Additive Manufacturing

The technology of the future

Develop a component using additive manufacturing processes safely

---

Name: Mohammed Ghalib Al-Hosni

Number: 1615

Class: 7A

Abstract

The following report showcases the research and design process of a product manufactured with the use of additive manufacturing technologies. The product chosen for said task is a charging cable holder, a functional product that is useful to the consumer as it eases cable management processes, a product that requires being cheap, dimensionally accurate, having a smooth surface, and a relatively decent underside grip. After the product has been chosen, a 3D model was created using a CAD software, said model was created to assist with the manufacturing process as well as being a good way of thoroughly analyzing possible models, which thus lead to singling out the optimal design for the product. After which, the material that is going to be used for the final product was chosen, with said material being PETg, it was chosen due to the attributes it is able to provide of a relatively strong, cheap, and durable polymer. From that, due to printer availability, the Ender 3 Pro was chosen was chosen as the printer that is going to carry out the printing process, with that, thorough analysis of the printer's specifications were done to figure the strength of the components and possible weakness that could be improved, which leads to then applying possible improvements to the printer's stock components. Two improvements were done the printer, with them coming of the form of component replacement such as the nozzle and the printer's bed, which were both replaced with a slightly bigger nozzle and a glass bed that was originally magnetic, said changes allowed for an overall better printing process in terms of both speed and failure prevention. Form then onwards, the structural integrity of the printing process was analyzed in order to prevent possible deformations such as shrinkage, warping, and issues related to overhangs. Said issues were afterwards resolved by using the optimal bed temperature, applying a base to the print which assisted in warping prevention, and the usage of the preferred support for the print as well carrying out the print with the product being in its optimal print orientation.

In terms of the manufacturing process, the first, the method in which 3D models are transferred to the 3D printer to initialize the printer process was touched upon, with the method being changing the model's file into an STL file, after which processing said STL file through a slicer software, which is a software the slices the model into each layer that the printer is going to be print. After which, the health and safety of the printer's user was taken into account, with the only significant point being that they have to check the temperature of the nozzle or bed before physically working on either of them, which can be done through the slicer software or the provided LCD screen on the printer. Before starting the printing process, the parameters that were going to be utilized were all taken into account and analyzed in order to use the optimal print settings relative to the requirements of the product, said parameters being the layer height, infill percentage and pattern, temperature, and the print speed,. Finally, possible finishing techniques such as sanding, flame treating, and epoxy coating were looked upon, with the one that was chosen being sanding due to the fact that it is able to significantly improve the texture of the product while still meeting the requirements of the product needing to be cheap, which is something that epoxy coating lacked in and flame treatment was too risky of a process that had high potential of ruining the product.

After the product has printed, it was noticed that the printer was able to achieve optimal dimensional accuracy with the utilized print settings however the print speed at around 40 minutes was relatively slow. In terms of the final product, it functions as intended, however, it was concluded that the overall speed of production needs to increase if the product were to be manufactured at a large scale.

Contents table

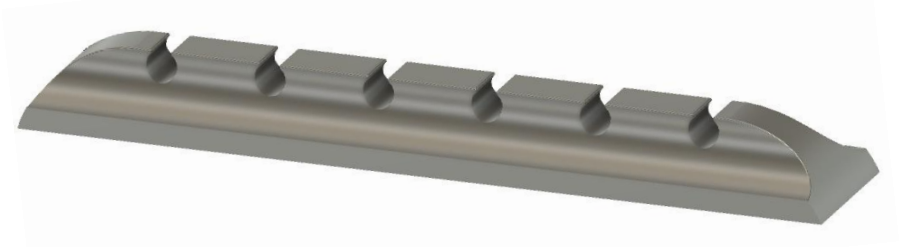
1.0 *Designing process*.....2
- 1.1 *Model image*.....2
- 1.2 *Application and function*.....2
- 1.3 *Dimensions*.....2
- 1.4 *Product requirements*.....2
- 1.5 *Design justification*.....3
- 1.6 *Material of choice*.....4
- 1.7 *Utilized printer and its specifications*.....4
- 1.8 *Analysis of major specifications*.....5
- 1.9 *Structural integrity*.....6

2.0 *Printing process*.....8
- 2.1 *Print initiration*.....8
- 2.2 *Health and safety*.....8
- 2.3 *Print parameters*.....9
- 2.4 *Post-processing*.....10
- 2.5 *The printed product*.....11
- 2.6 *Finishing the product*.....12
- 2.7 *Potential optimizations*.....12

3.0 *References*.....13

Designing process

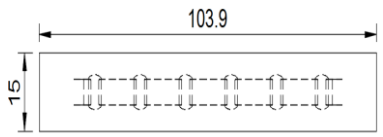
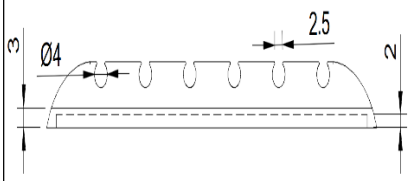
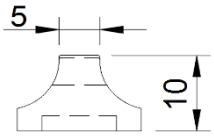
Charging cable holder



Application and function

The product's prime function is for it to be able to straighten wires and keep hold of them which thus preventing them from getting tangled around each other, which thus helps with protecting the cables from potential damage that could occur due to them being disorganized. As such, the product can be either used as a way of holding wires in place for easier access to said wires whenever the user requires them with the other potential purpose for the product being that it can be used as a means to ease the wire management procedure for certain application such tidying any utilized thin cables for phone charging set-ups or possibly computer cable management.

Dimensions

Top	Front	Side
		

Product requirements

Dimensional accuracy

Although it might not seem evident at first, the product does almost necessitate being dimensionally accurate, that is due to the fact that if it isn't, there would be a high probability of the cables to not be held in place as well as what might be expected of the product, causing cables to – as stated – get tangle around each other. Dimensional tolerances of the printer itself shouldn't be of major importance due to said tolerances being relatively small for them to have a significant effect on the product. However, accurate measurements of the average cable diameters would have to be done from the manufacturer's end to ensure a good grip on the majority of charging cables.

Cheap

Since the idea of the product non-essential and of relative basicness, potential consumers would expect it to be also be relatively cheap. As such, one of the main points that should be taken into consideration in terms expenditures is to try to keep it as cheap as possible while still maintaining sufficient overall production quality. The main sector in which money would be spent is the material of choice for the product, therefore it should be the main focus when trying to decrease the price of the product.

Good underside grip

The product should have a decent grip on the surface in which it is going to be placed on. That is the case since in the case the user should ever accidentally push the product softly, it should not fall off or get thrown a far enough distance to cause an inconvenience. Adding on the stated note, the product should be able be stable no matter the type of surface it is being placed on, whether that being glass, wood, ceramics, metal, or any other material that the user could potentially have the product situated on.

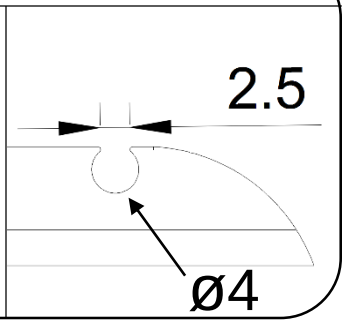
A smooth surface

Due to the design of the product being rather simplistic, for it to be aesthetically pleasing shouldn't be the main focus in terms of the design, material, or the finishing technique to be utilized. However, the aspect that should be worked out is the surface quality of the product in terms of the tactile feel and how smooth the surface is. That is the case since it would be attracting and preferable to the consumer if when the product is touched it was smooth and didn't cause any discomfort due to a rough and uneven surface.

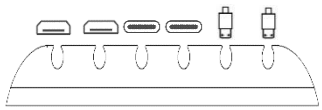
Design justification

The size and expansion of the cable insert opening

At the top of the product, the size of the hole in which the user inserts the cable is relatively small at around 2.5mm, even somewhat smaller than that of the diameter of the average cable itself, which – although differs from one type to another, which is usually around 2 to 3mm. However, although the size might be a bit smaller, even if a small amount of force is applied on the cable, it should be able to be forced in with minimal effort and without damaging the cable, as well as that, it also makes it so the cable can't be pulled out easily if the user or anything else unintentionally pulls on the cable faintly. As can be seen in the dimensional figures, after the initial opening at the top, the size of the hole expands to 4 mm, and this is to allow for slightly more freedom in terms of moving the cable after it has been inserted if the user wants to pull the cable to get it closer to them or shorten it. It also prevents an excess amount of friction to be borne on the sheathing of the cable, which after certain period of use, it could potentially damage it.



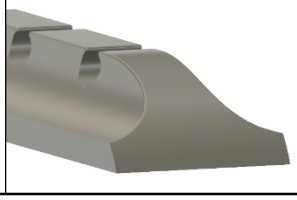
Amount of cable slots



As can be noticed in the design, the amount of cables that is possible to be inserted into the product is six. The reason for said number is so that if the user requires it, they are able to place at least two of every major type of phone charging cable, with said types being Micro USB, USB-C, and Apple's lighting charging cable. If the aforementioned variety in cable types is not required, having the extra amount doesn't have many significant drawbacks other than potential size prerequisites, and it would only be beneficial if the user ever becomes in need of them. As well as that, six cable slots ought to be enough for a charging set-up that a family might require.

Rounded design

The edges and sides of the product are rounded which assists with two goals for the design, the first being that is helps make the product like more aesthetically appealing to the consumer, that is the case since rounded edges look smoother and give off a tidy and – to a certain extent – modern look compared to that off straight edges and also provides a better tactile feel to the surface of the product whenever the consumer touches it. As well as that, it prevents a potential safety risk of the possibility that the consumer could harm themselves accidently due to the sharp edge that would be on the straight edges.



The gap on the underside of the product

As shown in diagram alongside this section and the dimensional figures, a gap has been situated on the underside of the product, the reason for which it is made is that it allows for the addition of a rubber pad by adding the possibility of joining one with the use an adhesive to the gap, thus following the stated product requirements since adding a rubber pad to the underside of any product is known to be an extremely and relatively simple method of applying a decent grip, which consequently assists in preventing the product from sliding easily.

The overall shape of the product

The design that has been decided for the product is for it to be rectangular in terms of the overall shape. Said shape has been chosen over other geometrical design possibilities such as for the product to be in a half circular shape since it allows for the product to be easily joined to the side of an object such as a table or a wall while still sustaining its practically. As well as that – personally speaking – the design is more aesthetic appealing compared to other design considerations that have been looked upon.

Material of choice

PETG

As has been mentioned by (Frey, 2021), PETG is known for being a reasonably cheap material with it being at around 20 USD per kg. The stated characteristics of the material is extremely helpful in terms of following the product's requirements by being able to keep the overall cost of the product at a relatively very cheap price. Another major point to the material is the fact that it is requires lower temperatures to work with compared to other popular materials utilized for 3D printing such as ABS, that is the case since – as stated by (Kondo, 2021) – ABS requires for the bed to be heated to temperatures of over 100°C, while PETG only demands around 70 to 80°C, making it far easier for the manufacturer work with and 30% cheaper energy wise, which would be of major importance if the product were to be produced in large quantities. Along with that, as stated by (TWI, 2021), PETG has extremely good layer adhesion, meaning the possibility of any issue occurring due to the deposited layer not sticking to the previous one is rare, thus also assisting with making the overall printing process for the manufacturer much easier in comparison to certain other types of filament that require much higher temperatures to be met for consistent layer adhesion success. The filament is also characterized for being relatively flexible, as such, this property should help if the wire were to be a bit bigger than the insert opening, along with that the material is also strong, thus allowing for the material to be able to sustain more damage and be more impact resistant. In addition, according to (Dwamena, Is PLA UV Resistant? Including ABS, PETG & More, 2021), PETG is more UV resistant relative to both PLA and ABS, which consequently means UV-resistance-based finishing techniques should not be taken into account as much as they should be for the aforementioned material, therefore decreasing the overall finishing expenses. The material is also known for being environmentally friendly due to it being recyclable as stated by (TWI, 2021). Additionally, the material can be used for food-based products, however that quality can't be of use for our product. Lastly, a major advantage of the filament is that it is offered in a multitude of colors, thus allowing for a greater freedom in choice.

On contrast to the stated advantages of the material, it does nonetheless bear certain drawbacks, with the prominent ones being that the material absorbs water fairly easily due to it having extremely poor water resistance as mentioned by (TWI, 2021), which causes the material

Utilized printer and its specifications

Creality's Ender 3 Pro

The printer available in possession at the moment is the Creality's open source Ender 3 Pro. The printer initially released in March of 2018, and since then it has been known for being one the most sought after 3D printers on the market by 3D printing enthusiasts, engineers looking to produce functional prints and product, cosplayers needing to print certain cosplay items such as masks or outfit parts, and even be people that have never touched the 3D printing space before. That is mainly due to it being an overall good quality printer that is being offered at a relatively very cheap price. As well as that, during 5 or so years, 3D printing in general has seen a major increase in consumers, and with Creality's Ender 3 Pro being one of the more notable printers, and the predominant face of the FDM printing space, the printer has been seeing a very massive increase in its market demand.

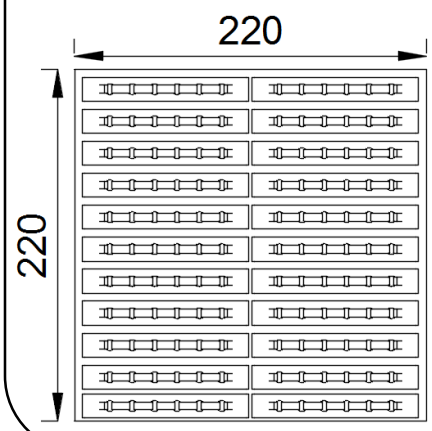
Specifications

Cost	$209
Printer size	440x440x465mm
Build volume	220x220x250mm
Stock bed type	*Magnetic bed*
Max. bed temp	110°C
Max. nozzle temp	260°C
Stock nozzle diameter	0.4mm
Max print speed	200mm/s
Stock extruder	*Aluminum bowden extruder*

Analysis of major specification

Build volume

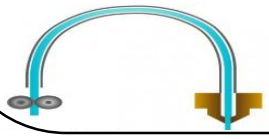
Due to the given build volume of the Ender 3 Pro, the printer allows for the print of around 22 full sized products at once during one cycle. This can be useful in the case the user requires more than one product due to the given six cable slots that comes with product not being sufficient to meet their needs. As well as that, the relatively big volume compared to the size of the product allowed us to be able to print a plethora of smaller test prints at once. This was really helpful since it allowed us to test all the seemingly realistic diameters and sizes for the cable insert opening and the hole in which the wire would sit on that we initially thought would be suitable for the product, which thus allowed us to find the optimal size for said sections in a rather quick manner.



The use of a Bowden extruder

The Ender 3 Pro utilized a bowden extruder, which is type of extruder in which the component itself is situated on a separate compartment from the print head, with the filament then being directed into the nozzle with the use of a tube. With the other type of extruder being a direct drive extruder, which – in contrast to bowden extruders – is placed right on top of the nozzle. The fact that the Ender 3 Pro functions using a bowden extruder is a major advantage, that is the case since as stated by (Fargo 3D Printing, 2017), due to the placement of the bowden extruder not being on the nozzle, this lifts a lot of weight that the print head would have had to move around with, thus consequently increasing the overall print speed and potential accuracy, allowing for the same quality of print to be done at a lesser amount of time compared to printers utilizing direct drive extruders, with the only major disadvantage to bowden extruder being that they consume more power as they push the filament a greater distance thus requiring more torque, they and are also aren't able to print flexible filaments as easily as direct drive extruders due to the potential of flexible filaments not moving properly through the tubing. However, for the purpose of the product and printing with PETG, neither disadvantage should be of any major inconvenience.

Bowden extruder



Direct drive extruder

Heating

In terms of PETG's heating requirements, the Ender 3 Pro shouldn't run into any issues, which is the case due to the fact that – as stated by (Sarcevic, 2021) – PETG only requires between 220°C to 265°C for the nozzle's temperatures and 70 to 80°C for the bed's, with the Ender 3 Pro being capable of easily achieving said temperatures. As well as that, over or under heating issues shouldn't be of concern as temperature parameters can be easily accessed and controlled through the slicer software or the LCD screen located on the printer.

Modified components

0.4 to 0.6mm nozzle

The stock nozzle that initially comes in the package with the Ender 3 Pro is a 0.4mm nozzle; a good average for both print quality as well as print speed, since the major difference between one nozzle diameter to the other is the quality-to-speed ratio, with smaller nozzles being able to provide better quality due to them having the ability to utilize lower layer, while bigger nozzles can print the same exact print sizes that smaller nozzles would, albeit much faster, which is a result of bigger nozzles being able to deposit wider layers, thus allowing for a section that would generally take ten print head movements to be done in around three or four, thus decreasing the time consumed for prints by a significant amount, with prints that would normally take 2 hours being done in less than 30 minutes. Although, as stated by (Zuza, 2018), larger nozzles create larger supports which are thus harder to remove and consequently cause more damage. However, when taking the product into consideration, it can be understood that it doesn't necessarily require being aesthetically appealing due to it being a functional print rather than a decorative one. As such, for the sake of saving time and lowering chances of print failures, a bigger nozzle would be utilized, specifically a 0.6mm one since it would be able to notably decrease the time while causing minimal damage that would be easily negated during the finishing process.

Magnetic to glass bed

One of the major components that directly affect failure chances as well as how well the first layer of the print adheres is the bed. With that being said, the magnetic bed that initially cam installed with the printer has been replaced with a glass bed. According to (Obudho, 2020), the main advantage to said change being that although the speed in which the bed heats is relatively slower, it is much more even due to how heat spreads through glass. Which evidently causes bed adhesion to be much better throughout its entirety as well as having a much lower risk of a failure. Additionally, glass beds make it so it is easier to remove prints off the bed after the printing process has concluded due to the utilized spatula used for the removing the print being able to slide easier on the it. However magnetic beds are flexible which in certain cases makes it so the user is able to remove the print off the bed by simply bending it, which might be much faster however if the bed adhesion was too strong there is a potential for the print to break if said method was used.

Structural integrity

Laminar filament flow

Laminar flow is a term used in fluid dynamics that is used to describe a very smooth movement of fluid that has little to no disturbance. In terms of 3D printing, for a nozzle to produce a laminar flow of the molten filament means that the said filament would be directed exactly towards the intended area with no potential of any spillage to occur due to the movement of the fluid during its descend towards the bed or the print's previous layer. As stated by (Hosch., 2021), laminar flow is generally possible and occurs within systems or processes that involve the flow of the fluid within a small channel. As such, when specifically considering the flow of filament through the nozzle as well the distance between the nozzle the and the previous layer, which are both relatively small, it can be deduced that printer should be able to produce a laminar without any major issues, with the exception of failures related to the z-axis, in which case said error would have to be investigated and fixed. As well as that, to achieve a laminar buildup in that all layers are set evenly with none of them being extended more than others, the printer needs to be set at a relatively slow speed in order to minimize potential accuracies on the x and y axis caused by quick nozzle movements.

Shrinkage prevention

According to (Dwamena, 2021), the optimal method of insuring prints do not shrink or at least have minimal shrinkage is to utilize the optimal bed temperature recommended for the material that is being used. Which in the case of PETG is – as stated earlier – 70 to 80°C, said temperature should be relatively easy to control through the slicer software. As well that any major cooling sources such as air conditioners should be kept away from the printer or turned off. That is the case since stated by (Rosli, Shuib, & Ishak, n.d.), this assists in minimizing the temperature difference of the molten filament compared to the one deposited at the bed; thus resulting in a decrease in the overall applied thermal stress, and consequently minimizing the risk of misshaping since is the main cause for material related deformations such as warping or shrinkage.

Cooling fan requirements

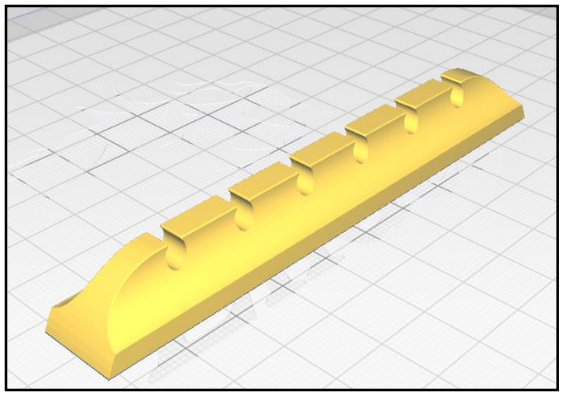
Rapidly cooling a deposited layer with the utilization of the cooling fan or fans is known for generally providing a better surface quality in a lot of cases by preventing warping and material stringing issues. However, according to (Dwamena, 2021), PETG has better printing results when printing without having a fan blowing on the deposited filament, that is the case since it helps significantly with layer to layer adhesion. Although not using fan would definitely damage the overall print quality, it would decrease the chances of print failure by a lot. And as stated earlier, said print damages could be easily be negated during the finishing process, which should be a better overall solution than having the product not print as intended or just entirely fail to print.

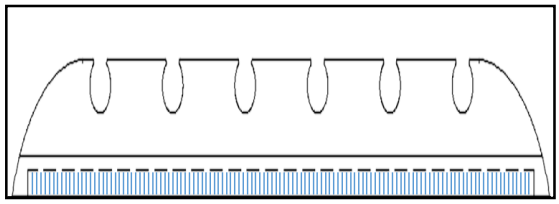
Support material

Support materials are platforms created to support overhanging section of a print, with the general rule being that they are a necessity for any overhangs that are of over 45°. Support material have a crucial role in ensuring a print doesn't fail, however the process of removing them after the printing process has concluded is can be tedious at times depending on thickness of the material and diameter of the nozzle used, as well as being time consuming, with the biggest drawbacks to applying supports being that they damage the surface they were placed n when they are removed, creating a rough and uneven surface. In addition to that, supports are considered wasted materials, which thus means that decreasing the amount of supports utilized is almost always preferable, which can be done by changing the orientation of the product that is going to be printed so that the orientation it is placed in has the least amount of overhangs. As stated by (Chakravorty, 2021), there are mainly two types of supports materials that 3D printers can utilized, with said types being tree-like support structures and linear supports.

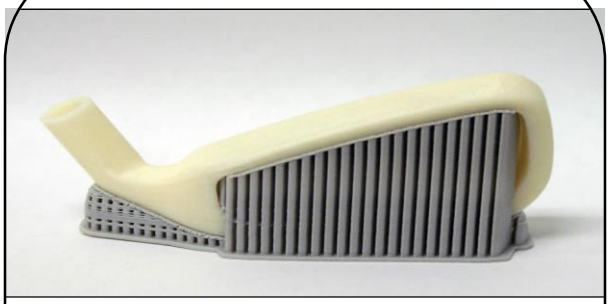
Optimal print orientation

The best orientation in which the support material necessitated is kept to a minimum while also the significant damage done to the product is relatively low can be achieved by simply having the product standing upright in the same orientation as was shown in the model showcase image at the beginning of the report. That is the case since the only support material that would be required for said orientation would be only in the gap made for the rubber pad, which is a location that wouldn't affect how smooth the surface of the product is to the user since said section should be inaccessible to the user without the them intentionally removing the rubber pad. The area would still nonetheless have to be smoothen out to ensure optimal adhesion of the pad to the product, since the uneven surface created by the removal of the supports could potentially affect that. Nevertheless the amount of finishing that would be required would be very little as well as easily done due to the fact that the area being finished is a flat surface, thus meaning it should be of any difficulty to get an even finish throughout the entirety of the affected area.



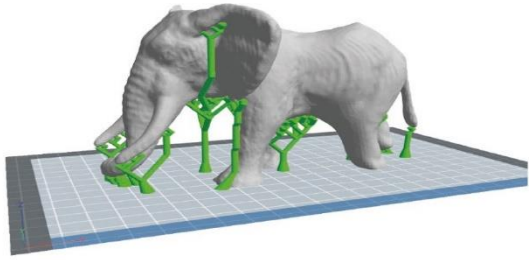


Linear supports



Linear support structures, sometimes also known as accordion structures, are multiple vertical supports that cover the entirety of the overhang as well as the area the supports are emerging from. Which thus causes double the damage to the surface quality due to said this type of structure causing damage to both the overhangs they are supporting and the base at which they were printed. These type of supports are also the ones most widely used due to them being relatively easy to create in terms of the technical stand point, as such they're seen in every slicer software, which is something that can't be said about tree-like support structure, however, Cura, the most widely used slicer software does support tree-like structure as a feature that the program possesses. In direct to contrast to tree-like structures, linear supports are much harder to pull due to the mentioned reasons, although, they are mechanically capable of supporting flat overhangs with angles of 180°.

Tree-like supports



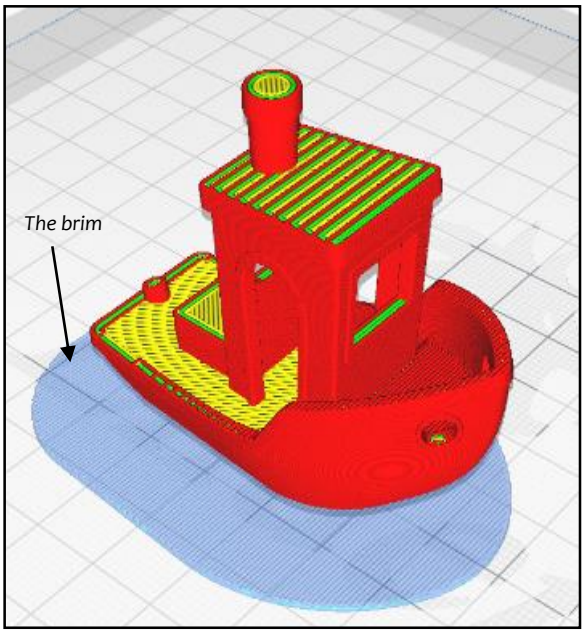
Tree-like supports structures consist of a support beam that moves vertically towards the overhang, while it is being printed, multiple branch-like structures are also printed from the beam itself, and emerging towards the ceiling they are going to support. The overall shape of the structure looks like a tree due to the branching out of the multiple supports, hence the name. This type of support structures are known for being much easier to remove than linear ones, while overall requiring a fewer amount of supports to be able to sufficiently hold the overhang, because although multiple of the aforementioned beams are placed, most of they are spread throughout the print rather than being next to each other. Thus consequently results in less damage being done to the surface of the print. However, due to the geometrical shape of this type of support structure, its mechanical properties makes it so it is unable to support any flat overhangs such us bridges.

Optimal support structure type

Due to the orientation in which the product is going to be printed at, the only section of the print that would require support material would be the gap in which the rubber pad would be placed in. Said gap is a flat surfaced area, which thus means that tree like support structures are not capable of supporting the material. As such – although generally worse due to them causing more damage to the surface – linear support structures would have to be the support material type to be utilized.

Warping

Warping is a similar deformation process to shrinking, in that they both occur due to the thermal stress applied on the material, however, warping only effects the edges of the initial layers that have deposited onto the print bed by causing them to curve upwards and lift up from the bed, which could affect the aesthetic appeal of the product, and in certain cases its functionality. The method generally used to full negate warping is to initially print a base known as the brim in which the rest of the print is going to be printed on. Doing this makes it so the warps instead of the print, thus keeping the main structure of the print safe, after which, it can just simple be removed as if it were supports. As well that, using the appropriate bed heating settings help with decreasing the overall damage of the warping since – as stated earlier – this decreases the difference between the temperature of the plastic at its molten stage and the temperature after it has been deposited and solidified, thus lowering the applied thermal stress. However, the base of the product is relatively solid and even if it were to warp slightly, it wouldn't affect the function of the product, therefore a brim would not be utilized in order to decrease the overall material consumption of the print.



Printing process

Print initiation

File conversion

Before the printing process is commence, the file in which the 3D model of the design that is intended for printing has to first initially be exported into an STL file, which can simply be done within the exporting option that almost all 3D modeling softwares have including Fusion 360, which is the software that was utilized to make the model of the product. Said STL file is a file form that changes a 3D model into a plethora of connected triangles that make up the final shape, which is a type of file that 3D printers are able to understand and work with along with PLY, OBJ, and many others, however, STL has been the standardized file form since 3D printing began as such it would be the one to be utilized.

The Slicer Software

However, converting a 3D model into an STL file is not enough to start the printing process, but rather the STL file converted model has to first be cut into slices that make up the product, which can be done with the use of a slicer software, with the thickness of said slices determining the thickness of each layer that is going to be deposited. The slicer software is also the program utilized to control any of the print parameters and settings such as the temperature, support material, scale up or down the model, and more. As well as that it can be used as an alternative to the printer's LCD screen to initiate, pause, and end the printing process. After the model has been sliced, the printer can just simply then be connected to the computer in which the slicer is being used to start the printing process. In certain types of 3D printers the sliced model could also be transferred into the printer with the use of a flash drive or an SD card, with the printer being controlled with the use of the provided LCD screen, which can be used as an alternative to a computer connection.

The Ender 3 Pro is – as stated earlier – an open source printer, meaning it can utilized any non-private software as such, the slicer software that is going to be utilized is Cura, which is known for being the main open source slicer software that has been unanimously agreed upon among the community as the preferred slicer, which is due to the magnitude of the options that the software provides. As such it will be the program to be used for the printing process of the product.

Health & Safety

Potential Safety Hazards

In terms of FDM printing, there are two major sectors in which potential safety hazards arise, and thus several health and safety precautions needs to be taken in order to provide and sustain an overall safe work environment. The issue first being the heat emitted from both the nozzle as well as the bed. As stated earlier, the heating that is going to be used for PETG is 235°C for the nozzle and 75°C for the bed, which are both temperature that could easily cause severe damage to the skin. As such, said components should never be touched during the printing process. And before handling either component after the process has concluded, the user should initially make sure that their temperature is low enough to the point where operating on them with bare hands would not cause any harm, which can be easily deduced by examining the temperature on either the LCD screen located on the Ender 3 Pro or by checking the slicer software. Therefore, due to the mentioned issues, the printer should kept out of reach from any children or pets, especially since the utilized printer does not have any protection guards closing off the build area from its surroundings.

The second issue that commonly occurs and unfortunately many people ignore is the fumes that are emitted by the filament as it is melting, with this issues being even more severe with the Ender 3 Pro due to it – as stated earlier – not having guards to be able to contain any emitted fumes. According to (Stone, 2021), fumes emitted by PETG aren't harmful due to component making up the filament being unharmful to humans in that most types of PETG are seen as 100% food safe, with certain studies even concluding any carbon emitted from the filament does not present any harm that could lead to significant health risk. As well as that, no odor is emitted from the aforementioned carbon emission, as such – although it is generally recommended for FDM 3D printing – ventilation of the room in which the printing process is going to be done is not necessary if the only utilized material for the print is PETG.

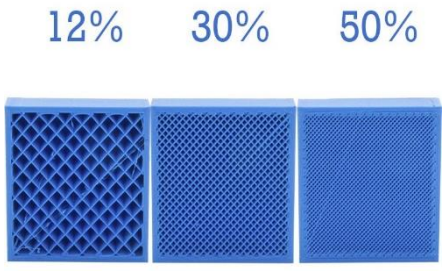
Although not as significant of an issue, it should still be kept in mind. In that if the utilized spatula for removing the print off the bed after the printing process has ended was made out of metal, the removal of the print should be done slowly and with care as to not damage the print and to also prevent chances of the user accidentally harming themselves by hitting their hand with the spatula.

Print parameters

Infill

Infill – as the name states – is the amount of material that is utilized to fill the inside of the print and support its outer structure. With lower infill percentage meaning the inside of the printed product is going to be more hollow, and vice versa for higher infill percentage. Thus printing a product a product with a higher infill percentage indicates that it is going to be stronger and more impact resistant due to it being more solid, while prints with a relatively low infill could potentially break and snap easily. There are around 14 different infill patterns that cura supports, which determine shape that the filling material is going to take when it is being printed, with different infill patterns having different mechanical attributes with each serving a specific purpose.

According to (Goldschmidt, 2021), the optimal infill pattern for functional prints is a cubic or octet pattern. As well as that functional prints require an infill percentage of over 50% to be able to be sturdy and resist potential impact damage that is going to be caused upon them while they are being used. However, although the product we're intending to print is a functional one, it does require the strength that a typical functional print would due to it only being used to hold charging cables, however it should still be capable of withstanding a decent amount of force before breaking. As such, the infill pattern that is going to be utilized is going to be one that any standard print would need, which is – as stated by (Goldschmidt, 2021), a grid-like pattern. In terms of the infill percentage, again, due to the print not requiring the strength that a typical functional one would, the optimal percentage that would minimize material consumption while still being sufficient in terms of preventing the product from snapping fairly easily would be a percentage of 40%, since it's just shy of being what's recommended for functional prints, thus not requiring less filament, but still being capable of providing comparable results in terms of strength.



Layer height

The layer height parameter represent the thickness of each layer that is being deposited and is directly responsible for the provided resolution in the vertical axis of the print. Utilizing bigger layer heights means that the overall print quality will be much lower due to the difference between one layer and another being fairly visible, which causes the surface quality to be aesthetically unappealing and rough. However, the increase in layer height would results in a decrease in print time due to less layers needing to be laid down for the overall print.

According to (Dwamena, 2021), the chosen layer height before causing potential damage to the print or the nozzle should be around 25 to 75% of the nozzle's diameter, which is 0.15 to 0.45mm for the utilized 0.6mm nozzle. With regard to the product, the optimal layer to be utilized is 0.2mm, which is the case since a low layer height helps with achieving an overall smoother surface, which is one of the requirements of the product. The reason for the chosen layer height being 0.2 rather than the lowest possible one which is 0.15 is just simply to avoid any risk of damaging the nozzle or print which would then lead to having to reprint the product.

Temperature

As mentioned earlier and as stated by (Sarcevic, 2021), PETG – the utilized material for the product – demands 220 to 265° from the nozzle to melt properly and 70 to 80°C for the bed. Thus, when considering the printer's capabilities, the preferred temperature settings would be around a nozzle temperature of 235°C and 75°C for the bed, since both of said temperature settings should act as a sufficient middle ground for providing good layer and bed adhesion.

Feed and Travel rates

The feed rate parameter controls the filament extrusion speed, in that increasing the feed rate causes more filament to be fed to the hot end of the printer, which thus causes more material to be deposited onto the bed or previous layer. Increasing the feed rate would increase the overall print speed, however it would also significantly impact the surface quality of the print due to the likelihood of more material than what is required being deposited, which increases the chances of said material to get compressed, thus creating uneven and rough surfaces. The travel feed rate works together with the layer thickness, in that it controls the planar resolution of the print by determining the print head's x and y-axis motor speed, while the layer height controls that of the z-axis. Just like the feed rate, increasing the travel feed rate causes the overall surface of the print to be rough, in that any edges that the print would not be even due to the inaccuracy of the print head due to it moving quickly, which consequently causes some layers within said edges to extend further compared to the ones below or above them.

Both the feed rate and travel food rate are both parameters that work proportionally, in that one were to be increased the other also has to, due to the fact that increasing one would also necessitate for the other to increase as well, if not this would cause the filament to not flow as intended whatsoever, as such there is one parameter used to represent both of them which is the overall print speed. As stated by (Ditsch, 2021), the maximum speed possible for the Ender 3 Pro to achieve is 200mm/s, as such, the print speed that is going to be utilized is around 40% of that, at around 80mm/s. That is the case since both the dimensional accuracy as well as that, the surface quality would be negatively affected if a fast print speed were to be chosen.

Post-processing

Finishing

Finishing is the any process done after a product has finished being manufactured, and is done to improve either the aesthetic appeal of the product or its physical or chemical attributes. It is known for being one of the crucial section when producing any product, and is one of the main points of consideration in research and design projects.

Necessary process: Support removal

The first post-processing procedure that has to be done in the case of almost every 3D print that contains them, is the process of removing any support structure that have been utilized for the print, since – in most cases – the product would not be able to function with them still being attached to it, and no other finishing techniques could be applied to the print due to the fact that the supports would be blocking certain sections of the print that would have to be finished. Generally, there are two method of support removal, the first being removing the support with the use of a combination of pliers and pulling the supports with your hand, with the second being utilizing a special separate material for the support structures known as PVA, which is a material that can be dissolved by placing it in hot water, making the support removal process fast and deals very minimal to no damage to the surface quality of the print, however, to utilize a base material as well as a separate one for the support requires for the printer to have the functionality of utilizing multiple filament by containing slots of two or more nozzles and extruder, which is something that the Ender 3 Pro does not possess. As such, the first stated method of removing the supports by hand with the assistance of pliers is the one that is going to be utilized.

Enhancing processes

Sanding

Sanding is a process done with the use of a gritted cloth known as sandpaper. Said sandpaper is then rubbed on the surface of the product, which evens outs the area it is applied on. Nevertheless, the sandpaper's grit level – which determines how rough the gritted surface is – needs to be taken into account before starting the sanding process, since if a very rough sandpaper were to be utilized, the likelihood of it damaging the surface and scratching it is really high. And if a smooth grit were to be used, it wouldn't harm the product, however it would not improve the surface quality, thus just simply wasting time. The main advantages of using sanding is that the process is fairly cheap, with a pack containing around 50 sheets of sand paper generally costing around 10 USD. However, on the other hand, the entirety of the sanding process, although effective and is able to produce smooth surfaces, it takes relatively a lot of time, especially if it were to be done by hand.

Heat treating

Heat treatment is done with the use of any heat source, generally a flame thrower. Said flame thrower is heated to a temperature that is equivalent or slightly to that of the plastic's melting point, after which, the produced flames are blown onto the print. Ideally, this would the outer layer of the material to melt, thus filling and gaps caused by the layer lines, resulting in a very smooth and even surface finish. However, the major issue with this finishing is process is that it carries a high risk with it, in that the flame were to be held onto the print for too long, there is a high chance of it deforming due to the removal of multiple layers, which in dimensionally demanding print, said deformation could deem to be non-functional. As well as that, if the process were to be applied unevenly, both the overall aesthetics of the print and the tactile feel would worsen due to the uneven surface that would be formed.

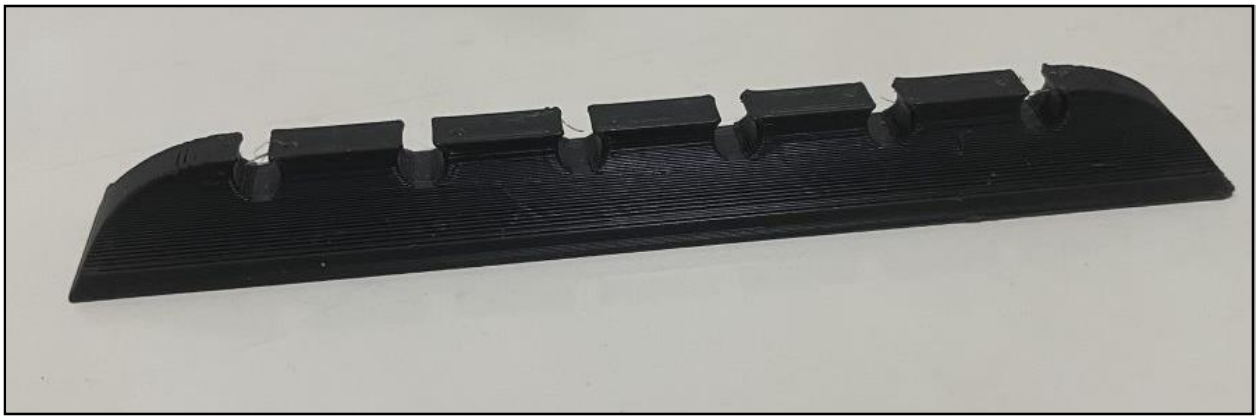
Coating

This finishing technique is the process of coating by brushing the surface of the print with the use of an epoxy material. Epoxy in general is known for being one of the best materials for tasks that require the filling of gaps on the surface of a product since it flows smoothly into tight spaces, said point is of extreme importance especially to 3D prints because the quality of fitting into said tight spaces allows for the epoxy to be able to flow easily between layer lines gaps and filling them, causing the overall print to be extremely smooth, with some type of epoxy also leaving a glossy surface finish. As well as that, epoxy adheres extremely well and is able to maintain its strength after it has been joined to the product, as stated by (Family Handyman, 2018). However, the major downside to epoxy is the fact that it is generally very expensive, at around 15 dollars for only 6 ounces according to the listen amazon prices for the XTX-3D epoxy resin, one of the popular resins usually utilized for 3D printing.

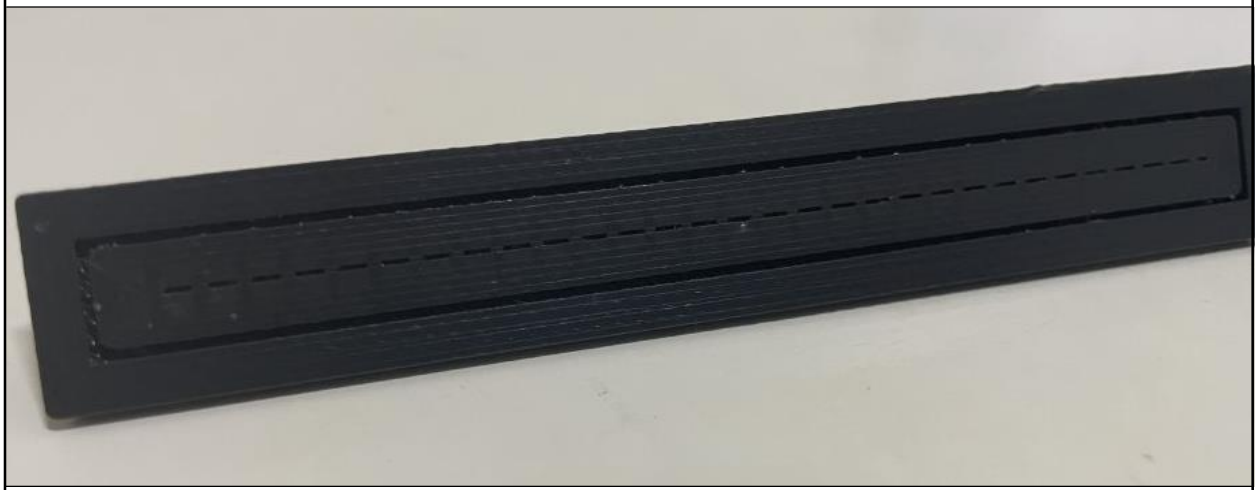
Conclusion

The best finishing technique that is able to improve the aesthetic appeal and tactile feel of the product while also not going against of the product's requirement is sanding. That is the case since although epoxy resin is probably the optimal finishing process to be utilized in order to enhance the product as much as possible and majorly increase the overall quality, the demanded price for the finishing technique is a major drawback that goes against the requirement of the product being cheap by significant margin, thus it cannot be utilized. On the other hand, heat treatment carries a lot of risk if the material were to be heated for longer due to the fact that – as stated earlier – it could easily cause the product to deform and not function properly, as well as that, the process requires for the person that is going to conduct the it to be experienced and very knowledgeable in terms of how it is going to be done on different types of materials. However, when looking at sanding, the process is slow, however it does not carry many significant issues besides that compared to the other finishing techniques stated, as such, sanding is going to be the finishing process that is going to be utilized for the product.

The printed product



Supports



Due to the way in which the supports were printed, there is very little space for removing them, and thus even if it were to be favorable to remove them by hand, it would have to be done with pliers, which should be an important point of consideration for subsequent prints due to the fact that removing supports with pliers, especially in a circumstance similar to this in that distance between the supports and the actual product is very small, can be very risky since if the person removing said supports were to not be careful they could very easily damage the product. As well as that removing supports by hand is a much faster method of doing so relative to removing them with pliers.

Impressions on accuracy and speed

The overall dimensional accuracy of the print compared to the digital model were extremely pleasing. Said impressive accuracy was mainly due to utilized print speed of 80mm/s. However, the print speed was relatively slow, at around 40 minutes, and thus contains some room for possible improvements.

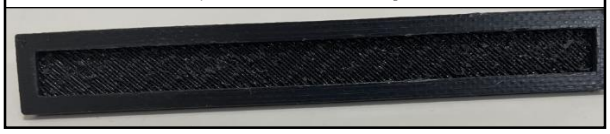
Section	Print	Model
Overall Dimensions	103.8x15x10	103.9x15x10
Rubber pad hole	10.2x0.8	10.2x0.8
Cable opening	2.7	2.8
Hole diameter	4ø	4ø

Finishing the product

Finishing requirements

The main areas that require finishing are the top of the product, specifically the sections that separate the cable openings, that is the case since they are the only parts that are relatively rough. As well as, slight stringing occurred on said locations, which would have to initially be removed. With regards to the underside however, it wouldn't necessarily require any finishing to be done to it, that is the case since when the supports are going to be removed, although the surface is going to be rough, it would still be flat enough for the rubber pad to adhere properly, as such no further finishing would be done to it since no significant benefits would come from finishing the section, and it would only result in wasted time and effort. In terms of the sandpaper requirements, the grit that is going to be initially started on is a grit of 120, after which it is gradually going to be replaced until 600 grit sandpaper is used. Said grit was utilized since it offers sufficient smoothness for 3D printing in general as stated by (Evans, 2017).

Before the rubber pad



Addition of the rubber pad

Before sanding

After sanding

Potential optimizations

The rubber pad hole

As stated earlier, this support structures that were made to assist in printing the rubber pad hole were extremely close to the edges of the print itself, which led to a relatively difficult time when removing them, in order to fix said issue, the gap utilized for the hole could be made deeper in order to make so it is easier to insert pliers between the supports and remove them. Another way of solving the issue is to print the product upside down, this would ultimately damage the surface quality more significantly, nonetheless the supports would be much easier to remove.

Increasing the print speed

As mentioned previously, the total print time for the product was around 40 minutes, which in terms of a singular print isn't necessarily bad, however if the product were to be mass produced the print speed would have to drastically increase to try and meet with customer demand. This could be done by increasing the overall print speed to around 120-150mm/s. The layer height could also be increased, which can be done by using a larger nozzle, possibly a 0.8mm diameter, however that could potentially sacrifice a lot in terms of the print quality.

Final verdict

To conclude, although the product can possibly be sold and produced at a large scale, the manufacturing process at the moment is very inefficient, with heavy focus on the overall production time per product. A better and more professional sanding job – potentially with the use of a sanding machine – would also certainly benefit the final product. However, in terms of the products function, it works as it suggest and has no issues in that regard.

References

Chakravorty, D. *(2021, November 5).* 3D Printing Supports – The Ultimate Guide. *Retrieved from all3dp: https://all3dp.com/1/3d-printing-support-structures/*

Ditsch, K. *(2021, May 3).* Best Ender 3 (Pro/V2) Print Speed Settings. *Retrieved from all3dp: https://all3dp.com/2/ender-3-print-speed-all-you-need-to-know/#:~:text=The%20maximum%20print%20speed%20for,rather%20than%20a%20print%20move.*

Dwamena, M. *(2021, December 15).* How to Get the Perfect Print Cooling & Fan Settings. *Retrieved from 3dprinterly: https://3dprinterly.com/how-to-get-the-perfect-print-cooling-fan-settings/*

Dwamena, M. *(2021, December 3).* Is PLA UV Resistant? Including ABS, PETG & More. *Retrieved from 3dprinterly: https://3dprinterly.com/is-pla-uv-resistant-including-abs-petg-more/*

Dwamena, M. *(2021, December 15).* PLA, ABS & PETG Shrinkage Compensation in 3D Printing – A How to. *Retrieved from 3dprinterly: https://3dprinterly.com/pla-abs-petg-shrinkage-compensation-in-3d-printing/*

Dwamena, M. *(2021, December 16).* Which Layer Height is Best for 3D Printing? *Retrieved from 3dprinterly: https://3dprinterly.com/which-layer-height-is-best-for-3d-printing/*

Evans, D. *(2017, January 29).* Ultimate Guide to Finishing 3D Printed Parts. *Retrieved from fictiv: https://www.fictiv.com/articles/ultimate-guide-to-finishing-3d-printed-parts*

Family Handyman. *(2018, December 17).* How to Use Epoxy Resin Like a Pro. *Retrieved from familyhandyman: https://www.familyhandyman.com/list/how-to-use-epoxy-resin-like-a-pro/#:~:text=Epoxy%20is%20one%20of%20the,loose-fitting%20joints%20in%20furniture.*

Fargo 3D Printing. *(2017, November 30).* Advantages and Disadvantages of Direct and Bowden Extrusion. *Retrieved from fargo3dprinting: https://www.fargo3dprinting.com/advantages-disadvantages-direct-bowden-extrusion/*

Frey, S. *(2021, October 27).* PETG Filament: Best Brands of 2021 – Buyer's Guide. *Retrieved from all3dp: https://all3dp.com/1/petg-filament-3d-printing/#:~:text=MatterHackers%20Build%20Series%20PETG%20is,material%20from%20a%20reputable%20company.*

Goldschmidt, B. *(2021, December 5).* Best Cura Infill Pattern For All Your Needs. *Retrieved from all3dp: https://all3dp.com/2/cura-infill-patterns-all-you-need-to-know/*

Hosch., W. L. *(2021, December 15).* laminar flow. *Retrieved from britannica: https://www.britannica.com/science/laminar-flow*

Kondo, H. *(2021, July 25).* PETG vs ABS: The Differences – Simply Explained. *Retrieved from all3dp: https://all3dp.com/2/petg-vs-abs-3d-printing-filaments-compared/*

Obudho, B. *(2020, July 7).* 3D Printer Glass Bed: What to Consider & Which to Buy. *Retrieved from all3dp: https://all3dp.com/2/3d-printer-glass-bed-what-to-consider-which-to-buy/*

Rosli, A. A., Shuib, R. K., & Ishak, K. M. (n.d.). Influence of Bed Temperature on Warpage, Shrinkage and Density of Various Acrylonitrile Butadiene Styrene (ABS) Parts from Fused Deposition Modelling (FDM). *Retrieved from scitation.*

Sarcevic, M. *(2021, Febuary 28).* Best PETG Print Settings – Simple Guide. *Retrieved from all3dp: https://all3dp.com/2/petg-print-settings-how-to-find-the-best-settings-for-petg/#:~:text=PETG%20will%20typically%20print%20at,will%20damage%20the%20hot%20end.*

Stone, W. *(2021, September 16).* Are PETG Fumes Toxic? What You Need To Know. *Retrieved from 3dprintingspot: https://www.3dprintingspot.com/post/are-petg-fumes-toxic-what-you-need-to-know*

TWI. *(2021, December 3).* WHAT IS PETG? (EVERYTHING YOU NEED TO KNOW). *Retrieved from twi: https://www.twi-global.com/technical-knowledge/faqs/what-is-petg*

Zuza, M. *(2018, June 7).* Everything about nozzles with a different diameter. *Retrieved from prusaprinters: https://blog.prusaprinters.org/everything-about-nozzles-with-a-different-diameter_8344/*

"You also have to acknowledge as an inventor or innovator, it's not about the product, it's not about the technology, it's about the team – picking the right ones that form the right culture, not necessarily your culture, but a healthy culture."

-Scott Crump

Inventor of FDM printing technologies