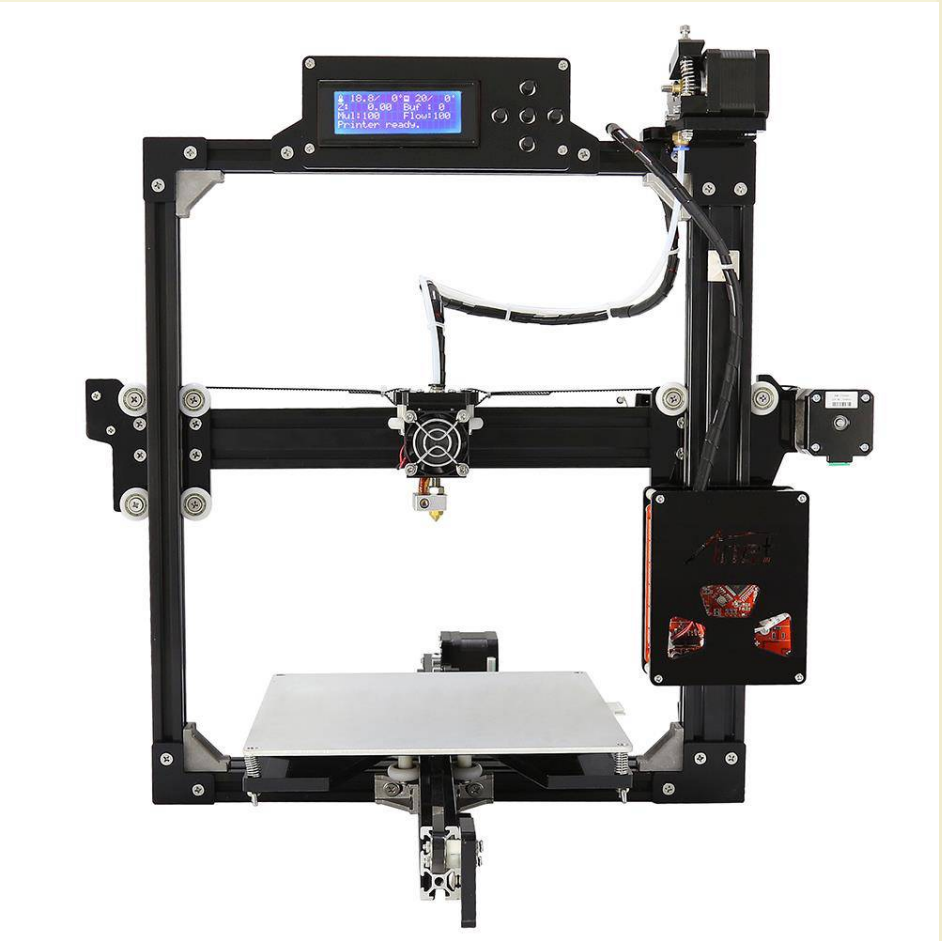
3 D Print Quality Troubleshooting Guide



Shenzhen Anet Technology Co,. Ltd.

Print Quality Troubleshooting Guide 1

Overview 1

1. Not Extruding at Start of Print 3

    1.1 Filament was not loaded into extruder before printing 4

    1.2 The distance between nozzle and bed is too close 4

    1.3 The filament stripped against the drive gear 5

    1.4 The extruder is clogged 5

2. Print Not Sticking to the Bed 6

    2.1 Build platform is not level 7

    2.2 Nizzle starts too far away from the bed 7

    2.3 First layer is printing too fast 8

    2.4 Temperature or cooling settings 8

    2.5 The build platform (tape, glues and materials) 9

    2.6 When all else fails: Brims and Rafts 10

3. Not Extruding Enough Plastic 11

    3.1 Incorrect filament diameter 12

    3.2 Increase the extrusion speed 12

4. Extruding Too Much Plastic 14

5. Holes and Gaps in the Top Layers 15

    5.1 Not enough top solid layers 16

    5.2 Infill percentage is too low 17

    5.3 Under-extrusion 17

6. Stringing or Oozing 18

    6.1 Retraction distance 19

    6.2 Retraction speed 19

    6.3 Temperature is too high 20

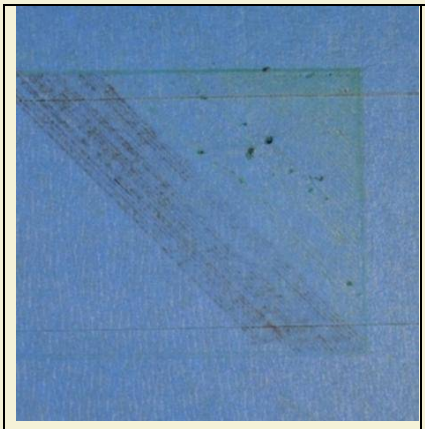
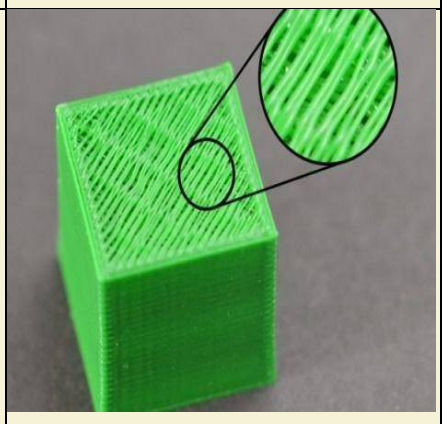
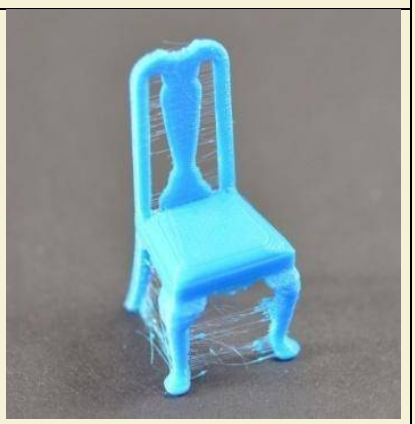
7. Overheating 21
7.1 Insufficient cooling 22
7.2 Print at too high temperature 22
7.3 Printing too fast 23
7.4 When all above fall, try to print multiple parts at once 23
8. Layer Shifting or Misalignment 24
8.1 Nozzle moves too fast 25
8.2 Mechanical or Electrical issues 25
9. Layer Separation and Splitting 27
9.1 Layer height is too large 28
9.2 Print temperature is too low 28
10. Grinding Filament 30
10.1 Increase the extruder temperature 31
10.2 Print too fast 31
10.3 Check for a nozzle clog 32
11. Clogged Extruder 33
11.1 Manually push the filament to extruder 33
11.2 Reload the filament 33
11.3 Clean out the nozzle 34
12. Stops Extruding in the Middle of a Print 35
12.1 Out of filament 36
12.2 The filament has stripped against the drive gear 36
12.3 The extruder is clogged 36
12.4 Overheated extruder motor driver 37
13. Weak Infill 38
13.1 Increase infill density 39
13.2 Lower print speed 39

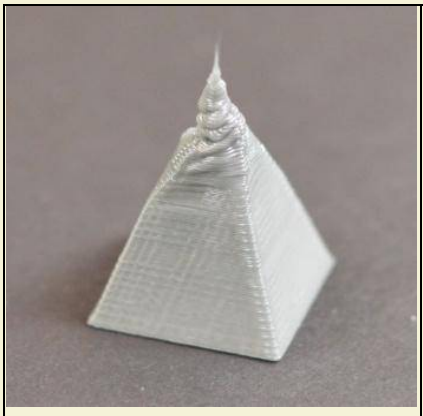
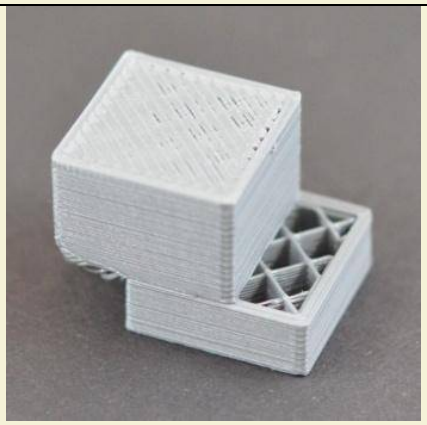
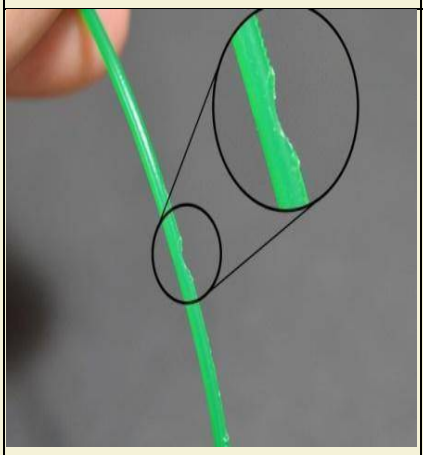
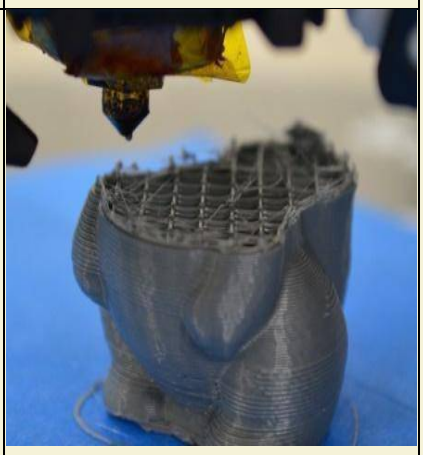
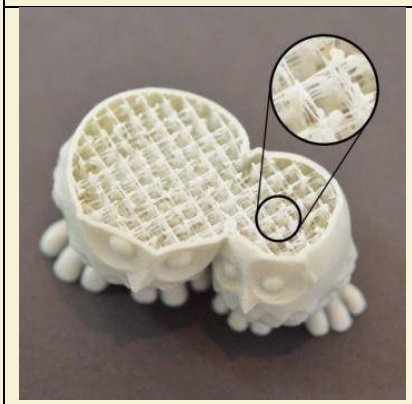
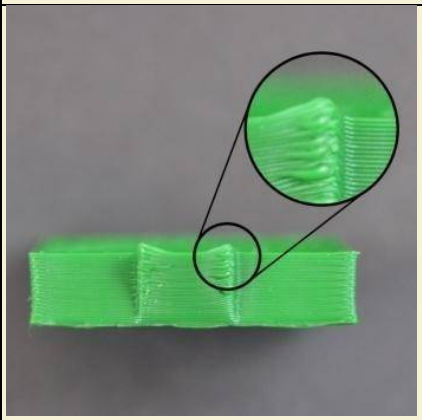
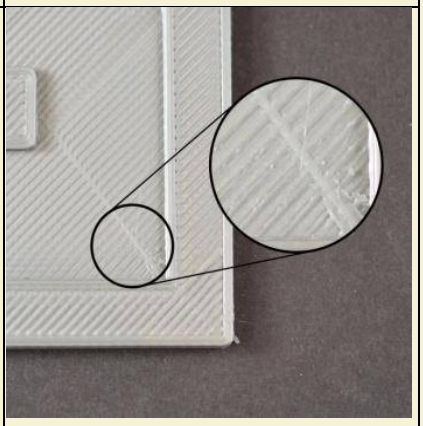
14. Curling or Rough Corners 40
15. Scars on Top Surface 41
15.1 Extruding too much plastic 42
16. Holes and Gaps in Floor Corners 43
16.1 Not enough perimeters 44
16.2 Not enough top solid layers 44
16.3 Infill density is too low 44
17. Lines on the Side of Print 45
17.1 Inconsistent extrusion 46
17.2 Temperature variation 46
17.3 Mechanical issues 47
18. Vibrations and Ringing 48
18.1 Print too fast 49
18.2 Firmware acceleration 49
18.3 Mechanical issues 49
19. Very Small Features Not Being Printed 50
19.1 Redesign the parts with thicker features 51
19.2 Install a nozzle with a smaller tip size 51
19.3 Force the software to print small features 52
20. Inconsistent Extrusion 53
20.1 Filament is getting stuck or tangled 54
20.2 Clogged extruder 54
20.3 Layer height is very low 54
20.4 Poor quality filament 55
20.5 Mechanical issues of extruder 55

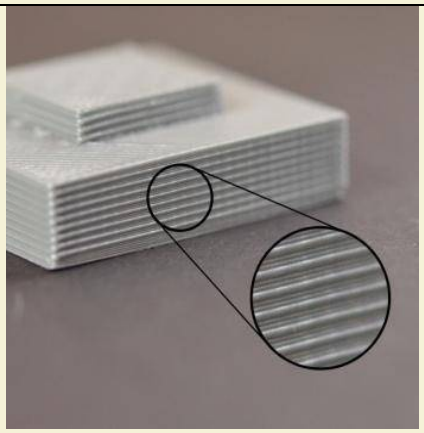
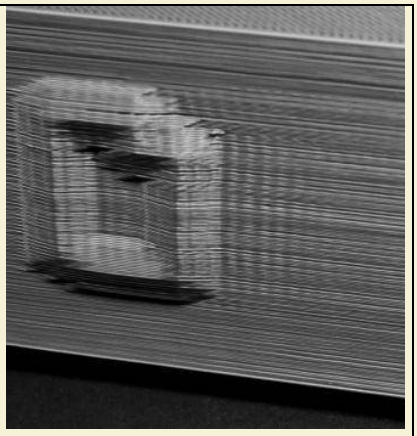
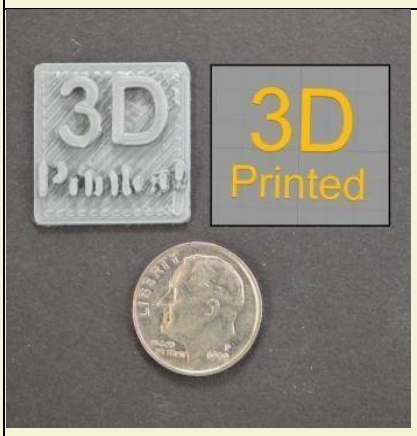
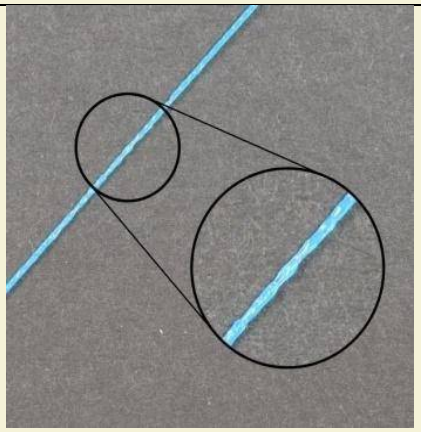
Print Quality Troubleshooting Guide

Overview

Use the thumbnails below to identify the print quality issue that you are seeing in your own 3D printed parts. You can click on the words under the thumbnail to jump that portion of the guide for immediate recommendations on how to resolve the issue. If you are not able to locate your issues from the thumbnails, scroll down and read through each section of the guide for more detail.

<table>
<tr>
<td> 
Not Extruding at Start of Print</td>
<td>
Print Not Sticking to the Bed</td>
<td>
Not Extruding Enough Plastic</td>
</tr>
<tr>
<td>
Extruding Too Much Plastic</td>
<td> 
Holes and Gaps in the Top Layers</td>
<td> 
Stringing or Oozing</td>
</tr>
</table>

 Overheating	 Layer shifting or Misalignment	Layer Separation and Splitting
 Grinding Filament	Clogged Extruder	 Stops Extruding Mid Print
 Weak Infill	 Curling or Rough Corners	 Scars on Top Surface

Gaps in Floor Corners	 Lines on the Side of Print	 Vibrations and Ringing
 Very Small Features Not Being Printed	 Inconsistent Extrusion	

1. Not Extruding at Start of Print

This issue is very common for new 3D printer owners, but thankfully, it is also very easy to resolve. There are four possible causes if your extruder does not extrude plastic at the beginning of your print. We will walk through each one below and explain how to solve the problem.

1.1 Filament was not loaded into extruder before printing

Most extruders have a problem of leaking plastic when they keep still 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 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 in Cura 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 on the Expert tab in Cura. You can also use control knob in Anet3D printer to extrude plastic before printing.

1.2 The distance between nozzle and bed is too close

If the nozzle is too close to the bed, 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 come out. 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. To solve this problem, you can adjust the

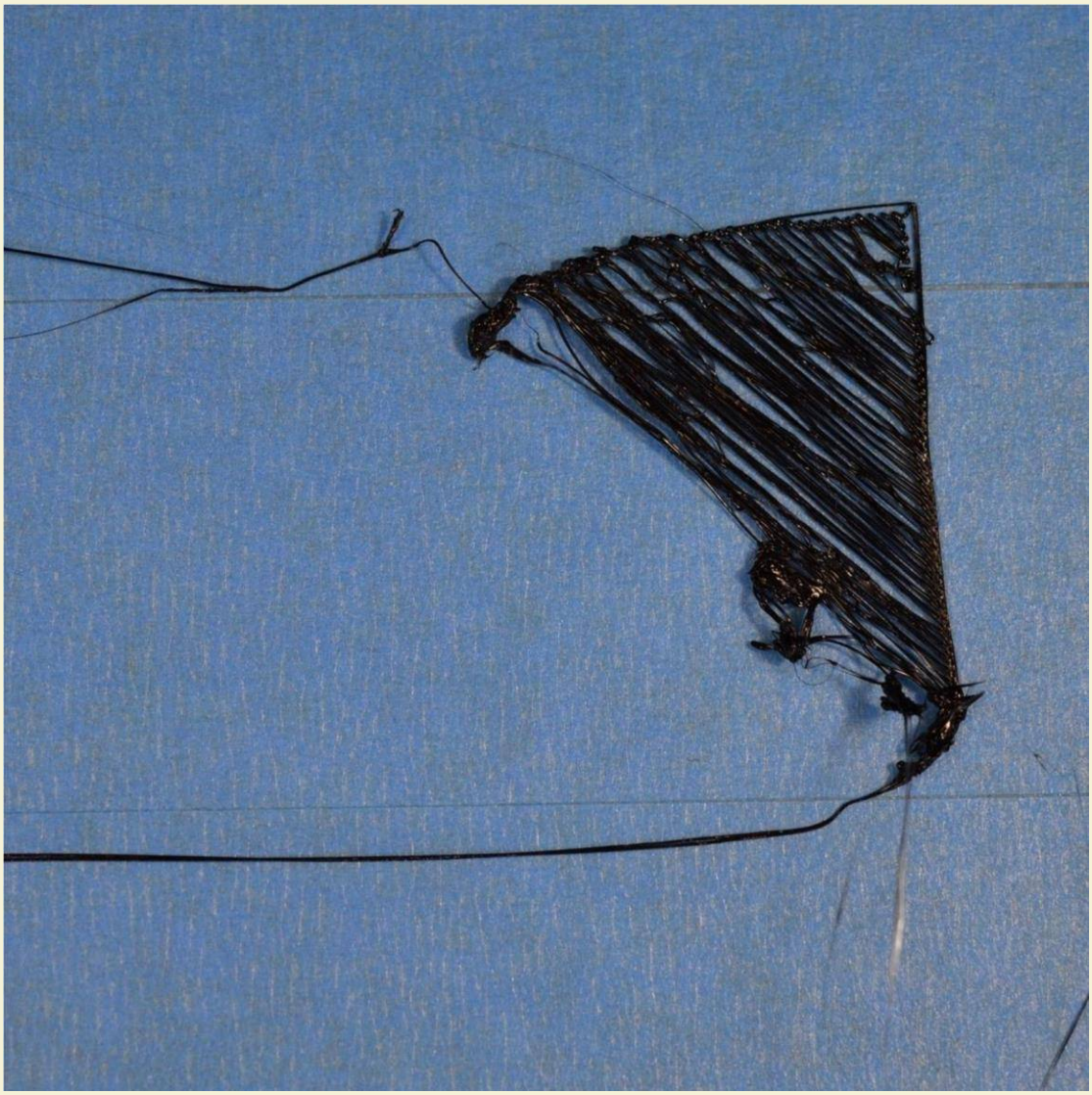
four screws at the corners of the hot bed, turn the screws clockwise and the hot bed will move down. Turn a little once to increase the distance until there is enough space for plastic extrusion (The distance is about the thickness of a piece of A4 Paper).

1.3 The filament stripped against the drive gear

Anet3D 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.

1.4 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. We successfully used relative diameter drill (or relative diameter copper line of electric wire) to unclog extruders by feeding it into the nozzle tip. You need to heat the extruder to 200 □ when you do this. Attention not hurt by the hot extruder. Please contact your printer manufacturer if you need disassemble the extruder.



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.

2.1 Build platform is not level

Anet3D printer uses 4 screws to control the position of the bed. If you have trouble to get your first layer to stick to the bed, the first thing you need 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. Anet3D printer already has a useful bed leveling video that guide you through the bed leveling process.

You can find this video by visiting website www.anet3d.com to Download Center and download the video or watch online.

2.2 Nizzle 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 adhesive against the build plate. You can adjust the four screws at the corners of the hot bed, turn the screws anticlockwise and the hot bed will move up. Turn a little once to reduce the distance until there is suitable room for filament extrusion(The distance is about the thickness of a piece of A4 paper). 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.

2.3 First layer is printing too fast

As you extrude the first layer of plastic, you want the plastic can properly bond to the surface of build platform 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. Cura provides a setting for this exact feature. If you click on “Advanced” and go to the Speed tab, you will see a setting labeled “Bottom Layer Speed”. For example, if you set a first layer speed of 20, it means that your first layer will print 20mm/s. If you feel that your printer is moving too fast on the first layer, try reducing this setting.

2.4 Temperature or cooling settings

Plastic tends to shrink when its temperature is down. 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. Anet3D printer also includes cooling fans that speed up this cooling process when they are being used. If this ABS part cooled down to a room temperature of 30□, 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.

In order to print high-temperature materials like ABS, Anet3D printer includes a heated bed to help combat these problems. If the bed it heated to maintain a temperature of 70-100□ for the entire print, it will keep the first layer warm so that it does not shrink. As a general starting point, PLA tends to adhere well to a bed that is heated to 40-6□, while ABS generally works better if the bed is heated to 70-100□. You can adjust these settings by the control panel. The specific setting process: Press the knob to enter the menu, and go to the Control tab, confirm, then go to the Bed tab, confirm, then turn the knob to adjust the temperature.

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. You can

read “Cura Tutorial” to learn how to resolve this issue.

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.

2.5 The build platform (tape, glues and materials)

Different plastics tend to adhere better to different materials. 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. 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). You can use

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!

2.6 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. Cura 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. This option can be enabled by going to the “Support” tab end enabling the “Platform adhesion type” option. Cura also allows users to add a raft under their part, which can also be used to provide a larger surface for bed adhesion. If you are interested in these options, please take a look at “Cura Tutorial”, which explains things in greater detail.

3. Not Extruding Enough Plastic

Cura includes some settings used to determine how much plastic the 3D printer should extruder. 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 and see whether 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.

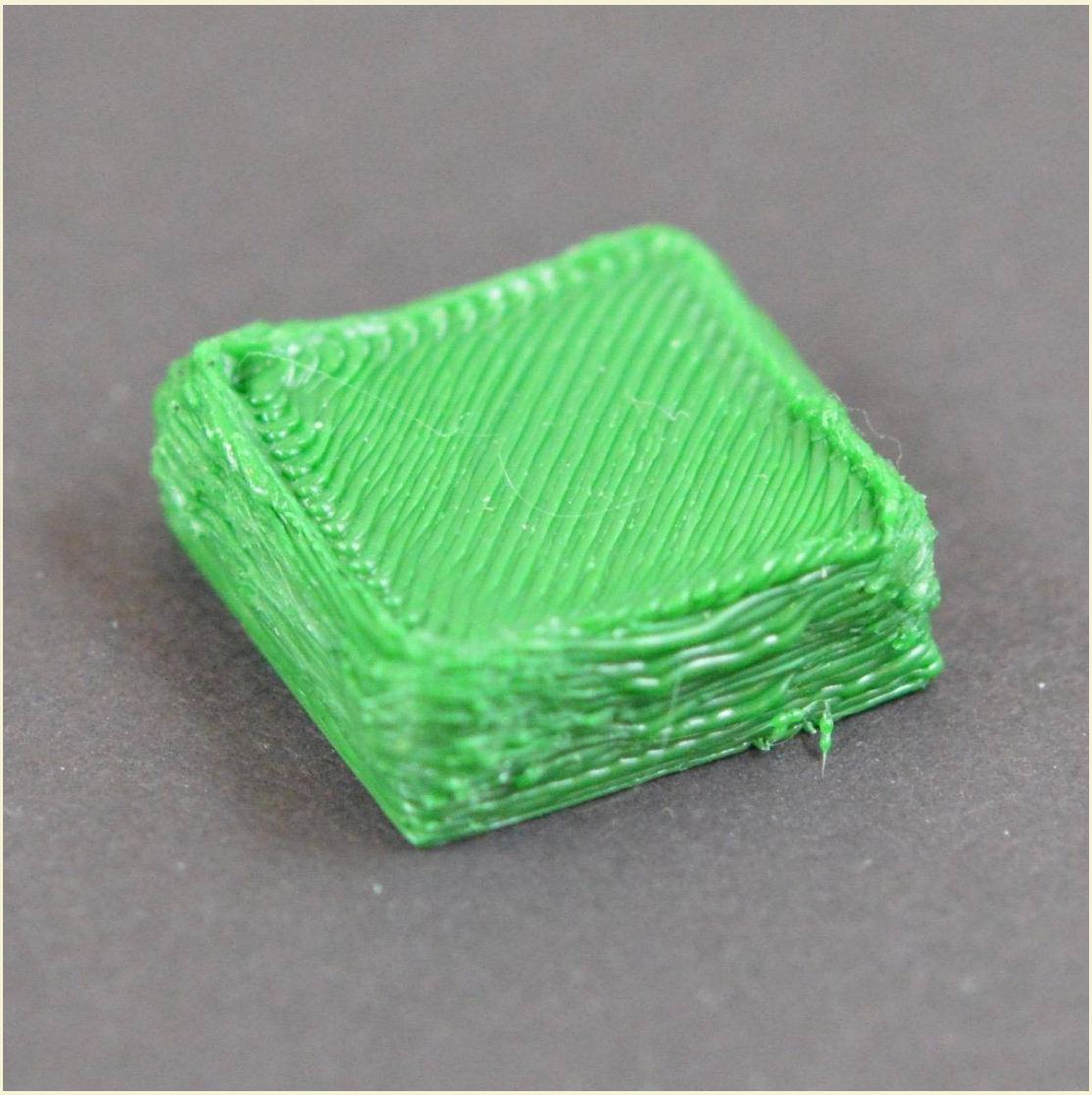
3.1 Incorrect filament diameter

The first thing you need to verify is that the software knows the filament diameter that you are using. You can find this setting by clicking “Basic” and going to the “Filament” tab. 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. The most common values for the filament diameter are 1.75mm and 2.85mm. The diameter of Anet3D printer filament is 1.75mm.

3.2 Increase the extrusion speed

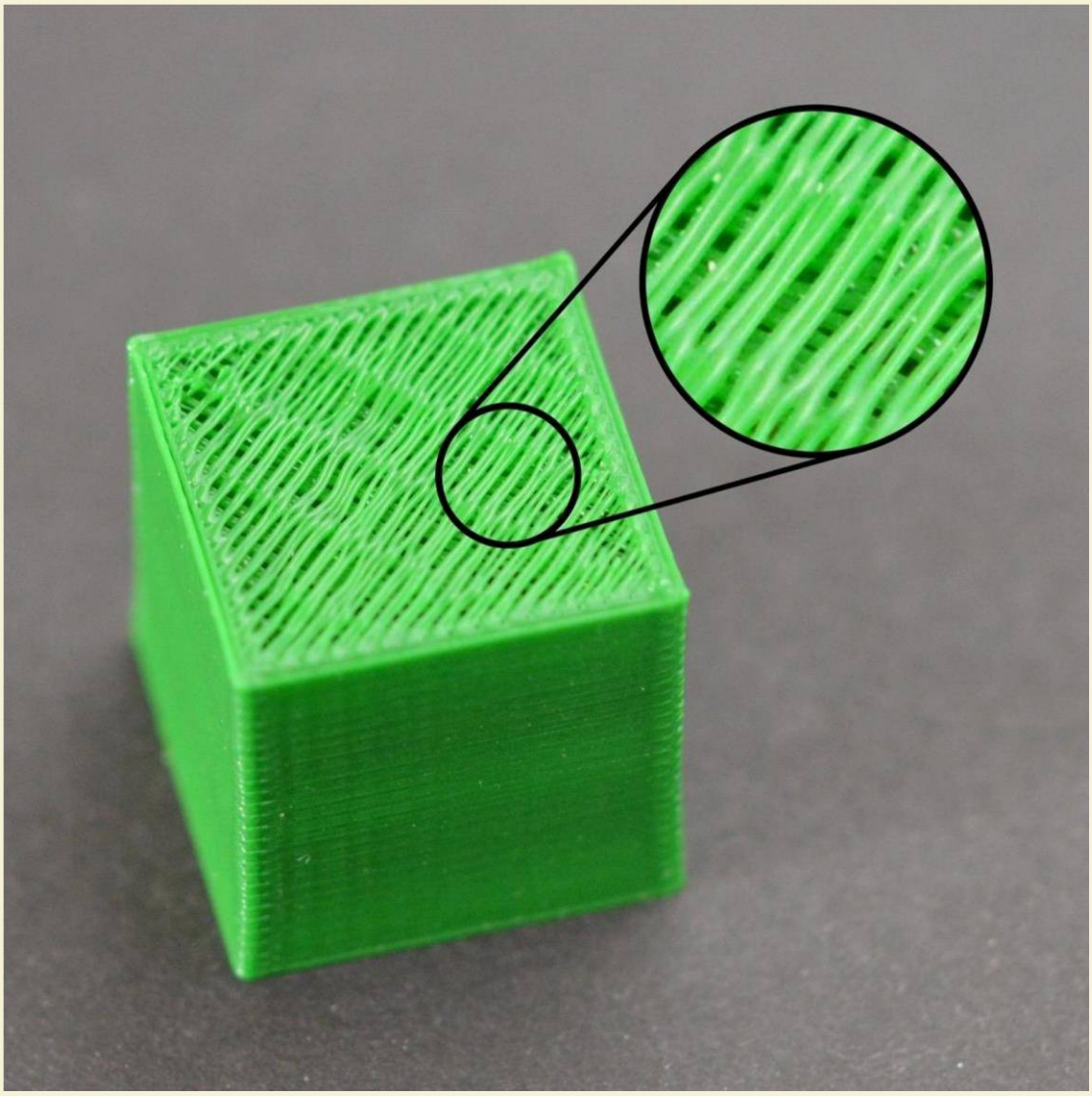
If your filament diameter is correct, but you are still seeing under-extrusion issues, then you need to adjust your extrusion speed. This is a very useful setting in Anet3D printer control panel that allows you to easily modify the amount of plastic that is extruded (otherwise known as the flow rate). You can find this setting by turning the knob to enter “Configuration” .Specific setting process as follows: Press the knob to enter the menu, choose “Configuration”, confirm，

turn to “motion”, confirm, find “Esteps/mm” tab, turn the knob to adjust extrusion speed. The default value is 95mm/s, you can change it to 97mm/s. that means the plastic will be extruded more 2mm/s than before. Try increasing your extrusion speed and then reprint the test cube to see if you still have gaps between your perimeters. Notice that increasing a little once to avoid extruding too much plastic.



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 extruder 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 in Anet3D control panel. 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 speed helps with under-extrusion, then you should decrease the extrusion speed 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, Anet3D 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. 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 0.6mm thick. So if you are using a 0.2mm layer height, you would need at least 3 top solid layers. If you are printing at a lower layer height such as 0.1mm, you may need 6 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 0.6mm solid layers, try printing with 1mm 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.

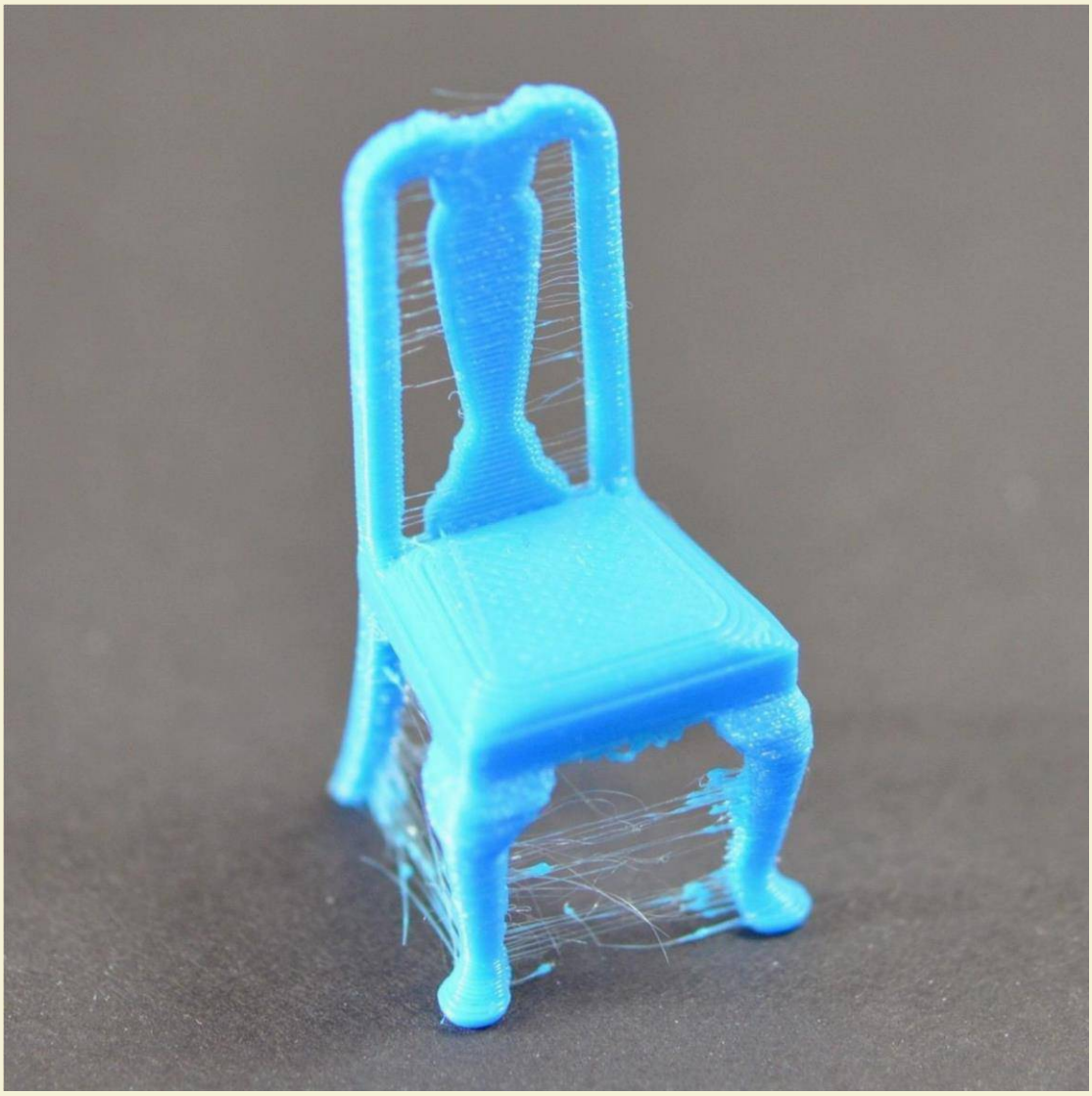
You can adjust the solid layer settings by clicking “Basic” and selecting the Bottom/Top Thickness tab.

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 top solid layers of your part will need to print on top of this foundation. If you infill percentage is very low, there will be large air gaps in your infill. For example, if you 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 seeing 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 10%, try using a 30% 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 in Cura 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. To ensure retraction is enabled, click “Basic” and click on the Quality tab. Ensure that the retraction option is enabled for your extruder. 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 Cura, you can click “Advanced” and click Retraction tab to set retraction distance. 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 3.5 mm. If you encounter stringing with your prints, try increasing the retraction distance by 0.5mm every time and test again to see if the performance improves.

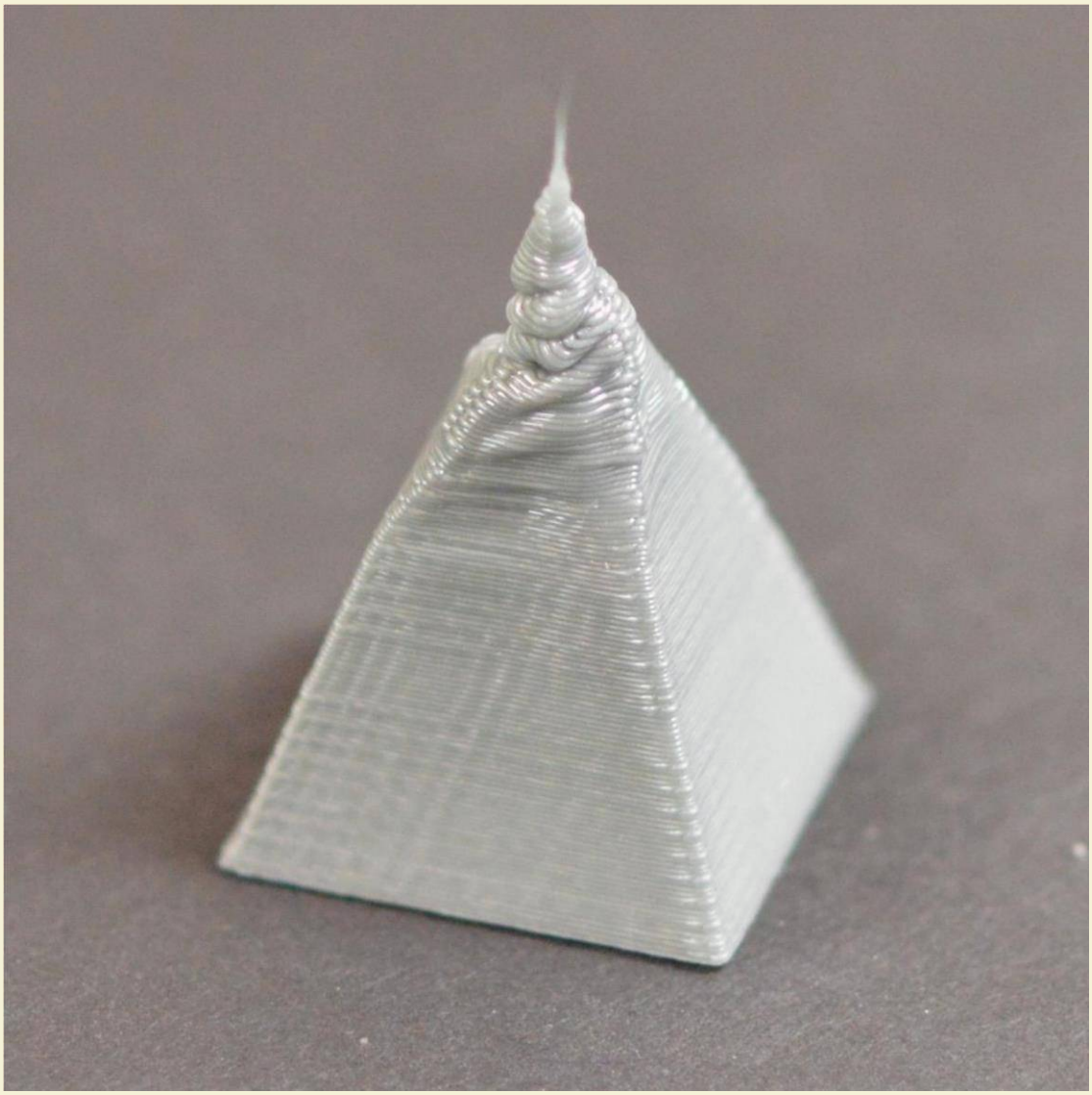
6.2 Retraction speed

Retraction speed 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

30-50 mm/s where retraction performs best. Cura has already provided many pre-configured profiles that can give you a starting point for what retraction speed works 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

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 extremely 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. You can adjust these settings through Anet3D control panel by turning the knob. First press the knob to enter the menu, choose “Control”, confirm and choose “Nozzle”, confirm. Then turn the knob to adjust the temperature.



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 above, 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. Anet3D printer includes a cooling fan, try increasing the power of the fan to cool the plastic faster. Please read “Cura Tutorial” to learn how to increase fan’s power

7.2 Print at too high temperature

If you are already using a cooling fan and you are still seeing this issue, you may need 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. Please read “Cura Tutorial” to learn how to set the temperature. 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.

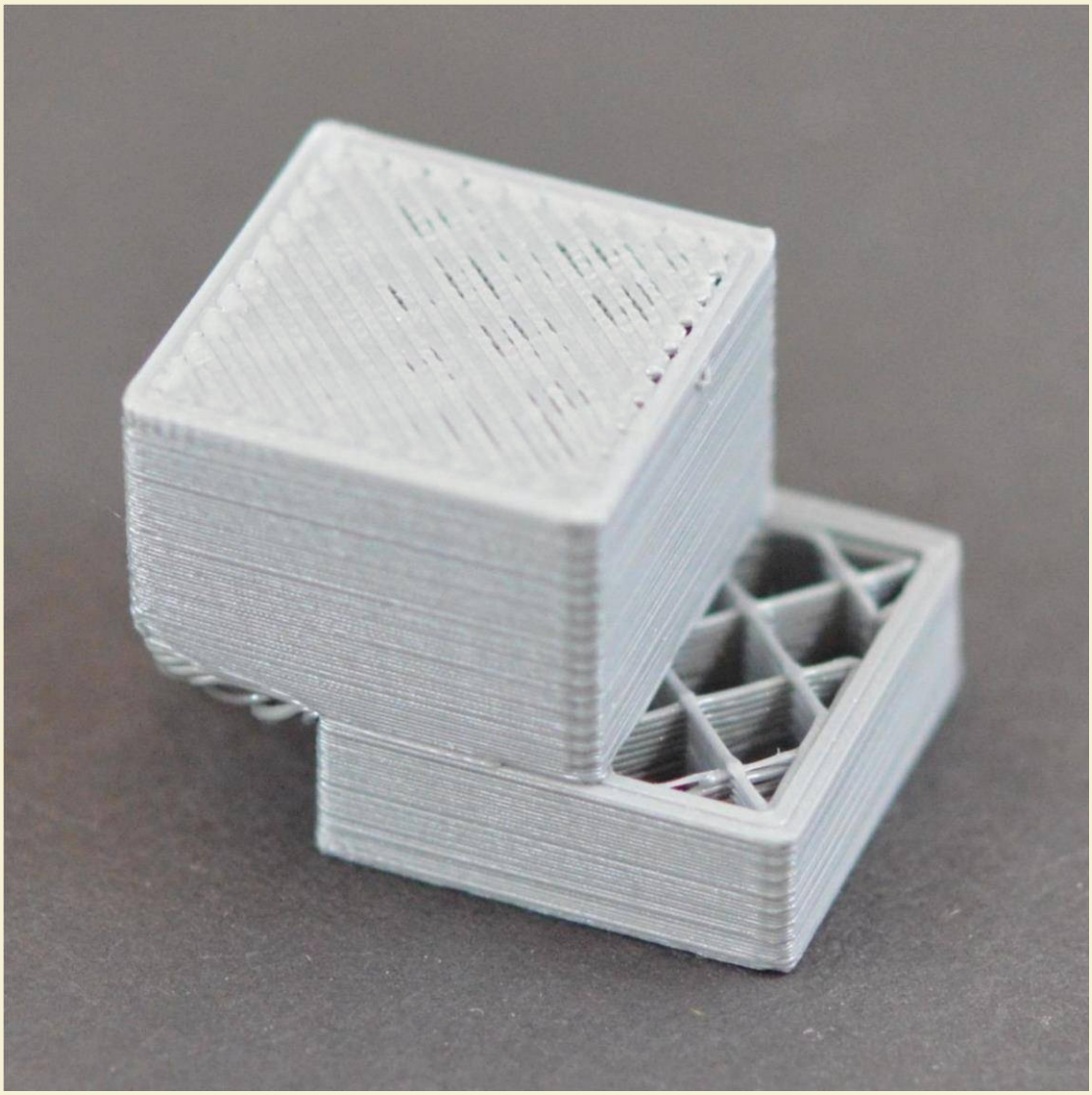
7.3 Printing too fast

If you print each layer very quickly, you might not need enough time for the previous layer to properly cool before you try to deposit the next layer 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. Cura includes a very simple option to do exactly that. Click “Advanced” and you will see “Minimal layer time” under “Cool” tab. This section 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 not less than 10 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 above fall, try to print 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. You can create a copy of the part you are trying to print in Cura by selecting model, (Select the model > Right clicking > Multiply object) 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 nozzle. The printer simply attempts to move the nozzle 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 nozzle 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 nozzle 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 Nozzle moves 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 to see if it helps. You can turn the knob on the control panel of Anet3D printer to reduce print speed.

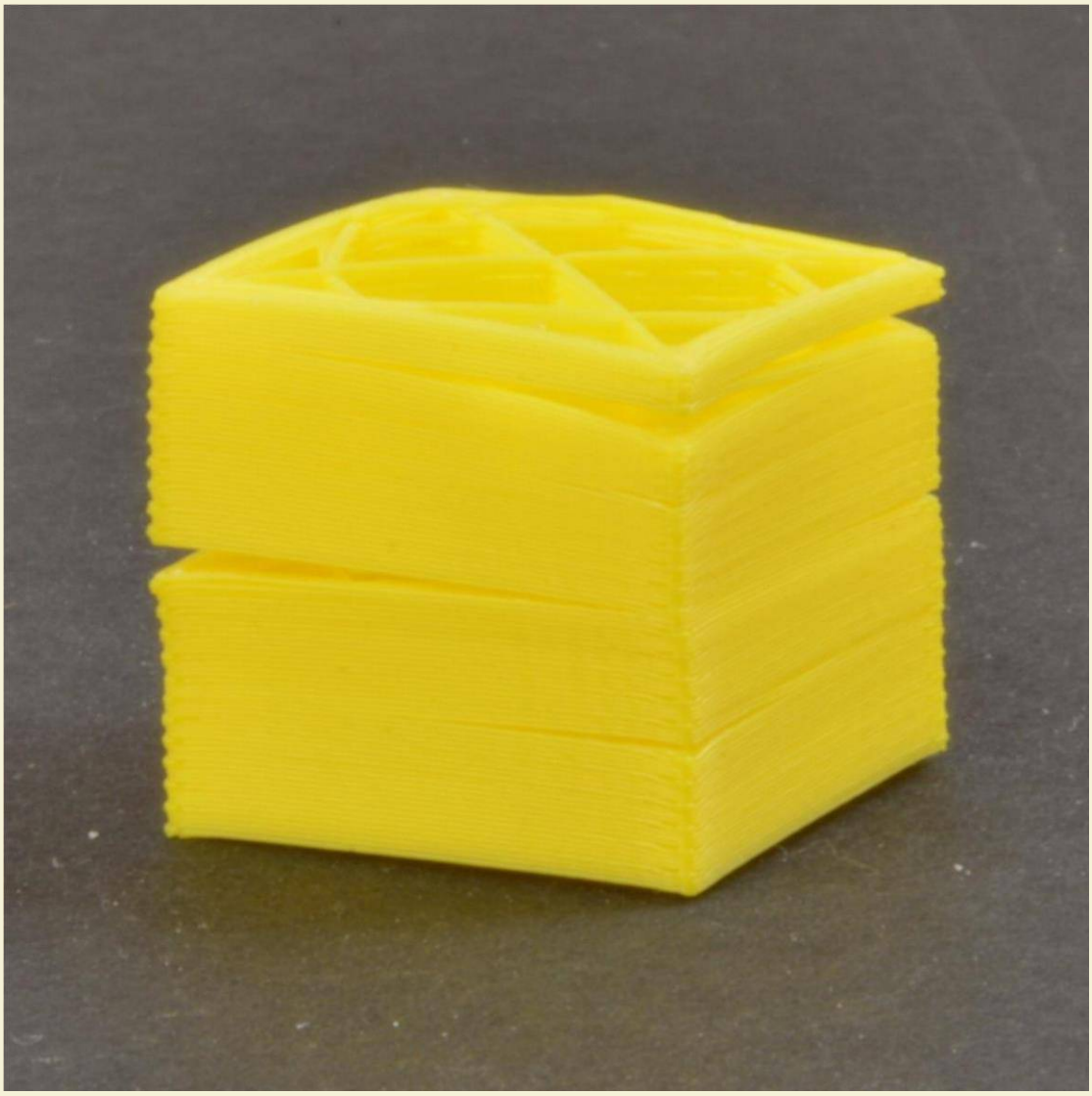
8.2 Mechanical or Electrical issues

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 nozzle. The belts are typically made of a rubber material and reinforced with some steel wire to provide additional strength. Over time, these belts may stretch, which can impact the belt tension that is used to position the nozzle. 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 nozzle 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.

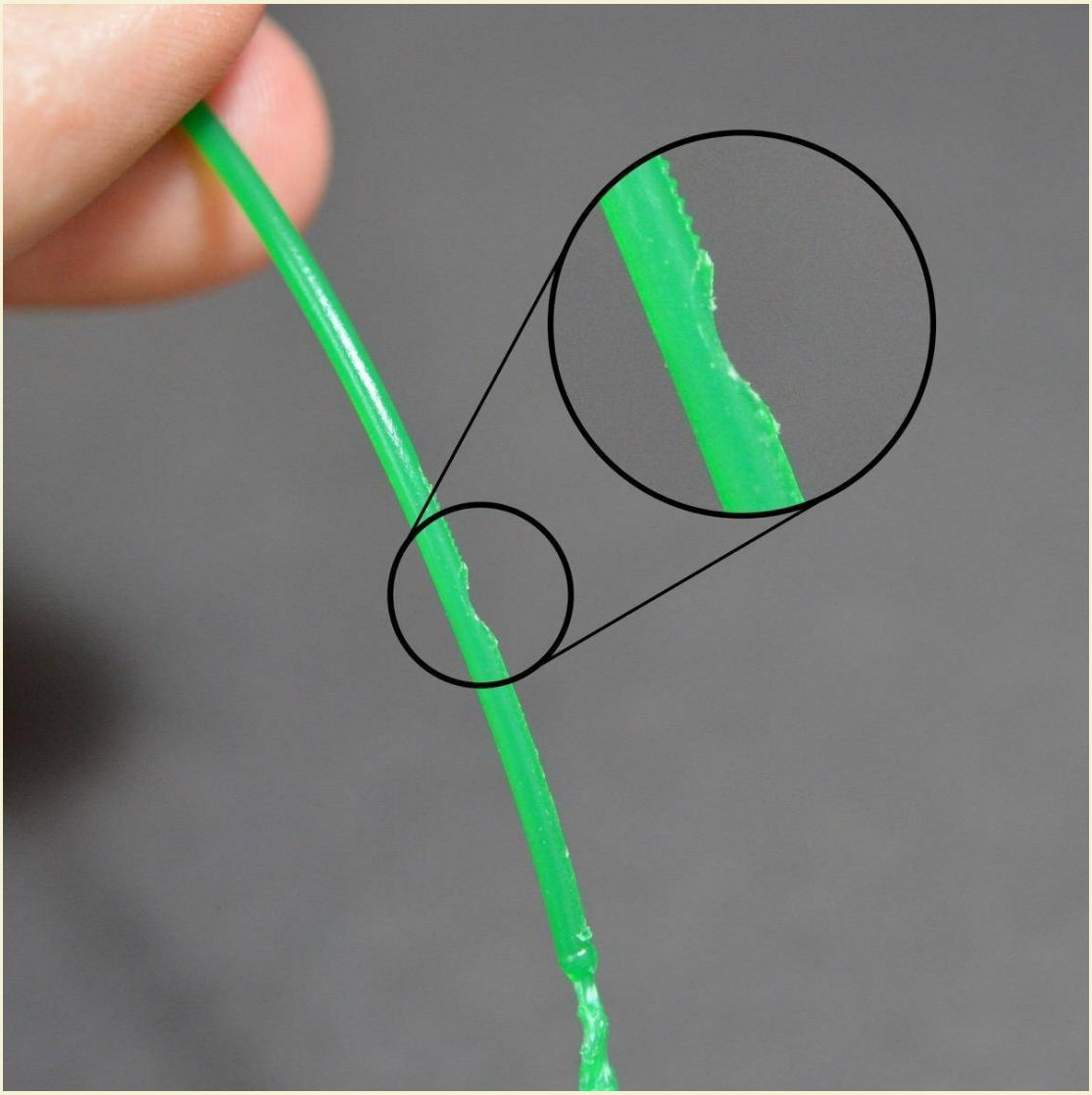
9.1 Layer height is too large

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 need 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 need 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. You can do this by clicking "Basic" and setting "Layer height" in Cura. Please read "Cura Tutorial" to learn more.

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, it is possible that your filament needs to be printed a higher temperature to create a strong bond. For example, if you tried to print ABS plastic at 190□, you would likely find that the layers of your part will

easily break apart. This is because ABS typically needs to be printed around 220-240□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 5- 10 degrees to see if the adhesion improves. Please read “Cure Tutorial” or “Anet3D printer Control Panel Instructions” to learn how to set print temperature.



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 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. Please read “Cure Tutorial” or “Anet3D printer Control Panel Instructions” to learn how to set extruder temperature. Plastic will always flow easier at a higher temperature, so this can be a very helpful setting to adjust.

10.2 Print 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. In Anet3D control panel, you can press the knob to enter the menu and choose “Configuration” and then choose “motion”, find“Esteps/mm” to adjust extrusion speed. The default value is 95mm/s. you modify it to 93mm/s. That means it will be 2mm plastic less extruded from the nozzle per second. Decrease only a little once to avoid Not Extruding Enough Plastic and then see if the filament grinding goes away.

10.3 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 to extruder

One of the first things you may need to try is manually pushing the filament into the extruder. Open Anet3D's Control Panel and heat your extruder to the appropriate temperature for your plastic. Next, use the Control knob to extruder a small amount of plastic, for example, 10mm. Please refer to "Tutorial of filament drawing out and push in". 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.

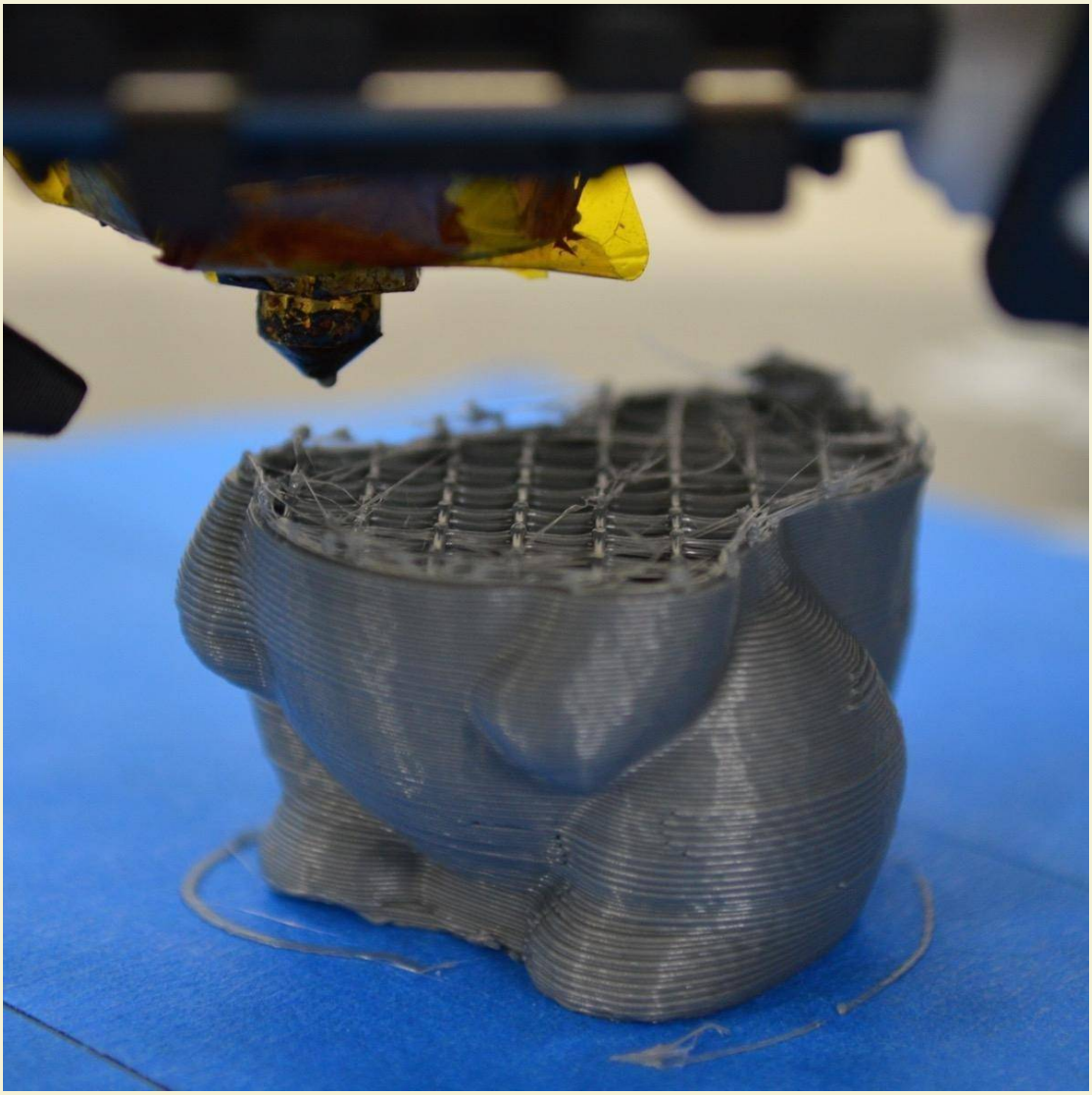
11.2 Reload the filament

If the filament still doesn't move, the next thing you should do is unload the filament. Verify that the extruder is heated to the appropriate temperature, then push the filament into extruder and draw it out quickly. 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. Please refer to "Tutorial of filament drawing out and push in".

11.3 Clean out the nozzle

If you can't 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 200□ and then manually pulling the filament out (hopefully along with any debris that was inside). You can use relative diameter drill (or relative diameter copper line of electric wire) to unclog extruders by feeding it into 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.

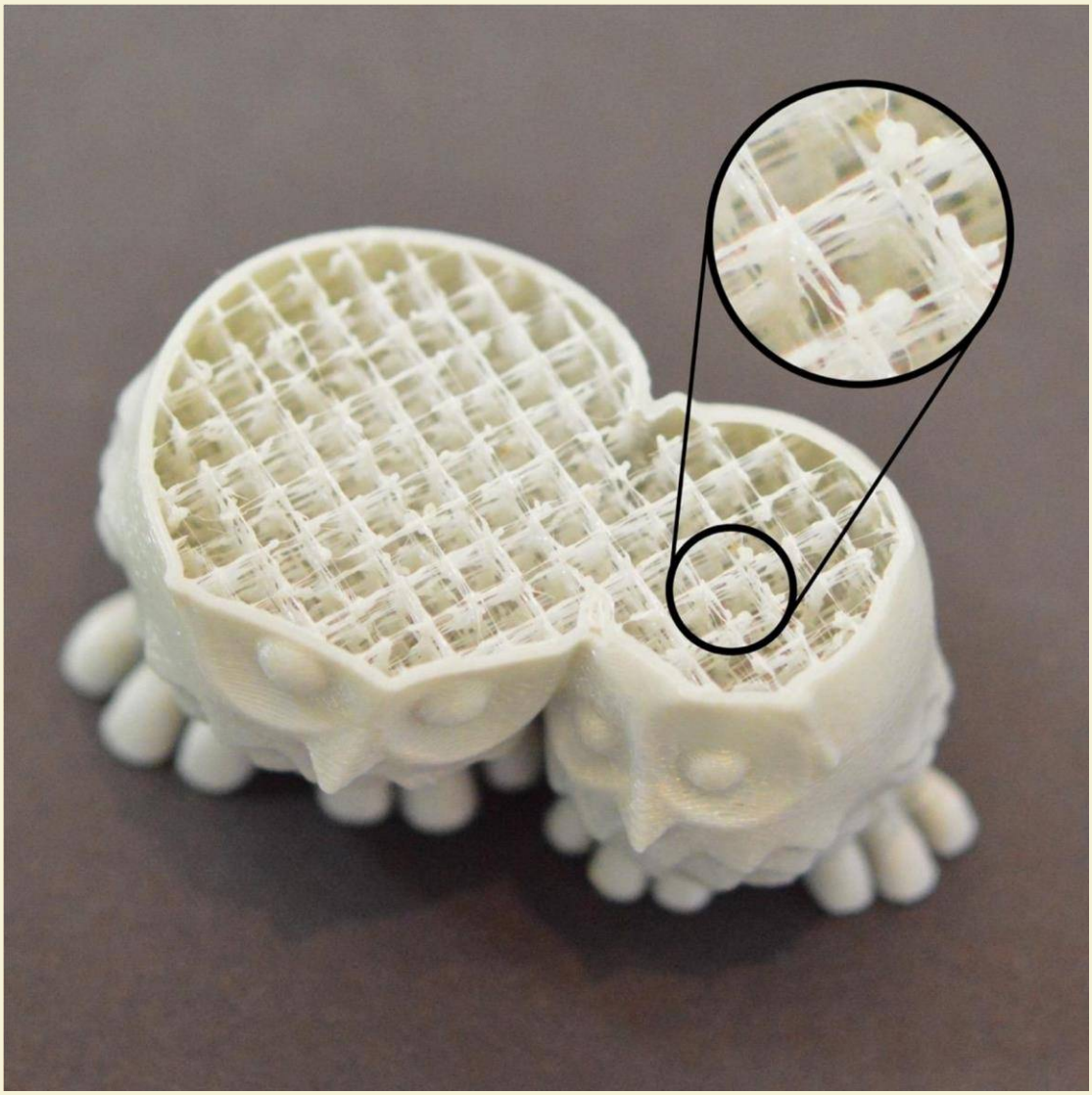
12.3 The extruder is clogged

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 nozzle, but the extruder motor will not be moving at all. Anet3D printer includes a cooling fan to cool down temperature for the extruder motor. You can also turn off the printer and allow the electronics to cool down.



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 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.

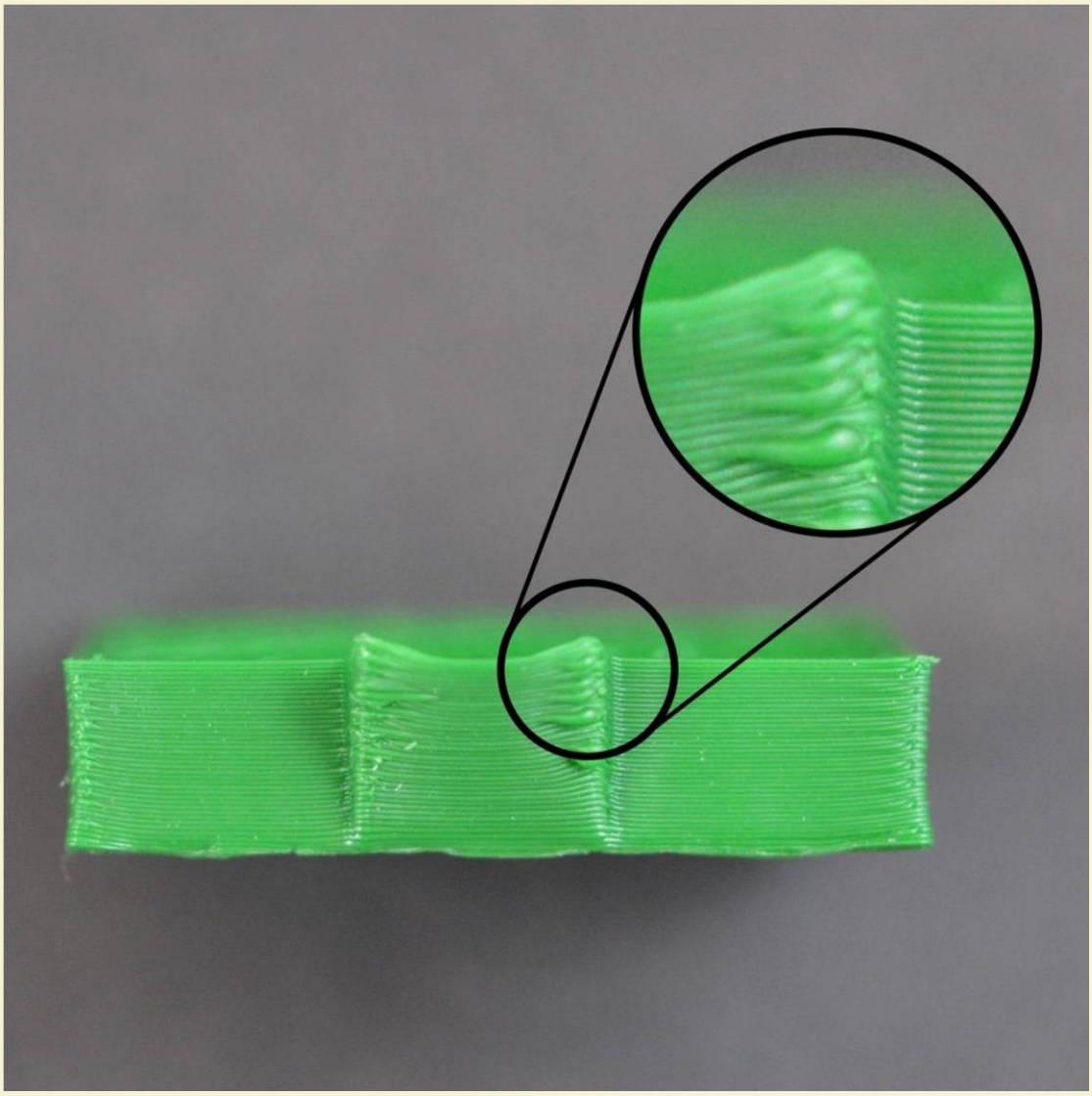
13.1 Increase infill density

In printing process, the fill density print has a great influence to its strength. If the infill density is very low, there will be a large number of empty gaps in the interior. Increase infill density appropriately can make print more firmly.

13.2 Lower 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. To do this, click "Advanced" and select the Speed.

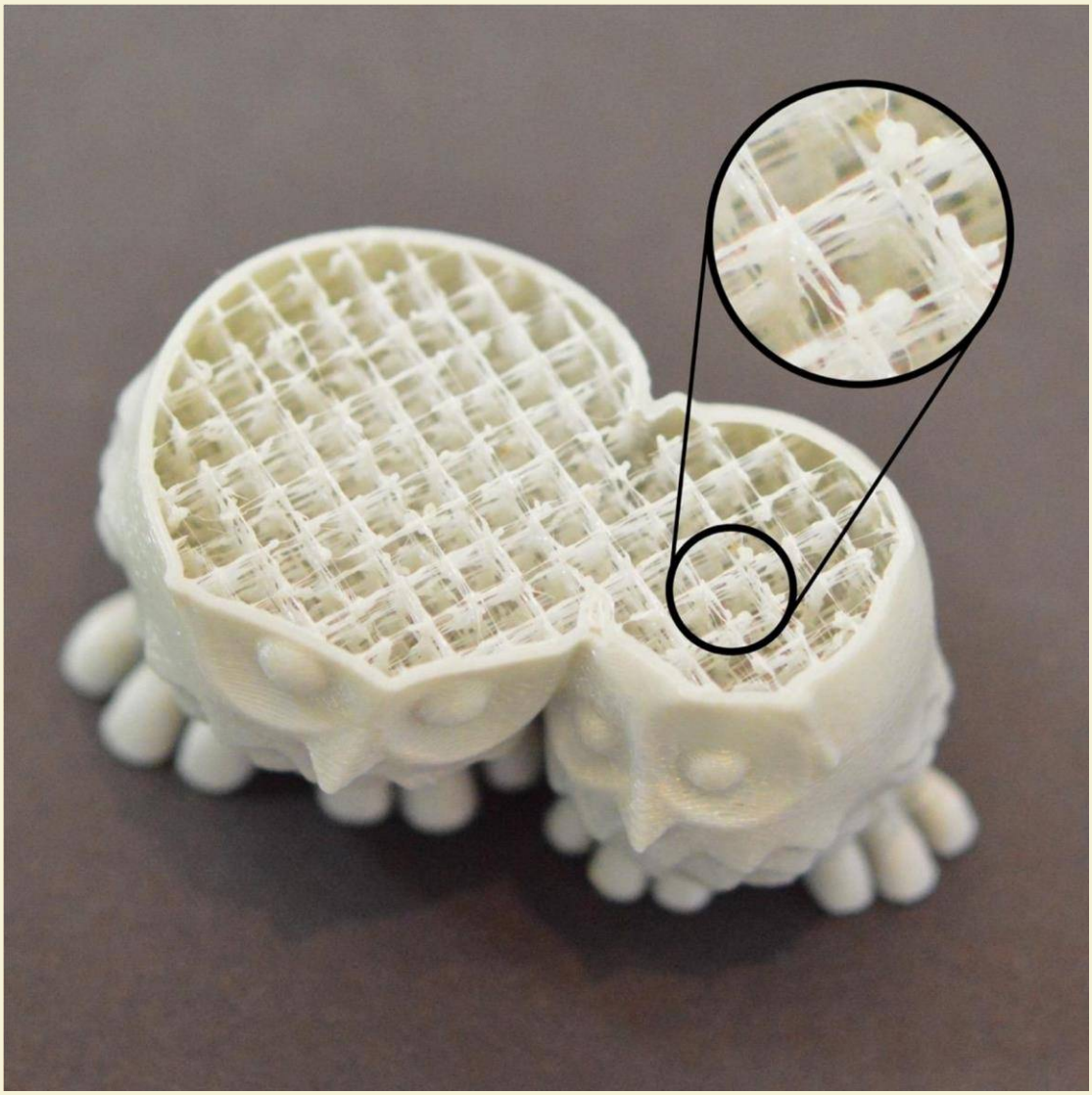
Adjust the "Infill speed", which directly controls the speed that is used for the infill. For example, if you were previously printing at 60mm/s, try decreasing that value by 50% to see if the infill starts to become stronger and more solid.



14. 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.



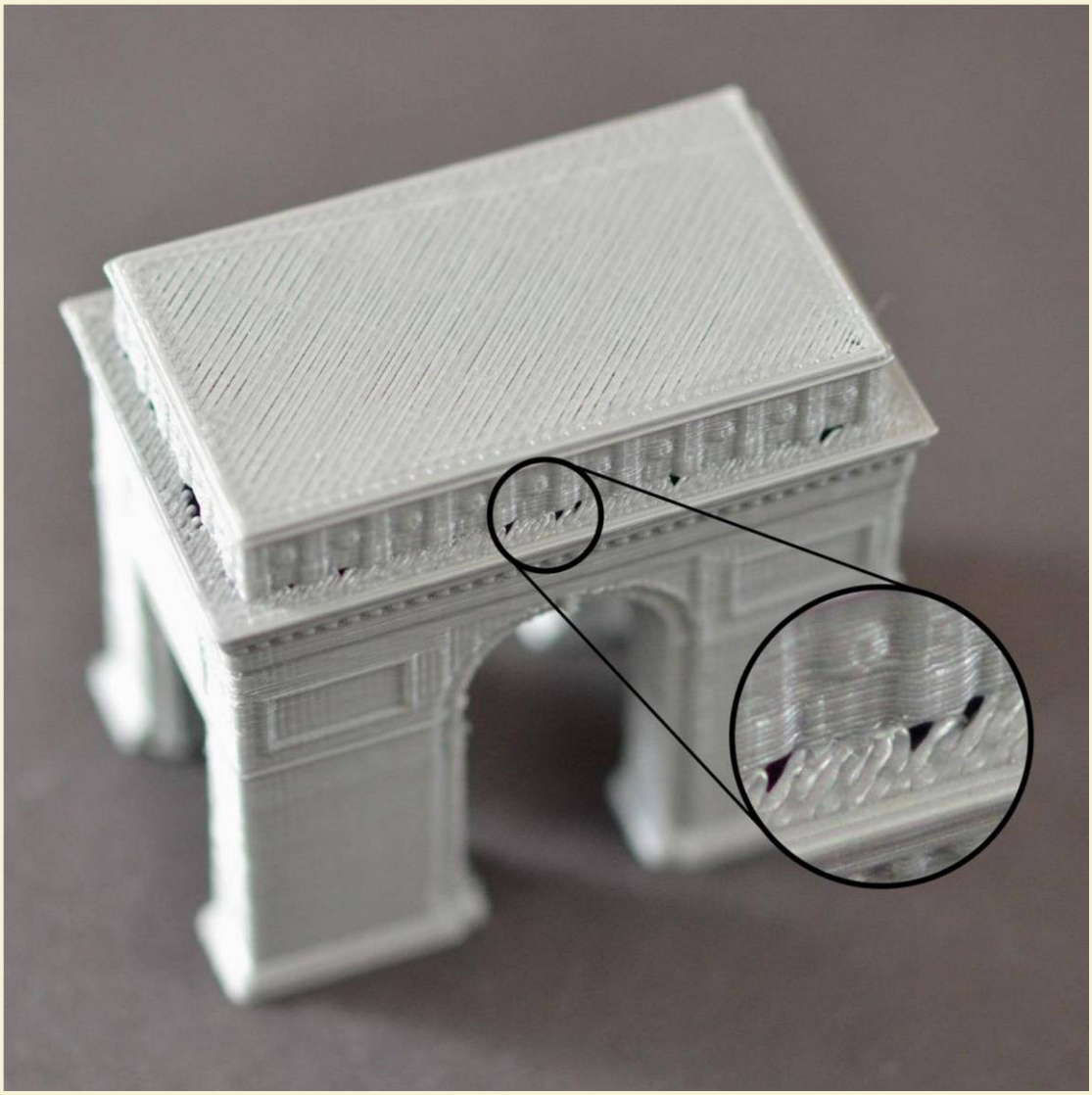
15. Scars on Top Surface

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.

15.1 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.



16. Holes and Gaps in Floor Corners

When building 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 in Cura to improve the print.

16.1 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. To adjust this setting, click “Basic” and click on the “Shell thickness” to set the value.. For example, if you were previously printing with 0.6mm perimeters, try the same print with 1.2mmperimeters to see if the gaps disappear.

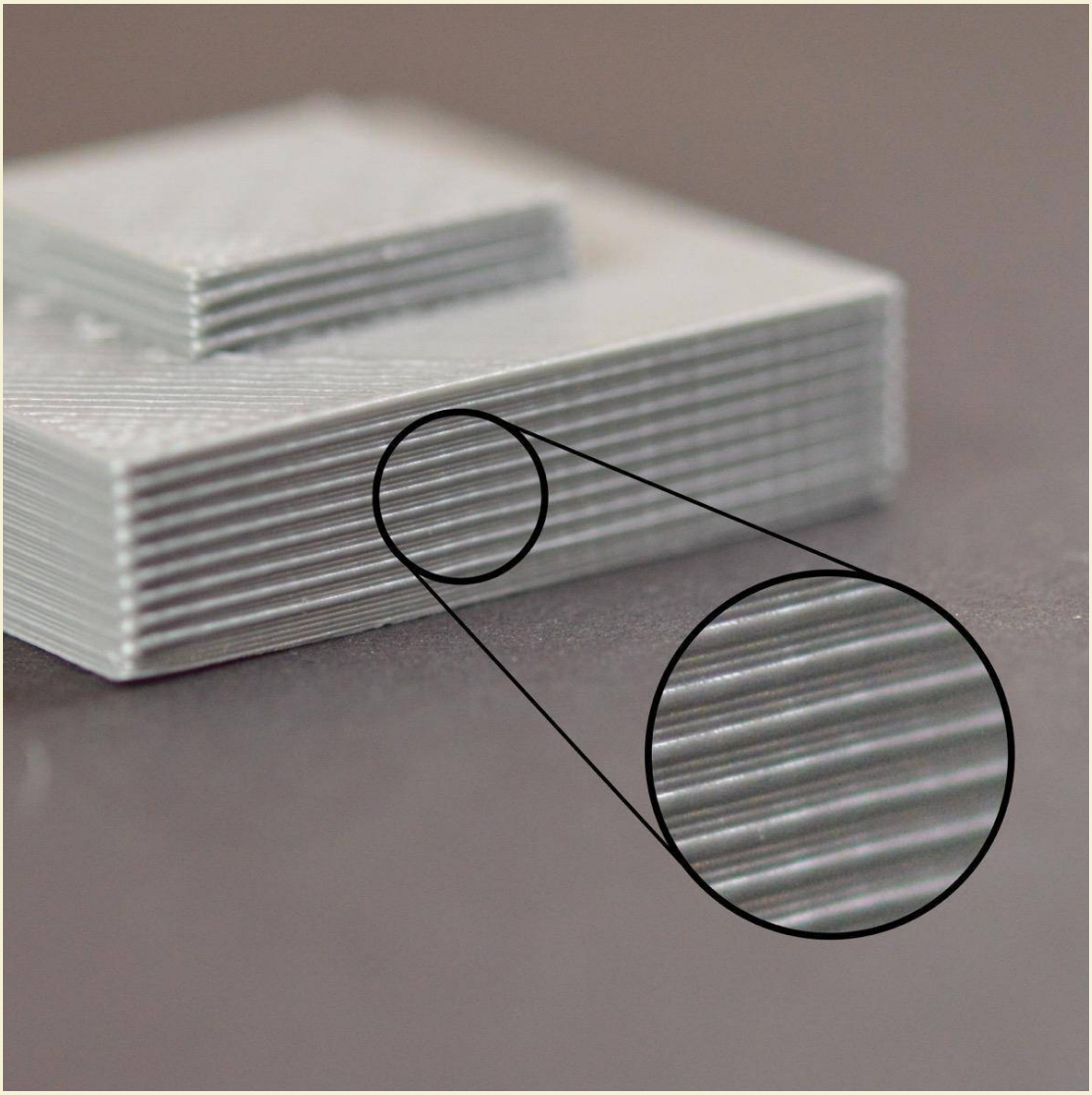
16.2 Not enough top solid layers

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. This setting can be adjusted by clicking “Basic” and selecting the “Bottom/Top thickness” to set appropriate value. If you were previously using only 0.6mm solid layers, try the same print with 1.2mm top solid layers to see if the foundation is improved.

16.3 Infill density is too low

The final setting you should check is the infill density. You will set the value under the“Fill Density” tab of “Basic”. 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.



17. 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.

17.1 Inconsistent extrusion

The most common cause for this issue is poor filament quality. If the filament does not have very tight tolerances, then you will notice this variation on the side walls of your print. For example, if your filament diameter varied by just 5% over the length of the spool, the width of the plastic extruded from the nozzle could change by as much as 0.05mm. This extra extrusion will create a layer that is wider than all the others, which will end up looking like a line on the side of the print. To create a perfectly smooth side wall, your printer needs to be able to produce a very consistent extrusion which requires high-quality plastic.

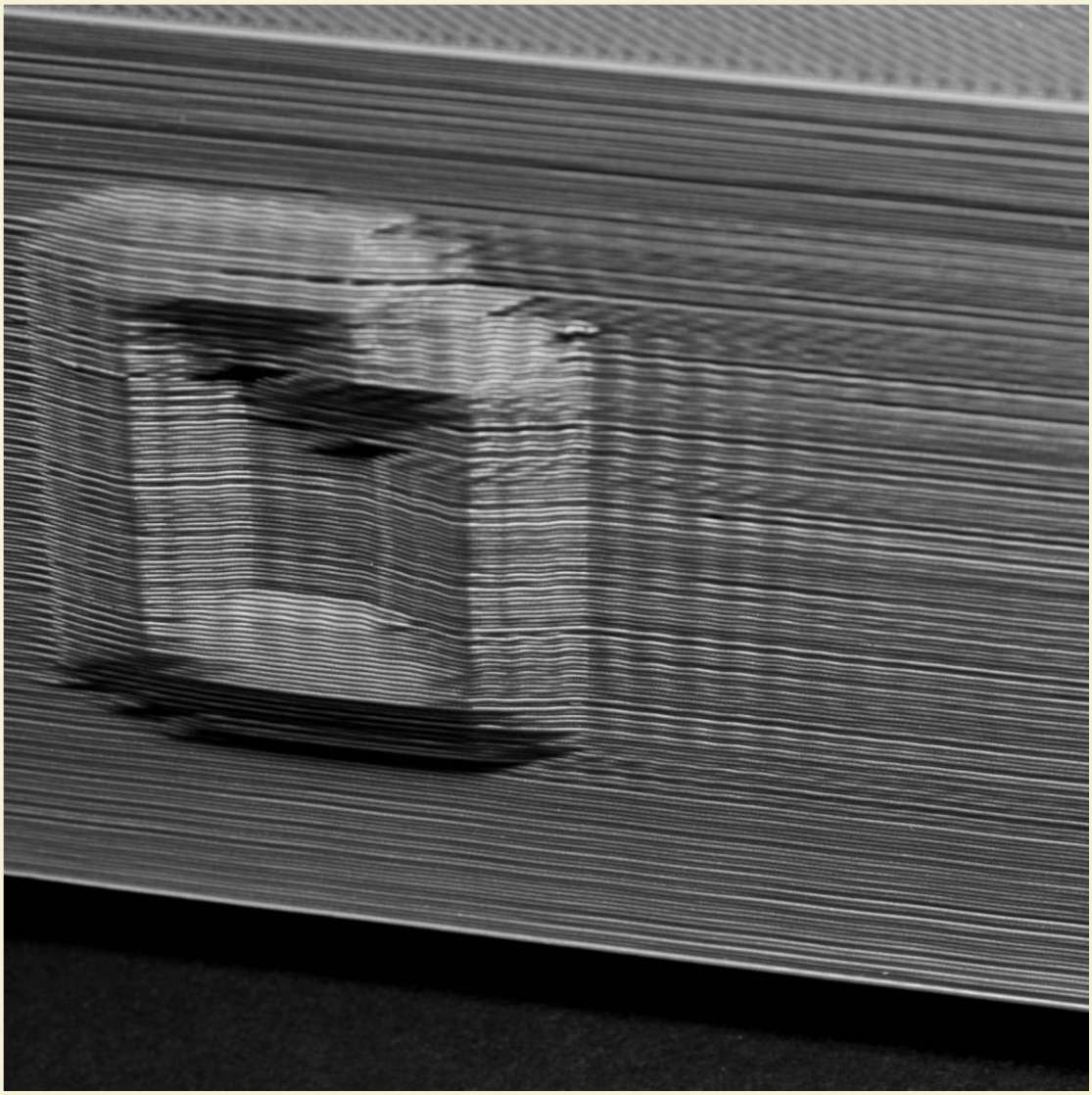
17.2 Temperature variation

Most 3D printers use a thermistor to regulate the temperature of the extruder. If this thermistor is not tuned properly, the temperature of the extruder may fluctuate over time. Due to the nature of how thermistor works, 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

+/-3degrees. During your print, you can use Anet3D printer's control panel to monitor the temperature of your extruder. If it is varying by more than 3 degrees, you may need to recalibrate your thermistor. Please consult your printer manufacturer for exact instructions on how to do this.

17.3 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.



18. 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 print 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.

18.1 Print 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. To do this, click “Basic” and select the“Print speed” tab to set suitable value.

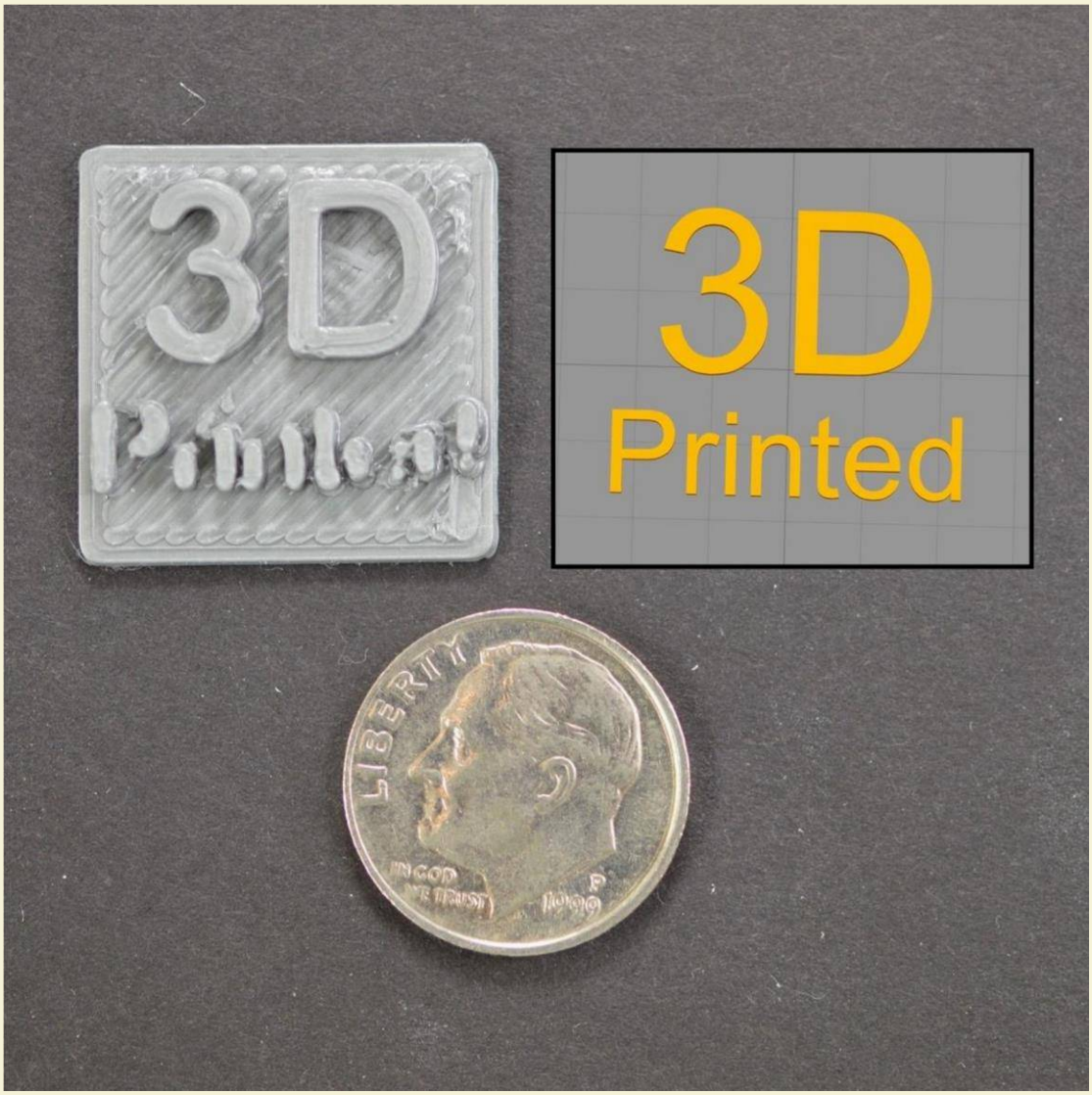
18.2 Firmware acceleration

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.

18.3 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. We’ve had

many users that eventually traced these issues back to mechanical problems with the printer, so it’s worth checking if none of the suggestions above were able to help.



19. Very Small Features Not Being Printed

Your printer includes a nozzle with a fixed size that allows you to accurately reproduce very small features. For example, many printers include a nozzle with

a 0.4mm diameter hole in the tip. 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 diameter. For example, if you were trying to print a 0.2mm thick wall with a 0.4mm nozzle. The reason for this is that you cannot accurately produce a 0.2mm extrusion from a 0.4mm nozzle.

The extrusion width should always be equal to or greater than the nozzle diameter. Because of this, when you click “Layers” in Cura, you may notice that the software removes these small features from the preview. This is the software’s way of telling you that you cannot print these very tiny features using the current nozzle on your 3D printer. If you are frequently printing very small parts, this may be a recurring issue that you encounter. There are several options that will allow you to successfully print these small parts. We will example each one in the section below.

19.1 Redesign the parts with thicker features

The first and most obvious 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 Cura to verify that your printer is capable of reproducing the 3D shape you created. If the features are visible in the Layers, then the printer will be capable of reproducing the revised features.

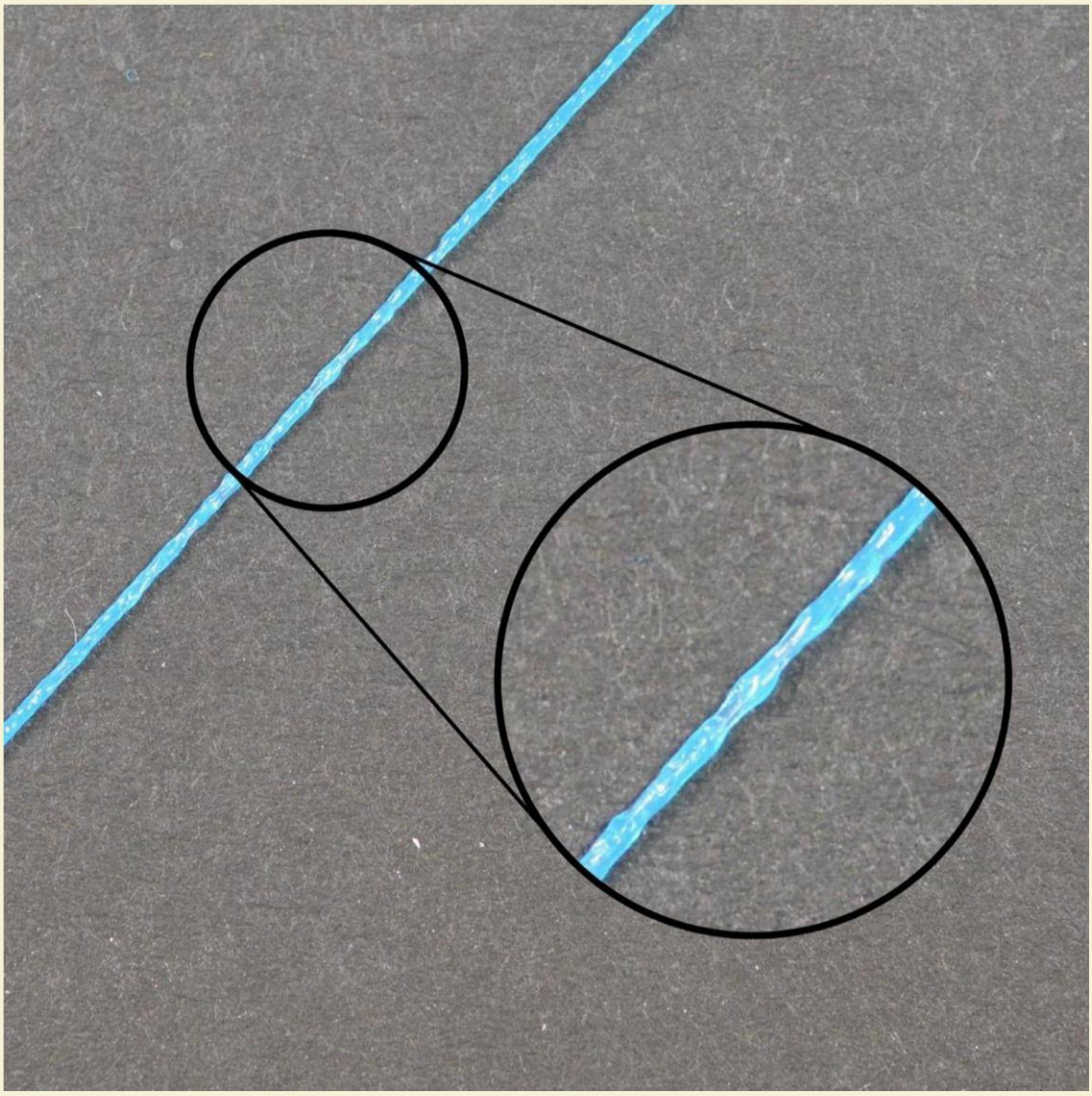
19.2 Install a nozzle with a smaller tip size

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.

19.3 Force the software to print small features

If you are not able to redesign the original 3D model, and you can't install a smaller nozzle on your 3D printer, then you have one last option. You can force the software to print these small features. However, it is likely that this may have some print quality consequences. If you click on "Advanced" and go to the Nozzle size tab, you can manually set the extrusion width that the software should use for your printer. For example, if you have a 0.4mm nozzle, you could select a manual extrusion width of 0.3mm to force the software to print features as small as 0.3mm in size. However, as we mentioned above, most nozzles are not capable of accurately producing an extrusion that is smaller than the tip size, so monitor your printer closely to ensure that the quality is acceptable for these fine features.



20. 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 summarized the most common

causes for inconsistent extrusion, and explained how each one can be addressed.

20.1 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 filament is extruded through the nozzle.

20.2 Clogged extruder

If the filament is not tangled and can easily be pushed 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 useAnet3D 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.

20.3 Layer height is very low

If the filament is spinning freely and the extruder is not clogged, it may be useful to check a few settings within Cura. 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. You can view this setting by clicking “Basic” and selecting the Layer height tab. If you are printing at a very small layer height, try increasing the value to see if the problem goes away.

20.4 Poor quality filament

One of the most common causes for inconstant extrusion that we have not mentioned yet is the quality of the filament that you are printing with. Low quality filament may contain extra additives that impact the consistency of the plastic. Others may have an inconsistent filament diameter, which will also cause inconsistent extrusion. Finally, 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 package 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, high-quality spool to see if the problem goes away.

20.5 Mechanical issues of extruder

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.

Consult Anet to see if your printer has a similar adjustment.