École Polytechnique Fédérale de Lausanne
Section de Génie Mécanique

Introduction to Additive Manufacturing ME-413:
Project report

Prof. Boillat Eric
Prof. Brugger Jürgen
Prof. Moser Christophe

Consumer additive processes: Towards domestic 3d-printers research

---

Group A-2

AVONI Leonardo	301524
TERZI Alex	258856
ZADOKS Dillon Joshua	338409
HUNG Wei-Sheng	322678

16 december 2020

Table of contents

1 Abstract

This report aims to study the domestic (desktop) 3D printers, which involves mainly two technologies - “Fused Deposition Modeling (FDM)” and “Photopolymer based process (SLA, MSLA, DLP...)”, among different printing methods in additive manufacturing. For a thorough research, each method has divided into crucial subparts of 3D printing, such as printing materials, supporting materials, and post-processing. At last, the report introduces the user interface of the printers and a real part demonstration to wrap up the study.

2 Introduction

The concept of additive manufacturing, or 3D printing has existed since 1945 and in practice—however primitive—since 1971. Nonetheless, it was not until 2006 that the commercial 3D printing finally came to the desktop. The first one is from Objet (now Stratasys), which let users send designs to its device in order to print them [24]. Though variety of additive manufacturing methods has developed, FDM and Photopolymer Based Processes in general have become the most popular options for desktop 3D printers nowadays [48]. Therefore, this report will focus on these two technologies and go through the process, supporting by our real practice for printing a smartphone case.

In section 3 and 4, the brief development of additive manufacturing has been addressed, as well as the latest desktop 3D printers. By reviewing these latest products (literally the best printers in 2021), some user-concerned factors are then discussed and gives readers an idea on recent desktop 3D printers.

Sections 5 to 9 deal with the different aspects of printing technology (for both FDM and photopolymer based processes). This includes for example the different printing materials, and the post processing needed.

Section 10 explains the user interface of how to actually print a part from a 3D model, by some specific modeling and slicer software. Some inportant parameters and print settings are introduced here to understand how to improve the printing quality.

Finally, section 11 aims at building a few real parts in order to compare the theoretical and practical printing behaviour.

3 State of the art

Additive manufacturing, generally called "3D printing" is a group of modern production techniques grouping all methods that allow to produce a part by adding material, without using any shape tool. This opposes to substractive methods (e.g. machining, EDM), that help producing a part by removing material, and replicative methods (e.g. casting, deep drawing...) where the shape of the final object is obtained by using a shape tool (e.g.the mold) [65]. Nowadays, this technology is more and more common in households, schools and industries, since it allows for rapid prototyping ([54]) and creative experimentation ([69]), while helping learn design and engineering ([66],[77]). Common uses of additive manufacturing are found in the Automotive industry, Aerospace, Medical fields, and even Civil engineering ([27],[74]). Additive manufacturing has also contributed during the COVID-19 global pandemic, allowing for creation of ventilator parts and other medical devices ([71]).

Evolutions in Additive Manufacturing have been possible thanks to research from the second part of the XX century, that led to a wide array of 3d printing processes such as FDM, SLA, MJ, BJ, SLS and many others ([63],[62],[6],[27],[68]). Nowadays, due to good understanding of the technology, and the presence of open-source options ([61]), the market, especially of desktop 3D printers is becoming viable ([34]). A number of advancements preconised by [63] like CAD software accessibility (allowing for complex part creation) and education around Additive Manufacturing is available, even for low-end printers. Such widespread bring a new way of thinking, especially for students and makers around the world: why don’t produce at home the neeeded parts instead of buying them? Research has proven that this method is economically viable ([76],[77]). However, creating parts using those kind of tools can be challenging; one must be able to succcesfully produce the CAD model, then needs to be able to slice it and set the machine in order to produce the component. Those steps require a certain level of knowledge ([54]). Also, one must pay attention to potential health threats when operating the printers ([79],[78]). Concerning specifically the case of desktop FDM and SLA printers, good knowledge has been reached. We know the precision levels allowed by those technologies ([73],[75]), we are also aware of the advantages/disadvantages between SLA and FDM ([67]). We also know what the obtainable resistance levels and potential printing errors can be ([52],[70]). Finally, dozens of low to mid range printers (from approx. 500 to 2000 USD) are nowadays available on the market, and able to produce quality results ([72]).

3.1 Latest 3D Printer Survey

In this part, some outstanding printers are surveyed and compared by some important features to customers, such as the size, price, or the print quality, etc. From two websites of the 3D printing communities [56][13], 4 products are listed here as overall the best products in 2021, 2 by FDM and 2 by MSLA (printing methods will be introduced in details in section 5). To be specified, the latest version of each product are chosen because in some examples, two communities select the same products but different version. The result is shown in the table 1.

Method	FDM		MSLA	
Product	Original Prusa i3 MK3S+ [31]	Creality Ender 3 Pro [21]	Elegoo MARS 2 [20]	Anycubic Photon Mono X [35]
Size (HWD) [cm]	38 x 50 x 56	20.3 x 38.1 x 53.3	41 x 20 x 20	47.5 x 29 x 27
Price [€]	769 in kit form 999 fully assembled	180.18	171.75	454.2 ($529)
Build volume (HWD) [cm]	25 x 21 x 21	25 x 22 x 22	15 x 8 x 12.9	24.5 x 12 x 19.2
Print speed	200+ mm/s (15 mm^3/s)	$\leq$ 180 mm/s normal 30-60	30-50 mm/h	60mm/h
Resolution [mm]	0.05	0.1	0.05(XY)	0.05(XY)

Table 1: Best 3D printers in 2021

4 Aim of the study

In this section, the features of a domestic printer is discussed, and it aims to find out some features that customers may consider the most important. In a straight-forward approach, some significant factors can be addressed by the comparison table 1, which are the best recent products reviewed by users and communities.

Among the FDM printers, Original Prusa i3 MK3S+ appears to be the top pick. The reasons are its exceptional print quality and the user-friendliness. The company Prusa also has its own filaments and slicer software, which makes the printer foolproof. Moreover, the printer is one of the line from RepRap – an open, collaborative project to develop 3D printers capable of making more 3D printers (Replicating Rapid-prototyper), which means it has large community resource and supports [59]. The Creality Ender 3 series is also a popular FDM printer because of its excellent price-to-performance, the price is almost six times cheaper than MK3S+. The other good feature is its highly modular and modifiable design, it can convert to laser engraver (figure 1) or other function depends on users.

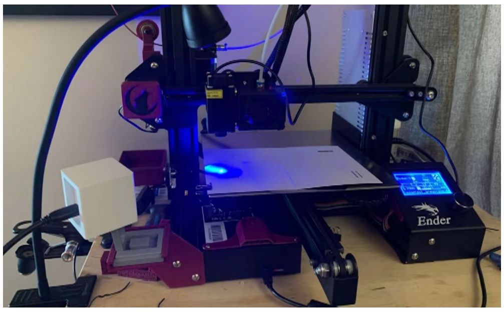


Figure 1: Laser engraver module of Creality Ender 3

Among the photopolymer based processes printers, MSLA-based stands out to be the most popular products with the advantage of fast-printing. Elegoo MARS 2 is one of the budget resin printer with good resolution. Its use of monochrome LCD provides fast curing time per layer, which improves from 7-8 seconds per layer to 2-3 seconds compared to older version. Its resolution is 50 μm by a 2K 2560 x 1620 pixel-sized panel, which is a standard resolution for this class of budget resin printer. From the article [57], Elegoo MARS 2 stands out because of its reliability and its nice appearance - a green plastic vat(figure 2). Anycubic Photon Mono X is considered the choice of supersize pick. Large build volume, 4K screen, and monochrome display, are all the trendy features of current resin 3D printing [55]. It has a special feature that it allows users to adjust and dial down the UV-light strength to prolong the lifecycle further. The cons of Anycubic Photon Mono X is its expensive price, but it still gets one of the largest build volume in resin 3D printers.

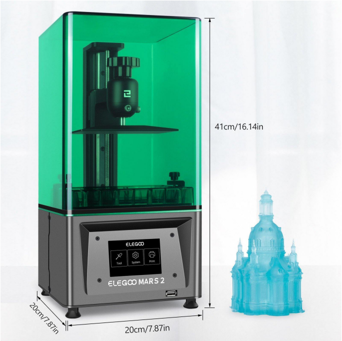


Figure 2: Elegoo MARS 2 [20]

To conclude, the price and the print performance are still the most significant factors. A worth-noticing point is that the open-source and the community resources become attractive to customers. In the selection of the resin printers, two main features are print speed and resolution. The print speed depends heavily on layer curing time in MSLA, which already enhanced by the monochrome screen. The resolution depends on the pixels of the screen. In fact, most of the recent MSLA printers already got monochrome screen and similar resolution. Interestingly, the appearance and design becomes a deciding factor for customers to choose similar class resin printers.

5 The Printing Methods

The domestic 3D printers available on the markets today mainly employ two different methods for manufacturing 3D printed parts: fused deposition modeling and a variety of photopolymer based processes. This section will explore the working principles, basic hardware configurations, the influential printing parameters, and an overview of the capabilities and limitations of each of the printing methods.

5.1 Fused Deposition Modeling

Fused deposition modeling (FDM), also commonly known as fused filament fabrication (FFF), is an additive manufacturing process where a part is manufactured by extruding thermoplastic filaments in layers that form the part. The two most common materials used in FDM processes are Polylactic acid (PLA) and Acrylonitrile Butadiene Styrene (ABS). There are a number of other materials that can be utilized in FDM including some more exotic materials such as wood filament and magnetic filament. This large selection of materials available allow the user to have more control over the properties of the final product. These are explored further in section 6 Materials - FDM.

Figure 3: Images of Prusa MK3S model FDM 3d printer [58]

Figure 3 shows a domestic FDM 3D printer with the common parts labeled:

1. Filament: The filament is the material that is used to build the part. It is heated and extruded through the nozzle to build individual layers of the part. The printer can easily change between different materials and has storage space for an extra spool of filament.
2. Build Plate: The build plate is the surface where the part is printed. It is usually heated to prevent warping of the finished part.
3. User Interface: The user interface is where the user can select parts to be printed, adjust printing parameters directly, or calibrate the printer.
4. Extruder: The extruder heats the filament and pulls the filament into the nozzle to be extruded onto the build plate.
5. Cooling Fan: The cooling fan cools the filament once it has been extruded onto the build plate or the previous print layer to provide more uniform cooling to prevent warping of the part.
6. Nozzle: The nozzle is the part of the extruder where the filament is finally melted and deposited onto the build plate. The nozzle diameter can range in size and determines the "resolution" of the print. Sizes usally range from 0.2mm up to 1.0mm.

FDM works by heating the working material just past its glass transition temperature. The glass transition temperature is the temperature at which the filament transitions from hard and brittle to a viscous and less brittle state. The material is then extruded through the nozzle layer by layer in precalculated locations. It then cools below the glass transition temperature and solidifies. The next layer is then extruded on top of the previous layer which adheres to the previous layer, slowly building up the part. To fill in an area the extruder can utilize different infill patterns which can be selected within the slicing software. FDM also allows the user to vary other printing parameters such as nozzle temperature, build platform temperature, build speed and layer height. A more in depth discussion of these parameters and their effects on the resulting part can be found in section 10.2 Detailed parameters for FDM printer.

The typical build size for a desktop 3D printer is 200 x 200 x 200 mm. Industrial printers can reach much larger sizes, however a larger build size can be emulated by printing a part in multiple smaller parts then assembled. Typically FDM printers can print with layer heights between 0.05mm to .4mm. A smaller layer height creates higher quality and smoother parts. A larger layer height creates parts faster and at a lower cost point. The most common layer height used is 0.2mm which creates a balance between print quality and print speed.

Warping is one of the most common issues when printing with FDM. This occurs when the material cools down after extrusion and begins to shrink. Different sections of the part cool at different rates which causes non-uniform changes in the dimensions causing the build up of internal stresses. This causes the bottom layers to pull upwards causing the part to warp. Warping can be prevented by monitoring and adjusting the build plate temperature and the chamber temperature. Warping can also be prevented by increasing the adhesion between the build plate and the part. Common methods include adding a layer of glue or hairspray to build plate causing the extruded material to stick to the build plate better. Different materials are more sensitive to warping.

FDM offers a large variety of materials and colors to the domestic 3D printer user. The amount of work for post-processing of each part is low and each part can be printed in a number of hours with high quality and consistency.

Advantages[65]:

1. Simple and clean to use
2. Large variety of available materials
3. Material is easy to store
4. Little post-processing

Disadvantages:

1. Anisotropy of printed parts
2. Low manufacturing speed
3. Not as accurate as photopolymer based options

5.2 Photopolymer Based Processes

Photopolymer based 3D printers are based on the same principles but use different methods to obtain the result. Generally a photopolymer printer will have a vat of liquid resin. This resin is special in that when a strong UV light is shined on the material, it will cure into a solid material. Photopolymer based printers take advantage of this by selectively shining the UV light and curing the resin into the desired shape. The process is similiar to FDM in that the process is done layer by layer.

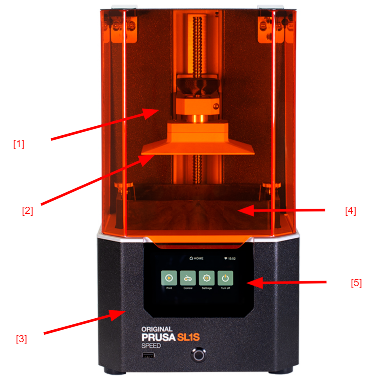


Figure 4: Images of Original Prusa Sl1s Speed [33]

5.2.1 SLA

The most commonly used photopolymer based process in home 3D printers is stereolithography (SLA). SLA is a very typical photopolymer process and an SLA printer usually consists of a tank of resin, a build platform, a light source (typically a laser), and an elevator. The elevator raises and lowers the build platform. The build platform is transparent and the galvanometers (a pair of precisely maneuverable mirrors) are located below the resin tank. The galvanometers are then able to precisely shine the UV light to cure the desired layer of resin.

Figure 4 shows the parts of a standard SLA printer:

1. Elevator: Used to move the part up or down once a layer has completed curing.
2. Build Plate: The part adheres to the build plate allowing the elevator to raise or lower the part.
3. The laser, galvanometers, and mirrors: The apparatus used to cure the resin cannot be seen directly on this printer. These are used to selectively cure the resin in the desired location to create each layer of the object.
4. Resin tank: This is where the resin is held while the part is being printed.
5. The user interface: The user interface is where the user can change print settings and adjust printer settings.

The instant the light hits the resin in the vat, the resin at that point is cured. Once the printer has cured the entire layer, the elevators raise the build platform up one layer height.

SLA printers can also be set up to print top-down as opposed to bottom-up. This simply moves the light source and galvanometers above the tank of resin. This means the build plate slowly lowers into the tank of resin until the item is complete and will be completely submerged in the resin.

5.2.2 MSLA

MSLA is so-called Masked-SLA. Its principle is mostly the same as SLA, but with different ways to expose each layer to UV light. MSLA printers expose a whole layer at once with an LCD photomask, whereas SLA printers steer a point of light to every element of a layer before moving on. As a result, the print speed of MSLA outperforms SLA because the speed of MSLA becomes independent of the model's sectional area but only height, and it is better for printing lots of parts at once.

5.2.3 DLP

Digital light processing (DLP) is another form of photopolymer based 3D printing. It uses the same working principles as SLA but differs in the way the light source is applied to the resin. Instead of a laser DLP uses a digital projection surface. In SLA each point is cured at a time, while in DLP the printer projects a complete layer at once. Due to this DLP printers are able to print faster than SLA printers.

DLP is able to cure entire layers at once by using a digital micromirror device. This controls precisely where the light should be projected. The digital micromirror devices contains a large amount of small micromirrors that are able to direct light precisely to create each layer of the print.

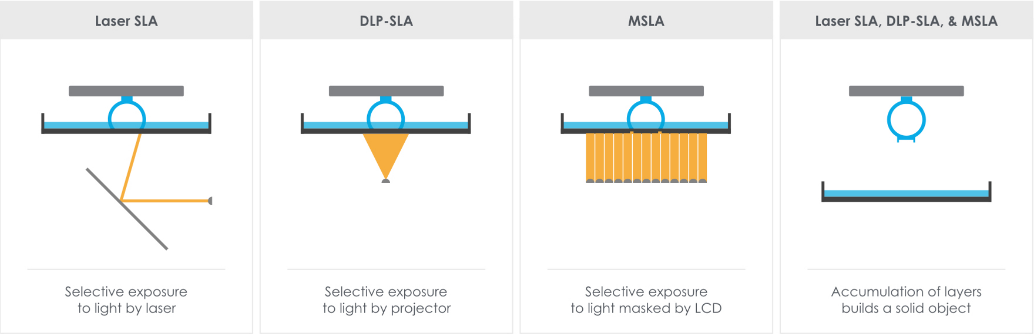


Figure 5: Photopolymer based processes [53]

Advantages[65]:

1. Very accurate (25µm)
2. Fast printing times

Disadvantages:

1. Limited to photoresist materials
2. Standard materials are expensive, toxic, and difficult to store
3. Post-processing is required

6 Materials - FDM

6.1 Printing materials

As we will see in the following subsections, fused deposition modelling can use a variety of materials, whether for the print or the supports, which are chosen according to the needs of the user.

However, the form in which the material is used is always the same for a 3D domestic printer: a spool of solid thermoplastic filament which is melt and extruded through the printer nozzle as the part takes shape. In addition to the material, the spools differ in other aspects: the diameter of the filament, which for the average user can be 1.75 mm, the most common, or 2.85 mm, and the weight. Standard spools are 1 kg, regardless of the material, but there are also smaller, for example 250 g, and larger of up to 8 kg [3]. Most materials are available in a wide range of colours, which vary mainly according to the supplier.

6.1.1 PLA

Polylactic acid (PLA), or more correctly poly(lactic acid) or polylactate, is the polymer of lactic acid. It is a thermoplastic polymer derived from renewable resources such as corn starch or sugar cane [37]. PLA is a versatile material that finds use in a variety of sectors. It is often used for the production of upholstery, carpets and awnings, as well as for the production of medical and surgical materials. In fact, implants are produced with this type of plastic, such as nets, screws or rods for bone or tissue surgery.

The simplified production process for this polymer goes through the following steps [22]. In the first stage, we start with corn starch, from which we obtain sugar which is then fermented and transformed into lactic acid. This is then fermented and transformed into lactic acid, which is then polymerised into polylactic acid, which in turn is transformed into granulated resin. The granulated resin is usually light in colour and is then put into a mixing machine. Colour pigments and possibly additives are added. The mixture is usually dried at 60° to 80°C to reduce the chances of the material bursting or clogging the nozzle. The material, still in granule form, is put into a machine, where it is heated, mixed and then extruded as a solid filament. This filament is then passed through a tub of hot water, which "cools" the material and causes it to take on a round shape. Finally, the round filament is passed into a tub of cold water and wound onto the spool.

The process of PLA degradability [10], which cannot be considered truly biodegradable at ambient temperature, is closely linked to the environment with three key factors: heat, humidity and pressure. In fact, PLA degrades best in warm environments (above 60°C, the glass transition temperature of PLA), in even shorter times if in the presence of bacteria, such as in soil. Sunlight does not accelerate the process (except for the heat), but only makes the object lose its colour. On average, after 6 months in these conditions the first signs of degradation of the object begin to appear, whereas under normal conditions a PLA print can last for years and years, as for normal plastic.

In addition to its industrial uses, PLA is one of the most widely used materials for home 3D printing, providing the basis for early experiences in this field. It is easy to use, can be printed at low temperatures, usually between 190°C and 230°C, and does not require a heated bed, although for large print sizes it is recommended.

It is a very economical material and creates parts that can be used for a wide variety of applications. However, compared to other materials it is stiffer and therefore sometimes more difficult to process once extruded. Its low melting temperature makes it unsuitable for many applications: in addition to its less-than-ideal mechanical properties, exposure to temperatures above room temperature can cause deformation.

Among its positive aspects are [28]:

- Extreme ease of printing (low extruder temperature)
- Good aesthetics (with 0.2 mm layers)
- Economical
- High printing speed (4000-10000 mm/sec)
- No need for heated platen
- Non-toxic and odourless during printing
- Environmentally friendly and biodegradable under certain environmental conditions

The cons we can highlight are:

- Sensitive to moisture before printing, so the coil must be used quickly or kept in a vacuum
- Not suitable for mechanical parts, as it is rigid but not particularly resistant
- If the material is not of good quality it can clog the printer nozzle

6.1.2 ABS

ABS, or Acrylonitrile Butadiene Styrene, is a thermoplastic widely used for everyday products [5]. It remains one of the most widely used in today's domestic 3D printing, as it guarantees high-quality finished objects. Also widely used in other additive manufacturing processes, its applications are much more expansive than PLA. It is used to make toys (for exemple LEGO bricks), safety helmets parts, control panels for household appliances, musical instruments such as flutes and clarinets, black water pipes (white, orange or grey pipes are made from PVC), boxes for electrical components, etc [12].

However, ABS is a non-sustainable material, derived from petroleum, and the fumes produced by its extrusion if inhaled can be dangerous[49]. The reason why this filament is preferable to PLA is mainly due to its accuracy, which can be further improved with post-processing techniques, and its mechanical specifications such as stiffness and lightness. Furthermore, it can be used in a temperature range between -20°C and 80°C. It resists well the action of concentrated acids such as hydrochloric and phosphoric, but suffers from the attacks of concentrated acids such as sulphuric or nitric concentrated acids, and is very sensitive to acetone, which makes it the main finishing technique after printing [26].

ABS is a very low cost material and its extreme strength allows it to be used for prototypes that need to be durable and rigid. However, compared to other materials, it is less likely to crumble, undergoes generally higher temperatures and has a lower coefficient of friction, making it easier to extrude. Despite this characteristic, it is not an easy material to print, as it is very sensitive to temperature, which can easily cause the part to shrink during printing. It is printed at higher temperatures than PLA, around 230 °C, and requires a heated bed and environmental protection. It is therefore not uncommon for this material to be used with closed printers to help maintain geometry.

To sum up, its positive sides are [28]:

- Good mechanical resistance
- Good aesthetic performance, especially with post-processing techniques
- Economical
- Greater precision than PLA
- Less prone to deterioration when exposed to the environment
- Easy post-process

On the downside, the cons are:

- Toxic fumes and unpleasant smell.
- Need for a heated bed (possibly up to 100°C).
- Nozzle temperatures between 220 and 245°C
- Sensitive to changes in temperature
- Not biodegradable

6.1.3 PETG

PETG is a transparent polyethylene terephthalate capoliester: a modified version of PET [36]. The "G" stands for "modified glycol", which is added to the composition of the material during polymerisation. The result is a filament that is clearer, less brittle and easier to use than its basic PET form, and above all very easy to extrude and recycle. Although the typical appearance of the material is semi-transparent, numerous varieties of coloured filaments are available on the market.
Outside the field of 3D printing, it is used for point-of-sale displays, signs, illuminated displays, signage and many kinds of construction details [4]. This material is difficult to attack by chemicals, as demonstrated by the containers in which petrol, fuel oil and solvents of various kinds are stored.
It is an extremely strong material that allows for durable prints as well. The low shrinkage coefficient makes this material excellent for 3D prints with large flat surfaces. It can be described as a good compromise between the printability of PLA and the strength of ABS.
It has high impact resistance, although it can be scratched easily, making it suitable for prints that have more practical than aesthetic functions. It has high mechanical strength and excellent flexibility: ideal for use for objects subjected to mechanical stress, it is a hard material, extremely resistant and flexible. This is also due to an excellent coefficient of adhesion between layers. The very strong adhesion between layers means that they are not so easy to peel off after printing. The shrinkage rate during processing is very low: it retains its dimensions faithfully as it cools and is therefore ideal for large prints. It is also extremely resistant to chemicals, acids and alkalis, and is completely odourless, which is important for home use [2].

The advantages of this material are therefore[28]:

- Easy to print (temperatures similar to PLA)
- Resistance comparable to ABS

- Does not emit particularly unpleasant odours
- Almost impervious to acids, solvents or saline environments
- Suitable for food contact

The negative aspects are:

- Less economical than PLA and ABS
- Needs more detailed print settings to avoid inaccuracies

6.1.4 Nylon

Nylon is a synthetic, polyamide plastic with remarkable mechanical properties [30]. When 3D printing first became popular, nylon was not listed as an extrudable material. It is a flexible and tensile material and is particularly well known for its use in the textile industry, but also in industries that need a strong and flexible material. There are many different types of nylon: 3D printing mainly uses nylon 6 and 6.6, which are the most common [15]. This material needs high temperatures, around 240 °C, to be printed.

Nylon is an incredibly strong, durable and flexible material, which makes it perfect for printing objects that will be subjected to stress or falls. It is stronger than other printable materials. Despite the less-than-perfect finish, it is generally considered a good compromise if the technical specifications of the print are to meet certain criteria. For processing, it has requirements at both ends of the spectrum: on the one hand, it needs high extrusion temperatures, but if these are too high they can lead to inaccurate prints, e.g. with bubbles inside, and on the other hand it does not need a heated printing plate. The latter feature, as with PLA, is however not recommended in the case of medium to large objects. However, the deformation during printing is comparable to that of ABS, which suggests the use of a closed environment printer.

As with PETG, nylon normally comes in transparent spools, but can be purchased in different colours. Furthermore, being a hygroscopic material (absorbing liquids), it is possible to leave the filament in liquid fabric dyes to make it take on that same colour [29].

Positives aspects[28]:

- Mechanical strength, durability, flexibility
- Affordable price

Negative aspects:

- Extrusion temperature close to 250°C
- Sensitive to moisture
- Worse final finish than other materials
- Tendency to warp during cooling (closed printing area recommended)

6.1.5 Others

There are many other types of filament[28], which are generally produced from a base of the materials we have seen above. These materials are often called "doped" because they have a varying percentage of contamination that makes them special. They are often used if you are looking for a particular aesthetic grade or specific applications. Below we will look at the main ones on the market.

WOOD FILAMENT

The wood filament is a PLA with contamination of wood fibres. There are different types with different types of wood (ebony, maple, bamboo, etc. [51]). This filament is specifically aesthetic, and if printed correctly, produces models that resemble wood straight out of the printer, which is excellent to the touch and often to the smell. It is, however, contraindicated for functional models as it is rather fragile, but very popular for architectural projects, product design, model making etc.

Suggestions:

- It is often printed with rather high layers to simulate wood striations
- The extrusion temperature is very subjective to the type of filament, because different fibres react differently.
- Sanding is often used as post-processing to further improve aesthetics.

METALLIC FILAMENTS

These are based on the same principle as wood-based filaments, but contaminated with metal powders. Excellent for obtaining, for example, a bronze, copper, gold, silver, brass, aluminium etc. effect. Once sanded to the touch they can achieve an excellent aesthetic result, looking as if they were really made of metal. In addition, when printed with 100 % filler, the weight will also be convincing. Generally, filaments are 50 % PLA or ABS and 50 % metal powder, but on the market you can find filaments with a composition of up to 85 %.

CONDUCTIVE FILAMENT

This filament consists of a base material contaminated with graphite. The effect obtained is that of electrical conduction, which makes it ideal for printing circuits and electrical connections at low voltage. It is often used to integrate mechatronics into small projects involving sensors, LEDs, Arduino, etc.

MAGNETIC FILAMENT

Unlike the previous filaments, it is contaminated with exclusively iron powders, which make the prints ferromagnetic. This means that objects can be attracted by magnets, making them excellent for practical applications such as magnets and more.

CARBON FILAMENT

This filament has a much more professional character than the previous ones. It is based on a contamination of PLA, ABS or PETG filaments in order to significantly increase the mechanical resistance while keeping the weight of the object very low. The main difference is that this carbon filament is not suitable for printing with standard (brass) nozzles as it tends to wear out very quickly. A special nozzle is generally required, often made of a harder material such as hardened steel. These nozzles, however, are available on the market at similar prices to standard nozzles.

PHOSPHORESCENT FILAMENT

Filament contaminated with phosphorescent material capable of absorbing photons, often used for aesthetic applications.

6.1.6 Comparison

The advantages and disadvantages for these main printing materials have been listed in the previous subchapters.

We can summarise the choice of material with the following points:

- The strength you would like the moulded part to have
- The aesthetics you need for your project
- The budget
- The printer you own

In the table 2, we have listed some of the mechanical and thermal properties of the materials shown, in order to provide some technical data [16]

	PLA	ABS	PETG	Nylon
Density	1.24 g/m^3	1.04 g/m^3	1.23 g/m^3	1.10 g/m^3
Coeff. of thermal expansion	68 $\mu m/(m.^\circ C)$	90$\mu m/(m.^\circ C)$	60 $\mu m/(m.^\circ C)$	95 $\mu m/(m.^\circ C)$
Ultimate strength	65 MPa	40 MPa	53 MPa	40-85 MPa
Suggested print temperature	190-220 °C	220-250 °C	230-250 °C	220-270 °C
Suggested bed temperature	45-60 °C	95-110 °C	75-90 °C	70-90 °C
Price	∼30 CHF/kg	∼30 CHF/kg	∼30 CHF/kg	∼100 CHF/kg

Table 2: Comparison of mechanical properties in printing materials

6.2 Supporting materials

It is often necessary to support our models that have unsupported volumes or surfaces, i.e. parts that would be printed in air. These supports if printed in the same material as the model will necessarily have to be removed by "brute force", for example with pliers or a cutter. If you set up the "support section" of the Slicer correctly, the removal of supports is also not too complicated, but you may have problems removing them due to the geometry of the print, especially in areas that are not accessible with tools[11]. A second problem is undoubtedly the finish of the surfaces resting on the supports, which is inexorably of a worse quality than surfaces not requiring supports.

For these reasons, two soluble materials used for this purpose will be illustrated, PVA and HITS. Generally speaking, they are not used very often, especially for home 3D printing because of the budget. In fact, to be able to print an object with two materials at the same time, you need a 3D printer with two extruders [8]. There are some on the market at affordable prices for the public, but they are not always the first choice for the average user.

6.2.1 PVA

PVA is generally the first choice for backing materials. The reason for this is simply that it is water soluble [38]. Simply immerse the finished print in a water bath, preferably lukewarm, to dissolve the substrates in a non-toxic manner with no disposal problems. PVA is in fact biodegradable and does not form dangerous by-products. The time required depends on the geometry and thickness of the substrates, but is generally in the range of 10 to 20 hours.

In addition, PVA substrates can be printed in contact with the surface to be supported (which is impossible to do with substrates made of the same material as the print). In this way, once dissolved, the surface is smoother and the print is better.

The principal downsides are[28]:

- Requires a dual extruder printer
- High price (about three times that of PLA)
- Extremely sensitive to humidity

6.2.2 HIPS

HIPS often fall into the category of soluble substrate materials, and this is certainly not wrong, as it is soluble with a substance called limonene [7]. However, it can also be used as the main printing material.

As a substrate material it has the same advantages as PVA, and is more similar in cost to PLA or ABS, but adds to the cost of limonene.

Advantages[28]:

- Interesting mechanical properties, especially impact resistance

- Easy to print and process
- Food safe
- Not sensitive to moisture
- Printing temperatures similar to ABS, but without material shrinkage

6.2.3 Comparison

In order to understand which substrate material is best, one has to consider above all which material will be used for the main print.
For example, PVA needs an extruder temperature of 190-210 °C and a print bed temperature of 50-60 °C, which is a perfect match for materials such as PLA and similar.
HIPS in turn needs an extruder temperature of 230-240°C and a table temperature of 80-90°C, which makes it impossible to use with PLA, just as it would be impossible to use PVA with ABS, as the table temperature would be too cold or too hot depending on what we are considering between the two.

A few more details can be found in the table 3 [16].

	PVA	HIPS
Density	1.23 g/m^3	1.04 g/m^3
Coeff. of thermal expansion	85 $\mu m/(m.°C)$	80$\mu m/(m.°C)$
Ultimate strength	78 MPa	32 MPa
Suggested print temperature	185-200 °C	230-245 °C
Suggested bed temperature	45-60 °C	100-115 °C
Price	~100 CHF/kg	~30 CHF/kg

Table 3: Comparison of mechanical properties in support materials

7 Post-processing - FDM

FDM printing has several limitations that the technology has yet to overcome [23], one of which is undoubtedly the aesthetic grade of the printed model. In fact, the FDM printing process, which works by layers, produces a layered and often uneven external printing surface visible to the naked eye. Of course the problem can be managed with thinner layers, but it is not possible to have a completely smooth surface. Another key factor is the removal of print support, no matter how well set, they will always have a negative effect on the surface of our print. Not to mention any imperfections due to excess filament or anything else. The first step we need to consider is, if present, the removal of the print supports. The supports are very useful for printing otherwise impossible overhanging volumes, but on the other hand they leave imperfections on the supported surface.

The most important things to save work in post-production are:

- Use as little support as possible, orienting the print in the most efficient way.
- Set them up properly, trying to keep them as far away from the surface to be supported as possible.
- If necessary, divide the model into several parts and print them separately in an efficient orientation.
- If possible, use soluble materials as seen above.

If you want to improve the surface, mainly for aesthetic reasons, with filament printers there are mainly two methods, physically or chemically.

PHYSICALLY

If you want to do this manually, the first thing to do is to sand. For this you need different grits of sandpaper. The grits work like this: the higher the number, the finer the paper; the lower the number, the coarser the grit [44].

If the work-piece to be sanded has media residues, coarse print defects or points to be levelled, an example of sanding could be:

- 100-200 grit for roughing
- 220-400-600 grit for the "middle" sanding process
- 1000-2000 grit for final sanding
- 5000 grit for polishing

A further tip is to always make small circular movements to remove the layers. In addition, heat sources, such as hair dryers, can be used on materials such as PLA that are very sensitive to heat, but this technique must be used with caution.

If you want to paint the work-piece, the best idea is to start with a primer. This is a special spray paint that often serves to prepare the surface of the material for subsequent painting. In our case it is very useful to even out imperfections from our layers, as the primer has filling properties and is very easy to sand. The primer should be applied in two light passes, with rapid movements so as not to create dripping and possible sanding between coats.

A very interesting alternative to primer is epoxy resins [1]. These resins are bi-component, i.e. they are based on the mixing of two different substances to create, via chemical triggering, a resin that can be used on our print. The procedure is a little more complex than the primer and consists of mixing component A with component B according to the dosages described on the product instructions. Once mixed, the resin has an average time of 10-15 minutes before it hardens, so it is a process that requires some speed.

CHEMICALLY

The main technique is to attack the surface with a substance to which the material is particularly sensitive. Acetone fumes are a very effective chemical sanding technique and can be applied to ABS (for PLA, tetrahydrofuran can be used) and provide a homogeneous surface with a mirror finish [43]. There are now commercially available machines for this technique at home, but they are more expensive. There are therefore more homemade methods that achieve equally good results.

One of these is to place the piece to be treated in a hermetically sealed box. Inside the box, acetone must be poured or material (kitchen paper) impregnated with acetone must be placed. When the box is closed, in about 60 minutes the vapours generated by the acetone will cause micro-fusions on the surface of the piece, the layers will level out and the surface will be smooth.

This procedure is recommended if the objective of our printing is only aesthetics. On tolerance parts or parts that need to be assembled, it is not ideal, as the acetone acts very aggressively on edges and other details.

8 Materials - Photopolymerization

Photopolymerization 3D printers use light-reactive thermosetting materials called photoresists, but are more commonly referred to as "resins". When photoresists, which unlike FDM printing filaments are initially in a viscous liquid state, are exposed to light at a certain wavelength, they polymerise the monomers into solidified geometries [46]. photoresists have the advantage of having a wide range of formulations: materials can be soft or hard, reinforced with secondary materials such as glass or ceramics, or have mechanical properties such as high heat distortion temperature or impact resistance. They range from specific materials such as those for prostheses, to final materials for prototyping, formulated to withstand extensive testing and to perform under stress.

The main ones on the market are briefly described below [50].

8.1 Standard

These types of photoresist are ideal if you want to make detailed parts that must boast high resolution, with both a smooth and matte surface finish.

8.2 Tough

This photoresist has mechanical properties comparable to ABS and is therefore suitable for making fasteners and assembled parts that must withstand compression, elongation, bending and impact without breaking.

8.3 Durable

Durable photoresists on the other hand has properties similar to those of polypropylene (PP), i.e. high impact and wear resistance, being more ductile than Tough photoresists This makes it excellent for making consumer products and moving parts. It also allows you to make prints that have an excellent smooth and glossy finish.

8.4 High Temp

This is basically the highest performing resin available. It has a heat deflection temperature of about 290 °C at 0,45 Mpa, making it suitable for prints related to hot air or gas flows, heat resistant attachments, housings and fasteners or to produce molds and inserts.

8.5 Flexible

This particular photoresist is used to simulate soft-touch materials. Characterized by rubber-like flexibility, it has a high ability to withstand bending, flexing and compression, as well as resistance to repeated cycles without tearing. This is the best choice if you want to make ergonomic models, components for robotics, medical devices and anatomical models. It is also quite successful in the preparation of stage equipment and models for special effects.

8.6 Others

There are also dental and medical photoresists, which represent a wide range of bio-compatible resins for the production of objects such as surgical templates, prostheses and dentures.
Jewelry photoresists are used to produce test pieces, masters for reusable molds or custom jewelry. They are composed of lost wax casting materials and vulcanized rubber molds, being easy to cast while still maintaining intricate detail and good shape retention.

8.7 Comparison

In the table 4, we have listed some of the mechanical and thermal properties of the main photoresists, in order to provide some technical data [17]

	Standard	Tough	High Temp	Durable
Elongation at break	6.2 %	24 %	2.0 %	49 %
Tensile strength	65 MPa	55.7 MPa	51.1 MPa	31.8 MPa
Tensile Modulus	2.80 GPa	2.80 GPa	3.60 GPa	1.26 GPa
Flexural Modulus	2.20 GPa	1.60 GPa	3.30 GPa	0.82 GPa
Price	50-100 CHF/L	100-150 CHF/L	∼200 CHF/L	150-200 CHF/L

Table 4: Comparison of mechanical properties in photoresist printing materials

9 Post-processing - Photopolymerization

Post-processing is necessary to obtain optimal properties from the photoresists [9]. This step allows the printed parts to achieve optimum material properties and the final product will have the desired characteristics without being sticky. Post-processing for photoresists printing requires a little extra care, especially with regard to safety: it is advisable to use nitrile or neoprene gloves to protect the hands and masks, e.g. FPP2, for the respiratory tract due to the solvents and vapours created. In addition, as you will be handling flammable substances, it is strongly recommended that you electrically ground the work table to avoid unwanted electrostatic sparkles.

9.1 Removing the object

When the printing is finished and the object has been printed, the uncleaned and uncured part must be removed from the build plate. It is important to gently remove each print from the platform and be careful not to damage it, as the material is not yet in its optimal state, especially mechanically.

9.2 Cleaning the object

If the printed object is not cleaned after printing, the photoresist on the surface can distort the shape of the model, hardened drips can be found and the part can be sticky. To ensure proper cleaning, it is recommended to rinse at least 4 - 5 minutes in IPA or (Bio) Ethanol, preferably using ultrasound or under agitation.
Ensure that there is sufficient IPA or (Bio) Ethanol to completely cover the part or change the position of the object and follow the cleaning step again. Be careful not to expose the parts to the cleaning solvents for too long (more than 20 minutes) as they may break. After this cleaning step, make sure that the parts are dry before post-curing. You can put the parts in a well-ventilated area for at least 30 minutes or you can also use compressed air for 2 minutes.

9.3 Post-curing in a UV chamber

There are different methods of post-curing: either by simply using light (natural light, UV nail lamp, UV curing stations, do-it-yourself UV curing boxes, etc.) or by using both light and heat. Heat speeds up the process and allows the formation of more complete bonds leading to an improvement in material properties that would be impossible to achieve with light alone.
For small prints, an inexpensive UV nail lamp could be an effective curing tool. However, if you want to maximise the mechanical properties, it is advisable to use a light box with a wavelength of 405 nm (the same wavelength as the lasers in SLA 3D printers), which should generally be purchased.
The duration of the post-print curing process depends on the equipment, the choice of photoresists and the geometry of the part. With the ideal post-press cure settings you can achieve the properties you need in a very short time. For standard photoresists the times range from 0 to 15-30 minutes. If you need materials with stability, stiffness and high temperature resistance, the properties of engineering photoresists improve when you subject the parts to a post-press cure of up to 60 minutes. Biocompatible materials require specific procedures to achieve both biocompatibility and excellent mechanical properties.

10 User interface

Now that we know how the printers work and what materials they use, we can proceed to the actual printing. This section details the steps and different settings needed in order to pass from a 3D model to a physical print. We will first deal with FDM procedure, then we will examine Photopolymerization techniques.
NOTE: The procedures explained next will refer to specific software and specific techniques. The goal of this section is to give an idea of what procedures can be asked from the end-user, and not what each individual button from each software do (because softwares can change).

10.1 FDM Procedure

In order to explain the procedure, we will use the following softwares:

- CATIA V5 as 3D modeling software (used to create the CAD model, [18])
- PrusaSlicer as slicer (used to obtain the G-code for the printer, [42])
- Prusa MK3S as FDM printer ([41])

We begin by creating a CAD model of the part to be printed. The model shown in Figure 6 has been created only for demonstration purposes, and has no mechanical use.

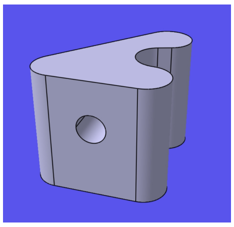
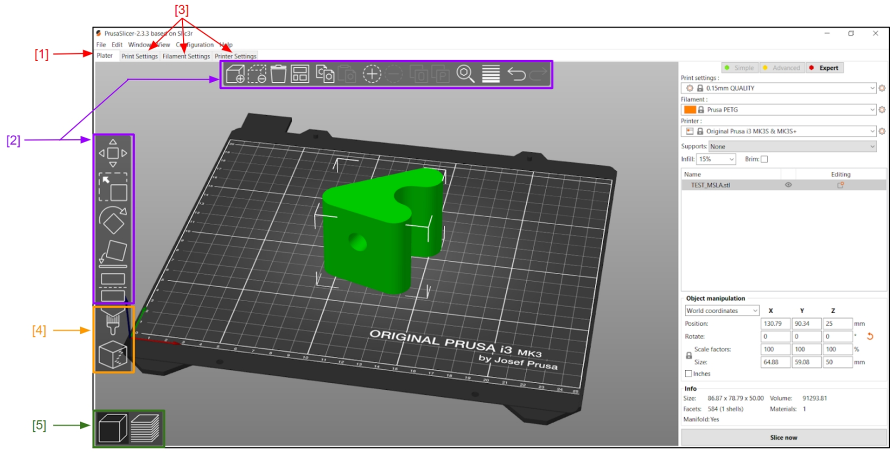


Figure 6: CATIA V5 object example. The object measures approximately 50mm in width

Once the CAD is finished, it is saved under a format acceptable by the slicer software. Generally, .stl is a good choice. We then proceed to import it in the slicer. The slicer view is shown in Figure 7



Print settings : 0.15mm QUALITY
Filament : Prusa PETG
Printer : Original Prusa i3 MK3S & MK3S+
Supports: None
Infill: 15%
Brim: ☐

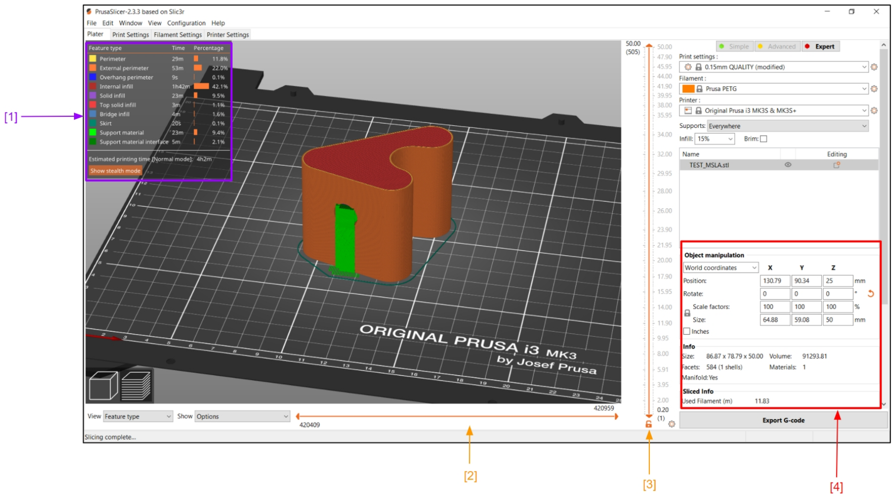
TEST_MSLA.stl

World coordinates
Position: 130.79 90.34 25 mm
Rotate: 0 0 0 °
Scale factors: 100 100 100 %
Size: 64.88 59.08 50 mm
☐ Inches

Size: 86.87 x 78.79 x 50.00
Volume: 91293.81
Facets: 584 (1 shells)
Materials: 1
Manifold: Yes

Figure 7: Screen view of PrusaSlicer in 3D editor view once the .stl model has been imported. The software provides visual rendering of where the part will be printed on the grey bottom plate. Palettes [2] provide tools to move, rotate, scale, generally modify the CAD models. We can also add several CAD parts if we want to, all to be printed simultaneously onto the bottom plate. Finally, there is the possibility to add support structures and modify seam settings using [4]. Also, detailed settings adjustments can be done using [3] buttons.

The idea is that we load all the .stl files we want to print in the slicer. We may want to change some of them in size, scale... We may also want to add support structures not automatically generated. Once we are done with the position of each file on the plate (in our case, it's only one file), we hit the "Slice now" button, that initiates the slicing procedure; at the end of which we get the result of Figure 8.



Feature type	Time	Percentage
🟨 Perimeter	29m	11.8%
🟧 External perimeter	53m	22.0%
🟦 Overhang perimeter	9s	0.1%
🟥 Internal infill	1h42m	42.1%
🟪 Solid infill	23m	9.5%
🟥 Top solid infill	3m	1.1%
🟦 Bridge infill	4m	1.6%
🟩 Skirt	20s	0.1%
🟩 Support material	23m	9.4%
🟩 Support material interface	5m	2.1%

Estimated printing time [Normal mode]: 4h2m

Print settings : 0.15mm QUALITY (modified)
Filament : Prusa PETG
Printer : Original Prusa i3 MK3S & MK3S+
Supports: Everywhere
Infill: 15%
Brim: ☐

TEST_MSLA.stl

World coordinates
Position: 130.79 90.34 25 mm
Rotate: 0 0 0 °
Scale factors: 100 100 100 %
Size: 64.88 59.08 50 mm
☐ Inches

Size: 86.87 x 78.79 x 50.00
Volume: 91293.81
Facets: 584 (1 shells)
Materials: 1
Manifold: Yes

Used Filament (m) 11.83

View: Feature type
Show: Options

Slicing complete...

Figure 8: Sliced model: we can see each individual layer of filament, as well as the scope of each filament (from [1]). We can also get access to key information such as the amount of used filament, and the impression time (from [4]). There are also two cursors ([2] and [3]), detailing respectively the path followed by the printing head, and the layer observed.

Slicing the model converts the solid .stl model into a group of extruded filament. Observing the model from the slicer view allow us to have a visual preview of what will be printed. In order to make this task easier, two cursors are used. Their respective function is shown in Figure 9.

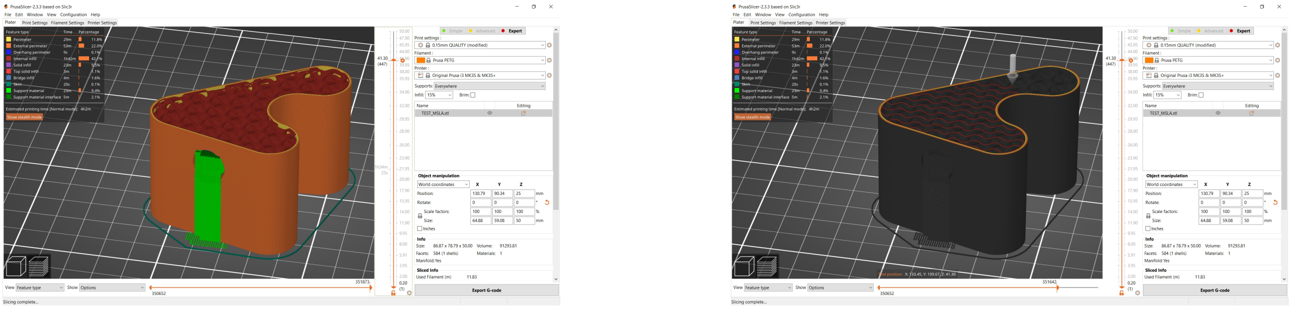


Figure 9: The left image shows the height cursor effect on the model. In that image, the top limit has been set to 41.30mm: we hence see the cross cut of the part at 41.30mm from the plate. As a consequence, we see the internal profile of the part: the infill structure.
The right image allow us to analyse the path followed by the extrusion head when creating the current slice.

Once we are happy with the theoretical printing, we can click "Export G-code", and sent it to the printer in order to print the part. However, before we do that, we should probably have a look at the parameters from [2] in Figure 7.

10.2 Detailed parameters for FDM printer

The previous paragraphs have developped how one takes the CAD parts, positions them, slices them, observes them and eventually export the G-code (the information needed to operate the printer). But what happens if we want to change the sliced version of our part? Some parameters can help to do that.

NOTE: The goal of the next part is not to describe each single parameter, as well as the impact on the final print (and trying to optimize the print quality), as is done in [64]. But rather to give an idea of what can be changed when printing using FDM. If the need to optimize the quality of the printing arises, one can create some test parts in order to understand what are the capabilities of the printer. More details are provided in [14].
Finally, the details about the parameters have been understood also thanks to [39].

The parameters that can be changed when FDM 3D printing are separated in "Print settings", "Filament settings" and "Printer settings" (those are the three buttons indicated by [2] on Figure 7). The filament settings concern the properties of the filament to be extruded (melting point, diameter, weight, cooling needed by the fans...). Those parameters are generally not modified, except when a new type of filament is used (the parameters are not the same when printing ABS or PLA for example).

The printer settings are about the general behaviour of the machine (maximum accelleration allowed, maximum height allowed...). We are not going to examine them in detail.

10.2.1 Most important print settings

Layer height

The layer height defines the thickness of the different layers. The more the layers are thick, the less total layers will be needed. However, the print quality will decrease.

Number of perimeters

Since most parts are printed with some void inside, we define a minimum number of perimeters for a certain part. On Figure 10

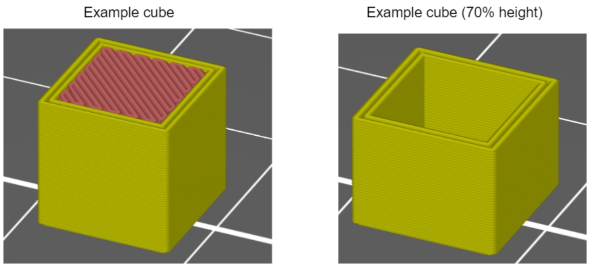


Figure 10: Small example cube; left: uncut, right: cut at 70% height. The cube has been set with a number of perimeters of 2 (hence we see two layers forming the perimeter)

The same parameters are present for the top and bottom layers (minimum number of layers on the top/on the bottom).
There is also the possibility for vase-like objects (objects that can be defined by $r(z, \theta)$, where r is the location of the wall in a cylindrical referential), to print them using a single perimeter, with a constantly growing z (no clear distinction between the layers, the object is built similar to the threads on a screw). This is called the vase mode.

Infill
Infill consist in the material put inside the part in order to fill the void, while still having acceptable mechanical properties. Infill can be managed using the fill density (see Figure 11)

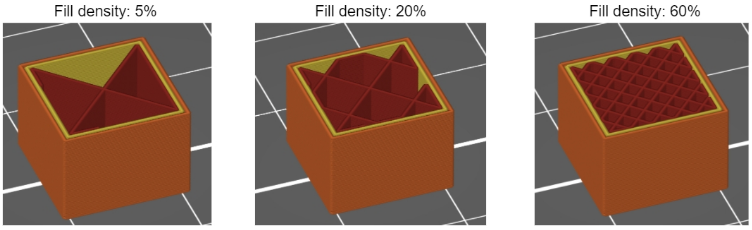


Figure 11: Impact of fill density on a print

The profile used for the infill in Figure 11 is called "grid". One can change the type of profile used for the infill. Some of the possible profiles are shown in Figure 12.

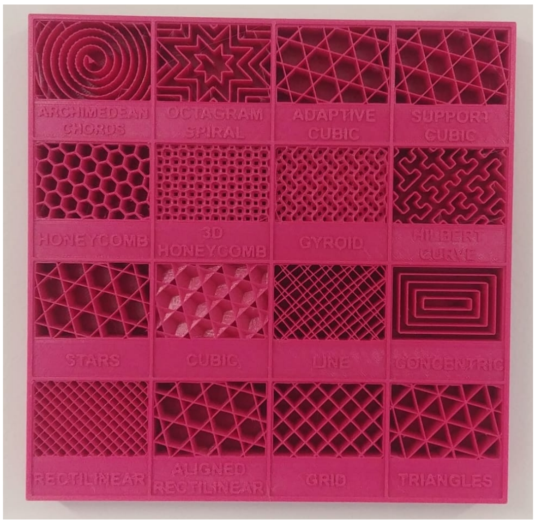


Figure 12: Types of possible profiles to use as infill. This model is exposed at SKIL (EPFL) [45]

Finally, when creating the infill, it is possible to set the print thickness to be different from the one used for the rest of the print (time saving). It is also possible to create after a certain number of layers a solid layer between the infills. For example: if we have a certain space to fill, one can decide to make 1 layer every five to be solid (fully filled). This is typically made for mechanical reasons.

Skirt and brim

The software allows the addition of a skirt and a brim. The brim allows for more surface area to be in contact with the plate, while the skirt is used to make a certain amount of plastic flow through the nozzle before the actual print. Having more surface area in contact with the plate limits the risk of the object getting separated from the plate during the printing. The software allows to customize the dimensions of those elements. For example, one can change the brim size. An example of those two features is shown in Figure 13.

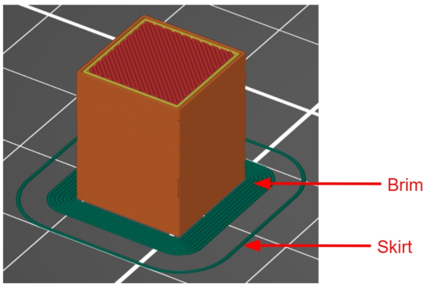


Figure 13: Brim and skirt for a small test cube

Support material

When creating certain types of parts, there may be cases where the printer will have to "bridge" (print mid-air). This case is shown in Figure 14

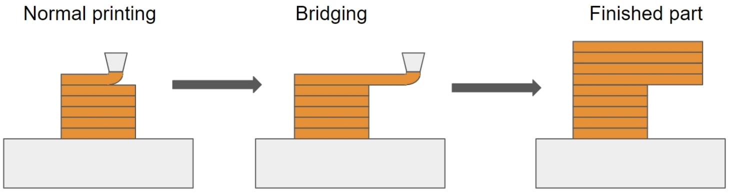


Figure 14: Bridging process. The filament is shown in orange, the nozzle is shown as a trapeze. Typically, when bridging, special care is given to cooling. The idea is to solidify the material as quickly as possible.

While it is possible to operate the printer in such a way, as shown by [25], the more the gap increases, the more gravity will pull down the bridged wire, modifying hence the shape of the object. The general idea when FDM 3D printing is to bridge the less possible.

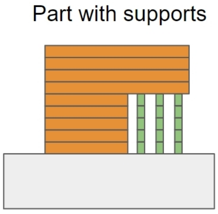


Figure 15: Example of support structures on a finished print (supports in green)

Sometimes, we have to build structures that would need bridging. When this is the case, we generally use support structures. (Figure 15). The idea of those structures is to provide support against gravity for the printing filament.

A more serious example is shown in Figure 16. In such figure, a "L shaped" part is being sliced. The green filaments represent the supports for the part of the L being mid-air. For most domestic 3D printers, the filament support material is the same as the one for the part. A particularity of the supports is that, when printed, they are generally easy to remove; they break off without breaking the part (when using rigid plastics like ABS, PETG...). This however may not be the case when using flexible materials.

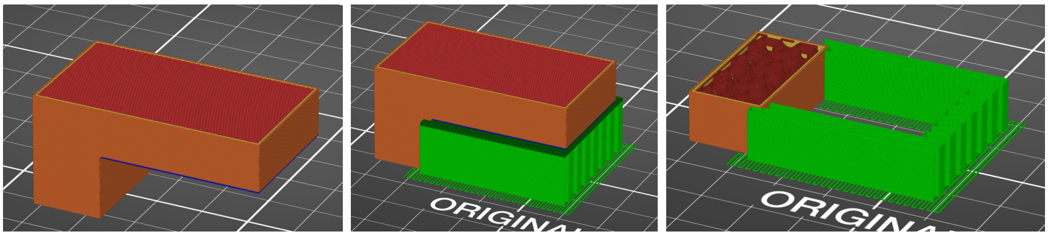


Figure 16: From left to right: test piece with a ledge, sliced without supports; test piece with supports; horizontal cut, showing that the supports are in this case only on the perimeter

When using the slicer, one can specify the exact location where supports are wanted. This can be done using the [4] tools shown in FIgure 7. Also, one can add what is called a "Raft", which is a platform under the model that can help adhering with the printing bed (the bottom plate). Such raft is shown for a test cube in Figure 17.

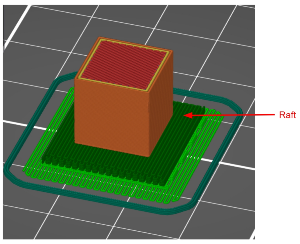
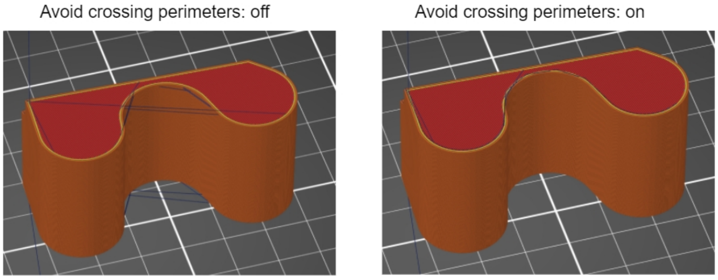


Figure 17: Raft under a test cube. NOTE: the difference between a raft and a brim is that a brim is placed around the base of the model, while a raft gets placed around and under the base.

Other quality parameters

There are some advanced parameters that can help improving the quality of the final solution. One of them is the "Avoid crossing perimeters" setting. If turned on, the software tries to minimize the amount of residuals wires of molten plastic that get dragged round when the nozzle moves outside the part. An example of the impact of such parameter is shown in Figure 18.

Other parameters can for example deal with the detection of very thin walls: if the walls are less thick than twice the layer thickness, the software will not print them, except we turn on the detection for thin walls. In that case, the walls get printed with a thickness of one layer.


Avoid crossing perimeters: off

Avoid crossing perimeters: on

Figure 18: Impact of the "Avoid crossing perimeters" setting. The blue filament represent the residuals wires of molten plastic that get dragged round when the nozzle moves

Additional settings include varying the maximum speed and accelleration of the nozzle, setting multiple extruded materials on the printer (for example [32]), and many others. For some slicing softares ([60]), there is also the possibility to include a texture on the printed elements. In case of objects made using the "vase" option, texturing can help improve mechanical properties ([47]).

10.3 Photopolymerization Procedure

In order to explain the procedure, we will use the following softwares:

- CATIA V5 as 3D modeling software (used to create the CAD model, [18])
- PrusaSlicer as slicer (used to obtain the G-code for the printer, [42])
- Prusa SL1 as photoresist based printer ([41])

Since most (if not all domestic 3D printers) use a top-down approach, we will detail how the design for top-down printing should be done, and what one should èay attention to when using a photoresist printer. Then, we will detail some of the parameters that can be changed on a photoresist-based printer.

NOTE: Even thoug several types of photoresist printing procedures exist (SLA, SLS, DLP, CDLP, MSLA...) the use of the slicer and launching the print is very similar. Hence we will treat the general case for photoresist-based printing.

As exlained in [65], the general way of printing top-down consist in proceeding as shown in Figure 19. The idea is to polymerize the resin from below (step 1), so that a solid layer is created (step 2). Then, the z-table is lifted, and the space between the polymerized part and the printer bed is filled with liquid resin (step 3).

Note that between step 2 and step 3, the part must be separed from the bed; this action requires some force because the part may be sticking to the printer bed.

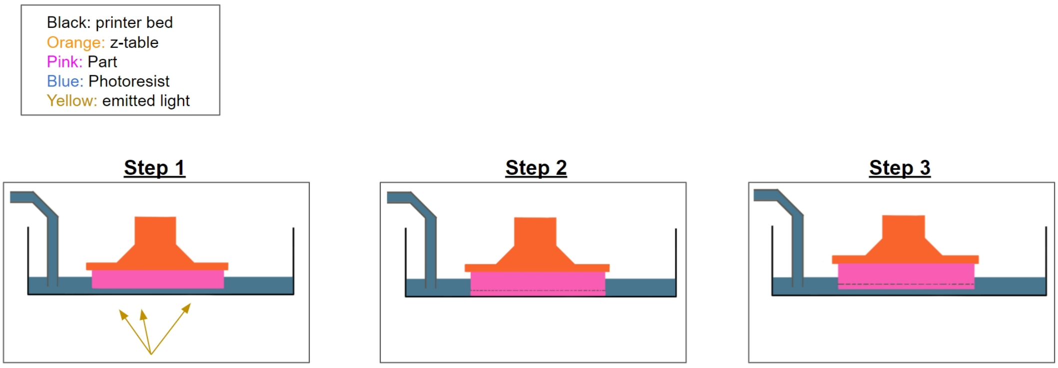


Figure 19: Printing method for a photoresist printer (one layer)

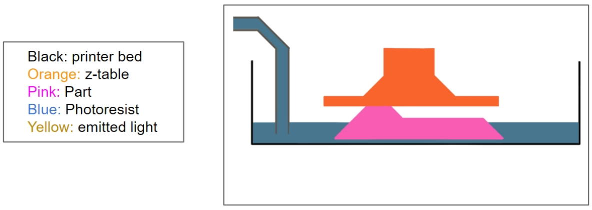


Figure 20: Part that will probably fall from the z-table into the photoresist bath when printed

The adherence between the polymerized plastic and the printer bed can cause problems. For example, if we attempt to print the part shown in Figure 20, the part may want to stick to the printer bed rather than to the z-table; hence the part could fall in the photoresist bath. In order to avoid those kind of events, supports are created. They will help better distribute the stresses during a print. For example, in the case of Figure 20, supports should be added as shown in Figure 21.

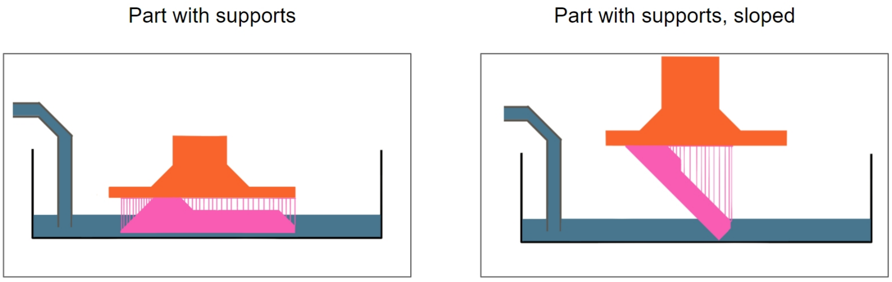
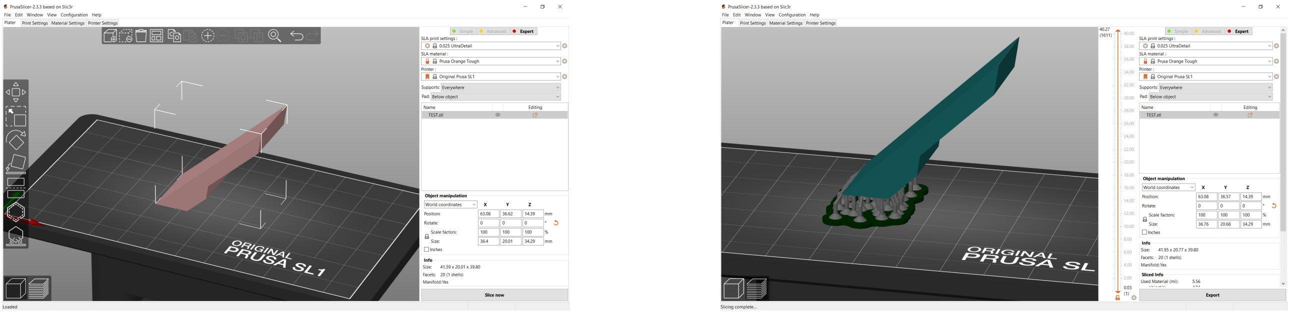


Figure 21: Improved printing for the test part (pink).Adding a slope to the part decreases in this case the surface in contact with the printer bed, hence decreasing the pull-down force.

If we had to implement this print in real life, we would had to print something similar to what is shown in Figure 22.



SLA print settings : 0.025 UltraDetail
SLA material : Prusa Orange Tough
Printer : Original Prusa SL1
Supports: Everywhere
Pad: Below object

Name		Editing
TEST.stl		

Object manipulation (before slicing)

World coordinates	X	Y	Z	
Position:	63.08	36.62	14.39	mm
Rotate:	0	0	0	°
Scale factors:	100	100	100	%
Size:	36.4	20.01	34.29	mm

☐ Inches

Info
Size: 41.59 x 20.01 x 39.80
Facets: 20 (1 shells)
Manifold: Yes

Object manipulation (after slicing)

World coordinates	X	Y	Z	
Position:	63.08	36.57	14.39	mm
Rotate:	0	0	0	°
Scale factors:	100	100	100	%
Size:	36.76	20.68	34.29	mm

☐ Inches

Info
Size: 41.95 x 20.77 x 39.80
Facets: 20 (1 shells)
Manifold: Yes

Sliced Info
Used Material (ml): 5.56

Figure 22: Example of test element on PrusaSlicer. Top image: object before slicing; bottom image: object after slicing. Notice the absence of supports around the part (not needed, since the base is relatively strong). Notice also the absence of "filament": each layer is printed on one iteration. Also, notice the fact that, in the slicer, the z-table is shown upside down compared to the printing location.

Now that we understand a bit better how the printing process work, we can have a look at the different elements that can be changed in the slicer when printing. We will not detail the post-printing processes as they have already been detailed in Section 9.
About the general user interface, we notice from Figure 22, that no big changes are present with respect to the FMD slicer interface. The only things that change is the fact that buttons [4] in Figure 7 now are about the support elements (different from FDM to photoresist printing). Hence, for the user interface display, one can refer to the FDM part.

10.4 Detailed parameters for photoresist printers

As in the FDM case, we will only explain some of the most important settings one can change when printing with a photoresist printer. Just like in the FDM case, we will only focus on the print settings, and not on the material or printer settings. Also, photoresist based prints are not as customizable as FDM, so the number of parameters will be reduced.

10.4.1 Most important print settings

Layer height This parameter is basically the same as for the FDM printer. However, the lenghtscale can be smaller (0.025mm for photoresist based printers, compared to 0.1 for FDM printers).

Supports Due to the different role of supports in photoresist based printing and FDM, the supports have a different shape. Such shape (diameter of the bars, layout, diameter of the base...) can be modified in the print settings. Also, one can decide to add support beams on the part, if they are not generated automatically. For example, a new version of the previous test part is shown in Figure 23

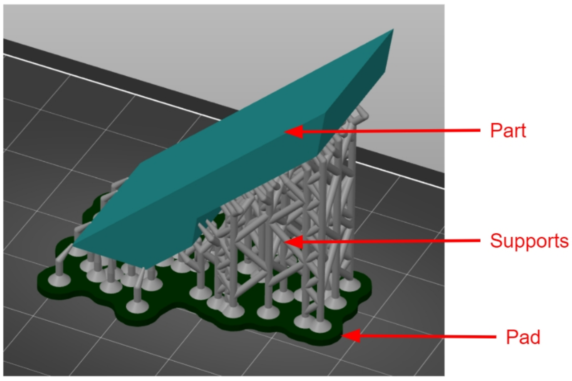


Figure 23: Additional print supports on part from Figure 22

Pad Notice that in Figure 23, a pad is shown. Such pad is there to maximize the contact surface between the part (and supports) and the z-table. One can change its shape (thickness, maximum distance from the object...).

Hollowing Sometimes, we may not want to print completely filled parts (for weight reasons for example). In that case, one can decide to "enable hollowing". That way, the part gets reduced to a shell, with eventual support structures inside. An example is shown in Figure 24.

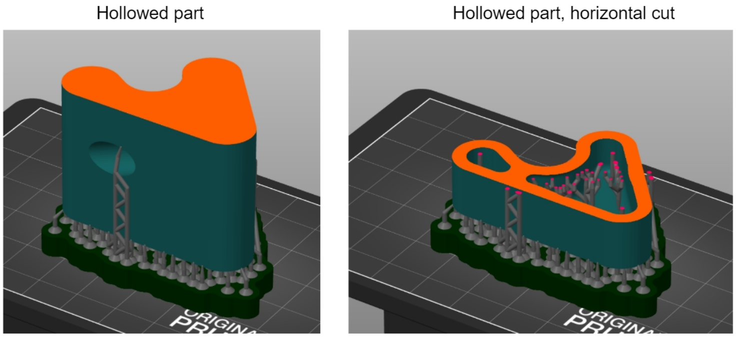


Figure 24: Test part sliced with "enable hollowing" on.

When enabling hollowing, one can change the thickness of the walls and other minor parameters. It is however difficult to change the position of the supports inside the part.

11 Real part FDM printing

Now that we have solid theoretical knowledge about domestic printing techniques, we decided to make a few attempts in printing an object using the FDM printers from SKIL (Prusa MK3S). This was made in order to acquire practical experience, especially with the mistakes that can be done before and during the printing process.

11.1 The idea

Our goal was to design a phone cover (for LG g6 phone). In fact, this would challenge not only the effective capacity of 3D printing an object, but also how well the printer manages to respect the dimensions of a product. When 3D printing a phone cover, it is essential that the cover snaps in place, hence the dimensions must be precise. NOTE: by an economical point of view, it is not worthy to 3D print our own covers (if we include the time used to design the product). Each cover costed in-fact at the end only for the filament a bit less than 4 CHF. We remember that this project was hence only done to acquire practice with 3D printing.

11.2 Version 1

The first version of the cover is shown in Figure 25

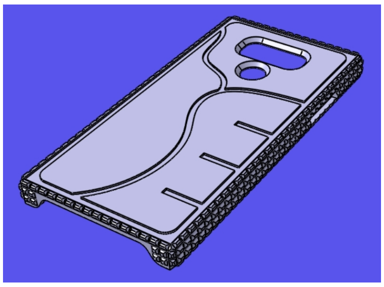
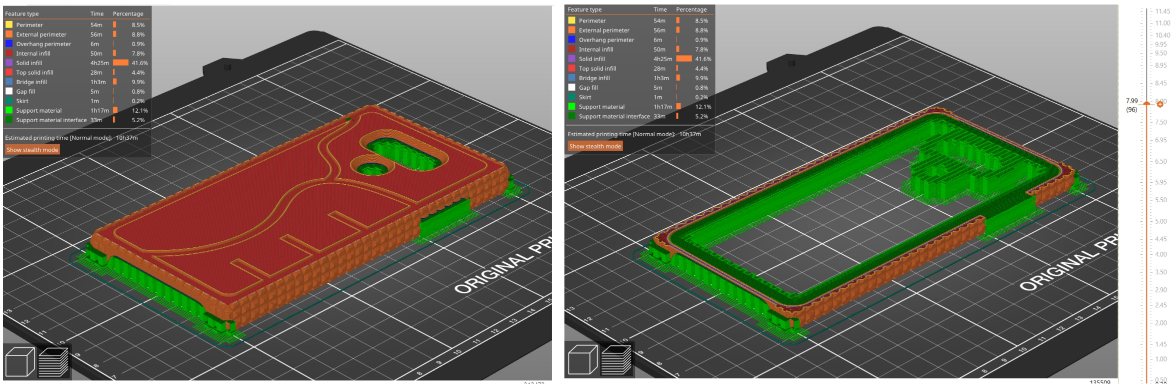


Figure 25: First design for the cover. The planned material of print is PETG

Once the model was uploaded in the slicer software (PrusaSlicer), we obtained the results from Figure 26



Feature type	Time	Percentage
🟨 Perimeter	54m	8.5%
🟧 External perimeter	56m	8.8%
🟦 Overhang perimeter	6m	0.9%
🟥 Internal infill	50m	7.8%
🟪 Solid infill	4h25m	41.6%
🟥 Top solid infill	28m	4.4%
🟦 Bridge infill	1h3m	9.9%
⬜ Gap fill	5m	0.8%
🟩 Skirt	1m	0.2%
🟩 Support material	1h17m	12.1%
🟩 Support material interface	33m	5.2%

Estimated printing time [Normal mode]: 10h37m

Figure 26: Sliced cover. Notice the presence of the support structures, even though those are not present everywhere under the cover

The cover has been placed as shown in Figure 26 in order to be able to print the drawings on the back. In fact, we thought that without any supports, the drawing would have been badly drawn. It turned out this is not the case.

Figure 26 also shows that supports have not been placed everywhere under the cover. We initially thought that it was normal "the software does it automatically, so it must be ok". But we later realized that the software does not exactly pay attention to the fact that filaments inside the print (in that case on the "phone side" of the cover, may be distorted due to bridging). Plus, even after acknowledging that, it was difficult to manually inserting supports inside the cover. Lastly, note that the very small pyramid-like profiles on the side of the covers did not have any support material. That is fine since the printer can bridge for a few millimeters (the profiles have a height of 0.5mm).

During the print, we used a relatively large fill density (50%). In fact, we thought that if the phone did not enter the cover, we had material to remove, without compromising too much the resistance of the cover. Lastly, we launched the print, using a black PETG filament.

NOTE: if one wants a robust part, without caring for aesthetics, a good choice would be to make the part white (without any colored pigments in the filament). Since we did not needed high mechanical performances, we went with black.

At the end of the printing, we had a lot of supports structures to remove. Moreover, filaments composing the interior of the cover (were no supports were present) had deformed towards the ground due to gravity. Moreover, even though the dimensions of the cover matched the dimensions of the phone; no tolerance (neither dimensional or of the machine precision) has been taken into account (we may have had a better result if we took the case to be 0.4mm larger on every dimension).

Due to those inconvenient, we had to remove material using a Dremel ([19]). The final result is shown in Figure 27

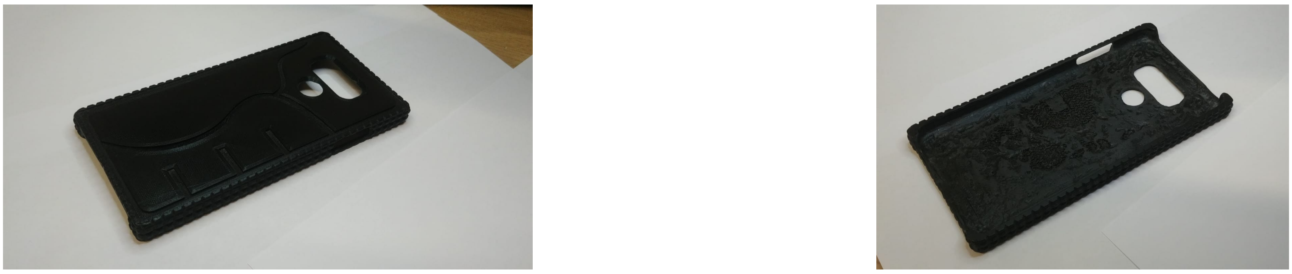


Figure 27: Front and back of the 3D printed cover. Notice the infill profiles on the inside of the cover, that got exposed when carving the part. For the print, we used a gyroid pattern

Except for the bridging problems, the printer had no trouble creating the part. If we had to make the part again, we will make the following modifications:

- Account for tolerances (whole part bigger of at least 0.4mm)
- Print the part with the back of the cover faced downwards (no needs of supports, no bridging)
- Eventually creating test parts such as the one shown in Figure 28, in order to check the tolerance. Those parts are smaller (cheaper and take less time to print), and can give an idea of the tolerances needed.

Figure 28: Test parts, printed each with different tolerances (-0.2mm and -0.4mm), in order to check which one fits best around the phone

After 3-4 weeks, we realized that this cover had also a few defects:

- A small crack has been spotted on one corner of the cover (maybe the material is too brittle?). An image is shown in Figure 29.
- The cover feels hard in the hands; maybe next time a softer material shall be preferred.
- It is hard to access to the volume buttons on the side due to the thickness of the cover

Figure 29: Crack on the phone cover. Notice how the crack continues between two layers (weak spot)

11.3 Version 2

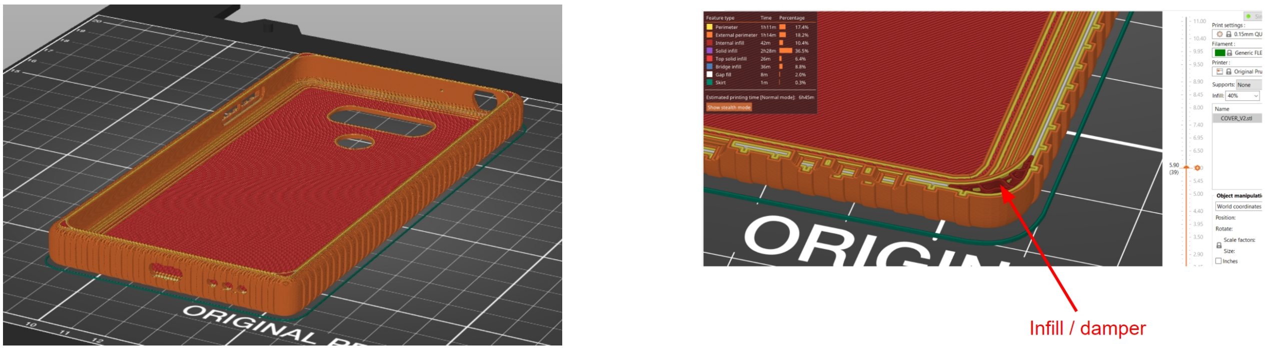
The second version of the cover is shown in Figure 30

Figure 30: Rendering of the front and back of the second phone cover. Notice the fact that the volume buttons have been added as a separate component (orange part)

This time, when uploading the model to the slicer, we placed it on the right direction (back of the cover towards the printer bed). Also, we had to avoid using support structures since, this time, the cover will be printed out of FLEX (generic flexible filament), that does not allow clear support separation. To make the object able to avoid supports, we did the following:

- We made the drawings on the back a bit curved (helps when printing them, avoids bridging)
- We reduced the size of each hole (jack hole, power hole, volume...), and we made each hole rounded in order to help when bridging

The images of the sliced model are shown in Figure 31



Feature type	Time	Percentage
Perimeter	1h11m	17.4%
External perimeter	1h14m	18.2%
Internal infill	42m	10.4%
Solid infill	2h28m	36.5%
Top solid infill	26m	6.4%
Bridge infill	36m	8.8%
Gap fill	8m	2.0%
Skirt	1m	0.3%

Estimated printing time [Normal mode]: 6h45m

Print settings : 0.15mm QU
Filament : Generic FLE
Printer : Original Pru
Supports: None
Infill: 40%
Name: COVER_V2.stl

Figure 31: Sliced model of the cover only (volume buttons are not present). The right image shows a cut of the cover: the 40% infill allows for the creation of a damping zone, that could absorb chocs when the phone is dropped

Before launching the print, we decided to test with a few test models if the tolerances picked were correct or not. Those test models are shown in Figure 32

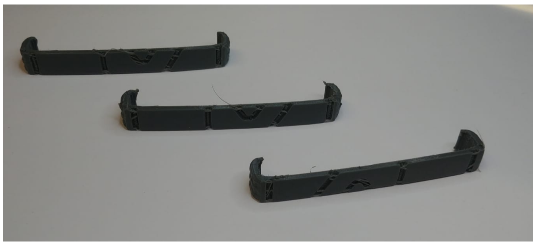


Figure 32: Test parts replicating the width of the phone cover. The three parts are bigger than the original by 0.2, 0.4 and 0.6mm

When printing the parts, we also realized that the Hilbert curve pattern that we planned on doing for the bottom layer (the one in contact with the bed), serving the purpose of creating some texture; actually came out smooth. This makes sense since the bed is heated and hence melts the top layer. An idea of what we wanted to do for texturing the cover using Hilbert curves is shown in Figure 33.

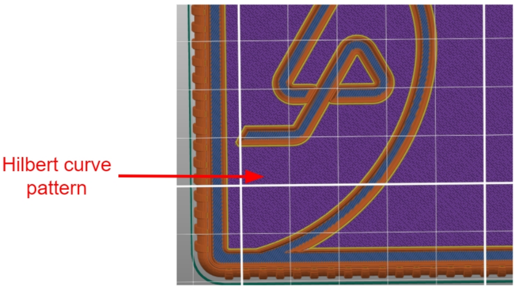


Figure 33: Hilbert curve profile, applied on the bottom layer. We thought it could help give some texture to the product.

For the actual print, we opted to use a textured plate on the printer bed ([40]). This helped add a texture to the final part, making it non-smooth. A few images of the printing process are shown in Figure 34

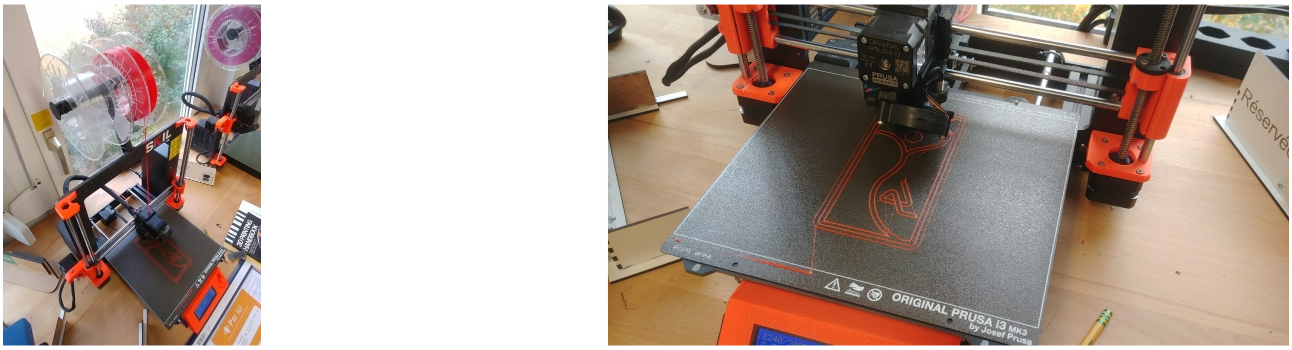


Figure 34: Images of the printing of the first layer of the cover. An orange FLEX filament (A95) has been used. Notice the textured plate on the machine.

Unfortunately, the first time the print has been launched, the cover detached from the bed (around layer 31/74), hence the print had to be stopped and relaunched. However, the same problem occurred again for the second print. Images of the first (incomplete) cover, and the second one are shown in Figure 35

Figure 35: Left image: unfinished cover, Right image: finished cover, even though the corners got detached from the bed

The fact that the print detaches from the bed is bad because now, the finished part is not able to host the phone. The detachment could maybe been avoided if a brim/raft had been used. Also, this phenomenon is more limited if a normal bed had been used (a textured bed decreases adherence).

At last attempt (in order not to throw away the part away), we did the following thing: we cut the material around the corners, inserted some 3D printed triangular FLEX part then used the welder to fix the triangular element in place. A schematic image of the process, as well as the mended cover are shown in Figure 36.

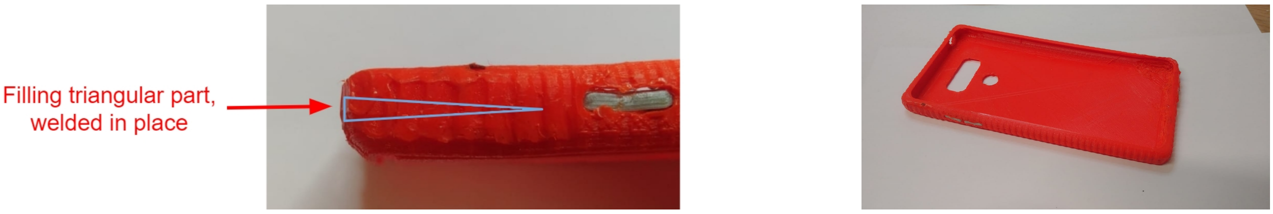


Figure 36: Left image: mending process, Right image: mended cover

The phone ended up by fitting. At the end, if we want to continue printing with flex and texture plate, special attention must be given to part/bed adherence and thermal drifting.

12 Conclusion

This work has allowed us to understand more in details the different aspects of domestic 3D printing, from the technology in use, to the materials, to the user interface. We note that for those kind of procedures, trial and error methods can be useful in order to vary the machine performances.
Of course, for simple prints and coarse tolerances, the standard settings can be good. However, the more the part complexifies, the more studies shall be done on the slicer.

References

[1] 3d printer geeks - can you use epoxy coating on pla printed objects? URL: https://3dprintergeeks.com/epoxy-coating-pla-prints/.

[2] 3d printing spot - are petg fumes toxic? URL: https://www.3dprintingspot.com/post/are-petg-fumes-toxic-what-you-need-to-know.

[3] 3djake - filaments for your 3d printer. URL: https://www.3djake.uk/3d-printer-filaments.

[4] Acmeplastics.com - what is petg? URL: https://www.acmeplastics.com/what-is-petg.

[5] Acrylonitrile butadiene styrene - wikipedia. URL: https://en.wikipedia.org/wiki/Acrylonitrile_butadiene_styrene.

[6] Additive manufacturing technologies: An overview. URL: https://www.hubs.com/knowledge-base/additive-manufacturing-technologies-overview/.

[7] All you need to know about hips as a support material in 3d printing. URL: https://www.3dnatives.com/en/hips-support-240820205/.

[8] All3dp - best dual extruder 3d printers. URL: https://all3dp.com/1/best-dual-extruder-3d-printer/.

[9] All3dp - how to clean resin 3d prints. URL: https://all3dp.com/2/sla-post-processing-best-ways-to-clean-your-resin-3d-prints/.

[10] All3dp - is pla actually biodegrable? URL: https://all3dp.com/2/is-pla-biodegradable-what-you-really-need-to-know/.

[11] All3dp - which one to use for my project? URL: https://all3dp.com/2/3d-printer-support-material-which-one-to-use-for-my-project/.

[12] Applications best suite for abs plasti sheet. URL: https://www.acplasticsinc.com/informationcenter/r/using-abs-plastic-sheeting-compared-to-other-materials.

[13] Best 3D Printers 2021 | Tom's Hardware. URL: https://www.tomshardware.com/best-picks/best-3d-printers.

[14] Best test models for 3d printing | top 3d shop. URL: https://top3dshop.com/blog/best-test-models-for-3d-printing.

[15] British plastics federation - nylons. URL: https://www.bpf.co.uk/plastipedia/polymers/polyamides.aspx.

[16] Comparison of mechanical properties in fdm printing materials. URL: https://www.simplify3d.com/support/materials-guide/properties-table/.

[17] Comparison of mechanical properties in sla printing materials. URL: https://www.hubs.com/knowledge-base/sla-3d-printing-materials-compared/.

[18] Conception et ingénierie | catia – dassault systèmes. URL: https://www.3ds.com/fr/produits-et-services/catia/.

[19] DREMEL® 4000 Outils électriques | Dremel. URL: https://www.dremel.com/ch/fr/p/dremel-4000-4000-4-65ez-v686.

[20] ELEGOO Mars 2 Mono LCD MSLA Resin 3D Printer. URL: https://www.elegoo.com/products/elegoo-mars-2-mono-lcd-3d-printer.

[21] Ender 3 Pro 3D Printer. URL: https://www.creality3dofficial.com/products/creality-ender-3-pro-3d-printer.

[22] Fabrication of pla filaments and its printable performance. URL: https://iopscience.iop.org/article/10.1088/1757-899X/275/1/012033/pdf.

[23] Guida pratica alla post-produzione per la stampa 3d. URL: https://www.binarioprint.it/guida-pratica-alla-post-produzione-per-la-stampa-3d/.

[24] History of 3D Printing: It's Older Than You Think [Updated]. URL: https://redshift.autodesk.com/history-of-3d-printing/.

[25] How far can a 3d printer "bridge"??? URL: https://www.youtube.com/watch?v=m4i1SbIIMaA.

[26] K-mac plastics - abs chemical resistance. URL: http://k-mac-plastics.com/data-sheets/abs_chemical_resistance.html.

[27] Mariano jiménez, luis romero, iris a. domínguez, maría del mar espinosa, manuel domínguez, "additive manufacturing technologies: An overview about 3d printing methods and future prospects", complexity, vol. 2019, article id 9656938, 30 pages, 2019. https://doi.org/10.1155/2019/9656938.

[28] Materiali stampa 3d – la guida completa per stampanti fdm. URL: https://www.binarioprint.it/materiali-stampa-3d/.

[29] Matterhackers - how to dye nylon 3d prints. URL: https://www.matterhackers.com/articles/how-to-dye-nylon-3d-prints.

[30] Nylon - wikipedia. URL: https://en.wikipedia.org/wiki/Nylon.

[31] Original Prusa i3 MK3S 3D printer. URL: https://shop.prusa3d.com/en/3d-printers/181-original-prusa-i3-mk3s-3d-printer.html?search_query=mk3s&results=91.

[32] Original Prusa i3 Multi Material 2.0. URL: https://www.prusa3d.com/original-prusa-i3-multi-material-2-0/.

[33] Original prusa sl1s speed. URL: https://www.prusa3d.com/original-prusa-sl1/.

[34] Personal 3d printers market size, share & growth | forecast - 2030. URL: https://www.alliedmarketresearch.com/personal-3d-printers-market.

[35] Photon Mono X. URL: https://www.anycubic.com/products/photon-mono-x-resin-printer.

[36] Polyethylene terephthalate - wikipedia. URL: https://en.wikipedia.org/wiki/Polyethylene_terephthalate#Copolymers.

[37] Polylactic acid - wikipedia. URL: https://en.wikipedia.org/wiki/Polylactic_acid.

[38] Polyvinyl alcohol - wikipedia. URL: https://en.wikipedia.org/wiki/Polyvinyl_alcohol.

[39] Prusa knowledgebase. URL: https://help.prusa3d.com/en/.

[40] Prusa knowledgebase. URL: https://help.prusa3d.com/en/article/textured-steel-sheet_196534.

[41] Prusa3d - open-source 3d printers from josef prusa. URL: https://www.prusa3d.com/.

[42] PrusaSlicer - Prusa3d.com - 3D printers by Josef Prusa. URL: https://www.prusa3d.com/prusaslicer/.

[43] Rifinitura abs con i vapori di acetone. URL: https://www.italia3dprint.it/guida-rifinitura-abs-con-i-vapori-di-acetone/.

[44] Sandpaper grit charts & grades. URL: https://www.grainger.com/know-how/equipment-information/kh-sandpaper-grit-chart.

[45] Skil. URL: https://www.epfl.ch/labs/skil/en/about/.

[46] Stereolithography - wikipedia. URL: https://en.wikipedia.org/wiki/Stereolithography.

[47] Texturing 3d prints for strength! URL: https://www.youtube.com/watch?v=3-ygdNQThAs.

[48] Top 3D printing technologies 2021. URL: https://www.statista.com/statistics/560304/worldwide-survey-3d-printing-top-technologies/.

[49] Toxicity of abs plastic in 3d printing. URL: https://www.whiteclouds.com/blog/toxicity-of-abs/.

[50] Types of 3d resins. URL: https://www.stampatutto3d.it/resina-3d-guida-completa-con-le-migliori-stampanti-a-resina/.

[51] Wood filament guide: 10 best wood filament. URL: https://printermaterials.com/best-wood-filament-brands/.

[52] Mechanical properties of fdm and sla low-cost 3-d prints. *Procedia Engineering*, 136:257–262, 2016. The 20th International Conference Machine Modeling and Simulations, MMS 2015 September 7-9, 2015, Terchová, Slovak Republic organized by: Faculty of Industrial Technologies in Púcho Alexander Dubček University of Trenčín. URL: https://www.sciencedirect.com/science/article/pii/S1877705816002113, doi:https://doi.org/10.1016/j.proeng.2016.01.207.

[53] Laser SLA vs DLP vs Masked SLA 3D Printing Technology, March 2017. Section: 3D Printing. URL: https://theorthocosmos.com/laser-sla-vs-dlp-vs-masked-sla-3d-printing-technology-compared/.

[54] How desktop 3d printers are transforming 3d printing, June 2019. URL: https://amfg.ai/2019/06/04/how-desktop-is-transforming-industrial-3d-printing/.

[55] Anycubic Photon Mono X Review: Supersize Pick, November 2020. URL: https://all3dp.com/1/anycubic-photon-mono-x-review-3d-printer-specs-2/.

[56] Best 3D Printers in 2021 Buyer's Guide, September 2021. URL: https://all3dp.com/1/best-3d-printer-reviews-top-3d-printers-home-3-d-printer-3d/.

[57] Elegoo Mars 2 Review: Green New Deal, April 2021. URL: https://all3dp.com/1/elegoo-mars-2-review-3d-printer-specs/.

[58] Ender 3 (Pro/V2) Laser Engraver Upgrade: How to Install It, January 2021. URL: https://all3dp.com/2/ender-3-laser-engraver-upgrade-all-you-need-to-know/.

[59] Original Prusa i3 MK3S+ Review: The Best Gets Better, February 2021. URL: https://all3dp.com/1/original-prusa-i3-mk3s-plus-review-3d-printer-specs/.

[60] Admin. Powerful 3d slicer software - ideamaker | raise3d. URL: https://www.raise3d.com/ideamaker/.

[61] Chandra Bortolotto, Esmeralda Eshja, Caterina Peroni, Matteo A. Orlandi, Nicola Bizzotto, and Paolo Poggi. 3d printing of ct dataset: validation of an open source and consumer-available workflow. *Journal of Digital Imaging*, 29(1):14–21, February 2016. doi:10.1007/s10278-015-9810-8.

[62] David Bourell and Joseph Beaman. Chronology and current processes for freeform fabrication. *Journal of the Japan Society of Powder and Powder Metallurgy*, 50, 11 2003. doi:10.2497/jjspm.50.981.

[63] David Bourell, Joseph Beaman, Ming Leu, and David Rosen. A brief history of additive manufacturing and the 2009 roadmap for additive manufacturing: Looking back and looking ahead. *Proceedings of RapidTech*, 09 2009.

[64] Arup Dey and Nita Yodo. A systematic survey of fdm process parameter optimization and their influence on part characteristics. *Journal of Manufacturing and Materials Processing*, 3(3), 2019. URL: https://www.mdpi.com/2504-4494/3/3/64, doi:10.3390/jmmp3030064.

[65] C.Moser Dr. Eric Boillat, Profs. J.Brugger. *Introduction to Additive manufacturing.* Centre d'impression EPFL, 2021.

[66] Rachael Elrod. Classroom innovation through 3d printing. *Library Hi Tech News*, 33:5–7, 05 2016. doi:10.1108/LHTN-12-2015-0085.

[67] Tyler Finnes. High definition 3d printing – comparing sla and fdm printing technologies. *The Journal of Undergraduate Research*, 13(1), January 2015. URL: https://openprairie.sdstate.edu/jur/vol13/iss1/3.

[68] Ian Gibson. The changing face of additive manufacturing. *Journal of Manufacturing Technology Management*, 28(1):10–17, January 2017. doi:10.1108/JMTM-12-2016-0182.

[69] Laura Griffiths. How desktop 3D printers became an essential industry tool, November 2018. URL: https://www.tctmagazine.com/api/content/38bed4c2-ee3e-11e8-897e-120e7ad5cf50/.

[70] Kadir Gunaydin and Halit Türkmen. Common fdm 3d printing defects. 04 2018.

[71] Guilherme Arthur Longhitano, Guilherme Bitencourt Nunes, Geovany Candido, and Jorge Vicente Lopes da Silva. The role of 3D printing during COVID-19 pandemic: a review. *Progress in Additive Manufacturing*, 6(1):19–37, February 2021. doi:10.1007/s40964-020-00159-x.

[72] Amit J. Lopes, Mireya A. Perez, David Espalin, and Ryan B. Wicker. Comparison of ranking models to evaluate desktop 3d printers in a growing market. *Additive Manufacturing*, 35:101291, 2020. URL: `https://www.sciencedirect.com/science/article/pii/S2214860420306631`, `doi:https://doi.org/10.1016/j.addma.2020.101291`.

[73] Aya Mansour, Haleimah Alabdouli, Hend Alqaydi, Huda Al Khawaja, and Waleed Khalil Ahmed. Evaluating the 3d printing capabilities. 2020.

[74] Cephas Mawere. The impact and application of 3d printing technology. *International Journal of Science and Research (IJSR)*, 06 2014.

[75] Aleksa Milovanović, Miloš Milošević, Goran Mladenović, Blaž Likozar, Katarina Čolić, and Nenad Mitrović. Experimental dimensional accuracy analysis of reformer prototype model produced by fdm and sla 3d printing technology. In Nenad Mitrovic, Milos Milosevic, and Goran Mladenovic, editors, *Experimental and Numerical Investigations in Materials Science and Engineering*, pages 84–95, Cham, 2019. Springer International Publishing.

[76] Emily E. Petersen, Romain W. Kidd, and Joshua M. Pearce. Impact of diy home manufacturing with 3d printing on the toy and game market. *Technologies*, 5(3), 2017. URL: `https://www.mdpi.com/2227-7080/5/3/45`, `doi:10.3390/technologies5030045`.

[77] P. Sevvel, D. Srinivasan, AJ. Balaji, N. Gowtham, V.G. Kalyana Varadhan, P. Kumaresh, and M. Kishore Bajrang. Design & fabrication of innovative desktop 3d printing machine. *Materials Today: Proceedings*, 22:3240–3249, 2020. 2nd International Conference on Materials Manufacturing and Modelling, ICMMM – 2019, VIT University, Vellore, 29th - 31st March 2019. URL: `https://www.sciencedirect.com/science/article/pii/S2214785320320642`, `doi:https://doi.org/10.1016/j.matpr.2020.03.284`.

[78] Brent Stephens, Parham Azimi, Zeineb El Orch, and Tiffanie Ramos. Ultrafine particle emissions from desktop 3d printers. *Atmospheric Environment*, 79:334–339, 2013. URL: `https://www.sciencedirect.com/science/article/pii/S1352231013005086`, `doi:https://doi.org/10.1016/j.atmosenv.2013.06.050`.

[79] Jinghai Yi, Ryan F. LeBouf, Matthew G. Duling, Timothy Nurkiewicz, Bean T. Chen, Diane Schwegler-Berry, M. Abbas Virji, and Aleksandr B. Stefaniak. Emission of particulate matter from a desktop three-dimensional (3d) printer. *Journal of Toxicology and Environmental Health, Part A*, 79(11):453–465, 2016. PMID: 27196745. `arXiv:https://doi.org/10.1080/15287394.2016.1166467`, `doi:10.1080/15287394.2016.1166467`.