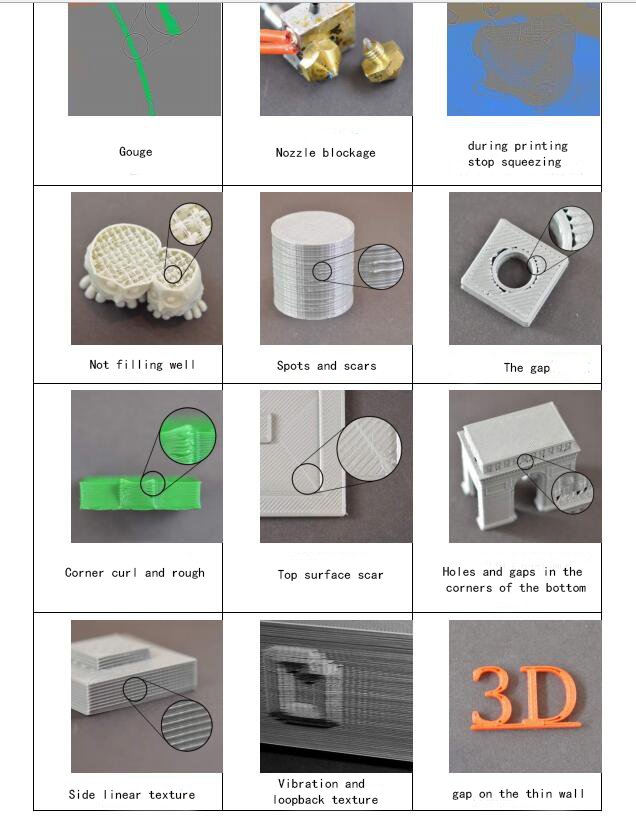
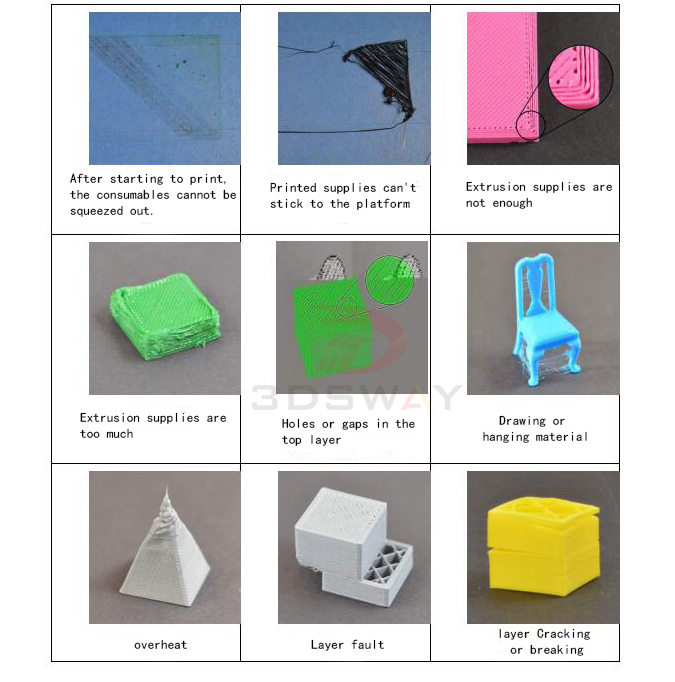
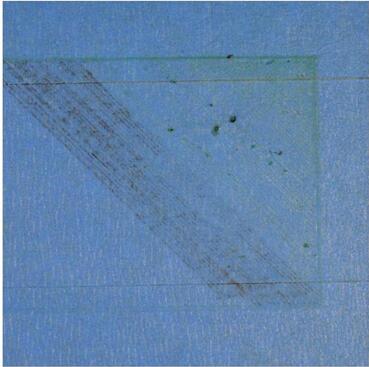
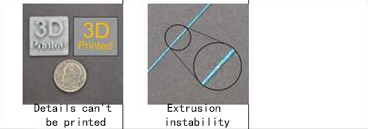
**3D printing item quality problem solving guide**

This guide is a good place to start if you want to improve the quality of your printer. We've compiled a list of the most commonly used 3D printing problems, which may be solved with software settings. In this tutorial, a lot of real images are used to make it easy for you to identify problems on various printouts.

Overview

Use the thumbnail below to determine what's wrong with your print. If you can't figure out what's wrong with you based on the thumbnail, you can go through each section and get more detailed information to help you improve your 3d printing quality.





1. **Start printing, no extrusion of consumables**

2. This is a relatively common problem for newcomers to 3D printing. However, fortunately, this problem is easier to solve. There are four possibilities for the extruder not to extrude consumables. We will explain each situation one by one and explain how to solve the problem by how to set it up.

**1.1 The extruder is not filled with consumables before printing begins.**

Most extruders have a problem: when the extrusion head is at a high temperature, it will leak. The consumables heated in the nozzle always tend to flow out, causing the nozzle to be empty. This problem with static feeds may occur at the beginning of the printing process. When you preheat the extrusion head, it may happen when the extruder is slowly cooled after the end of printing. If your extruder has some supplies due to the hopper, then the next time you squeeze, it may take a while for the plastic to start to squeeze out of the nozzle. When the extruder has a blank, you will find that the discharge will be delayed after the start of printing. To solve this problem, you need to ensure that the extruder is filled so that the nozzle is filled with plastic. The usual way to solve this problem is that Cura uses something called a "skirt." The skirt is the line around the print, which fills the extruder with plastic when it is printed. If you need to fill more, you can set the number of turns to increase the skirt. Some users may also use the control handle to manually extrude consumables before printing begins.

**1.2 The nozzle is too close to the platform**

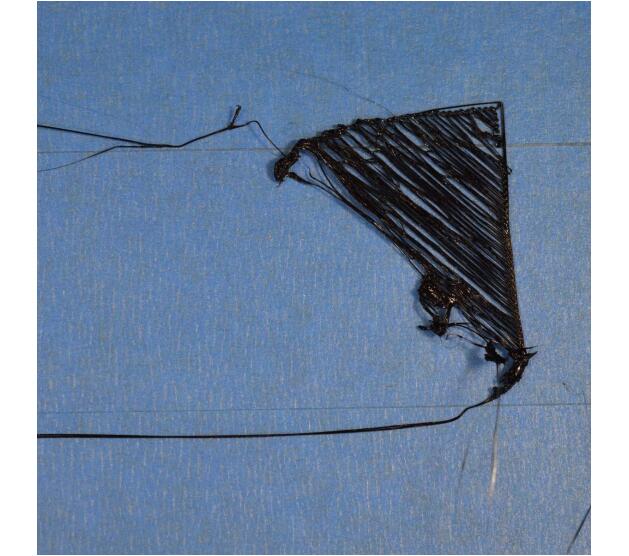
If the nozzle is too close to the platform, there will be insufficient space for the plastic to be squeezed out of the extruder. The hole at the top of the nozzle will always be blocked and the plastic will not come out. An easy way to identify this problem is to see if the 1st or 2nd layer does not extrude, but the 3rd or 4th floor is about to start normal extrusion. To solve the problem, you can adjust the distance between the nozzle and the platform until there is enough space between the nozzle platforms to allow the plastic to squeeze out.

**1.3 The wire is slipping on the extrusion gear (Gouge)**

Most 3d printers use a small gear to push the wire forward or backward. The teeth on the gears bite into the wire material to precisely control the position of the wire. However, if you look at the teeth marks on the plastic, you can see that there are some small sections of the wire that don't have teeth marks, which may be because the drive gears have removed too much plastic. When this happens, the drive gear can not grasp the wire, to drive the wire back and forth. Please refer to the section on "**Gouge**" to understand how to solve this problem.

**1.4 The extruder is blocked**

If none of the above suggestions can solve the problem, then it is possible that the extruder is blocked. The situation is as follows: When the outer debris gets stuck in the nozzle, the plastic silts up too much in the extruder, or the extruder is not sufficiently heat-free, and the consumables start to soften outside the expected melting zone. To solve the problem of the plug, we need to remove the extruder and clean the nozzle.



1. **The printed consumables can't stick to the platform.**

It is important that the first layer of print is tightly attached to the platform. Only in this way can the next layer be constructed on this basis. If the first layer fails to stick to the platform, it will cause problems in the back layer. There are many ways to deal with the first layer of non-stick problems. So, below, we will only check a few common situations and explain how to deal with them separately.

* 1. **Print platform is not horizontal**

Many printers have several screws or handles that adjust the position of the platform. If your printer has an adjustable platform and you are experiencing problems with the first layer, then the first thing you need to confirm is whether your platform is flat or not. If not horizontal, one side of the platform will be closer to the nozzle and the other side will be too far away. The first layer to be printed is perfect and requires a horizontal platform.

* 1. **The nozzle platform is too far**

When the platform has been adjusted, you still need to determine if the nozzle's starting position is appropriate to the platform's spacing. You need to position the nozzle at the right distance from the platform, keeping a distance of a A4 paper

* 1. **The first layer prints too fast.**

When the extruder prints the first layer on the platform, you want the first layer of plastic to stick properly to the surface of the platform so that the other layers can be printed next. If the first layer is printed too fast, the plastic may not have enough time to stick to the platform. For this reason, it is common to reduce the speed at which the first layer is printed

* 1. **There is a problem with the temperature or cooling settings**

When the temperature drops, the plastic contracts. For the sake of image, imagine a 100 mm wide ABS plastic punch, print, and extruder that is printed at 230 degrees, but the platform is cold, and the plastic squeezes out of the nozzle and cools quickly. Some printers also have cooling fans that speed up the cooling process when they are activated. If this ABS print is cooled to 30 degrees Celsius, this 100 mm wide print will shrink by 1.5 mm! However, the platform on your printer doesn't shrink that much, because it's always at the same temperature. Because of this phenomenon, when plastic cools, it tends to detach from the platform. This is an important factor to keep in mind as you print the first layer. If you observe that the first layer, which seems to stick quickly to the platform, but then drops off again as the temperature drops, then it's probably a matter of temperature and cooling related settings and settings. To print plastic that melts at high temperatures, like ABS, many printers are equipped with a heating platform to deal with the problem. During the printing process, if the platform is heated and stays at 90 degrees, it will keep the first layer hot, so it doesn't contract. It is generally thought that the PLA will make a good bed when it is heated to between 50 and 60 degrees in a hot bed, while ABS is better at 90 to 100 degrees. If you are in a more airy environment, you need to seal the printer so that the wind does not blow to the printer.

* 1. **Platform surface treatment (tape, glue, and material)**

Different plastics have different adhesion to different materials. Therefore, many printers have a special material platform dedicated to their consumables. For example, some printers will place a "BuildTak" piece that is well bonded to the PLA on the platform. Some printer manufacturers have chosen a heat-treated borosilicate glass platform that, when heated, bonds well to ABS. If you plan to print directly on these platforms, check the platform for dust, grease and the like before printing begins. Cleaning the platform with water or alcohol can have very different effects.

If your printer platform is not special material, there are still some other ways. There are several types of tape that can be bonded to a commonly used 3d open-up. Strips of tape can be easily glued to the surface of the platform, and can be easily removed or replaced to fit different materials. For example, the PLA can bond well with a blue latex tape. And ABS is glued to the Kapton tape, also known as the polyimide tape. Many users also apply glue to the surface of the platform. When all else fails, hairspray or GELS, or other sticky materials, can be quite useful. You can try which one works best for you.



1. **Extrusion supplies are not enough**

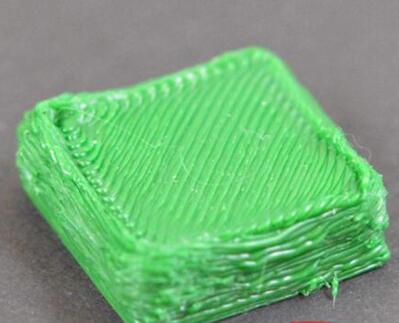
Test your printer to see if you can squeeze out enough of the method by printing a simple 20mm square cube with at least 3 layers of side lines printed. Check to see if the three edges at the top of the box are tightly bonded together. If there is a gap between the three connections, then you have a problem of undercharging. If the three sides are close together and there is no gap, then it's possible that you have a different problem. If you are sure that you are experiencing undercharging, here are a few possible ways to do this, as shown below

* 1. **Incorrect wire diameter**

The first thing you need to confirm is that the software knows the diameter of the consumables you use. Click to confirm the value of the setting, which is the same as the diameter of the wire you purchased. Even, you need to test your wire with a caliper to determine the value you set in the software, which is correct. The most common wire diameters are 1.75 and 2.85. Many wire rolls are packaged with the correct diameter.

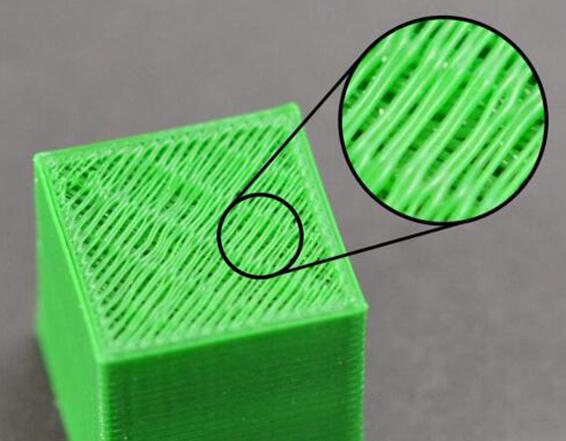
* 1. **Increase extrusion ratio**

If your consumables diameter is correct, but you still see the problem of insufficient discharge, then you need to adjust the extrusion ratio.



**4.Extrusion supplies are too much**

The software works with your printer. Make sure you squeeze the exact amount of plastic out of the nozzle. Accurate extrusion is an important factor for obtaining high quality printouts. However, most 3d printers do not have the means to monitor how much plastic is being squeezed out. If your extruder is incorrectly set, the printer is likely to squeeze out more plastic than the software expects. Overloading will cause problems with the external dimensions of the printout. To solve this problem, you can modify the same settings to solve the problem of overloading, just the opposite settings. For example, the problem of undercharging can be solved by increasing the extruded rate, and you can reduce the extruded rate to solve the problem of overcharging.



1. **Holes or gaps appear in the top layer**

To save on plastic, most 3D prints are made from a solid shell and a porous inner core. For example, the rate of filling the inner core of a printed piece is only 30% , meaning that only 30% of the core is plastic and the rest is air. Although the inner core of the Printout is partially hollow, we want the surface to be solid. To achieve this goal, Cura allows you to set how many solid layers are on the top and bottom of your print. For example, if you print a cube with five layers of solid layers up and down, the software will print up and down five completely solid layers, but the rest of the middle layer will be partially hollow. This technology can save a lot of plastic and time, but at the same time it can create a solid print. Of course, depending on what type of settings you use, you might notice that the top layer of the print is not completely solid. When you build these solid layers in the extruder, you may see holes or gaps. If you have a problem like this, here are a few simple settings that you can tweak to solve the problem.

**5.1 The number of solid layers at the top is insufficient**

Adjusting the number of solid filling layers at the top is the first to be used. When the fill layer is partially hollow, the solid layer will span the hollow part of the lower layer when the 100% solid fill layer is printed on a partially hollow filled layer. At this point, the plastic extruded from the solid layer tends to droop into the hollow. Therefore, it is usually necessary to print a few layers of solid layer at the top to get a flat, perfectly solid surface. The best way to do this is to print at least 0.5 mm at the top solid part. So if you use a layer of 0.25, you need to print the top 2 layers of the solid layer. If you print a lower layer, say 0.1 mm, you need to print five solid layers at the top to achieve the same effect. If you find a gap between the extruded threads at the top, the first thing you do is try to increase the number of solid layers on the top cores. For example, if you find this problem and only print three top solid layers, try printing five solid layers to see if they have improved. Note that adding a solid layer will only increase the size of the plastic inside the print, but not the external size.

**5.2 Fill rate is too low**

The padding inside the printout will form the basis of the upper layer. The solid layer at the top of the printout needs to be printed on this basis. If the filling rate is very low, there will be a lot of empty space in the fill. For example, if you use only 10% of the padding rate, the remaining 90% of the printout will be hollow. This will result in a solid layer that needs to be printed on a very large hollow space. If you try to increase the number of vertices, and you can still see gaps at the top, you might try to increase the filling rate to see if the gap will disappear. For example, your fill rate, which was previously set at 30% , tries to use a 50% fill rate, as this provides a better basis for printing the top solid layer.

**5.3** Extrusion supplies are not enough

If you've tried to increase the filling rate and the number of vertices at the top, but at the top of the print, you can still see the gaps. Then you may have a problem with crowding out. This means that the nozzle does not extrude the amount of plastic expected by the software. A complete solution to this problem can be found in the Extrusion supplies are not enough section.



**6.Drawing or hanging material**

A drawing occurs when tiny plastic threads are left on the printout. Usually, this is because the plastic droops out of the nozzle as the nozzle moves to a new position. In Cura, there are several settings that might help to solve this problem. The most commonly used method to solve the drawing problem is to "withdrawal" . If the withdrawal is turned on, then when the extruder completes the Printout of a region of the model, the wire in the nozzle is pulled back so that when it is reprinted, the plastic is pushed back into the nozzle and forced out from the top of the nozzle. To make sure that the withdrawal has been activated, make sure that each of your extruders have turned on the withdrawal option. In the following sections, we'll explore this important withdrawal setting, as well as several other settings for dealing with drawing problems, such as the temperature setting for the extruder.

**6.1 withdrawal distance**

The most important setting is the withdrawal distance. It determines how much plastic is pulled back from the nozzle. Generally speaking, the more plastic you pull back from the nozzle, the less prone it is to drool when the nozzle moves. Most direct-driven extruders require only a 0.5-2.0 mm pullback distance, and then some Bowden extruders may need up to 15 mm to pull back, as the extruder drives the gear and the heat nozzle farther apart. If you have a drawing problem with your printer, try increasing the back draw distance by 1 mm at a time, and watch for improvement。

**6.2 withdrawal speed**

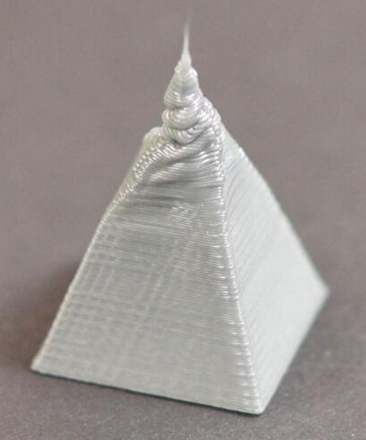
The next the withdrawal related setting is the the withdrawal speed. It determines how quickly and slowly the wire is pulled away from the nozzle. If the the withdrawal is too slow, the plastic will droop from the nozzle and start leaking before it can move to a new position. If the withdrawal too quickly, the wire may be disconnected from the plastic in the nozzle, or even drive the gear's rapid rotation, possibly shaving the surface part of the wire. There is a range, the effect is better, between 1200-6000 mm / Min (20-100 mm / s). Cura has provided some built in default configurations that give you a good starting point to determine how fast the withdrawal is going to work best. However, the ideal value is based on the material you actually use. So, you need to do experiments to determine whether different speeds reduce the amount of tension

**6.3 The temperature is too high**

If you have checked the withdrawal settings, the next most common factor that causes the tension problem is the temperature of the extruder. If the temperature is too high, the plastic in the nozzle becomes very thick, making it easier to get out of the nozzle. Then, if the temperature is too low, the plastic will remain hard and hard to squeeze out of the nozzle. If you feel that your withdrawal settings are correct, but the problem arises, try lowering the extruder temperature by 5 to 10 degrees. This will have a significant impact on the final quality of the print.

**6.4 Suspended moving distance is too long**

As we discussed above, the drawing takes place in the extruder and moves between two different positions. During the movement, the plastic droops down from the nozzle. The size of the moving distance has a great effect on the creation of the drawing. The short-range movement is fast enough that the plastic doesn't have time to fall back down from the nozzle. Then, a large distance of movement, more likely to lead to the drawing. CURA contains a very useful feature that makes the moving path as small as possible. The software is smart enough to automatically adjust the movement path to ensure that the nozzle is moving at a very low distance. In fact, most of the time, software can find the right path to avoid moving too far in a single breath. This means that there is no possibility of drawing the silk, because the nozzle is always on top of the solid plastic, and does not move outside the print.



1. **Over heat**

At least 190 to 230 degrees Celsius of plastic ejected from the extruder. When plastic is still hot, it is still soft, and can be easily shaped into different shapes. And then, when it cools, it quickly becomes solid, and it's shaped. You need to get the right balance between temperature and cooling, so that the plastic flows smoothly out of the nozzle, but can be quickly solidified to obtain the accuracy of the size of the print. If you don't get the balance, you'll run into some print quality problems, and the print is not as accurate as you'd like it to be. As you can see, the lines that extrude from the top of the pyramid don't cool down as quickly as possible. The following sections will examine several common situations that lead to overheating and how to avoid them.

* 1. **Insufficient heat dissipation**

The most common cause of overheating is that the plastic doesn't cool down in time. When the cooling is slow, plastic can easily be changed shape. For polyplastics, it is better to quickly cool the layers that have been printed to prevent them from deforming. If you have a cooling fan on your printer, try to increase the FAN's wind power to cool the plastic faster. This extra cooling helps to shape the plastic. If your printer doesn't have a complete cooling fan, you may need to try to install your own fan, or use a hand-held fan to speed up the cooling。

* 1. **Printing temperature is too high**

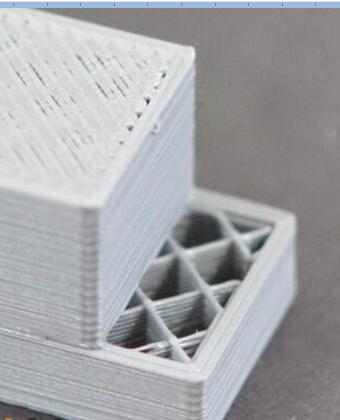
If you've already used a cooling fan, but there's still a problem, you might want to try to lower the temperature. If plastic were squeezed out of the nozzle at lower temperatures, it would probably solidify more quickly. Try to reduce the print temperature by 5 to 10 degrees to see the effect. Be careful not to cool too much so that the plastic is not hot enough to squeeze through the tiny holes in the nozzle.

* 1. **Printing too fast**

If you print each layer very quickly, you may not have enough time to let the layer cool properly, but start printing new layers on it again. This is particularly important when printing small models because there is only a small amount of time to print each layer. Even with a cooling fan, you still need to reduce the speed of printing for these very small layers to ensure that there is enough time for the layer to solidify.

* 1. **When all of the above methods are invalid, try printing multiple prints at once.**

If you've tried these three methods, but still have problems with cooling, there is another way you can try it. Copy the model you want to print (Edit Copy / Paste) , or import it to another model that can be printed at the same time. By printing two models at once, you can provide more cooling time for each model. The nozzle will need to be moved to a different position to print the second model, which provides an opportunity for the first mold to cool. It's simple, but it's a very effective strategy to solve the overheating problem



1. **Layer fault**

Most 3 D printers use an open loop control system. To put it bluntly, they don't have any feedback on the actual location of the nozzle. The printer simply tries to move the nozzle to a certain position and then expects it to get there. Most of the time, this is possible, because the stepper motor that drives the printer is very powerful and doesn't have a huge load to stop the nozzle from moving. Then, if something goes wrong, the printer won't be able to find it. For example, when printing, you suddenly bump into your printer, and you may cause the nozzle to move to a new location. The machine doesn't have feedback to identify the situation. So, it will continue to print, as if nothing had happened. If you find that the layer in the printer is in the wrong place, it may be due to one of the reasons listed below. Unfortunately, once these errors occur, the printer has no way to detect the problem and handle the problem. So we're going to talk about how to solve this problem.

* 1. **The nozzle moves too fast**

If you print at a high speed, the motor of the 3D printer will do its best to support it. If you try to print faster than the motor can handle, you usually hear a click, and the motor can't move to its intended position. In this case, the subsequent printed layer will be misplaced with all the layers that were previously printed. If you think your printer is printing too fast, try to reduce the print speed by 50% to see if it helps. If any one of these speeds is too fast, it could lead to dislocation. If you're willing to tweak more advanced settings, you can also consider lowering the acceleration settings in your printer firmware to make the acceleration, speed, and deceleration smoother.

* 1. **Mechanical or electronic problems**

If the speed is reduced and the problem of dislocation persists, it is possible that your printer has mechanical or electronic problems. For example, most 3 D printers use synchronous tape to drive the motor to control the position of the nozzle. The belt is usually made of rubber and reinforced with some kind of fiber. For a long time, the synchronous belt may be relaxed, which will affect the tension of the ribbon-bit nozzle of the synchronous belt. If the tension is not enough, the belt may slip on the synchronous wheel, which means that the synchronous wheel is rotating, but the belt is not moving. If the ribbon is too tight, it can also cause problems. The overstretched synchrony creates too much friction between the bearings, preventing the motor from turning. Ideally, the belt would be tight enough to prevent slipping, but not too tight, which would prevent the system from running. If you are dealing with the dislocation problem, you need to make sure that all the tension in the synchrony is appropriate, not too loose or too tight. If you think there may be a problem, please communicate with the printer provider to know how to adjust the tension of the belt. Most 3-d printers include a series of synchro-bands, synchronous wheels that drive the synchro band, and a stop screw (also called a Vertex) to hold onto the motor. The top wire locks the synchronous wheel on the motor's shaft so that the two can rotate synchronously. Therefore, if the top is loose, the synchronous wheel does not rotate with the motor shaft. This means that the motor may be spinning while the synchronous wheel and the synchronous belt are not moving. In this case, the nozzles will not reach the desired position, causing all the subsequent layers to stagger. So, if the problem of the lamination is repeated, you need to make sure that the fasteners on all the motors are tightened up. There are also some other common electronic problems that cause the motor to fail. For example, if the motor doesn't have enough current, the motor will not have enough torque to turn. It could also be that the motor's drive plate overheated, which would cause the motor to stop intermittently until the circuit cooled down. While this is not an exhaustive list, it provides only a few of the common electronic or mechanical recommendations that you may need to check when the dislocation problem keeps recurring.



1. **Layer cracking or breaking**

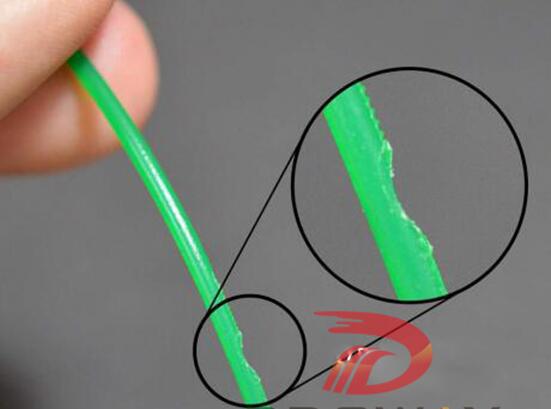
3d printing builds a model by printing one layer at a time. Each successive layer is printed on the previous layer and finally builds the desired 3-d shape. Then, in order for the final print to be solid, you need to make sure that each layer is fully bonded to the layer underneath it. If the layer does not adhere well to each other, the final printout may be cracked or broken. Next, we'll look at some of the typical reasons, and the solutions.

* 1. **Layer height is too high**

Most 3d printers have nozzle diameters between 0.3 and 0.5 mm in diameter. The plastic is squeezed out of this very small hole, forming a very thin filigree, and then building a detailed print. However, these small nozzles also cause the limit of the height of the layer. When you print another layer of plastic on the first floor, you need to make sure that the new layer is pressed down to the lower layer so that the two layers can stick together. In general, you need to make sure that the height you choose is 20 percent smaller than the diameter of the nozzle. For example, if your nozzle diameter is 0.4 mm, you can not use a layer higher than 0.32 mm, otherwise the plastic on each layer will not adhere properly to the layer below it. So, if you find that the print is cracked, and the layers don't stick together, the first thing you need to look at is that the height of the layer matches the diameter of the nozzle. Try to reduce the height of the layer, and see if you can get the layer to stick better.

* 1. **Printing temperature is too low**

Hot plastics always better to stick together than cold plastic.If you find that layers don't stick well together, and you can be sure that the height settings aren't too high, then it's probably your wire, which needs to be printed at a higher temperature to better bond. For example, if you try to print ABS plastic at 190 degrees Celsius, you may find that layers are easily separated. This is because ABS typically needs to be printed between 220 and 235 degrees Celsius to make layers and layers cohesive. So if you think this might be the problem, make sure you're using the right printing temperature for the wire you bought. Try Increasing the temperature by 10 degrees centigrade each time to see if the bonding has improved.



**10.Gouge**

Most 3D printers use one small gear and another to hold the wire so that the gear catches the wire. The drive gears have sharp teeth that can bite into the wire, and then rely on the rotation direction of the gears to push the wire back and forth. If the wire is not moving, but the gears continue to rotate, the gears may remove some plastic from the wire, leaving no room for the gear to grip the wire. Many people call this a "Gouge" . Too much plastic has been removed, causing the extrusion function to be abnormal. If this happens on your printer, you'll usually see a lot of plastic debris scattered all over the place. As you can see, the extruder is rotating, but the wire is not pushed inside the extruder. Here's the easiest way to solve this problem.

**10.1 Increase the temperature of the extruder**

If you've been having a problem with Gouge, try to raise the temperature of the extruder by 5 to 10 degrees, so that the plastic is easier to squeeze out. Plastic is always easier to squeeze out when the temperature is high, so this is a very useful setting that can be adjusted.

**10.2 Printing speed is too fast**

After raising the temperature, if you still have a problem with the Gouge, the next thing you need to do is to reduce the speed of printing. In doing so, the extruder's motor doesn't have to rotate at that speed, because the wire takes longer to squeeze out. Reducing the motor speed of the extruder will help to avoid the problem of Gouge. Adjust the "default print speed" to control the speed at which the extruder squeezes out the plastic. For example, if you printed before at 3600mm / Min (60 mm / s) , try to halve the value and see if the problem with the Gouge has gone away.

**10.3 Check if the nozzle is blocked**

After the reduced temperature and print speed, if you still have problems with the Gouge, it may be that the nozzle is blocked. please

Read the "Nozzle Blockage" section to learn how to deal with this problem.。



**11 Nozzle Blocked**

Your 3d printer, in its life cycle, needs to melt and extrude several kilograms of plastic. All the plastic has to be squeezed through a hole the size of sand, which complicates the problem. Inevitably, in the process, there will be some problems, the extruder can no longer push plastic through the nozzle. This blockage is often caused by something in the nozzle that blocks the normal extrusion of the plastic. The first time this problem came up, it was a bit overwhelming, but next, we're going to introduce a few simple steps to fix a blocked nozzle 。

**11.1 Manually push the wire into the extruder**

The first thing you might want to do is try to manually push the wire into the extruder. Heat extruders to the temperature needed for plastics. Then use the control handle to squeeze out a small amount of plastic. For example, 10mm. As the extruder motor rotates, the hand gently helps push the wire into the extruder. In most cases, this extra force allows the wire to pass through the position of the problem.

**11.2 Reinstalling the wire**

If the wire is still not moving, the next thing you're GonNa do is take down the wire. Make sure the extruder is at the right temperature, then use the control panel, draw the rod back, and pull it out of the extruder. As in the front, you may need to provide additional force if the wire is not moving. When the wire is pulled out, use a pair of scissors to cut the melted or damaged parts from the wire. Then re-install the wire, see if this new, undamaged wire can be squeezed out.

**11.3 Cleaning the nozzle**

If you can't squeeze this new piece of plastic through the nozzle, you may need to clean the nozzle before continuing the operation. Many users squeeze out the wire manually by heating the extruder to 100 degrees Celsius (hopefully nothing gets stuck in the nozzle!) Others prefer to use the e string on the guitar to push the object back out of the nozzle. There are many other ways, different extruders are different, so contact your printer provider for reliable guidance.

**12. During printing stop squeezing**

如果你的打印机，在开始的时候挤出正常，但后来突然停止挤出，通常有一些因素，可能导致这个问题。我们将逐个探讨常见的原因，并提供建议解决问题。如果你的打印机，在刚开始的时候，挤出有问题，请参考“ 打印开始后，耗材无挤出”

**12.1 Filment ran-out**

The situation is easy to figure out. But before you check on any other questions, make sure you have any consumables in the extruder. If the wire in the coil is depleted, you will need to install a new volume of consumables before you start printing.

**12.2 Filament and drive gear slip**

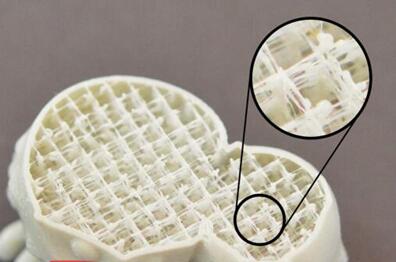
During the printing process, the motors of the extruder are constantly rotating to propel the wire into the nozzle, so that your printer can continue to extrude plastic. If you try to print too fast, or if you try to extrude too much plastic, it may cause the motor to shaving the filament until the drive gears can't hold the filament. If the extruder motor is rotating, but the wire is not moving, then it's probably that reason. Please refer to the section on "gouge" to obtain a more detailed explanation of the problem.

**12.3 Extruder blockage**

If it wasn't for any of the above, it was probably the extruder that was jammed. If this happens during the printing process, you may need to check and confirm that the wire is clean and there is no dust on the coil. When enough dust is attached to the wire, it may cause the nozzle to jam during the printing process. There are other possible reasons for blocking the nozzle, please refer to the "Nozzle blocked" section for more information

**12.4 Extruder motor drive overheating**

During the printing process, the motor load of the extruder is very large. It continues to rotate back and forth, pulling the wire back and forth. These rapid movements require a lot of electrical current. If the printer's circuit does not dissipate heat effectively, it may cause the motor drive circuit to overheat. This motor drive usually has overheated protection, and when the temperature is too high, it shuts down the motor. When that happens, the XY axis motors will spin and move the jets, but the motors in the extruder are completely stationary. The only way to solve this problem is to turn off the printer and let the circuit cool down. If the problem persists, you can also add an extra cooling fan



**13.Not filling well**

The padding part of the 3D printer plays a very important role in increasing the strength of the model. In 3d printing, the Shell that connects the outer layer is filled, while also supporting the surface that will be printed on it. If the padding is not strong or thin, you need to tweak several settings in the software to enhance this part.

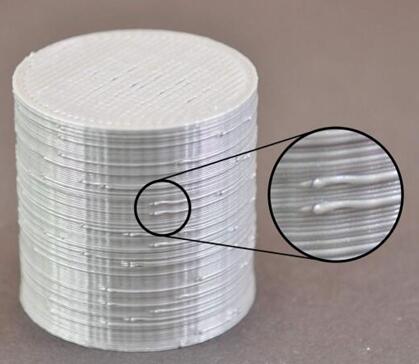
**13.1 Try replacing the fill texture**

First you need to study the settings that you use in the print, using the filler texture "internal fill texture" to determine what texture is used inside the print. Some textures are stronger than others. For example, grids, trigonometry and solid honeycomb are solid filling textures. Other textures, such as linear, or fast honeycomb may sacrifice strength in exchange for faster printing speed. If you have a problem with producing solid fillers, try different textures to see if they will be different.

**13.2 Reduce the printing speed**

In the 3D printing process, the filling speed is usually faster than the other parts. If you try to get the printer to print too fast, the extruder may not keep up with the problem of undercharging within the model. This undercharging will result in a weak, thin filling because the nozzle can not squeeze out enough plastic as the software would expect. If you try a few filler textures, but you still can't fill them in, try to reduce the speed of printing. The speed at which a direct decision is made to fill. For example, if you previously printed at 3600mm / Min (60 mm / s) , try to halve the value to see if the padding starts to become more robust.

**13.3 Increase the fill rate**

****

1. **Spots and scars**

During the 3D printing process, when the extruder moves to a different position, it must stop continuously or start to extrude. Most extruders can produce a consistent extrusion line at run time. Then every time the extruder is turned off and turned on, it makes a noticeable difference. For example, if you look at Your 3d printed case, you will see that there are small traces of the appearance in the area where the extrusion started. The extruder has to be printed from a position in the shell of your 3d model, and when the entire casing is printed, the nozzle will return to that position. This is often referred to as a spot or scar. As you can imagine, it's hard to connect two pieces of plastic without leaving any marks.

* 1. **withdrawal and sliding**

If you notice a small flaw in the prints, the best way to find out what's causing this is to look carefully at each edge of the print. Are these defects present when the extruder begins to print along the edges Or does it only appear along the edge when the extruder has to stop If a minor flaw happens to appear at the beginning of the start loop, it's likely that your pullback settings need to be tweaked a bit. At the bottom of the pullback distance setting, there will be an extra distance called extra start distance. This option determines the difference between the pullback distance when the extruder is stopped and the distance at which the reload starts again. The extruder may have extruded too much plastic if it is found to be in the beginning of printing along the edges. You can reduce the fill-in distance by entering a negative value in the reload setting. For example, if the pullback distance is 1.0 mm and the reload distance is-0.2 mm (note the minus sign) , then each time the extruder is stopped, it pulls back 0.1 mm of plastic. However, each time the extruder is restarted, it will simply push 0.8 mm of plastic back nozzle. Adjust this setting until the extruder begins to print the edges of the flaw does not appear. If this flaw only ends along the edge and the extruder is about to stop, then there is a different setting. This setting is called coasting. You can find it under the pullback settings on the extruder tab. At the end of the edge, the slide will close the extruder for a short distance to eliminate the pressure of the nozzle. Turn this setting on and add value until you don't see the blemish at the end of the edge, and the extruder will stop. Usually, the glide distance is set between 0.2 mm and 0.5 mm, which gives a very obvious effect.

* 1. **Avoid unnecessary withdrawals**

The above mentioned withdrawal and slippage settings can help avoid the flaws that come with each withdrawal, and in some cases, it's better to avoid it altogether. So the extruder does not have to reverse direction, but can make a beautiful and consistent extrusion. This is particularly important for the Bowden extruder machine, because the great distance between the extruder and the nozzle makes it more difficult to withdrawal. To adjust the settings for the Control withdrawal, you can open the advanced tab and look for "Ooze Control Behavior" . This paragraph contains a lot of useful settings that can modify the behavior of your printer. As we mentioned in the gouge section, the withdrawal is mainly used to prevent the nozzle droop from moving between the different printouts of the printout. However, if the nozzle does not move to an open area, the droop will occur inside the model and can not be seen from the outside. For this reason, many printers need to turn on the settings that "only draw back when moving to open space" to avoid unnecessary withdrawal

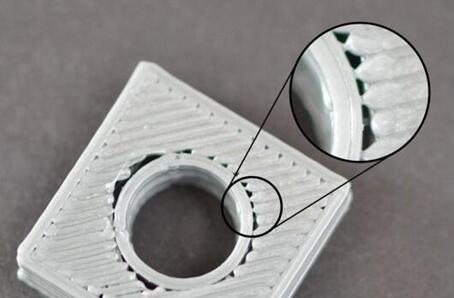
Another related setup can be found in the "mobile behavior" section. If your printer only withdrawal when it's moving to open space, it's better to try to avoid this open space There is a very useful setting in Simplify 3D that allows the movement path of the extruder to be turned so as to avoid intersecting with the contour edge. If you avoid intersecting with the contour by modifying the extruder's moving path, the withdrawal will be unnecessary. If you want to use this feature, simply open the "avoid moving paths intersect with the Outer Contour" option.

* 1. **non-fixed withdrawal**

Another very useful feature is the ability to implement non-fixed withdrawal. It was useful for Bowden's Extruder, and when it was printed, there was a lot of pressure in the nozzle. Usually when the machine stops, the extruder sits still, and because of internal pressure, it still squeezes out a small pile.

* 1. **Selecting the starting position**

If you still see imperfections on the printed surface, most of the time, the choice of these locations is to optimize the speed of printing. You can then randomize these locations, or you can set them to a specific location. For example, if you are printing a statue, you need to set all the starting points, starting at the back of the model so that they can't be seen from the front.



**15.The gap**

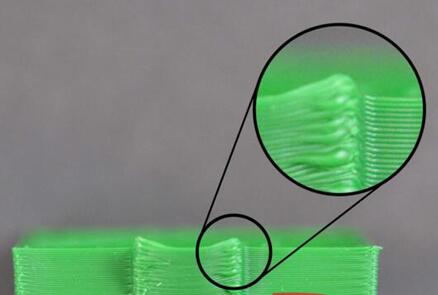
Reassemble a Printout, all composed of contours along the edges and padding. The outer edge path of the outline constitutes a solid and accurate surface of the printed piece. The padding is printed along the inside and is used to fill the remaining space in the layer. It is usually filled with a fast back-to-back texture so that the printing speed is faster. Filling has different textures, and it is important that the outline of the printout be solidly bonded to the filling. If you find that there are gaps on the edges of the padding, there are several settings you need to check.

**15.1 Outline overlap is not enough**

This value is based on the percentage of the width of your extruded silk, so it is easy to extend and adjust for different nozzle sizes. For example, if you set a 20% overlap, this means that the software will command the printer, and the padding will overlap with the innermost edge by 20% . This overlap helps to ensure that the two parts are cohesive. For example, you used to use the 20% overlap and try to increase it to 30% to see if the gap between the padding and the side disappears.

**15.2 Printing is too fast**

The speed at which your printer fills up is much faster than the outline. If the padding is printed too quickly, it will not have enough time to adhere to the outer contour. If you're trying to increase the overlap, but still see the gap between the outline and the padding, you need to reduce the speed of printing. For example, if you previously set the print speed to 3600mm / Min (60 mm / s) , try to reduce this value to half, and see if the gap between the padding and the contour will disappear. If the gap doesn't appear at low speeds, gradually increase the speed of printing until you find the best speed for your printer.



**16 corner curls and rough**

If a roll problem is found at the later stage of printing, passing means there is overheating problem. Plastic is squeezed out of the nozzle at a very high temperature, it doesn't cool down in time, and over time, it can deform. The curl can be resolved by rapidly cooling each layer so that it has no chance of deforming before it solidifies. Refer to the "over-heat" section for more detailed descriptions and solutions. If you find a curl in the first few minutes of printing, refer to "printed consumables not glued to the platform"

1. **Top surface scar**

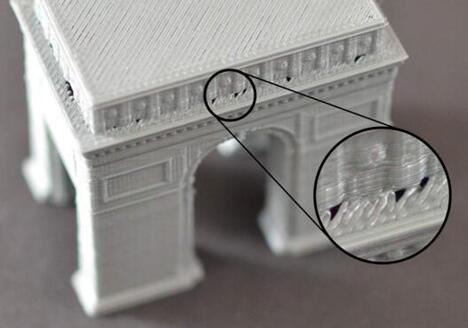
The advantage of 3D printing is that each print builds a layer at a time. This means that each layer is independent and the nozzle is free to move to any location on the platform while the print remains in the construction below. This provides a lot of opportunities, and you may see that the nozzle moves along the front surface, leaving a trail. It's easiest to see from the top of the print. The scar was created when the nozzle moved to a new position, dragging it across the plastic that had previously been printed. The following sections will explore several possible reasons and provide advice on which settings to adjust to avoid problems.

* 1. **squeezed too much filament**

First thing you need to know is that your extruder didn't extrude too much filament. If too much filament is extruded, each layer will tend to be thicker than expected. This means that when the nozzle moves through each layer, it can drag and drop the plastic that was previously overcrowded. Before you check on other settings, make sure you haven't extruded too much plastic. Please refer to the "squeezed too much filament" section for more information。

* 1. **Vertical lift (Z lift)**

If you're sure the extruder squeezes out the correct amount of plastic, but still has the problem of the nozzle dragging on the top surface, you need to look at the vertical lift settings. Opening this option will cause the nozzle to lift some distance before moving over the previously printed layer. When it reaches its target position, the nozzle will be moved back to its original height for printing. By moving up a certain height, the upper surface of the print can be avoided by scraping the nozzle.



1. **Holes and gaps in the corners of the bottom surface**

When the printing, each layer is built on the first layer. However, the amount of plastic used for printing is also a factor, so the strength of the foundation needs to be balanced against the amount of plastic used. If the foundation is not strong enough, there will be holes and gaps between layers. It is especially easy to appear at the edges where the size changes (for example, you print a 20cm square above a 40cm square) . When printing is converted to a smaller size, you need to ensure that there is sufficient foundation to support the 20mm box side wall. The reason for the poor foundation is usually a few. Next, we'll go through them one by one and then explain how to set it to improve the quality of the print

* 1. **Insufficient number of edges**

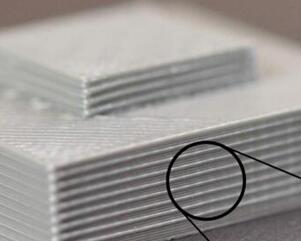
Adding more contour edges to the print will significantly enhance the base. Because the inside of the print, usually partially hollow, the Outer Wall is very thin and will have a big impact. For example, if you have previously printed 2 layers of outer edge, try printing the outer edge of the fourth layer to see if the gap has disappeared.

* 1. **Insufficient top layer**

Another common reason for the poor foundation is that you don't have enough solid padding at the top of your printer. Too thin a wall to support the structure on top of it. If you've been using the 2-layer solid layer before, try 4 layers of solid layer to see if it improves

* 1. **Fill rate is too low**

The last setup you need to check is the fill rate, and the top layer is built on top of the padding, so it's important to have enough padding to support these layers. For example, if you set the fill rate to 20% , try adding this value up to 40% to see if the print quality improves.



1. **Side linear texture**

The appearance of 3D printing is made up of hundreds of layers, which, if all goes well, will look like a smooth surface. Then, if there is a problem on only one level, it can be clearly found on the outside surface of the print. These incorrect layers cause the print to look like a linear texture. This flaw usually occurs periodically, which means that the line appears regularly (for example, every 15 layers) . Next, several common causes will be discussed.

* 1. **Extrusion instability**

The most likely reason for this problem is the poor quality of the filament. If the wire tolerances are large, you will find this change on the outside of the print. For example, the diameter of your entire volume will fluctuate only 5% , and the width of the plastic line squeezed out of the nozzle will change by 0.05 mm. This extra extrusion will cause the corresponding layer to be wider than the other layers, and finally a line will be seen on the outside of the print. In order to produce a smooth surface, your printer needs a stable extrusion condition, which requires high-quality consumables. Please read the unsteady extrusion to find out what else is causing the fluctuations.

* 1. **Temperature fluctuations**

Most 3D printers use pins to regulate the temperature of the extruder. If the PIN is out of tune, the temperature of the extruder will fluctuate over time. Given the principle of pin-and-foot control, the fluctuations are repeated frequently, which means that the temperature fluctuates like a sine wave. When the temperature is too high, the smoothness of the plastic extrudes is different from when it is cooler. This will cause the printer to extrude the layer is not the same, cause the outside surface of the printer to appear texture. A properly tuned printer should be able to control the temperature of the extruder to between positive and negative 2 degrees. During your printing process, you can use the device control panel to monitor the temperature of the extruder. If it fluctuates more than two degrees, you need to recalibrate the pin-pin controller. Please contact your printer provider for more detailed information about how to operate.

* 1. **Mechanical issues**

If you're sure that unstable extrusion and temperature fluctuations are not the culprits, it could be mechanical failure that causes the linear texture of the printed surface. For example, if the print platform wobbles during the printing process, it causes the nozzle position to fluctuate. This means that some layers are thicker than others. These thicker layers will produce a linear texture on the printed surface. Another common problem is that the z-axis is not properly mounted. For example, the return error, or the insufficient control of the motor subdivision. Even small changes in the platform will have a significant impact on the quality of the print on each layer.



1. **Vibration and loopback texture**

The loopback is a wavy texture on the surface of a printed piece because of the vibration of the printer. Usually, you see this texture when the extruder suddenly turns, as it is near a sharp corner. For example, if you print a 20mm square, each time the extruder goes to print the other side of the cube, it needs to change direction. When the sudden direction changes, the inertia of the extruder will cause vibration, which will appear in the printed piece. Next, we'll discuss the most common ways in which the loopback appears and explain why

* 1. **Printing too fast**

The most common reason for the loopback texture is that you're trying to print too fast. When the printer suddenly changes direction, rapid movement will lead to more force, thus creating a lingering vibration. If you think your printer is printing too fast, try slowing it down. You need to adjust the "default print speed" and "X / Y Axis Movement speed" . The first controls the speed at which the extruder squeezes out the plastic, and the second controls the fast moving speed of the extruder without the plastic being squeezed out. You need to adjust both settings to see the impact

* 1. **Firmware acceleration**

The firmware in your 3d printer motherboard usually has an acceleration control to avoid a sudden turn. This acceleration setting causes the printer to speed up and decelerate slowly as it turns. This feature is essential to avoid loopbacks. If you can modify the firmware, you can even slow down the acceleration to make the change more gradual. This is very helpful in reducing the loopback.

* 1. **Mechanical failure**

If nothing else can solve the problem of the loop, you may need to examine the mechanical factors, which may cause too much vibration. For example, the screw may be loose, the brace may be broken, it causes too much vibration. When you print, look carefully at your printer, and try to examine where the vibrations are coming from. Many of our users end up tracking the problem as a printer machine malfunctions. So, when the above suggestion is invalid, it is necessary to check it



1. **There is a gap on the thin wall**

Because you have a fixed-size nozzle on your printer. When printing is only a few times larger than the diameter of the nozzle, you may have problems. For example, you might use a 0.4 mm nozzle to print the thickness of 1.0 mm, so you need to make some adjustments to make sure that your printer is fully solid and doesn't leave any gaps. Next, we'll show you the settings.

* 1. **Adjusting thin wall behavior**

The first thing you need to check is that the software contains a very useful feature called "gap fill" . Like its name, it will allow your printer to fill the tiny gaps between the thin walls. To Open this setting, make sure that "allowing the gap to be filled" is check-in status when needed. When you open this option, if you still find a gap in the thin wall, there is another option to try. Open the "Fill Tab" to increase the value set by "contour overlap" , which will allow more space to be filled in the gaps between the thin walls. For example, if you previously set the value of the overlap to 20% , try changing the value to 30% to see if the thin wall will become more robust.



1. **The details cannot be printed.**

On your printer, there is a fixed size nozzle that you can use to accurately print very fine details. For example, many printers have a nozzle with a diameter of 0.4 mm. This works well enough when you print many pieces, but you run into problems when the print is so small as to be smaller than the diameter of the nozzle. For example, you're trying to print a 0.2 mm thin wall with a 0.4 mm nozzle. The problem is that you can't precisely produce a 0.2 mm extruder from a 0.4 mm extruder. The width of the extruded wire must always be larger or equal than the diameter of the nozzle. In this way, the software tells you that you can't use your printer, the current installation of the nozzle, to print such too subtle features. If you print small details often, this can be a common problem for you. There are a few options that will allow you to successfully print this fine detail. Now, let's go through each of them.

* 1. **Redesigning a thin-walled model**

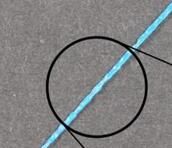
The most obvious option is to redesign your model to include only features that are larger than your nozzle diameter. This usually involves editing the 3D model in the CAD file to modify the size of the fine features. When you increase the thickness of the fine feature, you can re-import the model to see if your printer can print the 3D shape you created. If the feature can be seen in the preview mode, the printer can print the modified feature.

* 1. **Installing a nozzle with a smaller opening diameter**

Most of the time, you can't edit the original 3D model directly. For example, it can be designed by someone else, or you can download it from the Internet. At this point, you need to consider attaching a second nozzle to your printer to print out subtle features. Most printers and nozzles are detachable, making it easier to adjust after buying. For example, you can buy a 0.3 mm and a 0.5 mm nozzle at the same time, so there are two options. Contact your printer provider to get accurate instructions about how to replace smaller nozzles.

* 1. **The last method is to force the software to print smaller details.**

If you can't redesign the 3D model, and you can't replace a smaller nozzle on your 3d printer, then you have one last option. You can force the software to print these subtle features. Then, there may be some print quality issues. For example, if your nozzle is 0.4 mm, you may choose to customize the width to 0.3 mm to force the software to print as small as 0.3 mm. However, as mentioned above, most nozzles can not precisely produce a extruded silk smaller than its open hole diameter, so it is acceptable to look carefully at your printer and determine the quality of the print.



**23 extrusion instability**

To get your printer to print high-precision printouts, you need to extrude the plastic to a stable capacity. In the different parts of the printout, the extruded silk changes, which will affect the final print quality. By carefully observing the printing process, you can identify the problem of crowding out instability. For example, if the printer prints a line that is 20mm long, but you find that the extruded silk looks bumpy or uneven, or if the size seems to fluctuate, you might have a problem. We excerpted several common reasons for crowding out unstable, and described how to avoid them.

**23.1 The wire is stuck or entangled**

The first thing you need to look at is your printer's feed consumables. You need to make sure that the reel is smooth and that the plastic wire can be pulled out of the roll easily. If the wire is messy, or the drag of free rotation is too great, this will affect the smooth extrusion of the nozzle. If you have Bowden tubes installed on your printer (a very small hollow tube through which the wire passes through) , you need to check if the wire is able to pass through the tube without much drag. If there is too much resistance in the tube, you need to try to clean a tube, or add some lubrication to the tube

**23.2 Nozzle**

If the wire isn't stuck and can easily push into the extruder, the next thing to check is the nozzle itself. There may have been some small fragments, or foreign objects entering the nozzle, which prevented the normal extrusion. A simple way to test this possibility is to use the device control panel, which squeezes out some plastic from the nozzle. Observe to ensure that plastic extrusion is continuous and stable. If you find a problem, you may need to clean the nozzle. Please contact your printer provider to get the correct way to clear the inside of the nozzle.

**23.3 The height is too small**

If the wire rotates smoothly and the extruder is not blocked, it is useful to check the settings. For example, if you try to print very small layers, such as 0.01 mm, there will be very little room for plastic to extrude the nozzle. There is only a 0.1 mm gap under the nozzle, which means the plastic may be hard to squeeze out. Again and again, you use an appropriate height for your printing. If you are using a very small layer to print, try adding a layer height to see if the problem has been solved.

**23.4 Wrong extruded wire width**

The other setup that needs to be checked is the width of the extruded wire you set for the extruder. You can find this setting, each extruder has its own single extruded silk width, so make sure that you select the appropriate extruder from the list to view the settings. If the width of the extruder is really smaller than the diameter of the nozzle, this can cause problems. In general, the width of the extruded silk needs to be between 100% and 150% of the diameter of the nozzle. If your extruder width is much smaller than that of the nozzle (for example, the width of the extruded wire, for example, and the nozzle diameter of 0.4 mm) , the extruder will not be able to squeeze out the wire steadily.

**23.5 Consumables are of poor quality**

Another factor associated with crowding out instability that we haven't mentioned yet is the quality of the wire you're using. Poor quality wire material may contain other additives that affect the stability of extrusion. It may also be that the diameter of the wire is unstable, which can also lead to crowding out instability. In the end, a lot of plastic, over time, has a tendency to degrade. For example, the PLA tends to water in the air, which leads to a reduction in the quality of print. This is why plas are packed with desiccant to help remove moisture from the consumables. If you have a problem with your consumables, try replacing a new roll of unwrapped, high quality consumables, and see if it's possible to solve the problem.

**23.6 Extruder mechanical failure**

If you check all of the above, but there are still problems with crowding out instability, then you may need to check your extruder for mechanical failure. For example, many extruders use drive gears with sharp teeth, and the gears bite into the wire. This makes it easier to pull the wire before and after the extruder. This extruder can usually adjust the pressure of the gear on the wire. If set too loose, the drive gear will be unable to bite the wire deep enough, which affects the ability of the extruder to precisely control the position of the wire.