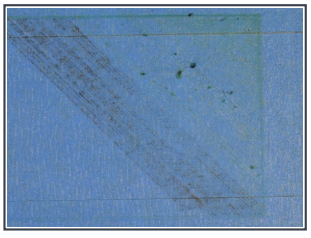
PRINT QUALITY TROUBLESHOOTING GUIDE

Print Quality Troubleshooting Guide

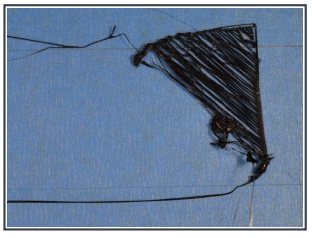
This guide is a great place to start if you are trying to improve the quality of your 3D printed parts. We have compiled an extensive list of the most common 3D printing issues along with the software ection of real-world ed parts. So let's get

e quality issue that you are seeing in your own 3D printed parts. You can click on the thumbnail to jump that 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, feel free to scroll down and read through each section of the guide in more detail. There are plenty of useful tips to learn that can help improve your 3D printed results!



Not Extruding At Start

Printer does not extrude plastic at the beginning of the print

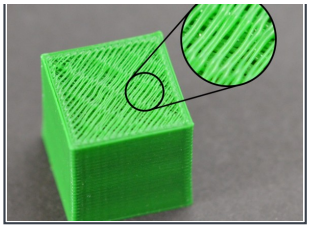


Not Sticking To Bed

The first layer does not stick to the bed and the print quickly fails

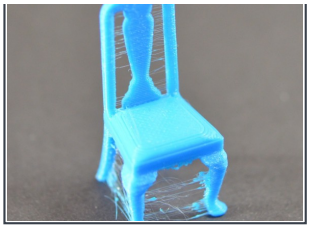
Under-E

Printer dc extrude e plastic, ga perimeter



Gaps in Top Layers

Holes or gaps in the top layers of the print

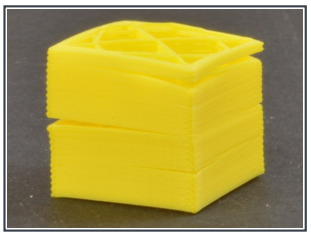


Stringing or Oozing

Lots of strings and hairs left behind when moving between different sections of the print

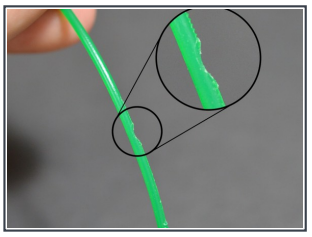
Overhe

Small fea
become c
and defo



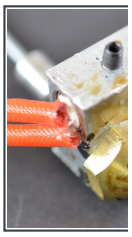
Layer Separation and Splitting

Layers are separating and splitting apart while printing



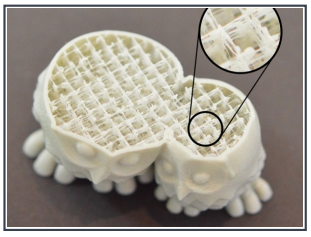
Grinding Filament

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



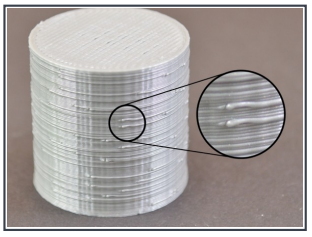
Clogged

Extruder
jammed a
longer ex
from the



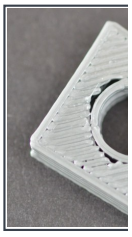
Weak Infill

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



Blobs and Zits

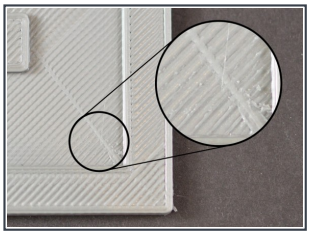
Small blobs on the surface of print, otherwise known as zits



Gaps Be
Infill an

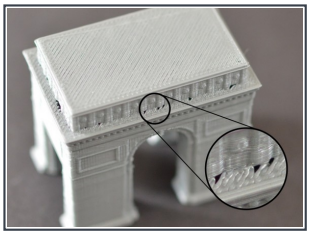
Gaps bet
outline of
and the o

infill layer



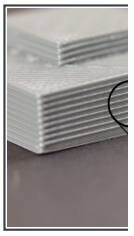
Scars on Top Surface

The nozzle drags across the top of the print and creates a scar on the surface



Gaps in Floor Corners

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

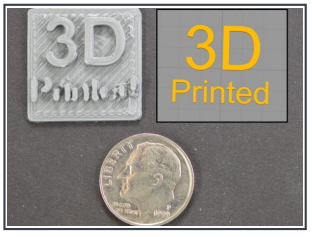


Lines or of Print

Side walls smooth, li visible on the print

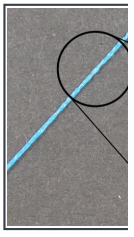
Gaps in Thin Walls

Gaps between thin walls of the print where the perimeters do not touch



Small Features Not Printed

Very small features are not printed or are missing from the software preview



Inconsis Extrusic

Extrusion tends to v not consi enough t an accura

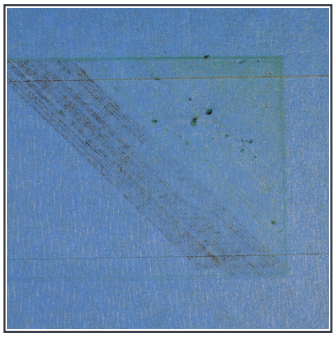
Poor Surface Above Supports

Poor surface quality on the underside of the part where it touches the support structures

List View

Each issue will now be listed one-by-one with details about what causes the problem and instructions for troubleshooting it. If you want to read more about a particular section, just click on the plus sign (“+”) to expand the section.

Not Extruding at Start of Print



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

+ Extruder was not primed before beginning the print

+ Nozzle starts too close to the bed

+ The filament has stripped against the drive gear

+ The extruder is clogged

Print Not Sticking to the Bed

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

+ Build platform is not level

+ Nozzle starts too far away from the bed

+ First layer is printing too fast

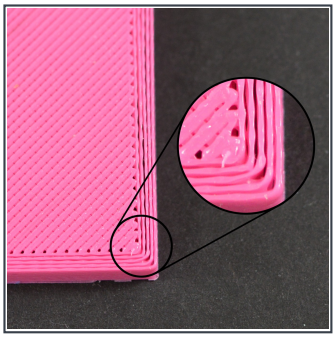
+ Temperature or cooling settings

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

+ When all else fails: Brims and Rafts

Not Extruding Enough Plastic

Each profile in Simplify3D includes settings that are 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-

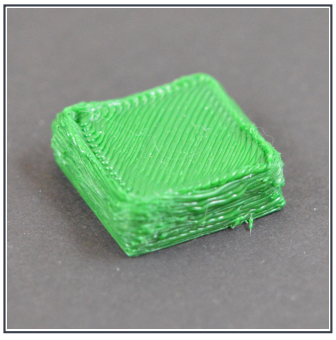


extrusion). If this happens, you may start to notice gaps between adjacent extrusions of each layer. The most reliable way to test whether or not your printer is extruding enough plastic is to print a simple 20mm tall cube with at least 3 perimeter outlines. At the top of the cube, check to see if the 3 perimeters are strongly bonded together or not. If there are gaps between the 3 perimeters, then you are under-extruding. If the 3 perimeters are touching and do not have any gaps, then you are likely encountering a different issue. If you determine that you are under-extruding, there are several possible causes for this, which we have summarized below.

+ Incorrect filament diameter

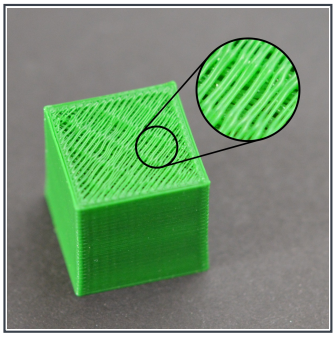
+ Increase the extrusion multiplier

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 Simplify3D. Please see the Not Extruding Enough Plastic section for a more detailed description. While those instructions are for under-extrusion, you will adjust the same settings for over-extrusion, just in the opposite direction. For example, if increasing the extrusion multiplier helps with under-extrusion, then you should decrease the extrusion multiplier for over-extrusion issues.

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, Simplify3D 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 thanks to Simplify3D's great infill options. 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.

+ Not enough top solid layers

+ Infill percentage is too low

+ Under-Extrusion

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 within Simplify3D that can help with this issue. The most common setting that is used to combat excessive stringing is something that is known as

retraction. If retraction is enabled, when the extruder is done printing one section of your model, the filament will be pulled backwards into the nozzle to act as a countermeasure against oozing. When it is time to begin printing again, the filament will be pushed back into the nozzle so that plastic once again begins extruding from the tip. To ensure retraction is enabled, click “Edit Process Settings” and click on the Extruder tab. Ensure that the retraction option is enabled for each of your extruders. In the sections below, we will discuss the important retraction settings as well as several other settings that can be used to combat stringing, such as the extruder temperature settings.

+ Retraction distance

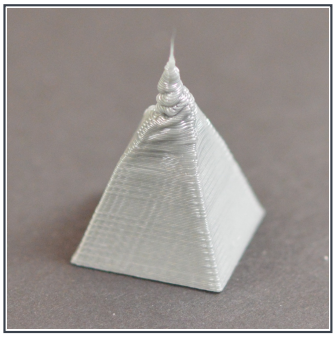
+ Retraction speed

+ Temperature is too high

+ Long movements over open spaces

+ Movement Speed

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 on the left, 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.

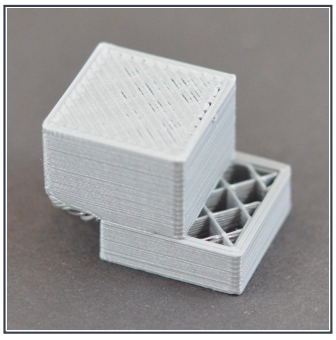
+ Insufficient Cooling

+ Printing at too high of a temperature

+ Printing too fast

+ When all else fails: Try printing multiple parts at once

Layer Shifting or Misalignment

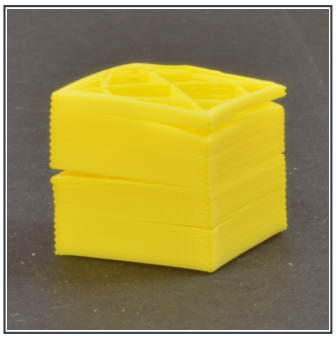


Most 3D printers use an open-loop control system, which is a fancy way to say that they have no feedback about the actual location of the toolhead. The printer simply attempts to move the toolhead to a specific location, and hopes that it gets there. In most cases, this works fine because the stepper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

+ Toolhead is moving too fast

+ Mechanical or Electrical Issues

Layer Separation and Splitting

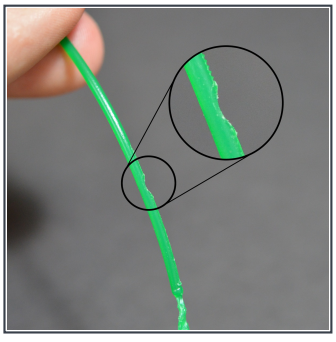


3D printing works by building the object one layer at a time. Each successive layer is printed on top of the previous layer, and in the end this creates the desired 3D shape. However, for the final part to be strong and reliable, you need to make sure that each layer adequately bonds to the layer below 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.

+ Layer height is too large

+ Print temperature is too low

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.

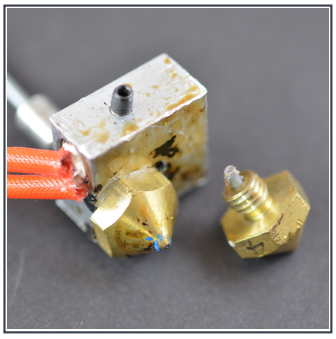
+ Aggressive Retraction Settings

+ Increase the extruder temperature

+ Printing too fast

+ Check for a nozzle clog

Clogged Extruder



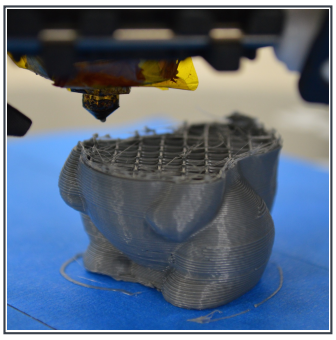
Your 3D printer must melt and extrude many kilograms of plastic over its lifetime. To make things more complicated, all of this plastic must exit the extruder through a tiny hole that is only as big as a single grain of sand. Inevitably, there may come a time where something goes wrong with this process 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.

+ Manually push the filament into the extruder

+ Reload the filament

+ Clean out the nozzle

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.

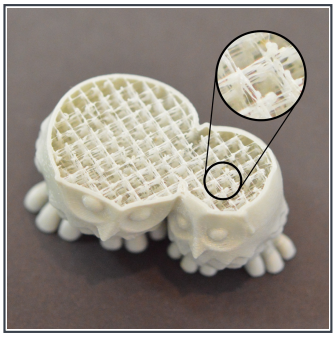
+ Out of filament

+ The filament has stripped against the drive gear

+ The extruder is clogged

+ Overheated extruder motor driver

Weak Infill



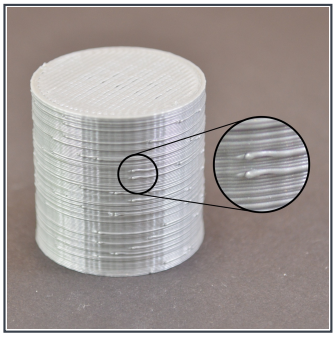
The infill inside your 3D printed part plays a very important role in the overall strength of your model. The infill is responsible for connecting the outer shells of your 3D print, and must also support and upper surfaces that will be printed on top of the infill. If your infill appears to be weak or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

+ Try alternate infill patterns

+ Lower the print speed

+ Increase the infill extrusion width

Blobs and Zits



During your 3D print, the extruder must constantly stop and start extruding as it moves to different portions of the build platform. Most extruders are very good at producing a uniform extrusion while they are running, however, each time the extruder is turned off and on again, it can create extra variation. For example, if you look at the outer shell of your 3D print, you may notice a small mark on the surface that represents the location where the extruder started printing that section of plastic. The extruder had to start printing the outer shell of your 3D model at that specific location, and then it eventually returned to that location when the entire shell had been printed. These marks are commonly referred to as blobs or zits. As you can imagine, it is difficult to join two pieces of plastic together without leaving any mark whatsoever, but there are several tools in Simplify3D that can be used to minimize the appearance of these surface blemishes.

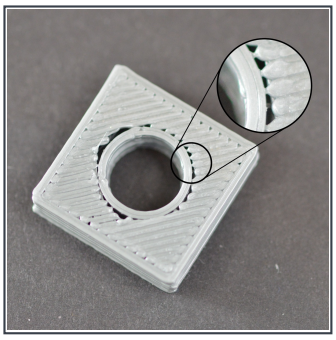
+ Retraction and coasting settings

+ Avoid unnecessary retractions

+ Non-stationary retractions

+ Choose the location of your start points

Gaps Between Infill and Outline

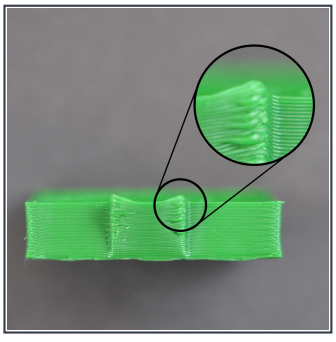


Each layer of your 3D printed part is created using a combination of outline perimeters and infill. The perimeters trace the outline of your part creating a strong and accurate exterior. The infill is printed inside of these perimeters to make up the remainder of the layer. The infill typically uses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

+ Not enough outline overlap

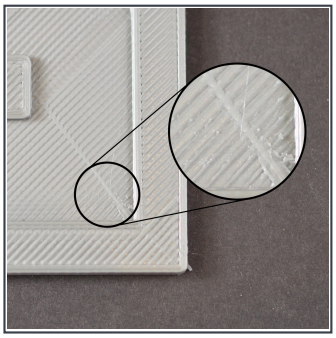
+ Printing too fast

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.

Scars on Top Surface

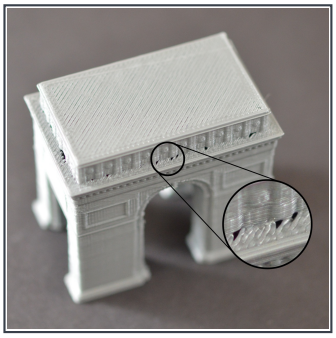


One of the benefits of 3D printing is that each part is constructed one layer at a time. This means that for each individual layer, the nozzle can freely move to any portion of your print bed, since the part is still being constructed down below. While this provides for very fast printing times, you 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.

+ Extruding too much plastic

+ Vertical lift (Z-hop)

Holes and Gaps in Floor Corners



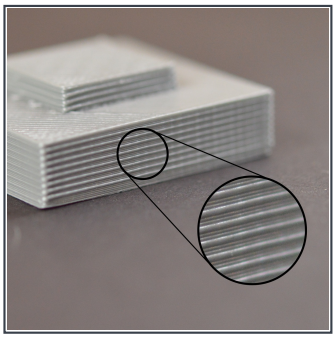
When building a 3D printed part, each layer relies on the foundation from the layer below. However, the amount of plastic that is used for the print is also a concern, so a balance must be achieved between the strength of the foundation and the amount of plastic that is used. If the foundation is not strong enough, you will start to see holes and gaps between the layers. This is typically most obvious in the corners, where the size of the part is changing (for example, if you were printing a 20mm cube on top of a 40mm cube). When you transition to the smaller size, you need to make sure that you have a sufficient foundation to support the sidewalls of the 20mm cube. There are several typical causes for these weak foundations. We will discuss each one below and present the settings that can be used in Simplify3D to improve the print.

+ Not enough perimeters

+ Not enough top solid layers

+ Infill percentage is too low

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.

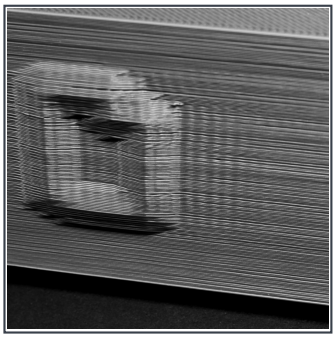
+ Inconsistent extrusion

+ Temperature variation

+ Mechanical issues

Vibrations and Ringing

Ringing is a wavy pattern that may appear on the surface of your print due to printer vibrations or wobbling. Typically, you will notice this pattern when the extruder is making a sudden direction change, such as near a sharp corner. For example, if you were printing a 20mm cube, each time the



extruder changes to printing a different face of the cube, it would need to change directions. The inertia of the extruder can create vibrations when these sudden direction changes occur, which will be visible of the print itself. We will look at the most common ways to address ringing, by examining each cause in the list below.

+ Printing too fast

+ Firmware acceleration

+ Mechanical issues

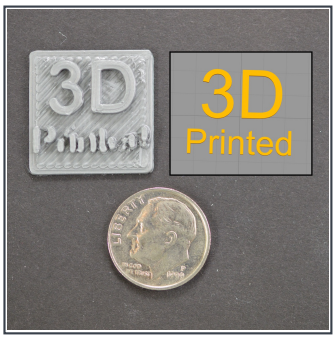
Gaps in Thin Walls

Because your 3D printer includes a fixed size nozzle, you may encounter issues when printing very thin walls that are only several times larger than the nozzle diameter. For example, if you were trying to print a 1.0mm thick wall with a 0.4mm extrusion width, you may need to make some adjustments to ensure your printer creates a completely solid wall and does not leave a gap in the middle. Simplify3D already includes several dedicated settings to help with thin wall printing, so we will describe the relevant settings below.

+ Adjust the thin wall behavior

+ Change the extrusion width to fit better

Very Small Features Not Being Printed



Your printer includes a nozzle with a fixed size that allows your 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 “Prepare to Print” in Simplify3D, 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.

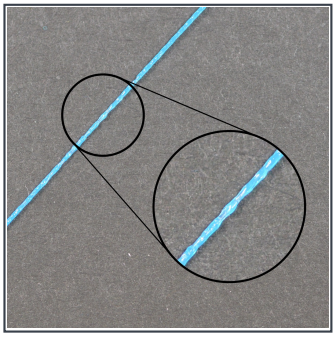
+ Redesign the part to have thicker features

+ Install a nozzle with a smaller tip size

+ As a last resort, force the software to print smaller features

Inconsistent Extrusion

For your printer to be able to create accurate parts, it needs to be capable of extruding a very consistent amount of plastic. If this extrusion varies across different parts of your print, it is going to affect the final print quality. Inconsistent extrusion can usually be identified by watching your printer closely as it prints. For example, if the printer is printing a straight line that



is 20mm long, but you notice that the extrusion seems rather bumpy or seems to vary in size, then you are likely experiencing this issue. We have summarize the most common causes for inconsistent extrusion, and explained how each one can be addressed.

+ Filament is getting stuck or tangled

+ Clogged extruder

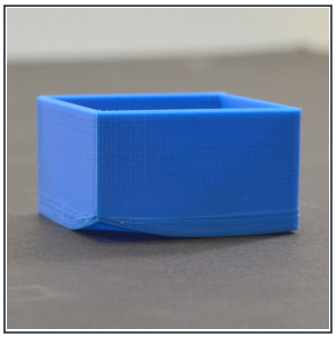
+ Very low layer height

+ Incorrect extrusion width

+ Poor quality filament

+ Mechanical extruder issues

Warping



As you start printing larger models, you may start to notice that even through the first few layers of your part successfully adhered to the bed, later on the part begins to curl and deform. This curling can be so severe that it actually causes part of your model to separate from the bed, and may cause the entire print to eventually fail. This behavior is particular common when printing very large or very long parts with high temperature materials such as ABS. The main reason for this problem is the fact that plastic tends to shrink as it cools. For example, if you printed an ABS part at 230C and then allowed it to cool to room temperature, it will shrink by almost 1.5%. For many large parts, this could equate

to several millimeters of shrinkage! As the print progresses, each successive layer will deform a bit more until the entire part curls and separates from the bed. This can be a challenging issue to solve, but we have several helpful suggestions to get you started.

+ Use a Heated Bed

+ Disable Fan Cooling

+ Use a Heated Enclosure

+ Brims and Rafts

Poor Surface Above Supports

One of the major benefits of Simplify3D is the ability to create innovative support structures which allow you to create incredibly complex parts that would be hard to manufacture otherwise. For example, if you have a steep overhang or part of your model within nothign below it, then a support structure can provide a foundation for these layers. The support structures created by Simplify3D are disposable and can be easily separated from the final part. However, depending on your settings, you may find that some adjustments are needed to perfect the surface quality on the underside of your parts, right above the support structure foundation. We will explain the key settings below and how they can affect your prints.

+ Lower Your Layer Height

+ Support Infill Percentage

+ Vertical Separation Layers

+ Horizontal Part Offset

+ Use a Second Extruder

Congratulations! You've reached the end of our list for the most common print quality issues that you are likely to encounter with 3D printing. If you are experiencing an issue that was not mentioned in this guide, there are still plenty of ways that you can get help and advice. A great place to start is the Simplify3D User Community. You can search for posts from others users with the same issue, or post pictures of your prints and receive suggestions from other experienced users. If you would rather speak with our team directly, you can also Contact Our Support Staff and we would be happy to assist you.