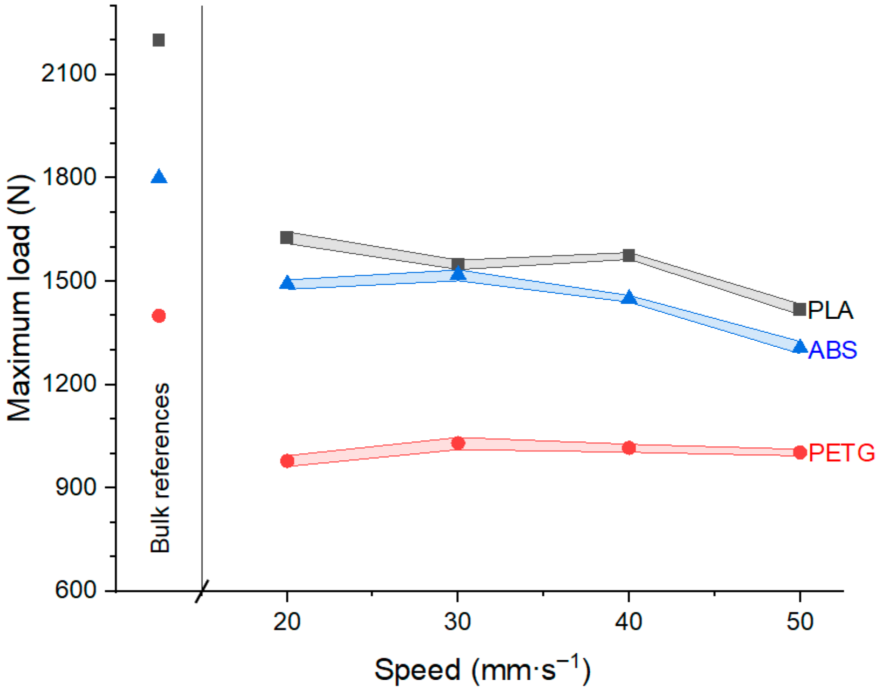


Figure 32: Effect of the nozzle speed on the samples' mechanical strength with "line" structure [21]

These two parameters are therefore linked, and increasing them will reduce printing time. However, it should be noted that increasing the nozzle size will alter the mechanical properties, lower the resolution, and reduce the surface finish.

To illustrate the reduction in time due to nozzle size, Figure 36 shows the cutting of a cube for nozzle diameters of 0.8, 0.6, and 0.25 mm. The estimated printing time by PrusaSlicer is 18 minutes, 21 minutes, and 1 hour and 20 minutes, respectively.

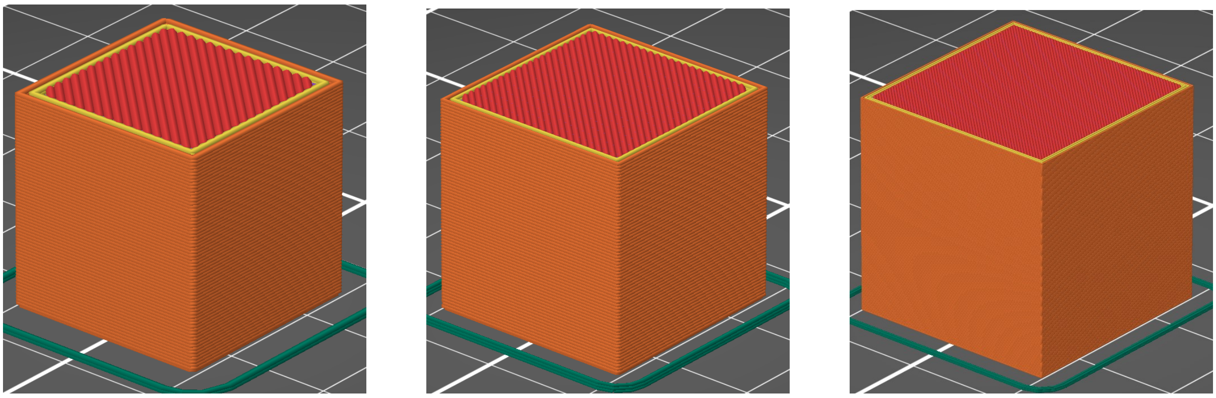


Figure 33: Nozzle diameter of 0.8 mm

Figure 34: Nozzle diameter of 0.6 mm

Figure 35: Nozzle diameter of 0.25 mm

Figure 36: Different cuts of 20 mm cubes depending on the size of the nozzle using PrusaSlicer

6.2.4 Filling form

With FDM, it is often possible to choose the shape of the part filling. A multitude of geometries exist, such as lines, triangles, honeycombs, stars, gyroids, etc. These shapes will influence the printing time. Figure 37 illustrates the effect of shape on printing time.

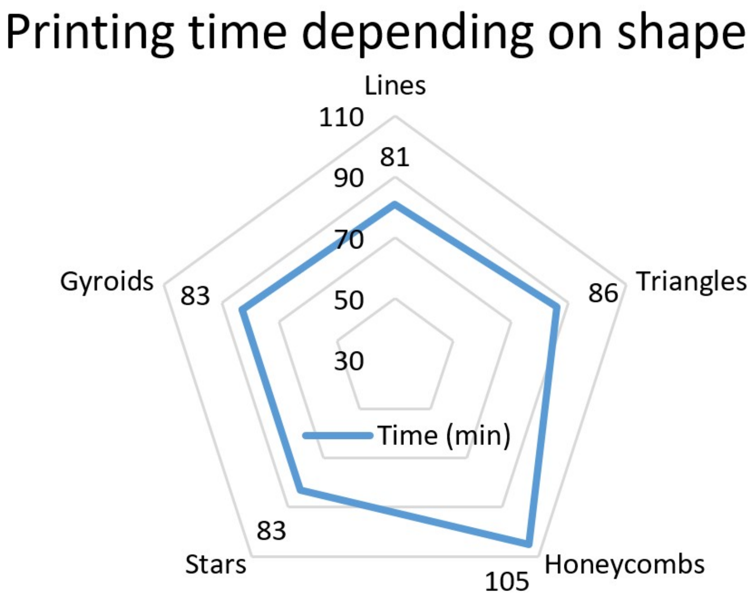


Figure 37: Influence of various shapes on the printing time of a cube with 20 mm edges and a fill rate of 50%, using the Prusa i3 MK3 model with a nozzle size of 0.25 mm in diameter, according to PrusSlicer predictions

6.2.5 Orientation

In some cases, increasing the number of layers increases printing time because the nozzle takes a certain amount of time to move vertically. To reduce this time, it is recommended to rotate the part to its “flattest” position in order to have as few layers as possible, as shown in Figure 41.

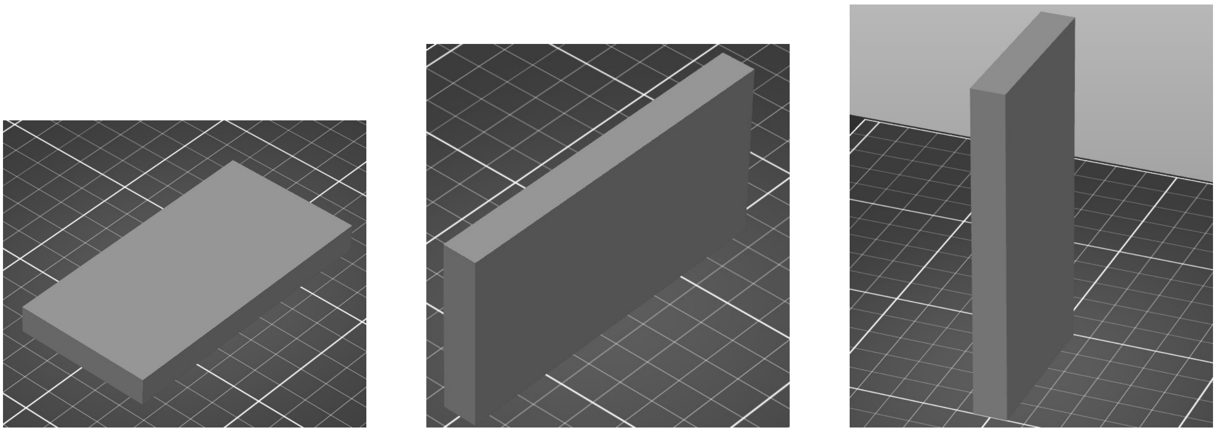


Figure 38: Best orientation : printing time of 5h41

Figure 39: Poorer orientation : printing time of 7h12

Figure 40: Worst orientation : printing time of 7h28

Figure 41: Estimated printing times for a rectangle measuring 10x50x100 millimeters, using the i3 MK3 model with a 0.25-millimeter diameter nozzle, oriented differently according to PrusaSlicer.

7 Conclusion

Ce rapport offre à un utilisateur novice en fabrication additive une vue d'ensemble complète sur l'impression FDM, en l'accompagnant depuis la conception orientée matériau et structure jusqu'à la gestion des formes et des géométries. Il aborde également les considérations liées au procédé d'impression, ainsi que l'optimisation du coût et du temps de fabrication. Enfin, il intègre des réflexions sur la durabilité et l'éco-conception, fournissant ainsi toutes les clés nécessaires pour une maîtrise avancée et responsable de ce procédé.

Parameter	**PLA (Generic)**	**ABS (Generic)**	**PETG (Generic)**
Filament-dependent settings			
Strength (MPa)	37–70	30–50	50–75
Stiffness (GPa)	3.5–4.0	2.0–2.7	2.0–2.5
Heat Resistance (°C)	50–60	90–100	70–85
Filament-dependent settings			
Nozzle temperature (°C)	220-230	240–250	250–260
Bed temperature (°C)	50–60	100–110	85–90
Printing settings			
Layer height (mm)	0.1–0.3		
Recommended wall thickness (mm)	1.5		
Infill density – rapid prototyping (%)	15-20		
Infill density – functional part (%)	60-80		
Infill pattern for compression	Grid, Cross, Triangle		
Infill pattern for other forces	Cubic, Concentric 3D, Cross 3D		

Table 5: Typical 3D printing parameters for PLA, ABS, and PETG (generic values).

To reduce the environmental impact of FDM, several approaches are possible:

- Reduce energy consumption: it is possible to reduce the energy consumption of printing by adding hot-end insulation, printer enclosures or heated bed insulation.
- Reduce the amount of material : to do this, it's possible to reduce the amount of support used by optimizing the print orientation or reducing the size of the support nozzle. It's also possible to reduce the fill rate of the entire part.
- Use more eco-friendly materials : ecological analysis is complex and depends on the criteria chosen. However, we could still establish the rule that, overall, it is preferable to choose PETG rather than ABS or PLA.

The table 6 summarizes the various parameters that influence printing time. Speed is often gained at the expense of precision, surface finish, and mechanical strength.

Parameter	Mechanical strength	Print time	Surface finish	Accuracy
Fill rate ↗	↗	↗	-	-
Nozzle speed ↗	↘	↘	↘	↘
Nozzle diameter ↘	↘	↗	↗	↗
Filling form change	It depends	It depends	-	-

Table 6: Qualitative summary of the parameters influencing FDM printing time and their consequences. ↗ : increases, ↘ : decreases, - : no effect.

8 Outlook

The study allowed for a comparison of several additive manufacturing processes and highlighted the particular potential of 3D printing. Although this technology is already widely used for prototyping and small-scale production, its mechanical performance and precision continue to improve thanks to advances in materials and printing systems. As a continuation of this work, it would be interesting to further compare metallic and polymeric processes, particularly in terms of durability, cost, and environmental impact. Material costs, which can vary significantly depending on the type and quality of the feedstock, remain a critical factor in selecting the appropriate process for a given application. Moreover, the integration of new approaches such as multi-material printing or topology optimization offers promising opportunities for designing lighter and higher-performance parts.

9 Bibliography

References

[1] Naghdali Choupani Ali Solouki Ahmad Makui and Hamidreza Seiti. *Analyzing the effects of printing parameters to minimize the dimensional deviation of polylactic acid parts by applying three different decision-making approaches*. Accessed on 20.10.2025. 2024. URL: `https://www.nature.com/articles/s41598-024-78952-9#citeas`.

[2] Mohammad S. Alsoufi and Abdulrhman E. Elsayed. *Warping Deformation of Desktop 3D Printed Parts Manufactured by Open Source Fused Deposition Modeling (FDM) System*. Accessed on 08.10.2025. 2017. URL: `https://www.researchgate.net/publication/318654219_Warping_Deformation_of_Desktop_3D_Printed_Parts_Manufactured_by_Open_Source_Fused_Deposition_Modeling_FDM_System`.

[3] Beatrice Aruanno. *EcoPrintAnalyzer: Assessing Sustainability in Material Extrusion Additive Manufacturing for Informed Decision-Making*. Accessed on 05.10.2025. 2024. URL: `https://www.mdpi.com/2071-1050/16/2/615`.

[4] Multiple unknown authors. *How to print overhangs*. Consulté le 7 octobre 2025. 2025. URL: `https://wiki.bambulab.com/en/filament-acc/filament/print-quality/overhang`.

[5] Tessa Axsom. *Recommended Wall Thickness for 3D Printing*. Consulté le 23 octobre 2025. 2022. URL: `https://www.fictiv.com/articles/recommended-wall-thickness-for-3d-printing`.

[6] BambuLab. *Filament Drying Recommendations*. Accessed on 16.10.2025. 2025. URL: `https://wiki.bambulab.com/en/filament-acc/filament/dry-filament`.

[7] Eric Boillat. *Schematic representation of the SLA*. Accessed on 23.10.2025. 2025. URL: https://www.epfl.ch/about/campus/wp-content/uploads/2021/06/HIM21_Boillat-Eric_Additive-manufacturing-Application-to-Microengineering.pdf.

[8] Eric Boillat. *Schematic representation of the SLM*. Accessed on 23.10.2025. 2025. URL: https://www.epfl.ch/about/campus/wp-content/uploads/2021/06/HIM21_Boillat-Eric_Additive-manufacturing-Application-to-Microengineering.pdf.

[9] custompartnet. *Schematic representation of the FDM*. Accessed on 23.10.2025. 2008. URL: https://www.researchgate.net/figure/Schematic-representation-of-the-FDM-fabrication-process-source_fig2_269631461.

[10] Piotr Czyżewski et al. *Influence of Extruder's Nozzle Diameter on the Improvement of Functional Properties of 3D-Printed PLA Products*. Accessed on 06.10.2025. 2022. URL: https://www.mdpi.com/2073-4360/14/2/356#B20-polymers-14-00356.

[11] Hrvoje Glavaš Grgić Ivan Mirko Karakašić and Pejo Konjatić. *Accuracy of FDM PLA Polymer 3D Printing Technology Based on Tolerance Fields*. Accessed on 20.10.2025. 2023. URL: https://www.mdpi.com/2227-9717/11/10/2810.

[12] Owen James Harding et al. *Methods to Reduce Energy and Polymer Consumption for Fused Filament Fabrication 3D Printing*. Accessed on 9.10.2025. 2023. URL: https://www.mdpi.com/2073-4360/15/8/1874.

[13] Gedeon Ang Hironori Kondo. *3D Printing overhang: how to 3D print overhangs*. Consulté le 7 octobre 2025. 2025. URL: https://all3dp.com/2/3d-printing-overhang-how-to-master-overhangs-exceeding-45/.

[14] Nicholas Hopkins, Liben Jiang, and Hadley Brooks. *Energy consumption of common desktop additive manufacturing technologies*. Accessed on 10.10.2025. 2021. URL: https://www.sciencedirect.com/science/article/pii/S2666790821000288?via%3Dihub.

[15] Ming-Hsien Hsueh et al. "Effect of Printing Parameters on the Thermal and Mechanical Properties of 3D-Printed PLA and PETG, Using Fused Deposition Modeling". In: *Polymers* 13 (May 2021), p. 1758. DOI: 10.3390/polym13111758.

[16] Nikko Industries. *How to Fix Over Extrusion on Your 3d Prints*. Consulté le 10 novembre 2025. 2022. URL: https://www.nikkoindustries.com/blogs/news/how-to-fix-over-extrusion-on-your-3d-prints.

[17] Jan Ramian Jakub Ramian and Daniel Dziob. *Thermal Deformations of Thermoplast during 3D Printing: Warping in the Case of ABS*. Accessed on 08.10.2025. 2021. URL: https://doi.org/%2010.3390/ma14227070.

[18] Rishi Kumara et al. *A Comparative Study on the Life Cycle Assessment of a 3D Printed Product with PLA, ABS PETG Materials*. Accessed on 05.10.2025. 2022. URL: https://www.sciencedirect.com/science/article/pii/S2212827122002190.

[19] Make$_{shaper}$. *How much does 3D filament cost ?* Accessed on 05.10.2025. 2025. URL: https://www.makeshaper.com/3d-printer-filament-price-cost.

[20] makerforged. *Thermoplastiques standard*. Accessed on 23.10.2025. 2022. URL: https://markforged.com/fr/resources/learn/3d-printing-basics/how-do-3d-printers-work/3d-printing-materials.

[21] Sergi Menargues et al. *Effect of Additive Manufacturing Parameters on PLA, ABS, and PETG Strength*. Accessed on 05.10.2025. 2025. URL: https://www.mdpi.com/2227-9717/13/9/2733.

[22] Majid R. Ayatollahi Mohammad Reza Khosravani Filippo Berto and Tamara Reinicke. *Characterization of 3D-printed PLA parts with different raster orientations and printing speeds.* Accessed on 07.10.2025. 2022. URL: `https://doi.org/10.1038/s41598-022-05005-4`.

[23] Khairul Izwan bin Ismail Mohd Aidil Nashruffi bin Mohd Khairul Nizam and Tze Chuen Yap. *The Effect of Printing Orientation on the Mechanical Properties of FDM 3D Printed Parts*. Accessed on 09.10.2025. 2022. URL: `https://www.researchgate.net/deref/https%3A%2F%2Flink.springer.com%2Fchapter%2F10.1007%2F978-981-19-2890-1_8?_tp=eyJjb250ZXh0Ijp7ImZpcnN0UGFnZSI6InB1YmxpY2F0aW9uIiwicGFnZSI6InB1YmxpY2F0aW9uIn19`.

[24] Everlyn Kerubo Mosomi, Oludolapo Akanni Olanrewaju, and Samson Oluropo Adeosun. *Pivotal role of polylactide in carbon emission reduction: A comprehensive review*. Accessed on 10.10.2025. 2024. URL: `https://onlinelibrary.wiley.com/doi/10.1002/eng2.12909`.

[25] OpenAI. *Présentation de ChatGPT*. Accessed on 23.10.2025. 2023. URL: `https://www.openai.com/chatgpt`.

[26] Madeleine P. *Wall Thickness in 3D Printing: All You Need to Know*. Consulté le 9 octobre 2025. 2024. URL: `https://www.3dnatives.com/en/wall-thickness-in-3d-printing-all-you-need-to-know-281120234/#!`.

[27] Benoît Pernet, Jacquelyn Kay Nagel, and Hao Zhang. *Compressive Strength Assessment of 3D Printing Infill Patterns*. Accessed on 23.10.2025. 2022. URL: `https://www.sciencedirect.com/science/article/pii/S2212827122001159?ref=pdf_download&fr=RR-2&rr=992f2d85eee33b51`.

[28] 3D Printing.com. *How 3D Printed Lattice Structures Improve Mechanical Properties*. Accessed on 23.10.2025. 2019. URL: `https://3dprinting.com/tips-tricks/3d-printed-lattice-structures/`.

[29] + Redyarsa Dharma Bintara. "The Characteristic of Overhang Object to Material Usage on FDM 3D Printing Technology". In: *Journal of mechanical engineering science and technology (JMEST)* (2019).

[30] Avery S. *Bridging in 3D Printing: Everything You Need to Know*. Consulté le 10 novembre 2025. 2023. URL: `https://www.3dnatives.com/en/how-to-avoid-briding-in-3d-printing-2107234/#!`.

[31] sciaky. *What is Directed Energy Deposition (DED) 3D Printing*. Accessed on 23.10.2025. 2022. URL: `https://www.sciaky.com/additive-manufacturing/what-is-ded-3d-printing`.

[32] Prusa Slicer. *Prusa Slicer*. Accessed on 23.10.2025. URL: `https://prusaslicer.net/`.

[33] Dassault systemes. *Binder Jetting and materials*. Accessed on 23.10.2025. 2018. URL: `https://www.3ds.com/make/guide/process/binder-jetting`.

[34] manufactur3d Team. *A Comprehensive List of All 3D Printing Technologies*. Accessed on 23.10.2025. 2021. URL: `https://manufactur3dmag.com/a-comprehensive-list-of-all-3d-printing-technologies/`.

[35] Unknown. *3D printing bridging*. Consulté le 27 octobre 2025. 2025. URL: `https://www.eufymake.com/blogs/printing-guides/3d-printing-bridging?srsltid=AfmBOoqfWUvVHdiTX6wPwo_XjASFbYy43vt17Zqpg6u2YIAtE7Hz7tsV`.

[36] Unknown. *Design rules for FFF 3D printing*. Consulté le 27 octobre 2025. 2024. URL: `https://www.hydraresearch3d.com/design-rules`.

[37] WEERG. *Guide impression 3D*. Consulté le 10 novembre 2025. URL: `https://www.weerg.com/fr/faq/lignes-directrices-de-conception-pour-impression-3d`.

[38] Wikipedia. *Thermoplastic elastomer*. Accessed on 9.10.2025. 2025. URL: `https://en.wikipedia.org/wiki/Thermoplastic_elastomer#Advantages`.

[39] zwickroell. *ASTM D695 Essai de compression composites et plastique*. Accessed on 23.10.2025. URL: `https://www.zwickroell.com/fr/secteurs-dactivite/composites/end-loading-compression-astm-d695/`.