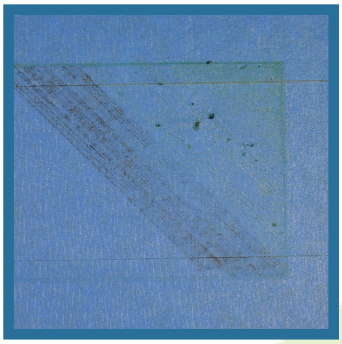
Your professional
3D Printing supplier

优塑
YOUSU

Printing Guide

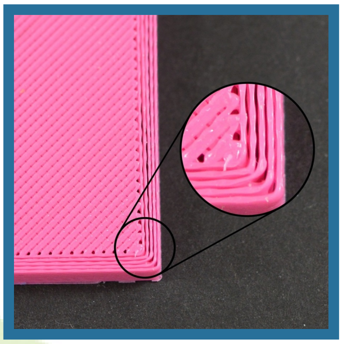
☐ PLA	☐ Silk PLA	☐ PETG	☐ TPU	☐ Nylon
☐ ABS	☐ PC	☐ Wood	☐ HIPS	☐ POM
☐ PEI	☐ PEEK	☐ FLEXIBLE		

amazon

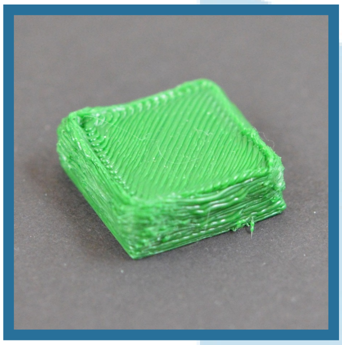


Not Extruding At Start: Printer does not extrude plastic at the beginning of the print

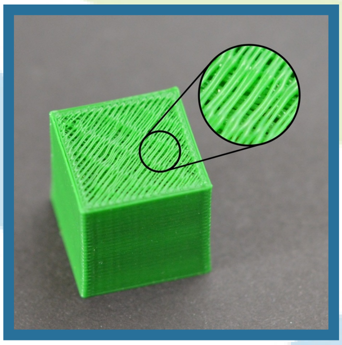
Not Sticking To Bed: The first layer does not stick to the bed and the print quickly fails



Under-Extrusion: Printer does not extrude enough plastic, gaps between perimeters and infill

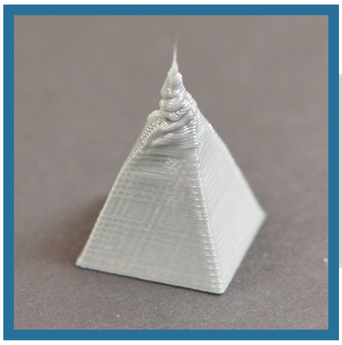


Over-Extrusion: Printer extrudes too much plastic, prints looks very messy

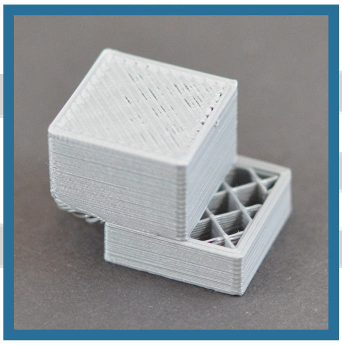


Gaps in Top Layers: Holes or gaps in the top layers of the print

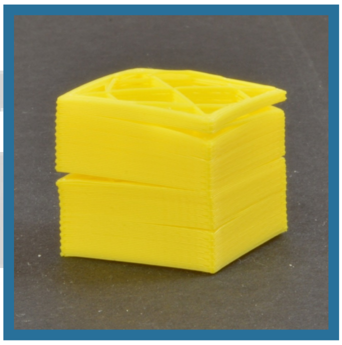
Stringing or Oozing: Lots of strings and hairs left behind when moving between different sections of the print



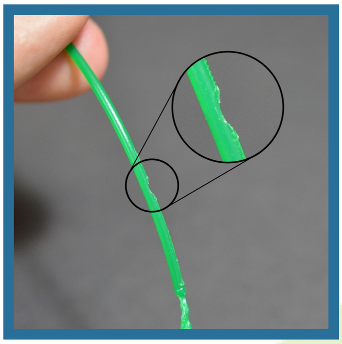
Overheating: Small features become overheated and deformed



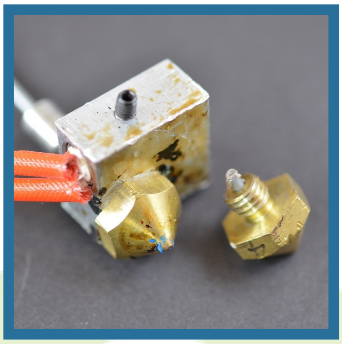
Layer Shifting: Layers are misaligned and shift relative to one another



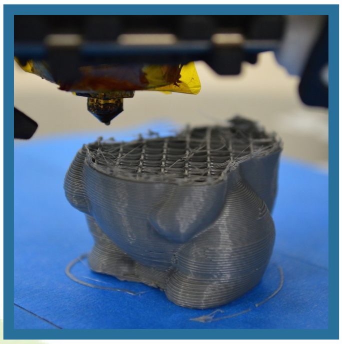
Layer Separation and Splitting: Layers are separating and splitting apart while printing



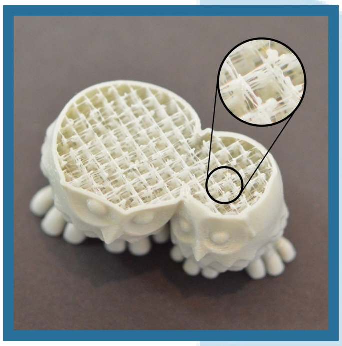
Grinding Filament: Plastic is being ground away until the filament no longer moves, otherwise known as "stripped" filament



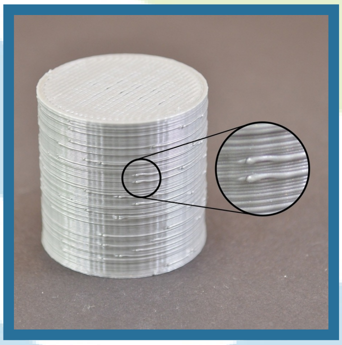
Clogged Extruder: Extruder is clogged or jammed and will no longer extrude plastic from the nozzle tip



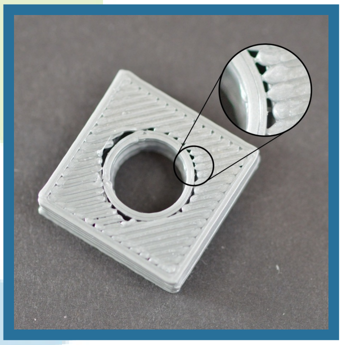
Stops Extruding Mid Print: Printer stops extruding plastic randomly in the middle of a print



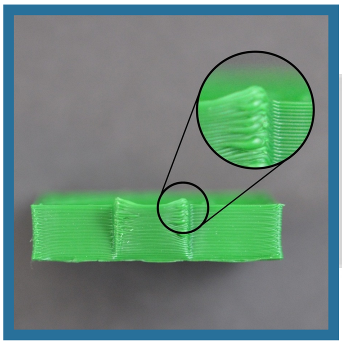
Weak Infill: Very thin, stringy infill that creates a weak interior and does not bond together well



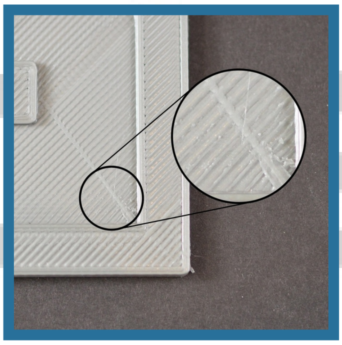
Blobs and Zits: Small blobs on the surface of print, otherwise known as zits



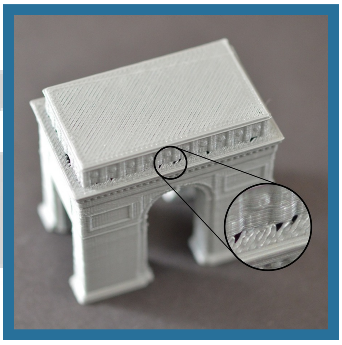
Gaps Between Infill and Outline: Gaps between the outline of the part and the outer solid infill layers



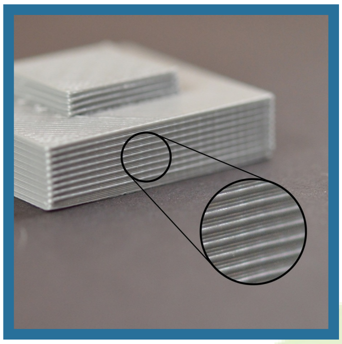
Curling or Rough Corners: Corners of the print tend to curl and deform after they are printed



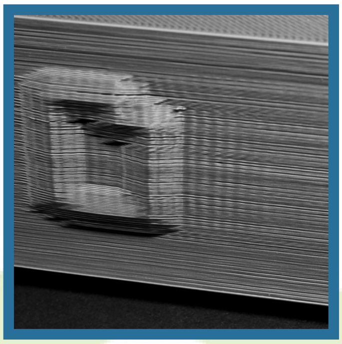
Scars on Top Surface: The nozzle drags across the top of the print and creates a scar on the surface



Gaps in Floor Corners: Gaps in the corners of the print, where the top layer does not join to the outline of the next layer



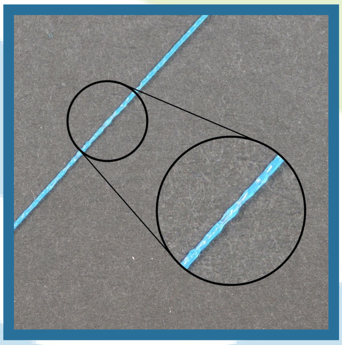
Lines on the Side of Print: Side walls are not smooth, lines are visible on the side of the print



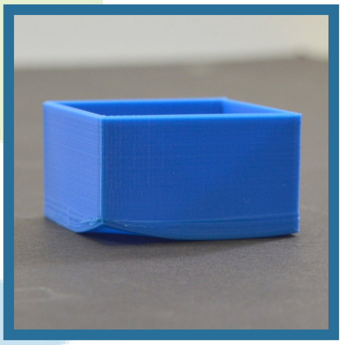
Vibrations and Ringing: Vibrations that cause oscillations on the surface of the print, otherwise known as -inging”

Gaps in Thin Walls: Gaps between thin walls of the print where the perimeters do not touch

Small Features Not Printed: Very small features are not printed or are missing from the software preview

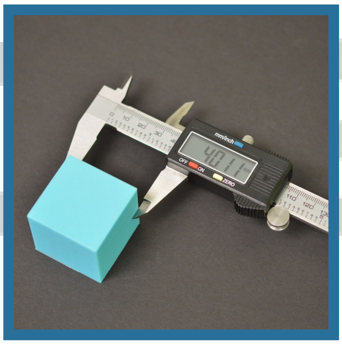


Inconsistent Extrusion: Extrusion amount tends to vary and is not consistent enough to produce an accurate shape

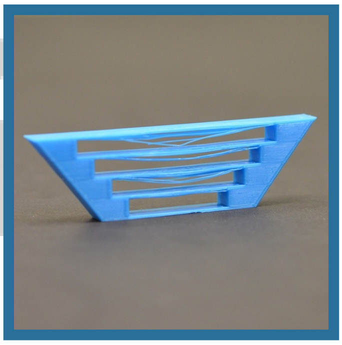


Warping: Warping of large parts, particularly with high temperature materials such as ABS

Poor Surface Above Supports: Poor surface quality on the underside of the part where it touches the support structures

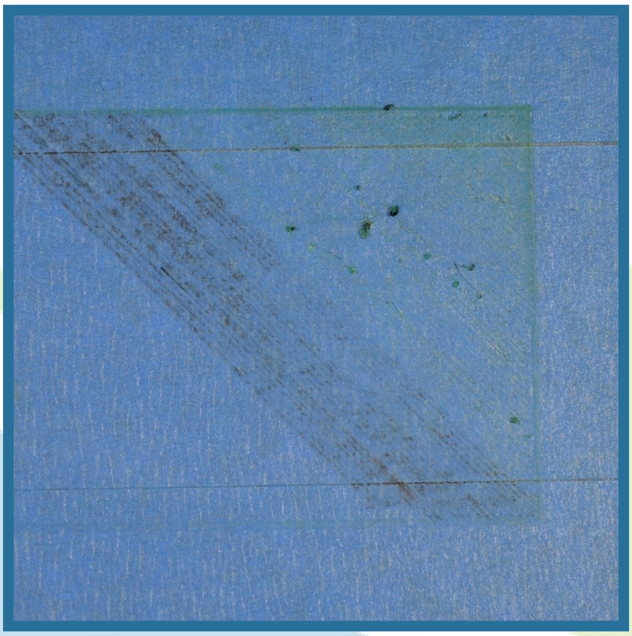


Dimensional Accuracy: Dimensional issues where the measured dimensions do not match the original design intent



Poor Bridging: Sagging, drooping, or gaps between the extruded segments of your bridging regions

1. Not Extruding at Start of Print



This issue is a very common one for new 3D printer owners, but thankfully, it is also very easy to resolve! If your extruder is not extruding plastic at the beginning of your print, there are four possible causes. We will walk through each one below and explain what settings can be used to solve the problem.

Extruder was not primed before beginning the print

Most extruders have a bad habit of leaking plastic when they are sitting idle at a high temperature. The hot plastic inside the nozzle tends to ooze out of the tip, which creates a void inside the nozzle where the plastic has drained out. This idle oozing can occur at the beginning of a print when you are first preheating your extruder, and also at the end of the print while the extruder is slowly cooling. If your extruder has lost some plastic due to oozing, the next time you try to extrude, it is likely that it will take a few seconds before plastic starts to come out of the nozzle again. If you are trying to start a print after you nozzle has been oozing, you may notice the same delayed extrusion. To solve this issue, make sure that you prime your extruder right before beginning a print so that the nozzle is full of plastic and ready to extrude. A common way to do this is by including something called a skirt. The skirt will draw a circle around your part, and in the process, it will prime the extruder with plastic. If you need extra priming, you can increase the number of skirt outlines. Some users may also prefer to manually extrude filament from their printer prior to beginning the print.

Nozzle starts too close to the bed

If the nozzle is too close to the build table surface, there will not be enough room for plastic to come out of the extruder. The hole in the top of the nozzle is essentially blocked so that no plastic can escape. An easy way to recognize this issue is if the print does not extrude plastic for the first layer or two, but begins to extrude normally around the 3rd or 4th layers as the bed continues to lower along the Z-axis. To solve this problem, you can use the very handy G-Code offsets. This allows you to make very fine adjustments to the Z-axis position without needing to change the hardware.

For example, if you enter a value of 0.05mm for the Z-axis G-Code offset, this will move the nozzle 0.05mm further away from the print bed. Keep increasing this value by small increments until there is enough room between the nozzle and the build platform for the plastic to escape.

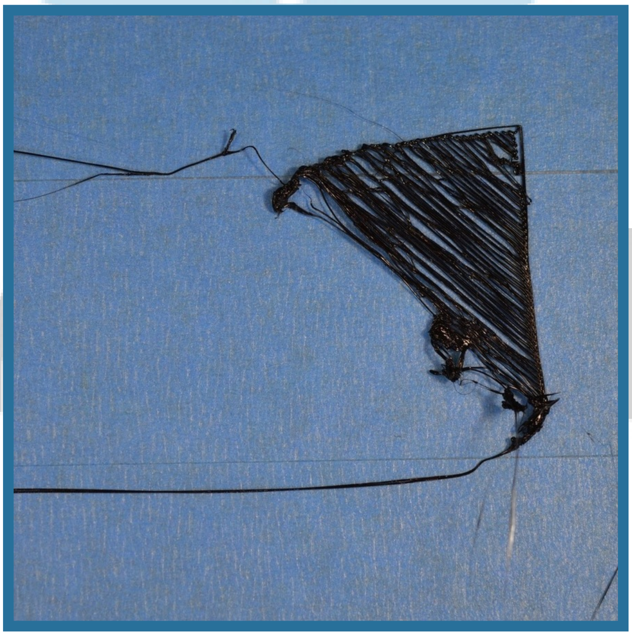
The filament has stripped against the drive gear

Most 3D printers use a small gear to push the filament back and forth. The teeth on this gear bite into the filament and allow it to accurately control the position of the filament. However, if you notice lots of plastic shavings or it looks like there is a section missing from your filament, then it's possible that the drive gear has removed too much plastic. Once this happens, the drive gear won't have anything left to grab onto when it tries to move the filament back and forth. Please see the Grinding Filament section for instructions on how to fix this issue.

The extruder is clogged

If none of the above suggestions are able to resolve the issue, then it is likely that your extruder is clogged. This can happen if foreign debris is trapped inside the nozzle, when hot plastic sits inside the extruder too long, or if the thermal cooling for the extruder is not sufficient and the filament begins to soften outside of the desired melt zone. Fixing a clogged extruder may require disassembling the extruder, so please contact your printer manufacturer before you proceed. We have had great success using the "E" string on a guitar to unclog extruders by feeding it into the nozzle tip, however, your manufacturer should also be able to provide recommendations. Also there are nozzle cleaning needles available in the market that fit this purpose.

2. Print Not Sticking to the Bed



It is very important that the first layer of your print is strongly connected to the printer's build platform so that the remainder of your part can be built on this foundation. If the first layer is not sticking to the build platform, it will create problems later on. There are many different ways to cope with these first layer adhesion problems, so we will examine several typical causes below and explain how to address each one.

Build platform is not level

Many printers include an adjustable bed with several screws or knobs that control the position of the bed. If your printer has an adjustable bed and you're having trouble getting your first layer to stick to the bed, the first thing you will want to verify is that your printer's bed is flat and level. If the bed is not level, one side of your bed may be too close to the nozzle, while the other side is too far away. Achieving a perfect first layer requires a level print bed. The manufacturer of your printer should give you instructions on how to perform the leveling correctly. There are also lots of articles and videos on the internet showing how to do so.

Nozzle starts too far away from the bed

Once your bed has been properly leveled, you still need to make sure that the nozzle is starting at the correct height relative to the build platform. Your goal is to locate your extruder the perfect distance away from the build plate — not too far and not too close. For good adhesion to the build plate, you want your filament to be slightly squished against the build plate. While you can adjust these settings by modifying the hardware, it is typically much easier (and much more precise!) to make these changes from your slicing software. You can use the Z-Axis global G-Code Offset to make very fine adjustments to your nozzle position. For example, if you enter -0.05mm for the Z-axis G-Code offset, the nozzle will begin printing 0.05mm closer to your build platform. Be careful to only make small adjustments to this setting. Each layer of your part is usually only around 0.2mm thick, so a small adjustment goes a long way!

First layer is printing too fast

As you extrude the first layer of plastic on top of the build platform, you want to make sure that plastic can properly bond to the surface before starting the next layer. If you print the first layer too fast, the plastic may not have time to bond to the build platform. For this reason, it is typically very useful to print the first layer at a slower speed so that the plastic has time to bond to the bed. The slicing software provides a setting for this exact feature. For example, if you set a first layer speed of 50 %, it means that your first layer will print 50 % slower than the rest of your part. If you feel that your printer is moving too fast on the first layer, try reducing this setting.

Temperature or cooling settings

Plastic tends to shrink as it cools from a warm temperature to a cool temperature. To provide a useful example, imagine a 100mm wide part that is being printed with ABS plastic. If the extruder was printing this plastic at 230 degrees Celsius, but it was being deposited onto a cold build platform, it is likely that the plastic would quickly cool down after leaving the hot nozzle. Some printers also include cooling fans that speed up this cooling process when they are being used. If this ABS part cooled down to a room temperature of 30C, the 100mm wide part would shrink by almost 1.5mm! Unfortunately, the build platform on your printer is not going to shrink this much, since it is typically kept at a fairly constant temperature. Because of this fact, the plastic will tend to separate from the build platform as it cools. This is an important fact to keep in mind as you print your first layer. If you notice that the layer seems to stick initially, but later separates from the print bed as it cools, it is possible that your temperature and cooling settings are to blame.

Many printers that are intended to print high-temperature materials like ABS include a heated bed to help combat these problems. If the bed is heated to maintain a temperature of 110C for the entire print, it will keep the first layer warm so that it does not shrink. So if your printer has a heated bed, you may want to try heating the bed to prevent the first layer from cooling. As a general starting point, PLA tends to adhere well to a bed that is heated to 60-70C, while ABS generally works better if the bed is heated to 100-120C. You can adjust all the temperature settings in your slicing software o directly on yor machine's control panel.

If your printer has a cooling fan, you may also want to try disabling that cooling fan for the first few layers of your printer so that the initial layers do not cool down too quickly. Usually, the slicing software allows you to adjust the fan speed setpoints. For example, you may want the first layer to start with the fan disabled and then turn on the fan to full power once you reach the 5th layer. In that case, you will need to add two setpoints into that list: Layer 1 at 0 % fan speed, and Layer 5 at 100 % fan speed. If you are using ABS plastic, it is common to disable the cooling fan for the entire print. If you are working in a breezy environment, you may also want to try to insulate your printer to keep the wind away from your part.

The build platform surface (tape, glues, and materials)

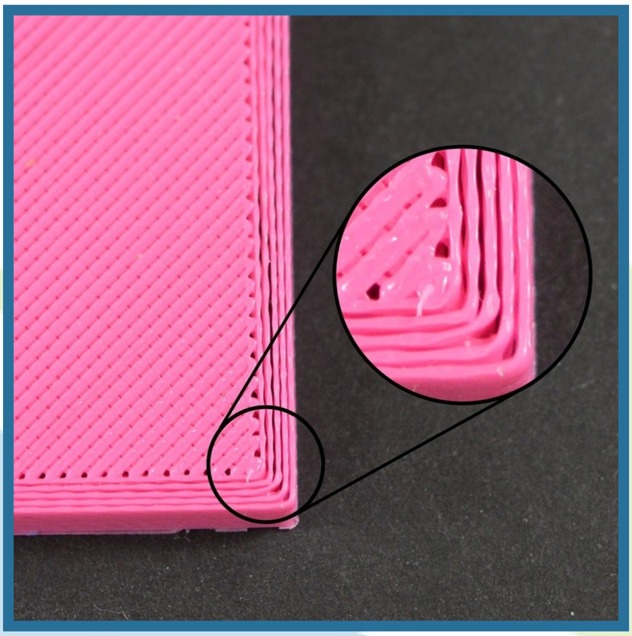
Different plastics tend to adhere better to different materials. For this reason, many printers include a special build platform material that is optimized for their materials. For example, several printers use a BuildTak sheet on the top of their bed that tends to stick very well to PLA. Other manufacturers opt for a heat treated glass bed such as Borosilicate glass, which tends to work very well for ABS when heated. If you are going to print directly onto these surfaces, it is always a good idea to make sure that your build platform is free of dust, grease, or oils before starting the print. Cleaning your print bed with some water or isopropyl rubbing alcohol can make a big difference.

If your printer does not include a special build platform material to help with adhesion, you still have options! Thankfully, there are several types of tape that stick well to common 3D printing materials. Strips of tape can be applied to the build platform surface and easily removed or replaced if you want to print with a different material. For example, PLA tends to stick well to blue painter's tape while ABS tends to stick better to Kapton tape (otherwise known as Polyimide film). Many users have also had great success using a temporary glue or spray on the top of their build platforms. Hair spray, glue sticks, and other sticky substances tend to work very well if everything else has failed. Feel free to experiment to see what works best for you!

When all else fails: Brims and Rafts

Sometimes you are printing a very small part that simply does not have enough surface area to stick to the build platform surface. The slicing software includes several options that can help increase this surface area to provide a larger surface to stick to the print bed. One of these options is called a "brim." The brim adds extra rings around the exterior of your part, similar to how a brim of a hat increases the circumference of the hat. The slicers also allows users to add a raft under their part, which can also be used to provide a larger surface for bed adhesion.

3. Not Extruding Enough Plastic



The slicing software includes settings that are used to determine how much plastic the 3D printer should extrude. However, because the 3D printer does not provide any feedback about how much plastic actually leaves the nozzle, it's possible that there may be less plastic exiting the nozzle than what the software expects (otherwise known as under-extrusion). If this happens, you may start to notice gaps between adjacent extrusions of each layer. The most reliable way to test whether or not your printer is extruding enough plastic is to print a simple 20mm tall cube with at least 3 perimeter outlines. At the top of the cube, check to see if the 3 perimeters are strongly bonded together or not. If there are gaps between the 3 perimeters, then you are under-extruding. If the 3 perimeters are touching and do not have any gaps, then you are likely encountering a different issue. If you determine that you are under-extruding, there are several possible causes for this, which we have summarized below.

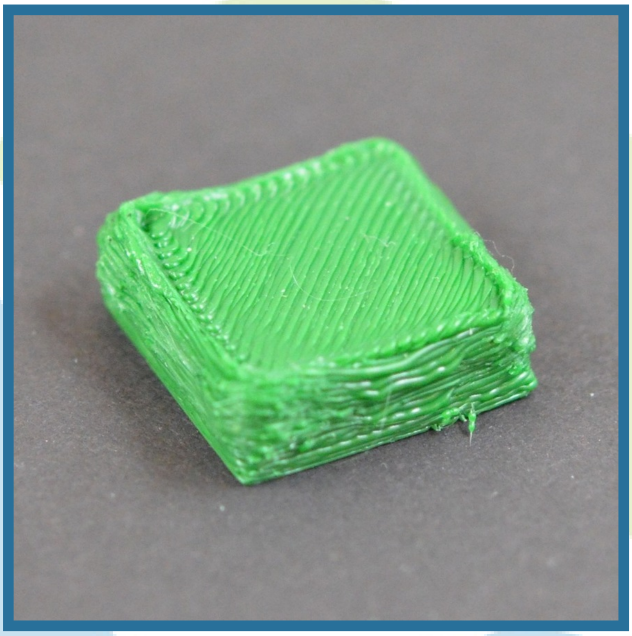
3.1. Incorrect filament diameter

The first thing you want to verify is that the software knows the filament diameter that you are using. Check to make sure that this value matches the filament that you purchased. You may even want to measure your filament yourself using a pair of calipers to make sure that you truly have the correct diameter specified in the software.

3.2. Increase the extrusion multiplier

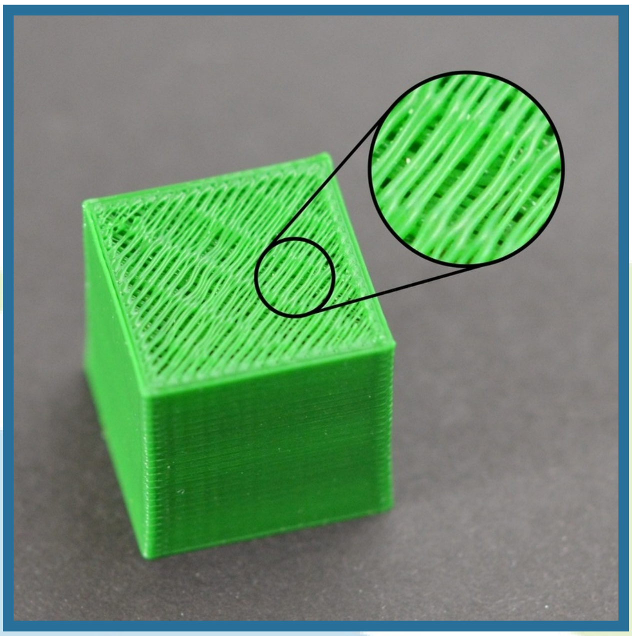
If your filament diameter is correct, but you are still seeing under-extrusion issues, then you need to adjust your extrusion multiplier. This is a very useful setting in the slicing software that allows you to easily modify the amount of plastic that is extruded (otherwise known as the flow rate). Each extruder on your printer can have a unique extrusion multiplier. As an example, if your extrusion multiplier was 1.0 previously and you change it to 1.05, it means you will be extruding 5 % more plastic than you were previously. It is typical for PLA to print with an extrusion multiplier near 0.9, while ABS tends to have extrusion multipliers closer to 1.0. Try increasing your extrusion multiplier by 5 %, and then reprint the test cube to see if you still have gaps between your perimeters.

4. Extruding Too Much Plastic



The software is constantly working together with your printer to make sure that your nozzle is extruding the correct amount of plastic. This precise extrusion is an important factor in achieving good print quality. However, most 3D printers have no way of monitoring how much plastic is actually extruded. If your extrusion settings are not configured properly, the printer may extrude more plastic than the software expects. This over-extrusion will result in excess plastic that can ruin the outer dimensions of your part. To resolve this issue, there are only a few settings you need to verify. Please see the Not Extruding Enough Plastic section for a more detailed description. While those instructions are for under-extrusion, you will adjust the same settings for over-extrusion, just in the opposite direction. For example, if increasing the extrusion multiplier helps with under-extrusion, then you should decrease the extrusion multiplier for over-extrusion issues.

5. Holes and Gaps in the Top Layers



To save plastic, most 3D printed parts are created to have a solid shell that surrounds a porous, partially hollow interior. For example, the interior of the part may use a 30 % infill percentage, which means that only 30 % of the interior is solid plastic, while the rest is air. While the interior of the part may be partially hollow, we want the exterior to remain solid. To do this, the slicing software allows you to specify how many solid layers you want on the top and bottom of your part. For example, if you were printing a simple cube with 5 top and bottom solid layers, the software would print 5 completely solid layers at the top and bottom of the print, but everything else in the middle would be printed as a partially hollow layer. This technique can save a tremendous amount of plastic and time, while still creating very strong parts. However, depending on what settings you are using, you may notice that the top solid layers of your print are not completely solid. You may see gaps or holes between the extrusions that make up these solid layers. If you have encountered this issue, here are several simple settings that you can adjust to fix it.

5.1. Not enough top solid layers

The first setting to adjust is the number of top solid layers that are used. When you try to print a 100 % solid layer on top of your partially hollow infill, the solid layer has to span across the hollow air pockets of your infill. When this happens, the extrusions for the solid layer have a tendency to droop or sag down into the air pocket. Because of this, you generally want to print several solid layers at the top of your print to ensure a nice flat, completely solid surface. As a good rule of thumb, you want the solid section at the top of your print to be at least 0.5mm thick. So if you are using a 0.25mm layer height, you would need at least 2 top solid layers. If you are printing at a lower layer height such as 0.1mm, you may need 5 solid layers at the top of your print to achieve the same effect. If you are noticing gaps between the extrusions in your top surface, the first thing you should try is increasing the number of top solid layers. For example, if you noticed the problem using only 3 top solid layers, try printing with 5 top solid layers to see if the problem is improved. Note that additional solid layers will occur within your part dimension and do not add size to the exterior of your part.

5.2. Infill percentage is too low

The infill on the inside of your part will act as the foundation for the layers above it. The solid layers at the top of your part will need to print on top of this foundation. If your infill percentage is very low, there will be large air gaps in your infill. For example, if you are using an infill percentage of only 10 %, the remaining 90 % of the interior of your part would be hollow, and this would create some very large air gaps that the solid layers would need to print on top of. If you have tried increasing the number of top solid layers and you are still seeings gaps in the top of your print, you may want to try increasing your infill percentage to see if the gaps go away. For example, if your infill percentage was previously 30 %, try using a 50 % infill percentage, as this would provide a much better foundation for the solid layers at the top of your print.

5.3. Under-Extrusion

If you have tried increasing the infill percentage and the number of top solid layers, yet you are still seeing gaps in the tops of your print, then you likely have an under-extrusion issue. This means that your nozzle is not extruding as much plastic as the software expects. For a full description of this issue and how to correct it, please read the Not Extruding Enough Plastic section.

6. Stringing or Oozing

Stringing (otherwise known as oozing, whiskers, or "hairy" prints) occurs when small strings of plastic are left behind on a 3D printed model. This is typically due to plastic oozing out of the nozzle while the extruder is moving to a new location. Thankfully, there are several settings that can help with this issue. The most common setting that is used to combat excessive stringing is something that is known as retraction. If retraction is enabled, when the extruder is done printing one section of your model, the filament will be pulled backwards into the nozzle to act as a countermeasure against oozing. When it is time to begin printing again, the filament will be pushed back into the nozzle so that plastic once again begins extruding from the tip. Ensure that the retraction option is enabled for each of your extruders. In the sections below, we will discuss the important retraction settings as well as several other settings that can be used to combat stringing, such as the extruder temperature settings.

6.1. Retraction distance

The most important retraction setting is the retraction distance. This determines how much plastic is pulled out of the nozzle. In general, the more plastic that is retracted from the nozzle, the less likely the nozzle is to ooze while moving. Most direct-drive extruders only require a retraction distance of 0.5-2.0mm, while some Bowden extruders may require a retraction distance as high as 15mm due to the longer distance between the extruder drive gear and the heated nozzle. If you encounter stringing with your prints, try increasing the retraction distance by 1mm and test again to see if the performance improves.

6.2. Retraction speed

The next retraction setting that you should check is the retraction speed. This determines how fast the filament is retracted from the nozzle. If you retract too slowly, the plastic will slowly ooze down through the nozzle and may start leaking before the extruder is done moving to its new destination. If you retract too quickly, the filament may separate from the hot plastic inside the nozzle, or the quick movement of the drive gear may even grind away pieces of your filament. There is usually a sweet spot somewhere between 1200-6000 mm/min (20-100 mm/s) where retraction performs best, but the ideal value can vary depending on the material that you are using, so you may want to experiment to see if different speeds decrease the amount of stringing that you see.

6.3. Temperature is too high

Once you have checked your retraction settings, the next most common cause for excessive stringing is the extruder temperature. If the temperature is too high, the plastic inside the nozzle will become less viscous and will leak out of the nozzle much more easily. However, if the temperature is too low, the plastic will still be somewhat solid and will have difficulty extruding from the nozzle. If you feel you have the correct retraction settings, but you are still encountering these issues, try decreasing your extruder temperature by 5-10 degrees. This can have a significant impact on the final print quality.

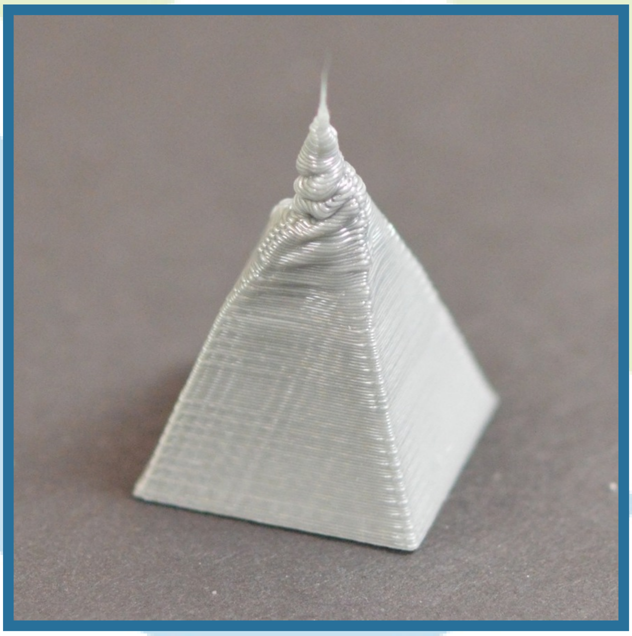
6.4. Long movements over open spaces

As we discussed above, stringing occurs when the extruder is moving between two different locations, and during that move, plastic starts to ooze out of the nozzle. The length of this movement can have a large impact on how much oozing takes place. Short moves may be quick enough that the plastic does not have time to ooze out of the nozzle. However, long movements are much more likely to create strings. Some slicing softwares include an extremely useful feature that can help minimize the length of these movements. The software is smart enough that it can automatically adjust the travel path to make sure that nozzle has a very short distance to travel over an open space. In fact, in many cases, the software may be able to find a travel path that avoids crossing an open space all together! This means that there is no possibility to create a string, because the nozzle will always be on top of the solid plastic and will never travel outside the part.

6.5. Movement Speed

Finally, you may also find that increasing the movement speed of your machine can also reduce the amount of time that the extruder can ooze when moving between parts. The X/Y Axis Movement Speed represents the side-to-side travel speed, and is frequently directly related to the amount of time your extruder spends moving over open air. If your machine can handle moving at higher speeds, you may find that increasing this settings can also reduce stringing between parts.

7. Overheating



The plastic that exits your extruder may be anywhere from 190 to 240 degrees Celsius. While the plastic is still hot, it is pliable and can easily be formed into different shapes. However, as it cools, it quickly becomes solid and retains its shape. You need to achieve the correct balance between temperature and cooling so that your plastic can flow freely through the nozzle, but it can quickly solidify to maintain the exact dimensions of your 3D printed part. If this balance is not achieved, you may start to notice some print quality issues where the exterior of your part is not as precise and defined as you would like. As you can see in the image, the filament extruded at the top of the pyramid was not able to cool quickly enough to retain its shape. The section below will examine several common causes for overheating and how to prevent them.

7.1. Insufficient Cooling

The most common cause for overheating is that the plastic is not being cooled fast enough. When this happens, the hot plastic is free to change shapes as it slowly cools. For many plastics, it is much better to quickly cool the layers to prevent them from changing shape after being printed. If your printer includes a cooling fan, try increasing the power of the fan to cool the plastic faster. This additional cooling will help the plastic retain its shape. If your printer does not include an integrated cooling fan, you may want to try installing an aftermarket fan or using a small handheld fan to cool down the layers faster.

7.2. Printing at too high of a temperature

If you are already using a cooling fan and you are still seeing this issue, you may want to try printing at a lower temperature. If the plastic is extruded at a lower temperature it will be able to solidify faster and retain its shape. Try lowering the print temperature by 5-10 degrees to see if it helps. Be careful not to lower the temperature too far, as otherwise the plastic may not be hot enough to extrude through the small opening in your nozzle.

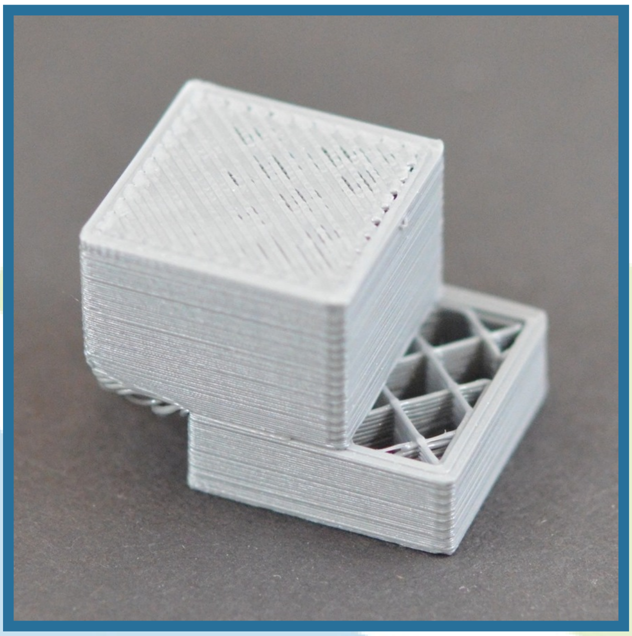
7.3. Printing too fast

If you are printing each layer very quickly, you might not allow enough time for the previous layer to properly cool before you are trying to deposit the next layer of hot plastic on top of it. This is particularly important for very small parts where each layer only requires a few seconds to print. Even with a cooling fan, you may still need to decrease the printing speed for these small layers to ensure you provide enough time for the layer to solidify. Usually, the slicing software includes a very simple option to do exactly that. This option is used to automatically slow down the printing speed for small layers to ensure they have enough time to cool and solidify before printing the next layer. For example, if you allow the software to adjust the printing speed for layers that take less than 15 seconds to print, the program will automatically slow down the printing speed for these small layers. This is a vital feature for combating these overheating issues.

7.4. When all else fails: Try printing multiple parts at once

If you have already tried the 3 items above and you are still having trouble achieving sufficient cooling, there's one more thing you can try. Create a copy of the part you are trying to print or import a second object that can be printed at the same time. By printing two objects at once, you can provide more cooling time for each individual part. The hot nozzle will need to move to a different location on the build platform to print the second part, which provides a short relief for your first part to cool down. This is a simple, yet very effective strategy for fixing overheating problems.

8. Layer Shifting or Misalignment



Most 3D printers use an open-loop control system, which is a fancy way to say that they have no feedback about the actual location of the toolhead. The printer simply attempts to move the toolhead to a specific location, and hopes that it gets there. In most cases, this works fine because the stepper motors that drive the printer are quite powerful, and there are no significant loads to prevent the toolhead from moving. However, if something does go wrong, the printer would have no way to detect this. For example, if you happened to bump into your printer while it was printing, you might cause the toolhead to move to a new position. The machine has no feedback to detect this, so it would just keep printing as if nothing had happened. If you notice misaligned layers in your print, it is usually due to one of the causes below. Unfortunately, once these errors occur, the printer has no way to detect and fix the problem, so we will explain how to resolve these issues below.

8.1. Toolhead is moving too fast

If you are printing at a very high speed, the motors for your 3D printer may struggle to keep up. If you attempt to move the printer faster than the motors can handle, you will typically hear a clicking sound as the motor fails to achieve the desired position. If this happens, the remainder of the print will be misaligned with everything that was printed before it. If you feel that your printer may be moving too fast, try to reduce the printing speed by 50 % to see if it helps. Adjust both the “Default Printing Speed” and the “X/Y Axis Movement Speed.” The default printing speed controls the speed of any movements where the extruder is actively extruding plastic. The X/Y axis movement speed controls the speed of rapid movements where no plastic is being extruded. If either of those speeds are too high, it can cause shifting to occur. If you are comfortable adjusting more advanced settings, you may also want to consider lowering the acceleration settings in your printer’s firmware to provide a more gradual speed up and slow down.

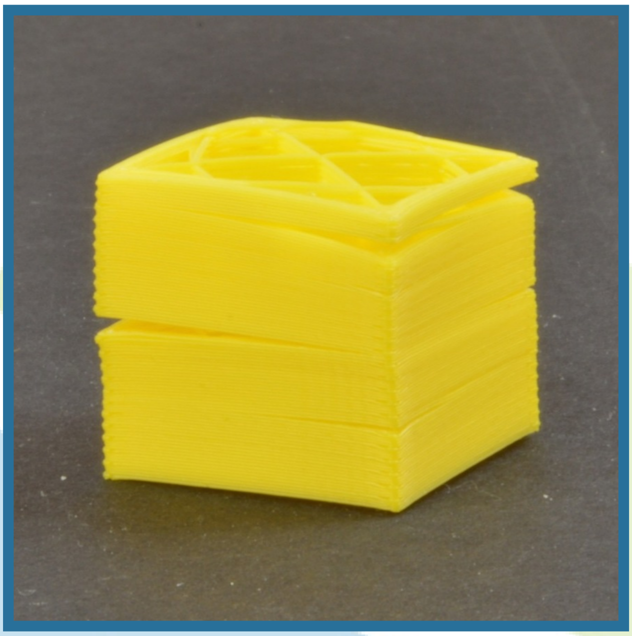
8.2. Mechanical or Electrical Issues

If the layer misalignment continues, even after reducing your print speed, then it is likely due to mechanical or electrical issues with the printer. For example, most 3D printers use belts that allow the motors to control the position of the toolhead. The belts are typically made of a rubber material and reinforced with some type of fiber to provide additional strength. Over time, these belts may stretch, which can impact the belt tension that is used to position the toolhead. If the tension becomes too loose, the belt may slip on top of the drive pulley, which means the pulley is rotating, but the belt is not moving. If the belt was originally installed too tight, this can also cause issues. An overtightened belt can create excess friction in the bearings that will prevent the motors from spinning. Ideal assembly requires a belt that is somewhat tight to prevent slipping, but not too tight to where the system is unable to rotate. If you start noticing issues with misaligned layers, you should verify that your belts all have the appropriate tension, and none appear to be too loose or too tight. If you think there may be a problem, please consult the printer manufacturer for instructions on how to adjust the belt tension.

Many 3D printers also include a series of belts that are driven by pulleys attached to a stepper motor shaft using a small set-screw (otherwise known as a grub screw). These set-screws anchor the pulley to the shaft of the motor so that the two items spin together. However, if the set-screw loosens, the pulley will no longer rotate together with the motor shaft. This means that the motor may be spinning, but the pulley and belts are not moving. When this happens, the toolhead does not get to the desired location, which can impact the alignment of all future layers of the print. So if layer misalignment is a reoccurring problem, you should verify that all of the motor fasteners are properly tightened.

There are also several other common electrical issues that can cause the motors to lose their position. For example, if there is not enough electrical current getting to the motors, they won't have enough power to spin. It is also possible that the motor driver electronics could overheat, which causes the motors to stop spinning temporarily until the electronics cool down. While this is not an exhaustive list, it provides a few ideas for common electrical and mechanical causes that you may want to check if layer shifting is a persistent problem.

9. Layer Separation and Splitting



3D printing works by building the object one layer at a time. Each successive layer is printed on top of the previous layer, and in the end this creates the desired 3D shape. However, for the final part to be strong and reliable, you need to make sure that each layer adequately bonds to the layer below it. If the layers do not bond together well enough, the final part may split or separate. We will examine several typical causes for this below and provide suggestions for resolving each one.

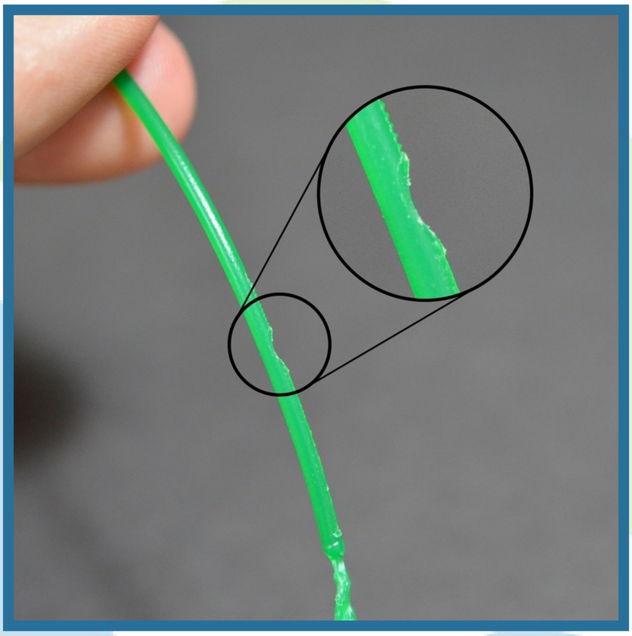
9.1. Layer height is too large

Most 3D printing nozzles have a diameter between 0.3-0.5mm. The plastic squeezes through this tiny opening to create a very thin extrusion that can produce extremely detailed parts. However, these small nozzles also create some limitations for what layer heights can be used. When you print one layer of plastic on top of another, you want to make sure that the new layer is being pressed against the layer below it so that the two layers will bond together. As a general rule of thumb, you want to make sure that the layer height you select is 20 % smaller than your nozzle diameter. For example, if you have a 0.4mm nozzle, you can't go too far past a layer height of 0.32mm, or each layer of plastic will not be able to properly bond to the layer beneath it. So if you notice that your prints are separating and the layers are not sticking together, the first thing you should check is your layer height compared to the size of your nozzle. Try reducing the layer height to see if it helps the layers bond together better.

9.2. Print temperature is too low

Warm plastic will always bond together much better than cold plastic. If you notice that your layers aren't bonding together and you are certain that your layer height isn't too large, then it is possible that your filament needs to be printed at a higher temperature to create a strong bond. For example, if you tried to print ABS plastic at 190C, you would likely find that the layers of your part will easily break apart. This is because ABS typically needs to be printed around 220-235C to create a strong bond between the layers of your print. So if you feel this may be the problem, verify that you are using the correct temperature for the filament you have purchased. Try increasing the temperature by 10 degrees to see if the adhesion improves.

10. Grinding Filament



Most 3D printers use a small drive gear that grabs the filament and sandwiches it against another bearing. The drive gear has sharp teeth that allow it to bite into the filament and push it forward or backward, depending on which direction the drive gear spins. If the filament is unable to move, yet the drive gear keeps spinning, it can grind away enough plastic from the filament so that there is nothing left for the gear teeth to grab on to. Many people refer to this situation as the filament being "stripped," because too much plastic has been stripped away for the extruder to function correctly. If this is happening on your printer, you will typically see lots of small plastic shavings from the plastic that has been ground away. You may also notice that the extruder motor is spinning, but the filament is not being pulled into the extruder body. We will explain the easiest way to resolve this issue below.

10.1. Aggressive Retraction Settings

One of the first things you will want to check are the retraction settings for your extruder. If the retraction speed is too fast, or you are trying to retract far too much filament, it may put excessive stress on your extruder and the filament will struggle to keep up. As an easy test, you can try reducing your retraction speed by 50 % to see if the problem goes away. If so, you know that your retraction settings may be part of the problem.

10.2. Increase the extruder temperature

If you continue to encounter filament grinding, try to increase the extruder temperature by 5-10 degrees so that the plastic flows easier. Plastic will always flow easier at a higher temperature, so this can be a very helpful setting to adjust.

10.3. Printing too fast

If you continue to encounter filament grinding, even after increasing the temperature, then the next thing you should do is decrease the printing speed. By doing this, the extruder motor will not need to spin as fast, since the filament is extruded over a longer period of time. The slower rotation of the extruder motor can help avoid grinding issues. Adjust the "Default Printing Speed," which controls the speed of any movements where the extruder is actively extruding plastic. For example, if you were previously printing at 3600 mm/min (60 mm/s), try decreasing that value by 50 % to see if the filament grinding goes away.

10.4. Check for a nozzle clog

If you are still encountering filament grinding after increasing the temperature and slowing down the print speed, then it's likely your nozzle is partially clogged. Please read the Clogged Extruder section for instructions on how to troubleshoot this issue.

11. Clogged Extruder

Your 3D printer must melt and extrude many kilograms of plastic over its lifetime. To make things more complicated, all of this plastic must exit the extruder through a tiny hole that is only as big as a single grain of sand. Inevitably, there may come a time where something goes wrong with this process and the extruder is no longer able to push plastic through the nozzle. These jams or clogs are usually due to something inside the nozzle that is blocking the plastic from freely extruding. While this may be daunting the first time it happens, but we will walk through several easy troubleshooting steps that can be used to fix a jammed nozzle.

11.1. Manually push the filament into the extruder

One of the first things you may want to try is manually pushing the filament into the extruder. Heat your extruder to the appropriate temperature for your plastic. Next, use the manual controls to extrude a small amount of plastic, for example, 10mm. As the extruder motor is spinning, lightly use your hands to help push the filament into the extruder. In many cases, this added force will be enough to advance the filament past the problem area.

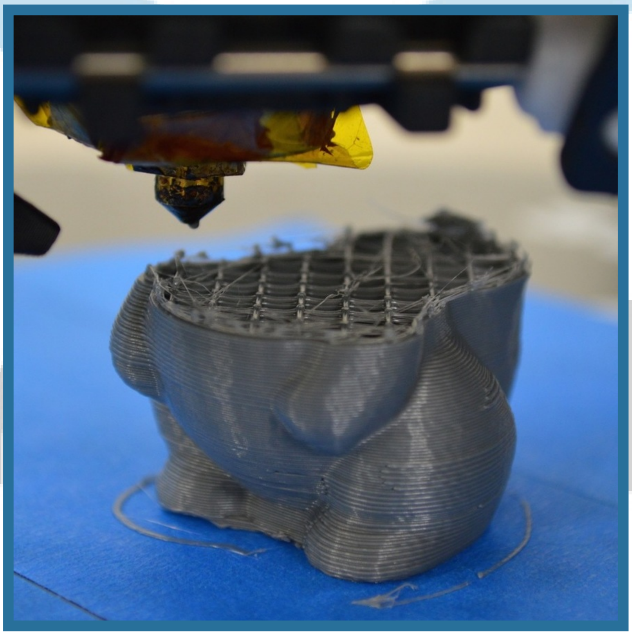
11.2. Reload the filament

If the filament still isn't moving, the next thing you should do is unload the filament. Verify that the extruder is heated to the appropriate temperature, and then use machine's control panel to retract the filament out of the extruder. As before, you may need to apply some additional force if the filament isn't moving. Once the filament is removed, use a pair of scissors to cut away the melted or damaged portion of the filament. Then reload the filament and see if you are able to extrude with the new, undamaged section of filament.

11.3. Clean out the nozzle

If you weren't able to extrude the new section of plastic through the nozzle, then it's likely you will need to clean out the nozzle before proceeding. Many users have had success heating their extruder to 100C and then manually pulling the filament out (hopefully along with any debris that was inside!). Others prefer to use the E string from a guitar to push the material backwards through the nozzle tip. There are plenty of other methods and each extruder is different, so please consult your printer manufacturer for precise instructions.

12. Stops Extruding in the Middle of a Print



If your printer was extruding properly at the beginning of your print, but suddenly stopped extruding later on, there are typically only a few things that could have caused this problem. We will explain each common cause below and provide suggestions for fixing the issue. If your printer was having trouble extruding at the very beginning of the print, please see the Not Extruding at Start of Print section.

12.1. Out of filament

This one is pretty obvious, but before checking the other issues, first verify that you still have filament leading into the nozzle. If the spool has run out, you will need to load a new spool before continuing the print.

12.2. The filament has stripped against the drive gear

During a print, the extruder motor is constantly spinning trying to push the filament into the nozzle so that your printer can keep extruding plastic. If you try to print too quickly or you try to extrude too much plastic, this motor may end up grinding away the filament until there is nothing left for the drive gear to grab onto. If your extruder motor is spinning, but the filament is not moving, then this is likely the cause. Please see the Grinding Filament section for more details on how to resolve the issue.

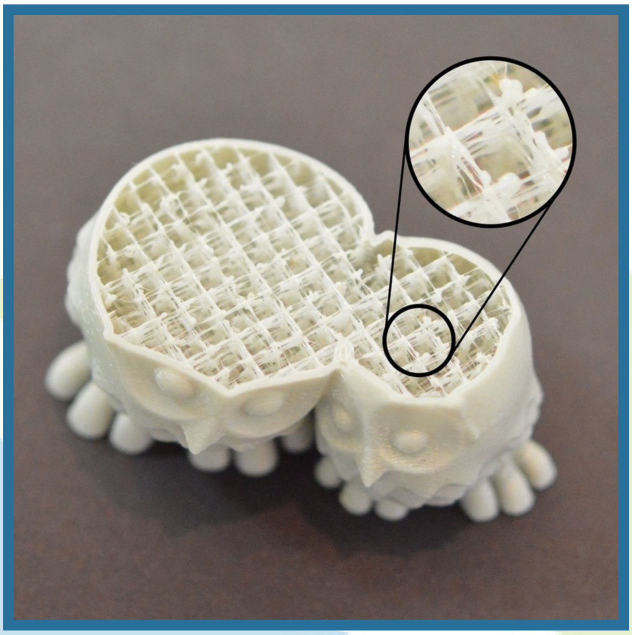
12.3. The extruder is clogged

If none of the above causes apply to you, then it is very likely that the extruder is clogged. If this happens in the middle of the print, you may want to check and make sure that the filament is clean and that there is no dust on the spool. If enough dust is attached to the filament, it can cause a clog as it builds up inside the nozzle. There are several other possible causes for a clogged extruder, so please see the clogged extruder description in the Not Extruding at Start of Print section for more details.

12.4. Overheated extruder motor driver

The extruder motor has to work incredibly hard during your print. It is constantly spinning back and forth, pushing and pulling plastic back and forth. This quick motion requires quite a bit of current, and if the printer's electronics do not have sufficient cooling, it can cause the motor driver electronics to overheat. These motor drivers typically have a thermal cutoff that will cause the driver to stop working if the temperature gets too high. If this happens, the X and Y axis motors will be spinning and moving the extruder toolhead, but the extruder motor will not be moving at all. The only way to resolve this issue is to turn off the printer and allow the electronics to cool down. You may also want to add an extra cooling fan if the problem continues.

13. Weak Infill



The infill inside your 3D printed part plays a very important role in the overall strength of your model. The infill is responsible for connecting the outer shells of your 3D print, and must also support the upper surfaces that will be printed on top of the infill. If your infill appears to be weak or stringy, you may want to adjust a few settings within the software to add additional strength to this section of your print.

13.1. Try alternate infill patterns

One of the first settings you should investigate is the infill pattern that is used for your print. Some patterns tend to be more solid than others. For example, Grid, Triangular, and Solid Honeycomb are all strong infill patterns. Other patterns like Rectilinear and Fast Honeycomb may sacrifice some strength for faster printing speeds. If you are having trouble producing strong reliable infill, try a different pattern to see if it makes a difference.

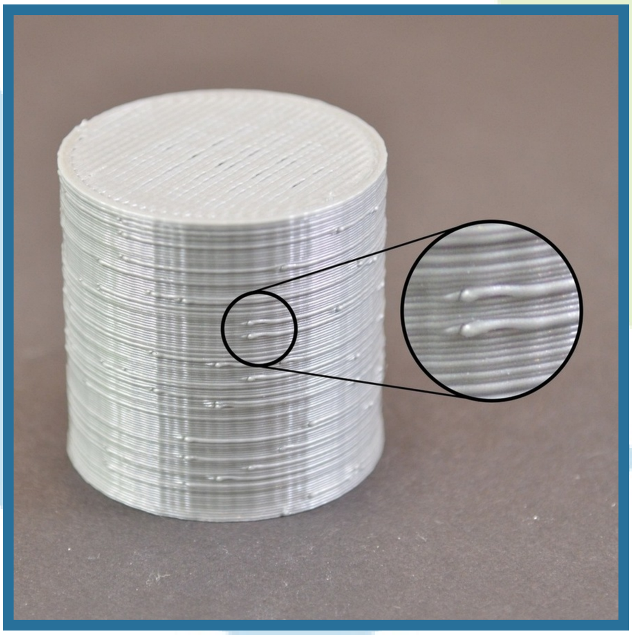
13.2. Lower the print speed

The infill is typically printed faster than any other portion of your 3D print. If you try to print the infill too fast, the extruder won't be able to keep up and you will start to notice under-extrusion on the inside of your part. This under-extrusion will tend to create weak, stringy infill since the nozzle is not able to extrude as much plastic as the software would like. If you have tried several infill patterns, but continue to have problems with weak infill, try reducing the print speed. For example, if you were previously printing at 3600 mm/min (60 mm/s), try decreasing that value by 50 % to see if the infill starts to become stronger and more solid.

13.3. Increase the infill extrusion width

Another very powerful feature within the slicing sofware is the ability to modify the extrusion width that is used for the infill of your part. For example, you could print the outline perimeters with a very fine 0.4mm extrusion width, but transition to a 0.8mm extrusion width for the infill. This will create thicker, stronger infill walls that greatly improve the strength of your 3D printed part. The "Infill Extrusion Width" is set as a percentage of the normal extrusion width. For example, if you enter a value of 200 %, the infill extrusions will be twice as thick as the outline perimeters. One thing to keep in mind when adjusting this setting is that the software must also maintain the infill percentage that you specify. So if you set the infill extrusion width to 200 %, the infill will use twice as much plastic for each line. To maintain the same infill percentage, the infill lines must be spaced further apart. For this reason, many users tend to increase their infill percentage after increasing the infill extrusion width.

14. Blobs and Zits



During your 3D print, the extruder must constantly stop and start extruding as it moves to different portions of the build platform. Most extruders are very good at producing a uniform extrusion while they are running, however, each time the extruder is turned off and on again, it can create extra variation. For example, if you look at the outer shell of your 3D print, you may notice a small mark on the surface that represents the location where the extruder started printing that section of plastic. The extruder had to start printing the outer shell of your 3D model at that specific location, and then it eventually returned to that location when the entire shell had been printed. These marks are commonly referred to as blobs or zits. As you can imagine, it is difficult to join two pieces of plastic together without leaving any mark whatsoever, but there are several tools that can be used to minimize the appearance of these surface blemishes.

14.1. Retraction and coasting settings

If you start to notice small defects on the surface of your print, the best way to diagnose what is causing them is to watch closely as each perimeter of your part is printed. Does the defect appear the moment the extruder starts printing the perimeter? Or does it only appear later when the perimeter is completed and the extruder is coming to a stop? If the defect appears right away at the beginning of the loop, then it's possible your retraction settings need to be adjusted slightly. Some slicers have a setting labeled "Extra Restart Distance." This option determines the difference between the retraction distance when the extruder is stopping and the priming distance that is used when the extruder is restarting. If you notice a surface defect right at the beginning of the perimeter, then your extruder is likely priming too much plastic. You can reduce the priming distance by entering a negative value for the extra restart distance. For example, if your retraction distance is 1.0mm, and the extra restart distance is -0.2mm (note the negative sign), then each time your extruder stops, it will retract 1.0mm of plastic. However, each time the extruder has to start extruding again, it will only push 0.8mm of plastic back into the nozzle. Adjust this setting until the defect no longer appears when the extruder initially begins printing the perimeter.

If the defect does not occur until the end of the perimeter when the extruder is coming to a stop, then there is a different setting to adjust. This setting is called coasting. Coasting will turn off your extruder a short distance before the end of the perimeter to relieve the pressure that is built up within the nozzle. Enable this option and increase the value until you no longer notice a defect appearing at the end of each perimeter when the extruder is coming to a stop. Typically, a coasting distance between 0.2-0.5mm is enough to have a noticeable impact.

14.2. Avoid unnecessary retractions

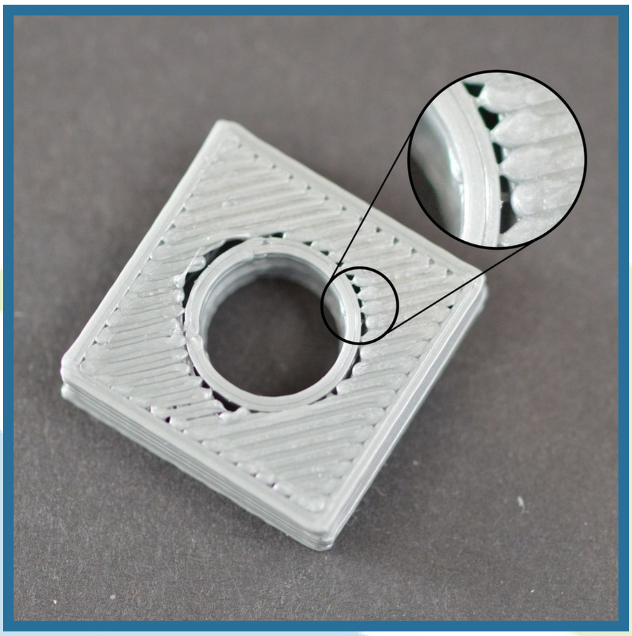
The retraction and coasting settings mentioned above can help avoid defects each time the nozzle retracts, however, in some cases, it is better to simply avoid the retractions all together. This way the extruder never has to reverse direction and can continue a nice uniform extrusion. This is particularly important for machines that use a Bowden extruder, as the long distance between the extruder motor and the nozzle makes retractions more troublesome. Some slicers allow to adjust the settings that control when a retraction takes place. As was mentioned in the Stringing or Oozing section, retractions are primarily used to prevent the nozzle from oozing as it moves between different parts of your print. However, if the nozzle is not going to cross an open space, the oozing that occurs will be on the inside of the model and won't be visible from the outside. For this reason, many printers will have the "Only retract when crossing open spaces" option enabled to avoid unnecessary retractions.

Another related setting can be found in some slicing softwares. If your printer is only going to retract when crossing open spaces, then it would be beneficial to avoid these open spaces as much as possible. Some slicers include an extremely useful feature that can divert the travel path of the extruder to avoid crossing an outline perimeter. If the extruder can avoid crossing the outline by changing the travel path, then a retraction won't be needed.

14.3. Choose the location of your start points

If you are still seeing some small defects on the surface of your print, some slicers also provide an option that can control the location of these points. In most cases, the locations of these start points are chosen to optimize the printing speed. However, you also have the ability to randomize the placement of the start points or align them to a specific location. For example, if you were printing a statue, you could align all of the start points to be on the backside of the model so that they were not visible from the front.

15. Gaps Between Infill and Outline



Each layer of your 3D printed part is created using a combination of outline perimeters and infill. The perimeters trace the outline of your part creating a strong and accurate exterior. The infill is printed inside of these perimeters to make up the remainder of the layer. The infill typically uses a fast back-and-forth pattern to allow for quick printing speeds. Because the infill uses a different pattern than the outline of your part, it is important that these two sections merge together to form a solid bond. If you notice small gaps between the edges of your infill, then there are several settings you may want to check.

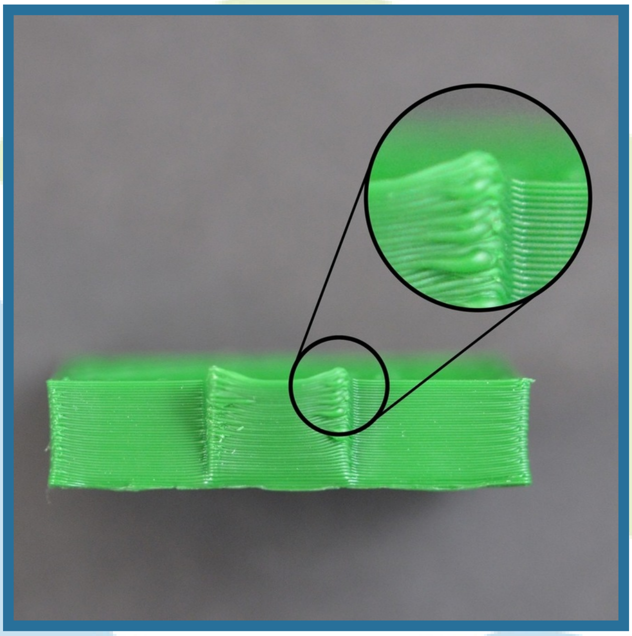
15.1. Not enough outline overlap

Slicing sofwares include a setting that allows you to adjust the strength of the bond between the perimeter outlines and the infill. This setting is called the "Outline overlap" and determines how much of the infill will overlap with the outline to join the two sections together. Usually the setting is based on a percentage of your extrusion width, so that it easily scales and adjusts for different nozzle sizes. For example, if you are using a 20 % outline overlap, it means that the software will instruct the printer so that the infill overlaps with 20 % of the inner-most perimeter. This overlap helps to ensure a strong bond between the two sections. As an example, if you were previously using an outline overlap of 20 %, try increasing that value to 30 % to see if the gaps between your perimeters and infill disappear.

15.2. Printing too fast

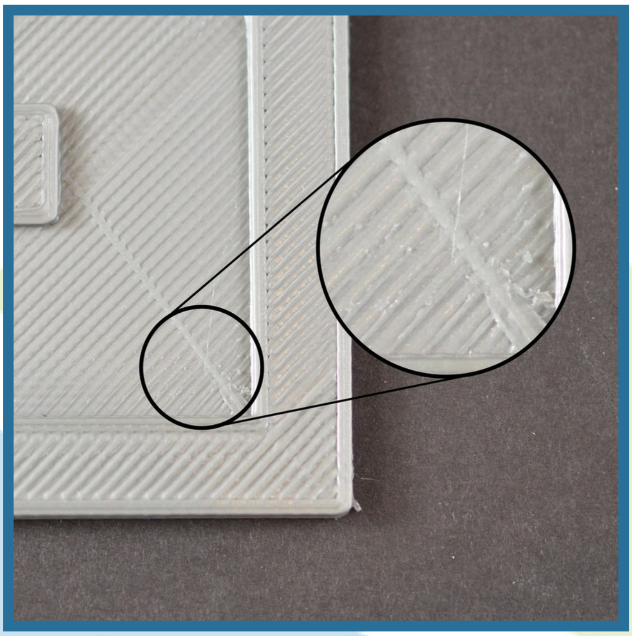
The infill for your part is generally printed much faster than the outlines. However, if the infill is printed too fast, it will not have enough time to bond to the outline perimeters. If you have tried increasing the outline overlap, but you are still seeing gaps between your perimeters and infill, then you should try decreasing the print speed. Adjust the speed of any movements where the extruder is actively extruding plastic. For example, if you were previously printing at 3600 mm/min (60 mm/s), try decreasing that value by 50 % to see if the gaps between your perimeters and infill disappear. If the gaps are no longer present at the lower speed, gradually increase the default printing speed until you find the best speed for your printer.

16. Curling or Rough Corners



If you are seeing curling issues later on in your print, it typically points to overheating issues. The plastic is extruded at a very hot temperature, and if it does not cool quickly, it may change shape over time. Curling can be prevented by rapidly cooling each layer so that it does not have time to deform before it has solidified. Please read the Overheating section for a more detailed description of this issue and how to resolve it. If you are noticing the curling at the very beginning of your print, please see the Print Not Sticking to the Bed section to address first layer issues.

17. Scars on Top Surface



One of the benefits of 3D printing is that each part is constructed one layer at a time. This means that for each individual layer, the nozzle can freely move to any portion of your print bed, since the part is still being constructed down below. While this provides for very fast printing times, you may notice that the nozzle leaves a mark when it travels on top of a previously printed layer. This is typically most visible on the top solid layers of your part. These scars and marks occur when the nozzle tries to move to a new location, but ends up dragging across previously printed plastic. The section below will explore several possible causes for this and provide recommendations for what settings can be adjusted to prevent it from happening.

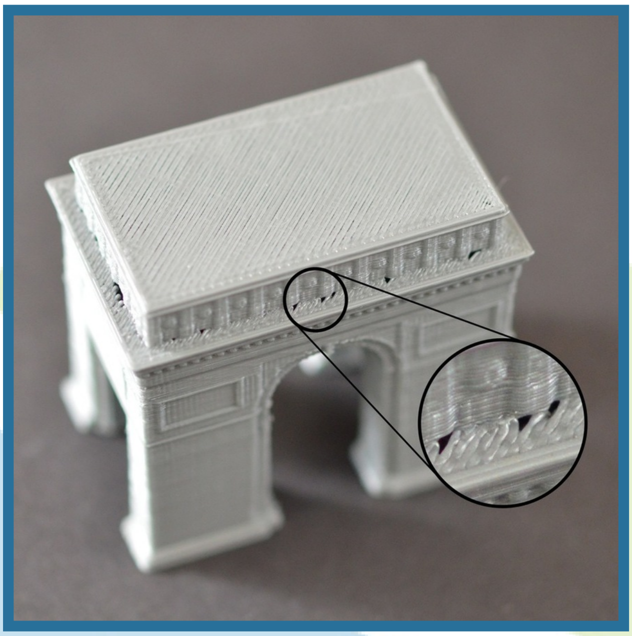
17.1. Extruding too much plastic

One of the first things you should verify is that you are not extruding too much plastic. If you extrude too much plastic, each layer will tend to be slightly thicker than intended. This means that when the nozzle tries to move across each layer, it may drag through some of the excess plastic. Before you look at any other settings, you should make sure that you are not extruding too much plastic. Please read the Extruding Too Much Plastic section for more details.

17.2. Vertical lift (Z-hop)

If you know you are extruding the correct amount of plastic, but are still having trouble with the nozzle dragging across your top surface, then it might be worth looking at the vertical lift settings. Enabling this option will cause the nozzle to lift up a set distance above the previously printed layer before moving to a new location. When it arrives at its final location, the nozzle will lower back down to prepare for printing. By moving at an elevated height, this can avoid the nozzle scratch on the top surface of your print. For example, if you set 0.5mm, the nozzle will always raise up 0.5mm before moving to a new location.

18. Holes and Gaps in Floor Corners



When building a 3D printed part, each layer relies on the foundation from the layer below. However, the amount of plastic that is used for the print is also a concern, so a balance must be achieved between the strength of the foundation and the amount of plastic that is used. If the foundation is not strong enough, you will start to see holes and gaps between the layers. This is typically most obvious in the corners, where the size of the part is changing (for example, if you were printing a 20mm cube on top of a 40mm cube). When you transition to the smaller size, you need to make sure that you have a sufficient foundation to support the sidewalls of the 20mm cube. There are several typical causes for these weak foundations. We will discuss each one below and present the settings that can be used to improve the print.

18.1. Not enough perimeters

Adding more outline perimeters to your part will greatly improve the strength of the foundation. Because the interior of your part is typically partially hollow, the thickness of the perimeter walls has a significant effect. For example, if you were previously printing with two perimeters, try the same print with four perimeters to see if the gaps disappear.

18.2. Not enough top solid layers

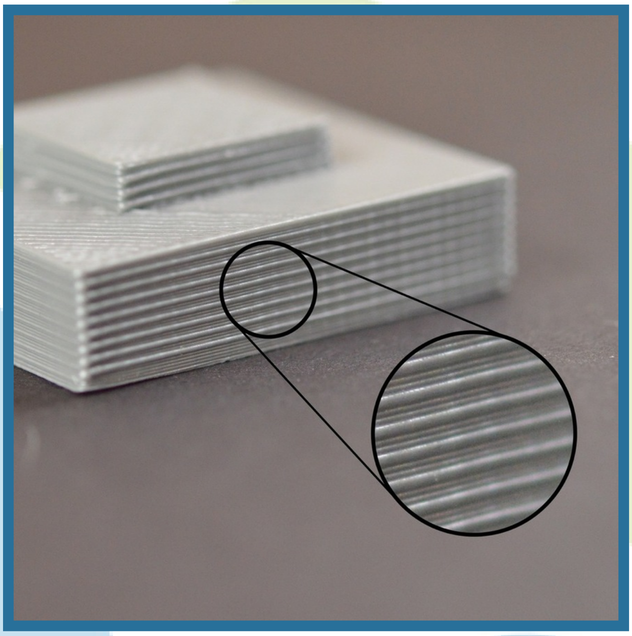
Another common cause for a weak foundation is not having enough solid layers for the top surfaces of your print. A thin ceiling will not be able to adequately support the structures that are printed on top of them. If you were previously using only two top solid layers, try the same print with four top solid layers to see if the foundation is improved.

18.3. Infill percentage is too low

The final setting you should check is the infill percentage used for your print. The top solid layers will be built on top of the infill, so it is important that there is enough infill to support these layers.

For example, if you were previously using a infill percentage of 20 %, try increasing that value to 40 % to see if the print quality improves.

19. Lines on the Side of Print



The sides of your 3D printed part are composed of hundreds of individual layers. If things are working properly, these layers will appear to be a single, smooth surface. However, if something goes wrong with just one of these layers, it is usually clearly visible from the outside of the print. These improper layers may appear to look like lines or ridges on the sides of your part. Many times the defects will appear to be cyclical, meaning that the lines appear in a repeating pattern (i.e. once every 15 layers). The section below will look at several common causes for these issues.

19.1. Inconsistent extrusion

Please read the Inconsistent Extrusion section.

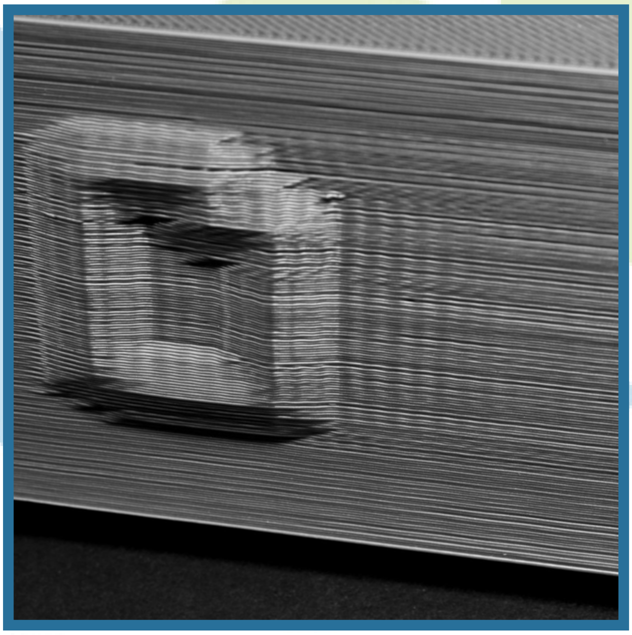
19.2. Temperature variation

Most 3D printers use a PID controller to regulate the temperature of the extruder. If this PID controller is not tuned properly, the temperature of the extruder may fluctuate over time. Due to the nature of how PID controllers work, this fluctuation is frequently cyclical, meaning that the temperature will vary with a sine wave pattern. As the temperature gets hotter, the plastic may flow differently than when it is cooler. This will cause the layers of the print to extrude differently, creating visible ridges on the sides of your print. A properly tuned printer should be able to maintain the extruder temperature within +/-2 degrees. During your print, you can use machine control panel to monitor the temperature of your extruder. If it is varying by more than 2 degrees, you may need to recalibrate your PID controller. Please consult your printer manufacturer for exact instructions on how to do this.

19.3. Mechanical issues

If you know that inconsistent extrusion and temperature variation are not to blame, then there may be a mechanical issue that is causing lines and ridges on the sides of your print. For example, if the print bed wobbles or vibrates while printing, this can cause the nozzle position to vary. This means that some layers may be slightly thicker than others. These thicker layers will produce ridges on the sides of your print. Another common issue is a Z-axis threaded rod that is not being positioned properly. For example, due to backlash issues or poor motor controller micro-stepping settings. Even a small change in the bed position can have a major impact on the quality of each layer that is printed.

20. Vibrations and Ringing



Ringing is a wavy pattern that may appear on the surface of your print due to printer vibrations or wobbling. Typically, you will notice this pattern when the extruder is making a sudden direction change, such as near a sharp corner. For example, if you were printing a 20mm cube, each time the extruder changes to printing a different face of the cube, it would need to change directions. The inertia of the extruder can create vibrations when these sudden direction changes occur, which will be visible of the print itself. We will look at the most common ways to address ringing, by examining each cause in the list below.

20.1. Printing too fast

The most common cause for ringing is that your printer is trying to move too fast. When the printer suddenly changes direction, these quick movements will create additional force that can cause the lingering vibrations. If you feel that your printer may be moving too fast, try to reduce the printing speed.

20.2. Firmware acceleration

The firmware that runs on your 3D printer's electronics typically implements acceleration controls to help prevent sudden direction changes. The acceleration settings will cause the printer to slowly ramp up in speed and then to slowly decelerate before changing directions. This functionality is vital for preventing ringing. If you are comfortable working with your printer's firmware, you may even want to try decreasing the acceleration settings so that the speed changes more gradually. This can help reduce ringing even further.

20.3. Mechanical issues

If nothing else has been able to resolving the ringing issues, then you may want to look for mechanical issues that could be causing the excessive vibrations. For example, there could be a loose screw or a broken bracket that is allowing excessive vibrations to occur. Watch your printer closely while it is running and try to identify where the vibrations are coming from. Many users eventually trace these issues back to mechanical problems with the printer.

21. Gaps in Thin Walls

Because your 3D printer includes a fixed size nozzle, you may encounter issues when printing very thin walls that are only a few times larger than the nozzle diameter. For example, if you were trying to print a 1.0mm thick wall with a 0.4mm extrusion width, you may need to make some adjustments to ensure your printer creates a completely solid wall and does not leave a gap in the middle. Slicing softwares usually include several dedicated settings to help with thin wall printing, so we will describe the relevant settings below.

21.1. Adjust the thin wall behavior

The first settings that you need to verify are the dedicated thin wall settings. The slicer usually includes several different options for the Internal Thin Wall Type. The default option typically uses something called "gap fill" to fill the small gaps between your thin walls. This will create a back-and-forth infill pattern that adjusts to fill the space between these thin gaps. However, the software may also include another useful option that can fill these thin walls with a single pass. This will use a dynamic single extrusion that will adjust in size to perfectly fill the gap between these walls.

21.2. Change the extrusion width to fit better

In some cases, you may find that you have better luck changing the size of the plastic that is extruded from the nozzle. For example, if you were printing a 1.0mm thick wall, you could achieve a fast and strong print if your nozzle was setup to create a 0.5mm extrusion. This works best for parts that have fairly consistent wall thicknesses. You can adjust the extrusion width that the software creates, choose a manual extrusion width and set a value of your choosing.

22. Very Small Features Not Being Printed

Most 3D printers have a fixed nozzle size that determines the part resolution in the XY direction. Popular nozzle sizes are 0.4mm or 0.5mm in diameter. While this works well for most parts, you may start to encounter issues when trying to print extremely thin features that are smaller than the nozzle size. For example, if you were trying to print a 0.2mm thick wall with a 0.4mm diameter nozzle. If you frequently need to print extremely thin features, we will explain the best options to consider for these micro-sized prints.

22.1. Enable single extrusion walls

Slicing sofware usually includes a specialty printing mode specifically for very thin walls and exterior features. To enable this special mode change the External Thin Wall Type to "Allow single extrusion walls".

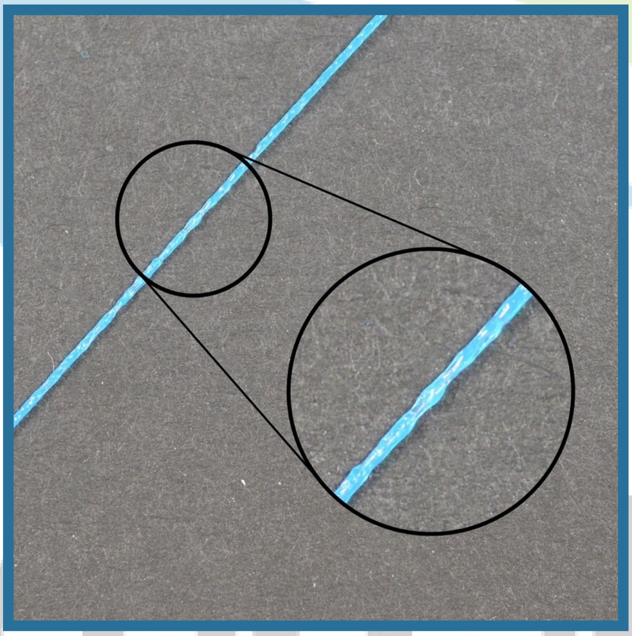
22.2. Redesign the part to have thicker features

If you are still having issues printing these thin features, another option is to redesign the part so that it only includes features that are larger than your nozzle diameter. This typically involves editing the 3D model in the original CAD package to modify the size of the small features. Once you have thickened the small features, you can re-import the model into the slicer to verify that your printer is capable of reproducing the 3D shape you created.

22.3. Install a nozzle with a smaller tip size

In many cases, you are not able to modify the original 3D model. For example, it may be a part that someone else designed or one that you downloaded from the internet. In this case, you may want to consider obtaining a second nozzle for your 3D printer that allows it to print smaller features. Many printers have a removable nozzle tip, which makes these aftermarket adjustments quite easy. For example, many users purchase a 0.3mm nozzle as well as a 0.5mm nozzle to provide two options. Consult your printer manufacturer for exact instructions on how to install a smaller nozzle tip size.

23. Inconsistent Extrusion

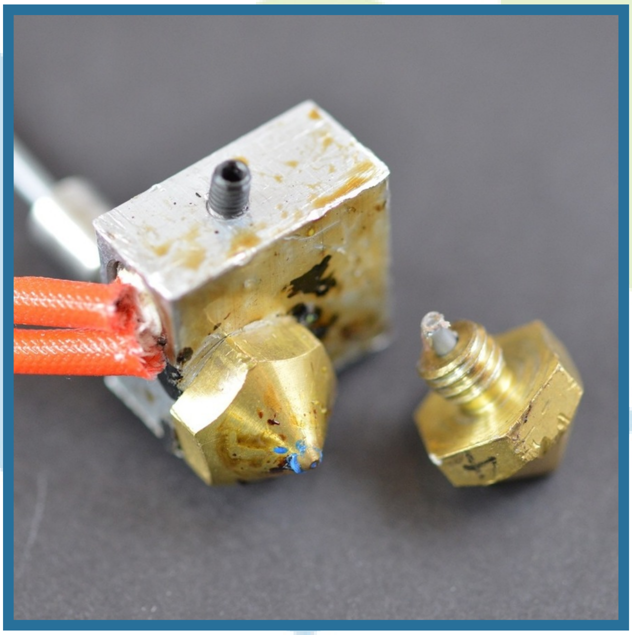


For your printer to be able to create accurate parts, it needs to be capable of extruding a very consistent amount of plastic. If this extrusion varies across different parts of your print, it is going to affect the final print quality. Inconsistent extrusion can usually be identified by watching your printer closely as it prints. For example, if the printer is printing a straight line that is 20mm long, but you notice that the extrusion seems rather bumpy or seems to vary in size, then you are likely experiencing this issue. We have summarize the most common causes for inconsistent extrusion, and explained how each one can be addressed.

23.1. Filament is getting stuck or tangled

The first thing you should check is the spool of plastic that is feeding into your printer. You need to make sure that this spool is able to rotate freely and that the plastic is easily being unwound from the spool. If the filament becomes tangled, or the spool has too much resistance to spin freely, it will impact how evenly the evenly the filament is extruded through the nozzle. If your printer includes a Bowden tube (a small hollow tube that the filament is routed through), you should also check to make sure that the filament can easily move through this tube without too much resistance. If there is too much resistance in the tube, you may want to try cleaning the tube or applying some lubrication inside the tube.

23.2. Clogged extruder



If the filament is not tangled and can easily be pulled into the extruder, then the next thing to check is the nozzle itself. It is possible that there is some small debris or foreign plastic inside the nozzle that is preventing proper extrusion. An easy way to check this is to use the printer's control panel to manually extrude some plastic from the nozzle. Watch to make sure that the plastic is extruding evenly and consistently. If you notice problems, you may need to clean the nozzle. Please consult your manufacturer for instructions on how to properly clean the inside of the nozzle.

23.3. Very low layer height

If the filament is spinning freely and the extruder is not clogged, it may be useful to check a few settings within your slicer. For example, if you are trying to print at an extremely low layer height, such as 0.01mm, there is very little room for the plastic to exit the nozzle. This gap below the nozzle is only 0.01mm tall, which means that the plastic may have a difficult time exiting the extruder. Double check to make sure you are using a reasonable layer height for your printer. If you are printing at a very small layer height, try increasing the value to see if the problem goes away.

23.4. Incorrect extrusion width

Another setting to check within the slicing software is the extrusion width that you have specified for your extruder. Each extruder can have its own unique extrusion width, so make sure you select the appropriate extruder. If you extrusion width is significantly smaller than your nozzle diameter, this may cause extrusion issues. As a general rule of thumb, the extrusion width should be within 100-150 % of the nozzle diameter. If your extrusion width is far below the nozzle diameter (for example, a 0.2mm extrusion width for a 0.4mm nozzle), then your extruder won't be able to push a consistent flow of filament.

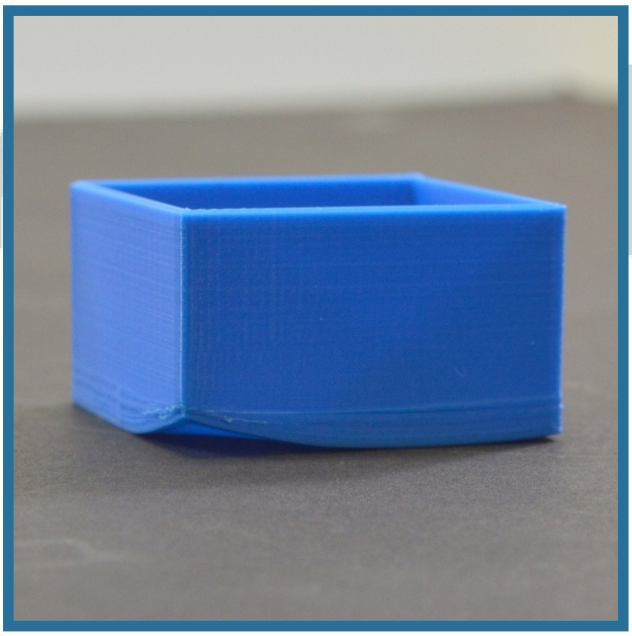
23.5. Moistured filament

One of the most common causes for inconstant extrusion that we have not mentioned yet is the condition of the filament that you are printing with. Many plastics also have a tendency to degrade over time. For example, PLA tends to absorb moisture from the air, and over time, this will cause the print quality to degrade. This is why many spools of plastic include a desiccant in the packaging to help remove any moisture from the spool. If you think your filament may be at fault, try swapping the spool for a new, unopened, spool to see if the problem goes away.

23.6. Mechanical extruder issues

If you have verified everything above and are still having problems with inconsistent extrusion, then you may want to check for mechanical issues with your extruder. For example, many extruders use a drive gear with sharp teeth that bite into the filament. This allows the extruder to move the filament back and forth easily. These extruders also typically include an adjustment that changes how hard the drive gear is pressed into the filament. If this setting is too loose, the drive gear teeth won't cut far enough into the filament, which impacts the extruder's ability to accurately control the position of the filament. Check with your manufacturer to see if your printer has a similar adjustment.

24. Warping



As you start printing larger models, you may start to notice that even though the first few layers of your part successfully adhered to the bed, later on the part begins to curl and deform. This curling can be so severe that it actually causes part of your model to separate from the bed, and may cause the entire print to eventually fail. This behavior is particular common when printing very large or very long parts with high temperature materials such as ABS. The main reason for this problem is the fact that plastic tends to shrink as it cools. For example, if you printed an ABS part at 230C and then allowed it to cool to room temperature, it will shrink by almost 1.5 %. For many large parts, this could equate to several millimeters of shrinkage! As the print progresses, each successive layer will deform a bit more until the entire part curls and separates from the bed. This can be a challenging issue to solve, but we have several helpful suggestions to get you started.

24.1. Use a Heated Bed

Many machines come equipped with a heated bed that can help keep the bottom layers of your part warm throughout the print. For materials such as ABS, it is common to set the heated bed temperature to 100-120C, which will significantly reduce the amount of plastic shrinkage in these layers.

24.2. Disable Fan Cooling

By now, you probably realize that cooling can be a problem for parts that tend to warp. For this reason, many users prefer to disable any external cooling fans entirely when printing with materials such as ABS. This allows all of the layers to stay warm for a longer period of time, increasing your chance of success.

24.3. Use a Heated Enclosure

While a heated bed can keep the bottom layers of your part warm, it may struggle to keep the upper layers of the part from contracting once you start printing taller and taller objects. In this situation, you may find it useful to place your printer inside of an enclosure that can help regulate the temperature of the entire build volume. Some machines may already include an external enclosure specifically for this reason. If your machine does include a heated enclosure, make sure to keep the doors closed during the print, which will keep the heat from escaping.

24.4. Brims and Rafts

If you have already tried all of the other suggestions, but your parts are still curling later on in the print, then you can also try including a brim or a raft with your print. These features will help hold the edges down and may warp less, since they are typically only a few layers tall.

25. Poor Surface Above Supports

One of the major benefits of modern slicing softwares is the ability to create innovative support structures which allow you to create incredibly complex parts that would be hard to manufacture otherwise. For example, if you have a steep overhang or part of your model with nothing below it, then a support structure can provide a foundation for these layers. The support structures created are disposable and can be easily separated from the final part. However, depending on your settings, you may find that some adjustments are needed to perfect the surface quality on the underside of your parts, right above the support structure foundation. We will explain the key settings below and how they can affect your prints.

25.1. Lower Your Layer Height

The overhang performance of your printer can be greatly improved by lowering your layer height. For example, if you reduced your layer height from 0.2mm to 0.1mm, your printer will create twice as many layers, which allows your printer to take smaller steps when creating an overhang. For this reason, your may find that you need support structures for any overhang above 45 degrees when using a 0.2mm layer height, but your overhang performance may improve to 60 degrees if you lower your layer height to 0.1mm. This has the obvious advantage of decreasing your print time and reducing the amount of support structures required for the print, but it will also allow you to create a smoother surface on the underside of your parts. If you find that you need to increase the print quality in this area, this is one of the first settings you will want to adjust.

25.2. Support Infill Percentage

Just like the interior of your part, you can also adjust the density of your support structures by changing the Support Infill Percentage. It is common to use a value around 20-40 %, but you may find that you need to increase this value if the bottom layers of your part are drooping too much. Many users also prefer to use Dense Support Structures for this task, as they allow you to use a lower density for the majority of your supports, and only use a higher infill percentage near the very top of the support structures.

25.3. Vertical Separation Layers

Creating removable support structures involves a fine balance between the amount of support provided to the model, and how easy the supports are to remove. If you provide too much support to the model, the support structures may start to bond to the part, making them difficult to separate. If you provide very little support, the disposable support structures will be easy to remove, but the part may not have enough of a foundation to print successfully. Usually the slicer allows your to customize the separation settings, so that you can choose the correct balance between these different factors. The first setting you will want to check is the Upper Vertical Separation Layers. This setting determines how many empty layers are left between the support structures and the part. For example, if you are printing your support structures with the same material as your part, it is common to use at least 1-2 vertical separation layers. Otherwise, if you used 0 separation layers and you are printing everything with the same material, the supports may bond to the part and can become difficult to remove. So this is one of the first settings you want to adjust as you try to perfect your print quality.

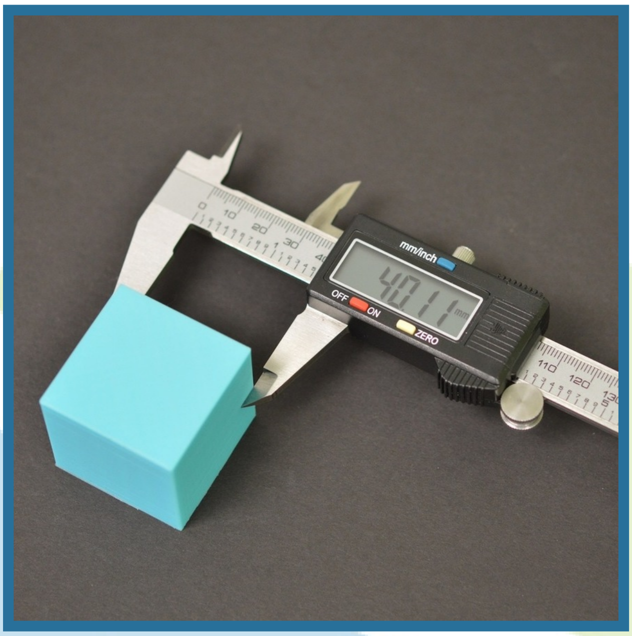
25.4. Horizontal Part Offset

The next separation setting you should check is the Horizontal Offset from your part. This setting controls the side-to-side distance between your part and the support structures. So while the Vertical Separation Layers can help keep the top of your supports from bonding to the bottom of your part, the Horizontal Offset will keep the sides of your supports from bonding to the side of your model. It is common to use a value between 0.2-0.4mm for this setting, but you may need to experiment and see what works best for your specific extruder and filament.

25.5. Use a Second Extruder

If your machine comes with 2 or more extruders, you can achieve a significant improvement by using a different material for your support structures. For example, it is quite common to print parts in PLA using water dissolvable PVA for the supports. Because the model and support structures are printed with different materials, they won't bond together as easily, which allows you to do a better job of supporting the part. If you are using a different material for the support structures, you can frequently decrease your Upper Vertical Separation Layers to zero, and reduce your Horizontal Offset from the part to around 0.1mm.

26. Dimensional Accuracy



The dimensional accuracy of your 3D printed parts can be extremely important if you are creating large assemblies or parts that need to precisely fit together. There are many common factors that can affect this accuracy such as under or over-extrusion, thermal contraction, and even the first layer nozzle alignment. Usually slicing software includes several tools to help cope with these common issues, so we will explain each one in more detail below.

26.1. First Layer Impact

Settings for your first layer can have an impact on dimensional accuracy. If your nozzle is too high or too low for the first layer of your print, it can drastically affect the next 10-20 layers of the part. For example, if you are printing a 0.2mm thick layer, but your nozzle is only positioned 0.1mm away from the bed, then this extra plastic may create a first layer that is a bit too large. Future layers can also be affected by the extra plastic on this layer, which creates several oversized layers at the bottom of the part. So before you spend too much time trying to perfect the dimensional accuracy of your prints, you need to verify that your measurements are not being affected by the first layer position. One common way to do this is by printing a model with 50-100 layers and then only measuring the top 20 or so layers. These top layers are far away from the very first layer that was printed on the bed, so it minimizes the impact of nozzle positioning. Before proceeding to the sections below, make sure that your measurements follow these guidelines.

26.2. Under or Over-Extrusion

Now that you know you are using accurate measurements that are not affected by the first layer position, the next setting you want to verify is your extrusion multiplier. This setting affects the flow rate for the entire print. If the extrusion multiplier is too low, you may start to see gaps between perimeters, holes in your top surfaces, and parts that are smaller than their intended size. If your extrusion multiplier is too high, you may notice top layers that tend to bulge upwards and parts that are larger than intended. So again, before proceeding to the sections below you will want to verify that your extrusion multiplier is properly calibrated. For more advice on these topics, please see the Under-Extrusion and Over-Extrusion sections.

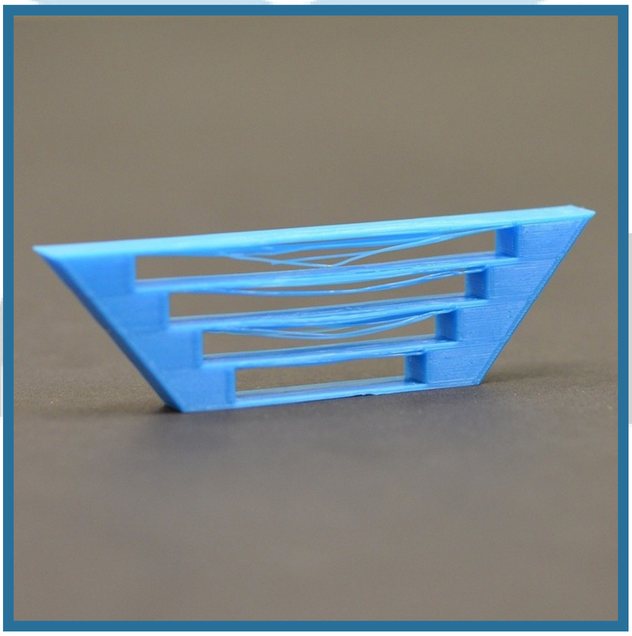
26.3. Constant Dimensional Error

If you have completed the steps above and the prints are still not sized correctly, some slicers offer the ability to precisely offset the edges of your print to account for these differences. This setting is called "Horizontal size compensation". For example, setting this value to -0.1mm will shrink your model by 0.1mm in the X and Y directions. This setting works best when the dimensional error is consistent, even when printing models of different sizes. For example, if the part is always 0.1mm too large, regardless of if the model is 20mm wide or 100mm wide, then this setting can easily account for that difference.

26.4. Increasing Dimensional Error

If you notice that the dimensional error tends to increase as you print larger parts, then there is a different setting you can adjust. For example, if your print was 0.1mm too small for a 20mm wide part, but increased to 0.5mm too small for a 100mm wide print, then it is likely the problem may be due to thermal contraction. This can be a common issue for high temperature materials like ABS, since plastic tends to shrink as it cools. Some slicers include options to help with this. First, you need to determine the shrinkage percentage. In the above example, the part is shrinking by 0.1mm over a 20mm print, so the shrinkage percentage is 0.1 / 20 = 0.5 %. The easiest way to fix this error is to scale te model to 100.5 %.

27. Poor Bridging



Bridging is a term that refers to plastic that needs to be extruded between two points without any support from below. For larger bridges, you may need to add support structures, but short bridges can typically be printed without any supports to save material and print time. When you are bridging between two points, the plastic will be extruded across the gap and then quickly cooled to create a solid connection. To get the best bridging results, you will need to make sure that your printer is properly calibrated with the best settings for these special segments. If you notice sagging, drooping, or gaps between the extruded segments, you may need to adjust your settings for the best results. We will cover each of the areas that you want to address to make sure your can print the best bridges possible on your 3D printer.

27.1. Verify bridging settings are being used

Bridging segments are identified by the slicing sofware. The slicing software usually allows to set specific parameters for the bridging segments. Some slicers allows also to ignore very small bridging areas and focus on the larger bridging regions that may need special settings. If you think your bridging area is not being included, make sure that the area of your bridging region is larger than the threshold value.

27.2. Check the angle used for the bridging infill

Usually the slicer will automatically calculate the best infill direction to use for your bridging regions. For example, if you are bridging between two pillars aligned on the X-axis, the software will automatically change the infill direction for that area to ensure the infill is also being extruded along the X-axis. This greatly improves your odds of success, so if you notice that you are getting poor bridging results, you want to double-check to make sure the infill is oriented in the correct direction. If you ever want to try a different infill angle for these bridging layers, some slicers allow to do so.

27.3. Adjust settings for optimal performance

The bridging regions are printed with special extrusion, speed, and cooling settings to achieve optimal performance. The extrusion and speed adjustments for these regions usually can be set separately. Typically, you will want to set the "Bridging extrusion multiplier" to 100 % or more, as lower values may have trouble properly sealing the bottom of these surfaces. The "Bridging speed multiplier" may require some experimentation, as some printers will perform better with slow bridging, while others get better results by moving quickly. Finally, you can set the bridging fan speed settings. Typically, you will want to set the bridging fan speed to a large value to make sure the bridges are cooled as quickly as possible. Experiment with these settings to find the best combination for your specific 3D printer and filament. There are many bridging test models available that can help with this calibration.

27.4. Use supports for longer bridges

In the event that you are unable to get the results you want after tuning the settings mentioned above, you may find that adding support structures will allow you to achieve the best quality. The support structures will provide an extra foundation for the bridging regions, greatly improving their odds of success. You can enable support generation for the entire model, or if your slicer allows it, add customizable support structures for extra control.

Website

Contact us

Guangzhou Yousu 3D Technology Co., Ltd

Tel:0086 20 32093406 Email:sales@ysfilament.com Http://www.ysfilament.com

:https://www.facebook.com/Yousu3D/ :@Guangzhou_YouSu

:@yousu_create3d