2019 Edition

3D Printing Failures

How to Diagnose & Repair all Desktop 3D Printing Issues

By: Sean Aranda
Edited by: David Feeney

Special Thanks to:
All of my subscribers
David Feeney and Bennett Berger
Hanna Henry for the cover design
©2019

Table of Contents

1• Introduction
4• Good Practice
7• Diagnosing Failures
13• Bed Adhesion
23• Build Plate Not Heating
27• Build Plate Not Reading Correct Temperature
29• Built Up Material in Nozzle
35• Extruder Stepper Skipping
42• Filament Snaps
45• Ghosting
48• Hotend Can't Reach or Maintain Temperature
53• Hotend Not Heating
57• Hotend Not Reading Correct Temperature
60• Important Accessories and Replacements
65• Layer Shifts
73• LCD Blank or Dark
75• Mandatory Maintenance for your 3D Printer
80• Materials and their Settings
95• Model Errors
99• Not Finding Home and Inverted Prints
103• Nozzle Clogs
110• Over/Under Extrusion
118• Parts Being Knocked Over
123• Parts Not Mating Together
128• Poor Layer Adhesion
131• Post-Processing
136• Print Pauses Mid-Print
140• Quality Options
143• Running Out of Filament

146• Settings Issues
161• Stepper Motors Overheating or Malfunctioning
167• Stripped Filament
172• Unlevelled Build Plate
177• Warping:
182• Z- Axis Wobble
190• Z- Height Calibration
195• Tips if Still Not Working
197• Limitations Involved with 3D Printing
199• Upgrades and Purchasing a New Printer
204• Resources
206• About the Author

Introduction

When I first started working with 3D printers, I was overwhelmed by the amount of knowledge required to have consecutive successful prints. Since I was aware of the acceptable failure rate for most other manufacturing machines, I was blown away by just how inconsistent 3D printing can be without frequent, proper maintenance.

Many desktop 3D printing companies advertise a plug and play machine that can be operated consistently without any engineering background, which is generally not true. 3D printing has been marketed by many as a magical solution to manufacturing and prototyping, but anyone who has used these machines will be able to tell you different.

From the time I began in this industry roughly 4 years ago, up to the point of writing this book, I have amassed over 75,000 hours of successful printing experience across over two dozen machines. I have repaired just about every section of the printer and have even built my own. Since prints can fail up to 20% of the time on certain machines without proper maintenance, you can be safe to assume I have had to fix just about every issue you will come across.

This 2019 edition of the book has been entirely re-written. Every single chapter has been updated to any new information I have run into, including the use of new software. There are dozens of new photos to help diagnose and fix problems and there 8 entirely new chapters. All of this information is to make sure everything is up to date.

In the diagnostic section in the beginning of this book I will go over some of the visual symptoms that you may see when your printer is malfunctioning, and what could be causing the problem. Whatever failure or ugly print you are experiencing, you should be able to compare it to the problem described, and then go to the designated chapters in order to fix.

My goal with this book is to take every failure I have witnessed and put it into one resource. This book should be able to help you fix close to 100% of the problems you are going to experience with your 3D printer. If you purchased this book and it does not help you with your specific problem, I offer you to contact me anytime at my YouTube channel "The 3D Print General", or to email me at Sean@3DPrintGeneral.com

Warnings for using your 3D printer

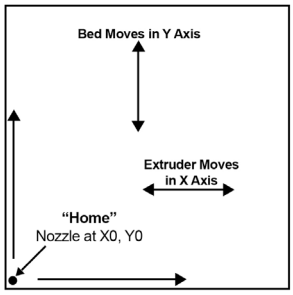
Since I personally believe this industry does not do a proper job of warning the dangers involved with 3D printing when advertising products to average

consumers, I feel it necessary to caution you of the real possibility of a fire while operating one of these machines. This is equipment that draws a lot of power, shakes and moves for hours, and has a lot of wires that can be dislodged or frayed. You shouldn't run your printer next to curtains or other flammable things, and you shouldn't leave your machine printing alone for hours if you are not confident in your build quality.

Another concern is that not much research has been done in relation to the amount of harmful particulates put into the air from melting plastic in this fashion. You can imagine that if you threw some Lego's into an oven and inhaled the smoke that was created, you would be doing some serious damage to your health. You should make sure that your 3D printer is in a well-ventilated area and that you do not stand over the hotend watching it while it prints. Some manufacturers have factored this issue into their build design and enclose the machine, ventilated with filters. Specific materials are worse for your health than others, but it is a good rule of thumb to assume that melting plastic and inhaling it is not going to be good for your lungs.

Notes about different types of printers

This book will only cover FDM desktop 3D printers and will not be able to help fixing SLA machines, SLS machines, or any other form of 3D printing. There may be some generic rules that overlap, but I was only factoring in FDM 3D printing when writing this book.



Many of the solutions I describe are specific to Cartesian machines. A Cartesian FDM 3D printer is one in which the build plate moves in the Y direction, the extruder moves in the X direction, and the carriage is moved up and down in the Z direction. Earlier Cartesian machines had the build plate move in the X and the extruder move in the Y, but the idea is still the same - the build plate moves in at least one axis other than Z. This is how most desktop FDM 3D printers function and is typically considered the simplest configuration to build and maintain.

A CoreXY machine is configured such that the bed moves in the Z direction and the extruder moves in both the X and Y direction. There are definitely some benefits involved with this method since you can achieve tighter tolerances at faster build speeds when properly maintained. A true CoreXY machine is configured such that the X and Y motors are statically mounted to the frame of the printer (ie. the motors do not move) to reduce moving mass in the X and Y axis. These machines are typically a little more difficult to maintain than classical Cartesian machines.

A variant of the CoreXY is the Hbot configuration. This style of machine features the same configuration as the CoreXY, with the exception that one of the motors on the X or Y axis moves. This provides a compromise between performance of the

CoreXY and the ease of maintenance of the classical Cartesian 3D printer.

A Delta 3D printer looks and functions quite differently than either of these, since the extruder is suspended above the build plate via three arms in a triangular fashion. These machines can normally print much faster than a Cartesian with the same specs. Their downfall is that they are typically considered the most difficult 3D printer configuration to calibrate and maintain.

You should be able to use most of the remedies in this book on CoreXY and Delta machines, but some of the dimensions described in the directions and firmware will be different.

There are also two types of extruder setups – direct drive and Bowden. A direct drive extruder is one where the stepper motor is directly feeding filament into the hotend, and is attached to the carriage. A Bowden option is where filament is fed from a stepper motor attached to the frame to the hotend carriage. A Bowden extruder will reduce the weight on your carriage, which means you can normally print much faster without seeing surface defects. The problem with Bowden machines is the difficulty involved with printing unique materials, with some very flexible options being entirely impossible. You will also need to ramp up the retraction settings in order to prevent a "hairy" print.

I do cover Bowden setups in this 2019 edition of the book, but in general I am discussing a direct drive. All fixes will work with Bowden machines, just some of the imagery may be different.

NOTE: I have done my best to have these photos show up in high quality, but Amazon's printing process does not seem to always be up to par. Email me at Sean@3DPrintGeneral and I will be happy to send you the HD version of these photos, along with a PDF version of this book - with proof of purchase.

Good Practice

Before I get into the specifics of fixing particular failures, I feel it necessary to go over some good practices to maintain your printer in top condition. Since 3D printers are mechanical machines that experience frequent rattling and movement, they require constant maintenance in order to continue performing at their peak. Look over the "Mandatory Maintenance" chapter as well to make sure you properly take care of your printer.

Keep a clean environment

If you do not make sure to always clean up your printer work area, you will be surprised just how quickly everything can get out of hand.

Make sure you always throw away excess material since it can eventually get in the way of your gears turning properly. Since there are fans blowing on different sections of your printer, you could even blow some stringy old material onto your current print.

Keeping your printer clean includes your build plate. This does not need to be cleaned after every print and will be determined by your specific build plate and adhesion method (as described in the "Bed Adhesion" Chapter). I like to clean off my build plate every couple of prints to make sure I have a proper first layer.

If you have an air compressor it can help immensely with cleaning the different parts of your printer from debris and dust.

Print replacement parts that are likely to break in advanced

You really don't want to wait for a printed part on your machine to break and to not have a replacement on hand. If you only have one 3D printer, you are going to have to order replacements, when it could have been as easy as printing an extra set when you first got your machine.

The first thing I do when I get a new machine is I print a replacement set of printed parts, the files for which are normally provided by the manufacturer. This is not much fun because the first day or two of printing will just be these parts, and not anything cool for yourself.

Keep these parts to the side and hope you will never need them, but you will be happy you did this if you ever do. I have kicked myself plenty of times in the past when I did not do this.

Slow your printer down

Many printing issues can be fixed or diagnosed easier if you run your machine a bit slower. There are possibilities you can run into a nozzle clog by doing this, but in general, you will have a much higher success rate by printing a bit slower.

You can do this by reducing your print speed in your slicer settings and by reducing your machine's acceleration, both of which are described later in this book.

I personally run my machines much slower than the manufacturer advertises.

Save slicer profiles as you go

Every time you make a tweak to your slicer settings, you should save it as a new file, organized somewhere on your computer. This is a good practice so that you can go back to profiles you know have worked in the past and can save you an immense amount of time when printing a unique material you know you have successfully printed in the past.

You can personalize profiles to specific machines and for each material you are printing with. This is the best way to hone in and perfect your settings for any given filament.

If you don't do this, you can still load a profile from a G-code. You should save your G-codes in a manner that can easily be remembered if the particular slicer settings are needed in the future.

Properly store your filament

Filament, especially nylon mixtures, can absorb and maintain moisture when kept in a humid environment. This is why when you purchase a new spool, the filament will always be vacuum sealed with desiccants.

If you do not plan on using your filament for some time, you should properly store it with a dehumidifier or vacuum seal it. If you do not, you may start to experience failures on a spool you have used successfully in the past.

Almost all filament wants to be stored in as close to 0% humidity as possible. I would set my dehumidifier to the lowest setting – 20% - which always worked great. I live in a very dry area where humidity rarely gets above 30%, so I do not have to deal with this nearly as much as someone living in a very humid area of the world.

Always watch the first layer of your print before leaving unattended

This is a very important tip. Never start a print and just walk away before watching that first layer print. Even with a $5,000 machine with auto levelling, it would be inadvisable to not watch this first layer print.

Any issue with finding the proper Z-height can lead to a failed print that can damage your machine. If the print starts too close to the build plate, you may have

to kiss your build plate and nozzle goodbye. If the print starts too far from the build plate, you are going to have a massive cleanup on your hands.

I would guess that 75% of the failures I experience can be diagnosed from the first layer. This is the fastest set of failures to diagnose, but if you don't watch the first layer printing, you will be left with quite a lot of headaches.

Dedicate machines if possible

If possible, it would be smart to dedicate specific machines for specific materials. If you have multiple machines, you should dedicate each machine to print only a couple of materials. If you have a printer that only prints ABS, you will not run into the issues that are experienced when switching materials constantly, such as black spots on your print and nozzle clogs.

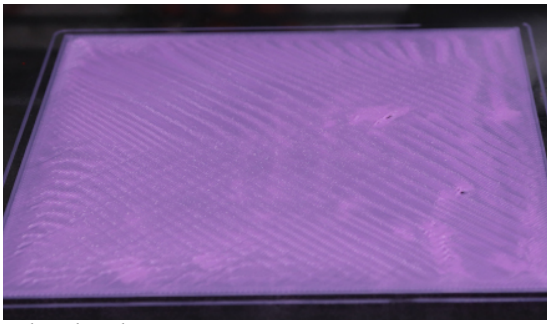
Don't get frustrated

3D printing can involve more annoyances than you may have first imagined. It might take you 20 re-prints just to hone in the proper Z-height and to level your build plate. Or you may start a 28 hour print that decides to clog 26 hours in.

There are countless ways that 3D printing can end up leaving you frustrated, but it is key to keep calm. Remember that there is a solution to the problem you are having and you just need to properly diagnose it.

Diagnosing Failures

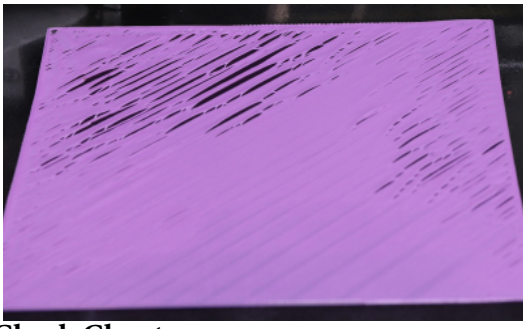
Nozzle Too Close To Buildplate



Check Chapters:

- Z-Height Calibration (p. 191)
- Unlevelled Build Plate (p. 172)

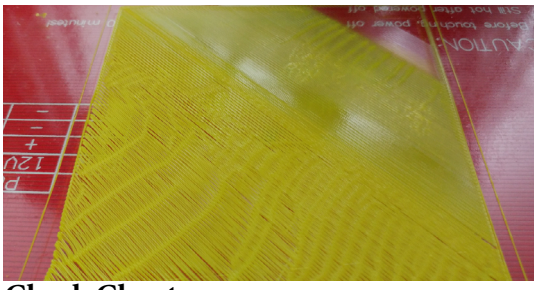
Nozzle Too Far From Buildplate



Check Chapters:

- Z-Height Calibration (p. 191)
- Unlevelled Build Plate (p. 172)

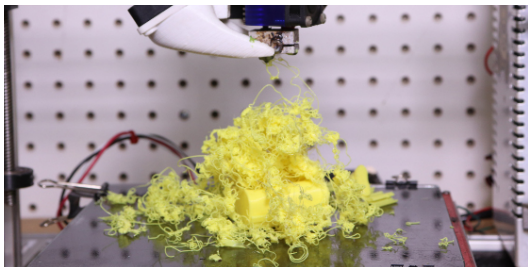
Uneven Build Surface



Check Chapters:

- Unlevelled Build Plate (p. 172)

Spaghetti Monster



Check Chapters:

- Bed Adhesion (p. 13)
- Z-Height Calibration (p. 191)

Built Up Material On Nozzle



Check Chapters:

- Built Up Material On Nozzle (p. 29)

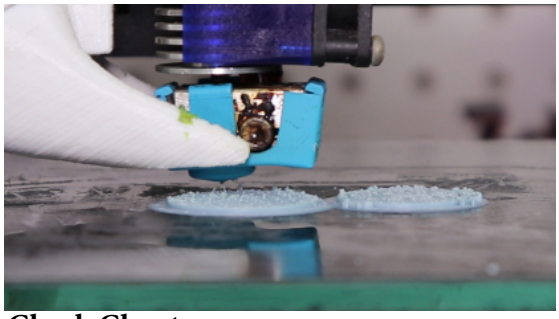
Nozzle Clogs



Check Chapters:

- Nozzle Clogs (p. 103)
- Settings Issues (p. 146)
- Bed Adhesion for Above Issue (p. 13)

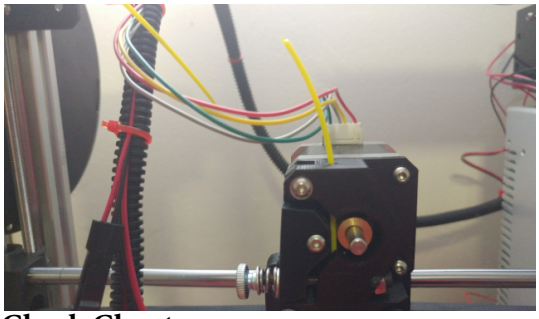
Ghost Printing



Check Chapters:

- Nozzle Clogs (p. 103)
- Stripped Filament (p. 167)
- Settings Issues (p. 146)
- Materials and their Settings (p. 180)

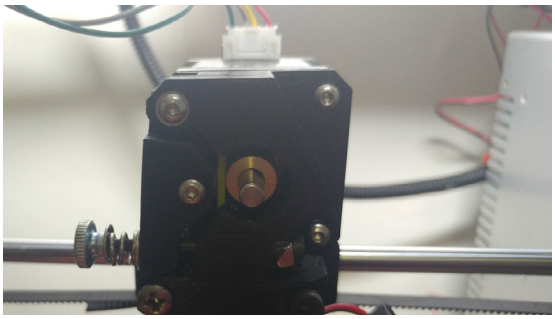
Filament Snaps



Check Chapters:

- Filament Snaps (p. 42)
- Settings Issues (p. 146)
- Materials and their Settings (p. 180)

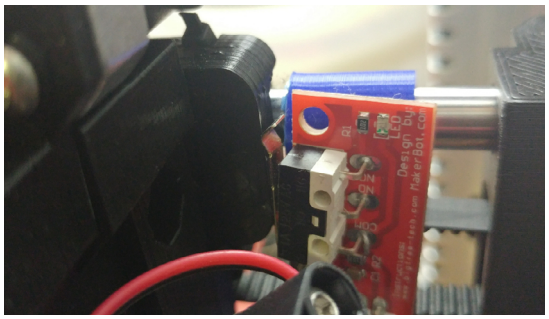
Running Out Of FIlament



Check Chapters:

- Running Out of Filament (p. 143)

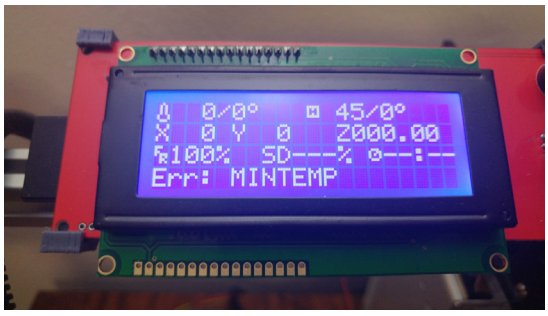
Endstop Not Engaging



Check Chapters:

- Not Finding Home (p. 99)

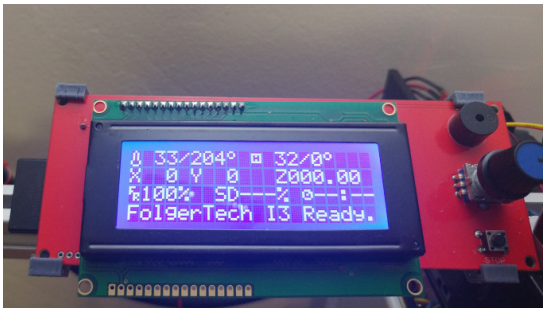
Error: MINTEMP



Check Chapters:

- Hotend Not Reading Correct Temperature (p. 57)

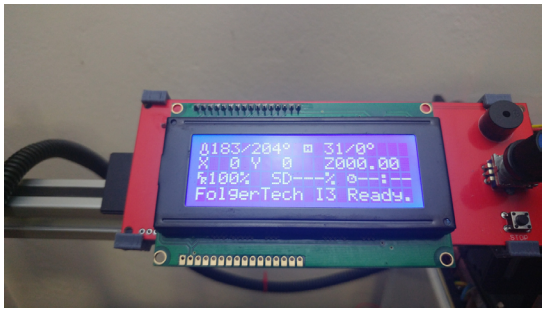
Hotend Not Heating



Check Chapters:

- Hotend Not Heating (p. 53)

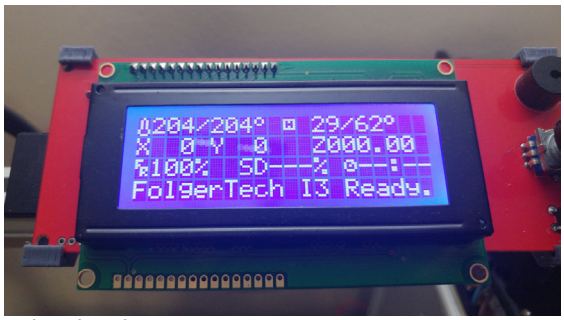
Hotend Cannot Reach Set Temp



Check Chapters:

- Hotend Cannot Reach or Maintain Temperature (p. 48)

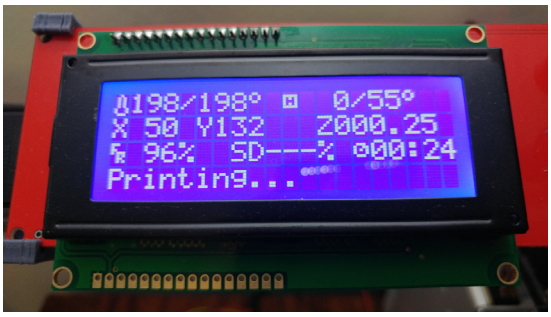
Build Plate Not Heating



Check Chapters:

- Build Plate Not Heating (p. 23)

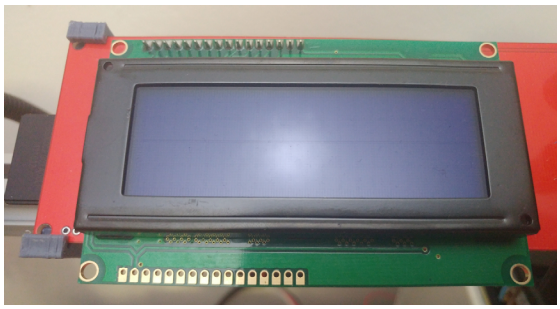
Build Plate Not Reading Temp



Check Chapters:

- Build Plate Not Reading Correct Temperature (p. 27)

LCD Screen Dark



Check Chapters:

- LCD Blank or Dark (p. 73)

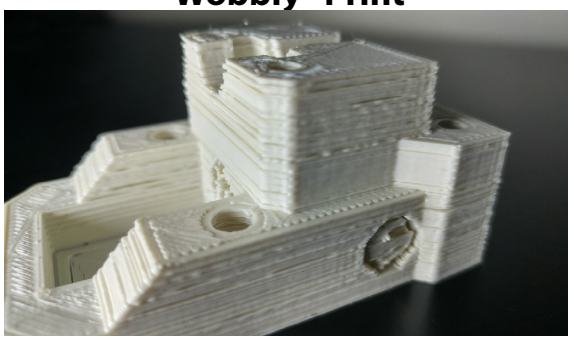
Black Spots On Print



Check Chapters:

- Built up Material on Nozzle (p. 29)
- Materials and their Settings (p. 80)

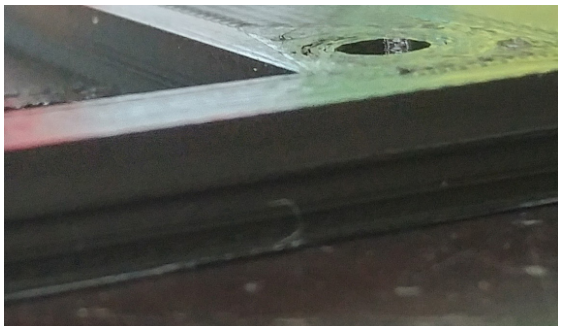
"Wobbly" Print



Check Chapters:

- Z-Axis Wobble (p. 182)

Single Layer Shift



Check Chapters:

- Layer Shifts (p. 65)

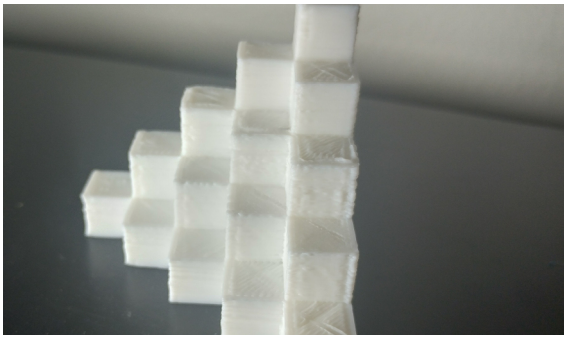
Multiple Layer Shifts



Check Chapters:

- Layer Shifts (p. 65)

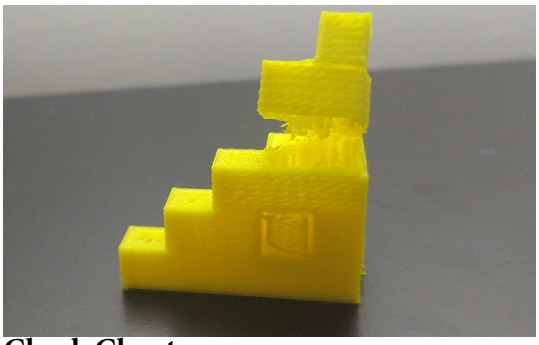
Over Extrusion



Check Chapters:

- Over and Under Extrusion (p. 110)

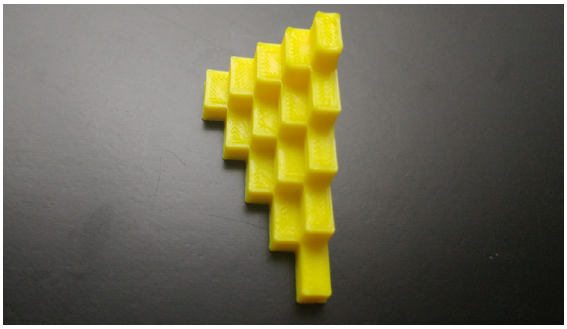
Under Extrusion



Check Chapters:

- Over and Under Extrusion (p. 110)

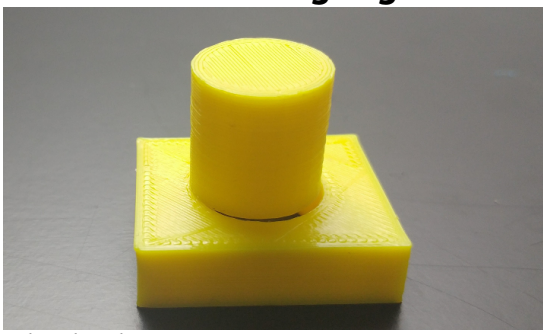
Parts To Wrong Dimensions



Check Chapters:

- Parts Not Mating Together (p. 123)

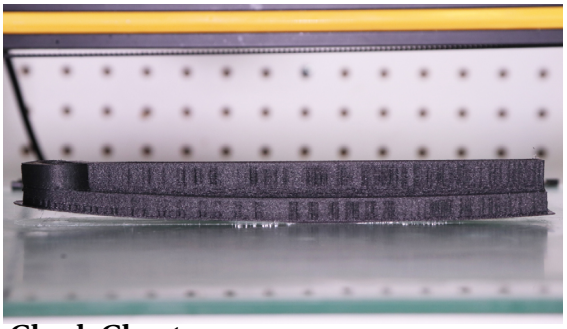
Parts Not Mating Together



Check Chapters:

- Parts Not Mating Together (p. 123)
- Over and Under Extrusion - Over Extrusion (p.110)

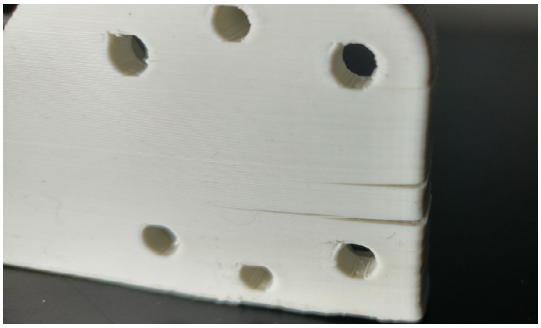
Warping



Check Chapters:

- Warping (p. 177)
- Bed Adhesion (p. 13)

Layer Delamination



Check Chapters:

- Warping (p. 177)
- Over and Under Extrusion (p. 110)

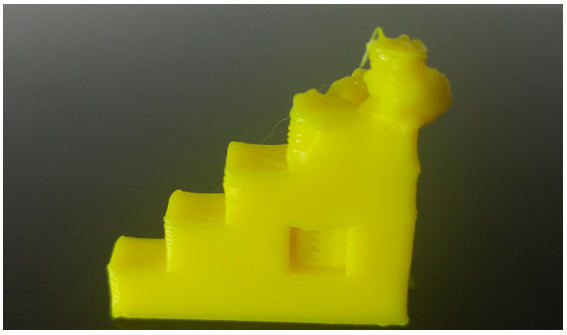
Poor Layer Adhesion



Check Chapters:

- Poor Layer Adhesion (p. 128)
- Over and Under Extrusion - Under Extrusion (p. 110)
- Settings Issues (p. 146)

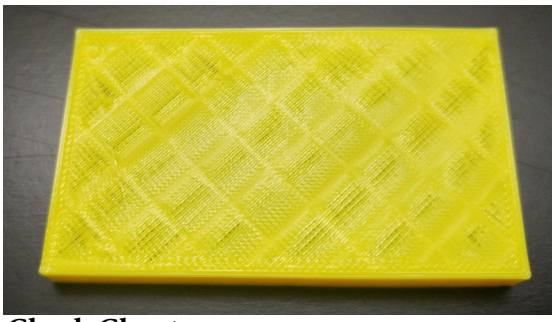
Ugly Top of Print



Check Chapters:

- Settings Issues - Lift Head Cooling (p. 155)

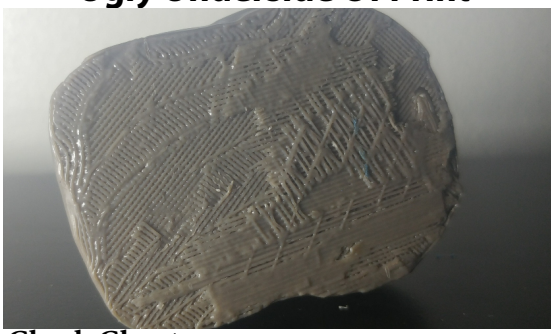
"Pitted" Top Side Of Print



Check Chapters:

- Settings Issues - Shell and Fill Settings (p. 148 p. 150)

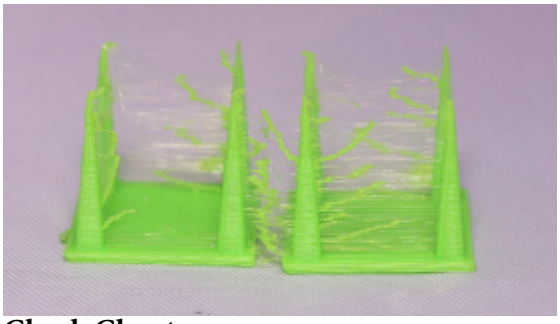
Ugly Underside Of Print



Check Chapters:

- Settings Issues - Support Settings (p. 156)

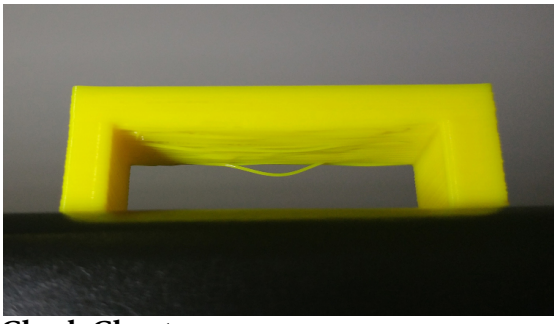
"Hairy" Prints



Check Chapters:

- Settings Issues - Material Retraction (p. 152)
- Materials and their Settings (p. 80)

Droopy/Ugly Undersides



Check Chapters:

- Settings Issues - Support Settings (p. 156)

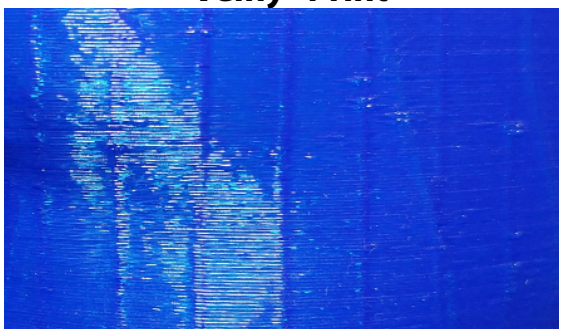
Text Not Legible



Check Chapters:

- Parts Not Mating (p. 123)
- Over and Under Extrusion - Over Extrusion (p. 110)
- Quality Options (p. 140)

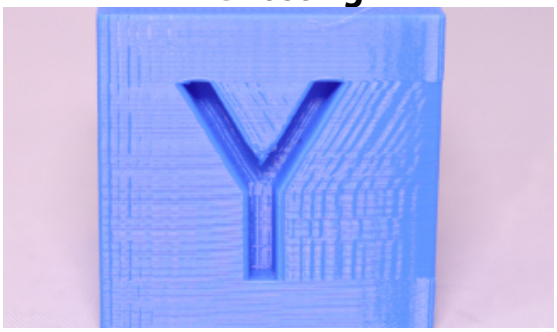
"Veiny" Print



Check Chapters:

- Settings Issues - Infill Overlap (p. 150)
- Ghosting (p. 45)

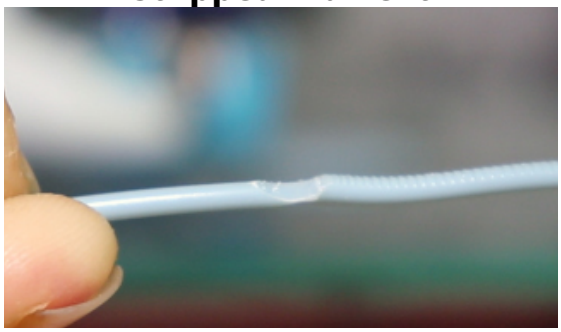
Ghosting



Check Chapters:

- Ghosting (p. 45)

Stripped Filament



Check Chapters:

- Stripped Filament (p. 167)

Bed Adhesion

Having proper bed adhesion is crucial to any successful print. Using different materials, nozzle diameters, and 3d models can lead to the need for different slicer settings and manual calibration for proper bed adhesion. While the issue may be frequent, it can normally be diagnosed early on into the print. The few times you lose bed adhesion mid print, it can lead to big issues, so make sure you take the proper precautions.

These tips are also crucial in avoiding any warping on your parts, as mentioned in that chapter.

Understanding the material you are using

Not only does each material affect your bed adhesion and settings required, each manufacturer can as well. I highly recommend against using low quality filament brands since their tolerances, humidity, and reliability are frequently subpar. I personally use Hatchbox or AIO Robotics for PLA, Fiberlogy, E-Sun and MatterHackers for PETG, 3DXTech and MatterHackers for carbon fiber blends, taulman3D for unique nylons, and NinjaTek for their various flexible materials. There are other reputable manufacturers as well, but I have found I like these companies best out of the ones I have tested. Review the "Materials and their Settings" chapter as well for further information.

You may also be able to switch to a different material with similar mechanical features but with better bed adhesion properties, as gone over further in the "Warping" chapter.

Slow down the speed on your first layer and turn off active cooling fan

Having your first layer extrude properly is needed to making sure that the rest of the print follows suit. Even after 4 years of experience and using higher end FDM machines, I will always watch the first layer to make sure it prints properly. If you just decide to start a print and walk away, you can come back to a very messy or warped print.

You will want to confirm that your first layer is running very slowly compared to the rest of your print (30%-60% print speed or ~15-30mm/s), and that no active cooling fan is on.

This makes sure that the first layer has its best chance of sticking to the bed, which is the most difficult layer for adhesion on any material.

Have a heated build plate

If you want the best adhesion possible, you are going to need a heated build plate. You may be able to get away with printing specific materials without a heated build plate, but even PLA sticks better when the bed is heated to its glass transition temperature.

The glass transition temperature of a material is always lower than the extrusion temperature and relates to when the material starts to become viscous without actually going through a phase transition. When the material becomes slightly viscous, it will be able to stick to the build plate much easier.

Below are the temperatures I set my heated build plate for various materials. Please keep in mind you will need to have your board actively cooled in order to have your bed reach and maintain some of these temperatures.

PLA: 60°C

ABS: 105°C

PETG: 70°-85°C

CFR-ABS: 105°C

CFR-Nylon: 80°C

Cheetah by Ninjatek: 40°C

Ninjaflex: 50°C

PCTPE: 45°C on a glass bed with a coat of PVA

Nylon 910: 45°C on a glass bed with a coat of PVA

Polycarbonate ABS: 120°C bed (and can only print in an enclosed machine)

Clean your bed before applying anything

Fingerprints and residue from previous prints on the plate can prevent the first layer from sticking to the plate properly. Isopropyl alcohol is used to properly clean uncoated glass and PEI beds, while acetone can be used on glass. As mentioned below, when cleaning a PVA solution you should only use soap and water, since that is what it is soluble in.

PEI beds require a clean surface in order to stick properly, and glass will always perform better after being cleaned, so make sure you clean this plate every 5-10 prints.

Having a proper initial Z-height

This is likely the most important step in getting a good first layer. The initial Z-height refers to how far the nozzle is from the build plate. You can easily imagine that if you start a print with your nozzle an inch off of the build plate that the material will not stick.

This is also covered in the “Unlevelled Build Plate” and “Z-Height Calibration”

chapters, but essentially the distance from the nozzle to the build plate needs to be honed in based off the initial layer height. A good starting point should have the nozzle roughly the thickness of a piece of paper in distance from the build plate.

This means that you can home your printer and grab a piece of computer paper. You should be able to slide the piece of paper under the nozzle with a slight amount of drag. If you cannot slide the paper under the nozzle, it is likely too close to the build plate, and if you get no resistance, the nozzle is likely too high.

While this is a go to method, I have actually gotten to the point where it is easiest for me to visually determine the proper layer height. Below are a couple of comparison photos, but essentially if the material is spreading out, the nozzle is too close, and if there is a gap between layer lines, it is too far.



Glass bed

When printing I personally use a glass bed. Glass is far superior to acrylic due to its ability to remain flat and its lack of warping while heated. I prefer to use a ¼" thick glass bed rather than the standard 1/8" glass, though it will add weight to your Y carriage on a Cartesian machine.

There are various methods to get your part to stick to a glass bed, since heating alone will not do the trick.

Hair Spray: This is my preferred method since it seems to work quite well and allows for the bottom of the print to retain a high quality shine. Honing in on the right hair spray is important and I have settled with Aqua Net Unscented Extra Super Hold. Make sure to always proceed with caution though, because hair spray is very flammable.

Apply a thin coat as if you were spray painting your bed. You should do this with your machine off, or with all fans covered as to not have the ambient spray be sucked into your hotend assembly. If possible, remove the glass plate and add the hair spray away from the printer, so that the sticky substance does not get onto the rest of your machine. Be sure to avoid spraying the rods, because they should remain as lubricated as possible.

Allow the hair spray to dry, heat the bed, and your parts should now stick with a lot more ease. Removal of these parts is normally quite easy as well, just wait for the bed to cool to room temperature. If you wait until room temperature, and you start your print the proper distance from the build plate, it will actually pop off without much effort at all.

You will run into issues on large ABS and other high shrinkage materials when it comes to warping. Applying hair spray will work for most circular shaped ABS parts, but any model with corners and a high density will experience warping, unless you switch to a different material or method of bed adhesion.

Blue Painters Tape: I do not use this method any longer but it will definitely work if you are having a lot of issues with your print sticking to the bed. This method is pretty much required if you are printing on a non-heated build plate.

Just grab a roll of blue painter's tape and apply it evenly to your build surface, while it is at room temperature. Apply as flat as possible and remove all bubbles. You may have to adjust your Z-Height if you were previously printing straight onto the build plate, since your nozzle will now drag on the tape during its first layer.

Most materials will stick great to blue painters tape, but removal of these parts can be quite a hassle. Not only will the underside surface not maintain a high quality look, attempting to scrape off the part can come to the point where you rip the tape along with the print, creating quite a lot of post cleanup.

You will still run into difficulty with large ABS parts, but not as much as you would with only hair spray.

Glue Stick: I really do not recommend this method, but if you are extremely frustrated after using the above two methods, you can give this a go. Parts stick very well when you evenly rub a glue stick onto surface of your build plate.

This method requires a lot of cleanup, does not allow for a clean underside, and makes the parts quite hard to remove.

This can also be used for some nylon prints, assuming the glue stick is a PVA base.

ABS Slurry: If you are printing a large, dense, and rectangular ABS part without any enclosures or special build plates on your machine, you will almost certainly need an ABS slurry.

To make an ABS slurry, grab a clean non-chemical reactant bottle, such as a disposable water bottle or glass mason jar (must have a sealed lid) , then toss in bits of ABS. You will want to use the same color ABS of the print you will be fabricating. You can use failed prints, past support material, or just break parts of the filament up right off the spool.

You will want to then add roughly 100mL of acetone per 10g of filament. This is why it is crucial that you are using a container that is not reactant to acetone. Then seal the container tight and wait a few hours until the large chunks of filament are

dissolved, and you are left with a nice slurry.

You will then brush this slurry onto a room temperature build plate. Make sure it is not heated because acetone vaporizes at low temperatures, is flammable, and is not good for your health.

You can now print large ABS parts without much issues. The bottom of these parts will not be clean and you will still experience layer separations if printing a tall piece in an unenclosed printer. You will only get one print from this coat of slurry, so you will then have to clean the build plate and reapply. Cleaning consists of acetone and a decent amount of scrubbing.

Be very careful when using this method with a glass bed. The adhesion to the bed will be extreme and it is not uncommon for glass beds to shatter or chip when attempting to remove a part that is particularly stuck to the surface. Use safety glasses and cut resistant gloves if using this method with a glass plate.

To be honest, due to the extreme difficulty of printing ABS without warping, I have essentially switched to PETG and Carbon Fiber blends when I need a strong mechanical part.

Coat of PVA: As mentioned above, most of the nylon materials you work with will require a coat of PVA being applied to a glass plate. I personally just use Elmer's glue mixed with water.

While I have read mixtures of up to 8 parts water to 1 part PVA, I actually go all the way up to 1 to 1. Get a cup and add some standard Elmer's glue, and then add roughly the equal amount of water. Mix this up thoroughly and then paint it onto the build plate when at room temperature.

This will be very liquidity so you will want to then heat your build plate to 45°C (or whatever temperature your print material calls for), and the water will evaporate slowly. Do not start your print until you have a dry surface. It will take about 5-10 minutes to evaporate entirely.

Your nylon parts should now stick much easier. You can also use this method for PLA and ABS, though I personally do not. In order to clean off this PVA you do not want to use alcohol or acetone, since it is not soluble in those. Just use soapy water to clean PVA off your build plate.

I have had difficulty on a couple of nylon mixtures, and ended up resorting to a glue stick, so keep that in mind as backup.

Third Party Sticky Sheets: There are a few companies that make sheets that are applied to your glass build plate. These advertise high bed adhesion with easy removal. But, from what I have found, most of these products allow your part to stick very well, but do not allow for easy removal.

The majority of these products will only last for around a dozen prints because it is extremely difficult to remove parts without damaging the sheet. I do personally use these though for when I have a lot of prints that have a very small surface area touching the build plate (imagine a print bed of 100 skinny posts). I would normally have to use a brim for these types of prints, but it can literally add hours of cleanup when across hundreds of parts.

Using these sheets allow for me to print these specific parts without any brim or fear that anything will get knocked over.

I have a BuildTak bed sheet on one of my machines, and while it works pretty well, I almost never use it, the only exception being the thin small parts mentioned earlier. Removing parts is very difficult and I have found that glass with hair spray works easiest. Keep in mind that these sheets will likely not work with Nylons.

PEI bed

Some printers come with what is called a PEI bed, or you can purchase one separately. These can be full beds or sheets that you apply, but they must be used on heated build plates only. A PEI bed will essentially create small suction cups when heated - which will help parts to stick, and then become flat when cooled - allowing for very simple removal. The major benefit to using a PEI bed is that, unlike glass, it does not require surface preparation before printing (aside from keeping it clean from normal dust and debris).

You will always want to wait until your part is easy to remove or until the build plate is at room temperature before attempting any removal. If you do not, you will be far more likely to damage this very delicate material. I have broken a few PEI beds in the past, so be careful. You will definitely want to make sure you have the proper initial Z-height, since printing too close can really damage the material.

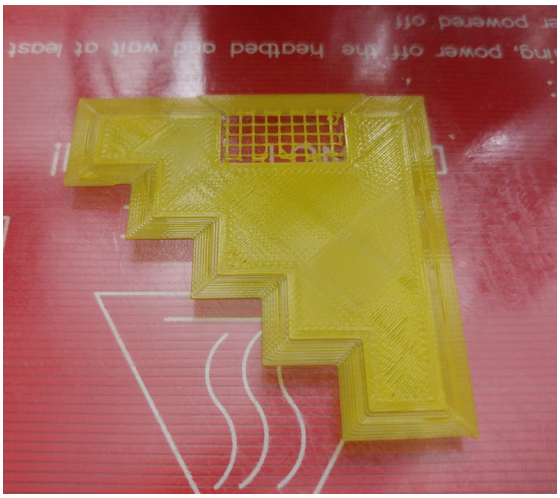
You will also want to periodically clean this bed with alcohol, otherwise you will slowly lose adhesion properties. I suggest cleaning every 5 prints. If you notice parts are not sticking like they once did, cleaning can normally fix this. Do not use acetone when cleaning a PEI bed!

High quality PEI beds are normally pretty expensive, can be easily scratched by the nozzle and will eventually degrade from normal use over a few hundred prints. Lower cost PEI films will bubble up, and eventually will get torn from repeated removal - over the course of a hundred prints you will likely need to replace this part. I personally do not use PEI beds any longer, but some of the best printers come with these stock.

Using a brim

A brim refers to the lines around the print that act as an anchor. This will be needed for all high shrinkage materials such as ABS, and for most parts that have a very small surface area touching the build plate.

You can add a brim in your slicer settings, as explained further in the "Settings Issues" chapter. As you will read there, I prefer to only print a brim on the outside of a part. I do this because a standard brim will add anchor lines to every section of the print touching the build plate, including small holes. This adds for a lot of cleanup, and most prints only require this anchor to be applied to the outside perimeter.



How thick the brim is will be based off of the nozzle diameter. A brim of 15 lines will be twice as thick with a 0.8mm nozzle vs a 0.4mm nozzle. For most parts requiring a brim I will use anywhere from 10-30 lines. Anything more than this is unnecessary.

If this brim doesn't help with very small parts being knocked over, you can try using a raft, as explained further in the "Settings Issues" chapter. You can also review the "Parts Being Knocked Over" chapter for further information.

Initial layer thickness

You can also increase the thickness of the first layer of the print in order to help bed adhesion. This is not preferred because it will tweak the tolerances of the bottom of your print by the amount you increased. If you have pieces that must mate together, you should not proceed with this method.

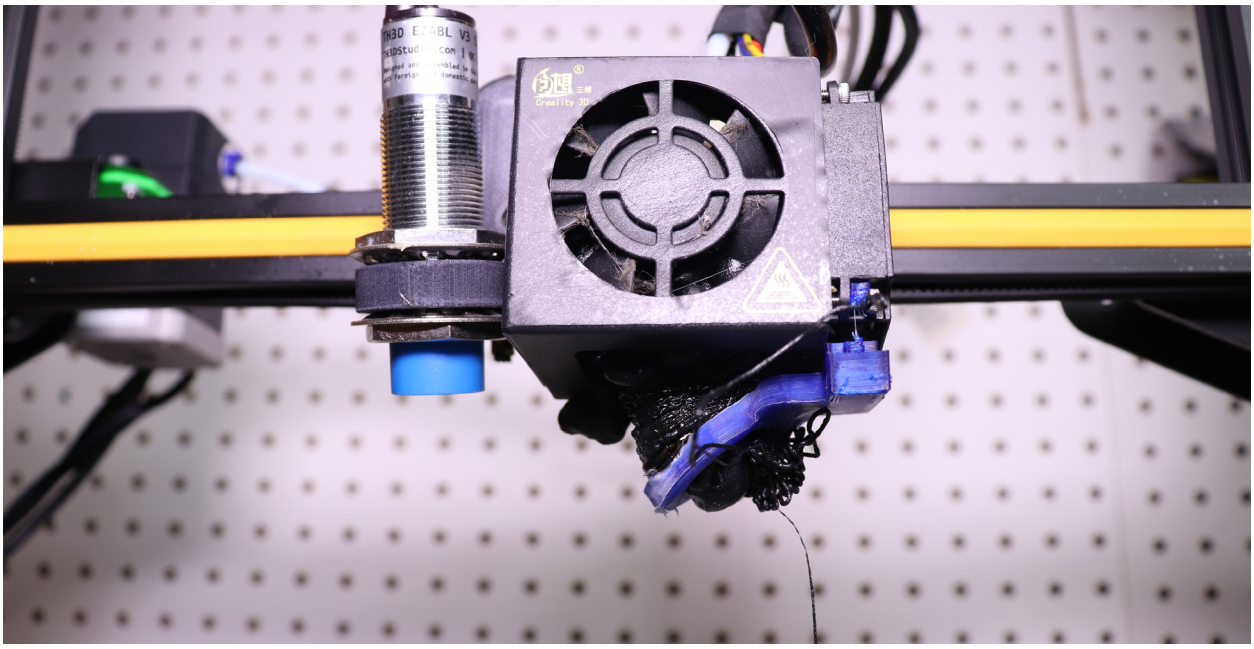
If you are OK with the bottom layer being slightly thicker than the rest of your print, this method definitely works to help anchor your part and increase bed adhesion.

Initial Layer Height

Rather than tweaking the initial layer thickness, you can tweak the initial layer height to help the first layer lay down properly. If you are printing at 0.1mm layer heights, it is very difficult to get the first layer to stick. The lower the layer height, the more precise your Z-height needs to be. If your bed is not perfectly flat, (which is extremely common, particularly on larger printers) then you may need to use this parameter to get any successful prints to adhere to the bed.

I have standardized to having the first layer be 0.25mm – 0.3mm on a 0.4mm nozzle, so that I have a lot more leeway on the initial Z-height. This means my first layer will be 0.25mm-0.3mm, while the rest will be lower for a higher quality print.

What to do when this happens

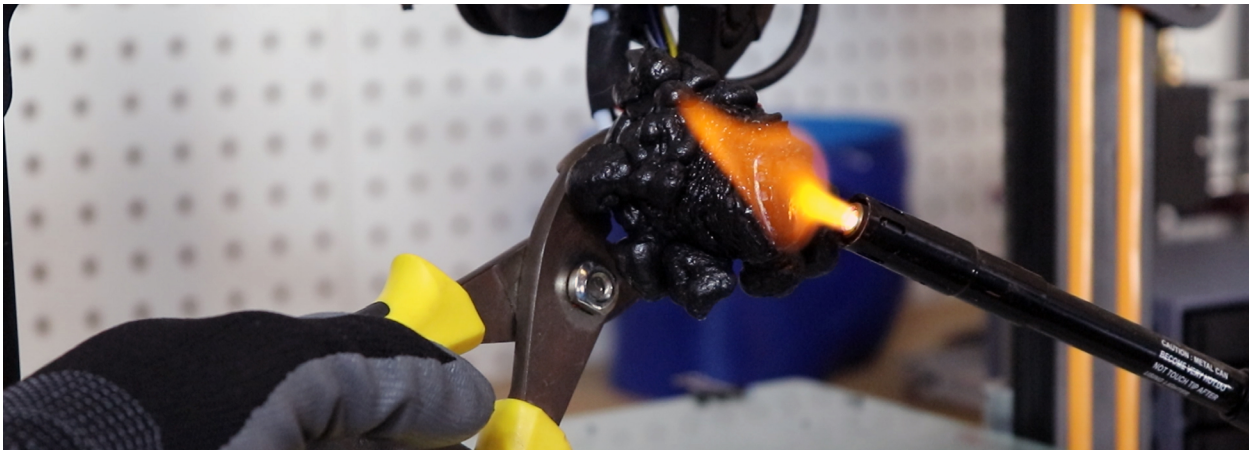


What you see above is the result of an overnight print that lost its bed adhesion a couple of hours in. The ways to prevent this from happening are all of the methods mentioned earlier in this chapter, but once it happens, you have quite the cleanup on your hands.

If your issue is not quite as bad as mine, you can just heat the nozzle to a bit higher than extrusion temperatures, and clean up using a set of pliers. If it is as bad as mine, you may end up needing to purchase some new parts.

You will need to disassemble all of the parts affected. If you do not want to spend time torching material, you can just purchase a new hotend, thermistor, and heater and assemble them in.

To salvage as many parts as possible, you will need to get to a well-ventilated area with a mask and torch. Make sure your affected areas are disconnected from the printer, or that the machine is off and disconnected.



You need to clearly take the proper precautions here, but there isn't really an easier way to get this stuff off when it's this far gone. Make sure you avoid ALL wires, unless you just plan on replacing them. Essentially you just need to torch for a couple of seconds, and then use your pliers to dig in.



After digging in for a few passes, you should be able to eventually get the entire chunk off at once. If you are able to heat your hotend, make sure it is hot for easiest removal (which is obviously not possible if you already cut them for replacement).

It ends up that I needed to use a new heater block, heater, and thermistor in the example above. Even after taking the time to remove the material from the heater block, I was unable to remove the heater and thermistor because the screws were stuck in place. I could have spent another couple of hours torching until everything could be removed, but I figured it was not worth the time.

You can purchase a replacement heater block, heater, and thermistor on Amazon or MatterHackers, and you will then need to re-connect it all. You can just cut the wires and then solder the new set. Please check the "Important Accessories and Replacements" chapter of this book for ways to connect wires.

Reprint any parts you may need and be careful reassembling. It ends up I also needed a new barrel cooling fan, so make sure you purchase all replacement parts required. This need for new parts is why it is really important to get your bed adhesion right before walking away from it for hours at a time.

It is very possible you have a clogged nozzle or barrel after this, so if you do – follow the steps in that chapter.

I also have a video on this exact issue where I go into further detail – titled "Repairing a Gunked Up Hotend" on my 3D Print General YouTube channel.

Summary of Fixes and Precautions:

- Know the material you are using (what are the print settings and limitations, including shrinkage rate)
- Heat your build plate to either the glass transition temperature of the

material you are using, or to a specific temperature suggested by the filament manufacturer

- Frequently clean your bed before applying any extra adhesion to remove fingerprints and residue from previous prints.
- Use either a ¼" glass build plate with a coat of hair spray, or a PEI bed without any extra adhesive substance
- Hone in the proper initial Z-height, since the first layer requires the nozzle be a proper distance from the build plate
- Create an ABS Slurry if you are printing a large ABS part in a non-enclosed printer on a glass bed
- Slow down the speed and turn off active cooling for your first layer
- Print with a brim if the corners are curling, parts are being knocked over, or if you are experiencing warping
- Increase the initial layer thickness if having lots of problems with bed adhesion
- Increase the initial layer height to max out your nozzle diameter (75% the nozzle diameter) so that the tolerance on your initial Z-height is a lot easier to hone in.

Build Plate Not Heating

As with your hotend not heating, this failure is very easy to diagnose since your build plate will not heat up, or will turn off instead of maintaining a temperature. This issue comes up more frequently than your extruder not heating because more power is required to heat your build plate due to its size. This is particularly true for Cartesian printers which require the bed to have high acceleration moves in the X and/or Y axis which can accelerate wear on the wiring.

After you confirm you are using the correct volt/amp for your heater/board combination, you can move forward to checking the following issues.

Many of these instructions are similar to the "Hotend Not Heating" chapter in this book.

Heater malfunctioning

The heater for your build plate is normally either a thick material stuck to the underside of your bed, or is its own separate surface. This heater will have thick gauged wire soldered onto it, which will transfer the power to provide heat.

The easiest diagnosis for when your heater is malfunctioning is to see if this connection has been frayed or destroyed. Frequent usage of your machine or other unforeseen problems can cause this cord to be pulled and tugged from its connection on your heater. Make sure your power is turned off and your machine is unplugged, then take your build plate off so you can easily see the underside of the heater. If you notice that a wire has been pulled from the bed, or is not connected strongly, you will have to solder it back on properly, or purchase a replacement connector.

Most heaters are very simple resistive elements, so it is pretty rare to find that the heater itself is not working. It is recommended to proceed to the next steps before purchasing a new one.

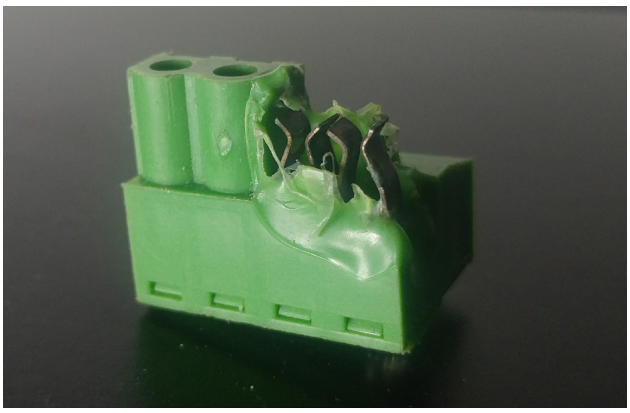
DO NOT REPLACE A HEATER CORD WITH A NORMAL SMALL GAUGED WIRE (thinner than 14 gauge) YOU WILL BE SUSCEPTIBLE TO A FIRE AND FURTHER BURNT OUT WIRES. KNOW THE AMPERAGE OF YOUR HEATER AND SIZE YOUR CORDS ACCORDINGLY.

Burnt out connectors or wires

If the wires are connected to your build plate properly, you will want to check the cords for any disconnections or burnt out areas. When a build plate goes out, it is common for this to happen. When a connector or wire is burnt out you will not get any heating out of your build plate, unlike what I describe in the next section. This is something that has occurred on a few of the machines I have worked with.

If you find any burnt areas, you will have to cut that section off and solder the wires back together – making sure you get continuity on your multimeter after. Make sure to have your machine off and unplugged, and use the proper shrink wrap or correct gauge solder seal connectors. Absolutely no metal can be exposed after this process. Then confirm that the build plate can reach its farthest point from the board, because you just made the wire shorter than it was.

If your newly soldered wire cannot reach the board at the build plate's farthest point, you will experience layer shifts on large builds. In this case it would be recommended to replace the entire wiring harness or adding a length of wire to the harnesses after cutting out the burnt section. Avoid accumulating multiple solder joints on a single wiring harness as that will lead to increased resistance on the line and lower heating times.



If you see that the connector that is plugged into the board is burnt out, you will need to replace it.

This is extremely common on the power input for RAMPS boards, and can be referred to as a design flaw. The connector that comes with the board is easily overheated and can melt, which is exactly what happened to me on one printer that I built. I had no active cooling fan on the board, so if I didn't notice it early, I could have had a real issue on my hands.

Historically, RepRap boards don't use connectors that are rated high enough. If you are using a RAMPS board, I can't suggest enough that you upgrade the board terminals to a well-made 16A replacement (which can be found at Digikey or Radioshack, as well as on my website 3DPrintGeneral.com).

Remember that the area you are replacing may have some melted plastic, so you will need to clean it out entirely before attempting to reconnect. If replacing the connector does not fix this burnt out area, you may unfortunately need to replace the entire board.

Only after everything is wired properly and there is no exposed metal should you plug everything back on and test to confirm the build plate is heating again.

Board overheating

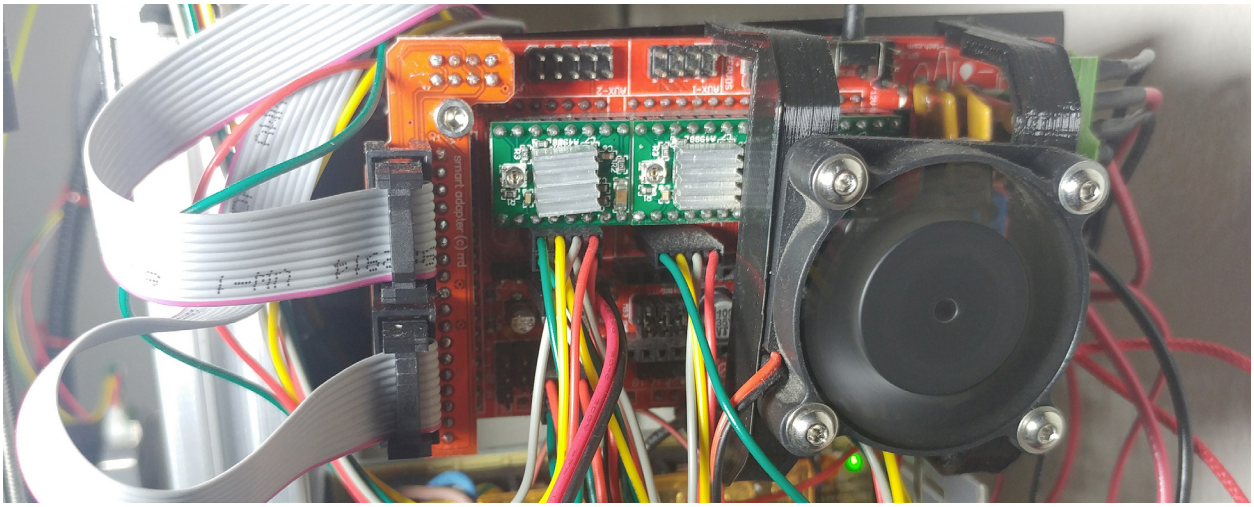
During this particular failure the bed will normally start to heat - and then randomly turn off. This may not even cause a failed print at all - depending on when the heater goes out, how large your model is, what bed adhesions you are using, and what material you are printing in. When your board overheats mid print, you likely will not have a lot of cleanup on your hands -as you would when the

hotend loses heat - but you can experience a warped part.

If you only pay attention to the beginning of a print and then come back when it is completed, you may not even notice that your board overheated. Run a print on a hot build plate (such as printing in ABS) and pay attention to when, and if, the heated build plate starts dropping in temperature.

You can normally tell if this is happening by checking the temperature of your board when the heater cuts out. Be careful though, because if the board is overheating, it will be hot enough to burn your fingers.

I go over a few of these methods elsewhere in this book, but if you are experiencing a board that is overheating, you will need to get 1-3 active fans blowing right onto it. If your board is enclosed with the rest of the machine, the build plate heating will drastically affect the ambient air temperature, causing much more frequent times your board can overheat. This is why you will want your board located outside of any enclosed printers. Small heatsinks will always help affected areas, but from what I can tell, setting up some active fans make the biggest difference.



Most manufacturers come with fans blowing on the board stock, but they can go out or not blow as fast as they should over time. Make sure to regularly check that any active cooling fans are free from excessive dust and debris which can cause them to fail.

If your board is constantly overheating, or you are using a RAMPS board with thousands of hours of printing on it, you will want to replace it entirely. RAMPS boards are normally extremely cheap at under $10, but higher end boards such as a RAMBo board may be closer to $150. These higher end boards will experience less overheating when wired correctly.

Purchase a new heated build plate

As mentioned earlier, it is very rare that you will need to do this. In fact, I have used over 20 machines for over 75,000 hours of printing and have only had to

replace two heated build plates. Since these are normally a bit more expensive than replacing many other parts, confirm that you have tested all of the above methods before purchasing a replacement.

Summary of Fixes and Precautions

- Confirm you are using the correct Volt/Amp for your heated build plate/board
- Check to see if cords have been ripped off or have poor connection to the build plate
- Check for visual damage (burnt out cords/connectors)
- Solder or rewire sections of wire that are disconnected or burnt out – always using the proper gauge
- Confirm you have continuity in the wires from the board to the heated build plate via a multimeter
- Replace any burnt out connectors to the board
- Actively cool your board
- Replace board if overused or burnt out

Build Plate Not Reading Correct Temperature

At times this problem can be pretty difficult to diagnose, and just like when your hotend isn't reading the proper temperature, if left unattended it can lead some serious issues.

These instructions are very similar to the "Hotend Not Reading Correct Temperature" chapter.

Build plate reading 0° or you receive a bed not heating error

There is a thermistor for your heated build plate that works as a thermometer - and just with the hotend's thermistor - it can become damaged or disconnected. These thermistors are not very expensive, but can be difficult to replace on certain setups - so while it is good to have a spare, you will likely want to test everything else first.

If your thermistor has no physical damage that you can see, you will want to check for continuity in the wire. If there is a frayed wire, or a section of a wire you can diagnose has no continuity via a multimeter, you can cut and solder skipping the section – or rewire entirely. If your thermistor is still intact, replacing the wiring will likely fix your issue. When repairing the wiring you will then need to confirm that the build plate can reach its farthest point from the board, because you just made the wire shorter than it was.

Note: if you are in a pinch, soldering the thermistor line will work but it may lead to additional process issues since you are changing the resistive value of that line and therefore the temperature that the board is reading. Your material profiles may require slightly different temperatures after doing this. It is typically recommended to completely replace this section of the wiring if possible.

If your newly soldered wire cannot reach the board at the build plate's furthest point, you will experience layer shifts on large builds.

Confirm thermistor is attached properly

Well-built machines will have their thermistors attached snugly to the build plate, but cheaper machines may only be held onto your heated platform by some Kapton tape. Constant moving and shaking of your build plate can make it so this tape becomes disconnected, and your thermistor will inevitably shake out of its holder.

This will cause your thermistor to read a lower temperature than your build plate

actually is, since it is not actually touching it. This problem can become severe if the thermistor gets far enough from the build plate, which can cause your heated platform to continue to rise in temperature until your board overheats.

You need to confirm that your thermistor is attached properly to the build plate and that there is no chance of it being ripped out mid print.

Having a build plate that is off by 5 degrees typically will not affect the quality of your print, but it can definitely become an issue when the differential gets further than this.

Still experiencing issues

If you are still experiencing issues, or you notice that your bed is continually 10 or more degrees off from where it should be, you will likely want to replace your thermistor and rewire it from the build plate to the board.

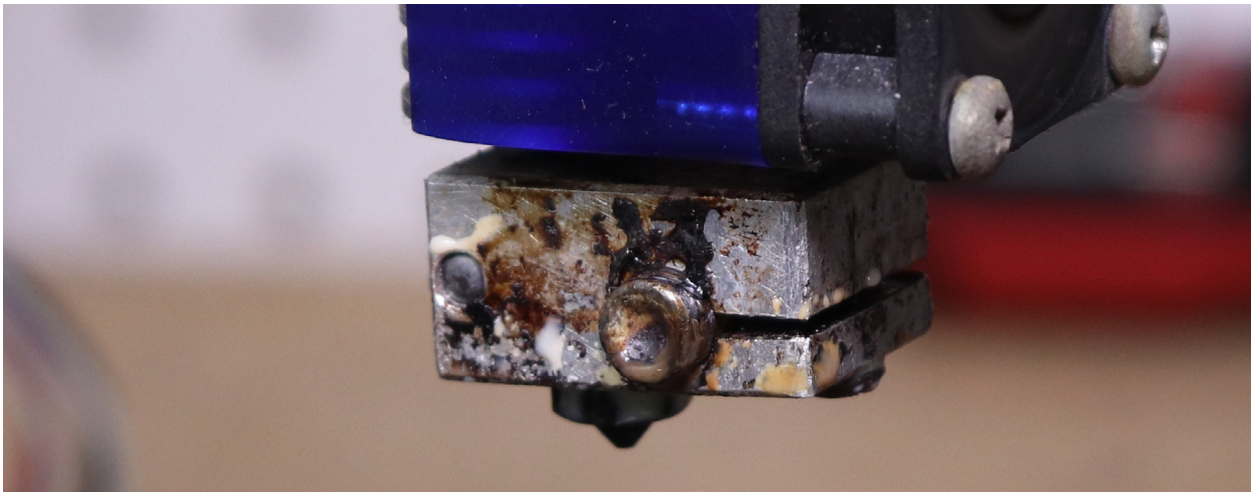
If the problem continues, flash your machine with the original firmware, and then replace the board if necessary.

Summary of Fixes and Precautions

- Make sure the thermistor is actually connected to the build plate and not hovering next to it.
- If thermistor is noticeably damaged, replace it
- Check for breaks or frays in your wire and solder or rewire as needed
- Make sure that if you do rewire, you give enough slack to allow the build plate to move to its furthest point
- Replace and rewire thermistor from the heated platform all the way to the board
- Flash firmware to factory settings
- If still malfunctioning - replace board

Built Up Material in Nozzle

If you do not have a specific hotend for every material you are using, you will likely experience some black dots on your prints from time to time. Even when you are using only one material, this can still be a frequent occurrence.



One of the biggest issues with this failure is that you will often not be able to diagnose it until it happens. This means that a black spot might show up on an important section of your piece 10+ hours in. This is why it is crucial you maintain the cleanliness of your hotend by frequently purging and cold pulling out any residue from your nozzle, along with using a nylon brush.

Ensure your hotend and nozzle is set up properly and tight

Every hotend setup needs to be assembled in a slightly different fashion, but nearly all of them require you to not overtighten. This precaution to not overtighten can lead to gaps between the nozzle and heater block if you assembled everything while at room temperature.

When the heater rises in temperature, the metal expands and can cause your once tight nozzle/heater block to actually have minor gaps. This gap can cause material to ooze out and make its way onto your print. Since this material has been stuck on a hot nozzle before finally being pushed onto your model, it will likely be black and burnt, regardless of the color you are using.

If you notice that your heater block is loose when hot, or that you constantly have to brush off the nozzle or hotend from excess material, you will likely need to tighten these parts.

I always suggest doing the final tightening of your nozzle and heater block when heated to 240°C, using proper gloves and tools. Remember that you have a high chance of burning yourself, so only do this with extreme caution.

You also want to make sure to not overtighten anything. I have broken quite a few heater blocks, nozzles, and heat break barrels due to over-tightening and not being careful. These parts, especially when hot, can easily snap under pressure. When you are doing this make sure to only tighten until you know that the nozzle and heat block are not loose, and will not unscrew during the print.

If you still are experiencing material oozing from the gap between your nozzle and heater block you will likely need to upgrade or replace your nozzle, heater block, or entire hotend. Poorly made or worn out parts will not have tight tolerances, and can lead to these gaps in your threads.

This is why it is important to only purchase name brand parts and not to buy aftermarket knock-offs. For example, if you want an E3D hotend, only purchase from verified dealers such as Filastruder or Matterhackers because there are many counterfeit products on the market.

Purging material

After very long prints, or when switching between filaments, you are going to want to purge out the material that may have oxidized inside of the hotend. There are a couple of ways that you can do this.

If you are using the same material, you can go ahead and heat the hotend to its printing temperature. You then push down the filament for about an inch, and then pull up quickly. Cut off the end and you should be good to go with your next print. Repeat this step as necessary if you are switching to a different color in order to ensure that you do not get any mixed colors during your print.

If you are switching to a material that prints at a higher temperature (such as switching from PLA to ABS) purging is normally simple. You will want to do the same procedure as above, just perhaps do it multiple times in order to confirm that no remaining residue within the hotend. If this hotter material is in a different color, then you will follow the same procedure, it will just be more apparent when you haven't purged enough. Since you are printing at a higher temperature, the majority of the previous material should be removed.

A real issue occurs when you are switching to a material that prints at a lower temperature than your previous filament (such as switching from ABS to PLA). It is likely you will not be able to purge all of the residue material with the above method, because the ABS needs to be purged at a temperature it can properly

extrude at.

If you like to live dangerously, you can purge this material by pushing the colder filament through the hotend when it is set to the higher temperature (such as extruding PLA at 235°C when switching from ABS). Push the material through at a steady pace and then pull it out very quickly, making sure to not let it sit. If you attempt this method, you are going to have a higher chance of a nozzle clog, and you may not get 100% of the residue material.

The proper way to get rid of this material would be to do a Cold Pull as described below, or to purge by using a cleaning filament/nylon material.

Purging with cleaning filament or nylon can be done by heating your extruder to a temperature of around 240°C – 250°C, and then extruding the cleaning filament through as you would with the examples above. Quickly pull the filament out as to not leave any residue, and you will see just how much gunk cleaning filament was able to pull out.

Though less common when using cleaning filament than with other printing materials, you still run the risk of leaving residual material in the hotend that will come out later as a black, burnt spot.

Cold pull

When switching to a material that prints at a lower temperature than your previous filament you will likely want to do a cold pull. Cold pulls are also very beneficial to do as regular maintenance on your machine.

I personally like to perform cold pulls with either a cleaning filament or Nylon mix, but you can perform them with the material you are trying to clear out. My favorite material to do this with is actually Nylon 910 by taulman3D. Not only is that material great to print with, it seems to work even better than cleaning filament I have used in the past for removing the oxidized material in the hotend.

What you will want to do is heat the hotend to the temperature of the material you are using to do the cold pull (250°C for Nylon 910). Push the filament through for an inch, or as much is required for you to no longer see the previous material coming out the nozzle.

Then quickly set your hotend to 150°C - 160°C. You don't want to leave the material sitting in the hotend for a long period of time because it can oxidize itself, or even cause heat creep in your barrel. Once the nozzle cools to this newly set temperature you will want to pull out the filament. This can be difficult if there is a lot of built up residue material, but it normally doesn't require too much effort.

Once you pull you should see excess burnt or colored material on the filament you just cold pulled. Repeat this process until you no longer see this residue.

This is the best way I know of, other than purchasing a new hotend/heater block, to get rid of the excess and oxidized material.

Use a wire brush to clean nozzle and hotend



You should have a brass or steel wire brush on hand to clean the nozzle and hotend periodically. Before a print, especially one where you see material has built up on the heater block/nozzle, you will want to clean it with a wire brush (when heated). When the nozzle is hot you can brush off any excess material that has built up.

Many people suggest using a nylon brush instead of a wire brush since it is far less abrasive. I will use a nylon brush frequently and only periodically use a wire brush on a very dirty hotend. If using a metal wire brush it is better to use one made of copper or brass as opposed to steel to reduce the amount of nozzle abrasion that occurs.

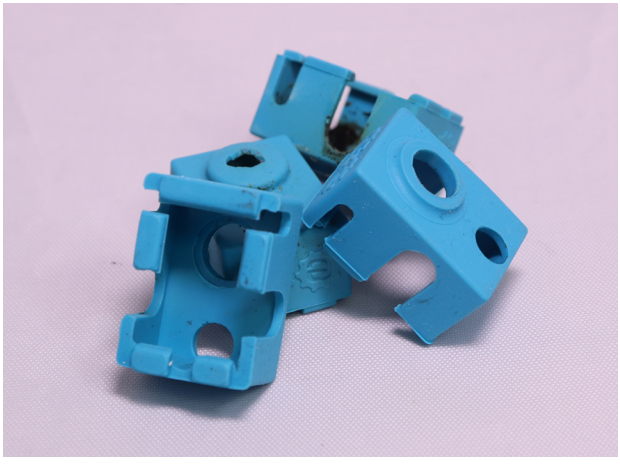
This is crucial in maintaining a clean nozzle and reducing the amount of burnt spots experienced on a print.

Don't leave filament resting in a heated hotend

You shouldn't heat your hotend until you are ready to extrude. If you leave filament in a heated hotend for long periods of time you will increase your chance of nozzle clogs and oxidization of the material. This is a sure fire way to increase the amount of times you will see burnt black spots on your print.

Make sure all of your end G-codes have the script M104 S0, which turns off your hotend after completing a print.

Use a silicon sleeve if available



One great improvement done by E3D for their hotends is adding a simple silicone sleeve for the heater block and nozzle. This blue sleeve fits right over their heater block and does not melt at normal extruding temperatures. Many other manufacturers also offer this feature.

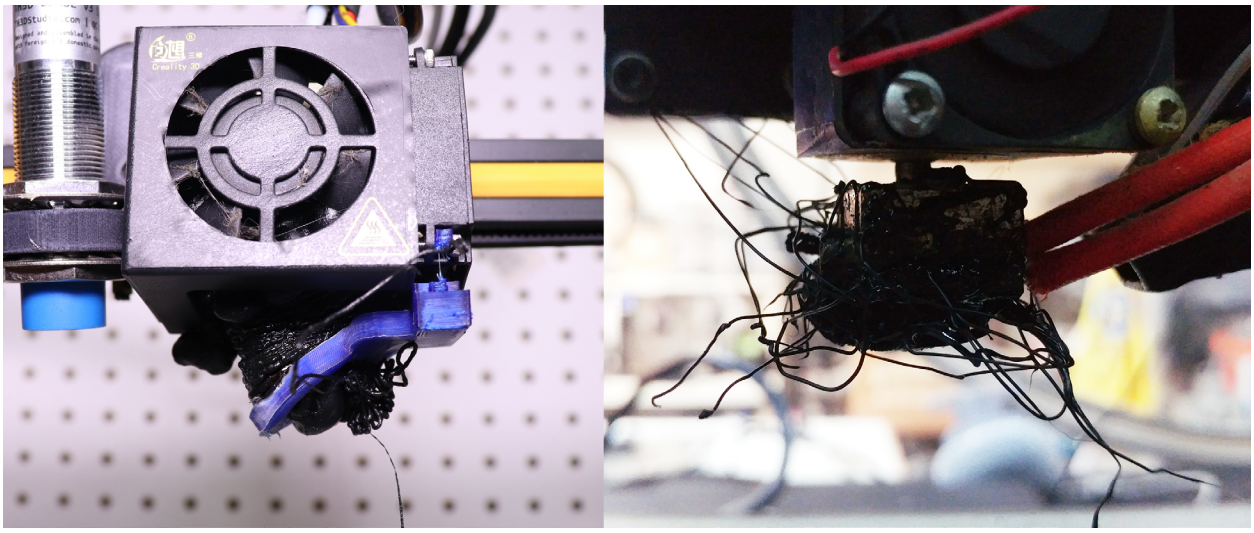
As mentioned in the "Hotend Can't Reach or Maintain a Temperature" chapter, these sleeves help to keep the hotend from sporadically changing

temperatures, but that is not their best feature. The greatest part about these sleeves is the extreme reduction in black burnt spots on your prints. Your heater block and nozzle will remain in shiny clean condition so long as you print using the sleeve.

Black burnt spots are still possible due to the fact you can have oxidized material built up in the heater block, but the problem should be vastly reduced. The only negative I have found from these sleeves is when attempting to print with a large diameter nozzle in a draft resolution. When I print on a 0.6mm nozzle at 0.4mm layer heights I always remove this sleeve. This is because the thick layer heights increase the probability that the sleeve will get in the way of the print and be knocked off.

E3D now makes silicone sleeves with a large hole for the entire nozzle to fit through, which can be used when printing at these larger layer heights.

Excess material built up on nozzle



The two examples shown above have an extreme amount of material stuck to the nozzle. This is caused via letting a part that is not stuck to the build plate continue to print. Once a part is knocked off or does not have the proper bed adhesion, your printer will continue to print, and thus, build up material all over the hotend.

Please refer to the "Bed Adhesion" and "Parts Being Knocked Over" chapters for ways to prevent and clean this.

Summary of Fixes and Precautions

- Make sure you have a well-made hotend and that everything is tightened when heated to 240°C. Take proper precaution to not overtighten.
- Purge old material by pushing down new material an inch and pulling out

quickly. Cut off any old material that is stuck onto the filament and repeat the process until there is no longer any excess residue.

- Use cleaning filament or Nylon 910 since it works best for this process.
- Cold pull by extruding cleaning filament/ Nylon through the hotend at 250°C. Allow the nozzle to then cool to 150°C – 160°C, and pull the filament out. Repeat this process until you no longer get discoloring.
- Don't leave filament in a heated hotend for extended periods of time.
- Use a silicone sleeve to reduce black spots on your print.

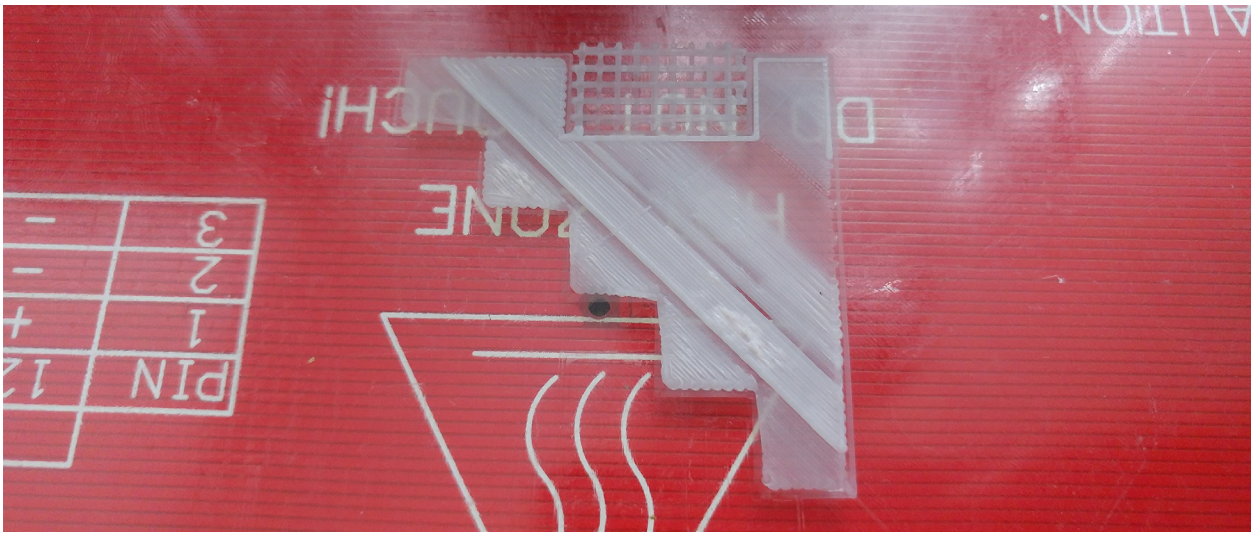
Extruder Stepper Skipping

(Extruder Making a Clicking Noise)

Your extruder stepper motor skipping can happen on any setup, but it seems to be most common on non-geared extruders. A geared extruder allows for a mechanical advantage that will have your motor spin faster, but have much less resistance against it.

When using a non-geared extruder you will be forced to run your prints slower than you could otherwise, so that you do not experience these skips.

Is your nozzle too close to the build plate?



If you hear this skipping on your first layer, it is very likely your nozzle is too close to the build plate. When the nozzle is too close on this first layer, there is no room for the material to extrude. Not only will this cause damage to your nozzle and can even crack your printer bed, it will cause you to hear frustrating clicking noise. This can also lead to stripped filament.

As mentioned in the "Z-Height Calibration" chapter, confirm your nozzle is homing at the proper height from the build plate before starting any print.

Slow your prints down

A common reason your extruder might be making that skipping noise is that you are running your prints too fast. Your nozzle can only push out so much filament depending on its diameter. So just as with bottlenecking in traffic, you will experience stoppage if you try to push too fast.

This can result in grinding of your filament or extruder stepper skipping. The

general rule of thumb is to not print faster than 100x the nozzle diameter. So if you are using a 0.4mm nozzle, you should limit your print speeds to 40mm/s, and adjust according to your performance. This may be slow to some experienced people in the industry, but is the rough estimate I use for printing on a non-geared direct drive extruder.

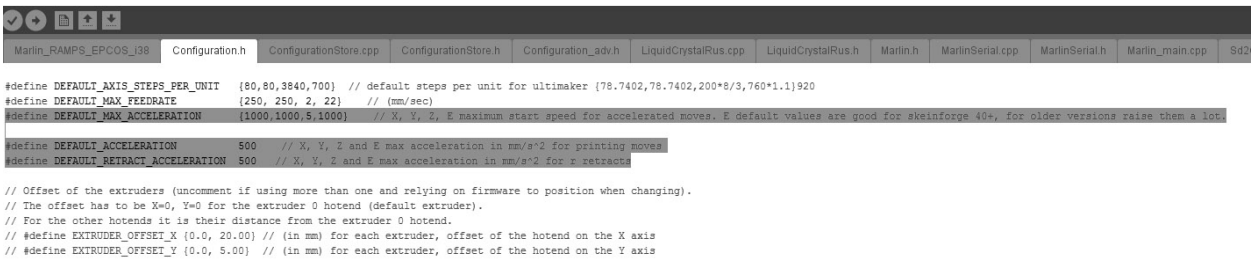
You can test this out mid print if you have a LCD screen on your machine. Most LCD setups are designed so that when you turn the nob mid print, it will change the feed rate (speed). If you hear clicking and would like to see if reducing the speed can fix the issue, turn this knob counter-clockwise. Go to 90% and lower to see if the skipping is decreasing.

If you still are seeing this problem, you may want to check there is not too much moisture in your filament, as covered in the "Stripped Filament" chapter.

Reduce acceleration

The acceleration of your machine is something that is set in the firmware, but can also be tweaked with certain slicing software - including the new Cura. You can tweak this mid-print tough if you have an LCD display. Under Control – Motion you should find the option for Acceleration, depending on how your board was flashed. You likely do not want this number over 1000, and for some machines you won't want this over 500. Manually reduce this number and see if it helps.

If you do not have an LCD screen, or you would like to permanently change this number, you can find the acceleration in your printers firmware in Marlin, under the configuration.h tab.



It is easiest to tweak this in the slicing software, as covered in the "Settings Issues" chapter.

Nozzle Clog

If you have a clog in your barrel or hotend, then you may come across this issue. Your extruder will be trying to push filament down but will click when it is unable to do so. Make sure you check the "Nozzle Clog" chapter if you are experiencing this.

Increase the extrusion temperature

Before attempting this, make sure that your issue isn't being caused by heat creep

(refer to the "Nozzle Clogs" chapter of this book). If you are experiencing a clog in your barrel due to heat creep, increasing your extrusion temperature will only make the problem worse.

If you are not experiencing any heat creep and the barrel of your machine remains close to room temperature, you can try increasing the extrusion temperature a bit to decrease the chances of your stepper motor skipping. You will normally not want to go outside the recommended print temperatures, but there has been a few times I have had to do this to print properly.

This increase in temperature, so long as you are still within the materials accepted extrusion temperature range, will allow more filament to feed through the nozzle at a faster rate.

Loosen the tension on your idler

Most extruder setups have an idler that allows for you to adjust tension – pinching your filament against the hobbed gear or bolt. This tension is necessary to prevent filament grinding and to make sure the proper amount of material is being pushed through the extruder.

While a decent amount of tension is required, you can of course go too far and have this idler be too tight. When too tight you can actually flatten the filament, making it too wide to feed. When material is too fat to feed you will experience similar issues as you would with Heat Creep in the "Nozzle Clogs" chapter, or you may experience stripped filament, but it can also result in the skipping of your extruder motor.

Pinching too tight on a motor that does not have a lot of torque can also cause skipping at the point of contact. While a tight idler allows for good grip on your filament, it is harder for the extruder motor to spin, especially on non-geared setups.

If you notice that the tension on your idler is very tight and you are experiencing skipping of your stepper, try loosening it a bit. Just keep in mind if you go too loose, you can experience under extrusion as the hobbed gear or bolt will begin to slip on the filament.

Making sure filament path is clear

The first step in making sure your filament path is clear is to check for nozzle clogs and residue in your hotend setup. Any problem you find in the "Nozzle Clogs" chapter of this book can lead to extruder motor skipping, so you will want to make sure the path in your barrel is clear from old material and debris. You can do this by torching out the old plastic in a well-ventilated area as described in that chapter.

Aside from old material and debris, I am also talking about the actual pathway that your filament is traveling after and before being fed through the extruder. If you have a 3D printed carriage that is warped, or one that is not to tolerances, you may

have a pathway that does not allow your filament to pass smoothly through it. Any big turns that are required to get your filament to go down your barrel will add to the difficulty involved with feeding filament. Resistance at the spool or pathway leading to the extruder will also cause problems.

You may need to print parts (or purchase parts) for a new extruder on your machine with tighter tolerances and a clearer path to the hotend. This is yet another reason you only want to buy hotends from reputable manufacturers that have tight tolerances.

Adjust the current to your extruder stepper

If you are running a board with digital current control, when you flash firmware onto your printer there is a section that tells the board how much current is flowing to the steppers. You will essentially want this to the minimum possible that still gets the job done. This is because going too high can cause overheating, the driver to run less efficiently, cause wear on the stepper motor, and results in an increase in noise.

Other boards have a manual current control, including the common RAMPS board. For these you will need a multimeter and a screwdriver. Please keep in mind this is only something I have had to do a couple of times and is very rare that you will need to change.

You first want to have your machine turned off and disconnect the extruder stepper from the board. Clip the negative lead of the meter to one of the ground pins on the board. The best place is normally a negative terminal on the main power connector.

Current limits are determined in the motor and driver data sheet. You will not want to run higher than either the driver continuous current limit, or the motor current rating limit, so it is often good to have a driver that has a higher continuous current rating.

Once you know what current limit you want, you can then go to the current limiting section of your stepper driver data sheet in order to figure out the VREF you will want. Current limit will equal VREF x some number (2.5 for the common Pololu A4988). So if your max current limit is 1AMP, you will have a Vref of roughly 0.4. This would be your target number that your multi meter will read out.

You can then clip the positive lead of your meter to the metal shaft on the screwdriver. Setting your multimeter to 20V DC, you can then touch the potentiometer with the screwdriver head (white circle on stepper driver). You will then see a voltage number on your multimeter. You can turn the screwdriver clockwise to increase the voltage, and counterclockwise to decrease. You will not want to overtighten or screw too fast since there are often not stops on the ends.

If you experience overheating after this process, you will want to reduce this current, but increasing it should help with skipping – just make sure to not go over

the rated limits. I suggest you make sure you have upgraded to a geared extruder and tend to the notes below before going this route.

Clean the teeth on your hobbed gear/bolt

Your filament is constantly being pinched to geared teeth that grip it and push it downward. Over time these teeth can be filled with material, causing them to be smooth rather than to have a grip. You will want to get a very small wire brush or metal pick and clean this out periodically. Having a good grip on your material is essential to avoiding stepper skipping and material slippage.

Please check the "Stripped Filament" chapter for further information if you continually experience debris on the teeth of your extruder.

Too small of a nozzle

As with under extrusion, the smaller the nozzle you use the more likely you will experience stepper motor skipping. I had a near impossible time printing with a 0.25mm nozzle and a non-geared extruder. While I didn't hear clicking (since I was running the print extremely slow), minor motor skips led to a print looking as though it was grossly under extruded. E-steps were right on, yet the print looked as though it was under extruded.

This is due to an increased bottlenecking effect at the point of the nozzle. I was forced to upgrade to a geared extruder to print with these finer nozzle diameters. E3D actually advertises that their lower diameter nozzles require a geared extruder to work.

Upgrade to a geared extruder

As mentioned in the first paragraph in this chapter, a geared extruder, such as a Greg's Wade extruder, will help to prevent the stepper motor from skipping. In fact, a geared extruder is pretty much mandatory for printing flexible materials and for using very fine nozzle diameters.

These gears add a mechanical advantage for increased torque without adding pressure on the stepper motor - though you will need to increase and re-adjust your E-steps after switching (Refer to the "Over and Under Extrusion" chapter in this Book).

Upgrading to a geared Greg's Wade extruder requires a very minimal amount of extra parts. You can likely find the required printed parts for your printer and hotend assembly on Thingiverse. These parts normally include a large herringbone gear, a small herringbone gear that is connected to your stepper, mount for your hotend and stepper, and some minor hardware purchases.

Along with various washers and bolts, you will need three 608ZZ regular skateboard bearings and an 8mm hobbed bolt. Hobbed bolts can be purchased from NewEgg for about $2, or you can spend a bit more from places like Amazon or

specialty 3D printing stores. You can even make one yourself by looking up a "how to" online.

Assembly is very easy and all of the printed parts can be completed in about 24 hours on a standard Prusa I3. You will just need to recalibrate your e-steps after upgrading, but you will want to factor in some basic gear ratios first. A classic Greg's wade with a 39:11 gear ratio has e-steps of 500 or more, unlike 90 where a standard direct drive starts at. If you start checking your e-steps before changing from 90 to 500, you will drastically under extrude on your first attempt.



Personally, I now only use purchased extruders. I have standardized to either an E3D Titan, or my new favorite - a Bondtech BMG extruder. The Bondtech BMG has a dual drive, meaning that both sides pinching the filament have teeth that grab onto the filament, rather than just one pushing against a bearing. This increased grip really helps to have proper extrusion and ease to print with smaller diameter nozzles and flexible filaments. These unfortunately currently cost $100, and can be purchased over at MatterHackers.com

Overheating extruder stepper motor

All stepper motors can get overheated, especially when running fast and in an enclosed machine. It is very smart to purchase some heatsinks and add them to all of your stepper motors. If you are hearing a clicking noise from your extruder, and the motor itself is too hot to touch, you will need to reduce this temperature. A heatsink is the easiest way to do this, and is another reason working with an enclosed machine can add to printing failures when not done properly. You can also wire a fan to blow on this stepper motor if still too hot after adding a heatsink

Upgrade to a more powerful stepper/driver/board/power supply

This isn't needed unless you are still experiencing problems or just want to run your machine faster than the settings I suggest. Of course, if you have a malfunctioning stepper, driver, board, or power supply you will need to replace it - but upgrading these parts will not be needed if you took the steps listed above. That is unless you would like to run your printer at extremely high speeds and print temperatures.

Printing at 270°C extrusion temperatures and speeds above 200x the diameter of the nozzle will require a 24v machine, and possibly an upgraded board/driver/stepper combination.

But you can of course also just order a more powerful board, stepper driver, and stepper so that you can run more current, causing far less skips in the motor. Just make sure you understand and purchase everything for the voltage provided by your power supply.

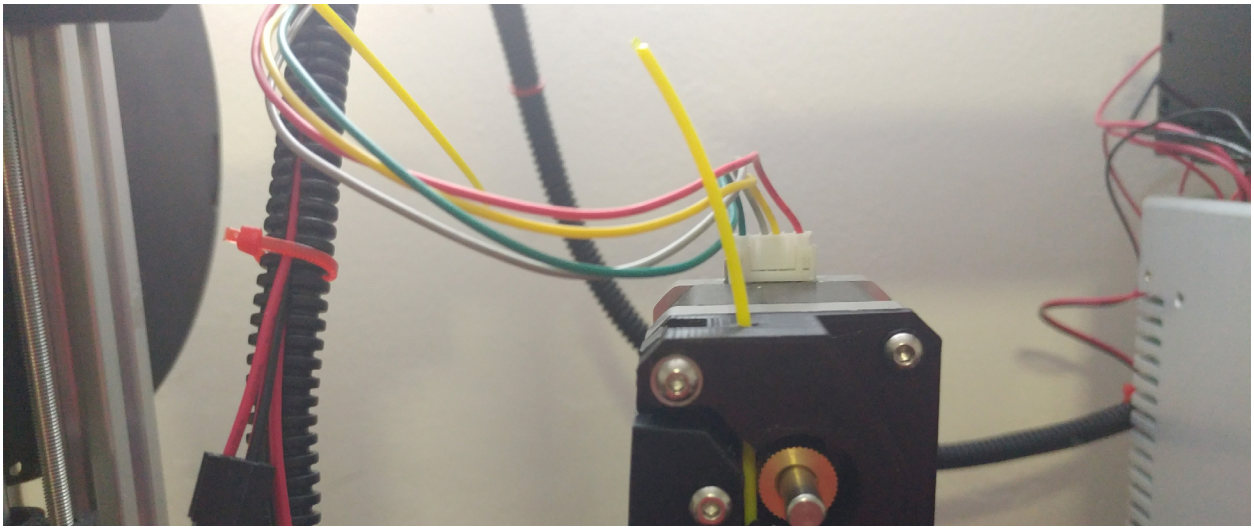
Summary of Fixes and Precautions:

- Confirm your first layer isn't too close to the build plate.
- Slow your prints down (manually change during print with the knob on the LCD screen to see if this fixes the problem).
- Reduce acceleration.
- Print at a slightly higher extrusion temperature.
- Loosen the tension on the extruder idler.
- Have a clear, straight filament path. Reprint or find more suitable extruder carriage on Thingiverse if required.
- Increase the current to your stepper if not at limit and not experiencing overheating.
- Maintain a clean hobbed gear/bolt.
- Increase nozzle diameter if using smaller than a 0.4mm nozzle.
- Upgrade to a geared extruder such as a Greg's Wade Extruder. Designs can be found on Thingiverse and minor parts purchased on Amazon, eBay, or NewEgg.
- Purchase an upgraded geared extruder such as the E3D Titan or the Bondtech BMG.
- Make sure your extruder stepper motor is not overheating and add a heatsink to help prevent the issue.

Filament Snaps

This is another easy to diagnose issue that can be extremely frustrating since it can happen hours into a print.

Understand the material you are using



Certain materials have lower elongation than others, meaning they will snap easier on the spool. Almost all of the carbon fiber blends made by 3DX Tech snap extremely easily on the spool and will require you to use Teflon tubing to guide the filament and help prevent breaks mid print.

Teflon tubing to guide your filament to the extruder will help generally with filament breaks and tangles.

PLA has a much lower elongation than ABS, so you will experience more frequent snaps during the print with it. You can test this yourself since you should be able to easily snap a piece of PLA off of its spool, while ABS will require a tool to cut.

3.00mm filament is also far more likely to experience these issues than 1.75mm filament, as explored in the "Upgrades and Purchasing a New Printer" chapter. 3.00mm PLA near the end of the spool can be impossible to print without snapping occurring, which is definitely a benefit for the thinner diameter.

Review the "Materials and their Settings" chapter for more information.

Properly store your filament

Depending on the climate you live in and material you are using, not properly storing your filament can cause it to not print properly or break very easily. If you do not plan on using a spool of filament for a week or longer, you should store it in

a controlled environment.

You can store opened filament in an enclosed environment with a dehumidifier, or vacuum seal it with a desiccant bag. The dehumidifier should be set to as low as you can (normally around 20%). This is even more important if you live in an area of the world with high humidity.

This is extremely important when using nylon filaments because they can easily absorb moisture within hours of being exposed to a high humidity environment. Low humidity filament containment and feeding systems are also manufactured for in use of highly hygroscopic materials such as nylons

If you do not properly store your filament, you will need to go through the steps required to get the moisture out, or even purchase a new spool.

Change manufacturer

If the manufacturer of your filament does not have high reviews, you may experience frequent breaks. This is when spending a bit more for a name brand can pay for itself. Hatchbox, E-Sun, Matterhackers, ColorFabb, IC3D, taulman3D, AIO Robotics, Fiberlogy, and Proto-Pasta are just a few of the filaments providers that should have no problems straight from the manufacturer.

Without naming any brands, I have had consistent failures and filament breaks from a few specific lower priced companies.

Loosen idler tension

The tension on your extruder idler may be pinching your filament too tight. If this is the case, the grinding of filament may lead to your material snapping entirely.

This would normally lead more to stripped filament, as discussed in that chapter, but it is possible for it to grind until snapping.

Most extruder setups allow for the adjustment of this tension. You want it to be tight enough for there to be plenty of pressure, but not too much that it grinds into the filament or deforms it. If you think this may be too tight, see if loosening the idler fixes your problem.

Check for nozzle clog

You can be experiencing a nozzle clog if your filament is snapping, so if this is the case, be sure to read the "Nozzle Clogs" chapter in this book.

Summary of Fixes and Precautions

- Know the possibility that the specific material you are using might break.
- Switch to a 1.75mm extruder and hotend setup for less frequent filament snaps.

- Properly store your filament.
- Always buy from a well-respected manufacturer for 3D printing filament.
- Replace your spool and try a new manufacturer if needed.
- Check for a nozzle clog and fix if that is the underlying culprit.

Ghosting

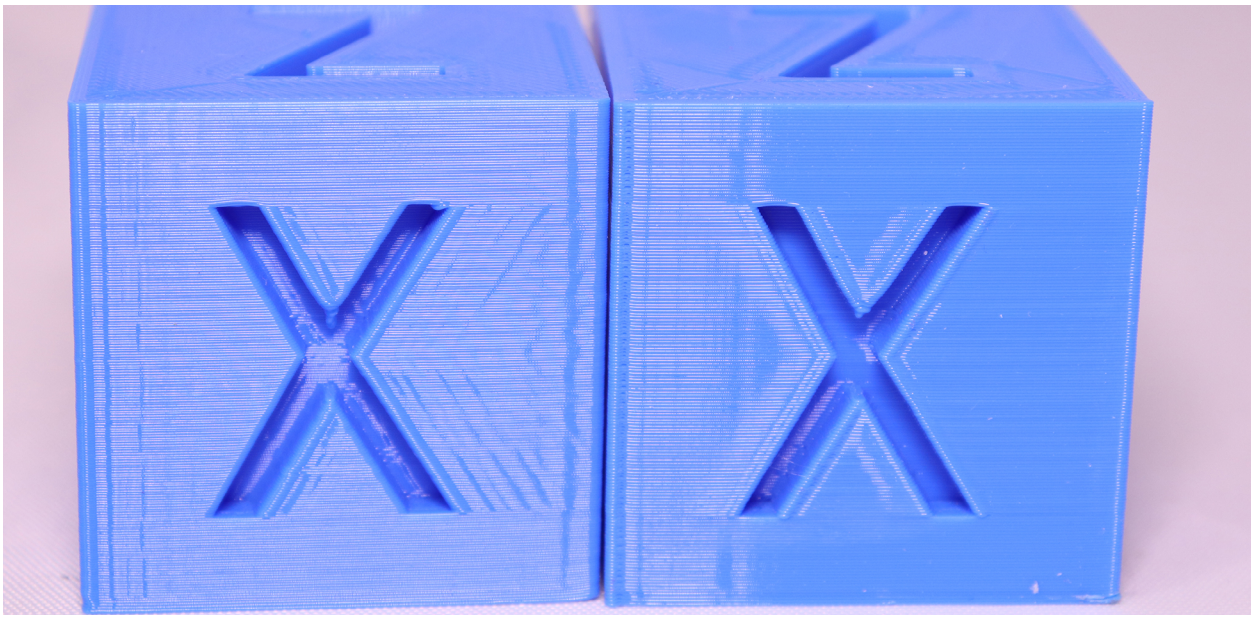
Ghosting in printing essentially refers to an “echo” where details in your print can be seen outside where it should be.

The number one reason this occurs is having your acceleration and jerk settings too high. This is extremely common in 3D printing, especially since many printer manufacturers will set the default value for these numbers too high, since it will allow them to advertise faster printing times.

A part with some minor ghosting is still entirely useable, so printer manufacturers may get away with calling something a successful print that we as makers would be upset with.

Refer to my YouTube video titled “Reduce Ghosting in 3D Printing” for further explanations and examples.

Reducing jerk and acceleration



As you can tell from the photo above, there is a drastic difference in ghosting between the two examples. These parts were printed with every slicer setting being the same, except for Jerk and Acceleration.

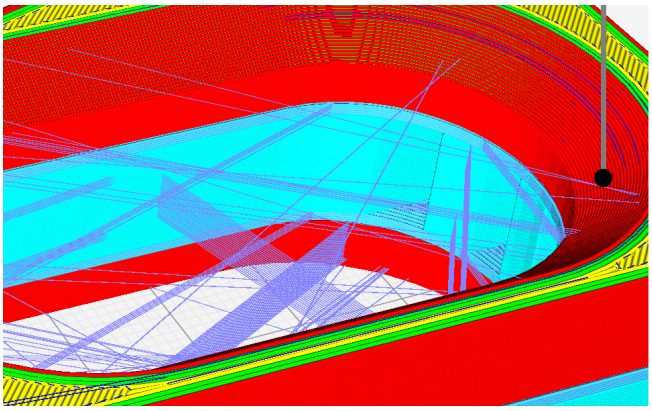
The failed print on the left had accelerations of 3,000 m/s ^{2} and a jerk of 30 mm/s while the print on the right had accelerations of 500 m/s^2 and a jerk of 12 mm/s – all other settings being the same. It has become clear to me that jerk and acceleration are the key factors when it comes to reducing ghosting.

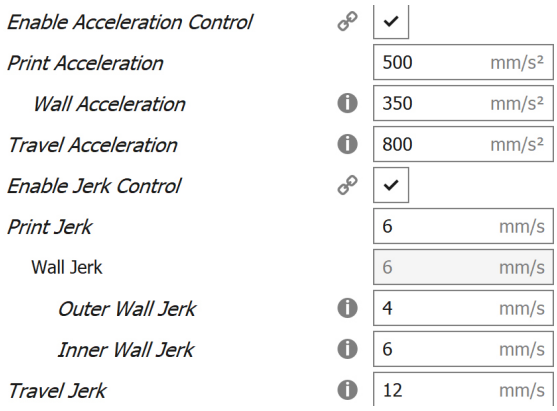
The print on the left took 1 hour 36 minutes and the print on the right took 1 hour

50 minutes, so it is clear you will need to wait longer for your print to finish, but it is definitely worth it to have the quality you expect.

Controlling jerk and acceleration right in Cura

As mentioned elsewhere in this book, the new version of the Cura slicing software is updated frequently, and it now allows for jerk and acceleration control.





This allows you to change these numbers without having flash your firmware. In Cura, this is located under the "Speed" section. If you prefer not to use Cura, then you will have to check within your slicer if this option is available. If it isn't, you will have to manually change the Jerk and Acceleration within Marlin and reflash your firmware. These numbers are right within Configuration H:

Marlin_RAMPS_EPCOS_i38-_gregs | Configuration.h § | ConfigurationStore.cpp | ConfigurationStore.h | Configuration_adv.h | LiquidCrystalRus.cpp | LiquidCr

```
// default settings

#define DEFAULT_AXIS_STEPS_PER_UNIT   {80,80,3840,904.5}  // default steps per unit for ultimaker {78.7402,78.7402,200*8/3,760*1.1}920
#define DEFAULT_MAX_FEEDRATE          {250, 250, 2, 22}    // (mm/sec)
#define DEFAULT_MAX_ACCELERATION      {1000,1000,5,1000}    // X, Y, Z, E maximum start speed for accelerated moves. E default values ar

#define DEFAULT_ACCELERATION          500    // X, Y, Z and E max acceleration in mm/s^2 for printing moves
#define DEFAULT_RETRACT_ACCELERATION  500   // X, Y, Z and E max acceleration in mm/s^2 for r retracts

// Offset of the extruders (uncomment if using more than one and relying on firmware to position when changing).
// The offset has to be X=0, Y=0 for the extruder 0 hotend (default extruder).
// For the other hotends it is their distance from the extruder 0 hotend.
// #define EXTRUDER_OFFSET_X {0.0, 20.00} // (in mm) for each extruder, offset of the hotend on the X axis
// #define EXTRUDER_OFFSET_Y {0.0, 5.00}  // (in mm) for each extruder, offset of the hotend on the Y axis

// The speed change that does not require acceleration (i.e. the software might assume it can be done instanteneously)
#define DEFAULT_XYJERK                8    // (mm/sec)
#define DEFAULT_ZJERK                 0.4     // (mm/sec)
#define DEFAULT_EJERK                 5.0    // (mm/sec)
```

Add small cushions under printer feet

This is a very minor, but easy addition to your printer to help reduce ghosting. Part of the problem with ghosting is the rattling that occurs within the printer without anywhere to disperse it. If you have a very sturdy machine on a sturdy print area, most of the rattling ends up locating within the machine.

An easy fix is to grab some small foam cubes and place them under your printer feet. This will help to disperse vibrations through the machine into the pads,

allowing you to print higher acceleration and jerk speeds with reduced ghosting.

While I have these pads on my printer, I still do not print faster than roughly 800m/s^2 accelerations and 15mm/s jerk.

Summary of Fixes and Precautions

You should refer to my video "Reduce Ghosting in 3D Printing" for a further review of these issues, located on my 3D Print General YouTube channel.

- Reduce your acceleration and jerk via Cura or your preferred slicing software.
- If not available in your slicing software, reduce via marlin and re-flash.
- Add small foam cushions below your printer feet to help disperse vibrations.
- Make sure everything is harnessed tight and securely (extruder, hotend, frame, etc).

Hotend Can't Reach or Maintain Temperature

This problem can arise in a couple different fashions. Your printer may be able to reach its set point but then cannot hold it during the middle of the print, or your hotend may not ever be able to reach the temperature you give it.

Confirm you are using the proper wattage heater

I had a consistent issue on one machine where it had extreme difficulty holding 230°C, and could never reach 250°C. I had done all of the tests mentioned below but the hotend just couldn't make it. It turned out I was using a 30W heater instead of 40W, which the machine was rated for.

Replacing the heater with the proper wattage fixed this problem instantaneously. Wiring for the heaters is typically color coded by wattage so make sure the colors match when changing out your heater.

Check firmware for MAXTEMP

It is also smart to confirm that the maximum temperature for your extruder is set higher than you will be printing in your firmware. Your printer will be restricted from reaching any temperature higher than this.

```
// When temperature exceeds max temp, your heater will be switched off.
// This feature exists to protect your hotend from overheating accidentally, but *NOT* from thermistor short/failure!
// You should use MINTEMP for thermistor short/failure protection.
#define HEATER_0_MAXTEMP 270
#define HEATER_1_MAXTEMP 245
#define HEATER_2_MAXTEMP 245
#define BED_MAXTEMP 200
```

This should be automatically set to a safe number for your machine and hotend, but if it was ever turned to 230, you would not be able to reach any temperatures above that.

PID autotuning

PID refers to a proportional integral derivative controller that is used in 3D printing to control the temperature of the nozzle. When the PID settings are not correct, a fan can drastically change the temperature of the hotend.

An active cooling fan, or even something as minimal as the barrel fan can drop a hotend from 210°C to 160°C if the PID settings are off.

This rapid cooling will make the nozzle too cold for the filament to feed, or may even prevent your print from starting at all.

If you notice that the temperature is fluctuating consistently around the setpoint that is also a sign that the PID settings need to be tuned. Erratic fluctuation is more likely to be a sign of intermittent electrical failure or the temperature sensor getting lodged from its cavity.

You can change your overall PID settings for your machine in your firmware, but I actually suggest getting a specific PID number for the particular material and fan settings you are attempting to print. You can then add these PID numbers into your start G-code.

The first thing you will want to do is to mimic the material you will be printing with (or at least the material you are having difficulty with). If you will be using an active cooling fan, manually turn it on.

While still cold, you will then want to give this command through your printer terminal (Repetier, Octoprint, etc)

M303 E0 Stemperature

So if you are testing out PLA, and you want to print the PLA at 200°C, you would type in:

M303 E0 S200

This will run a PID Autotune. After a decent amount of time your program will feedback values for P I and D:

```
bias: 92 d: 92 min: 196.56 max: 203.75
Ku: 32.59 Tu: 54.92
Clasic PID
Kp: 19.56
Ki: 0.71
Kd: 134.26
PID Autotune finished ! Place the Kp, Ki and Kd constants in the configuration.h
```

As mentioned in that readout, you would open up marlin and go to your configuration.h tab in order to change the PID settings for your entire machine.

```
// If you are using a preconfigured hotend then you can use one of the value sets by uncomment:
// Ultimaker
    #define  DEFAULT_Kp 22.2
    #define  DEFAULT_Ki 1.08
    #define  DEFAULT_Kd 114
```

Some machines also let you change this within the EEPROM of the LCD screen. Just make sure you store settings if you go this route.

As mentioned earlier, I actually suggest getting a separate PID value for each material you plan on using for the most precise results. This is because the numbers you are inputting may be drastically different when printing at 200°C without an active cooling fan vs printing 250°C with an active cooling fan. This is particularly true with lower end machines.

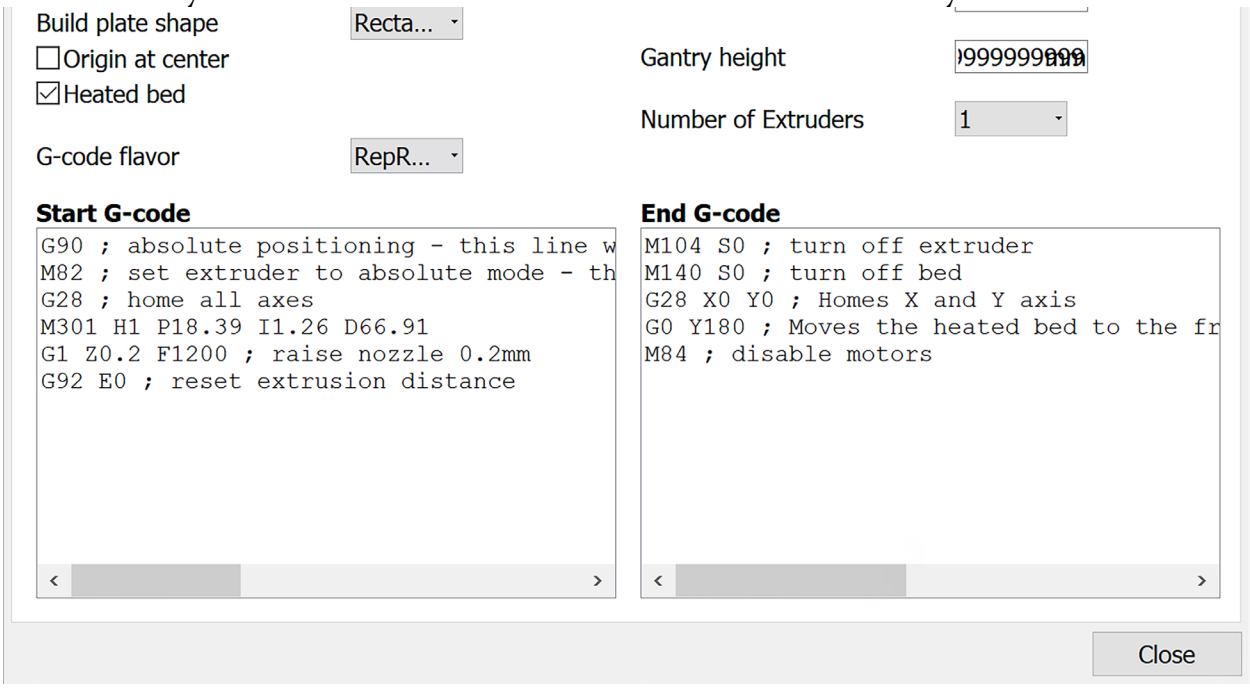
If you would like a specific PID setting for a specific material, run the exact same M303 code mentioned above in the settings desired. For example, if you would like to get a good PID number for PETG, turn on your active cooling fan and run

```
M303 E0 S250
```

Once you are read out a number go to the start G-code section of your slicing program. You will then want to add this line before your G90 input:

```
M301 H1 P##.## I##.## D##.##
```

Here is an example Start G-code for PLA with an active cooling fan I was read out for one of my machines. Remember that each machine will vary:



You would then save this .ini profile so that whenever you are printing with this material in the future, you will not experience dropping or difficulty maintaining the nozzle temperature.

If your PID Autotune has failed, and you have confirmed you are using the correct wattage heater, you can try the next solution

Readjust or reprint active cooling fan

The most common culprit for hotend temperatures dropping mid- print is that your active cooling fan kicks in and it is blowing directly onto your nozzle. With

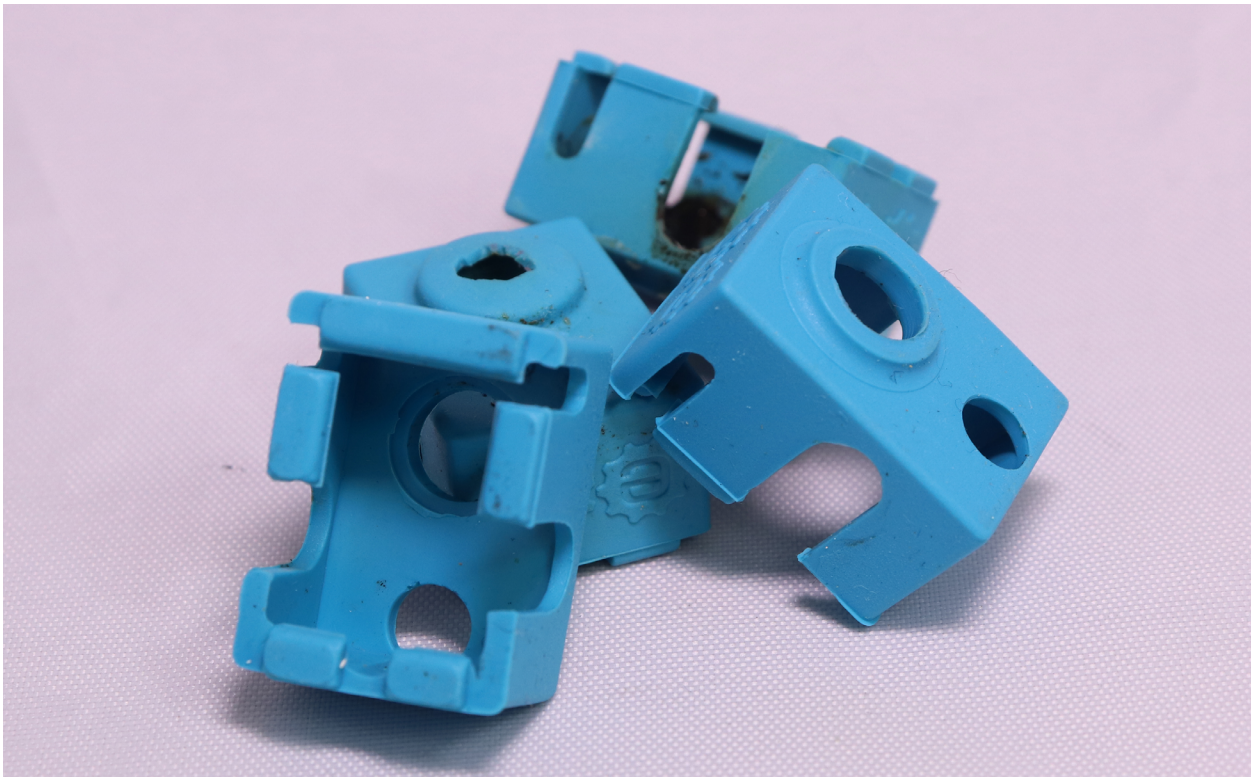
your hotend at room temperature, turn on your active cooling fan. Then place your finger onto your nozzle and hotend and see if you feel a lot of cool air. If you are, this air may be what is dropping your hotend temperature during your print.

You may be able to overcome this differential by running your PID tuning while the active cooling fan is on, as mentioned above, but it doesn't always work. The best active cooling fans are ones that go around your hotend and blow directly downward. You only want the air to be blowing onto your print and not onto your nozzle.

If your active cooling fan is blowing directly onto your hotend, you will need to print a new duct. Search on Thingiverse and elsewhere for a file that will fit your extruder/hotend setup. There are likely dozens already created. Make sure that this duct angles the air downward as to avoid the heater block and nozzle.

These fan ducts can get in the way of your printer finding home, or can even get in the way of raised bed clips, so be careful and confirm your machine can move to all areas of your build plate before starting a print.

Add a silicon sleeve



As mentioned in a couple chapters in this book, you should add a silicone sleeve to your heater block/nozzle, if the manufacturer makes one. Not only does this silicone sleeve prevent burnt black spots on your print, it can actually help insulate the hotend. Any air blowing around the heater block will not have as much effect on cooling a nozzle that has a silicone sleeve around it.

If you have confirmed you are using the proper wattage heater, that your machine can reach the temperature you are trying to reach, air is not blowing onto your heater block, you've added a silicone sleeve and your PID Autotuning is still failing, you will likely need to replace and re-flash. Buy a new heater and thermistor, rewire to your board, and flash your machine again. Then attempt to run the PID Autotuning code.

If still failing you may actually require a new board or power supply. There is a problem that exists with the CR-10 boards where it has difficulty holding temperatures of 230 degrees and above. TH3D Studios can fix this easily over at TH3DStudios.com

This is precisely what happened with my CR-10 and Timothy at TH3D helped fixed it for me quickly.

Summary of Fixes and Precautions

- Confirm you are using the proper wattage heater for your machine.
- Check that you're not exceeding the MAXTEMP for your firmware.
- Run the PID Autotune sequence for the temperature you are having difficulty holding, along with turning on any fans that will be running during the print.
- Change the PID in marlin to set the standard for your machine, or add the code to your start G-code for each given profile.
- Make sure your active cooling fan isn't blowing directly onto your heater block and nozzle. If it is, you will need to adjust it or reprint a new fan duct so that it aims directly onto your print.
- Add a silicone sleeve if available.
- If all else fails, replace heater and thermistor. Flash firmware.
- Absolute final solutions would include replacing your board and power supply, but this should not be necessary in the vast majority of times you experience this problem.
- CR-10 printers have this issue on some boards and it can be fixed via TH3DStudios.com

Hotend Not Heating

This failure is not common and easy to diagnose since your extruder will not heat up. If the temperature is reading out properly, then this is an issue with your heater. Many of these instructions are similar to the ones in the "Build Plate Not Heating" chapter in this book.

After you confirm you are using the correct Volt/Amp for your heater/board combination, you can move forward to checking the following issues.

Heater malfunctioning

The heater is the cartridge attached to thick wires (normally red or blue) that is connected to the heat block of your hotend. As with all parts on a printer, this cord can wear out and malfunction. Luckily, replacement heaters are fairly inexpensive.

It is best to have a spare heater available so that you can easily test if it is your heater or your board that is malfunctioning. All you would need to do is to disconnect the heater from the board and connect your replacement heater, and then see if heats up without any issue. If it does, then it may not be worth your time to even figure out what is wrong with your old heater. This is why it is great to have all of your important parts use a connector. This way you do not have to cut wires if you just want to test a part out.

If you do not have a spare heater, you can start by checking the continuity of the wires from the cartridge to the board using a multimeter. This may be difficult for you if you do not have any connectors or exposed wires that you can access near the cartridge, which is why I suggest testing by using a spare. Even if you do find that one of the wires has a break in it, you will likely need a replacement cord, since it needs to be a thick gauged wire. For a 12V 30A heater, you will want at least a 14 gauge wire.

DO NOT REPLACE A HEATER CORD WITH A NORMAL SMALL GAUGED WIRE (thinner than 14 gauge) YOU WILL BE SUSCEPTIBLE TO A FIRE AND FURTHER BURNT OUT WIRES

Burnt out and/or disconnected wires and connectors

Burnt out and/or disconnectedconnectors or wires and connectors

This is far more common if you are experiencing issues with your build plate not heating, but can occur with your extruder heater as well. You should easily be able to see any burnt out connectors on your board with the naked eye, and if you can't, you will be able to once you disconnect the hotend from the board.

When a connector or wire is burnt out you will not get any heating out of your hotend, unlike what I describe in the next section. If there is a wire you can easily

see was burnt out in one section, you can often fix this without replacing the entire wire.

A disconnect may be as simple as noticing that a connector had vibrated loose. When there is a partial or intermittent break in the wiring, typically referred to as fraying, this issue can be much more difficult to diagnose. This should only ever be an issue on older machines with many printing hours. When this does happen, try to replace the entire wiring harness with a new one if available from the manufacturer. When one section of wiring begins to fray the rest typically will follow shortly thereafter.

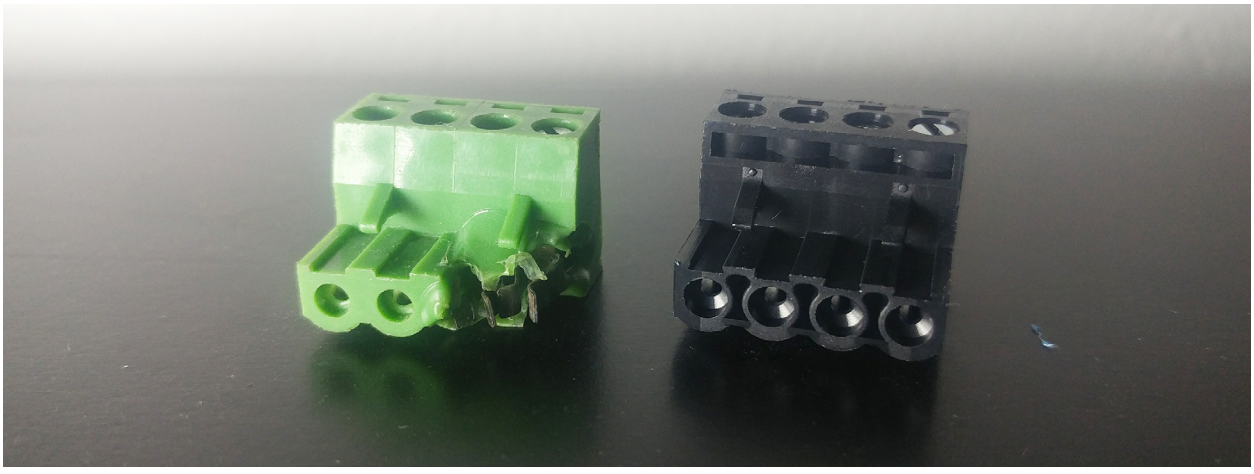
With the printer off and unplugged, cut off the burnt section out of the wire, and then solder the two sections that are not burnt out back together (or use solder seal connectors). Be sure to use the proper shrink wrap because absolutely no metal can be exposed after this process. I mention what I use in the "Important Accessories and Replacements" chapter. Then you will need to confirm that the extruder can reach its farthest point from the board, because you just made the wire shorter than it was.

If your newly soldered wire cannot reach the board at the extruder furthest point, you will experience layer shifts on large builds.

If you see that the connector going from your hotend heater to the board is burnt out, you will need to replace it.

This is extremely common on the power input for RAMPS boards, and is referred to as a design flaw. The connector that comes with the board is easily overheated and can melt, which is exactly what happened to me on a printer that I built. Since I had no active cooling fan, if I didn't notice it early, I could have had a real issue on my hands.

Historically RepRap boards don't use connectors that are rated high enough. If you are using a RAMPS board, I can't suggest enough that you upgrade the board terminals to a well-made 16A replacement (which can be found at Digikey or Radioshack, which I link to over at 3DPrintGeneral.com).



Remember that the area you are replacing may have some melted plastic, so you will need to clean it out entirely before attempting to reconnect. If replacing the connector does not fix this burnt out area, you may actually need to replace the entire board.

Only after everything is wired properly and there is no exposed metal, plug everything back on and test to confirm the hotend is heating again.

Board overheating

Just as with the section above, this is more common with the build plate not heating, but you will normally notice this issue if your heater starts to heat and then cuts out at a certain point.

Remember that this is different than your printer not being able to maintain a specific temperature, which I go over in the "Hotend Can't Reach or Maintain a Temperature" chapter. Rather, this will occur when your hotend is heating up just fine and then entirely cuts out, going all the way back to room temperature.

Not only will this cause a failed print, it will cause a severe nozzle clog if your printer continues to try and cold extrude.

You can normally tell if this is happening by checking the temperature of your board when the heater cuts out. Be careful though, if the board is overheating, it will be hot enough to burn your fingers.

I go over a few of these methods elsewhere in this book, but if you are experiencing a board that is overheating, you will need to get 1-3 active fans blowing right onto it. If your board is enclosed with the rest of the machine, the build plate heating will drastically affect the ambient air temperature, causing much more frequent times your board can overheat. Small heatsinks will always help affected areas, but from what I can tell, setting up some active fans make the biggest difference. Whenever using active cooling fans, make sure to keep them well maintained. Check for dust accumulation and make sure the area around them is free of debris (use a filter/grill if possible to protect the fan blades). Broken fan blades can add a lot of noise to your system and if the cooling fan fails, your board will end up overheating again.

If your board is constantly overheating, or you are using a RAMPS board with thousands of hours of printing on it, you will want to replace it entirely. RAMPS boards are normally extremely cheap at under $10, but higher end boards such as a RAMBo board may be closer to $150. These higher end boards will experience less overheating when wired correctly.

Summary of Fixes and Precautions

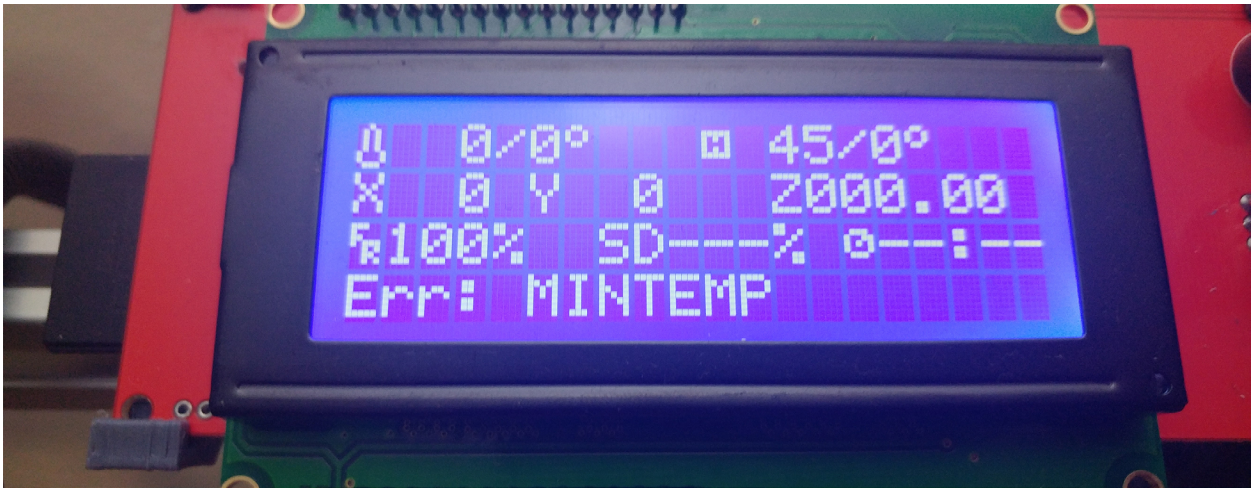
- Confirm you are using the correct Volt/Amp for your Heater/Board.
- Check for visual damage (burnt out cords/connectors).
- Test out a new heater if you do not see anything noticeable.
- Solder wires or rewire any section that is not connected or burnt out - always using the proper gauge.
- Replace any burnt out connector to the board.
- Actively cool your board.
- Replace board if overused or burnt out.

Hotend Not Reading Correct Temperature

This failures refers to your hotend being a different temperature than what it actually is due to issues with your thermistor, and can lead to bigger problems if not properly addressed. This is noticeable when filament feeds at different temperatures than it is supposed to, or if you are receiving a message of "Error Min Temp".

These instructions are similar to the "Build Plate Not Reading Correct Temperature" chapter.

Error Min Temp (or Error Stopped Temp Sensor)



If your thermistor is not connected properly to your printer, it will stop working and read "Error Min Temp". This is most common on fragile, non-cartridge thermistors, but can happen on just about any setup.

If your printer is reading this error you will need a new thermistor the majority of the time. You can order replacement thermistors from your hotend manufacturer. I personally use E3D hotends and you can easily order replacement cartridge thermistors from their website, or from Filastruder.com for US orders.

If your thermistor has no physical damage that you can see, you may be able to fix this without buying a new one. This is when there is an easily recognizable break in the wire, or if you can diagnose that there is no continuity.

If there is a frayed wire or a section of a wire you can diagnose has no continuity, you can cut and solder the wire skipping this section - or rewire entirely. If your thermistor is still intact, replacing the wiring will likely fix your issue. You will then

you will need to confirm that the extruder can reach its farthest point from the board, because you just made the wire shorter than it was.

If your newly soldered wire cannot reach the board at the extruder farthest point, you will experience layer shifts on large builds, or rip the thermistor off the hotend.

Other possibilities include your readout showing a negative temperature. I have had this happened instead of an "Error Min Temp" when working with the CR-10.

Not reading proper temperature

This issue is extremely difficult to diagnose if you are not looking for it and when the differential is not extreme. When drastically different from the actual heat you can have a major problem on your hand if the hotend does not stop heating. If entirely not reading correctly or if your thermistor is hovering an inch next to your hotend, your printer will think that its hotend is not able to reach its set temperature and will just continue heating it. Many machines have settings to turn off your heater if it does not reach its target temperature after a few minutes, but not all do. I have personally seen melted metal from a heater block that never recognized it was actually getting hot, since the thermistor was not connected to it.

Other less severe symptoms may include consistent nozzle clogs or over/under extrusion. If you are only printing PLA you may not ever notice the temperature is reading incorrectly, due to the fact most PLAs have such a wide extrusion temperature range. I have seen some PLA formulations running on E3Ds that need to be extruded between 180-215, any higher than that and you get clogs from heat creep.

If you have a device on hand that can read temperatures back to you up to 200°, then you can obviously use this to test the accuracy of your thermistor. If you don't, and you aren't experiencing issues, a simple way to test this is to try and extrude ABS.

ABS has a smaller temperature range of 220°C - 240°C (ideal 230°C), which will allow you to notice this problem a bit easier than with PLA. MAKE SURE you are using a spool that is reliable and has no moisture in it. Set your hotend to 220°C and if you cannot extrude any ABS, it is likely that your thermistor is reading hotter than it actually is. Do the opposite test if your hotend seems to be heating too high, by attempting to feed ABS at 240°C. If the ABS crystalizes, bubbles, clogs easily, or comes out extremely burnt, your thermistor is likely reading a lower temperature than your hotend actually is.

For this issue you will want to confirm that the thermistor is properly connected to the hotend. If the thermistor is hovering right next to the hotend, and not inside its holder, it will likely read out a temperature lower than the hotend actually is.

If connected properly, and there are no breaks in your wire, you will likely have to replace the thermistor by purchasing a new one, as mentioned above. This is why

having a spare on hand is very beneficial

If none of these fix your problem, there is a small chance that your board itself is malfunctioning and needs replacement. Flash your firmware to factory settings, and if that doesn't work, replace the board

Summary of Fixes and Precautions

- Keep a spare thermistor for your hotend on hand if possible.
- Make sure the thermistor is actually connected to the hotend. If hovering next to it your hotend may not stop heating and can cause serious problems.
- If thermistor is noticeably damaged, replace it.
- Check for breaks of frays in your wire and solder or rewire as needed.
- Make sure that if you do rewire you give enough slack to have the extruder move to its furthest point.
- Check that your hotend is reading the proper temperature by extruding ABS at its low and high temperature ranges.
- Replace and rewire thermistor from the hotend all the way to the board if all else fails.
- Flash firmware to factory settings.
- If still not reading proper temperature, you may need to replace your board.

Important Accessories and Replacements

You should always have a few accessories and replacements on hand so that you do not need to order parts every time something breaks down. Some of these tools I use every single day, while others are needed a lot less frequently.

You can visit 3DPrintGeneral.com for more parts that I suggest and have personally used. This is consistently updated.

Accessories:

I consider all tools that are frequently used on 3D printed parts and the machine itself as a 3D printing accessory. I actually have a video similar to this chapter titled "Mandatory Accessories for a 3D Printer" on my 3D Print General YouTube channel if you wish to see more.

Pliers: I can't think of anything more important to support removal than a set of pliers. You will use these just about every time you print with support.

Razor Blade: Something you need to be extremely careful with, but cleaning up a print will always be easier with some razor blades. I use them just about every time I print with a brim, and it also helps to clean off any "hairy" residue.

Model Cutters: These sharp, thin scissors are a lifesaver when it comes to printing very thin parts. If you have a very thin part of your print that is being held up via support material, just going at it with a set of pliers will likely break that fragile area. Using model cutters will allow you to cut this support material off without damaging your print.

Scraper: A scraper will come with just about any 3D printer you buy, but it is smart to have one that you prefer. I like a very strong metal that comes to a thin, flat front. I have tried out flimsy scrapers that will bend on difficult parts, and thick scrapers that are difficult to get under your print without damaging it. Get a strong metal scraper that tapers at the very front. I like the set by AMX3D if you wanted a suggestion.

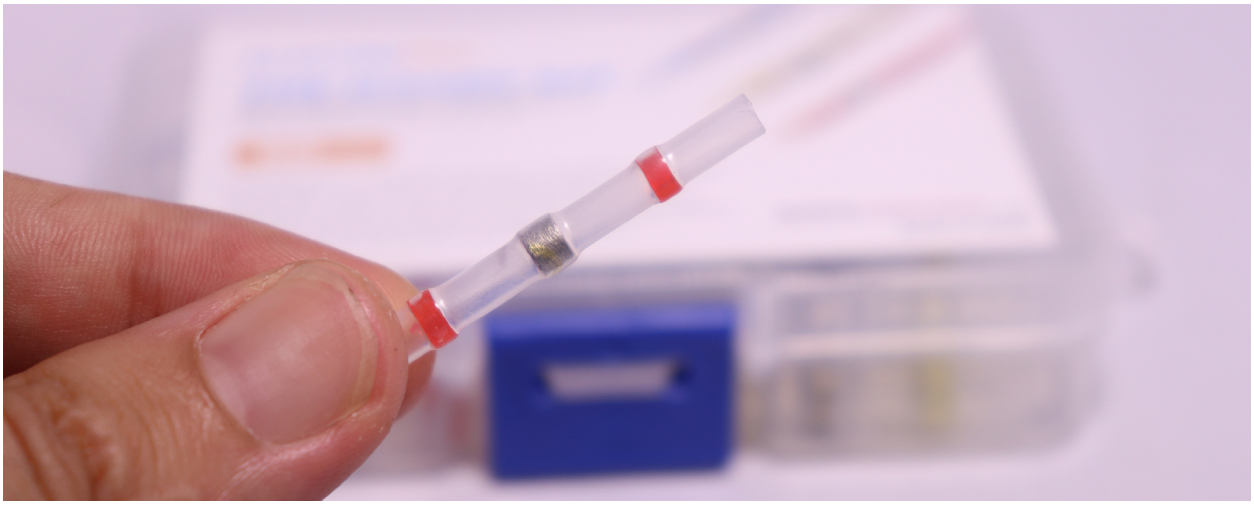
Metric Allen Screwdriver Set: Just about every printer on the market uses a lot of Allen screws. Having an Allen wrench set is smart, but having an Allen screwdriver set is even better. It can be difficult to get to some of your screws via a normal Allen wrench, but is far easier with an Allen screwdriver.

The most common size of screw used an M3, but it is smart to have an M2.5, M3,

M4, and M5 Allen screwdrivers. I don't know of any 3D printers that use standard sizes, almost all use metric. In order to make just about any fixes or repairs to your printer, you will need this screwdrivers/wrench set.

Solder Set or Solder Seal Wire Connectors: Wires become frayed, and you will often need to cut them to attach new parts. There are plenty of times in this book where I go over how to fix parts on your machine, and most of them require to you cut and fix frayed wires.

The most common way to do this is to get wire strippers, solder, a soldering iron, heatshrink, a heat gun, and maybe some tin. There are plenty of tutorials on how to solder wires, which involve stripping the wire, adding some tin to help solder stick, and then using your soldering iron to melt the solder to the two exposed wires. This makes a strong connection which you cover with some heatshrink so that no wires are exposed.

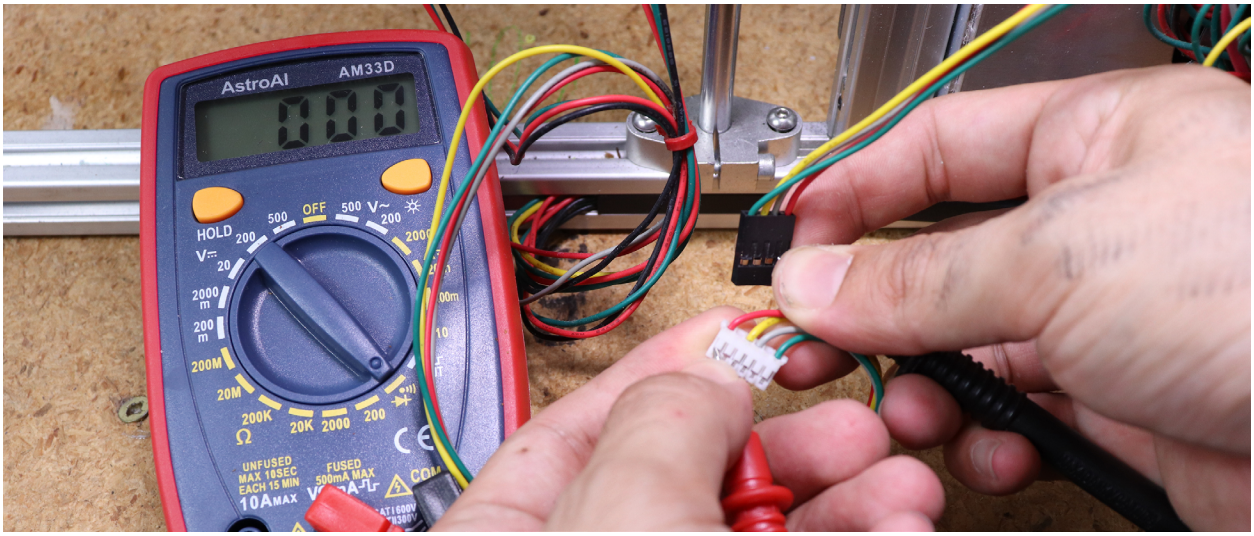


I recently discovered solder seal wire connectors, which I think have become my new favorite way to do this. With solder seal wire connectors you do not need any soldering iron, solder, tin, or heatshrink. These connectors have solder in the middle where you push the two stripped wires to. You then use a heatgun (or even a lighter) to melt the solder and shrink the wrap around the wires. I definitely suggest having a set of these for when you need to connect wires if you do not want to pay for a full soldering kit.

Zip Ties: Obviously zip ties are not ideal for important parts on your machine, but they can definitely help out in a pinch. They can also help to clean up your wires by zip tying them all together. I always have a large box of different size zip ties on hand since I use them quite frequently, and never know when I will actually need them.

Calipers: Calipers help immensely to accurately measure parts and filament. If you are doing any sort of designing, you are definitely going to want calipers. They also help immensely to properly figure out your E-steps.

Multimeter: A multimeter has a ton of features and uses, but I really just use the connection setting for the majority of my 3D printing needs. You can check voltage, amperage, and much more, but just figuring out if a wire is working properly is its best function.



You can easily touch a metal tip to one side of your wire, and the other metal tip to the other side, and you will be told if there is continuity. This can help immensely when trying to diagnose what isn't working on your printer.

Loctite Super Glue Gel: My personal favorite way to combine parts strongly is actually Devcon plastic welding, but Loctite Super Glue gel is the easiest to use. I believe this has become the favorite of most 3D printing enthusiasts to combine plastic parts together. Keep in mind combining different types of materials may require different adhesives, but Loctite Super Glue Gel works wonders on PLA, and it dries extremely fast.

You will have to remove the outer container when running low, since there is still about a quarter of the gel left in the container after you are unable to get any more out.

White Lithium Grease: There are different preferences when it comes to lubing up important rods on your machine, but my favorite is white lithium grease. Periodically lubing threaded and smooth rods on your printer will help to get consistent clean prints.

Wire and Nylon Brushes: Nylon brushes are key to cleaning up a dirty nozzle and heater block. The only issue is that the nylon may melt on hot heaters. I really like to use brass brushes, but the added abrasion is not great for the hotend. If you have a very dirty heater block and nozzle, go ahead with a brass brush, but try to use nylon brushes when possible.

It can be quite difficult to clean up a dirty heater block and nozzle without a set of these brushes.

Heatsinks: As mentioned in the "Stepper Motors Overheating or Malfunctioning" chapter, it is very smart to have some heatsinks on any parts that may be getting too hot. They help to disperse heat and are very smart to add to the extruder, X, and Y stepper motors. Use with a thermal transfer adhesive backing or paste.

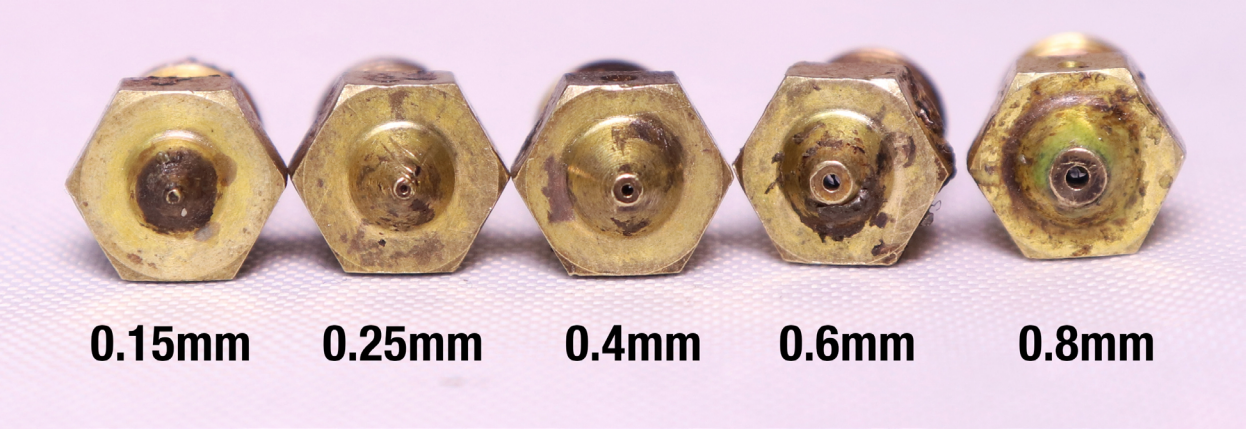
Replacements:

While it is nice to have replacements for every part in your machine, this is clearly not practical. Below are a list of inexpensive parts that should be held on reserve in case you were to ever need them.

Thermistors: Thermistors essentially act as thermometers for your hotend, making sure the heater goes to the proper temperature. These are very fragile parts, and many heater blocks just hold them in via a small screw and washer. These are very inexpensive and nice to have backups in case you ever get an "ERR: MINTEMP" or negative temperature readout.

Heater: Heaters do not need to be replaced very often, but it is another inexpensive part that may come in handy. It can help to save you time diagnosing any hotend heating issues.

Nozzles: This is probably the most important replacement to have on hand. Brass nozzles wear out quite easily, and even hardened steel ones will eventually need to be replaced. If you are ever having difficulty diagnosing why your prints are coming out ugly, it is well worth swapping out your nozzle. Brass nozzles cost under $5 and hardened steel ones are closer to $15, but you will be extremely happy once you change your nozzle and it fixes your problems.



It would also be smart to have a few different nozzle diameters on hand. A good set would be a couple 0.25mm, 0.4mm, and 0.6mm nozzles, and maybe a 0.8mm for large prints. Review my video "The Importance of Replacing Nozzles" on YouTube for further information.

Fans: Brushless fans can become damaged if accidentally hit when spinning, and may just burn out over time. If this is your barrel cooling fan, you are destined

for repeated clogs. Make sure you have a couple of spare fans on hand for when you need them. The majority of fans on a printer are 40mm x 40mm x 10mm, but some use different sizes. E3D setups use a 30mm x 30mm x 10mm fan. Just make sure you purchase the correct voltage for your machine.

Silicone Sleeves: These silicone sleeves go over your heater block and will help to prevent black spots on your print and to maintain print temperature. I definitely suggest to always print with these silicone socks, and to have a couple backups. These will eventually degrade and break over hundreds of hours of printing.

Teflon Tubing: Teflon tubing is used to help guide your filament to the hotend. It helps to prevent filament tangling and to make sure everything is guided correctly. Every so often these can be damaged when pushed into your hotend, so having some extra tubing on hand would be a smart idea.

Endstops: Endstops are another inexpensive part that will every so often need replacement. This doesn't necessarily need to be purchased before needing it, but can save you a couple of days shipping time when you run into an issue.

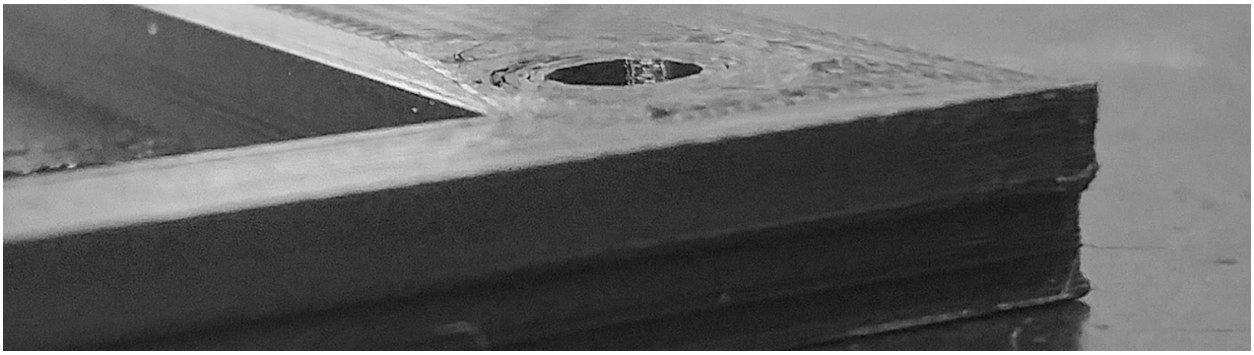
Layer Shifts

Layer shifts refer to when the print looks fine other than the fact that one or multiple layers are shifted in the X or Y direction. Layer shifts can be something as simple as a loose wire or can be as difficult as recognizing that your stepper motor pulley is not functioning properly.

This can result in a print that has one, or multiple layer shifts.

Single Layer Shift

This issue is normally a bit easier to diagnose and fix than a print with multiple layer shifts.



Obstruction during print.

The most common cause of a single layer shift is that there was an obstruction during the printing process. This can be from tangled or too tightly wound filament, or from a cord that is the way of an axis from moving properly. You will see this occur more frequently on larger prints than small ones.

It is another reason I suggest only buying highly made filaments from reputable manufacturers. I have actually had brand new filaments that had a tangle in the middle of the spool, causing a layer shift mid print.

You will want to make sure that your printer has a clear path before starting a print, and that all cords and wires are not in a position to obstruct after moving throughout the entirety of your print area. Confirm your filament is tight on its spool and that you maintain it in a way that will not allow it to tangle. If you have a spool that is becoming unwound during the print, get some Teflon tubing and mount it to your frame as tight as possible to help.

Zip tie all cords in a fashion that get them out of the way of the toolpath. Anything in the way of the extruder or build plate that is stronger than your stepper motor

will cause a skip, and then likely result in at least a layer shift.

Endstops in wrong spot or frame not set up properly

You will run into issues with a single layer shift if you are printing something large and your full print area is not set up properly. If you are using a slicing program where your machine settings are not to the proper dimensions of your printer, the machine will think that it can print further than it actually can. The stepper will skip when your extruder or bed hits its max build area, and the print will continue, assuming that it went the entire tool path.

You can check to make sure everything is set up correctly by homing your machine. When you home the printer and it goes to the very corner of your build plate, you likely have everything set up properly (or the center for printers with a home of 0,0).

When you home the machine, if there is accessible print area in front of or to the side of the build plate (depending on your homing setup), you are not going to be able to use the entirety of your build area.

This can easily be fixed if your X or Y endstop is just in the wrong position by slightly adjusting their location. If these endstops are fixed on your machine, the frame itself may not be set properly. Adjust any t-nuts that may be holding your bed in the position it is in and slide until in the proper homing position. Retighten and make sure nothing can rattle. You need to make sure that your printer homes in the correct spot in order to take advantage of the entire build area.

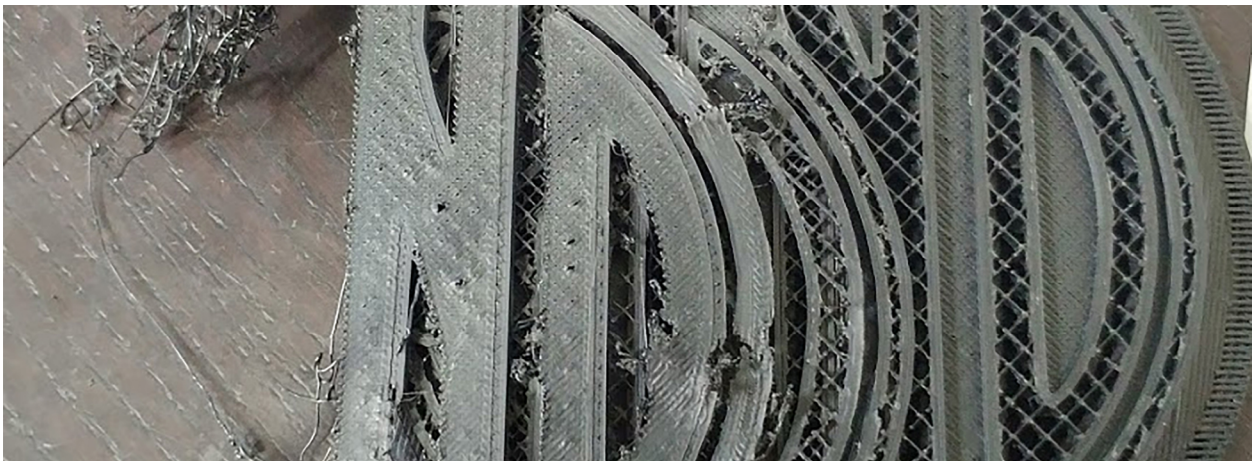
Errors in G-Code or model

You will want to check out the "Model Errors" chapter for a further description, but essentially files can be corrupted or exported improperly. While in your slicing program, check the model layer by layer to see if there are any holes or missing walls. Also be sure to check for actual errors on software such as Cura or the old version of Netfabb.

Your G-Code can actually be corrupted as well. This is not common but I have had prints that just would not print properly no matter what I did. This normally happens if you transfer a file before it has completely saved.

This is a lot harder to diagnose, but if you have a part that has caused a layer shift in the exact same spot after reprinting, then it is worth your time to reslice and reupload to your machine.

Multiple Layer Shifts

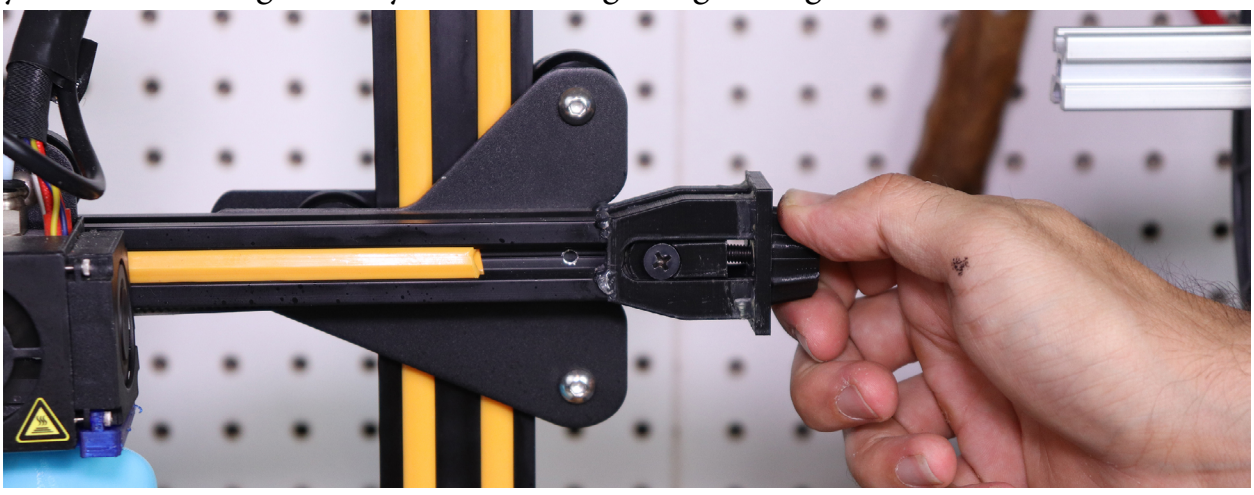


Belts too loose (or too tight)

Belt harnesses on many machines are built in a way that will cause loosening over frequent printing. Some harnesses only hold the belt tight via a zip-tie.

A loose belt will cause slippage and excess play. This is a very common problem when you are using a heavy bed that moves back and forth frequently (you will see frequent layer shifts in the Y-axis).

You will want to make sure your X and Y carriage have belts that are very tight. You can over tighten them, but from my experience a loose belt is far more common. If your belt is too tight it may cause binding and ghosting.



It is smart to print an adjustable belt tensioner for your carriages. This will allow for easy tightening when things get loose over time – otherwise you will likely have to disassemble. I go over this further in the "Z-Wobble" chapter. After adding one of these you actually can over tighten, so be careful.

Dry rods or broken bearings

Most printers have self-lubricating bearings for their carriages, but even those can get dry over frequent machine use. If you are having difficulty manually moving the X or Y axis when the machine is not printing and stepper motors are disabled, check to see if the rods are extremely dry or that the bearings are not broken. A broken bearing is easily replaced and a dry rod can be fixed with some white lithium grease. Also check to make sure that the rods themselves do not have scarring from the bearing wearing down. If the rod is heavily scratched then that may need to be replaced as well.

Just rub a minor amount of white lithium grease to the rods (both threaded and smooth) and then move the carriages around so that it spreads. If you notice a drastic increase in smooth movement of your axis's, then you may have fixed your issues of multiple layer shifts.

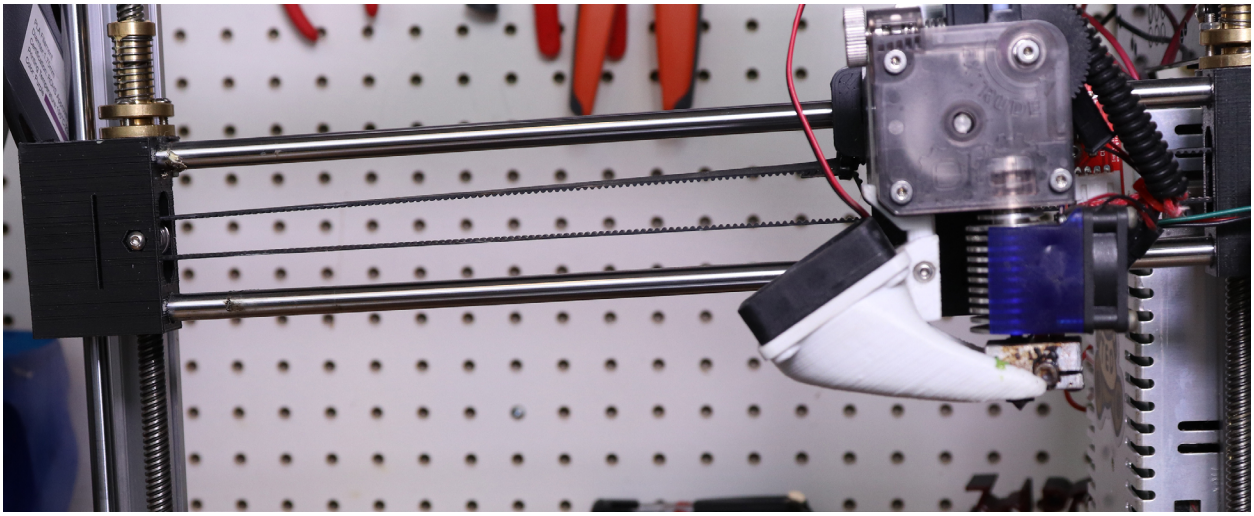
Reapplying lithium grease and checking the resistance on the rods/bearings is good practice regardless of experiencing layer shifts since it can help to allow for a clean print

Bent rods

Rods, especially thin threaded Z-Rods, can become bent over time. This bent rod can cause one or multiple layer shifts as the carriage or bed travel over these bends.

If you notice any rods that are bent, replace them immediately.

Level Build Plate



If one side of your X-axis is much higher than the other, your printer may have difficulty evenly moving the extruder. Having one side of your machine be unlevel from the other could end up causing your stepper to skip multiple times, leading to many annoying layer shifts. This will also put unnecessary pressure on your rods which may cause them to bend and need to be replaced. Make sure took keep your X-axis level.

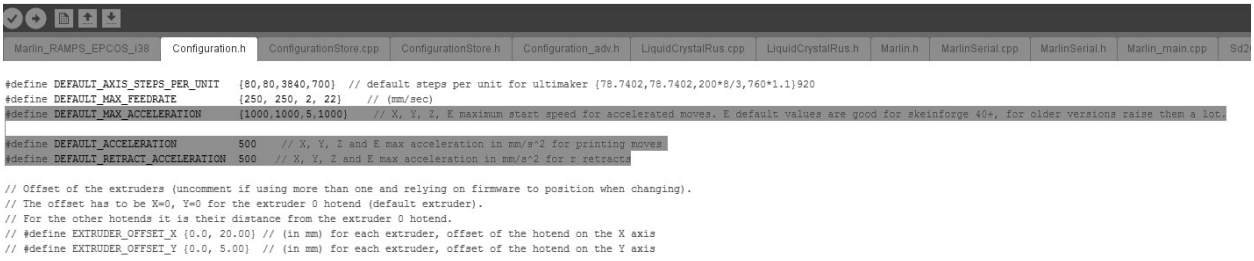
Acceleration or speeds too high

Your motor's torque at a given speed must be greater than the force needed to accelerate or decelerate the carriage at a given acceleration rate and maximum speed. If you require a higher torque than the motor can supply at that given speed or acceleration, the layers will shift.

You can decrease the speed in your slicer settings or by manually turning down the knob on your LCD screen. Acceleration can be decreased on the LCD screen as well, but to permanently change it you will want to go into your firmware. You can also just change the acceleration right in your Cura slicing. You will see this in the "Speed" section of Cura, as explained in the "Settings" chapter.

As mentioned in the "Extruder Stepper Skipping" section, you likely do not want this number over 1000, and for some machines you won't want this over 500. Manually reduce this number and see if it helps.

If you do not have an LCD screen, it is not offered in the slicing program you are using, or you would like to permanently change this number, you can find the acceleration in your printer's firmware in Marlin under the Configuration.h tab.



Driver current is too low or too high

Having the current going to your stepper drivers too low can cause insufficient motor torque and result in layer shifts. A driver current that is too high may overheat the stepper and cause a thermal shutdown. This shutdown will result in that stepper no longer working until it is cool once again. The same is true if the driver on your board is too hot.

Make sure that you have a fan on your board, a heat sink on your stepper motor, and a small heatsink for your stepper drivers. You can then check the current via the method described in the "Extruder Stepper Skipping" chapter, but with the X or Y motor in question. This should not be an issue with factory made machines, but will be more common on the inexpensive DIY ones.

You first want to have your machine turned off and disconnect the stepper from the board. Clip the negative lead of the meter to one of the ground pins on the board. The best place is normally a negative terminal on the main power connector.

Current limits are determined in the motor and driver data sheet. You will not want to run higher than either the driver continuous current limit, or the motor

current rating limit, so it is often good to have a driver that has a higher continuous current rating.

Once you know what current limit you want, you can then go to the current limiting section of your stepper driver data sheet in order to figure out the VREF you will want. Current limit will equal VREF x some number (2.5 for the common Pololu A4988). So if your max current limit is 1AMP, you will have a Vref of roughly 0.4. This would be your target number that your multi meter will read out.

You can then clip the positive lead of your meter to the metal shaft on the screwdriver. Setting your multimeter to 20V DC, you can then touch the potentiometer with the screwdriver head (white circle on stepper driver). You will then see a voltage number on your multimeter. You can turn the screwdriver clockwise to increase the voltage, and counterclockwise to decrease. You will not want to overtighten or screw too fast since there are often not stops on the ends.

Remember, this shouldn't really be necessary on factory made, high end machines, since everything should be set to the proper ratings stock.

Stepper still skips

If your stepper still skips, you may have a malfunctioning motor or have wiring issues. Check the continuity for each wire going from your stepper to your board. If there is a break somewhere you will have to replace it or cut it and rewire.

The stepper or driver itself may be malfunctioning, so try switching the continuity to a different axis and see if the motor still skips when moving that axis. If it does, replace the stepper to fix your layer shifting and skipping issues.

Cheap or worn-out pulleys

Pulleys for your motors need to have sharp defined teeth with the proper spacing in order to work properly moving your belt the correct amount of steps. If you buy a poorly made pulley, or notice that yours have had the grooves worn down, you will want to upgrade and purchase a new set. Go for products made from aluminum or stainless steel, since small items such as this are not that much more expensive and can make a huge difference.

Bed too heavy

This normally is not an issue if you have your drivers putting out the proper current and everything is lubed, but if you have an abnormally heavy bed you may experience shifts in the Y-Axis (on Cartesian machines). If you are trying out an experimental bed with a lot of wiring and extras on top of a ¼" thick glass bed, this axis may have difficulty moving without experiencing some layer shifts.

Bed corners tightened to their max

When attempting to get a level build plate you may run into a time that you end

up over tightening one or multiple corners. You do not want these corners at their maximum spring tightness because over time it can actually warp the metal build plate. These warps will make your problems even worse.

You will notice an issue when you try to move the Y carriage (on Cartesian setups) with the printer off and stepper motors disabled. When one or multiple corners are over tightened, the bed will be difficult to move.

I suggest starting fresh by loosening all of the corners until they have equal minor tension on the springs. If you notice a big difference in how easily your bed moves, then this is likely the culprit. Get the Z-rods even by checking the distance differences for the nozzle to the bed in the X direction. Just hold one rod in place while you twist the other, leveling the X-carriage (as explained further in the "Unlevelled Bed" chapter.

Only then should you adjust the corners for a level build plate. If you have a very warped metal plate you can experience certain corners that will just not get level no matter what you do. In this instance you will actually need a new metal plate. This is why you do not want to leave any corner overtightened for long periods of time. You will be slowly putting pressure on an item that may get bent over time. Using a thicker ¼"glass build plate helps to make this issue less common.

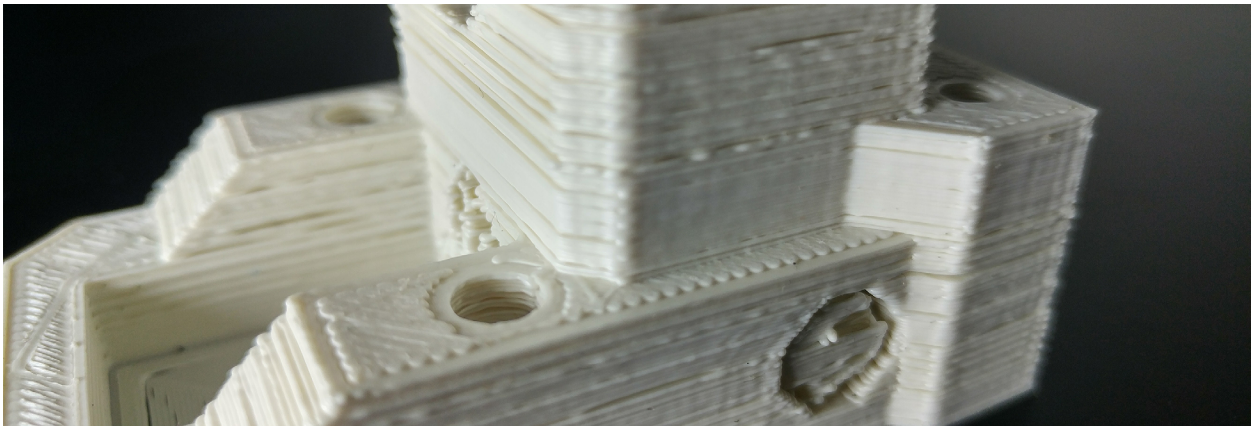
Running into the print

A less common problem with layer shifting can be when the printer head runs into the layer it just printed. This can cause a skip in the motor and will have the print continue where it left off. You are more likely to get a print knocked off the build plate in this occurrence, but sometimes it can cause a layer shift.

This is most common when you are over extruding at very low layer heights with a very strong bed adhesion.

As explained in the "Parts Being Knocked Over" chapter, you will want to add a Z-hop to that of your layer height, and you will want to make sure you have your printer head avoid printed parts in your slicer settings.

Z-wobble



Some may consider Z-wobble in the class of layer shifts, but I have it as its own category. If you are experiencing a wobbly looking print in the Z-Axis (never ending extremely small layer shifts), please refer to the "Z- Axis Wobble" chapter

Summary of Fixes and Precautions

- Clear the printing path of your carriages from any obstructions. – Zip tie all wires and loose cords, and maintain a clean printing area.
- Confirm end stops are in the correct spots and that the frame is built correctly-correct so that when you home the nozzle it is in the furthest part of the corner that it can be.
- Check for errors in model or reslice if G-Code is corrupted.
- Make sure belts are tight enough (but not too tight).
- Make sure the rods are not dry or that any bearings are broken.
- Replace any bent rods.
- Reduce your acceleration and/or speed.
- Increase or decrease the current going to your stepper drivers.
- Check to see if your stepper or drivers are malfunctioning.
- Replace or upgrade your pulleys.
- Do not overtighten bed clips.

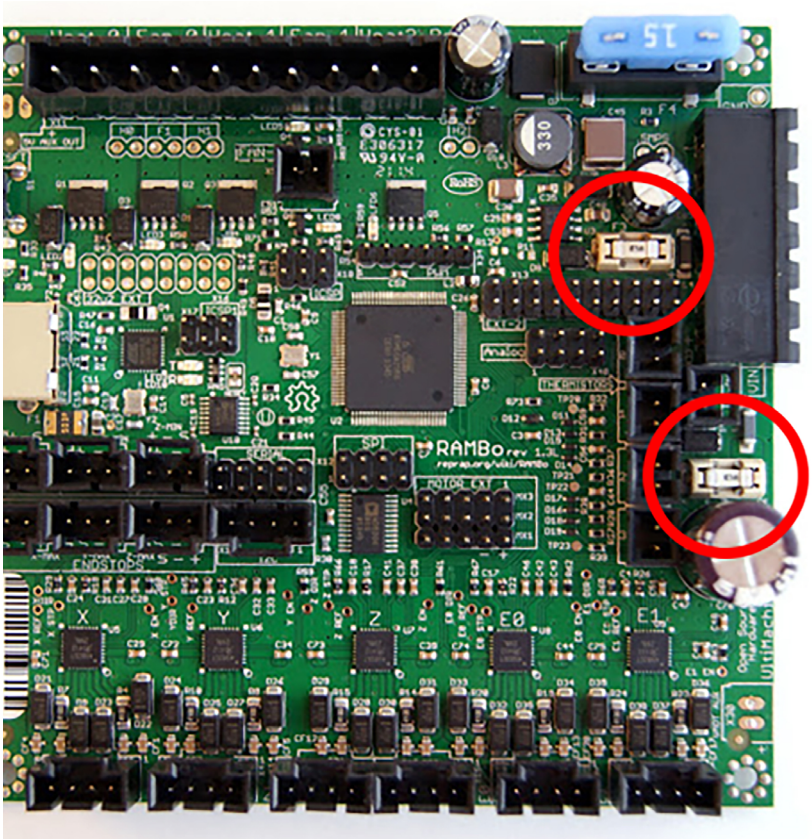
LCD Blank or Dark

If your LCD goes out there are really only a couple of solutions available. Most of the time you can still print with a blank LCD screen, but you can no longer use an SD card to load G-code. If your LCD goes out and you cannot control your printer by using a computer, it is likely your board needs a new NANO fuse, or the screen itself has burnt out.

Replace NANO fuse

I am not familiar with every board on the market but RAMBo boards use two small NANO fuses. Companies such as LULZBOT use RAMBo boards on their TAZ machines.

These NANO fuses can become quite a nuisance because they do go out more frequently than you would expect. When one of these fuses goes out it can cause your LCD screen to go dark. You also will not be able to control any axis or activate any heaters via Octoprint or Repetier when you are have a blown NANO fuse.



When this first happened to me I had to call the manufacturer, because I went through all of the procedures and could not get the LCD screen to work. I had not thought to replace one of these small fuses. You can buy replacements on Digikey or Mouser by searching for a 5A NANO fuse, such as ones made by Littlefuse (also linked on my website). The specific part number is 0448005.MR and they are roughly $2 each. I highly recommend having extra on hand if you are have a board that uses these NANO fuses.

Be sure your machine is off and unplugged before working with these fuses. Carefully remove one and replace it with one of these fuses you purchased. I suggest using a set of tweezers. Re-plug your machine in and see if it fixes your problem. If not, switch the second NANO fuse instead.

I believe every time my screen has gone out yet I could not control the printer it was due to one of these NANO fuses blowing.

Check connections to LCD screen

If you can control your printer but your screen is out or dark, you will want to check all the connections to your LCD screen. Confirm that nothing has been disconnected from either the screen or the board and that there are no breaks in the cords. I have not personally experienced this since the LCD screen does not rattle around during the print, but I can only assume it to be possible. Use your multimeter if required.

Replace LCD screen

I understand this is not ideal, but LCD screens do burn out, especially on a machine that is over a year old with frequent usage. Luckily, LCD screens for basic RepRap machines will only be about $10, but ones for machines such as a Lulzbot TAZ will be closer to $70.

Personally, I have only had to replace two LCD screens in my printing history. So while it is not common, it is definitely possible.

If you have a brand new machine, or one only a couple of months old, your LCD screen should not be burnt out. If this is the case you may want to check with the manufacturer if it is under warranty, or if they will replace due to it being faulty.

Finally, as with many other solutions in this book, if your screen is still dark you will want to re-flash the firmware, and then replace the board if all else fails.

Replace the LCD cable

If you still can't get the LCD to come back alive, it is very likely that the issue is with the LCD cable. If you have another printer available, swap the LCD screens and see if you can get it working. I've seen one case in which the old screen lit right up when swapping it like this. In this case, the system was back up and running with the original screen after replacing the cable.

Summary of Fixes and Precautions

- Check to see if you can still control your printer when plugged into your computer.
- If you cannot control your machine, it is likely a Nano fuse was blown. Replace one or both.
- If you can control your machine, check to see if there are any breaks in the wire to your LCD or if any connections are dislodged.
- Replace the LCD screen if you do not see any problems with the wiring.
- Re-flash firmware, and finally, replace the board if all else fails.

Mandatory Maintenance for your 3D Printer

The majority of the tips covered in this section are covered elsewhere for specific issues, but these are all mandatory to consistently achieve clean prints. I will perform all of these once every couple of months as precautionary steps to not have to worry about fixing a problem after it occurs.

All of these steps are also covered in depth in a video I made titled "Mandatory Maintenance for your 3D Printer" on my 3D Print General YouTube channel.

Check there is no gaps or rattling from the bearings

Most machines have bearings that allow the X and Y axis to move along their rods. These bearings are attached to your carriages normally via printed parts.

What you will want to do is grab your hotend when the machine is off and try to rattle it around. There should be no movement via this rattling. If there is, and you have made sure your hotend is setup properly, this rattling may be coming from gaps in your bearings.

Many machines use plastic bearings that can actually be stretched over time. Bearings should be gripping tight onto your metal rods and if there is any freeplay, it can result in ugly prints. I have had to replace plastic bearings on all machines I have used with them after about 6 months of consistent printing. It is very noticeable after switching to new bearings that the rattling is entirely removed.

If you are using metal bearings, this is very unlikely to be the issue. That said, you will want to make sure these bearings themselves are harnessed tightly to your carriage. This may mean you will need to print new parts or use new zip-ties, since the bearings should be held as tightly as possible to your carriages.

This same exact process should be done for your printer bed, assuming you have a Cartesian machine. In the video I made mentioned in the introduction to this chapter, I had to replace all of the zipties holding the bearings to my build plate since they stretched over time.

Tighten all belts

Other than confirming all harnesses are tight and that there is zero rattling on the extruder and build plate, the next most common reason for Z-wobble is a loose belt.

As explained elsewhere in this book, it is possible to overtighten a belt, but it is pretty difficult to do so on low end machines where the belt is just held together via

zip ties. Both the X and Y axis belts should be very springy to the touch with zero-droop.

If there is any droop in your belt, you will need to tighten. For low end, non-upgraded machines, cut the zip tie that is holding the belt together, grab some pliers, and pull tight as you put on a new zip tie. Make sure the belt is tighter than it was and that the zip tie is pinching everything so that the belt won't slip.

Even better than doing this would be to print a manual way to tighten your belts. There is likely a file on Thingiverse for your specific machine setup, you would just need to search. I have added such a way to tighten the X-axis belt on my CR-10 by a model designed by donnyb99 on that site. I disassembled the X-axis belt and where the bearing is connected, and then added this printed part. I can now easily tighten the X-axis belt via a simple turn on the knob.

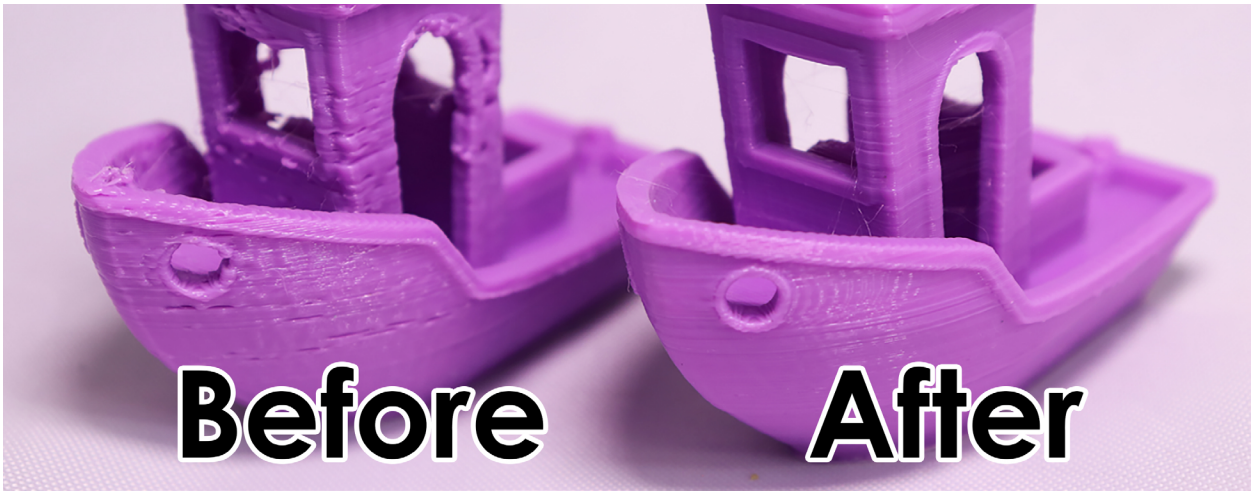
Be careful when adding one of these, since you will now be able to over tighten, which I had mentioned is difficult to do without this. Just turn the knob until the belt is very springy to the touch. There is no real scientific way to do this, you just want to make sure there is zero droop whatsoever.

Clean up wiring

Just because you aren't experiencing an issue doesn't mean you shouldn't keep your wiring neat. Tangled wires from disorganization can lead to potential layer shifts or ripped out wires. If your barrel cooling fan's wires are ripped out mid print, you will get a nozzle clog – one that could have been prevented if you made sure your wiring was organized.

Make sure everything is tucked away and that there is no chance for any wire to get snagged mid print.

Replace nozzle (especially if brass)



Above is a photo from a print with a worn out nozzle next to a brand new one.

As mentioned elsewhere, hardened steel nozzles do not need to be replaced nearly as often as brass ones, but replacing a new nozzle can never hurt. I would replace brass nozzles just about every month or two to ensure that I don't run into any issues in the future, but I have since upgraded to only using hardened steel nozzles. As discussed in the "Over/Under Extrusion" chapter, even hardened steel nozzles can be worn out over time leading to very ugly prints even with all the proper slicer settings.

I also have a video on this on my YouTube channel titled "The Importance of Replacing Nozzles" in which I go over this in further detail.

Check E-Steps

This is the exact same process covered in the "Over/Under Extrusion" chapter, but you shouldn't wait until you see a problem before checking. Once every month or so I will check my E-steps to make sure they are on point, and will tweak if necessary. I don't wait until I see an under or over extruded part since I would rather prevent the problem before it happens.

For most under and over extrusion issues, you will want to check and calibrate your E-steps. To do this is actually quite simple.

You will want to start off by measuring out 100mm of filament. I prefer to use White PLA because it is the easiest to write on, has a low printing temperature, and is cheapest - though you could use any material you have at your disposal.

You can do this in whatever method is easiest for you. I found it easiest to measure this 100mm when the filament is already fed into the extruder. You can also do this on a desk before feeding, but 3.00mm filament, and 1.75 near the end of its spool, are quite hard to keep from rolling back up.

Be as precise as you can by using a fine tip sharpie and holding the material as straight as possible. Use calipers if you have them at your disposal. After heating your hotend, you then want to push the filament down until the lowest dot you made lines up with the top of your extruder, or somewhere else you can easily line up the starting point (because you will need to compare it to where it finishes).

The next thing you will want to do is to tell your printer to extrude 100mm. This is done with a simple G-code command in your terminal.

If you normally print via SD card you will need to hook up to a computer for this. If you print via Octoprint or a similar online program, you can send the G-Code commands from their terminals.

When hooked up to Repetier Host, or whatever program you use to control your machine, and your hotend hot, you will want to give your machine the command:

```
G92 E0
```

This sets your extruder to 0. Next you will want to give either the command:

For 3.00mm Filament: **G1 E100 F30**

or

For 1.75mm Filament: **G1 E100 F60**

This will tell your extruder to feed 100mm, and is why it was important you lined up your starting dot with either the top of your extruder or something similar.

Once your extruder has finished you will want to mark your filament at the same spot you lined up your original dot (top of extruder in my examples). If your 100mm dot lines up perfectly, then your e-steps are right on - but even 1mm means that your printer is extruding incorrectly by 1%.

After marking where 100mm actually was, you will want to compare it to where you measured 100mm to be at the beginning of this process. If higher on the filament, your printer is over extruding, if lower on the filament, your printer is under extruding.

After measuring this difference you will want to write down somewhere how much your extruder actually fed. If your printer over extruded by 2.1mm, you will want to mark down 102.1mm. If it under extruded by 2.1mm, you will want to mark down 97.9mm. You will need this number later on.

The next step in this process is to determine what your current E-Steps are. You can do this by either checking the firmware for your machine, by going into the "Motion" section of your EEPROM (LCD Screen), or just by giving it the command "M503" in your printer terminal. Direct drive non-geared extruders have e-steps of around 90, while Greg's wade and other geared extruders have E-Steps of 500 or more.

If you are checking in the firmware that you use to flash your machine, you will want to open it up. While in Marlin you will go to the "Configuration.h" tab and scroll all the way down to where it says "DEFAULT_AXIS_STEPS_PER_UNIT", with e-steps being the 4th and final number (if using one extruder).

You could also type M503 into your terminal. You then take this number and multiply it by 100. You will then divide this new number by the number you wrote down earlier.

For example, if your current e-steps are 90.5 as shown above, you will multiply it by 100 to get 9050. We will then divide 9050 by how much you extruded earlier. So if you extruded 102.1mm, you will take 9050 and divide it by 102.1 to get 88.64.

90.5 x 100 = 9050

9050 ÷ 102.1 = 88.64

88.64 in this above example would be your new e-steps. As you can tell it is lower than it was before, because in this example you were correcting for over extrusion.

You will now set your e-steps. You can do this through your terminal, EEPROM,

or by flashing your firmware. If you are going to do this through your terminal you will want to give the M92 command, by typing M92 E88.64. You will then want to type M500 in order to save these settings.

While you can set this number on LCD screens under the "Motion" section, it will only save permanently if you have the option to save your settings after doing so. Otherwise your e-steps will reset once you turn your machine off.

Thomas Sanladerer has a great tutorial video going over all of this on his channel which you can find by searching "calibrating your extruder" on YouTube.

Clean your build plate

Cleaning your build plate frequently will help to prevent bed adhesion issues. Don't wait until your print won't stick before you clean your bed – proper precaution is important.

Lubricate Rods and Leadscrews

Grab a paper towel and clean off all of your metal rods. Then get some white lithium grease and rub it over both your smooth rods and threaded rods. While the bearings you use are advertised as "self-lubricating", it won't last forever.

After adding some grease, move all of your axis to all positions. This will help spread the grease out and you should notice a distinct difference in moving your carriages around.

Tighten all screws and bolt on your machine

Frequent rattling of your machine can get screws and bolts loosened on your machine. If you haven't checked in a while, you will be surprised just how many aren't tight anymore.

When I say check every screw, I mean every single one. The ones holding your frame together, the ones holding your build plate, the ones holding your extruder – all of them. The last time I did this I had waited about 2 months. I found over 5 screws that had become entirely loose. Consider adding non-permanent thread locker to your screw threads to reduce the impact of vibrations over time.

These loose screws can lead to Z-wobble or entirely failed prints. Take the precautionary steps to prevent this from ever happening.

Materials and their Settings

By this point you are well aware that different manufacturers with different printer setups will require different specific slicer settings. That said, I will review some of the settings that work best for me on both a geared direct drive and geared Bowden machine. These will be good starting points, though the speeds should be turned down a bit when working with non-geared extruders.

While Bowden machines can move faster without experiencing rattling, since the carriage is much lighter without the extruder, it is limited on specific materials due to the distance between the extruder and hotend. This also means it is more likely to print stringy and have difficulty travelling over small distances without leaving a trial, requiring an increased speed and distance for retraction.

All of the settings below are editable in Cura 3.3 and higher, and likely editable in your slicer as well. Make sure to read the PLA section entirely regardless of what material you are printing with, since there are some overall notes factored within.

If you would like further information about 3D Printing materials, I would suggest you visit OptiMatter by SD3D (OptiMatter.com). OptiMatter provides a wide range of data on materials used for 3D printing and can even help to calculate the best printing configuration for you.

NOTES ABOUT ALL MATERIALS:

This will be covered in the "Settings" section in this book, but first I will cover a few specific slicer settings that will be used regardless of the material. I almost always use Triangles or Hexagonal infill patterns, since I see no need for many of the other options, and I will always have at least 1mm of top layers. I keep acceleration and jerk lower than most other operators because I notice this is one of the most important things to keep low for a clean print, but it does increase the print time required. Both acceleration and jerk can be set higher on a Bowden machine though, due to its decreased carriage weight.

Make sure you always properly purge the filament when switching between materials in order to reduce the chance of black blobs on your print. Read the "Built up Material on Nozzle" chapter for a further explanation.

As of writing this book, Cura also has an experimental feature called "Coasting" that I use for almost every Bowden print. You will see throughout this section that Bowden machines come with an increased difficulty reducing stringy and hairy

prints. Coasting replaces the last part of an extrusion path with a travel path, meaning the oozed material is used to print the last piece of the extrusion path to reduce stringing. This is great, but if you go too high, you can experience under extruded prints. For most materials I use the following Costing settings on Bowden machines:

```
Coasting Volume: 0.064 (nozzle diameter cubed)
Minimum Volume Before Coasting: 1mm3
Coasting Speed: 90%
```

PLA

Polylactic acid (PLA) is the most common and easiest to use 3D printing material. While PLA has a very high tensile strength, it has a very low bend-to-break ratio, making it one of the most brittle materials upon impact. The low glass transition temperature means that the printed part cannot withstand heat higher than 50°C, which means parts will deform when left in the dash of a hot car. These two factors mean that PLA is not going to be your choice for practical mechanical parts.

While you aren't going to be using PLA as a load bearing mechanical material, it has numerous other applications. Its ease to print, low shrinkage rate and inexpensive price makes it the perfect choice for models, prototypes for sizing, molding, cosplay, and other fun prints. Being a stiff part that holds its dimensions makes it a great choice for anyone wanting to make a negative cast for molding. For these exact reasons I personally use PLA far more than any other material. All of my fan art around my house and most of the examples used in my YouTube videos are made via PLA.

PLA Slicer Settings:

Print Temperature (180° - 220°C)

This print temperature range is huge and is only this large due to the vast variations among individual manufacturers. Most PLA manufacturers either ship their filament with a processing temperature range of 180°- 205° or 205°-220°, you just have to check with your individual spool. Even then, these ranges are larger than the average material you work with, and you will find that you will be able to achieve a successful print almost anywhere within these ranges. My personal choice is to go to the higher side of the material range. I like to print around 200°C in the materials with the lower range, and around 215°C with PLA that has the higher range on both Bowden and Direct Drive machines - though you can play around a bit and lower if you are experiencing blobs.

Build Plate Temperature (60°C)

PLA is a bit unique in the fact it can actually print on a bed that is not heated, but it will stick and work best on one that is heated to its glass transition temperature of around 60°C. While working with a 60°C build plate, you can actually just

print directly onto a clean glass bed, but it is best to add additional adhesive. My favorite adhesive solution is hair spray since it is easy to add, remove, and keeps the underside of the print clean. When working with a printer that does not have the ability to heat it's bed, you will need to go out of your way to make sure the part can stick on the initial layers beyond just hair spray. This is when you will likely want to use blue painters tape or another form mentioned in the "Bed Adhesion" Chapter, along with including a brim on just about every print. Having a heated bed will definitely reduce your headaches, but it is not mandatory for this material.

While working with a heated bed, I reduce the amount of times I use a brim to as minimal as possible. This is because most models should stick just fine without warping and removing a brim for PLA is not nearly as easy as it is on other materials. I have definitely regretted using a brim in the past when printing 30 small parts on a single bed – leading to an extra couple of hours of post-print manual cleanup.

Retraction: (Direct Drive: Distance: 2mm – Speed: 20mm/s – Minimum Travel 0.8mm) (Bowden: Distance: 4.8mm – Speed: 55mm/s – Minimum Travel – 0.8mm)

Please note that retraction may need to be tweaked for individual models. Some models require retraction to be turned off entirely in order to print properly, but these are good starting points for your average model.

This is the setting that has the most difference between a direct drive and Bowden setup. Due to the pressure built up between the extruder and hotend in the teflon tubing, Bowden has increased propensity for stringiness and will require increased retraction settings on every material (some more than others). With a geared direct drive setup, these retraction settings work great on most PLA manufacturers that I have used, with minimal tweaking required for specific models.

Bowden machines need more tweaking than direct drives when it comes to this regard. Some models may be under extruded with these settings and need you to reduce the distance and speed. This is why I often tweak with the minimum travel. If you have a part that requires a lot of retraction to reduce stringiness, but there are a couple very small areas on your print that will be under extruded, you can bump this number up so that no retraction occurs over those small sections.

I almost always also keep the COASTING option turned on for Bowden prints, available in Cura 3.3 and newer, mentioned in the Notes about All Materials section a few paragraphs earlier.

Speed: (Direct Drive: Speed: 45mm/s - Outer Wall: 20mm/s – Top/Bottom: 35mm/s – Initial Layer: 20mm/s – Travel Speed: 80mm/s) (Bowden: Speed: 60mm/s – Outer Wall: 30mm/s – Top/Bottom: 30mm/s – Initial Layer Speed: 25mm/s – Travel Speed: 100mm/s)

This section will get a lot of dispute from other experienced users, and definitely is

different than the manufacturer advertised speeds. Personally, I don't mind a print taking an extra hour or two if it means that the quality is increased and the chance of failing is decreased. Reducing these numbers (as well as jerk and acceleration) from the recommended settings has done nothing but improve my print quality and reduce my failure rate.

PLA is the material I run the fastest due to its ease of printing. A light Bowden allows for fast printing, and if your setup has very little rattling with a geared extruder, you can likely go 65mm/s without any reduction in quality. Due to its rigidity, PLA is an excellent material for fast processing on a Bowden setup because it will not compress much, if at all, in the teflon tubing.

Anyone saying they can achieve print speeds over 120mm/s is either lying, getting poor quality prints, or does not understand how acceleration works. For any print to reach its top speed, enough distance is required and acceleration speeds set high enough to reach these speeds.

If your printer starts rattling when moving in-between parts, reduce your travel speed. If you don't see any ghosting and your parts are coming out extremely clean, you can try bumping speeds up to see if you get the same results.

These settings are also for when I am printing 0.2mm layer heights on a 0.4mm diameter nozzle. When printing with a very small nozzle at very small layer heights, these speeds will drastically need to be reduced. When I print on a 0.25mm nozzle I reduce everything by about 20%.

While you should be able to achieve faster speeds than this when using a large nozzle diameter at large layer heights, you run into the issue of overheating your extruder stepper and getting skipped steps. You can keep speeds as they are if you have a powerful extruder, but it is also likely you will need to reduce these speeds when going over 0.6mm diameter nozzles. When I print with a 0.8mm diameter nozzle at 0.5mm layer heights I also have to decrease my speeds by about 20%. This can be helped by using a hotend such as the E3D Volcano setup.

Support Settings: Overhang Angle: 50-55° - Density: 12% - Z Distance: 0.2mm – X/Y Distance: 0.8mm – Enable Support Interface – Interface Thickness: 1mm – Interface Density: 80% - Interface Pattern: Concentric)

These settings are the ones that will require the most additional tweaking for your individual printer and model. These are also for single nozzle parent support material. If you are using dissolvable support material, you will likely want to increase the density and decrease the Z distance quite a lot. The 12% density may be too little for specific models, as well as the overhang angle. If you have support interface turned off, make sure you increase the density and decrease the Z Distance.

I have found that the support interface works great with PLA with difficult overhangs, it will just require you to be extra careful when removing the support

material. Since PLA is so rigid and brittle, the part can easily break when trying to remove stubborn support that was not processed with the right settings. With the settings I provided above, it results in a very clean underside with support material that can kind of be "cracked" off in a couple of seconds, with a minimal amount of razor cleanup.

There is a video on my YouTube channel titled "Cura Support Settings – PLA" in which I go over this further in depth.

NOTES ABOUT PLA:

PLA is the easiest of the materials you will read about in this section to work with. You can get just as clean results with all of these settings tweaked slightly. As with almost all materials, PLA also can absorb moisture when not stored properly. If you want to check if your PLA is good to work with, it should be breakable with a quick bend on the spool. If it requires you to get scissors out to cut the material, since you cannot break it, it likely has absorbed moisture and needs to be dehumidified.

Since PLA is one of the least warping material, you should also always have your active cooling fan on full blast after the initial layer. This makes a drastic difference in surface quality and overhangs achievable.

ABS

Acrylonitrile butadiene styrene (ABS) was the second most used material by myself, as it was always the go-to material for mechanical parts with a low price tag. ABS is what Legos are made out of, and its properties make it a great choice for mechanical parts as well as those requiring a high heat resistance - handling a heat deflection temperature up to 95°C (or 205°F). It is also soluble in acetone, allowing you to blend layers together and add an injection molded shine in an acetone vapor bath.

One thing you will hear from just about every experienced 3D printer is the difficulty to print this material without warping and delamination. Small parts may be achievable, but any large model will require a well-built enclosure with ambient air of over 50°C for any hope of success. Even with this environment, a level PEI build plate, and a model with 30 lines of brim, it is difficult to get a non-warped part weighing over 500 grams. This is why you will need an ABS Slurry for very large prints, and this added difficulty has turned ABS into a material I only use for small models.

You will want to keep your active cooling fan OFF when printing with ABS to help prevent warping.

ABS Slicer Settings:

Print Temperature (220° - 240°C)

ABS has a slightly higher printing temperature and lower range than PLA - depending on the manufacturer of course. I seem to prefer to print ABS at 230°C, but you may have better results tweaking within this range. Always check with your

manufacturer's suggestions first.

Build Plate Temperature (105°C)

This is one of the reasons it is so difficult to print ABS – bed adhesion. With a high glass transition temperature, you will want to run your print bed at a hot 105°C.

The issue with this will come from having drastically different ambient air temperature. Having cool air surrounding your print will cause those areas to want to shrink before the bottom of your print, leading to warping and delamination.

I have no problems printing small parts in ABS on an open printer, but it would be smart to print anything of size on an enclosed machine. I have been able to reach ambient temperature of over 55°C in an enclosed machine, which works perfectly to prevent warping and delamination. You just need to make sure the electronics used inside the machine can handle those temperatures.

Even when having your print bed at 105°C, warping will still occur. I personally just use Aqua Net hairspray on a glass bed for adhesion, but this won't work on large parts. If your part absolutely needs to stick and has to be in ABS, you will need to make an ABS slurry if your printer is not entirely enclosed.

Retraction: (Direct Drive: Distance: 1.8mm – Speed: 15mm/s – Minimum Travel 1.2mm) (Bowden: Distance: 3.5mm – Speed: 35mm/s – Minimum Travel – 1.2mm)

For ABS I reduce my retraction settings a bit from PLA. It seems that ABS doesn't have as many issues with "stringiness" or "hairy" prints as other materials, so these numbers don't need to be quite as high. This is because ABS has a lower heat capacity than most of the materials we work with. PLA, PET and elastomers have much higher heat capacities. The lower the heat capacity the faster the material can cool. This leads to less stringing but also lower interlayer adhesion. ABS is also still fairly rigid so it doesn't have to deal with extreme compression like what would be dealt with when processing a nylon or elastomer on a Bowden setup. If you are experiencing stringing with these settings, increase accordingly, or use "Coasting" if applicable.

As with PLA, each model may require different retraction settings, with even some models having you turn retraction entirely off.

Speed: (Direct Drive: Speed: 45mm/s - Outer Wall: 25mm/s – Top/Bottom: 40mm/s – Initial Layer: 25mm/s – Travel Speed: 80mm/s) (Bowden: Speed: 55mm/s – Outer Wall: 30mm/s – Top/Bottom: 30mm/s – Initial Layer Speed: 25mm/s – Travel Speed: 100mm/s)

As with all materials, speed is also going to be limited by the machine you are using. Travel speed can be bumped up if you are using a strong, reliable machine, but you will always be limited based on acceleration and deceleration speeds. This is why a large printer can normally reach higher top speeds than a small one, having a longer period of time to accelerate and decelerate.

My CR-10 Bowden machine is limited to a travel speed of around 100mm/s, since the motors start making some loud noises over this. This being said, I print ABS at around the same speeds as PLA. Some machines advertise speeds much higher than this, but I always suggest starting low and increasing until failure.

These are settings for a 0.4mm nozzle and 0.2mm layer heights. These speeds will have to be decreased with smaller nozzles and layer heights, and can be increased with higher ones.

Support Settings: Overhang Angle: 45-50° - Density: 15% - Z Distance: 0.2mm – X/YDistance: 0.8mm – Enable Support Interface – Interface Thickness: 1mm – Interface Density: 80% - Interface Pattern: Concentric)

ABS is another material that I prefer to use the support interface on in order to improve underside quality. From my tests its seems that ABS cannot handle angles that great, so I will often set the overhang angle to 45°.

Since I keep the cooling fan OFF on ABS, it also seems to have more difficulty bridging gaps. Because of this I bump the density up slightly to 15% just to make sure the support interface lays down cleanly. I personally see no reason to go higher than 80% support interface density on any material, so I keep that the same regardless.

NOTES ABOUT ABS:

ABS is slowly being used less and less in 3D printing as easier, strong alternatives present themselves and their prices continue to drop. The rate of me getting a failed print while using ABS is much higher than just about every other material (with a few exceptions such as PCABS and other very high temp filaments).

One of the best parts about printing in ABS is the fact is soluble in acetone, meaning you can smooth the outer sections and make it more watertight, for a more injection molded look and feeling. You can do this via a crock pot on low for 10-20 seconds with your print lifted, you just have to be very careful doing this. For a further explanation, please read the "Post-Processing" chapter in this book.

PETG

Polyethylene terephalate with a glycol modification (PETG) is a plastic resin of the polyester family that is used in beverage containers, food packaging, and countless other everyday applications. While it is used for food containers, it is not recommended to use any 3D print for food parts, due to the minor gaps created in-between layer lines where bacteria can grow.

PETG has slowly become the replacement for ABS as the go to filament for strong mechanical parts at a low price tag. You can get a 1KG spool of PETG for only a couple of dollars more than PLA, and you can easily print a large piece without warping - so you will likely want to use this for the majority of your ABS applications.

Assuming you do not need your part to withstand higher than 70°C, then PETG will likely be your choice over ABS.

PETG is very susceptible to stringiness due to its high heat capacity, and you can end up under extruding by attempting to rectify it via increasing retraction, so you may need to clean this stringing up post print on specific models.

PETG Slicer Settings:

Print Temperature (240° - 260°C)

Some say that you can print PETG below 240°C, but I personally have always printed at 250°C successfully (with minor tweaks for small/large nozzles and layer heights).

Always check your filament manufacturers recommended settings, but I have had to go above this a couple times for PETG.

Build Plate Temperature (70° - 85°C)

You can actually print PETG without a heated bed, but it isn't simple. When heating to 70°C, and having a fully level bed, PETG sticks great and has no problems with warping whatsoever. I have printed parts that take up the entire 300mm x 300mm x 400mm build area of the CR-10 in PETG with very minimal problems with warping or delamination.

I personally prefer Fiberlogy PETG which actually requires closer to an 80°C print bed.

Retraction: (Direct Drive: Distance: 2.2mm – Speed: 22mm/s – Minimum Travel 0.7mm) (Bowden: Distance: 5mm – Speed: 55mm/s – Minimum Travel – 0.7mm)

Many of the settings for PETG are similar to PLA, but I have found a slightly increased retraction to help out immensely to reduce stringiness. As mentioned earlier, PETG seems to ooze out more than other materials, so you will want a bit of an increased retraction over PLA, and you will also want to have Coasting turned on.

If you see under extrusion after increasing these numbers, reduce these settings and you will just have to clean off the string post-print with a blow dryer or a razor.

Speed: (Direct Drive: Speed: 45mm/s - Outer Wall: 25mm/s – Top/Bottom: 40mm/s – Initial Layer: 25mm/s – Travel Speed: 80mm/s) (Bowden: Speed: 55mm/s – Outer Wall: 30mm/s – Top/Bottom: 30mm/s – Initial Layer Speed: 25mm/s – Travel Speed: 100mm/s)

This sounds a bit crazy, but I keep everything the same as I do when printing with PLA. These numbers worked perfectly so I saw no reason to change anything.

Keep in mind, as with all materials, these settings are for a 0.4mm nozzle and 0.2mm layer heights, with speeds needing to be tweaked if you are using something different.

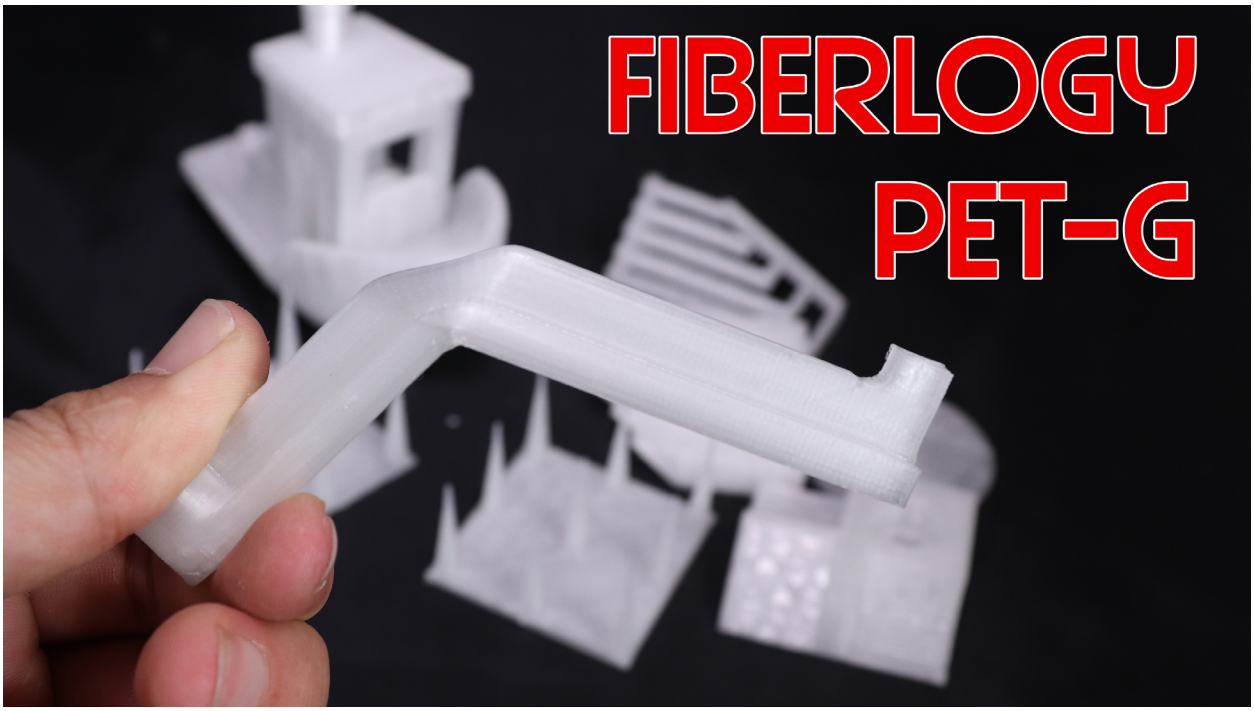
Support Settings: Overhang Angle: 60° - Density: 12% - Z Distance: 0.2mm – X/Y Distance: 0.8mm – Enable Support Interface – Interface Thickness: 1mm – Interface Density: 80% - Interface Pattern: Concentric)

While I keep the actual support settings the same as PLA, the angle which support is required is much higher than PLA. When printing overhang tests, it seems that PETG achieves some of the cleanest overhangs of any material. This means you can go all the way up to 60° without requiring any support material, saving you a ton on material and time required to print lots of models.

With the support interface you may need to get your razor out, but removing support should essentially "crack" off and leave you with a very clean underside.

NOTES ABOUT PETG:

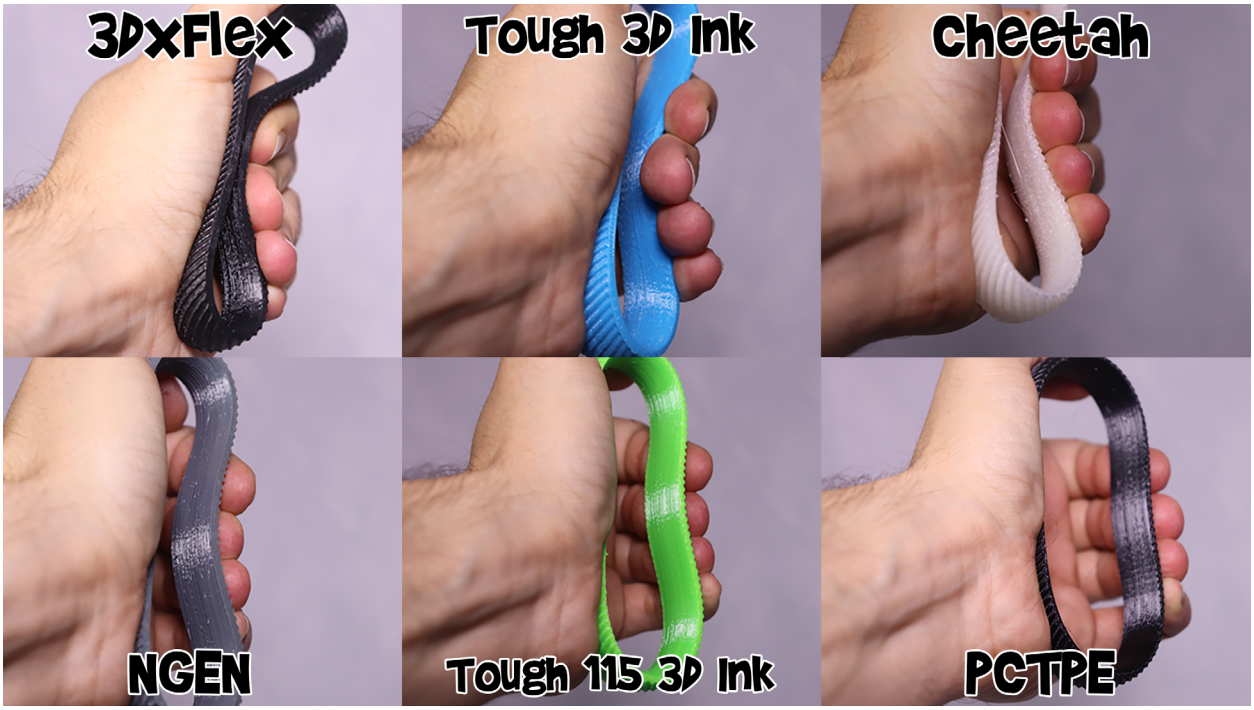
There is a video on my YouTube channel titled "PETG Cura Settings" in which I go over this more in-depth. I also have a video reviewing my current favorite PETG titled "Fiberlogy PETG Review".



I have always used the active cooling fan "on", but others have said to keep it off for increased layer adhesion. If you are noticing difficulty with layer adhesion, try turning the active fan off.

PETG also seems to have difficulty bridging gaps (when compared to PLA), so you will require support settings turned on for small gaps.

Flexible Filaments



Flexible filaments come in a wide variety of properties and print settings, so it is just about impossible to give an all-inclusive profile. I have made two YouTube videos so far in which I go over 10 popular flexible titled “3D Flexible Filament Comparison” and “3D Flexible Filament Comparison Part 2” to get a bit more of detailed description of options available.



While there are quite a lot of flexible options out there, my personal favorite for a lot of applications is Cheetah by NinjaTek.

NinjaTek makes NinjaFlex, one of the most flexible options available on the market, but it is extremely difficult to print. Cheetah was created in order to fix this by allowing you to print faster, hence its name. I will be using Cheetah below as the example because of this, and because of the fact you can print this on a Bowden machine such as the CR-10 with an upgraded extruder.

Some overall notes for flexible filaments:

- All flexible filaments print better on a direct drive extruder vs a Bowden machine, and some materials are entirely impossible to print on a Bowden extruder.
- You will need a geared extruder rated to print flexible filaments. You can print a basic Greg's Wade and purchase minimal parts, or you can purchase an upgraded extruder. My favorites are the E3D Titan, and the Bondtech BMG Extruder. The Bondtech BMG has a dual-drive, meaning there are two gears gripping onto the filament. This, along with a 3-1 ratio makes it my favorite extruder to use when having difficulty printing flexible filaments
- Slow your print down! Print at speeds about 25% lower than recommended manufacturer settings to start with, and reduce further if needed. Some very soft materials such as 3DXFlex and NinjaFlex I have to print at 25mm/s in order to not experience a failed print.
- The harder the flexible filament, the easier it is to print. Almost any flexible filament with a shore hardness of 90A or lower is very difficult to print
- Parent support material is extremely hard to remove. Flexible filaments with a large bend-to-break ratio have high layer adhesion, making parent support material very hard to remove cleanly. Because of this, I do not use support interface. You should always factor in the difficulty to remove support material when designing your flexible part.

CHEETAH Slicer Settings:

Print Temperature (235° - 245°C)

Flexible filaments have a wide range of printing temperature depending on the particular manufacturer. For Cheetah I have found good results printing at 240° while something like Tough 3D Ink by M3D prints better at 250°C.

Build Plate Temperature (45°C)

Many flexible filaments seem to prefer a print bed temperature of just 45°C, but there are of course exceptions to this. NGEN by Colorfabb requires a print bed of 80°C and PCTPE by taulman3D requires a coat of PVA be applied to prevent warping. Cheetah seems to print just fine without any warping with a 45°C glass bed and Aqua Net Hairspray.

Retraction: (Direct Drive: Distance: 2mm – Speed: 20mm/s – Minimum Travel 1mm) (Bowden: Distance: 3mm – Speed: 40mm/s – Minimum Travel – 1mm)

I have retraction turned down from other materials due to the difficulty to print flexible filaments. Since speeds in general are reduced, retraction speeds should be reduced as well.

If you are printing a part that requires a lot of retraction and are doing it in flexible filament, even Cheetah, you are going to be left with some string you will have to clean up post print. You just need to grab a razor and scrape off the residue left on the print.

Speed: (Direct Drive: Speed: 40mm/s - Outer Wall: 20mm/s – Top/Bottom: 35mm/s – Initial Layer: 20mm/s – Travel Speed: 80mm/s) (Bowden: Speed: 35mm/s – Outer Wall: 17.5mm/s – Top/Bottom: 17.5mm/s – Initial Layer Speed: 20mm/s – Travel Speed: 100mm/s)

I am able to print cheetah faster than any other flexible filament, so these will be the upper bound for flexible materials. Remember you require an upgraded extruder (and likely hotend) if you want to print Cheetah on a Bowden machine such as the CR-10. I currently have a Bondtech BMG on my CR-10 and it is able to print Cheetah like a champ with the settings provided.

That being said, even with this $100 upgraded extruder and a $60 upgraded hotend, I am unable to print extremely soft materials such as 3DXFlex on my Bowden CR-10 regardless of how slow I run the print. If you plan on printing a lot of flexible filaments, it is worth your money to upgrade to a geared direct drive setup.

Support Settings: Overhang Angle: 50° - Density: 15% - Z Distance: 0.2mm – X/Y Distance: 0.8mm – Disable Support Interface)

These settings are definitely not perfected, but they work well for me while printing with Cheetah. As mentioned in the earlier notes, removing parent support material both cleanly and easily is near impossible with certain flexible filaments.

The overhang angle when support material is required will need to be found out for each material. NGEN by colorfab can get closer to 60°, so make sure you print an overhang test to find out when you will need support.

NOTES ABOUT CHEETAH:

Cheetah has been the easiest to print flexible filament for me, so make sure you decrease most of these numbers when working with other materials. Some flexibles require a specific print bed since they have a nylon mixture.

Always store your flexible filaments in dry area vacuum sealed with some desiccants. This is true with all materials, but flexibles and nylons absorb water the fastest and will need to be vacuum purged if left out in a semi-humid area.

Nylon Filaments

Nylon can also be referred to as a polyamide and are generally strong (and often semi-flexible) options for 3D printing. Almost all nylon options offered on the market are more expensive than PETG, but they are best for many applications. If you require impact resistance, you are likely going to want to go with nylon due to its durability to flexibility ratio.

Nylons come in a wide variety of options, but almost all have a bit of difficulty sticking to build plates without extra help. As mentioned elsewhere in this book, you will likely want to grab some Elmer's glue, do a 1-1 mixture with water, and lightly brush it onto a clean glass build plate. After evaporated, nylons will stick great and pop off fairly easily with a scraper.

This may not be needed for all nylons, but I prefer doing this for most to prevent warping or parts being knocked off the print bed.

For the example below I will be using Nylon Alloy 910 by taulman3D, since it is one of my favorite strong nylons to work on.

Some overall notes for nylon filaments:

- Nylon is hygroscopic and must be kept dry. This is not an option as leaving out your nylon spool can actually make it absorb too much moisture in a matter of a day or two. You should keep all nylon spools vacuum sealed, and in a dry area (30% humidity or lower).
- Many nylons have a shelf life and will not print as well after a year of being on the shelf.
- Nylons are very durable and have an excellent strength to flexibility ratio.
- Nylons often do not come in many color options, but most can be dyed (since it can absorb moisture). I have tried this in the past to mixed results
- Just as with flexible filaments, removing parent support material can be very difficult with Nylons

Nylon Alloy 910 Slicer Settings:

Print Temperature (230° - 240°C)

I found a happy medium with Alloy 910 by printing at 235°C. Check with your manufacturer suggested settings before trying out a different Nylon, since some can print as low as 220°C, while others will require 250°C+.

Build Plate Temperature (45°C)

As with most flexible filaments, the majority of nylon materials I have worked with in the past prefer a 45°C print bed, and the vast majority require a glass bed with a PVA mixture to print. Refer to the beginning of this Nylon section to read further on how to do this properly.

Most nylons shouldn't give you much issue warping, but Alloy 910 has warped on me in the past. You just need to make sure you evenly apply your PVA mixture and

that your bed is levelled at the correct z-height.

Check with the manufacturers recommended print bed settings to be sure.

Retraction: (Direct Drive: Distance: 2mm – Speed: 20mm/s – Minimum Travel 1mm) (Bowden: Distance: 3mm – Speed: 40mm/s – Minimum Travel – 1mm)

Without going too much into detail, I kept my settings the same as I did with Cheetah and got great results with Nylon Alloy 910. Since it worked well on both of my Direct Drive and Bowden setups (with upgraded extruders), I figured I wouldn't tweak it much further.

This should be a good starting point for most Nylons and you will have to tweak further if you see too much stringiness, or if you see any under extrusion.

Speed: (Direct Drive: Speed: 35mm/s - Outer Wall: 17.5mm/s – Top/Bottom: 30mm/s – Initial Layer: 17.5mm/s – Travel Speed: 80mm/s) (Bowden: Speed: 30mm/s – Outer Wall: 15mm/s – Top/Bottom: 15mm/s – Initial Layer Speed: 15mm/s – Travel Speed: 100mm/s)

These settings worked for me when working with Alloy 910, and are a great starting point for most nylon filaments. I don't recall working with a specific nylon that required drastically slow print speeds, so proceed with what I have above, unless you read otherwise by the manufacturer.

Support Settings: Overhang Angle: 50° - Density: 15% - Z Distance: 0.2mm – X/Y Distance: 0.8mm – Disable Support Interface)

These settings stay the same as flexible filaments due to the difficulty to remove the support material cleanly. If you are experiencing ugly undersides but the support material is easy to remove, decrease your Z distance. If you cannot remove the support material and it is stuck to the print, increase your Z distance.

If you notice a lot of drooping and that the particular nylon you are working with is unable to bridge gaps, increase the density. If you think you will be able to remove the filament, go ahead and test out support interface – I just personally haven't had great results in being able to easily clean it all up.

NOTES ABOUT NYLON ALLOY 910:

Alloy 910 is made by taulman3D and is a go to for me whenever I really need a part to have strength. I have used this material twice now for 3D printed planetary gear skateboard wheels, and they held up to a LOT of impact. They wouldn't break no matter how hard I jumped on my skateboard, and they eventually cracked under the pressure of a few professional skateboards (which can be seen in a couple Braille Skateboarding YouTube video). Compared to an ABS set that broke the moment I jumped on my board, it is clear nylons have their place in 3D printing.

Finding the right elasticity, shore hardness, and tensile strength for your part may be difficult, but there seems to be a wide variety of nylon options on the market today. You just need to think about what you need for your particular application.

Carbon Fiber Reinforced and Filled Filaments

Carbon fiber reinforced materials are filled with continuous fibers or fiber particles that result in parts with improved physical properties and high stiffness. There is a variety of carbon fiber reinforced options out there for 3D printing, but they all require drastically different print settings. Because of this, I will not be going over particular slicer settings. In general, you will need similar settings to the material that Carbon Fiber is reinforcing. This means when working with Carbon Fiber Reinforced PLA, you will want to use similar settings to that of PLA. The same is true with ABS and Carbon Fiber Reinforced ABS, and so on.

Essentially, you will want to use carbon fiber reinforced materials when you require properties of the original material, but with a higher tensile strength.

Carbon Fiber Reinforced ABS has the high glass transition temperature of ABS, yet is much more stiff and strong. It is also easier to print. Carbon Fiber Reinforced Nylon is an awesome combination, one that is perfect for a wide variety of strength applications.

Personally, I have never found a use for Carbon Fiber Reinforced PLA, since it turns out more brittle than standard PLA (due to a lower elongation), but I am sure there applications for everything.

You will HAVE to work with hardened steel or ruby tipped nozzles when working with carbon fiber nylons, since the hardness will add abrasion and lead to your brass nozzle being worn out rapidly fast. A hardened steel or ruby tip nozzle will have a higher hardness than the carbon fiber blend and should not wear down your nozzle (or it will at least take much, much longer).

I am not going to include specific slicer settings for carbon fiber blends because they all vary drastically depending on what the carbon fiber is blended with.

Model Errors

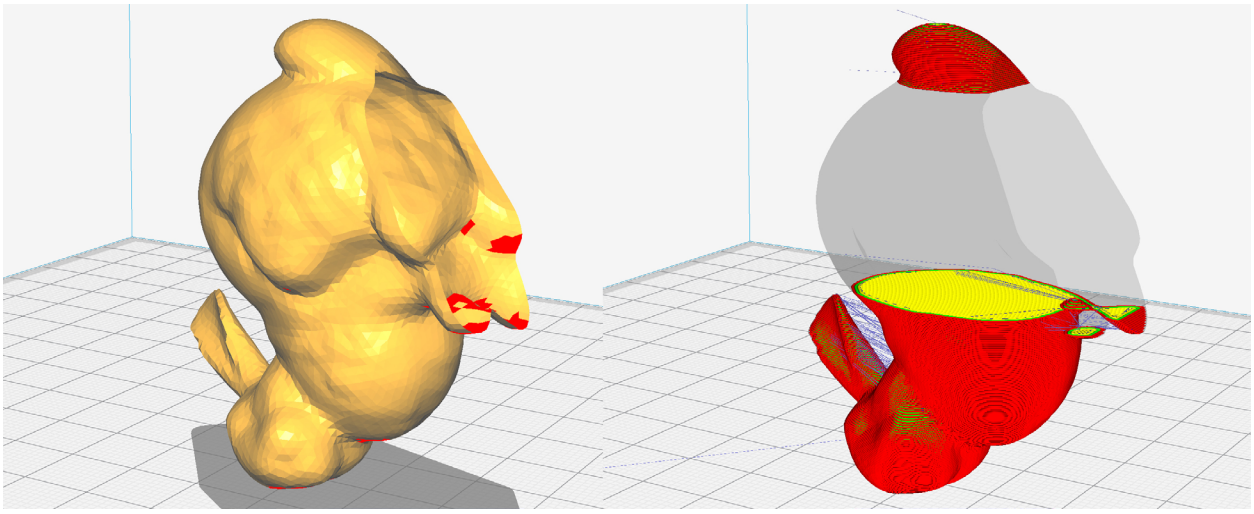
I am personally not a designer, nor am I extremely familiar with all the designing software out there, so I am not the best for giving advice when it comes to combining parts and properly exporting them in your preferred program. That said, there are some common issues that come into play with models exported for 3D printing.

Holes, one sided walls, and other model errors

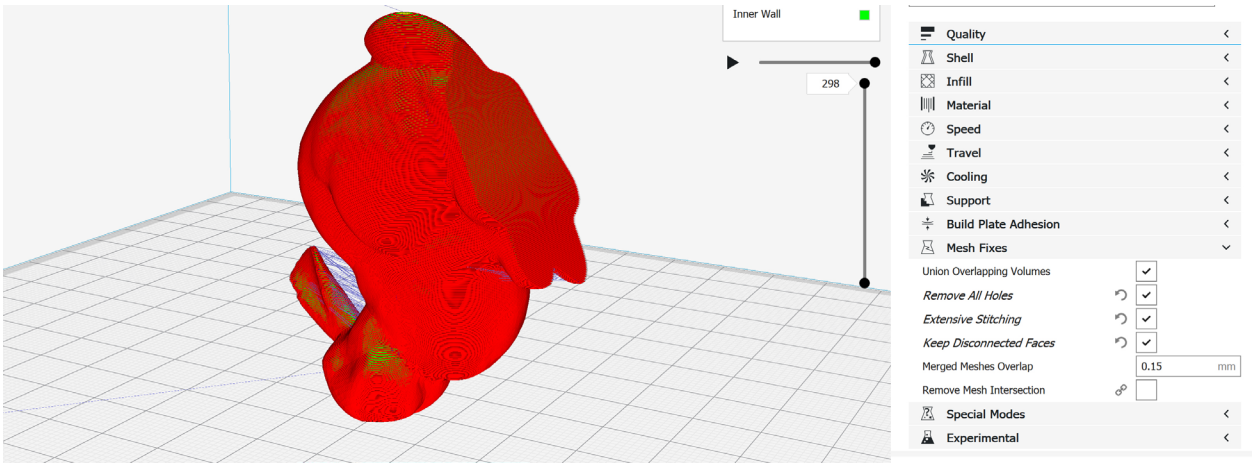
Other than the problems described below, there can actually just be errors within your model. This is extremely common if you used SketchUp to do your designing. I do not have the exact reasoning for this, but it seems to happen with .STL's exported from SketchUp more so than any other program.

Someone who takes all of the proper steps to design a model for 3D printing, including combining and exporting properly, should never experience a model with errors. But there are some free and paid programs out there that can help you diagnose and fix model errors.

I personally use the free version of Netfabb Basic. This actually isn't available any longer (without some hunting), but there is the ability to fix models via Cura. Below is an example of a model missing a face and being too thin to print since it is not solid.



When you open up your settings you are given "Mesh Fixes" as a settings option. You can play around here and go back and forth from layers view to see if your problem was corrected.



If you are able to download a previous version of Netfabb, this is also very easy. When you bring an .STL into Netfabb you will see a red exclamation mark telling when there is a model error. You would then click on the "Repair" button up on the top toolbar

Choose "Automatic Repair", execute the default repair, and then click "Apply Repair". This fixes the vast majority of model errors so that it can be 3D printed.

If you still see the exclamation mark after attempting this repair, you can also try out Microsoft's free online model repair tool at tools3d.azurewebsites.net. There are likely other free methods to fix models, such as using Meshmixer, but I personally do not use those.

If you are still experiencing issues, you may need to redesign or re-export the part.

Parts that are not actually 3 dimensional

A common practice for those who do not have much designing for 3D printing experience will be to take a 2D object and fold it around itself.

If you were to take a 2 dimensional rectangle and then fold it around itself so that it is a cylinder with an open top and bottom, the object will look 3 dimensional to the human eye, but will not be recognized as such by the 3D printer or slicing software.

This is because the object you are looking at has walls that are infinitely thin – they do not have any actual depth to them.

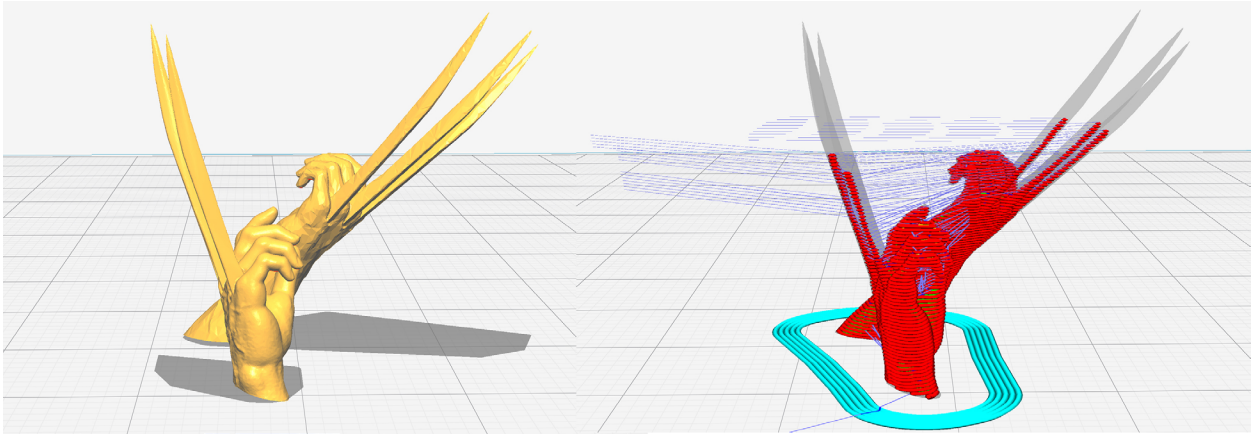
You will need to close off these objects and make them solid, or do some other designing practices that allow them to become actual 3 dimensional objects with solid walls.

Wall thickness

These next few may not be considered a technical error in the model, but essentially result in the same problems. You will always need to consider the thickness of the nozzle diameter you are using when designing the thinnest part of your print.

An FDM 3D printer can only print walls that are at least as thick as the nozzle diameter. I normally suggest at least 2-3x as thick as the nozzle diameter, but the absolute minimum requirement is the diameter itself (or the line width). So, if you were using a 0.4mm nozzle, but some of the walls on your print are 0.3mm thick, those walls will not be recognized, and therefore will not print.

Along with increased detail, a smaller nozzle will allow you to print walls that were not visible on nozzles with larger diameters. You can always see if your walls are recognized by your printer by checking the "layer mode" in your slicing program.



In the example above, the tips of Old Man Logan's claws are thinner than 0.4mm – the diameter of the nozzle I was using. Increasing the size of this part will allow you to print more, since the diameter of the blades will increase. If you are unable to print larger, you will need to use a smaller diameter nozzle.

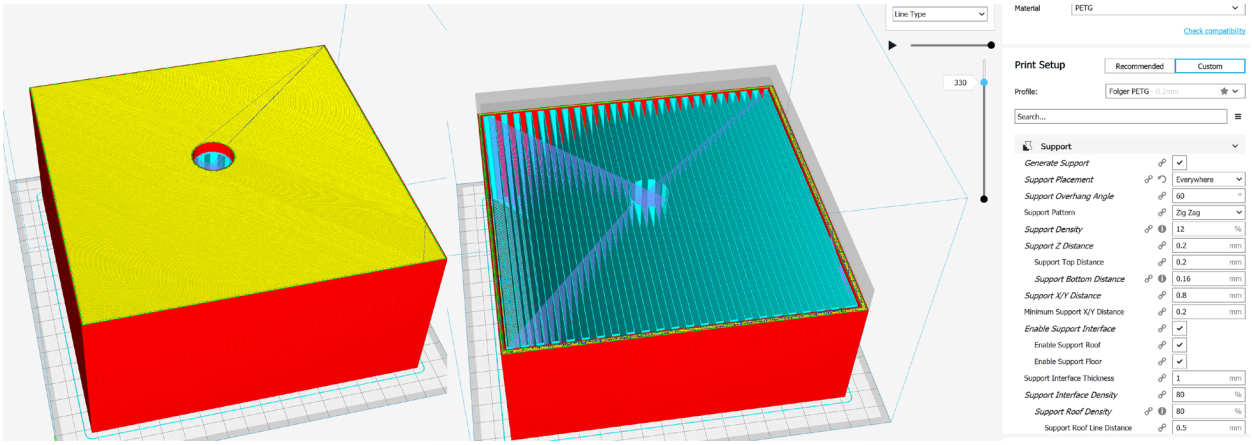
If something is thinner than the thinnest diameter nozzle available, you have a part that cannot currently be FDM 3D printed.

Unprintable models

A model can be considered unprintable for many reasons, one of which being it requires support material that is impossible to remove. This is also explored in the "Limitations Involved with 3D Printing" chapter at the end of this book.

If you were to design a hollow cube with a single small hole giving you access to the inside, you will likely never be able to print this model.

This is because a cube of any decent size will require either infill (not hollow), or removable parent support material. If your hollow cube does not give you access to the inside, you will not be able to get that parent support material out. The other option would be to attempt to bridge the large gap, but no printer or material will be able to bridge 100mm+. This is why this hollow cube may be possible if small, but just about impossible if large.



You can see this a bit better with the above example. It would be just about impossible to remove that support through the hole, but you wouldn't be able to bridge this large gap without support material.

While this basic model may have no actual errors, you will not be able to print it successfully.

Summary of Fixes and Precautions

- Try to avoid designing on Sketchup.
- Always combine parts and export them using the proper methods of the program you are using. Do not cut corners.
- Use Cura, Netfabb, Meshmixer, or a similar program to diagnose and fix printing failures due to poor models.
- Make sure the walls of your print are at least as thick as the diameter of the nozzle you are using.
- Inspect your print in your slicing programs "layer mode" to see what toolpath the printer will actually follow.
- Always take note if you have the ability to remove the support material that is being laid down.

Not Finding Home and Inverted Prints

When you start a print, your extruder and bed will go to its "home" position. It does this by moving the carriages until hitting their respective endstops. If there are any obstructions in the way of this process, or if there are any malfunctioning endstops, you can run into some serious structural issues.

The other way this can happen is via your firmware being set up improperly. If your printer is going to the incorrect location for home, you may also be experiencing inverted prints.

Clear axis's of any obstructions

Each axis and machine setup will come with different obstructions that can prevent your printer from finding home. When this happens, your stepper motor for that axis will continue to turn even though it cannot move any further. If this is on the X or Y axis, you may not experience any long term damage to your printer, but when on the Z axis you can have a nozzle that continues to dig into your build plate, resulting in issues to your frame, nozzle, and possibly even cracking your bed.

The most common obstruction I have experienced when dealing with a printer's X axis is when I leave the extruder idler open and start a print. This idler is just long enough to run into the printer's frame before engaging the endstop, causing quite a lot of noise when starting a print. You can also get an obstruction in the X axis from tangled filament or an extruder setup that is unable to engage the endstop.

For the Y axis, the most common obstruction seems to be wires and power cords. Since the build plate remains at ground level during a print on a Cartesian printer, and since it requires thick cords to power its heating, they can get in the way during and after a print. Throughout a previous print these cords may have gotten in the way of the build plate hitting its endstop, and you wouldn't know until you try to home or start a new print. You can also get a Y-axis obstruction if the endstop is unable to be engaged.

The Z axis really does differ on every machine and setup, but if you are using a mechanical screw to check your Z-height, there is a possibility it has been dislodged and not engaging the endstop. Confirm that the screw lines up properly with its endstop and that it is able to engage it to the point that the endstop clicks.

Clear any obstructions and confirm all endstops can be engaged when homing before moving on to the next solution.

Malfunctioning endstop

A malfunctioning endstop is one that does not send a signal to the controller (ie. light up) when engaged (pushed down until it clicks). This could be due to a broken endstop, or due to the wiring being disconnected.

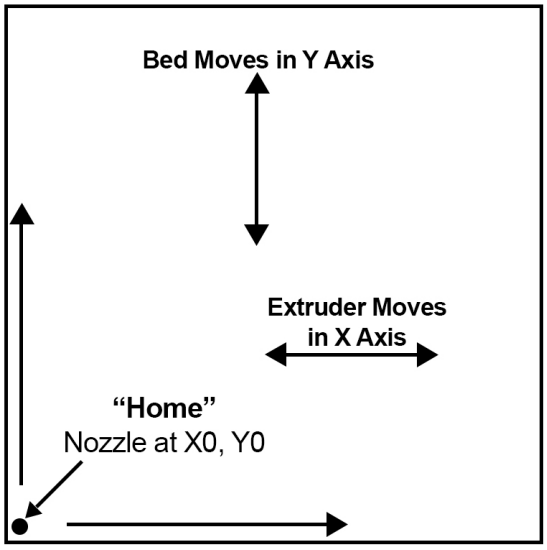
First confirm that all wires are plugged into the end stop and board, and that none have been pulled out from their connectors. Then check for any easy to see visual issues such as frayed or burnt out wires. If you do not see any, pull out your multimeter and check each wire for continuity, as explained in the "Stepper Motors Overheating or Malfunctioning" chapter. If one wire does not read out any noise or continuity, attempt to find where the break is, cut and solder, or replace the wire entirely.

If all wires show continuity, your endstop itself may need to be replaced. Luckily, these are very cheap and I recommend having a spare on hand to test and fix this issue fast.

Homing to wrong side or endstop on wrong side

These next steps should only really be an issue on a machine you are building yourself. If this is occurring on a machine that was sent to you from the manufacturer, it was likely their fault.

3D printers can have very different understanding of where "home" is depending on the manufacturer. For some - home is when the X carriage is all the way to the left and the bed is all the way back, others it can be the exact opposite. CoreXY printers will have the bed homing by either moving up to the nozzle, or down to a specific set point. Though this can vary, the standard for the majority of Cartesian (non CoreXY or Delta) machines is having the nozzle home to the front left side of your build plate:



If you have your endstop on the incorrect side for where your machine is homing, it will clearly never be able to find it. You will hear a lot of noise as the extruder or build plate continues to try and move and cause your stepper motor to skip.

If you are unaware of where "home" should be on your printer, it is a good bet that it is where your printer moves to when starting a new print. If you are sure that home should be on one side, yet your printer always moves to the other, you may have your settings incorrect in either the firmware or when slicing your G-Code. You will definitely be able to tell if this is the case if your prints are coming out inverted.

If your prints are not coming out inverted, it is likely easiest for you to just change the location of your endstop. If you would rather go the firmware route, open up Marlin when re-flashing your machine, and go to the Configuration.h tab. Scroll down until you see "ENDSTOP SETTINGS:" followed by each axis's home direction being defined by either 1 or -1. If the axis that is homing on the incorrect side for you, and it says 1, change it to -1. If the axis's that is homing on the incorrect side for you says -1, change it to 1.

```
// ENDSTOP SETTINGS:
// Sets direction of endstops when homing; 1=MAX, -1=MIN
#define X_HOME_DIR -1
#define Y_HOME_DIR -1
#define Z_HOME_DIR -1
```

Inverted prints

If your prints are coming out inverted, it is normally when one of your axis's is inverted via firmware, slicing program, or your motor is wired/setup improperly. You would really only need to change one of these regardless of which was causing you this issue. If you setup your stepper motor on the wrong side, so that it is turning your belt the incorrect direction, it will likely be easier to just invert this axis in your firmware rather than rebuilding that part of your machine.

To do this open up marlin to flash your machine. While in Configuration.h, scroll down to where it says "#define INVERT_X_DIR" followed by either true or false, and the other axis's below it. Whichever axis is inverting for you, switch the language (turn "false" to "true", or vice versa).

```
#define INVERT_X_DIR false    // for Mendel set to false, for Orca set to true
#define INVERT_Y_DIR false    // for Mendel set to true, for Orca set to false
#define INVERT_Z_DIR true     // for Mendel set to false, for Orca set to true
```

If you are currently homing properly, but your axis is inverted, you will want to change the language on both sections of the Configuration.h tab mentioned above. This will correct the inverted axis but still allow home to be in the same location, meaning you do not need to relocate any endstops. If you only change the invert, but not the endstop settings, you will be homing in the incorrect spot.

When finished, plug in your machine, choose the correct board and port, and upload.

Summary of Fixes and Precautions

- Confirm that there are no obstructions from any axis hitting its respective endstop.
- Confirm that the endstop lights up when engaged.

- If the endstop does not light up, check that all wires are showing continuity. Replace any that are not.
- If all wires show continuity, replace the endstop in question.
- If homing to opposite side of your machine but parts are not inverted, you can either physically move your endstop to the other side or you can change the proper language in Marlin.
- If homing to opposite side of your machine but parts are inverted, change the proper language in Marlin
- If parts are inverted but you are homing to the correct side of your machine, change both the language in both sections of Marlin mentioned above.

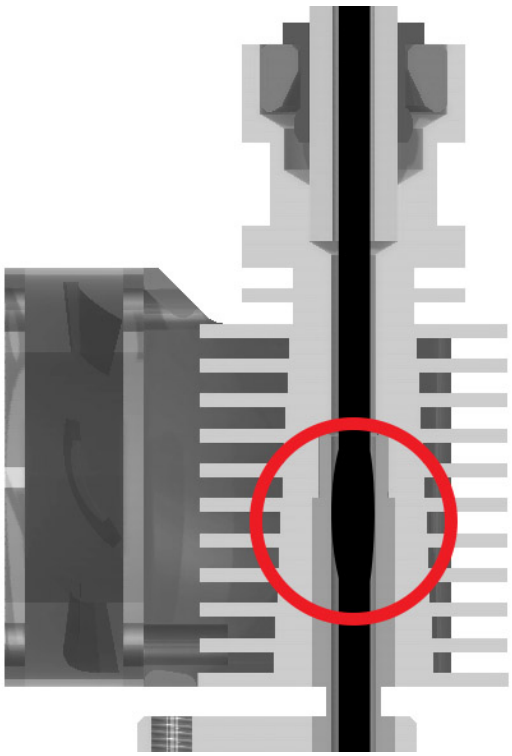
Nozzle Clogs

Nozzle clogs are just what they sound like and are a fairly consistent problem with FDM machines. They can take a very long time to fix and can normally not be diagnosed until the print has been going for a while. Most of the time these nozzle clogs will actually be clogs in the barrel, but we will still refer to this issue as a nozzle clog.

If the nozzle clog is frequently happening, even after doing these fixes, you will need to review the "Settings Issues", "Hotend Can't Reach or Maintain Temperature", and "Hotend Not Reading Correct Temperature" chapters.

IMPORTANT NOTE: Anytime you switch filament you will ALWAYS want to push the material down around 1 cm before removing. Some minor heat creep can occur when the printer is cooling down after a print and is only noticeable when you attempt to remove it. This practice can save you from an untold amount of barrel clogs.

Heat Creep (Ghost Printing Where You Cannot Easily Remove the Filament):



Heat creep refers to when heat creeps up the filament and causes unwanted expansion. If filament within the barrel gets too hot, it causes the plastic inside the barrel to heat beyond its glass transition temperature, and will often create a little bubble. This bubble will cause a ghost print and result in filament that you cannot remove.

If the heat creep is minor, you may be able to use a small Allen wrench to push the filament through until it makes it past this bubble, but you do not want to put too much pressure on your machine or else you will cause extra problems for yourself.

When the heat creep is more excessive, you will want to disassemble your hotend. If your filament hasn't already grinded to the point where it has snapped, you can break it off now. Disassemble as far as you can until you are left with the section that will not allow you to move the filament. This is

normally in the barrel, but can be in the hotend itself.

If in the barrel, you will want to remove all plastic parts that may be connected (fan, printed adaptors for your machine, Teflon tubing), and any wiring that you can. You will then want to get a lighter or a propane fueled torch. Be sure to not use a high temperature torch due to the possibility of it melting your barrel.

Before proceeding I must warn that this is dangerous and should only done at your caution. You will want to be in a very well-ventilated area and wear a mask for your protection. Do not attempt this procedure if you do not feel comfortable handling the following equipment.

Hold the barrel with a wrench or pliers that have rubber grips up to the flame (while wearing the proper gloves). If possible, pinch the filament in-between another set of pliers with your other hand. You can then carefully pull until the barrel gets hot enough that the filament will melt and the material will be easily pulled or pushed through. Clean out the barrel from remaining debris if you see any by pushing a small Allen wrench through.

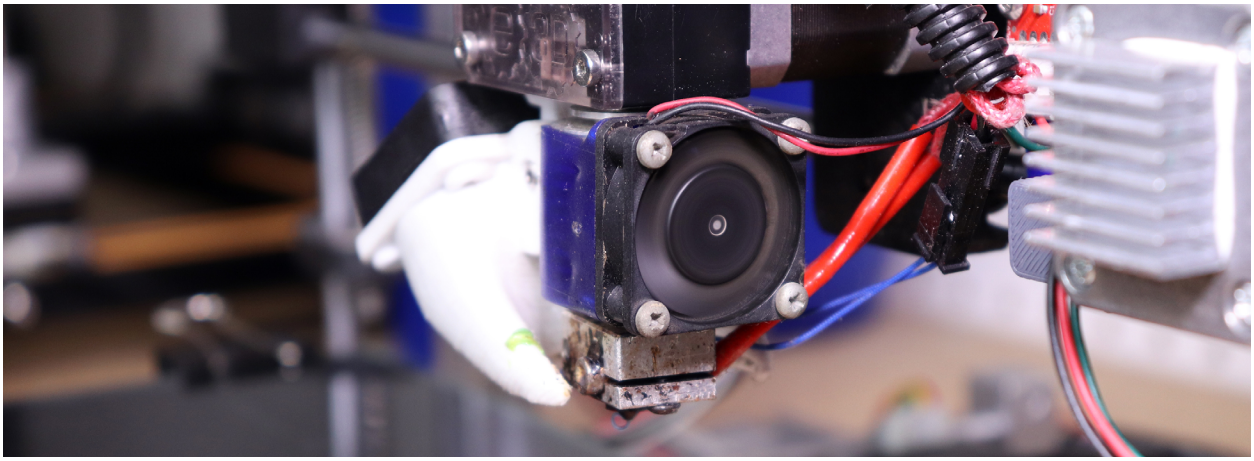
If the clog is in the unlikely section of your hotend, you will want to do this same process, only you must make sure all wiring is out of the way or removed. This type of clog is rarer because the hotend can normally heat itself hot enough to push filament through without the need of a torch.

Be sure to allow the barrel to cool all the way down to room temperature before reassembling. Once reassembled you can now move forward with future prints, but be sure to read on to prevent this from happening in the future.

How to prevent heat creep:

Since heat creep means that the temperature from the hotend is creeping up the material, there are a few prevention tips that can be tried.

Confirm your barrel cooling fan is on and not obstructed:



The majority of hotends will have a fan that blows on this barrel. I wire my barrel cooling fan directly to the power output on my board so that it is always on, something you may want to consider. If this fan is not blowing or is obstructed by debris when your hotend is heating, you will need to diagnose the problem.

First turn off and unplug your machine. Make sure your fan is of the proper voltage for your machine (12v/24v) and that all the blades are intact. If you have an extra fan of the proper voltage available, connect that fan instead and turn back on your machine. If it works, you can skip the next section since the issue was a burnt out fan.

If the above solution does not work, unplug and turn your machine off once again. Inspect the wiring of your fan. You may easily see that a wire has come loose or been disconnected. If you do not see a problem, you will have to use a multimeter tool to check the continuity of the wiring from your board to the fan. If one of the wires does not show continuity, you will either need to re-solder the unconnected section, or rewire entirely.

This is simple on many machines, but can be quite a headache on others. Having the proper connectors and ability to solder will make this process exponentially easier. Be sure you see continuity on the wire all the way from your board to the fan, there are no exposed wires, and turn your machine back on.

If the fan still does not turn on, there may be a problem with the section of the board that that fan is wired to. With the machine unplugged and off, wire the barrel fan directly to the power input section (red to + and black to -). Turn your machine back on, and if you are using a new fan and new wiring, everything should be working again. If not, reconfigure marlin and re-flash your machine for one final attempt before possibly needing a new board.

Links to where you can purchase fans and other accessories are explained in the "Important Accessories and Replacements" chapter.

Make sure you are not printing too slow or too hot:

This will involve you checking the settings on your slicing program. If you run your machine too hot, too slow, or a combination of the two, it will allow heat to move up the filament easier. Try lowering the heat a bit or speeding the print up to see if this helps with the problem.

Reference the "Settings Issues" chapter of this book for further help with tweaking your slicer settings for a more accurate print.

Final attempts to fix:

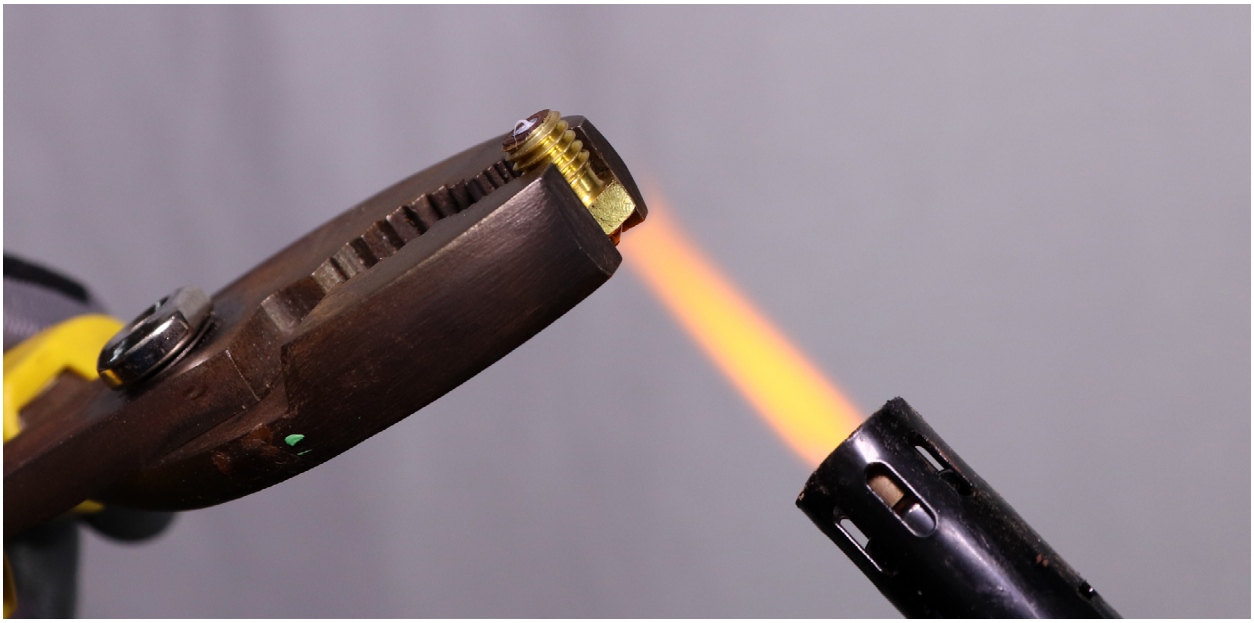
If you have done everything above and are still experiencing heat creep, you may want to invest in a new barrel or an entire new hot-end. I have personally used E3D hotends for a while, and while they do have an occasional heat creep, it is not a consistent issue. Your barrel may have been degraded over time causing the

diameter of the hole to be inconsistent. Or the company you may have purchased it from may not have the tolerances and quality that other products guarantee.

Finally, your thermistor may be reading the incorrect temperature. Your hotend may think it is 230°C, when it is actually running at 250°C. To fix this issue, please refer to the "Hotend not Reading Correct Temperature" chapter of this book.

Clog in the Actual Nozzle (ghost printing, can't push filament through but may be able to remove the filament):

A clog in the nozzle and not your hotend or barrel is normally caused by one of two issues: settings issues with the diameter of your nozzle, or improper purging of the previous filament. You will want to treat a clogged nozzle in the same way you would with a clogged barrel that I go into further detail above: Remove the nozzle and carefully apply a torch in a well-ventilated area until you can push the clogged filament through.



Too fast for the diameter of nozzle you are using:

It's easy to understand that the larger diameter of nozzle you are using, the easier it is to print without any clogs. E3D makes nozzles from as large as 0.8mm in diameter (up to 1.2mm in their volcano nozzles) all the way down to 0.15mm. I have personally used every diameter of nozzle they offer and can personally attest to the increase in nozzle clogs the lower you go. It took me roughly two dozen failed prints on their 0.25mm nozzle before I honed in the proper speed and retraction settings for this nozzle diameter without any clogs or issues. This meant that I torched out the nozzle quite a few times before I got everything right, each time

having to wait hours into the print before the issue showed itself. I repeated this process with the 0.15mm nozzle, and with each type of material. Once honed in, you should only use the same manufacturer, since each company you purchase filament from may require slightly tweaked settings.

There is no one step solution for this - you will need to manually tweak your settings until you achieve a print without any clogging. Refer to the "Settings Issues" chapter of this book for further detail on how you may be able to fix this yourself. You can always move up in nozzle diameter if you can't seem to get things to print properly. I have found that a 0.4mm diameter nozzle achieves very accurate prints with very minimal issues of clogs, and a 0.6mm nozzle is extremely easy to use and fast to print, but not the greatest when it comes to X/Y accuracy of text and other detail.

If you want to print really fast with large nozzle diameters – you will need a hotend that allows for this, such as the Volcano setup by E3D.

Filament Grinding (ghost printing and you can remove the filament and push it through again):

The filament will often grind to the point of snapping with the two possibilities above, the difference here being there is no heat creep requiring you to disassemble your machine. You can just use some pliers to remove the filament, reload the material, and push everything through without any clogs.

When this failure gets extreme, it can lead to you needing to read the "Stripped Filament" chapter. This is likely due to having too much moisture in the material.

If you are using flexible material, you may see the filament curled up before it grinds. Make sure you slow the print down if this happens, and confirm you have an extruder with enough torque for flexible material. Also confirm that there is a straight, clear path through the hotend. Bowden machines cannot print many flexible materials, even when using a strong geared extruder. The longer the tubing run between your extruder and hotend, the more difficult it will be to print with flexible materials.

While the cleanup is easy, you are still left with a failed print. If you just click "re-print", you will likely end up with the exact same results.

How to prevent filament grinding:

Loose idler tensioner, blockage, wrong hot end temperature, and too fast of retraction and print speeds are all common causes, yet easy to correct. The one that is the most annoying is from having too much moisture in your filament. Make sure you also read the "Stripped Filament" chapter in this book.

If you were to speed up your machine 10x what it normally prints at, you can

imagine how your extruder gear would be turning far faster than material can be pushed out the nozzle. This would cause the filament to grind until the point of snapping (if not just causing your extruder motor to skip).

Before clicking re-print, try slicing your part with both your print speed and retraction speeds slower. Watch the print and see if there are any sections where the extruder is moving too fast, and attempt to fix in your slicing program or by turning the knob on your LCD screen. The general rule of thumb is to limit the maximum print speed to 100x the nozzle diameter, printing at no greater layer heights than 75% of the nozzle diameter.

Next, adjust your idler tension. Start by loosening the idler and then feed filament through and tighten until you no longer experience slipping, but not so tight you experience grinding or stepper motor skipping. Filaments vary in diameter, so although the idler will absorb some variations, some material will require fine adjustments.

You should also check the temperature of your hotend and make sure it is not running too cold. If you try to print ABS at 200°C, you might be able to get some filament to extrude at first, but it will eventually get stuck. If slowing your printer down didn't help, try increasing the hotend temperature a bit.

Finally, as with most other problems, if you still are experiencing issues, re-flash your machine.

Summary of Fixes and Precautions:

- Push filament down before pulling it out.
- Don't keep material heated for long periods of time without printing.
- Make sure the barrel cooling fan is working and that all the blades are intact.
- Confirm wiring from the board to the barrel cooling fan is connected properly.
- Clean your barrel cooling fan and remove all dust.
- Check your settings that you are using the proper extrusion temperature for the material you are printing.
- In a well-ventilated area, torch metal parts when required.
- Make sure your thermistor is reading the correct temperature.
- Adjust the idler tension.
- Tweak the print speed.
- Confirm you understand what is said in the "Settings Issues", "Hotend Can't Reach or Maintain Temperature", and "Hotend Not Reading Correct Temperature" chapters.

- Read the “Stripped Filament” chapter if experiencing this issue frequently.
- It is smart to have replacement heater blocks, heaters, and thermistors so you do not need to wait for delivery to fix your machine when required.
- Re-flash if all else fails.

Over/Under Extrusion

Over and under extrusion is normally caused when your stepper motor is not turning the proper amount of steps, and results in either too much or too little plastic being extruded. This will lead to either ugly or brittle parts.

Keep in mind that a minor amount of under or over extrusion can be unnoticeable to the human eye, so it is smart to check this intermittently regardless. Unnoticed under or over extrusion can cause issue with parts that need to fit together or are used under significant loads.

**For most of your issues, you will want to skip down to the Check and Fix your E-Steps section of this chapter.

Over Extrusion

Over extruded parts will be fairly easy to tell either to the naked eye, or because parts are not fitting together. If you factored in your machine tolerances (which I go over in the "Parts not Not Mating Together" chapter) and parts are still not mating together, you likely are experiencing a bit of over extrusion.

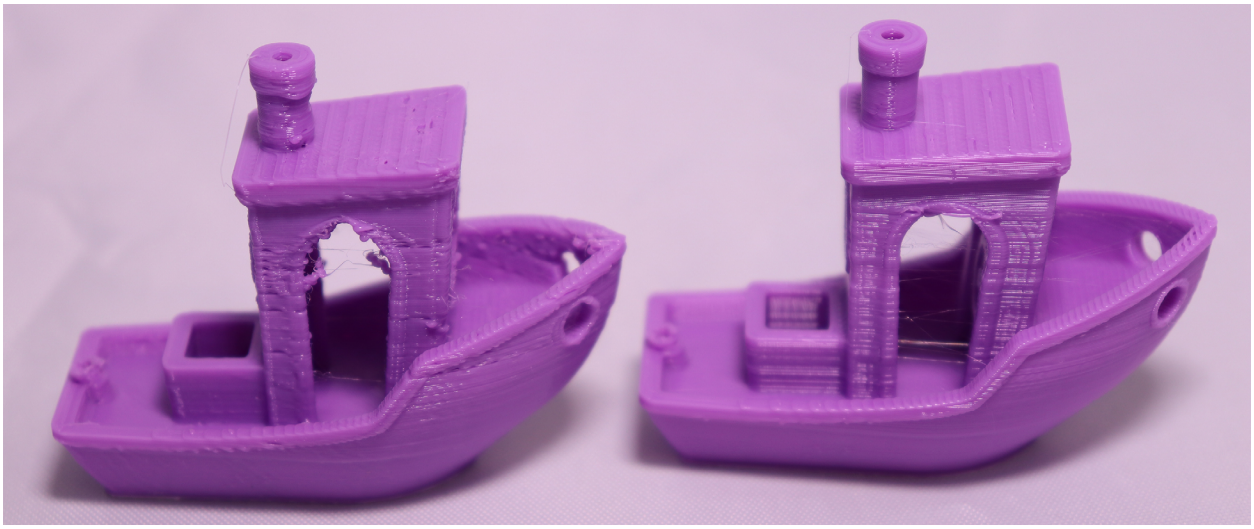
Check to see if nozzle is degraded

Nozzles can become degraded over time from material constantly running through them, as well as from times you started a print too close to the build plate. Some materials such as the Carbon Fiber Reinforced blends require a hardened steel nozzle to prevent rapid degradation.

When your nozzle degrades, the diameter will get slightly larger than it was when factory shipped to you. This can lead to some parts of your print looking as though there was over extrusion (or may just look ugly). You won't really be able to tell if you have this issue, unless your nozzle looks like the one on the left below. The nozzle on the right has been used, but it clearly has not been as degraded as the one on the left:



Normally a nozzle won't be this noticeably degraded, so you will want to keep track of how much printing you are doing on a machine. If you have gone 6 months of daily printing on a brass nozzle, it is definitely time for a replacement. It is smart to always keep 1-2 spares on hand to test if this solves your issues of over extrusion.



Above the two parts have the exact same G-code, but the one on the right is with a new nozzle. The part on the left was printed with a hardened steel nozzle, so even these wear out after hundreds of hours of printing with abrasive materials. While the print on the right is not perfect, you can see just how much a nozzle with the proper tolerances will help to achieve clean prints.

Before replacing your nozzle though, you should check your e-steps, as explained further in this chapter. It is also well worth the investment to get a hardened steel nozzle so you do not have to worry about this issue for a long time.

Under Extrusion

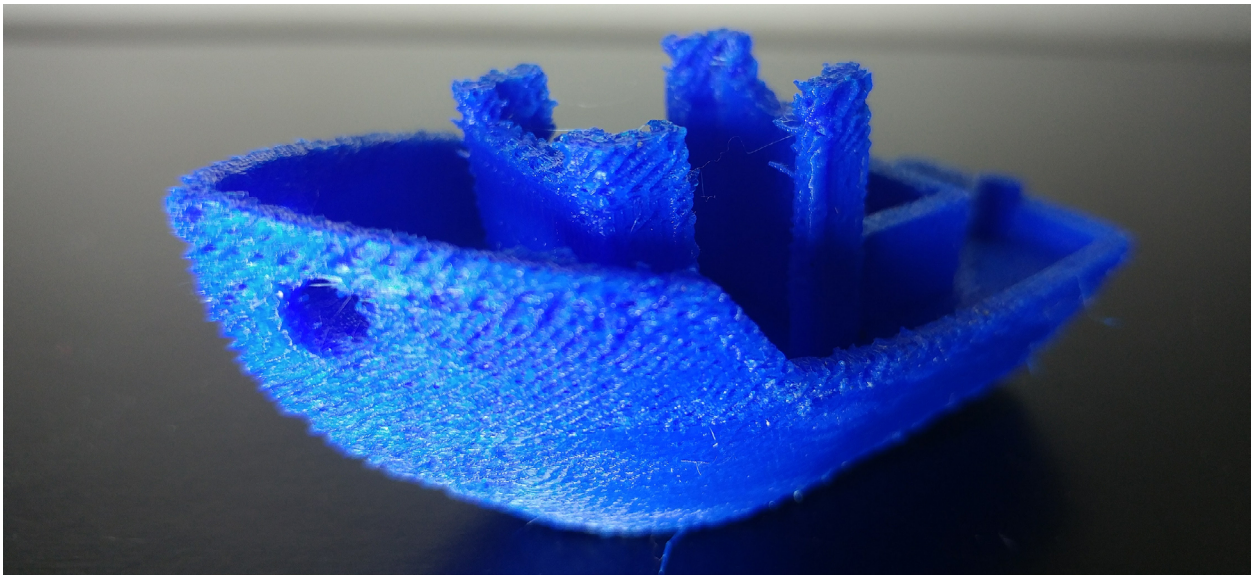
Unless extreme, under extrusion is much more difficult to diagnose with the naked eye. This is because parts will bridge gaps just fine, the surface quality will look good, and parts will mate together with ease. The biggest issue comes with the integrity of your parts. They will delaminate easier and will break under far less pressure than the part should be able to handle.

Check tension on extruder idler

Make sure there is enough tension on your filament by the idler that pinches the filament to your hobbed bolt/gear. When this idler is too loose, you may experience less filament feeding through the hotend than there should be. Too tight and you will experience further skipped steps in your extruder motor, or even stripped filament.

Not enough torque to extruder for nozzle diameter

When using a direct drive non-geared extruder on a 0.25mm diameter nozzle, I was experiencing what looked like massive under extrusion. I checked my e-steps and tweaked my slicer settings multiple times but could not fix the problem. What was occurring was minor skips in the extruder motor, and minor grinding of the filament.



As explained in the "Extruder Motor Skipping" chapter, I did not hear the normal clicking noise you hear from extruder motors skipping, but I was essentially experiencing the same issue. This is because a non-geared extruder does not have enough torque to push through such a small nozzle.

You will even see on E3D's website that they say a geared extruder is required to print using their extremely fine diameter nozzles. After upgrading to a Greg's Wade setup, I was able to print through a 0.25mm and 0.15mm nozzle without this under extrusion issue.

Confirming you have the proper filament diameter set

This sounds silly – but it has happened to me. Without thinking, I have had my filament diameter set to 2.85mm on Cura, but I was actually using 1.75mm. This will 100% result in a print that looks far under extruded.

This is an easy fix – just make sure you have this set correctly in your slicing software.

Reduce Retraction and/or Coasting

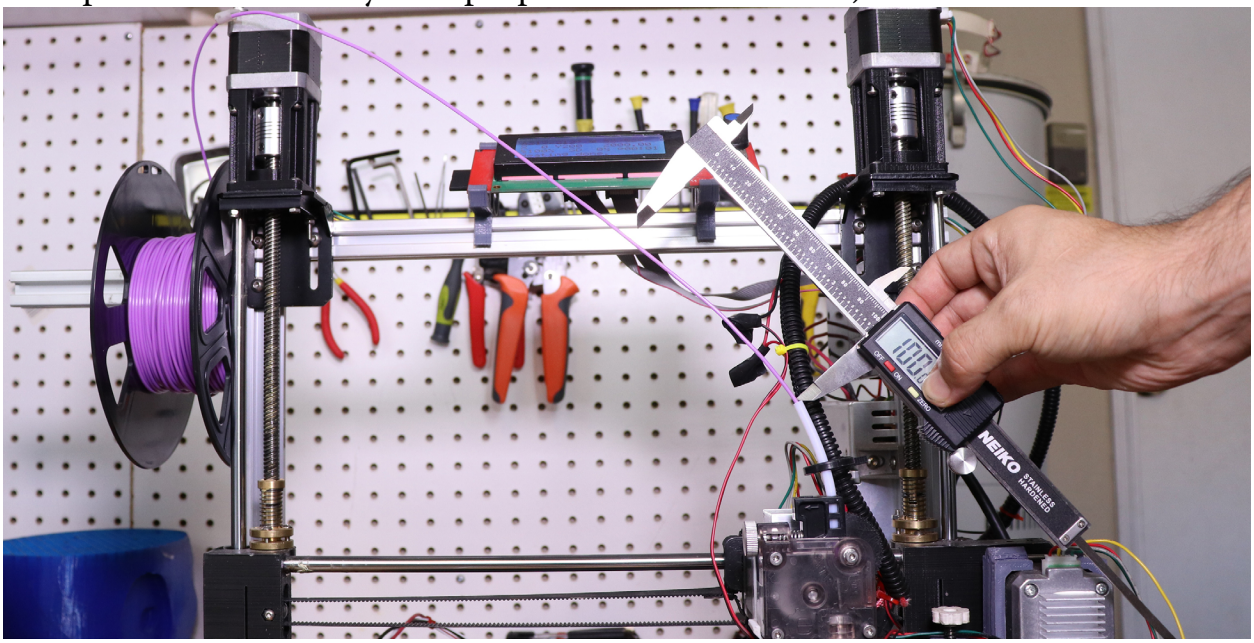
Having retraction set too low will result in stringy prints, but too high and you may

experience under extrusion.

Check and Fix your E-Steps

For most under and over extrusion issues, you will want to check and calibrate your e-steps. To do this is actually quite simple.

You will want to start off by measuring out 100mm of filament. I prefer to use White PLA because it is the easiest to write on, has a low printing temperature, and is cheapest - though you could use any material you have at your disposal (my example below is actually with purple AIO Robotics PLA).



You can do this in whatever method is easiest for you. I found it easiest to measure this 100mm when the filament is already fed into the extruder. You can also do this on a desk before feeding, but 3.00mm filament, and 1.75 near the end of its spool, are quite hard to keep from rolling back up.

Be as precise as you can by using a fine tip sharpie and holding the material as straight as possible. Use calipers if you have them at your disposal. If not – you can download an app to your phone, but you want to make sure you are as precise as possible. Being off by 1mm will result in your measurement being off by 1%. If you find it difficult to get an accurate reading, you can try increasing all of the 100mm distances to 200mm which will give you twice the resolution when reading your measurements. It will also take twice as long for the operation to complete.

After heating your hotend, you then want to push the filament down until the lowest dot you made lines up with the top of your extruder, or somewhere else you can easily line up the starting point (because you will need to compare it to where it finishes).

The next thing you will want to do is to tell your printer to extrude 100mm. This is done with a simple G-Code command in your terminal.

If you normally print via SD card, you will need to hook up to a computer for this. If you print via Octoprint or a similar online program, you can send the G-Code commands from their terminals.

When hooked up to Repetier Host, or whatever program you use to control your machine, and with your hotend heated to extrusion temperatures, you will want to give your machine the command:

`G92 E0`

This sets your extruder to 0. Next you will want to give either the command:

For 3.00mm Filament: **`G1 E100 F30`**

or

For 1.75mm Filament: **`G1 E100 F60`**

This will tell your extruder to feed 100mm, and is why it was important you lined up your starting dot with either the top of your Teflon tubing, extruder, or something similar, so that you can tell exactly if it has under or over extruded. If your 100mm dot lines up exactly with your starting dot after this, you know your E-Steps are right on.

Once your extruder has finished, you will want to mark your filament at the same spot you lined up your original dot (top of Teflon tubing in my example photo). If your 100mm dot lines up perfectly, then your e-steps are right on - but even 1mm means that your printer is extruding incorrectly by 1%. You could do this same test with 200mm or 300mm measured and fed out, so that you avoid any chances of minor measuring mistakes.

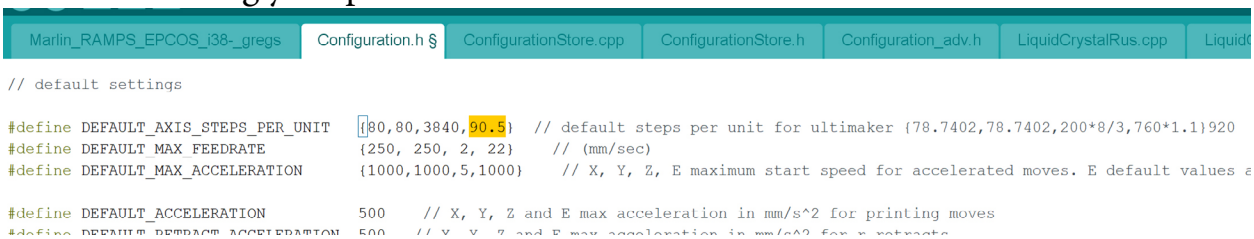
After marking the actual location of where the G1 E100 command lands, you will want to compare it to where you measured 100mm to be at the beginning of this process. If the 100mm mark is higher than the original, your printer is over extruding, if lower, your printer is under extruding. If the two dots line up you are perfectly calibrated.

After measuring this difference you will want to write down how much your extruder actually fed. If your printer over extruded by 2.1mm, you will want to mark down 102.1mm. If it under extruded by 2.1mm, you will want to mark down 97.9mm. You will need this number later on.

The next step in this process is to determine what your current E-Steps are. You can do this by either checking the firmware for your machine, by going into the "Motion" section of your EEPROM (LCD Screen), or just by giving it the command "M503" in your printer terminal. After typing "M503" into Octoprint, Repetier, or another terminal, you hit enter and will be read out your current E-Steps. Direct drive non-geared extruders have e-steps of around 90, while Greg's wade and other

geared extruders have e-steps of 500 or more.

If you are checking in the firmware that you use to flash your machine rather than giving the M503 command, you will want to open up Marlin. While in Marlin you will go to the "Configuration.h" tab and scroll all the way down to where it says "DEFAULT_AXIS_STEPS_PER_UNIT", with e-steps being the 4th and final number (if using one extruder). This is only if you plan on working in Marlin, rather than using your printer terminal.



You can also get this number by entering M503 into your terminal. You then take this number and multiply it by 100. You will then divide this new number by the number you wrote down earlier.

For example, if your current e-steps are 90.5, as shown above, you will multiply it by 100 to get 9050. We will then divide 9050 by how much you extruded earlier. So if you extruded 102.1mm, you will take 9050 and divide it by 102.1 to get 88.64.

90.5 x 100 = 9050

9050 / 102.1 = ***88.64***

88.64 in this above example would be your new e-steps. As you can tell it is lower than it was before, because in this example you were correcting for over extrusion.

You will now set your e-steps. You can do this through your terminal, EEPROM, or by flashing your firmware. If you are going to do this through your terminal you will want to give the M92 command, by typing M92 E88.64 and hitting enter. You will then want to type M500 and hit enter in order to save these settings.

While you can set this number on LCD screens under the "Motion" section, it will only save permanently if you have the option to save your settings after doing so. Otherwise your e-steps will reset once you turn your machine off. This is the same idea as typing "M500".

Thomas Sanladerer has a great tutorial video going over all of this on his channel which you can find by searching "calibrating your extruder" on YouTube.

Flash your firmware

Finally, you can reflash your firmware with your new E-steps. As mentioned in the "Tips if Still Not Working" chapter, you will want to reflash your firmware regardless periodically if experiencing non-stop issues.

You will do this by downloading the proper firmware for your machine if you

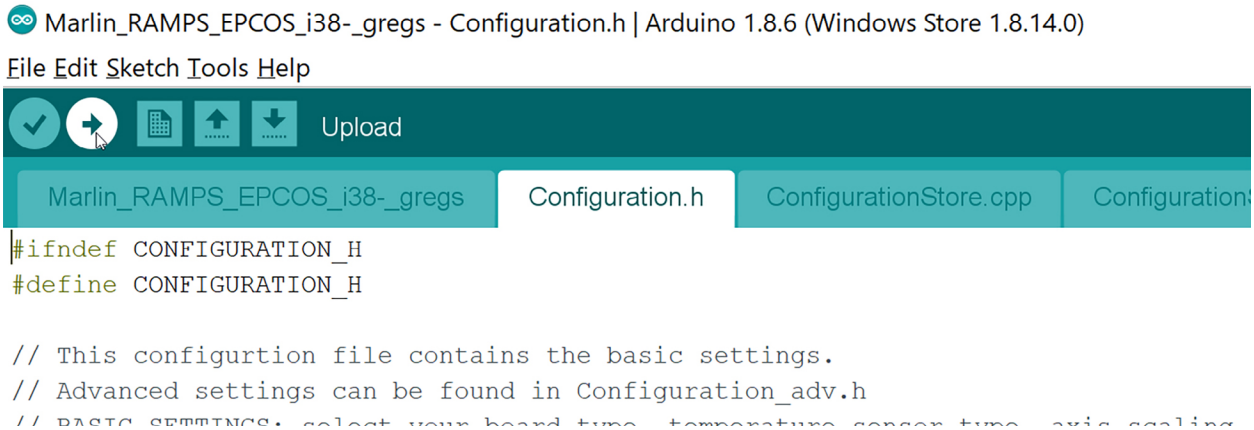
haven't already. You can normally find this firmware by searching for your machine online, but you likely already have it from when you first set up your printer.

Some printers, such as the CR-10 series, do not come with the ability to flash your firmware via Marlin. If you would like to do this on such a machine, you will want to bootload via an Arduino Uno. TH3D Studios has a great tutorial on this and ability to purchase all parts required (TH3DStudio.com). If you have a machine such as this, you can still change your E-Steps via a terminal, or as a last case scenario, edit the "flow" settings on Cura, as explained in the section below.

Double click the Arduino file to open up Marlin. If you are not changing your E-Steps, you can just proceed below to the next paragraph. If you are changing your E-Steps, go to the "Configuration.h" tab and scroll down to where it says "DEFAULT_AXIS_STEPS_PER_UNIT" as explained earlier. You will then change the fourth number to your newly determined e-steps.

Flashing your firmware is quite simple after this. Hook up your printer to your computer and then choose the proper port in Marlin by clicking "Tools", scrolling over "Port" and choosing the port to which your machine is hooked up, normally the highest number.

You will then choose the board you are using. If you are using a RAMPS board, you will choose "Arduino/Genuino Mega or Mega 2560". If not – you will need to check with what board you are using and what the manufacturer says. Once ready, you will want to click on the arrow "upload" button:



The code will then compile and upload to your machine. If you experience errors while compiling, read the specific error you are fed out. If could be something very simple to understand and fix, or it could be something much more complex.

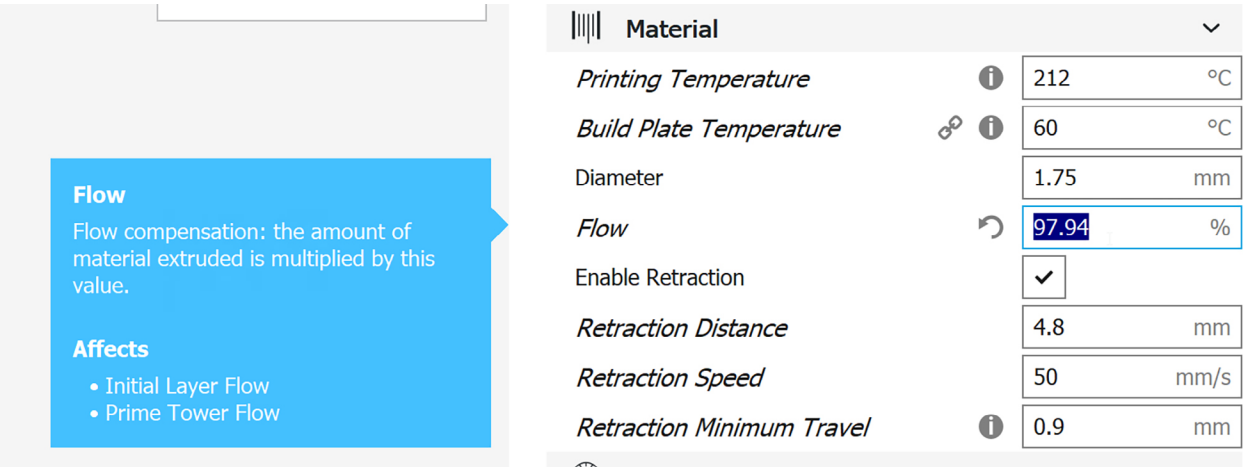
If confused, copy and paste the error message that is read out to you into a Google search. There is likely someone who has experienced and fixed the same problem. This is exactly how I fixed my issues with compiling when I was fed out an error that did not make sense to me.

Assuming you have the proper firmware for your machine, you shouldn't have these issues.

Changing Flow % in slicing software

If you have a machine that will not allow you to use Marlin and cannot save the settings using the M500 command, you could literally change the Flow % in Cura.

This is not ideal, but can get the job done if needed. Essentially, you will not need to know any E-Steps, you will just need to know the percentage that you over or under extruded in your test you performed earlier. If your extruder fed out 102.1mm, then you would divide 100 by 102.1 to be read out .9794. You can then set your flow percentage to 97.94%



Summary of Fixes and Precautions

- Step one should be to check your e-steps.
- Read the guide above on checking your e-steps or watch Thomas Sanladerer's YouTube video on "Calibrating your extruder".
- Correct your E-steps via your terminal, EEPROM, or by re-flashing your firmware.
- If you cannot change your E-steps via one of the three methods above, you can also change your flow %.
- Replace nozzle if degraded or if you have done countless prints on a standard brass nozzle.
- Check the tension on extruder idler.
- Confirm you are giving enough torque for the nozzle diameter you are using.
- Confirm you have the correct filament diameter set in your slicer settings.
- Understand the possibility that your hotend is not reading the proper temperature as mentioned in that corresponding chapter.

Parts Being Knocked Over

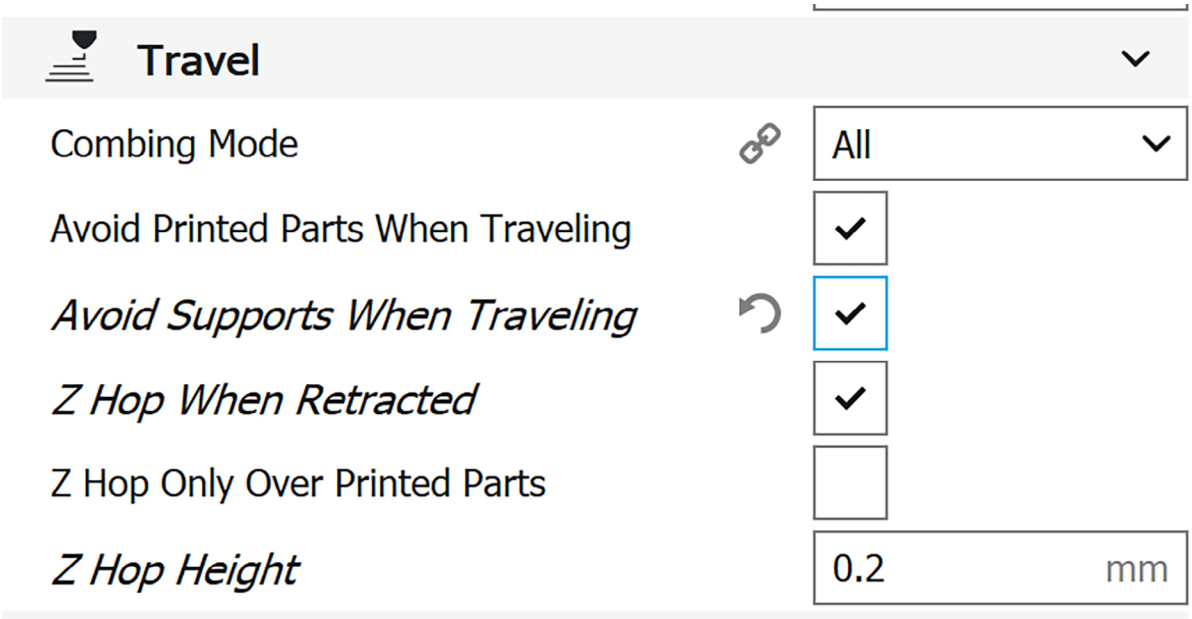
Your part being knocked off the build plate can happen quite frequently, especially when dealing with tall, skinny prints.

Proper bed adhesion

Goal number 1 when it comes to having parts not being knocked over is having the proper bed adhesion. This includes knowing the right mixture or bed sheet for the material you are using, as well as having the proper z-height with a level build plate.

To review this further, make sure you check the "Bed Adhesion", "Unlevelled Build Plate", and "Z-Height" chapters. If any part of your print is too far from the build plate, you will be very susceptible to your part being knocked over.

Adding a Z-hop



As mentioned elsewhere in the book, a Z-hop refers to the printer head moving up while travelling. The printer head (or build plate when working with a CoreXY machine) will have the nozzle move away from the print by the determined amount in your slicer settings. This will mean that the nozzle will not be running into your print when travelling, reducing the odds of it being knocked over.

I always have my Z-hop height set to the layer height of my print. As you can see from the photo, it is currently set to 0.2mm on this profile, meaning my layer height for this print is 0.2mm. I have just found this to work best, because if you have a Z-hop set lower than the layer height, you run the risk of still bumping into the top of your print.

Avoid parts when traveling

As you can see in the photo above, I also have both the avoid supports and avoid printed parts when travelling checked. This is a fairly new feature offered in Cura, in which the path of your printer head will avoid both printed parts and supports while moving. This can increase print time, but I find it best to use to avoid any parts being knocked over.

To be honest – I am not quite sure if both Z-hop and this feature are required, or if one cancels out the need for the other. I figure I have had so many parts get knocked over in the past that it would just be best to take all precautions to prevent this.

Working with very thin tall prints

This is also covered in the Z-Wobble chapter, but essentially a very tall skinny part is far more likely to wobble and get knocked over during a long print. This should not be an issue on CoreXY machines where the build plate only moves downward, but on Cartesian machines where the build plate rattles back and forth, the very top of a tall skinny print will sway back and forth.

This swaying will not only cause ugly Z-wobble prints, but it can cause a print to be knocked over. This is because the nozzle may try and start the print slightly to the side of where the top of the print swayed to. Get this happening a couple of times and this part will easily fall over.

This is especially noticeable when working with some support material. Often a part will require a very tall support structure, but may not need to cover a large surface area. These thin towers can fall over before they get to where you need them to be.

Unfortunately, when working with a Cartesian machine, there isn't much that can be done to prevent this. Of course you need to have a proper brim for great bed adhesion, but if a part is extremely thin and tall, there will be swaying. There are essentially two things you can do in this scenario – cut the part into two sections or add further scaffolding to help anchor.

There have been some thin parts I have printed in the past that I was just forced to slice in half and glue together post printing. This is clearly not ideal for mechanical parts, but anything this thin will be flimsy regardless.

You can also add manual support structures to the model, anchoring the part to the build plate every so often, to make sure no swaying occurs. If you are just building a thin tower, you will need support structures attaching to the side. This can be done in programs such as Cura.

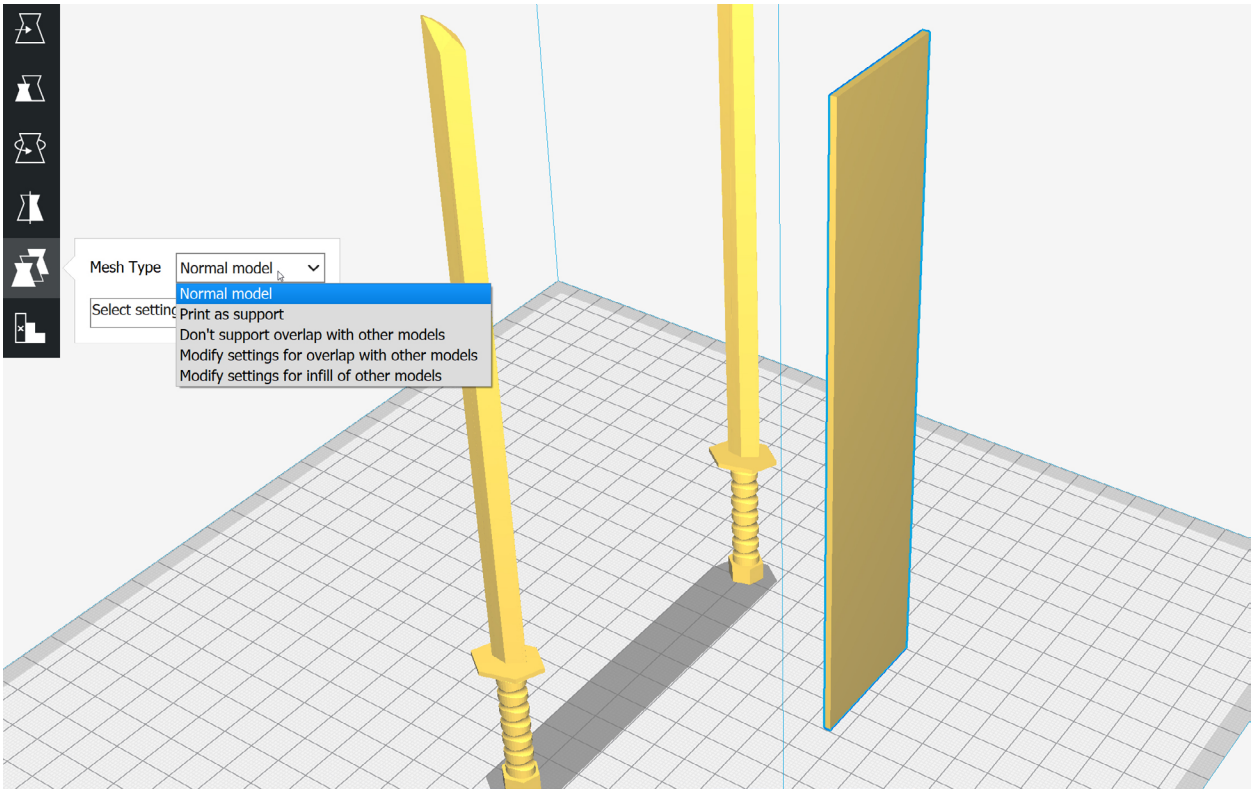
Anchoring Prints

I am positive there are better ways to do this, but the easiest way would be to anchor

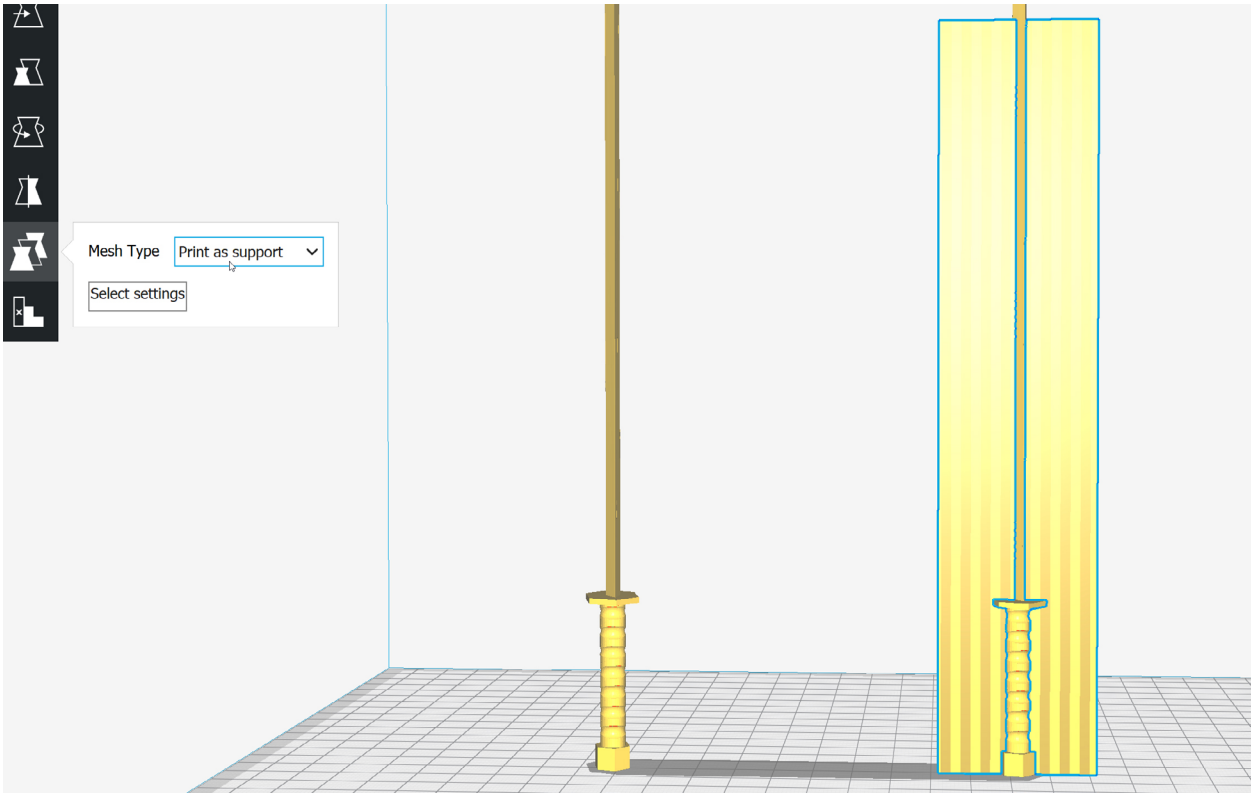
your print in Cura. I have rarely done this, but it definitely helps with a tall, skinny print wobbling back and forth. This is also explained in the Z-Wobble chapter.

Below is an example of two skinny swords from a Deadpool print that I made for my YouTube channel. When not adding any anchors, my Cartesian machine would wobble the build plate back and forth and cause the top half of these swords to look extremely ugly (if they didn't get knocked off).

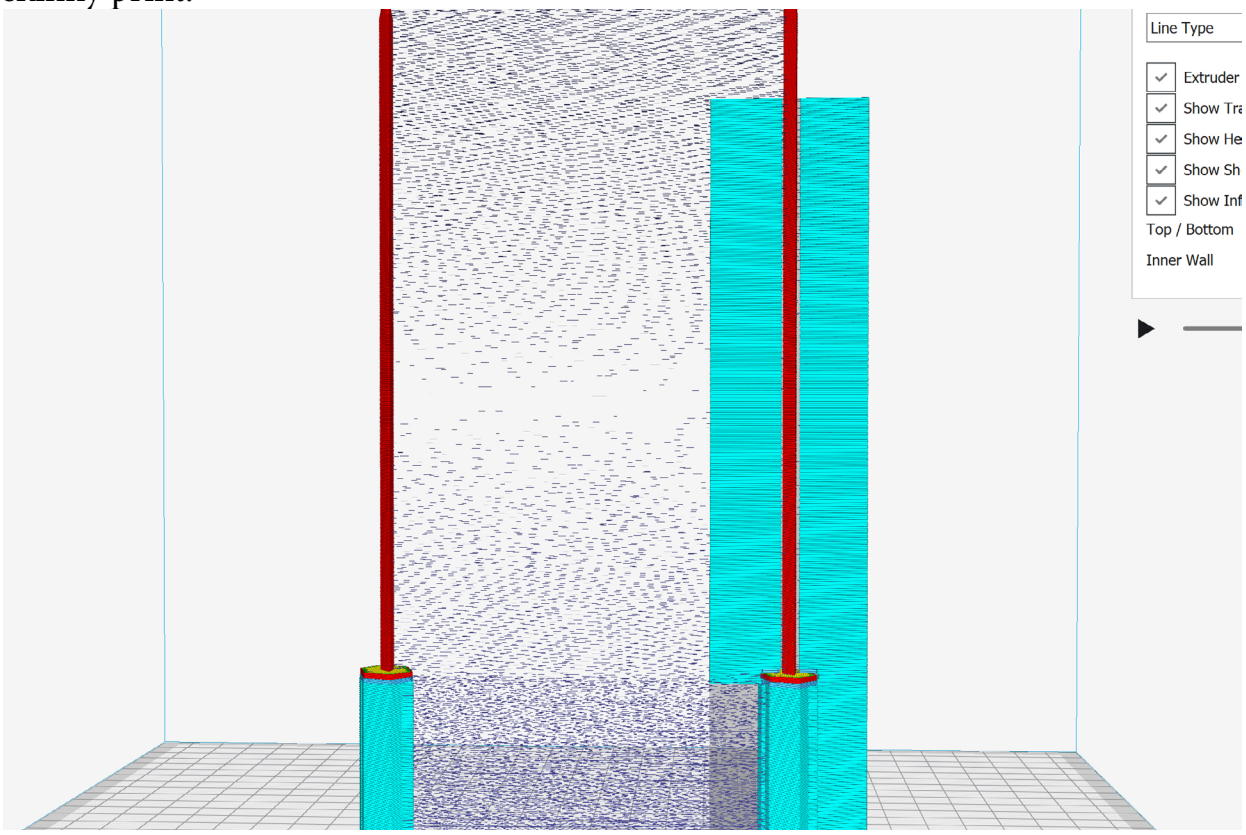
Cura now allows you to bring in a second model that intersects with your main print. They also allow you to print a part entirely as support. This means you can drag in a second object that acts only as support for your main structure.



This rectangle in the example above is thin, so that it won't take up too much material, yet will extend the anchoring for the sword (I added a second sword to compare how it will slice). After bringing in a shape that will work for your model, you can choose the model and click (Print as support)



After turning the shape into “Print as support”, you can then drag it over your tall, skinny print.



As you can see in "Layer Mode", this entire shape is now support structure that can help to anchor your tall skinny print to help prevent this wobbling back and forth.

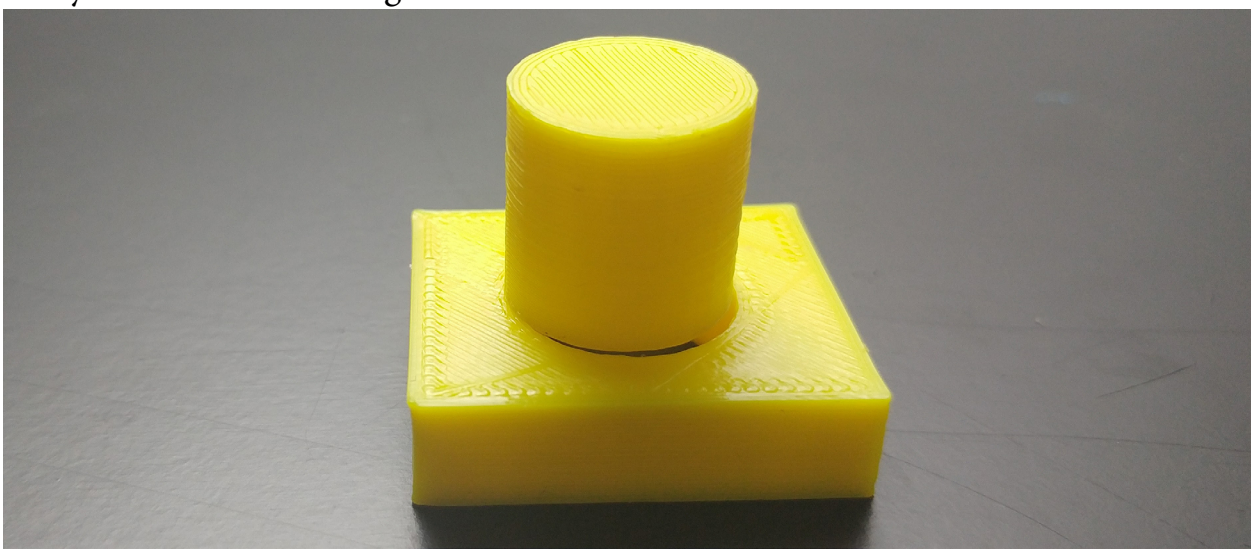
As mentioned – there are many other ways to do this, this is just the simplest way since it allows you to do this right in your slicing software.

Summary of Fixes and Precautions

- Ensure you have the proper bed adhesion – including having a level build plate with the proper starting Z-height
- Add a Z-hop (normally the same distance as your layer heights)
- Utilize the ability to avoid parts and support when printing in Cura's slicing software
- Plan ahead when working with a tall and skinny print. You may need to cut the part in half or add anchoring to make sure the part doesn't wobble back and forth

Parts Not Mating Together

This issue can be as minor as that when you print two parts that are meant to fit together they don't without some sanding, to as far as one or multiple axis's being off by over 100% in scaling.

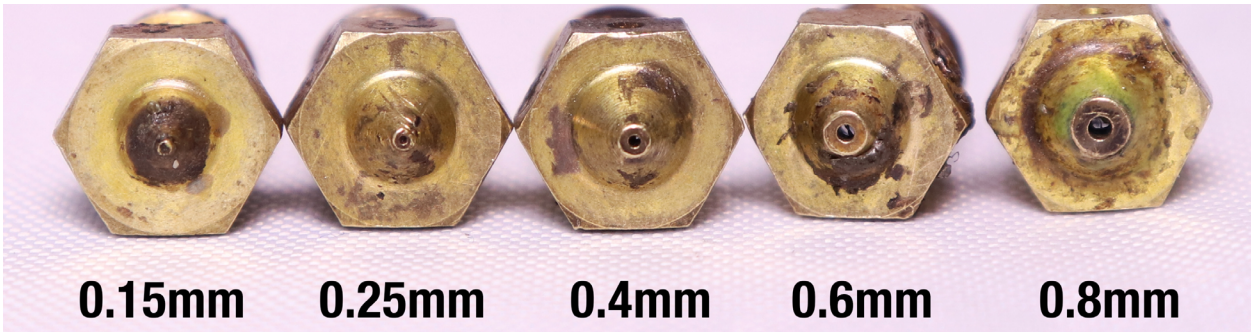


You will have to diagnose how bad this problem is. If it is minor, proceed with the next couple of steps.

Confirm you factored in the appropriate tolerances

There are clearly tolerances involved with 3D printing, and they can vary drastically on the nozzle diameter and layer heights of your particular print. While there may be sources that say the tolerances are tighter than the numbers I am saying, these are what I use and what has worked for me throughout all of my prints.

The tolerance of the print that I will use is the layer heights you are using in the Z-direction, and half the diameter of your nozzle in the X/Y direction. So, as an example, if you were printing 0.1mm layer heights on a 0.4mm nozzle, your tolerances will be roughly 0.1mm in the Z-direction and 0.2mm in the XY direction.



Since 3D printing is additive, it will almost always err on the side of adding more material than less material. This means that parts will be slightly larger and holes will be slightly tighter.

So, if you were to design two parts that mate perfectly together without any clearance, no matter how good your settings are, you will likely not be able to fit the printed pieces together without a lot of sanding.

I always suggest parts that need to be perfect to their dimensions, or ones that need to mate together, have the size of their holes increased, and the overall size of the part decreased.

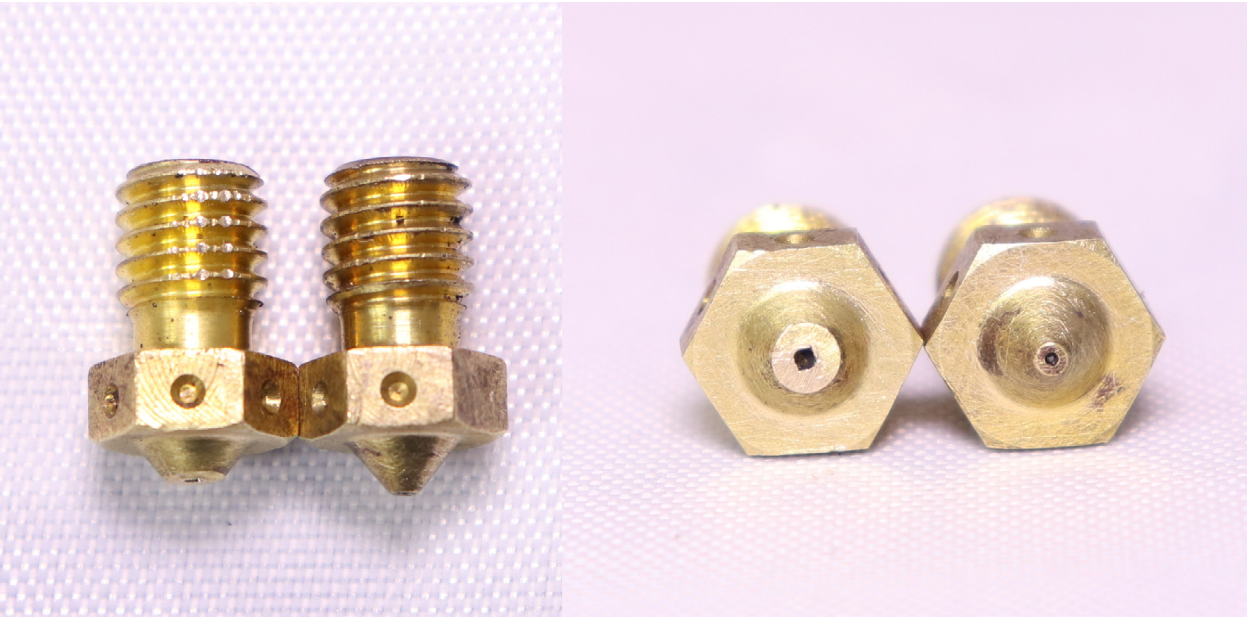
For instance, using the same example as above, if you are printing at 0.1mm layer heights on a 0.4mm nozzle, you will want to increase the diameter of holes being printed in the XY direction by 0.2mm, and decrease the size of the part in the XY direction by 0.2mm (and 0.1mm in the Z direction).

With this tolerance, parts will mate tightly together. If you would like the fit to be a little loose, you will want to increase this to 0.3mm in the above example.

This is another reason printing in a high resolution on a fine nozzle is beneficial for printing a part with accurate dimensions, but will of course result in a much longer print.

Replace your nozzle

Your nozzle will be degraded over time, especially when using one made of brass. Abrasive materials at hot temperatures will slowly make the diameter larger than what you think it is.



The example above is an extreme version of what I am talking about. The nozzle on the left had been worn out from being used when too close to the print bed, and

from being used for a couple of months (not from my machine). Not only is the tip extremely worn down, the hole itself is clearly larger and not as precise.

While you are trying to lay down 0.4mm layer lines, your nozzle may actually be closer to 0.6mm in diameter. This can lead to ugly parts, and can definitely lead to your parts not mating together. If you are using a brass nozzle, make sure you replace it every couple hundred hours of printing. I definitely suggest upgrading to a hardened steel nozzle so this does not become an issue nearly as quickly.

Check to see if you are over extruding

You will definitely want to visit the "Over and Under Extrusion" chapter in this book if you factored in tolerances but parts are still coming out slightly too large.

Be sure you are as accurate as possible when checking your E-Steps because even a slight over extrusion can cause problems when you are trying to print two parts that are meant fit together accurately.

Tighten belts

As mentioned elsewhere in this book, it is possible to overtighten your belts, but you will really have to be trying to do that. If you are experiencing parts that are not to the correct dimensions your belts may be too loose.

You do not want a lot of slack in your belts because not only will quality of your parts decrease, you can experience actual dimensional issues. This happened slowly over time on just about every machine I have used. A part printed on one machine over time would not mate with a part printed on another.

It turned out that tightening the belts fixed the issue immediately. This is another reason having a way to easily tighten your belts will be extremely useful and necessary for practical preventative maintenance.

Confirm stepper pulley has proper amount of teeth

This is a strange one and should only occur if you recently changed the pulley attached to your stepper motor. This happened to me once, and it took me about 3 prints of confusion to realize what was going on.

If your stepper pulley has the incorrect amount of teeth for your machine, your scaling for that dimension will be very far off. From what I can tell, the vast majority of FDM machines use 20 teeth pulleys for their stepper motors.

Read the *Settings Issues* chapter

If you have confirmed everything above, and parts are not mating together, make sure you read the "Settings Issues" chapter. You would be amazed just how much tweaking your settings can affect the quality of your print. Along with making sure you don't have your flow rate over 100%, minor fluctuations in temperature for specific materials can affect the viscosity which can cause a print to ooze more than

it should. This will result in a part that is not to the proper dimensions.

Print a large calibration cube and check XYZ steps/mm



This really should not be needed if everything is built properly and you are using the correct firmware for your machine - but it may be smart to do on very large machines.

Find a calibration cube on Thingiverse, or create your own that is large and uses the least amount of filament as possible. This can be a hollowed out cube that is only shells and has not top or bottom, or whatever you can think of that has specific dimensions of at least 100mm. If you have the filament, space, and time, you should go even larger.

This is because the tolerances involved with your machine will effect these numbers the smaller the part you print. This process would then be the same as when you checked your E-steps.

I cannot stress enough how important it is to check your E-steps and to check everything else described above before printing your calibration part. There is no point changing the X, Y, or Z steps/mm if a belt is loose or if you are over extruding. Your X,Y, and Z-steps should be a calculation based off of the parts you are using – and there is no reason to tweak this UNLESS you built the machine yourself and you were wrong with your calculations in flashing firmware.

Find your X, Y and Z steps per mm in your firmware – they are located directly to the left of the E-Steps/mm (as explained in the “Over and Under Extrusion”

chapter). Take that number and multiply it by the number of mm your calibration print should be in that dimension (if printing a 100mm cube, multiply by 100). Pull out your calipers and measure your print in that direction for what it actually printed.

Make sure you do not scrape off your print before marking or remembering which direction is which. Divide that newly found number by the actual number read out by the calipers. This is your new steps per mm for that dimension.

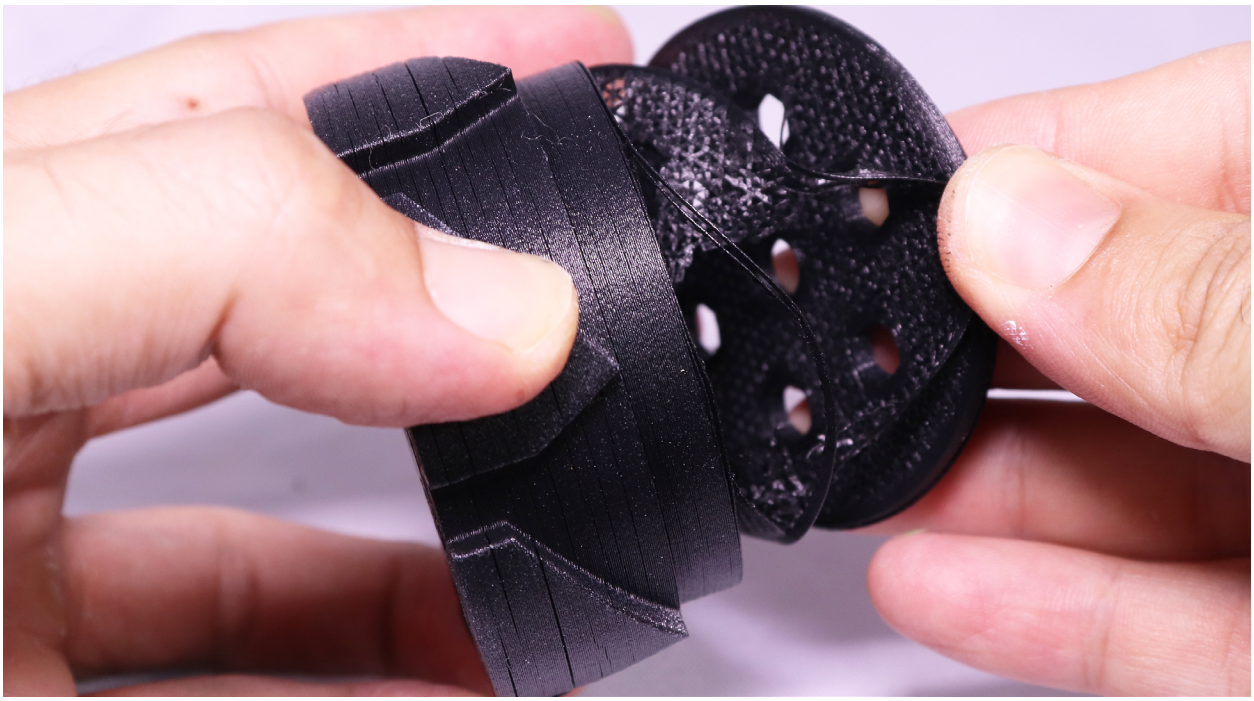
As mentioned, if you have a printer that is using the proper firmware and parts, with settings that are correct for the material you are using, this should generally not be needed.

Summary of Fixes and Precautions

- Make sure you factored in clearances for parts that need to be mated together or precise. These tolerances have to do with your layer heights and nozzle diameter.
- Replace any worn out nozzle.
- Make sure you are not over extruding by checking the E-steps.
- Tighten all loose belts.
- For parts that are far off in their accuracy, confirm you have the proper amount of teeth for your stepper motor pulley for your machine/firmware.
- Read the "Settings Issues" chapter since many issues in minor dimensional accuracy have to do with having the proper settings for the material you are using.
- For large machines, or machines you do not have the firmware for, you may want to check the X,Y and Z steps/mm by printing a very large calibration cube. Only do so after confirming all of the above.

Poor Layer Adhesion

Having strong layer adhesion is not only mandatory for watertight parts, but it is needed for clean, strong prints. If your individual layers do not stick together well, you are bound for a part that will break and peel apart.



Understand the Material Being Used

Each material requires its own settings, including temperatures and speed. You need to make sure that you are using the proper settings for the type of material you are using.

Refer to the "Materials and their Settings", as well as the manufacturers print settings, before moving forward in trying to fix this problem.

Increase the Extrusion Temperature

One of the most common reasons this happens is because you are printing at too low of an extrusion temperature.

Following the manufacturer guidelines is normally a sure fire way to prevent this, but a few times I have had to go above these recommended settings to make sure I had a strong enough layer adhesion.

The photo above was a failed print I had when printing in NylonX by MatterHackers. They recommend 250 – 265 degrees Celcius for printing, so I went ahead and tried 250 degrees. I was left with the part you see above, something with

far too weak of layer adhesion. After upping this to 265 degrees the part printed just fine.

Try slightly increasing your extrusion temperature to see if it helps with this problem.

Under Extrusion

Another reason for poor layer adhesion is under extrusion itself. If your extruder is depositing less filament than it thinks it is, you are bound to have weak layer adhesion.

Confirm you have the proper E-steps set by referring to the "Over and Under Extrusion" chapter in this book.

Not Enough Torque

As with the "Extruder Stepper Skipping" chapter, you could be working with a stepper motor/extruder setup that does not have enough torque. If your extruder motor skips, you will essentially be left with an under extruder or poor layer adhesion print.

The best way to remedy this is to upgrade to a geared extruder, if you haven't already. You can do this via printing a Greg's Wade extruder or by purchasing one. The two that I use are the Titan extruder by E3D, and my current favorite is the Bondtech BMG dual drive extruder.

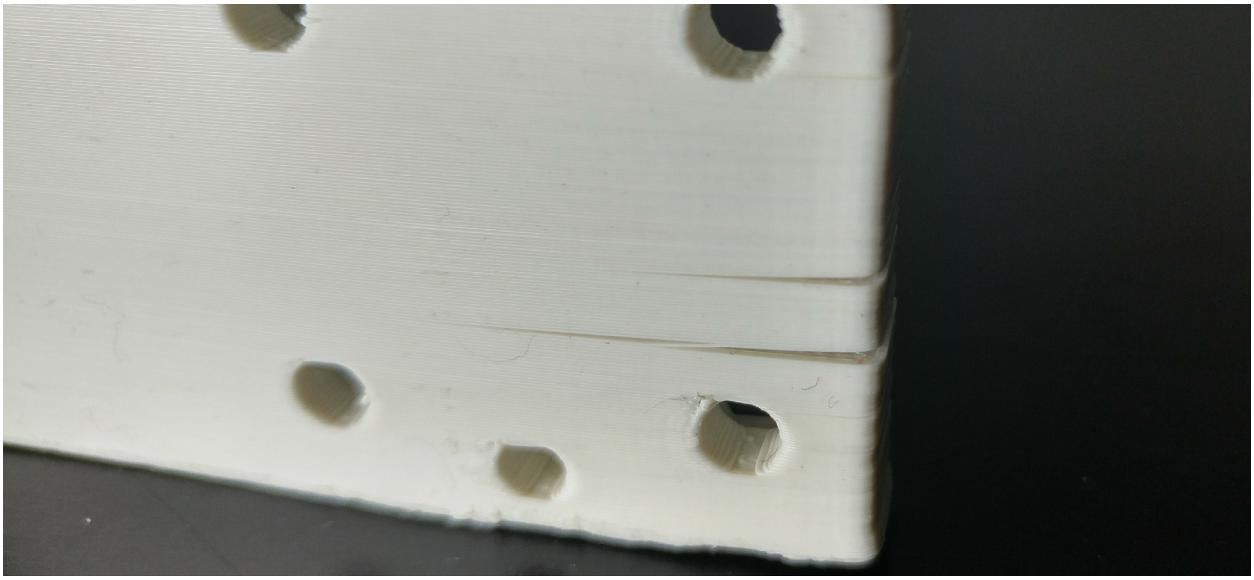
Turn Off Active Cooling Fan

While having your active cooling fan turned on will benefit the majority of prints and materials, many filaments require you keep this active cooling fan off. Another reason the print from the photo above failed was that I kept my active cooling fan on. It seems that MatterHackers states their NylonX should be printed without any active cooling fan. This not only helps to prevent warping, but will also help to achieve strong layer adhesion. Generally the higher the heat capacity and density of the polymer, the more beneficial a cooling fan will be. For lower density polymers such as ABS, HIPS, etc. that are below 1.2g/cc, the fan is typically recommended to be turned off.

Make sure material is stored properly

As with many other problems, make sure your material is stored properly. Old, wet material, or poorly made, off-tolerance filament can lead to issues that are very difficult to diagnose. If you consistently get poor layer adhesion from one spool regardless of what you do, try using a different spool. If the issue does not continue, it is likely problems with that particular material.

Delamination



I personally consider delamination a different problem than just poor layer adhesion. You can have extremely strong layer adhesion but still experience delamination when working with high warping parts.

Because of this – I cover this specific issue in the "Warping" chapter, since delamination is far more a symptom of high warping materials rather than poor layer adhesion.

Summary of Fixes and Precautions

- Check the manufacturer recommended print settings. Do not go faster or change the temperature outside their ranges to start your tests.
- If experiencing poor layer adhesion while staying within the recommended settings, try increasing the extrusion temperature slightly.
- Confirm you have enough torque and that your extruder motor is not skipping.
- Turn off active cooling fan if the particular material does not call for it.
- Make sure there is no issues with the material you are using.
- If experiencing delamination – check the "Warping" chapter.

Post-Processing

This is not going to be classified as a failure, but rather tips to help you combine, smooth, sand, and paint your 3D printed object. There will be no tips on how to print your part in this section, just how to post-process it.

I have had a decent amount of experience with post-processing, and will let you all know my results below.

Loctite Super Glue Gel

Combining 3D printed parts can be a difficult task, which is why Loctite Super Glue Gel has become the preferred favorite of many 3D printing enthusiasts. This stuff is quite strong and can dry in under a minute. When you have a part that can't easily be clamped together for drying, or a small part that can easily break, you will likely want to go with using Loctite. Remember that this stuff dries fast so you can accidentally glue your finger to the part if you are not careful.

After you have used everything you can get out, make sure you remove the tube from the hard plastic bottle. There is still about 30% of the glue in there that you can't get out until removed from the casing.

Devcon Plastic Welder

This is a two part welder that you mix together and apply for parts you want to have a very strong hold. Devcon is just one manufacturer of Plastic Welders, I just haven't tried them all. This is my go-to for any part I want to be as strong as possible. In fact, I have found that Devcon Plastic Welder is stronger than the actual layer adhesion, since on all of my tests the part would break before the bond did. The issue comes with the fact that it takes about 10 minutes to get a decent hold, and close to 24 hours before you get a fully cured weld.

Because of this I use Loctite Super Glue Gel for those hard to hold and small spots, and Devcon Plastic Welder for just about everything else.

The only issue I have found is that every so often I get a bottle where one part will come out at a different flow rate than the other. This has led me to wasting a decent amount of the product.

Notes about Super Glue and Plastic Welding

These two products work great with PLA and many other hard plastics, but they do NOT work well with Nylons. I have only done a minimal amount of testing on bonding nylon parts together and I have yet to find something that I find acceptable. If you are designing a cosplay or fan art piece, use PLA or other hard plastics. If you are designing a mechanical piece to be printed in two parts in nylon,

you should have those two parts screw or clamp together. Do not trust a super glue or plastic welding hold for parts that will be used in mechanical applications.

Sanding

Sanding is key regardless of the methods you choose to go with below, and will help you to clean up any print. For the majority of materials (including PLA) I start with 220 grit sandpaper. This is just about the right roughness to help smooth your print out without deforming the look. You can go slightly lower than this for very hard plastics, just be careful.

From 220 you can move to 800 and even 2000 grit sandpaper. If you start with these high grit sandpapers, you won't get much progress, so it is smart to start with 220 and work your way up. You will likely want to wet the sandpaper as well since it helps in the process.

Bondo

This stuff is very hard to sand without damaging your part, so make sure you are only using it to seal large seams that need to be strong. I have used this a few times in the past and you can really get a smooth seal, it just comes with a lot of work.

When bondo hardens it cannot be easily hand sanded. This means you will want to have an electrical sander (or a dremmel sander for small areas). This issue with this is that you will have to be very careful to not accidentally damage the printed part. Not only can you accidentally dent the printed part, sanding gets hot, so hot it can warp a PLA part.

But when it comes to having a strong seal where no one can see the seam, you will want to use Bondo. Otherwise you might want to try out Spackle.

Spackle

This is really for display pieces only, since it can accidentally be dented or scratched off with abrasion. For very small seams, or even small gaps in layers, I will rub on some spackle using my fingers. I am able to get a very small amount of this stuff into areas, and then after a half hour use some 800 grit sandpaper to clean it all up.

The problem with spackle is that even after it dries it won't be that hard. This means you can accidentally dig a finger nail into it. This is why this is best for very small gaps on display pieces and should never be used on anything that needs to be used mechanically.

Acetone vapor bath for ABS and carbon fiber reinforced ABS parts

One of the best parts of printing in ABS is the ability to acetone vapor finish your parts. PLA and other materials are not soluble in acetone, making them unable to

be post processed this way. Not only can it make a print stronger and more water tight, it also gives a great finishing shine to parts that resemble an injection mold quality. Prints are smoother to the touch and overall easier to work with after acetone vapor finishing.

Along with a high glass transition temperature, this is another reason you would still use ABS in 3D printing today.

You do have to keep in mind though that this can definitely be overdone and lead to a destroyed print, so use caution when proceeding with these steps.

*****CAUTION*** Acetone is EXTREMELY flammable, and this process should only be done in a well-ventilated area with absolutely no open flames.**

Step 1: Prepare your slow cooker in such a way that you are using a metal grate (often comes with the slow cooker). This metal grate needs to be entirely taped off on the bottom with packaging tape and elevated slightly off the surface of the cooker. Make sure to use clear packaging tape as ABS parts will not stick to it unless overexposed to acetone vapor. This will help prevent the acetone from bubbling up and accidentally directly splashing your part during the heating stage.

Before turning anything on or putting in the metal grate with your part, be sure to add a few tablespoons of acetone.

Step 2: Place your part on the newly taped off metal grate and put into your slow cooker. You will then close your slow cooker and put it at the LOWEST settings. This is a VERY fast process, so do not try this without reading all of the steps and researching further.

Leave your part on the metal grate in the warm cooker for no more than 45 seconds. I had a method down where I would leave it on for 30 seconds, off for 15, and then on for 10 final seconds. This process will vary with each slow cooker you use.

I actually used to use a big broiler with a similar setup that I would put onto a hot plate for larger ABS prints. This process requires roughly 10 minutes though, due to the large volume of the broiler.

I recommend that you always error on the side of less time. You can always do another round of acetone vapor if you determine that the print requires it.

Step 3: After your quick acetone vapor bath is complete, remove the grate that is holding your part and put the two off to the side (onto a countertop that you do not care about). After allowing for 30 minutes or more for drying, you can remove the part to be vacuum purged.

This step is not needed but definitely helps with the strength of the part as well as the time required to dry. Doing the process above without a vacuum purge will require 24 – 48 hours before the part is to full strength.

XTC 3D for PLA and PETG

I like to be entirely honest with my reviews, and while other's have seen some amazing results from this stuff, I just personally have not. I am not sure if I am doing something wrong with the mixing or what, but my results are never quite what I see other's getting.

The idea behind this is XTC is that it is a protective and smoothing coat for finishing 3D parts that does not melt plastic. XTC-3D should fill in gaps and retain a smooth, shiny finish.

I have definitely had some semi-successful results in the past, it just often becomes more work than it is worth. First, you have to make sure the part is sanded as best as you can with standard 220 grit sandpaper. You then need to wear gloves and be in a well-ventilated area as you mix the two part goop. You then paint on an even coat, wait for it to entirely dry, and give it a further couple passes of sanding. After a decent amount of elbow grease, you should be left with a very smooth outer surface.

With some practice I am sure this can be perfected, I just personally prefer other methods at this point.

Polysher and PolySmooth

PolyMaker has also made a product called the Polysher which works exactly as an acetone vapor bath, but is made specifically for 3D prints utilizing isopropyl alcohol. Rather than being soluble in acetone, their proprietary PolySmooth filament is soluble in alcohol, making a vapor bath work perfectly to remove layer lines.

I was given a Polysher for review (which is up on my YouTube channel) and was very happy with the results. It is much safer than going with an acetone vapor bath, but you are limited to only using their PolySmooth material (or another alcohol soluble filament), and you are also limited on size to their Polysher.

This is great if you plan on doing a lot of display pieces such as miniatures or action figures, since it works perfectly to result in an injection mold look - but I would be cautious when working with mechanical parts due to the lack of detailed information on PolySmooth.

For further information, search for "Polysher by Polymaker Review" on YouTube for my 3D Print General review.

Primer

If you plan on painting your part, you will want to spray a coat of flat grey primer (or primer filler) before moving forward. Make sure you have combined your parts, sanded them smooth, and cleaned them entirely off of debris before going to this step. It would be smart to use a powerful air blower if you have a compressor on

hand as to make sure all dust is blown off.

In a well-ventilated or outside area, with a tarp laid down, spray a light, even coat of primer about 6 inches from your part. This will allow for acrylics and other paints to stick to your part properly.

After waiting a couple of hours to fully dry, you can go ahead with further sanding, spray painting or hand painting.

Painting

I am sure the best way to paint parts with high detail would be use an air brush kit, but I personally do not have any experience in that regard. I stick with acrylic hand painting.

You can get a 24 color set of acrylic paints online for pretty cheap, it just requires some practice to get details down. You can also use model paint, it is just a tad more expensive. I have a few tutorial videos in this regard on my 3D Print General YouTube channel if interested in learning further. Remember to always allow time to dry and finish with either a clear satin or glossy spray coat to make sure everything sets as it should.

Keep in mind that not all materials are great to paint. I have attempted to paint flexible filaments in the past, and it seems that acrylic paints will crack when being flexed.

Using shoe polish for shading

This is something I have only been doing recently, but it can add a great shading effect to parts that call for it. You need to make sure your part is as smooth as possible, the thicker the gaps in the layer lines, the worse this effect will be. You also need to make sure the previous painting is entirely dry with a clear coat on top, as to reduce any chance of paint chipping.

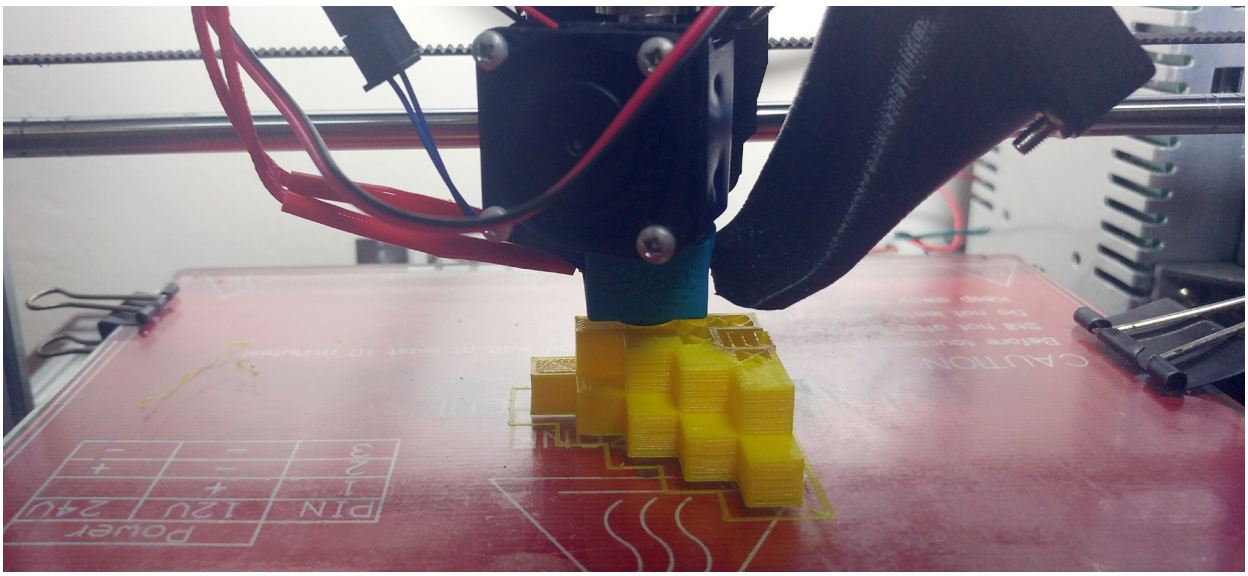
Essentially, if you get some black shoe polish and spread it out on a paper plate, you can then brush it onto your print with a paintbrush into the crevices and indents that should show shading. This works great for detailed figurines in which you want muscles or folds in shirts to show through better.

The black shoe polish will naturally go into these indents, and all you have left is to clean off the excess. Grab a sponge and get it wet. You then want to use the rough side to LIGHTLY brush off shoe polish. If you didn't use proper clear coats or wait the proper amount of time for paints to dry, you run the risk of chipping off paint. The sponge won't be able to get into those small indents and you should be left with a part looking as if it is shaded.

For a far more detailed explanation, you should check out Cosplay Chris on YouTube, as he has perfected this approach with his "Custom Collectables" playlist.

Print Pauses Mid-Print

This is when your printer thinks it is either still printing or the print is complete, while in reality it looks as though the machine is frozen. This is different than ghost printing (Nozzle Clogs/Running Out of Filament) because the printer never actually completes the tool path it was given in the G-Code.



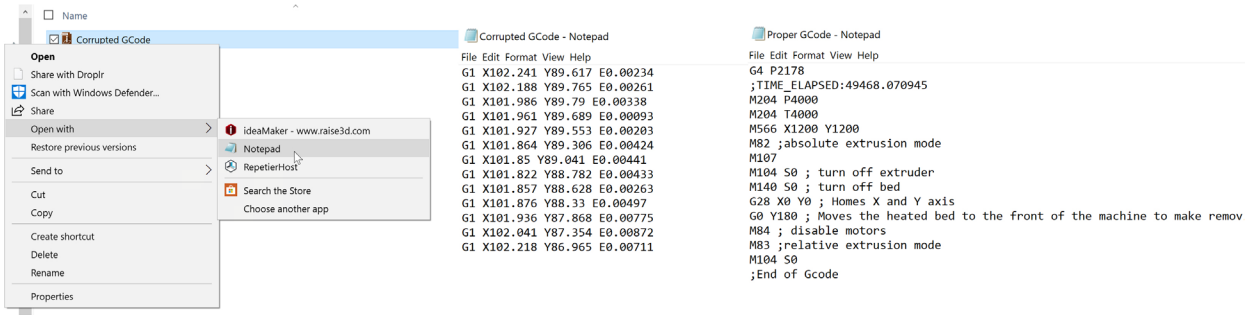
This issue is not very common but can be another one of those frustrating failures that can be extremely difficult to properly diagnose.

Corrupted G-Code

As mentioned in some other chapters, there is a possibility that your G-code could be corrupted. I have had the experience of exporting a large G-code and then preemptively dragging the incomplete file over to my printer. You have to make sure to wait until the file is fully exported before dragging to be printed on your printer.

Since I didn't realize the G-code wasn't done saving, the print stopped right at the point I dragged it over. This caused the remainder of the tool path to not be followed, and for the end G-Code to never be triggered - meaning I had a hot nozzle and build plate with a half-finished print paused just like the photo you see above.

I had assumed that it was a connectivity issue, so I just reprinted, only to experience the exact same issue at the exact same spot on the print. Re-slicing and exporting the G-code fixed this on the third try.



You will notice some larger G-Codes can take 5-10 minutes to slice, so you do not want to get over zealous when trying to get your print started. An easy way to tell if your machine was sliced at the incorrect time is to just open the G-Code in a notepad to see if the ending script is there.

Connectivity issues

If you are running your printer via a Raspberry Pi, this is something you may experience. A connectivity issue could be a problem with your USB cable, or with your Raspberry Pi itself (disconnects from internet or local network do not cause failures, but you won't be able to reach the octoprint UI from another device to do anything with the printer if the pi disconnects from the network).

There is even the chance that your Raspberry Pi's power cable has been disconnected or bent to the point where it is not producing power. If your Raspberry Pi loses power mid print, the print will just pause exactly where it last left off.

I have experienced some very confusing situations in which I could not properly diagnose why the connectivity problem occurred. I would start a print overnight only to see it paused in place the next morning. A simple reboot of the machine and of Octoprint would fix this just about every time.

A good precaution for this is to always make sure you are using the most up to date raspberry pi and Octoprint firmware. Make sure you are using a strong, USB power cord and Ethernet cable, and that they have a good clasp (so that they stay in their ports).

I cannot stress enough how important it is to only use high quality USB cables with ferrite cores (cylindrical objects on the cord near the connectors) and make sure to have spares on hand. These cables degrade and go out more often than you would expect. High quality cables can typically be sourced locally for less than $10 and are well worth the extra $1-$2 premium over generic lower quality cables.

If your pi is in a place where it can be rattled or moved repeatedly, you will experience this issue more often.

This is also true if you were to remove an SD card mid print or were to disconnect your printer from the computer when printing via Repetier.

Power malfunction

Power malfunctions can relate to either your power supply malfunctioning, overloading a circuit breaker, or having an entire power outage - with an outage being one of the hardest to diagnose.

3D printers are energy hogs, especially when heating a large build platform. You will not want more than two power hungry printers on one standard circuit, otherwise you are prone to blowing it.

When I would run 12 printers simultaneously, even though we had upgraded our electrical, I would never start the bed heating process for more than 3 machines at one time. That is because the initial amount of power that is required is more than the energy required to maintain the temperature.

Next, the power supply on your machine may actually be malfunctioning. If you are running an inexpensive machine, you are likely running a very inexpensive power supply. These blow out all the time and have issues with their fans dying. I have had a power supply whose fan died without me noticing, which would cause the power supply to overheat and shut off periodically. This will leave you confused coming back to a power supply that is working and a print that is paused in place.

If your breaker were to blow mid print, or your power supply were to give out entirely, you will be left with a similar looking problem (power off on machine, print failed mid print). Flip your printer switch off and go to your breaker to see if any switches are flipped. If so, you know that it is an issue with overloading the circuit.

And then the worst of all scenario is an actual blackout. This is not your fault and can happen periodically in any city. Two times in the past couple of years I had started extremely long prints when I experienced a blackout that affected multiple city blocks. Every company and house was without electricity for a short period of time, so the machines were powerless as well.

One of these times it happened in the middle of the night when I was not near the facility. When I came in the next morning everything powered up just fine yet every single print was frozen in place. The only reason I was able to tell there was a blackout was by checking a time-lapse video that I had been running.

If your printer is extremely important and cannot be shut off in a blackout, it would be smart to invest in a backup power supply generator that can run the printer for an hour while power hopefully returns.

NANO fuse issues

Finally, you may see similar issues, but some parts of the printer still power on when it is a NANO fuse issue. If your printer turns on, yet either the LCD screen doesn't work or some other strange part is not working – preventing parts to complete – there is a chance you have a blown NANO fuse on your board.

I go over this in full detail in the "LCD Blank or Dark" chapter. I have only had this problem on Lulzbot TAZ machines, so make sure you know if your board requires NANO fuses.

Summary of Fixes and Precautions

- Make sure your G-Code was exported properly and re-slice if needed.
- Check connectivity of printer to raspberry pi. Any disconnections in that chain will cause this type of failed print.

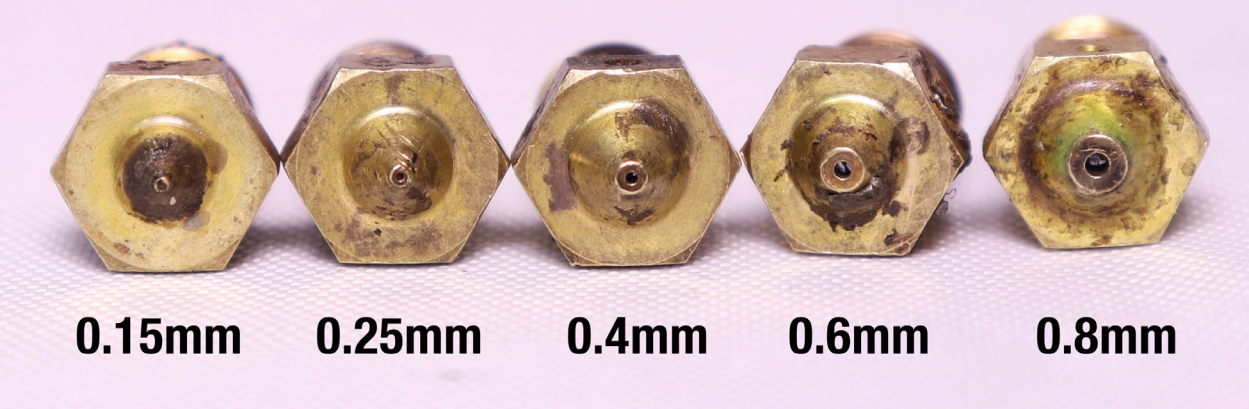
  Or

- Check to make sure your SD card is not freely moving and remains connected throughout the print.
- Check to see if power supply is working and fans turn on when hot.
- Make sure you are not overloading any circuits.
- Check with neighbors if experiencing a blackout.
- Have extra NANO fuses on hand if your board requires them.

Quality Options

The quality, and amount of time to print, will vary based off of two factors – your nozzle diameter and the layer heights you are printing at. Below we will take a look at a few of these options.

Nozzle Diameter



The nozzle diameter will determine the line width of your print segments, which will affect the tolerances in the X/Y direction. Any part of your print that is thinner than your nozzle diameter will not be printed, so you can imagine how a thinner nozzle diameter can lead to a higher quality print.

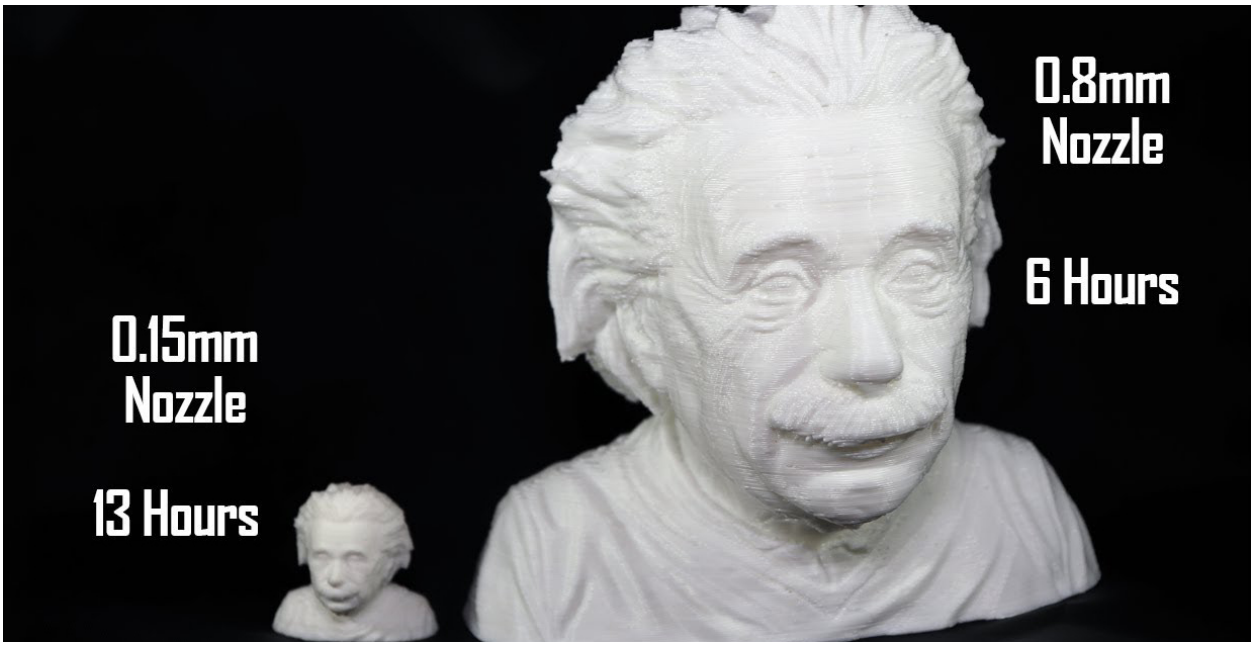
The biggest issue with this comes with the print time required. The fact that you have to slow your print down to prevent nozzle clogs, that you have to print at lower layer heights, all along with the actual lines being thinner, your print can be exponentially longer.

The general rule of thumb is to allow for a clearance of ½ the nozzle diameter for parts that mate together.

When printing with a very small nozzle you will need to be using a geared extruder. You need the proper amount of torque to push through 0.15mm or 0.25mm diameter nozzles.

Personally, I have standardized to using 0.25mm, 0.4mm, and 0.6mm nozzles. It seems that the 0.15mm nozzle is very hard to dial in and takes an extraordinarily long amount of time to print with, and the 0.8mm nozzle is just too low of tolerances for what I am looking for. I have a couple of videos going over printing in different nozzle diameters at my YouTube channel – The 3D Print General – if you would like further information.

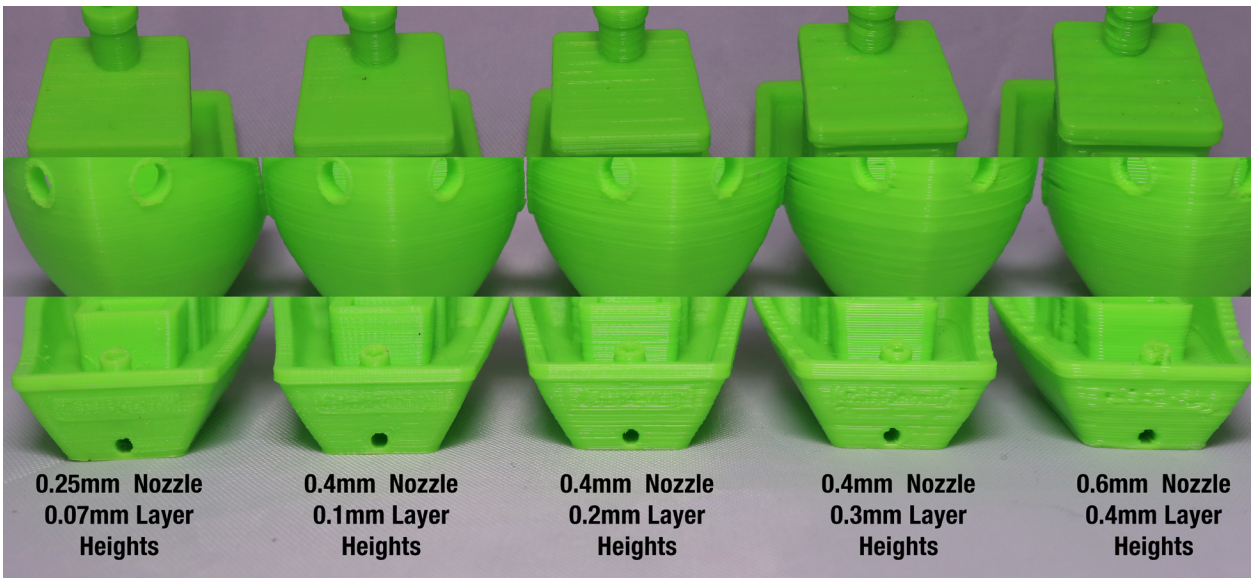
How the nozzle diameter effects layer heights



As stated elsewhere in the book, you have a range of layer heights that will result in reliable prints based off of your nozzle diameter. Essentially you want your layer heights to stay within 25-75% of your nozzle diameter. This means a 0.15mm nozzle should print within 0.04mm – 0.11mm layer heights, and a 0.8mm nozzle should print within 0.2mm – 0.6mm layer heights.

When you go outside this range, the extrusion reliability and quality will often go down. When you try to print with a small nozzle with too large of layer heights, you will surely clog and grind filament more frequently, and when you try to print too low of layer heights on a large nozzle, you won't be printing at quite the tolerances and quality that you could with a proper nozzle diameter.

Layer Heights



Layer heights refer to how thick each individual layer is in the Z-direction. Large layer heights reduce the quality in the Z-direction, but allow for a much faster print. When printing at the same speeds with same nozzle diameter, a print that is 0.2mm layer heights will take half as long as the same print with 0.1mm layer heights.

It seems that the speeds you can print at with a standard extruder/hotend setup works on a bell curve. You need to slow down your print as you go to very low layer heights and when using a small diameter nozzle in order to prevent bottlenecking and nozzle clogs. You also need to slow down your print speeds when going with a very large nozzle with large layer heights in order to get the proper viscosity. If you print too fast with large layer heights and nozzle diameters, the particular material may not have enough time to melt. This can be remedied with a hotend meant for this - such as the E3D Volcano.

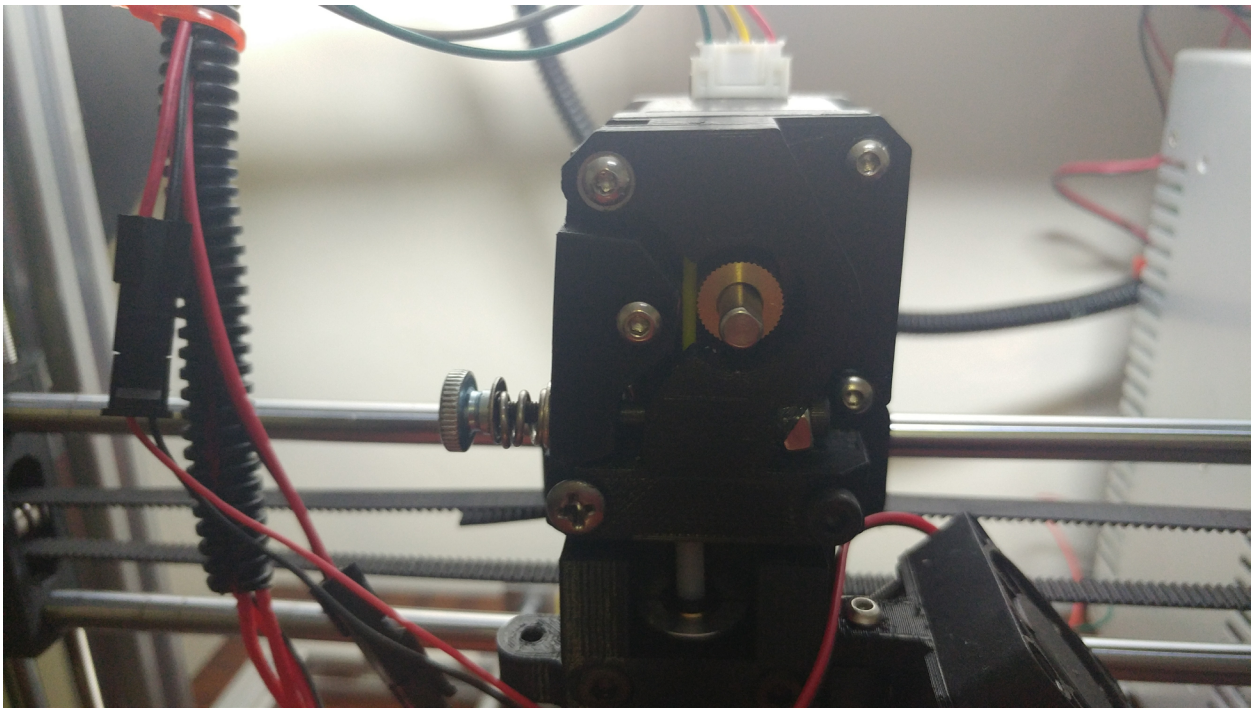
It seems I can print with the fastest linear speed on my standard setup with a 0.6mm nozzle at around 0.25mm layer heights. Once I bump up to the 0.8mm nozzle I need to slow down my print speeds, and the same is true when moving to a 0.4mm or 0.25mm nozzle. The larger nozzle may still allow the print to finish faster even with the lower print speed due to the additional volume of material that is being deposited with each move.

For over 90% of my prints I have standardized to a 0.4mm nozzle. I have this nozzle in hardened steel and will print the vast majority of my prints at 0.1mm – 0.25mm layer heights with this 0.4mm nozzle. This will work for the majority of 3D printing applications.

Running Out of Filament

This problem is by far the easiest to diagnose but also one of the most frustrating when it occurs. You can think you have enough filament for that 400 gram print, when 20 hours in, with only 10 layers left, you run out of material.

You can avoid this by taking these precautions



Weigh an empty spool

It is always good to have the weight in grams of an empty spool for the filament manufacturer you are using. These do have tolerances, but it is a good starting point.

After you have the weight of an empty spool you can then weigh the spool you are about to use for your next print. Subtract the weight of the empty spool and you should have a rough estimate of how many grams of the material are left. Make sure to provide a buffer of at least 20 grams to account for tolerances in the spool itself.

Pause at layer height if you know you will run out

If you are going to start a print with a spool that you know does not have enough filament to complete, you can add a “Pause at Layer Height” to the slice of your model. There is a plugin available for Cura and similar features on Simplify 3D that will allow you to have the print pause at a specific layer height and raise in the

Z-axis, allowing you to switch the filament out to another spool that can complete the remainder of the print.

This is especially useful for very large prints and you don't have a full spool available.

You can also use this feature if you would like the top portion of your print to be a different color than the bottom.

Understand the density of the material you will be printing with

When you use a slicing program it will give you an estimate of the amount of material it will be using. If it gives you this number in grams, it will not be accurate if you are using a material that is different than is in your machine settings. If you are given this estimation without having to set anything up, it is likely based on the density of PLA.

If you are given the estimation in meters and not grams, you will have to do a minor calculation to find out the estimation in grams for your material.

PLA is 1.25 grams per cubic cm. If you were using 1.75mm diameter filament, one meter of filament would be 2.41 cubic cm in volume.

1 meter of PLA 1.75mm filament would then be equal to 2.41 x 1.25, or 3.0125 grams. A 1,000 gram spool should be roughly 331 meters.

Using that 2.41 cubic cm in volume for 1.75 filament, you can use the data below to figure out how many grams your material expects to use.

Density of material

PLA: 1.25 g/ccm - 3.0125 grams per meter of 1.75mm filament

ABS: 1.04 g/ccm - 2.5064 grams per meter of 1.75mm filament

PET: 1.38 g/ccm - 3.3258 grams per meter of 1.75mm filament

Most Nylons: 1.13 g/ccm - 2.7233 grams per meter of 1.75mm filament

The same is true for 2.85mm filament, you would just use 6.38 cubic centimeters as the volume per one meter of filament.

If you are using Octoprint, you can also try installing the Filtracker plugin which uses unique QR codes to monitor remaining filament length and provides alerts when attempting to print an object without enough filament remaining.

Summary of Fixes and Precautions

- Weigh your spool before starting print.
- Add settings to pause at a layer height to allow you to switch filament if you know your spool does not have enough.
- Know the density of the material you are using to calculate the estimated amount of grams for your print.
- If you are running Octoprint, try out the Filtracker plugin.

Settings Issues

This is a very vague chapter since it can deal with a variety of issues related to having a clean print with the proper dimensional accuracy. Every single material by every single manufacturer on every single machine will have slightly different slicer settings in order to achieve the highest quality print. That being said, I go over my personal settings for each material in "Materials and Their Settings" chapter in this book.

This section has actually been entirely re-written now that new slicer options are available. When covering this in my previous book I went over the old Cura – Version 15.04.6 to be specific. Since that book, Cura has entirely redone their interface, along with including the ability to tweak just about anything you can think of. I originally only covered Cura settings because it was the best free slicer in my opinion, but as of their new updates, I find it better than almost any paid option as well.

It may take a little while to get used to everything, but just about anything you want to tweak is now available. That is why this chapter is such a long and all-encompassing one.

Printers such as Zortrax will require their own slicer and not have these options, but you really should not be experiencing any settings issues on those machines when using their proprietary material.

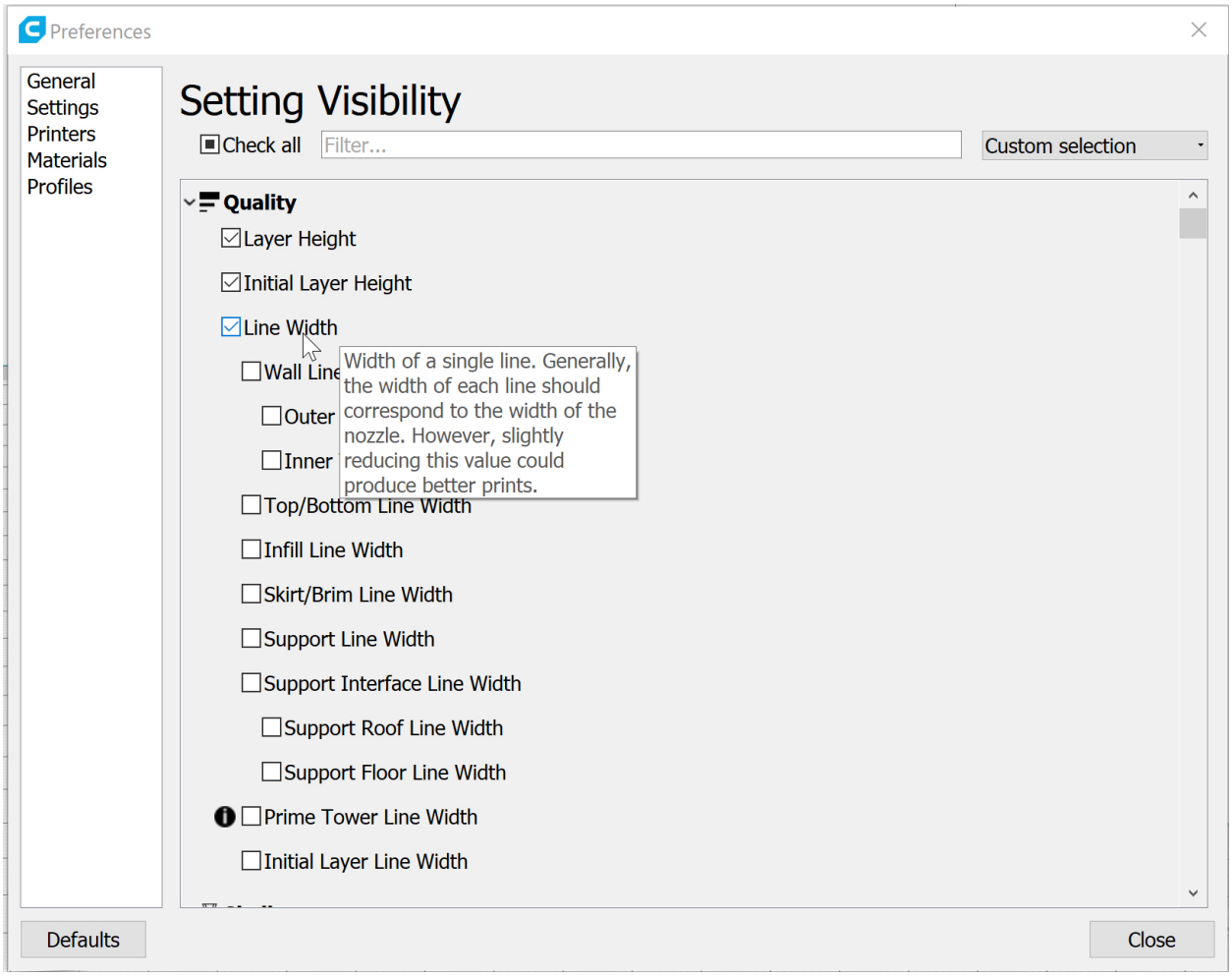
I will specifically be using Cura 3.3.1, but they are frequently releasing updates with increased options.

The following examples will be with printing in PLA on a direct drive machine, though you can get more specific settings in the Materials and Their Settings chapter. If you would like some further detail you can always visit my 3D Print General YouTube channel which goes more in-depth on a couple of these options.

About Using Cura 3.3.1 and Newer

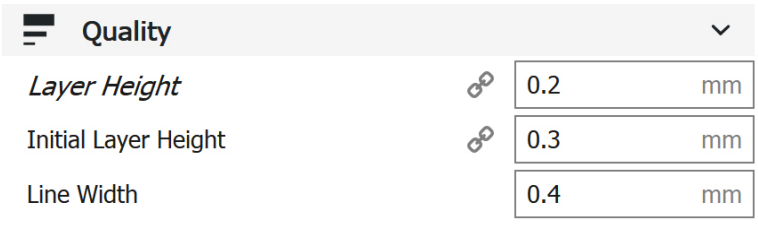
In order to see expanded options, make sure you are in the "Custom" print setup, and not "Recommended". You can save profiles for specific materials and qualities, but we will just be going over what each options does.

You can then click on the gear icon next to any section to option the expanded settings. When in the custom selection you can choose exactly what options you want to tweak. For anything you are not sure what it does, you can always scroll over to be given a definition. I will personally only be going over the options I tweak, because there are literally hundreds of settings available.



Quality

All of the factors I will be going over in this Quality section are the same for both a Bowden and Direct drive printer.



The layer height of your print is very dependent on the nozzle size of your printer as well as the quality of the print you will like to achieve. A print at 0.1mm layer heights will take four times as long as one at 0.4mm layer heights, with the same nozzle diameter and printer speeds. You can get up to roughly 75% your nozzle diameter in layer height, but remember that you sacrifice quality for time the higher you go with the layer heights. You can also achieve as low as roughly 25% the nozzle diameter without lowering your results.

Changing your layer height should not affect the amount of material you are using, just the time involved to print. The longer the print though, the higher the chance of experiencing a failure at some point. Not only will the print be longer due to the

smaller layer height, you will actually have to run your extrusion speeds slower as well for a clean print.

The layer height is also affected by your Z-Axis leadscrews/threaded rods. The diameter and pitch can affect the quality due to where your carriage is on its rotation. There is a handy calculator over at www.PrusaPrinters.org/calculator if you don't want to do any math. You just choose your Motor step angle (labelled on your Z axis motors), your desired layer height, and your leadscrew pitch. Most of my machines are 1.8° step angle and a leadscrew pitch of 2mm/revolution on an M8 leadscrew. This means I can tweak my layer heights on a 0.01mm basis without any issues. But if you have an M5 threaded rod with a pitch of 0.8mm/revolution, you will have to tweak on a 0.014mm basis for best results. This means instead of 0.25mm layer heights, you should actually go for 0.248mm or 0.252mm.

This is a bit technical and will definitely lead to cleaner results, but I have printed outside of these suggested ranges in the past without much of a difference. This is for when you want to make sure everything is as tuned in as possible.

The Initial Layer Height is actually for bed adhesion more so than quality. This allows you to have a thicker first layer in order to make sure everything sticks properly. This number should always be at least as thick as your normal layer height, and should only be increased up to 75% of the nozzle diameter. I always take advantage of this because getting the first layer to stick is easiest with a thick first layer.

This is a big reason dealing with small nozzles is so difficult to get the first layer to stick properly. I have used a 0.15mm diameter nozzle before and it took me over a half hour of restarting the print to get that first layer z-height distance correct. This is because the thickest I could print my initial layer height at is 0.11mm after factoring in the RepRap Calculator limitations.

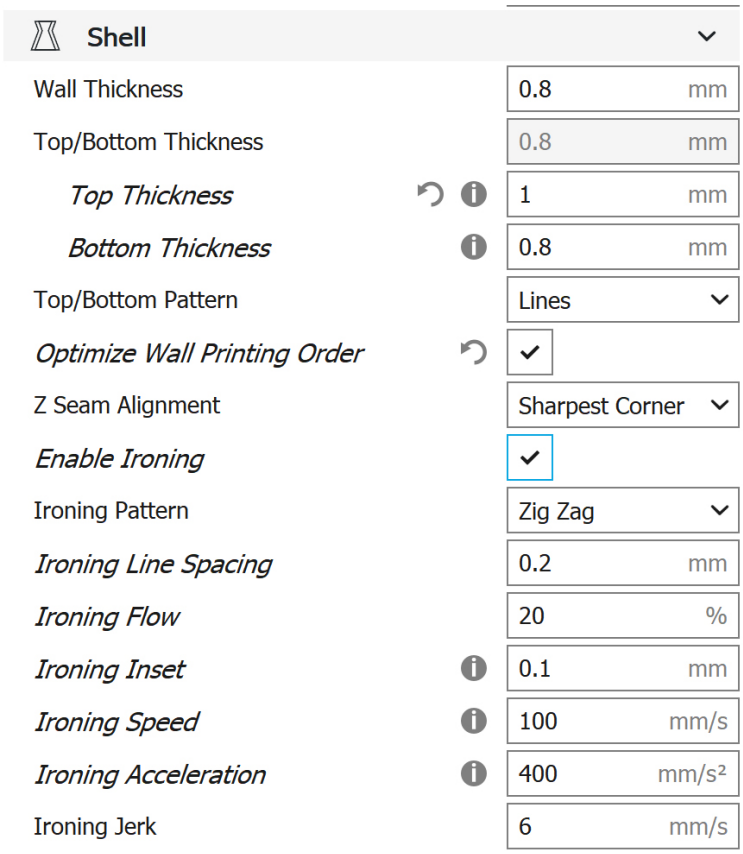
The Line Width is just your nozzle diameter. You can actually go outside this range, but I would always just suggest using the nozzle diameter for the line width you want to achieve. If you want thin 0.25mm line widths, I would highly recommend using a 0.25mm diameter nozzle instead of just tweaking this for a 0.4mm nozzle.

There are roughly a dozen other settings you can tweak in Cura under Quality, but I personally do not change any of them, since they all deal with changing the line width of specific sections of the print. There may be particular applications in which you want to do this, but I always stick to the line width of the nozzle diameter I am using.

Shell

Within the shell section you can tweak how your outer walls and top/bottom layers are printed. The first option I have open is the Wall Thickness. The wall thickness should always be a multiple of your nozzle diameter (or line width) and should be as thick as required for your individual part. Unless printing something in vase

mode (spiralize the outer contour), you will almost always want at least a minimum of two shells (3-4 on very small nozzle diameters). This is not only for strength, but for infill overlap to make sure the outer surface is not affected by the infill.



If your particular part requires strength from the outer direction, such as when I print skateboard wheels, you can increase the shell thickness rather than increasing the infill percentage. On the many skateboard wheels I have tested and worked with, I increased the wall thickness until it was 100% filled via shells. This is why it is important to understand which direction your part requires strength in before choosing to increase infill percentage, wall thickness, or both. For an average part I will keep this to a multiple of 2-3x the nozzle diameter.

The Top/Bottom Thickness is how many layers of 100% fill will be left before and after the infill is printed, and is a multiple of your layer height. This number will have to be tweaked depending on how much infill you use and how thick your model is, but I always suggest having a minimum of 1mm in top layers. This is to make sure you do not have a "pitted" top in which you can see the infill below. While this is very important to keep the top layers at a minimum of 1mm, if you are still seeing a pitted effect, it is probably better to just increase the infill percentage. The bottom layers are not as important since they do not have to cover an overhang, I still will do at least 4x the layer height, with a minimum of 0.6mm being laid down.

The Top/Bottom pattern is as it is described, and I almost never tweak this from "Lines". "Zig-Zag" may have some applications, but I don't know when you would want to use "Concentric".

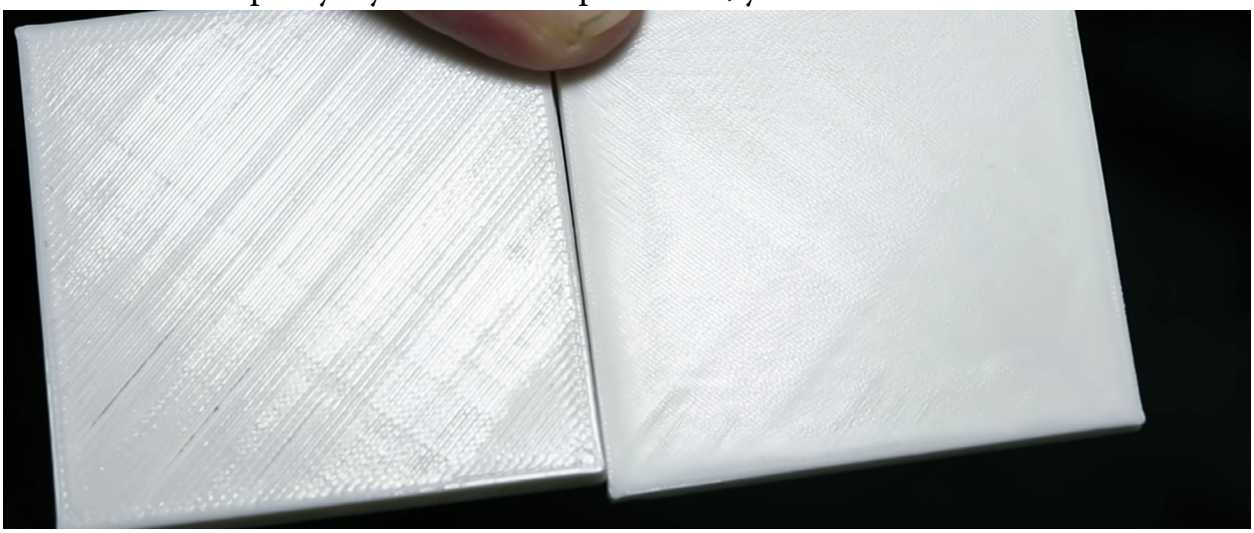
Optimize Wall Printing Order reduces the number of retractions and the distances travelled and will benefit most parts. It should improve speed and quality on most prints, but if you are starting to see defects going up in the Z-direction of your print or an increased slice time – turn it off.

The Z Seam Alignment is where the printer will decide to go up to the next layer.

If you have ever seen a seam going up your print, it is because your printer was moving up in the z-direction to start a new layer right at that spot. These seams are just about impossible to avoid, but you can determine where they are placed. If you are printing a part with corners or lots of curves, you will likely want to choose "Sharpest Corner" for the highest quality. For a smooth or cylinder shape, you may want to choose "Random" for the best quality, so there isn't a giant seam. "Shortest" will choose the fastest printing method, which will likely lead to a seam on one part of your print.

Ironing is a very unique setting that you will not always want turned on. It is also the only option in this section that will need to be tweaked whether you are using a Direct Drive or a Bowden (Bowden will need to run closer to the Cura recommended settings, while Direct Drive has to be bumped up to what is shown in the previous photo). Ironing was a Cura Experimental feature that was moved to the "Shell" section.

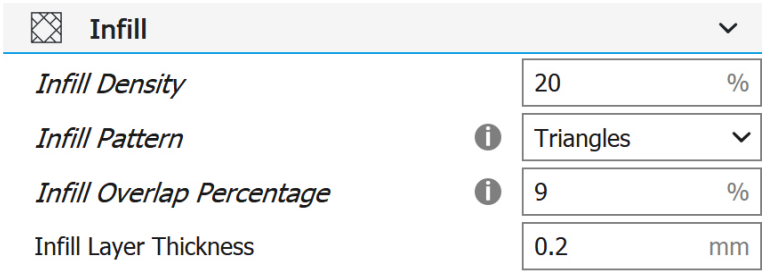
Ironing is a very cool feature in which I cover in-depth on a video on my 3D Print General YouTube channel. It has your nozzle go over the top surfaces an additional time, but without extruding any material in order to melt the plastic on top further, creating an extremely smooth top surface. This will ONLY work on parts that have flat tops and is not needed on parts that are entirely curved, but if you ever wanted to remove those pesky layer lines on top surfaces, you can now do so.



This will definitely need honing in for your particular machine, as it took me a couple dozen prints to get everything you see dialed in for my direct drive printer.

Infill

This section is what you will want to change if you need to increase the strength of your part in the top/bottom direction, reduce the "pitted" effect on your print, or decrease the time required for your part to print. Everything explained below will be the same on both a direct drive and Bowden extruder.



You will have to recognize how thick your part is before realizing how much infill you require. If your whole part is made up of thin walls, it is likely the infill percentage will not make any difference at all, but if you are dealing with a large block, it will be a deciding factor in a successful or unsuccessful print.

Most models can print successfully at 10% or lower, but not for large models. Think about the top surface contour of your print. If it is a flat top that needs to bridge a large gap, you are likely going to need to increase this infill to a minimum of 20%.

To be honest, most parts you print will not require higher infill than 50%, and you will get diminishing results when you go much higher than that. You will use a ton of material to print at 100% infill and your printer will take a lot longer to complete, when you will likely get a lower quality print without much increased strength. I will rarely print something at 75% infill and almost never at 100%. The majority of my decorative or prototype pieces are at 10-15% infill, parts I use are 20-40% infill, and very strong parts I go around 50%. There are many parts that can actually be printed below 10%, even 0%, if the geometry calls for it and you put enough top layers in your Top/Bottom Thickness.

The Infill Pattern refers to the structure shape of the infill. While you are given plenty of options, I almost always keep this on "Triangles". Hexagons are one of the strongest shapes in this regard, but your printer nozzle has to go over the same line twice in order to actually make the shape. Because of this (and because Cura does not offer the Hexagonal option), I go with the second best shape being a triangle. Triangle infill can be printed extremely fast and still has a lot of strength properties.

Others online say they get great results from some of the other options, I just stick with "Triangles".

Infill Overlap Percentage is the percent that your infill overlaps onto the shell walls. When this number is set too high, along with not having enough shells, you can get what I called a "Veiny" print in the diagnostic section, and too low of a percentage can result in infill that rattles around detached from the walls. I almost always reduce the standard set by the slicer and prefer a number around 8% or 9%. This, in combination with 2-3 shell walls, will result in a strong print without any infill veins showing throw the outside.

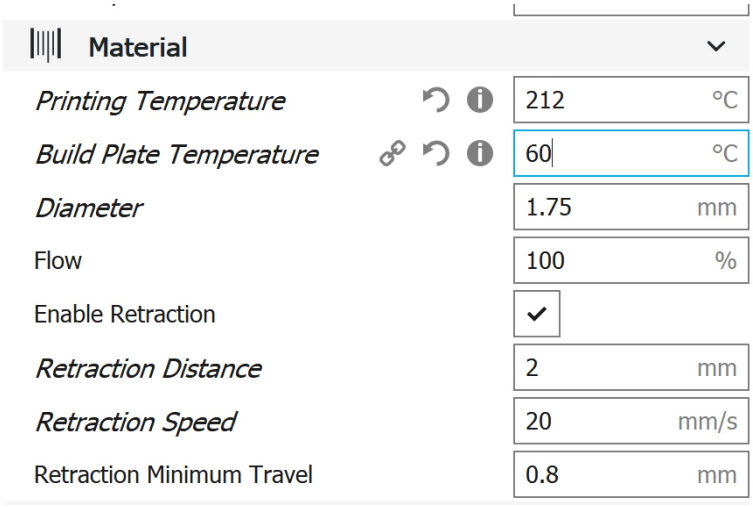
Please note this is for opaque materials. If you are dealing with a translucent material you are going to need to increase the shell walls drastically if you do not want to be able to see the infill. That is not a veiny print, but rather a print you can see through.

Finally, the infill layer thickness is just what it sounds like. For well over 90% of

prints I just keep this the same as the rest of my layer heights. But let's say we are printing a large detailed print in which the strength does not really matter. We may set our layer heights to be 0.1mm, but the quality only matters on the outside walls. You can set your infill layer thickness to be 0.2mm, meaning that it only prints the infill every two layers – drastically speeding up the time required to print.

There are a good dozen other options available in which I personally do not tweak.

Material



Printing Temperature is very straightforward and will be the temperature in which your hotend is set to. This is entirely dependent on the material you are using and the printing temperature will be tweaked depending on the nozzle diameter and layer height. As covered thoroughly in the "Nozzle Clogs" chapter, if you have the hotend set to the wrong temperature, you can have quite an annoying cleanup on your hands.

Bed temperatures for specific materials are covered in the "Bed Adhesion" chapter, so below you will find the generic temperature ranges I like to use for extruding different materials. The range is in reference to the differences in nozzle diameter, layer height, and manufacturer. This is all explained further in the relevant "Materials and their Settings" chapter of this book.

PLA: *180°C – 220°C*

ABS: *225°C – 235°C*

PETG: *245°C – 2552°C*

CFR – ABS: *245°C – 253°C*

Cheetah by Ninjatek: *223°C - 235°C*

Ninjaflex: *223°C – 235°C*

PCTPE: *232° - 235°C*

Nylon 910: *245°C – 252°C*

Polycarbonate ABS: *267°C – 275°C*

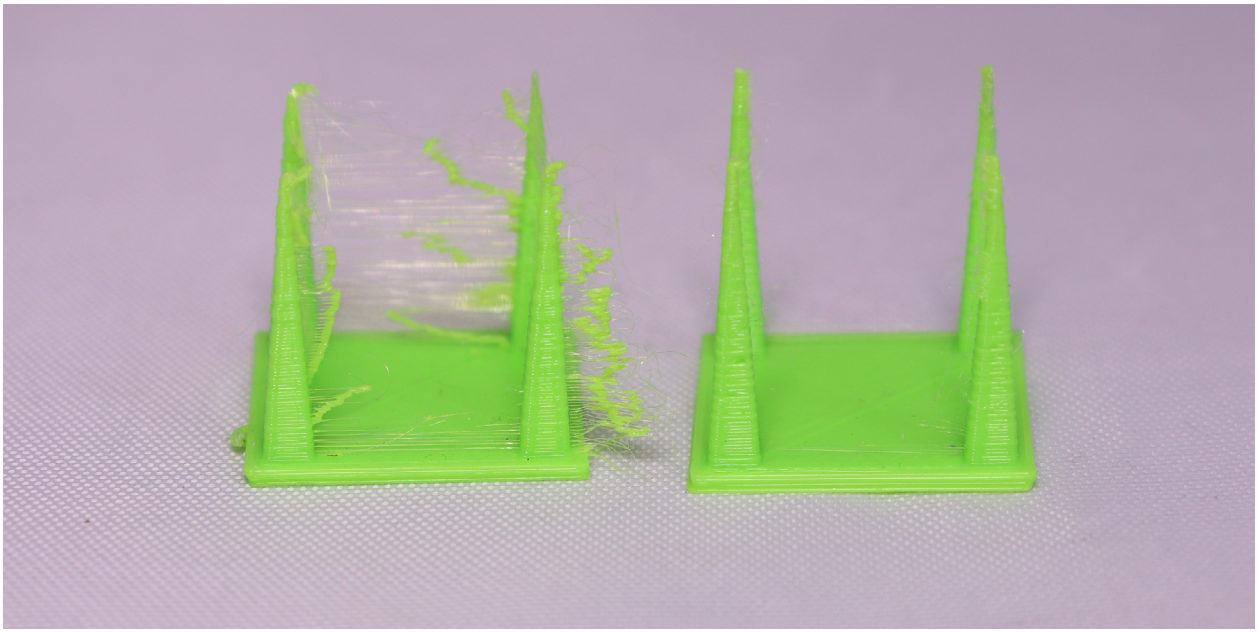
As mentioned elsewhere, you may need to tweak these settings depending on the manufacturer you purchase from, and the machine you are using.

The diameter is the diameter of the material and extruder you are using. This will either be 1.75mm or 2.85mm depending on your setup. If you purchase from a subpar manufacturer, you may get tolerances too large, so make sure you stick with what has been reviewed and tested to work.

Assuming you have your E-Steps dialed in, as explained in the "Over and Under Extrusion" chapter in this book, you will not need to change the flow from 100%. But, if you wanted to change your extrusion rates on the fly, you can change it in the flow section. 101% will extrude 1% more than what your current E-Steps are set to.

Retraction should be enabled on the vast majority of your prints and will need to be tweaked based off of the material and if you are using a Bowden or Direct Drive setup (with the numbers shown being for PLA on a Direct Drive).

Having your retraction dialed in is the number 1 way to reduce the stringiness and "hairy" prints. There are a few models that may require retraction turned off, but you will almost always want it on.



The print on the left is with retraction turned off, and the right is with it turned on to the settings I have shown above. Please keep in mind this model is designed to torture test your machine, and you may not need quite as high of retraction settings on every model.

The retraction distance refers to how far the material goes when retracting, retraction speed refers to how fast, and minimum travel refers to the amount of travel required for any retraction to happen at all. If you have a part with extremely small pegs (for instance - a very small fence on an architectural model) and they are still coming out "hairy" after using the settings I example, you may need to decrease this number.

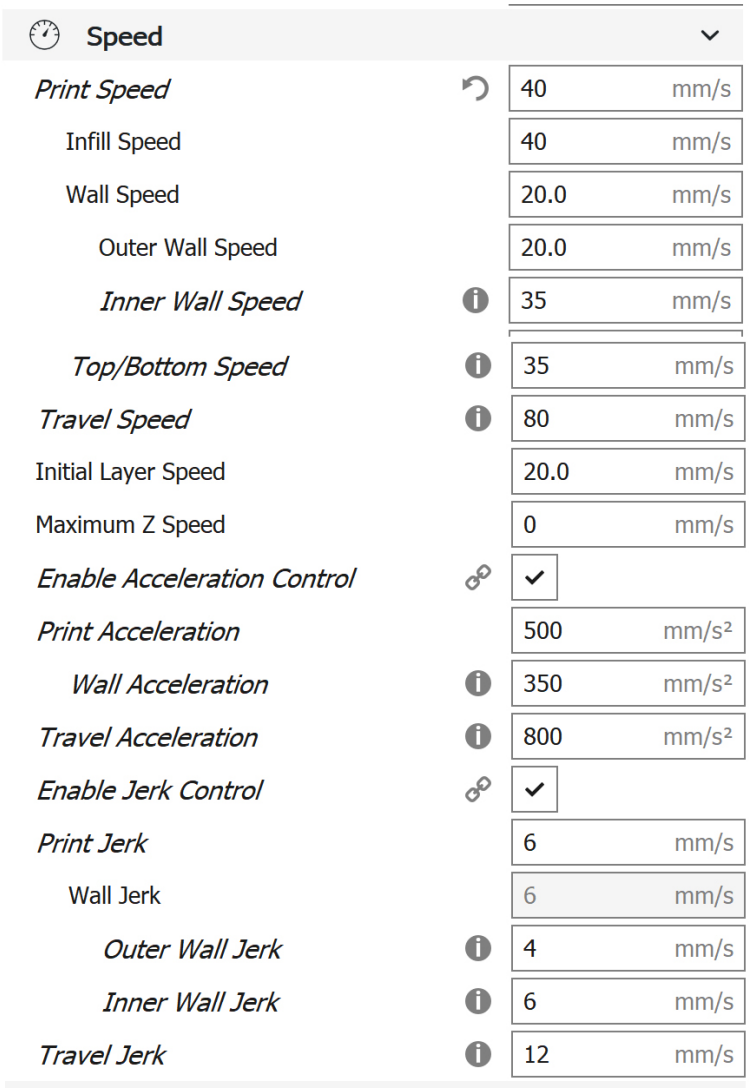
Bowden machines will need these numbers bumped up, as explained in the

"Materials and their Settings" chapter.

Particular materials also require different retraction settings. I have found that PETG is far stringier than PLA, requiring a higher retraction settings.

It is very easy to clean up a minor amount of stringiness via a heat gun and a razor.

Speed



All of the numbers you see will need to be tweaked for Bowden machines, as explained in the "Materials and their Settings" chapter. You will also notice my numbers are MUCH lower than what is suggested and told by other people. These are starting numbers and should be increased until you start to see problems. My acceleration and jerk settings are also particularly low.

These are the settings that will be most tweaked depending on your machine, quality, and the material you are using. As mentioned elsewhere, if you are experiencing issues with the speed your machine is printing at, you should reduce it so that it is no faster than 100x the nozzle diameter. If you have a well-built geared extruder machine that is printing at mid-range heights (~50% nozzle diameter), you can actually get this number to 300x the diameter of the nozzle, but I would not recommend anything near those speeds if you are experiencing issues. On a Bowden, I will run this closer to 60mm/s for PLA.

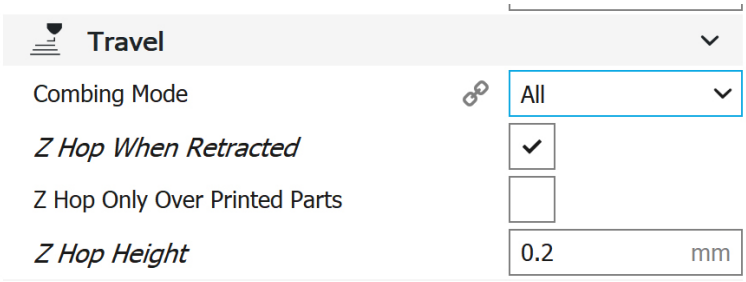
The infill speed should almost always be the same speed as your print speed, while your outer wall should be slowed down a bit to make sure it has the best surface quality. You will also want to make sure your initial layer is about 50% your print speeds, with it tweaked if you are having difficulty getting that first layer to stick. A

generally good range for this is anywhere between 15mm/s and 25mm/s depending on the material being used.

Your travel speed is how fast the carriage moves when not extruding filament, and this can be bumped up fairly high, especially on Bowden machines. As long as your printer isn't rattling, you can likely get up to 100mm/s without much of an issue, even higher with a light carriage.

You can set your acceleration controls in your printer firmware, but it can always be tweaked right here in Cura. If you have ghosting prints, Z-wobble you just can't get rid of, or ugly outer surfaces - reduce your acceleration and jerk. Reducing these will obviously slow your print down, but it will help immensely. Start with the numbers I show above for a direct drive machine and increase them if everything turns out fine. If not, there is likely something else that is wrong (such as a loose carriage or other issue explained in this book).

Travel



Combing refers to the printer head following the path of the print rather than attempting to clear gaps. This will help prevent "hairy" sides of a print. When combing is set to "off", the extruder moves straight from the starting point to the end point and will always retract.

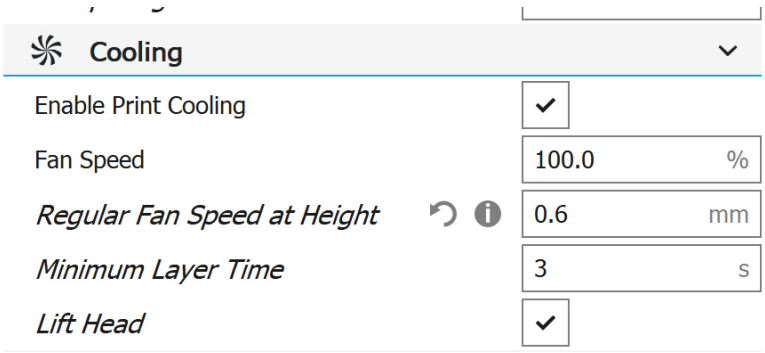
Z hop when retracting refers to the amount the printer will raise in the Z-direction after retracting, as to not knock over small pieces when traveling between sections. If you are constantly getting parts knocked over even though you took all of the precautions mentions in the "Bed Adhesion" chapter, you may need to increase this number. The larger the nozzle diameter and layer height, the more you will likely want to increase this. Also refer to the "Parts Being Knocked Over" chapter if required.

If you ever hear some random loud clicks or noises during your print that you are having trouble figuring out where they are coming from, it may be from your nozzle hitting the print or support as it goes over it. Even if your print does not get knocked over, you will want to add a Z-hop height and increase it until this does not happen. Almost every print you will want at least a minimal Z-hop height.

Cooling

Cooling refers to when your active cooling fan will be engaged. This is crucial for getting the cleanest print possible. If you are printing PLA without an active cooling fan, it is certain you are not achieving the best results you can.

You do not want your active cooling fan to blow on your first layer since it can hurt



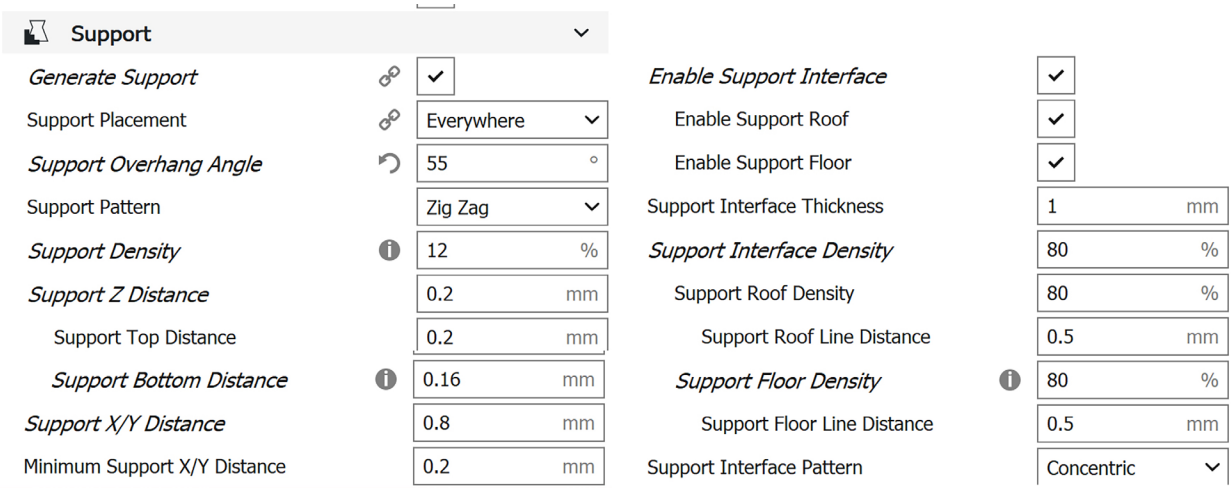
with your bed adhesion. This is why I set the "Fan full on at height" number to 0.5mm – 0.6mm on most prints. It is rare that a material will call for a fan speed in-between 0 and 100%, so it is normally set to 100% on most prints.

If I am printing a large ABS part that comes to a small point, I may actually turn on the active cooling fan and turn this number to the height in which the point starts. If you have an active cooling fan on a medium-large ABS part, or any other high-warping material, it is likely you will end up with a failed print. This is why you will want the active cooling fan turned off for the majority of ABS prints, unless the geometry calls for it.

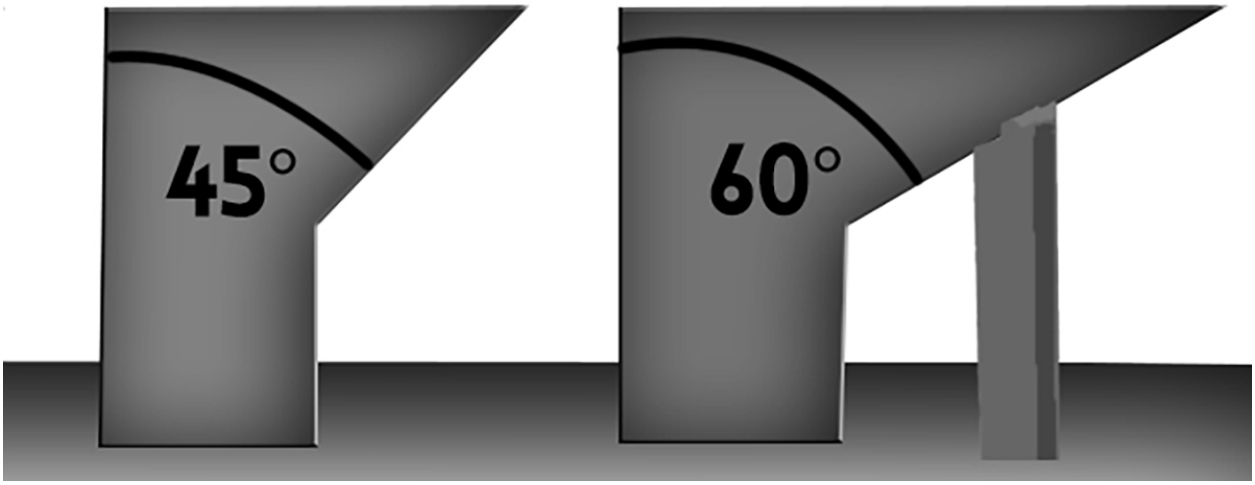
Minimal layer time refers to the amount of time required before starting a new layer. If a layer completes its tool path faster than this amount of time, your printer will pause until the time has passed before starting the next layer. This setting should almost always be accompanied with a "Lift Head" that you see right below it.

Most of the time you will not require this number to be higher than 5, since 5 seconds is long enough for the vast majority of materials to properly cool. If set to 1 second, you will get very ugly, melted pointed tops. Any print that doesn't come to a point will not even be affected by this number, unless set very high. If you are just printing one skinny part, you will notice that it may come out hairy due to constant head lifting from the minimum layer time not being hit. In these instances you can print 2 or 3 of the model without increasing the print time at all, and may actually result in a cleaner print.

Support



The overhang angle for support refers to the minimum angle required before support material is laid down, vertical being 0°. This may seem a bit confusing so it is best described by the image below:



Certain slicing software, such as the one for a Zortrax machine, will go in the opposite direction, having horizontal be 0°. We will be ignoring this for our book, but if you are using something like this, you will need to invert the settings I am describing.

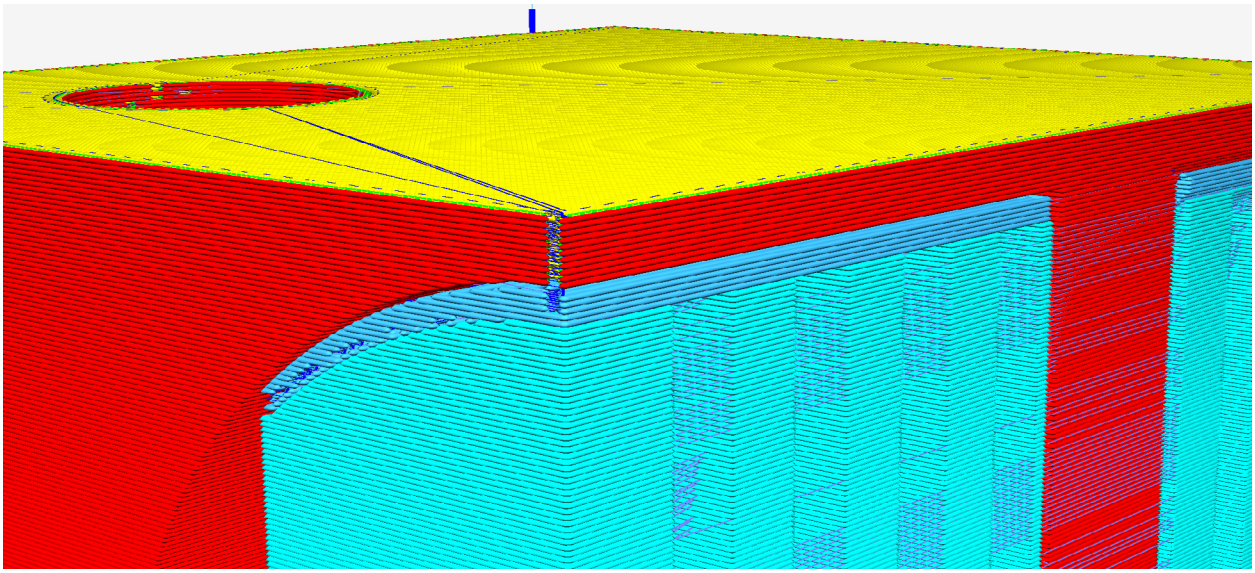
The general rule of thumb is support material will be needed on overhangs of 45° or greater. PLA can actually cleanly lay down angles of a higher degree if everything is set up properly and you are running an active cooling fan. There are models you can find on www.Thingiverse.com that allow you to test the highest angle your printer and material can achieve.

For ABS, I have this number set to 45°, and I will hone it in for any angles I notice scarring. In general, for most materials I will not go lower than 40° or higher than 60°.

You will usually want your support placement to be "Everywhere", unless the model you are printing was designed to only require "Touching Buildplate"

The support pattern I almost always have set to Zig Zag, though you can use Lines to save filament, or Grid to increase quality of the print, but when using the Support Interface explained later, it won't make much of a difference at all. The support density should be tweaked as well if you are not using the support interface.

Essentially the support interface generates a dense interface between the model and the support creating a skin at the top of the support in which the model is printed, and is a newer feature on Cura. This is automatically done on Zortrax machines and is a great addition to Cura. In the last edition of my book this was not available yet, so I went over my support settings just using the normal support lines.



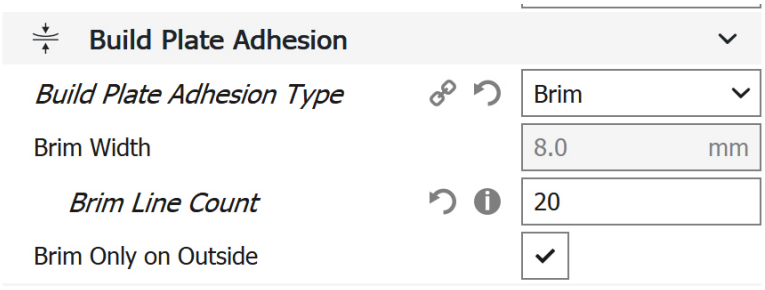
As you can see in the image above, there is this support interface in-between the normal support and the model. This means that the support pattern and support density will not affect the underside quality of your print when having this turned on, so I will reduce my support infill to as low as I can get it.

I use this support interface on almost all main materials, but if you are using flexible filaments or materials with absurd layer adhesion, you will likely want this turned off.

The numbers you see are what works for me on PLA for both a direct drive and Bowden extruder, and may need to be tweaked further for your machine.

If the underside of your print is still ugly after using my suggested starting points, either increase the support interface density and/or decrease the Z distance for support. If it is proving very difficult to remove this support material based off of my suggested settings, you should increase the Z distance for support. The support X/Y distance of 0.8mm seems to work just fine on almost all models and materials, and any closer the support material may be difficult to remove.

Build Plate Adhesion



Having a skirt will allow for the material to purge a bit at the beginning of your print and for you to double check that the bed is level, but will not help to anchor your print. You will only need one line and you will want the start distance far enough from your print so that it does not interfere. A distance of 3mm works fine.

If you are having difficulty getting your part to stick to the bed, or if you are printing with a high warping material, you will want a Brim as mentioned in the "Bed Adhesion" chapter. If you choose "Brim", it will add the number of lines you choose touching the perimeter of your print. These lines are as thick as the nozzle diameter you are using. So if you choose 15 lines and are using a 0.4mm nozzle, you will be adding 6mm of brim around your print.

This brim acts as an anchor to prevent warping and help bed adhesion, and is removed post print. I have "Brim Only on Outside" checked for most prints, because it will prevent extra brim from being laid on holes on the inside of your print, and only be printed on the outside to act as an anchor.

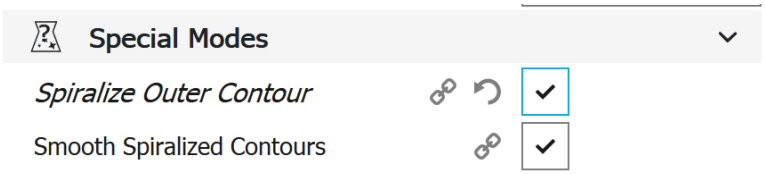
Finally, your last option for platform adhesion, is to add a raft.

I honestly do not use this option often at all. Many printers, such as the ones made by Zortrax, will leave an easy to remove raft on all prints. The Zortrax does it because the build plate is perforated and would leave an ugly underside to your print if it didn't.

If you are printing a part that only has a lot of very small posts touching the build plate, you may want to consider adding a raft. Material has a lot easier time sticking to itself when printing than it does sticking to the build plate on its initial layer. By adding a raft with specific settings, you can make sure the bottom layers are nice and thick with plenty of adhesion, and then stick your print to the top of it.

These rafts are then removed after printing, but can damage your part if too close, and can leave ugly scarring if your part is too far.

Special Modes

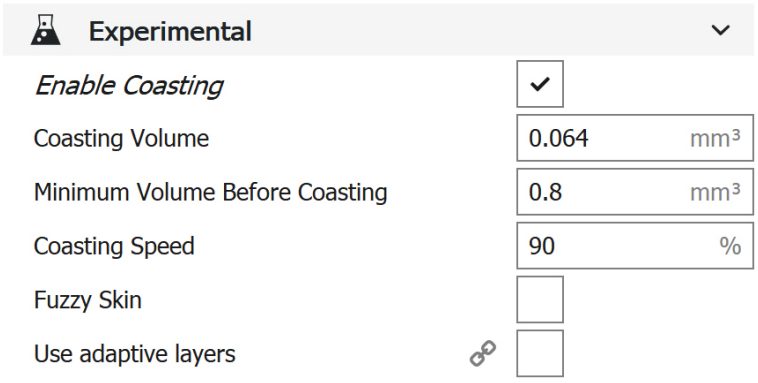


There are a few "special" modes offered by Cura, but I will only use one on rare occasion: Spiralize Outer Contour. Spiralize the Outer Contour is basically "Vase Mode" for Cura. This will make it so only the outer wall and bottom layers of your part are printed. No inner walls, no infill, no top layers. This will only want to be used when you are printing a Vase or something that needs similar properties. I have a Christmas tree that I printed large in this mode that looks great, along with a couple of vases given to my mother. Other than rare use cases like this, it is likely you will want this option unchecked.

Experimental

This section is constantly changing and is where "Ironing" once appeared a few months back. I attempt to go over some of these on my 3D Print General YouTube channel, but Cura is always getting tweaked. I have tried a few of these but have found the most useful to be Coasting and Fuzzy Skin. Coasting is so useful that I

expect it to be moved out of the experimental section in upcoming updates.



Coasting replaces the last path of an extrusion path with an oozed travel path. The oozed material is used to print the last piece of the extrusion path in order to reduce stringing. If you are experiencing "hairy" or stringy prints regardless of tweaking your retraction settings, you will definitely want to try this out. In fact, I have this turned on for almost every Bowden print that I do. The volume should just be your nozzle diameter cubed, and the minimum volume before coasting will make it so coasting is turned off for small areas. This is important to get to 0.8mm cubed or above to prevent small parts from looking under extruded. If that is happening to you, go ahead and increase this number.

Fuzzy Skin randomly jitters the outer wall so that the surface has a rough and fuzzy look. This is a very rare use-case, but it works great to give something a grippy feel that looks stronger than it is. I have personally used this for handles on cosplay guns.

Settings on other slicing programs

Other slicing software may have these features in different sections, or they may not have them at all. For years Simplify 3D was the way to go if you wanted full control of your slicer settings, but as of the last year or so Cura offers just about everything they do – free of cost. There are so many options available that I have well over half of them turned off. You can spend a couple of weeks just doing test prints to figure out what each one does, allowing for full customization.

I am definitely not paid by Cura/Ultimaker and have had no contact with them other than really enjoying their updates to the slicing software.

Stepper Motors Overheating or Malfunctioning

This is a fairly broad printing failure but is essentially when one of your motors is not turning properly or is overheating. Many of these issues are covered in the "Layer Shifts" chapter in this book.

Stepper motors overheating

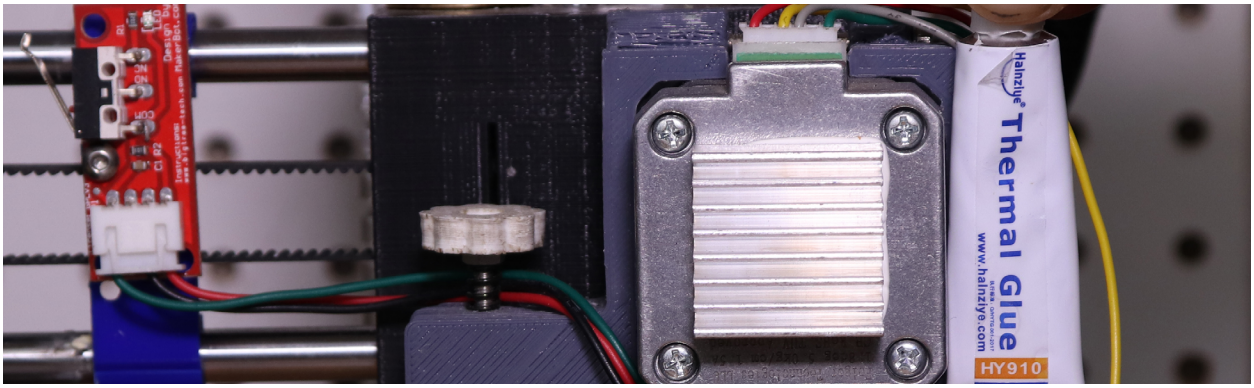
A stepper motor can be running too hot for multiple reasons. If you have an enclosed printer and are running a long ABS print, not only will the stepper motors be getting hot from standard usage, they will be trapped with ambient air of 35°C – 50°C. Even if a print is being completed without any issues, you do not want to run your motors too hot so that they can remain as free of maintenance as possible.

Industrial grade stepper motors have magnetic cores which begin to degrade when they reach temperatures above 80°C. During warm days or prints on an enclosed machine, the stock stepper motor surface temperature will hover between 70°C and 75°C for long print durations at moderate extrusion speeds.

Along with reducing the current to the stepper driver as mentioned in next section, you can provide some external ways to cool these motors.

The extruder motor is one of the most common to overheat, so it can only be beneficial to add a passive heatsink. Increasing extrusion rates while maintaining a constant output temperature will require additional torque from the extrusion motor. This additional torque will require more power, which creates thermal dissipation through electro-mechanical inefficiencies – meaning this motor will become hotter the faster it runs.

After the extruder motor, the X and Y steppers are second most likely to overheat, since the Z will only move intermittently. Photo below is a heatsink on the X-axis motor.

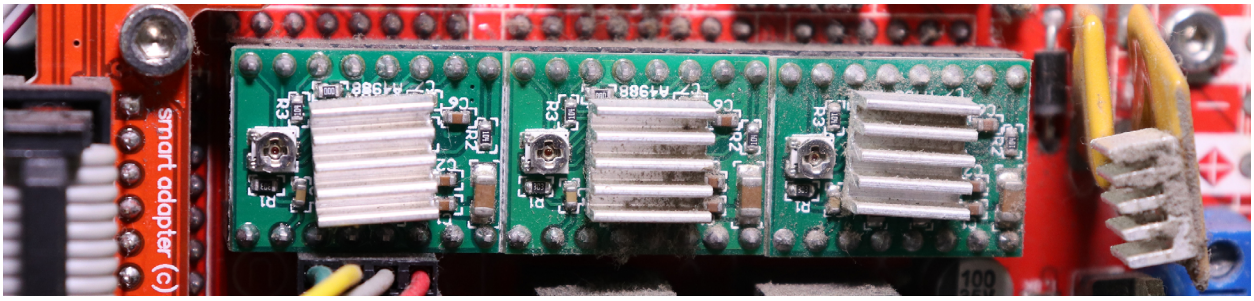


You can either screw a heatsink into your motor, get thermal glue, or some use some strong thermal double sided tape to stick it right on there. These will increase the heat dissipation of the individual motor by expanding the area available for convective heat transfer.

Finally you can connect a fan that will actively blow on that stepper motor. You can wire this fan directly to the power supply on your board so that it turns on when your printer is on, allowing for cooling of any overheating stepper motor. This is not needed on most machines, but if you are running hot, this is a pretty sure fire way to help remedy.

Stepper drivers overheating

Along with your stepper itself overheating, the stepper drivers can overheat. Most drivers will come with very small heat sinks that definitely help, and if yours does not, I highly recommend applying them. Though this helps, it is not enough in itself to entirely prevent stepper drivers from overheating.

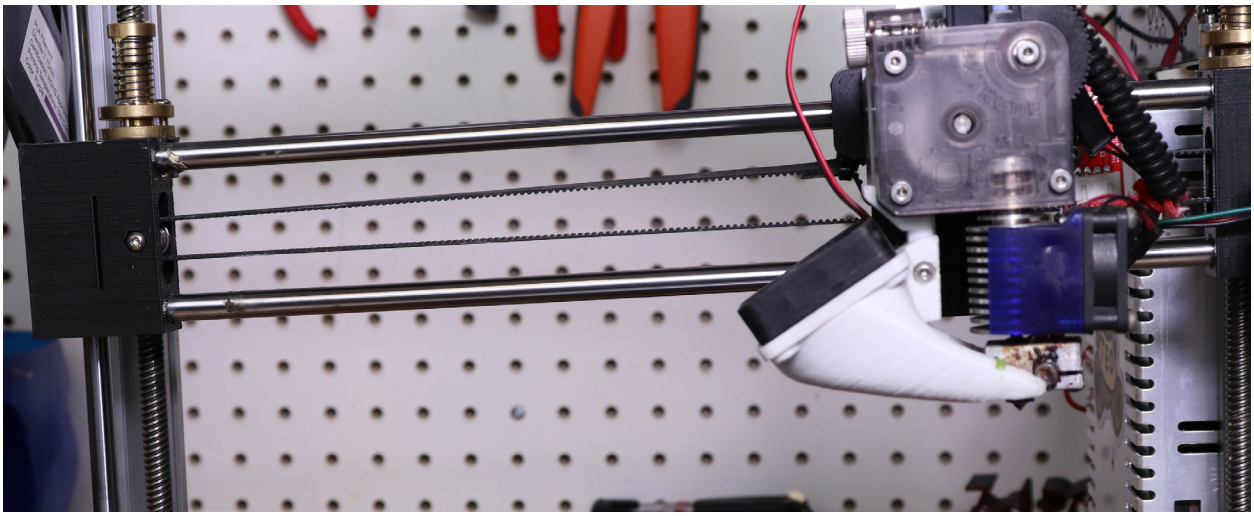


As mentioned elsewhere in this book, you will always want 1-3 active cooling fans blowing onto your board. This can do wonders when it comes to overheating drivers and other parts on your board. Just remember that if you do not have a filter on these fans you will see dust accumulate which can cause the fans to fail over time. Always make sure your printer is off when it is not being used to not collect unnecessary dust, and to clean these fans/your board periodically. Almost every pre-built printer comes with these fans on the board standard, so periodically check to make sure this is spinning well.

If still experiencing overheating, you will want to check that the current going to that driver isn't over the rated limit via the methods described in the "Extruder Motor Skipping" chapter, or in the "Layer Shifts" chapter.

Unlevelled X carriage

In the "Unlevelled Build Plate" chapter in this book I mention that it is important to level the right and left side of your X carriage in relation to how close it is to the build plate. These can become unlevelled over time, but you will see some massive differences if one Z stepper driver is malfunctioning.



If you notice your printer is looking like the photo above, you likely have a malfunctioning stepper (or stepper driver) for the Z-axis motor(s). It could also mean one of the parts on your frame is broken, so inspect closely.

This, as well as any other stepper malfunctioning, will have to be remedied as follows:

Plug in a different stepper motor (or plug the current stepper motor into a different driver)

You can test to see if it is your stepper or driver/board that is malfunctioning by swapping the stepper motors. Unplug the stepper in question and take those same wires and plug them into a different stepper you know is working properly (either on your machine or a spare that you have on hand). Attempt to move the stepper again by moving the axis in question and see if it is working properly. If it is, this means you will have to replace the motor that was malfunctioning.

If when you plug in this different stepper that you know is working, and it does not spin when moving the axis in question, you have a problem with either your stepper driver or your board. This is assuming you have confirmed that all wires in question have continuity (as described below).

You can achieve this same outcome by using the motor in question but plugging it into a different driver that you know is working.

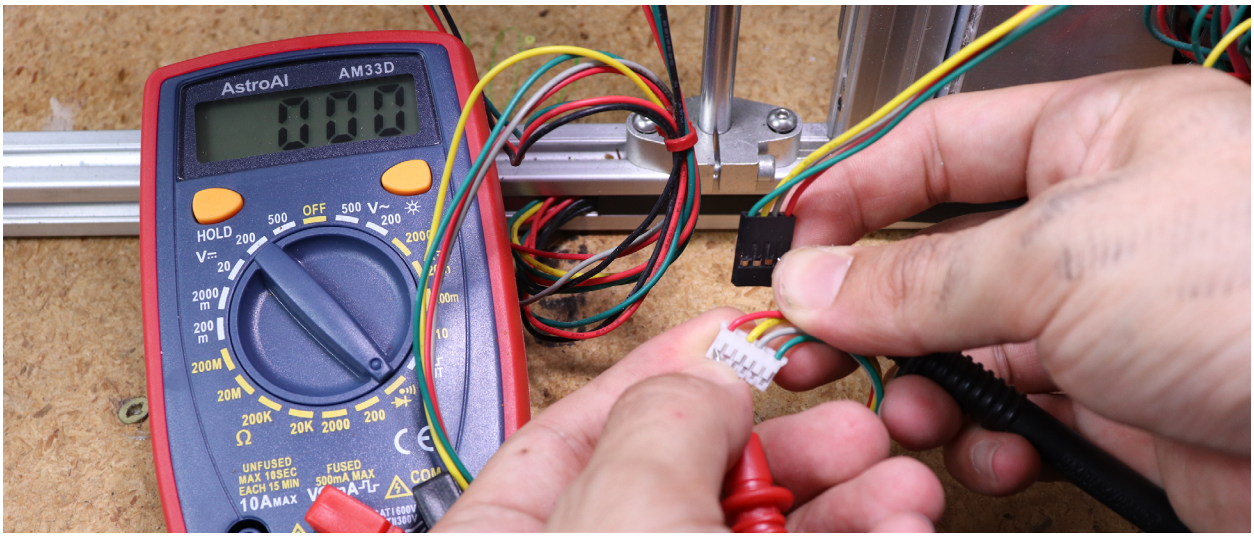
Replace a motor if it is the problem, and move on to the next step if it is working fine.

Check the continuity and wiring of the stepper in question

With the frequent rattling and movement of your machine, wires can easily get

caught mid print and become either disconnected or frayed to the point they are not providing continuity. The first thing you will want to do when you see a stepper is malfunctioning is to check the continuity of each wire.

If not visually frayed, you will need to take your multimeter and switch it to the continuity tab. Turn off and unplug your machine. Then unplug the connector for the stepper in question and its connector to the board.



Take either lead and touch it to one of your wires inside the connector to your motor (be sure it is touching metal). Take the other lead and repeat the process for the connector that is going to the board for the same color wire you are testing on the motor side.

If there you hear a beep that means there is no break in that wire – there is continuity from the starting point of the wire to where it connects to the motor.

Continue the same process for each colored wire. If you notice that one of the wires is not beeping, that means there is a break in the wire somewhere. Follow that cord from the motor to the board and see if there is anything you can see with the naked eye (frayed, cut, or burnt out sections of the wire). If it easy to spot, cut that section, re-solder, and confirm that there is continuity.

If there is no physical damage you can spot with your naked eye, you will want to replace that wire entirely. Once you confirm all wires have continuity from the board to your motor, everything should be working properly again. If not, move on to the next step.

Test your stepper driver by swapping it

Once you have confirmed that your wires all have continuity and that the stepper motor itself is working, you will want to test the stepper driver in question. This is simple and can be done with any spare stepper driver you may have, or by just testing a driver for a different motor you know is working.

Unplug the driver in question and plug your spare/new driver in. If everything works just fine, you know it was the driver that was malfunctioning. Always make sure you are using the proper stepper motor for your machine setup.

Final fix - replace your board

Finally – if all wires are showing continuity, the motor is proven to be working fine, the stepper driver is functioning just fine, it's likely that you unfortunately have a malfunctioning board. RAMPS boards are very inexpensive but some printers other boards such as a RAMBo that may cost close to $150.

Once you have made it this far in your testing, and you made sure you are using all the proper parts, I can almost guarantee the issue remains in the board itself. Boards that require NANO fuses will not work at all if the NANO fuse is blown, so if you are able to heat your extruder and move different axis's, then it is not a NANO fuse problem.

Buy a replacement board, or get a free replacement if under warranty, and embark in the annoying process involved in unplugging and re-plugging everything (with your printer off and unplugged of course). Some machines can be extra frustrating to do this on because of the limited space you have. You unfortunately will not be able to just test the one motor in question until everything is wired and your board is flashed with the appropriate firmware.

Once everything is plugged in, and the correct firmware is flashed, test everything out. If all the proper tests above were taken, everything should now be working fine.

Remember to test all wires and motors before going this final route.

Summary of Fixes and Precautions

- If motor is extremely hot to the touch during a print, add a heat sink
- If motor is still overheating, add an active cooling fan
- Add a heat sink and active cooling fan to an overheating stepper driver
- Check current on stepper driver if still overheating
- If motor is malfunctioning regardless of heat, check continuity of all wires from it to the board
- Replace or fix all wires that do not show any continuity
- Test a different motor or plug a different driver's cords into the stepper in question to test if the stepper itself is the culprit
- If stepper itself is not working, replace it
- If stepper is functioning, check the driver by testing a spare

- If driver itself is not working, replace it
- If all above fails, purchase and install a new board

Stripped Filament

If you are noticing that your filament is stripping during the print, it may result in a part that look under extruded. This is because the shaved filament will not be fed through the extruder.

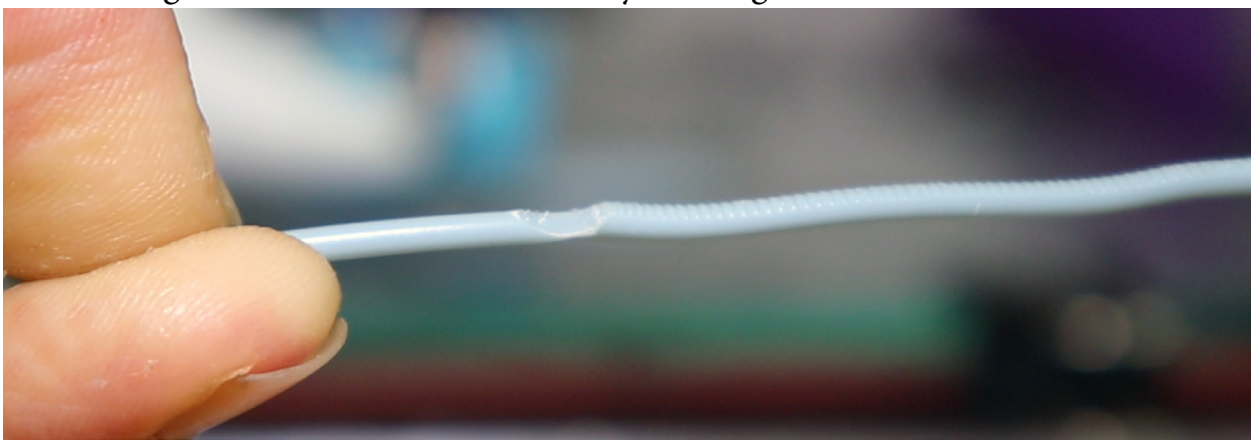
Constant filament stripping without any cleaning being done can lead to your extruder slipping, due to the teeth on your hobbed gear/bolt becoming less sharp as filament wedges itself in between the grooves. All of this will cause further under extrusion and failed prints.

Moisture in your Material

The number one reason this problem will consistently occur is due to the material having absorbed too much moisture. When this first happened to me I did all of the steps below, but no matter what I did, the particular spool of PLA would just grind until it could no longer print. This was on a direct drive Titan extruder that I had checked multiple times for any problems.

Well, I eventually put a different spool of PLA on and it printed great. It ends up that the spool of PLA had absorbed too much moisture and could not print properly. It is easy to tell when this happens to PLA, because you won't be able to snap the material on the spool via bending. You can notice this if you see bubbling when extruding, but another is if the material continually grinds down.

All filaments require you to store them in a low humidity area – either via vacuum sealing with desiccants, or being left with a dehumidifier set to around 25%. You should make note of where you live, since this won't be nearly as much of a problem for us living in the desert, versus those of you living in humid climates.



If your filament keeps getting grinded until it looks like the photo above, you will either need to replace the spool, or get it dried out.

You can purge out the excess moisture a couple of ways – the best doing a

minor vacuum purge after heating to a bit under the materials glass transition temperature. Most of us do not have a vacuum, so here are the steps for what you can do with the tools you have at home.

First – check the glass transition temperature of the particular material you are using. PLA is right around 60 degrees Celsius, so you don't want to get it hotter than that. Turn your print bed to this 60 degrees, and then place the spool of filament on the print bed. Cover it with a cardboard box (you can use the one it was shipped in). Leave it there for roughly 1 hour. The heat will help to evaporate the moisture from the spool, and the fact the cardboard can absorb moisture will help as well. You can throw in some desiccants to help.

It took me 3 rounds (3 hours) of this and it fixed my moisture filled spool of PLA perfectly. I was able to print as if the spool was brand new without any grinding filament.

This may not always work, so you want to make sure to use the proper precautions to avoid ever having to do this. Nylons are much more likely to absorb moisture, and thus would be a bit more difficult to complete this process.

Clean your extruder

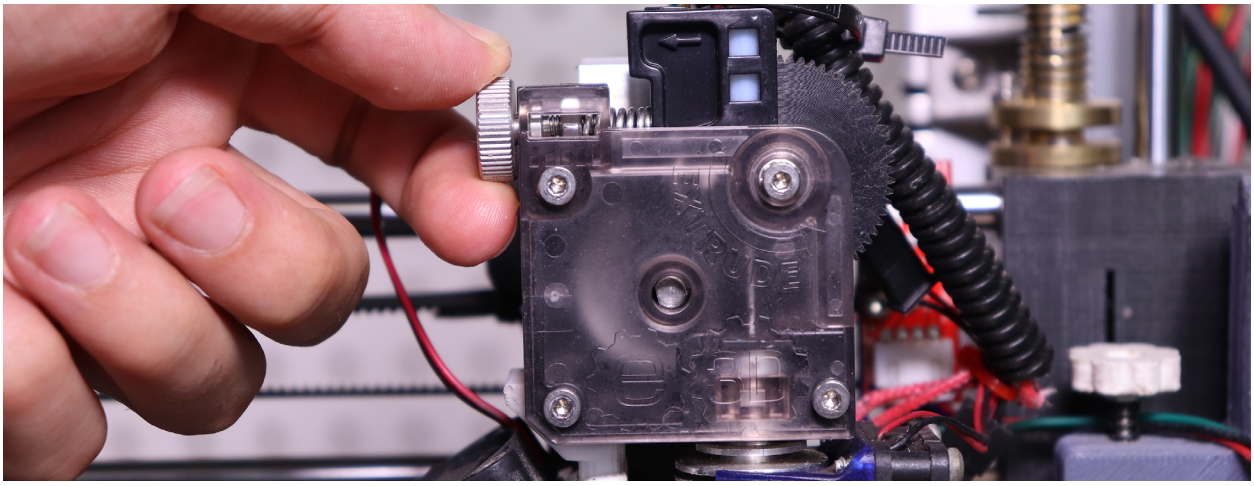
As I mention in the "Good Practices" chapter early in this book, keeping your parts clean is important to have consistent prints without failure. This definitely includes your extruder.

If you are experiencing stripped filament, or would like to prevent it as best you can, grab a small wire brush and pick and clean out the teeth on your hobbed bolt/gear.

If these grooves are not defined, they will not grip properly to your filament. This can lead to filament slipping and stripping.

Some high-end extruders even come with the ability to self-clean themselves so that you don't have to worry about this.

Check idler tension



As mentioned for other issues in this book, you will want to make sure you have the proper tension on your extruder idler. Too tight of an idler can cause the extruder motor to skip and to even cause filament grinding to the point of snapping.

While grinded filament can occur from too tight of an idler, it can actually also occur from an idler that is too loose. This is because no consistent grip is attained by your hobbed bolt/gear, and certain rotations will just rub against the filament, rather than pushing it through your hotend.

It is important to get this tension right since it can cause various issues in the quality and consistency of your prints.

Increase hotend temperature

I always recommend this with caution because if you go too hot, materials will easily get clogged in the barrel, causing you the need to read the "Nozzle Clogs" chapter in this book. But as mentioned in that chapter, running the hotend too cold for the speed you are feeding can lead to under extrusion.

Your extruder stepper may be turning the proper amount of steps, but bottlenecking in the nozzle will cause your hobbed bolt/gear to just rub against the filament when the pressure is too high.

If you are under the high end of the temperature range for your material, you can attempt printing the same G-code at a slightly higher nozzle temperature in order to help the viscosity of the material.

Don't go over the materials temperature range, since running PLA at 230°C can give you further problems with clogging in the barrel and oxidizing material.

Lower the speed/acceleration

Just like with the solution described above, lowering the speed and acceleration can reduce the amount of bottlenecking caused in the hotend/nozzle.

You can imagine that if you were to speed your extruder up 10 fold you will clearly grind your filament, since it cannot be fed through your nozzle at that speed.

You do want to make sure you have a properly cooled barrel and that you understand what material you are using in order to reduce the threat of heat creep. Certain materials have a very small range of temperature and speed they can properly be extruded at without a clog being formed.

If you would like a tutorial on how to reduce the acceleration on your machine, please refer to the "Extruder Stepper Skipping" or "Settings Issues" chapter in this book

Printing Fast on Large Diameter Nozzles

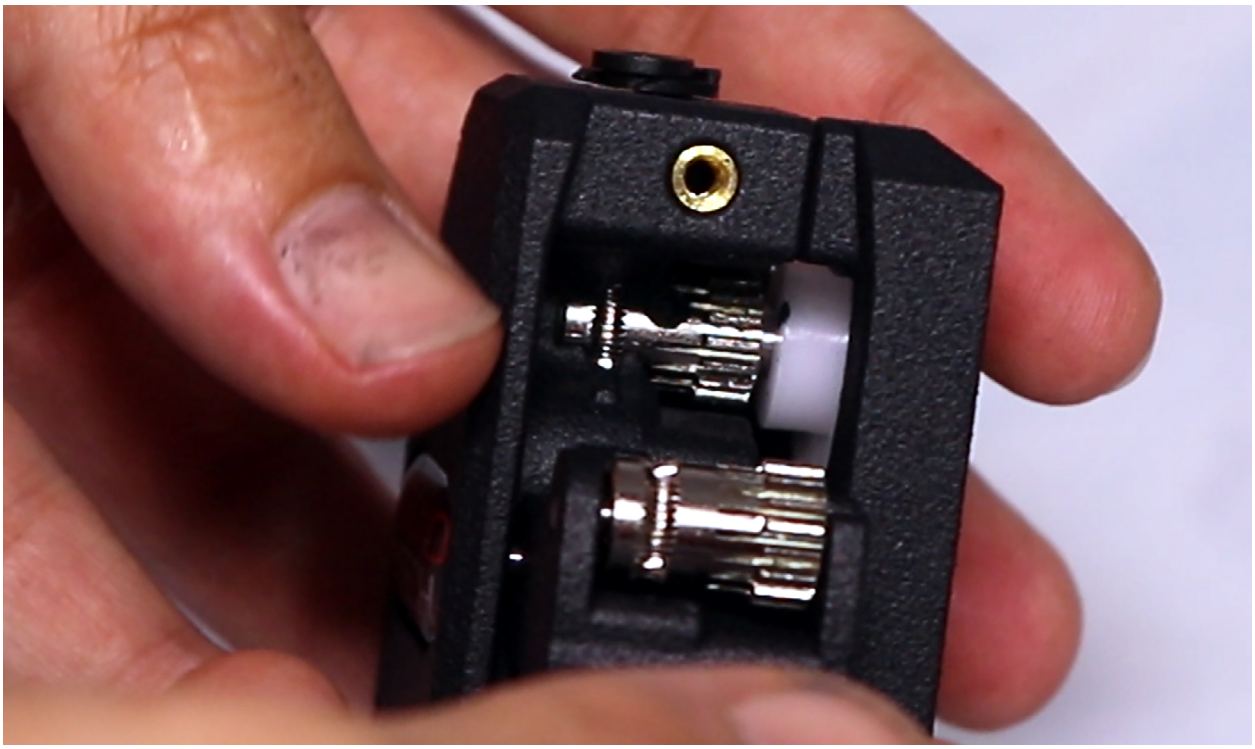
You will need to slow down prints when attempting to print large layer lines on a large diameter nozzle with a standard setup. This is because the material needs a

certain amount of time to heat and reach it's proper viscosity to extrude. Since you are pushing so much material through the nozzle, it needs to be slowed down to prevent filament grinding.

If you will be printing on large diameter nozzles (Larger than 0.6mm) frequently, it is smart to upgrade to a hotend setup that is specifically designed for high volume extrusion. E3D has their Volcano setup which is meant for just this purpose – printing fast with large diameter nozzles and layer heights. You will clearly need a geared extruder to even achieve these speeds, which is why they also sell their Titan extruder. You can use the Bondtech BMG in conjunction with a Volcano hotend for the fastest printing possible.

Replace hobbed bolt/gear

Please keep in mind that cheaply made hobbed gear and bolts may not have the proper spacing, sharpness, and depth required to grab onto the filament. These components also wear over time, particularly if you are running carbon or glass filled materials. If you are experiencing constant filament grinding regardless of the steps you took above or the material you are using, you should purchase a new extruder hobbed bolt/gear from a reputable manufacturer. Choose stainless steel options over aluminum for this part, if available.



I currently use the fairly new Bondtech BMG extruder which actually has two gears grabbing onto the filament, rather than just one pushing against a bearing. This dual drive extruder has made it so that I haven't had any grinding or extrusion issues. I cannot recommend this extruder enough, though it does currently come

with a hefty price tag ($100 at MatterHackers.com).

Summary of Fixes and Precautions

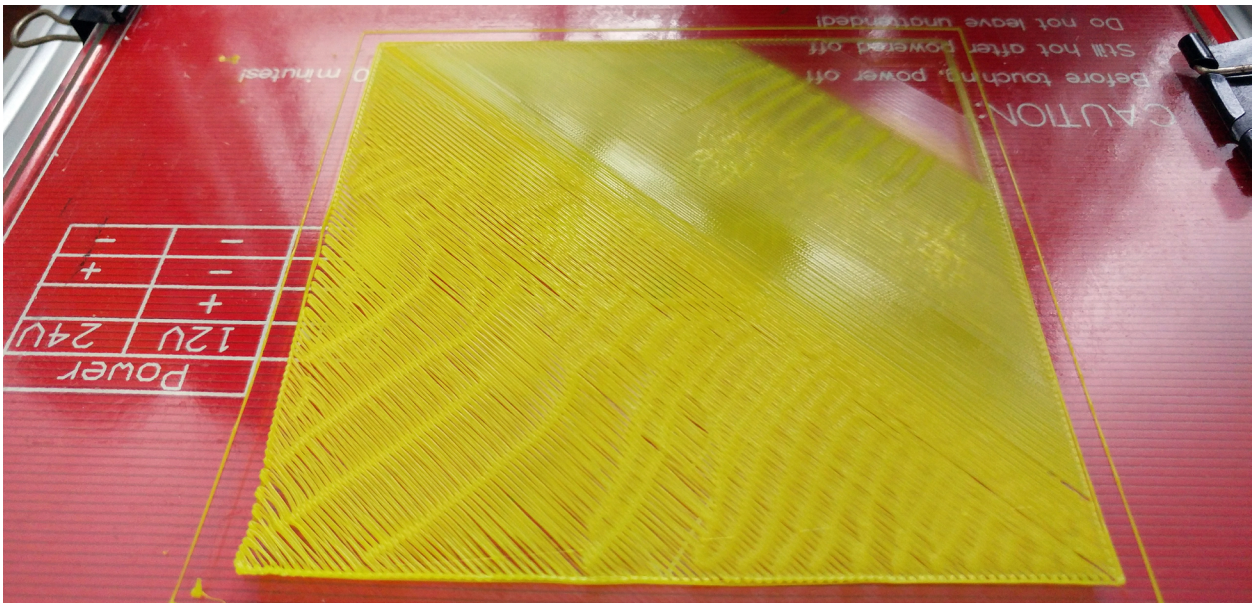
- Confirm the material you are using hasn't absorbed too much moisture.
- Replace filament, or dry out your material if it has absorbed moisture.
- Clean your extruder by making sure there is no filament stuck in the teeth of the hobbed bolt/gear on your extruder.
- Check to make sure you have the proper tension on your extruder idler.
- If you are below the maximum suggested printing temperature for your filament, try increasing it slightly.
- Reduce the speed and acceleration on your print. Make sure your barrel is properly cooled.
- Replace poorly made or worn-out extruder hobbed bolts and gears.

Unlevelled Build Plate

An unlevelled build plate will mean that the nozzle is too close in some areas and too far away in others. If the bed is extremely unlevelled, there may be structural issues with the way the build plate is mounted to the frame, however, in most cases it is minor (less than a mm height difference from one corner to another). In these situations you just need fine tune adjustments to the spring loaded corner mounts of the build platform.

Many new FDM 3D printers come with an auto bed leveling system that actually works, and there are some third party bed levelling sensors that are useful, unlike attempts in the past. I go over these a bit further in the "Upgrades and Purchasing a New Printer" chapter, as well as a later section in this chapter.

You can easily diagnose this problem by either running a bed calibration G-code, or by paying close attention to the first layer of your print. By getting used to what a print should look like on its first layer, you will easily be able to which corners are too close, and which are too far.



The photo above is easy to diagnose this – the front left corner has the nozzle too far from the build plate and the top right corner is far too close. Somewhere in the middle the nozzle is the proper distance.

An unlevelled build plate, along with Z-height calibration, are the two most common failures when starting a new print. Most printers either have bed clips or some form of mechanical leveling system. These screws become loose over time, and even one long print can cause your bed to become unlevelled for the next run.

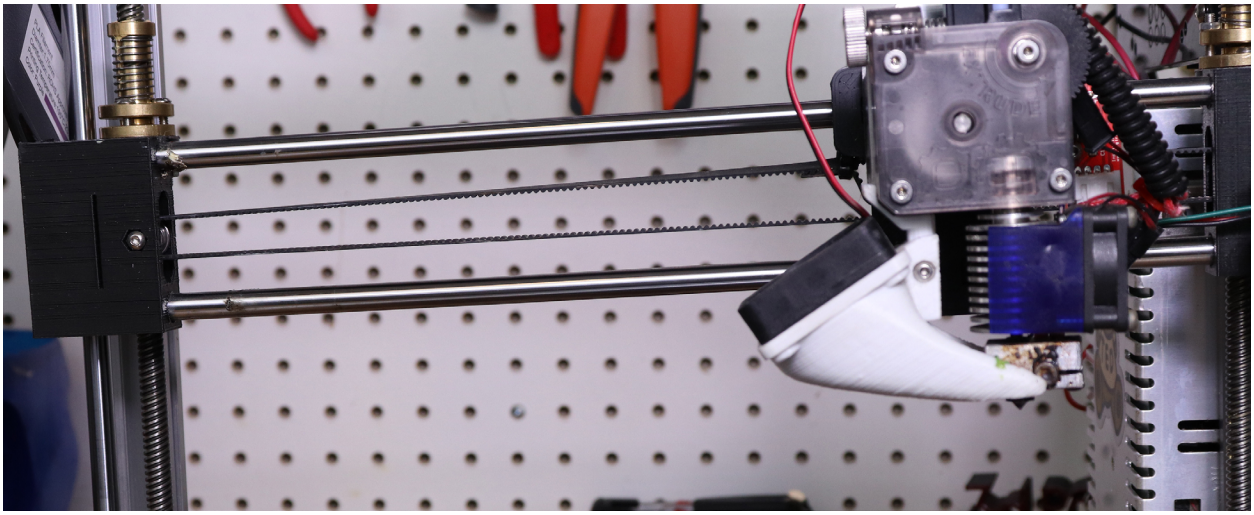
If you continue to only tighten corners without adjusting your entire bed, it can

result in a warped metal build plate, and eventually constant layer shifts - as mentioned in that chapter in this book.

Levelling left and right side before touching corner bed clips

Before you even bother adjusting the corner bed clips, you will want to level the left and right part of your X-carriage. Variations over time, moving your printer, as well as times that you home the Z axis too close to the bed, can all cause your two Z- rods to become unleveled from each other in relation to the X carriage.

If this is just a maintenance problem, and not an issue with your stepper motor (as mentioned in the "Steppers Motors Overheating or Malfunctioning" chapter), you can level this while your steppers are disengaged. Hold onto the left coupler that attaches the left threaded Z rod to its respective stepper motor. While holding onto the left coupler, twist the right threaded Z rod's coupler in the correct direction to level your X carriage.



You can measure the distance from the bed to the X carriage rod on the right and left side of the build plate, and continue the above process until they are even.

Loosen/tighten corner bed clips

If you have confirmed that the left and right side of your X carriage are level to each other in relation to the build plate, and you are still experiencing an uneven first layer, you will want to play around with the corner bed clips of your printer.

I always suggest that you loosen before you start tightening these corners. That means you should try to loosen the corners that are further from the nozzle rather than tighten those that are too close. So if one corner is too close to the nozzle, I would suggest raising the Z-height slightly and loosening the remaining 3 corners until level. This is because you do not want to be in the situation where you cannot tighten a corner any further. This leads to a warped metal plate overtime and will

cause layer shifts in the Y-direction since it becomes rough to move. This is not nearly as much of a problem on CoreXY machines since the build plate only moves down in the Z-direction.

This is not always possible though, because you can imagine the above situation where 1-3 corners are too close to the nozzle and the remaining corner(s) are as loose as they can go. You can always use a longer screw and larger spring on those corners, but if that is not feasible, you will want to tighten those corners that are too close to the nozzle.

On newly built, or strong framed machines, this can be as simple as doing a slight tweak while a new print begins. Printers that have a lot of build time racked up can lead you to having a massive headache going forward with the above method. If you are unable to level the build plate easily with the above two methods, I sincerely suggest entirely starting over.

This means that you should take your build plate apart, make sure everything is assembled properly, and put it back together. Make sure that all corners have the same length screws and same exact springs and are all equally tight. Then confirm once again that the left and right side of your X carriage are level. Find the proper Z-height and then run a bed leveling G-Code that you can easily find online if not provided by your printer (just search on Thingiverse for a bed level test). Starting from scratch like this can often make this issue go away.

If you still cannot level your build plate after all of this, you likely have an issue with the frame of your printer. Take the metal bed plate entirely off and lay it on a flat surface. You can then push on each corner to see if it has warped. If this is the case, you will need to order a new metal plate from your printer manufacturer.

If the plate is flat, confirm that your frame is built properly. This means that if you are using t-slotted aluminum rods, confirm that everything matches up in a 90° angle and that it lays flat on the tabletop you are using. Some printers have little feet that make sure your printer lays flat but they break off easily.

Confirm that all printed parts on your frame are not warped. Check to see that all bearings are popped into their holders and that everything is equal distance from each other. It would be smart to go over the "Mandatory Maintenance" chapter to make sure everything is working as it should.

If you are using an acrylic framed printer this problem will become more prevalent over time. Metal framed printers experience less warping and bending over time.

One or multiple corners of your build plate may not be laying perfectly flat because of the issues above.

Larger layer heights/nozzle diameter can make this process easier

If you are using a 0.25mm or smaller nozzle with 0.1mm layer heights or lower,

you will notice extremely slight variations in how level your build plate is. As mentioned in the "Z-Height Calibration" chapter, the lower the layer height and nozzle diameter, the more difficult it is to get your first layer to stick to the build plate.

On any print like this I would suggest increasing the initial layer height on your slicing program. Increase the first layer to be 0.2mm and you will have a much easier time leveling the build plate. The same is true if you are using a very small nozzle.

0.4mm nozzles can get most jobs done and will allow you to get that bed adhesion that is so hard to come by on 0.15mm – 0.3mm nozzles. On a 0.4mm nozzle I have 0.25mm or 0.3mm layer heights for the first layer, which can save you an immense amount of headaches.

A build plate on a printer using a 0.6mm nozzle with a 0.3mm initial layer height will be relatively extremely easy to level when compared to a 0.15mm nozzle with 0.05mm layer heights.

Please refer to the "Settings Issues" chapter for a further explanation.

Do not frequently move your printer

FDM 3D printers are not meant to be travel ready and should stay in one spot as much as possible. Every time you move your printer, especially when traveling in a car, minor adjustments can occur that will require you to recalibrate.

The most common issue with moving your printer is having to readjust an unlevelled build plate. When we would take printers to fairs and events in the past, we would ALWAYS have to re-level the bed when setting up. Take care not to bump into, move, or vibrate a printer, even when it is not in use.

Purchase an auto bed leveler

You can purchase a part that will allow you to auto bed level your machine. You will still need to get the proper Z-height, but your printer makes minor movements in the Z-motors to make sure the bed is level.

I personally upgraded my CR-10 to use TH3D's EZABL which will take a reading of the bed before printing to make sure that first layer is perfectly level. This part takes 16 points on the bed in order to get a mesh, though you can change this number to be more or less precise. After honing in the Z-off set, it then uses the Z-motor (or motors) to vary the height of the print as the nozzle moves. This means if one corner is too far from the nozzle when compared to another, the z-motors will tweak to make sure the nozzle is closer in that section.

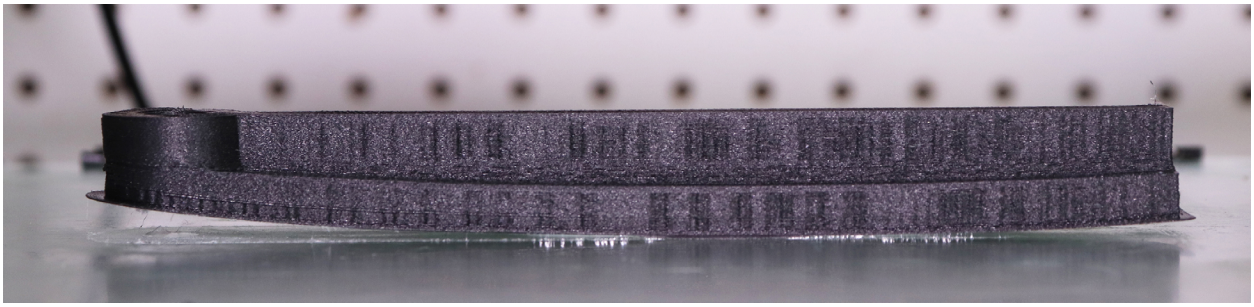
This minor motor movement can be seen when looking closely at the first layer. I am extremely happy after upgrading to this since I no longer have to deal with this particular failure on this machine.

Summary of Fixes and Precautions

- Confirm the left and right side of your build plate are equally level to the X-carriage.
- If they are not, while stepper motors are disengaged, hold onto the left threaded Z-rod's coupler and twist the right threaded Z-rod's coupler until level.
- Loosen the bed corner clips that are too far from the nozzle relative to the remaining corners. Tighten only when needed.
- Start from scratch by dissembling build plate and making sure all corners are equally tightened, using same sized screws and springs with equal amount of tension.
- If still experiencing issues, confirm that your metal build plate is not warped, the printed parts on your machine are not warped, and that the frame is put together properly.
- If any part is warped or off tolerance, you will need to purchase or print replacements.
- Increase your initial layer height, or even increase the diameter of your nozzle if you don't want to deal with this issue as frequently. A large initial layer height makes leveling the bed much simpler.
- Take care not bump into, move, or vibrate a printer - even when it is not in use.
- Upgrade to an auto bed leveler to prevent this problem from happening.

Warping:

The warping of parts is just about inevitable if you don't understand the material or machine you are using. Warping is when corners or entire parts of the print curl upward due to uneven cooling of your part, or from improper bed adhesion.



Understand the *Bed Adhesion* chapter, *Z- Height Calibration* chapter, and the *Unlevelled Build Plate* chapter

You need to entirely understand all three of these chapters in order to even start to try and fix your warping problems. A print will have an exponentially higher chance of warping when either part of the print, or the entirety of the print, is too far from the build plate.

This is fairly easy to understand because the further the nozzle, the less bed adhesion that is involved, the higher chance it will curl up later in the print.

You will need a brim on any material that has a high shrinkage, rate such as ABS. For large non-circular ABS prints you will need an ABS slurry if you cannot maintain an ambient air temperature of at least 45°C – all things covered in those three chapters.

Understand the material you are using, and possibly use an alternative

Much of this is covered in the "Materials and their Settings" chapter, so please review this as well when understanding your material needs.

You will almost never experience issues with warping when using a material such as PLA, because PLA has an extremely low shrinkage rate. Very large, highly dense PLA parts should use a heated build plate and a brim, but it is very uncommon to get warping with PLA on a level build plate.

ABS is an entirely different matter, with shrinkage rates from 0.4% - 1.2%,

depending on your particular spool and manufacturer. Since ABS also requires a higher temperature for its build plate due to its higher glass transition temperature, the differential between the bed and the ambient air is also increased.

While ABS is great for its price and functionality, this factor may make it impossible for you to achieve certain parts on your machine without warping. This is why it is important to understand the factors and features you are looking for on your print and if you can use an alternative material.

If you require mechanical functionality but do not care about acetone vapor finishing or a high glass transition temperature, you should definitely try out PETG. PETG is a great material for the vast majority of mechanical applications you were attempting to get out ABS, and has a very low relative shrinkage rate. It also has a competitive price tag.

PETG has some serious adhesion when a bed is set to 70°C, and I have yet to experience warping on a level build plate when using this material.

If you really need strength, you can try out Carbon Reinforced Nylon, since that is the strongest material I have tested to date – just keep in mind it has a high price tag. This has a chance of warping if you do not use the proper precautions, but it is far easier to print than ABS.

If you require a high glass transition temperature from your material, I haven't personally found a great alternative to ABS with a low pricetag. One of your best options would be Carbon Fiber Reinforced ABS by 3DX Tech. This material has a great glass transition temperature of 90°C and warps far less than standard ABS, while maintaining its ability to be acetone vapor finished. The biggest drawbacks to this material is the price compared to ABS, increased degradation of your nozzle, and a lower bend to break ratio. The problem with most materials that have a high glass transition temperature is their likelihood to warp.

Certain nylon materials that do not have a high glass transition temperature will still have a high probability of warping. For these materials you will need to use a coat of PVA on a glass build plate, as described in the "Bed Adhesion" chapter.

I have attempted to print Polypropylene and Acetal (Delrin) in the past, to essentially no avail. Along with requiring a high temp hotend, Polypropylene warped on just about every print larger than a cubic inch, and Acetal was impossible to get any print started without using a sheet of cardboard on top of the build plate. They just do not want to stick to standard build platforms.

Polycarbonate ABS has an amazing glass transition temperature of around 120°C but will require an enclosed environment on any print larger than a cubic inch. This material's warping probability is just too difficult to overcome on most machines.

Using an ABS slurry, as described in the "Bed Adhesion" chapter, can help with particular ABS blends to truly stick to the build plate.

Printing in an enclosed environment

When you are printing a part on a heated build plate you are automatically working in an environment with uneven ambient temperatures. When the room is around 30°C and your heated build plate is 110°C, there is a quick change in temperature for parts close to the build plate yet exposed to the surrounding air.

Printing in an enclosed machine allows for the ambient air to remain a bit hotter, due to the trapping of the heat given off by the build plate.

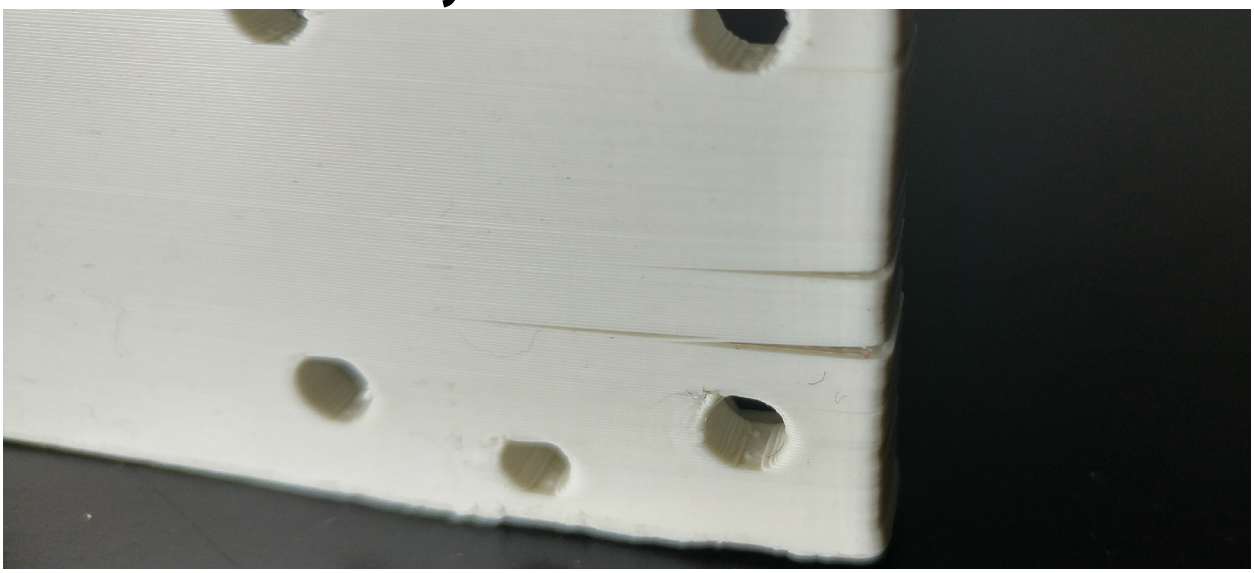
You can purchase a printer that is enclosed, or somewhat enclosed, which works pretty great if you can afford them. You can also build a DIY enclosure with laser cut acrylic and a few printed parts. Or you can find some other build that someone has posted instructions for online.

When printing a part with a high likelihood of warping in an enclosed machine, you will want to let the bed sit at its printing temperature for around 5-10 minutes to allow the ambient air to heat up. A good ambient temperature for ABS would be 45°C, and ideal up to 60°C. You obviously would not want to print PLA in that type of environment though, since that is right around its glass transition temperature.

Many issues arise when you allow ambient air to rise this high. Stepper motors and other electronics will overheat and cause your printer to malfunction. This is why you will need to have your power supply and board outside the enclosed chamber, have enough heat sinks spread throughout, and keep an active fan on anything that is heating too hot. Read up on the "Stepper Motors Overheating or Malfunction" chapter as well.

Even then you may still experience issues, so be sure to understand some basic thermodynamics and mechanical engineering before getting your ambient air to 50°C or higher.

Delamination of layers



You may not consider this failure as "warping", but it has nearly all of the same reasons for happening. This is why it is not included in the "Poor Layer Adhesion" chapter.

If you have incredible bed adhesion, such as when you use an ABS slurry, but are printing a large part in an open environment, you can experience delamination instead of warping.

Delamination is when two layers will separate from one another, even when taking in all the layer adhesion precautions. This is because of the same temperature gradients explained earlier, but occurs when bottom layers are stuck extremely well to the build plate.

The bottom of your print may not curl upward taking the entire print with it, but rather layer adhesion becomes the breaking point for this shrinkage.

If this is happening to you, you will need to check your settings or drastically change your environment/material being used.

I have only experienced delamination on very large PLA prints when the ambient air is quite cold, while it can be unavoidable on tall ABS prints not in an enclosed environment.

Your settings can be tweaked to help prevent this. The denser your part is on the inside, the more likely this will happen, so try printing your part with less infill and a couple more shell walls (if possible).

Finally, confirm that your E-steps are on (as explained in the "Over and Under Extrusion" chapter). If you are under extruding by a decent amount, you could potentially experience more frequent occurrences of delamination.

Make sure the build plate is not losing heat mid print

If your board is overheating or you having issues with continuity to your heated build plate, the temperature may drop mid print. If you only watch the beginning of your print and come back when it is finished, you may not even notice this is happening other than returning to a warped part.

Make sure your bed is maintaining its heat throughout the entire print, and if it is not, refer to the "Build Plate Not Heating" chapter

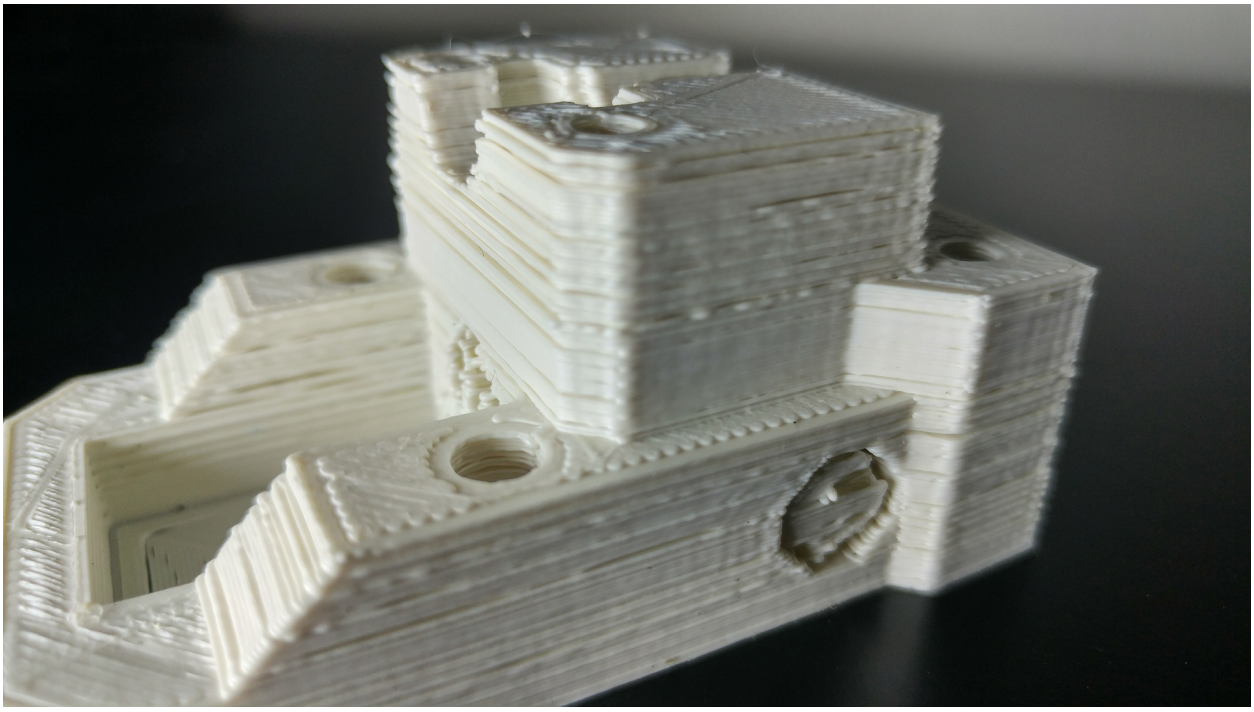
Summary of Fixes and Precautions

- Make sure your bed is level and that your Z-height is correct when starting your print.
- Use your preferred method of Bed adhesion. Some large ABS prints on a non-enclosed printer will require an ABS Slurry.
- Print with a Brim.

- Know the proper print settings for the material you are using.
- Attempt using a material with a lower shrinkage rate with similar applications (PETG, CFR-ABS).
- Use an enclosed 3D printer, or build an enclosure for your machine, but understand the possibility of parts overheating.
- Delamination occurs on taller parts that have good bed adhesion. You will need to reduce the density of your part, print an enclosed environment, or use a different material in order to fix this problem.
- Make sure the build plate is maintaining its temperature throughout the print.

Z- Axis Wobble

When you are experiencing Z-axis wobble you will see what looks like repeated minor layer shifts, sometimes with every single layer. If the wobble is minor, it may just look like the print surface is not clean. You can hone in all of the slicer settings properly, but it will not fix this issue.



Tighten extruder carriage and hotend setup

This is probably the most common cause for a wobbly looking print. Your extruder carriage is moving and rattling constantly over prints and screws loosen over time. This is especially common on 3D printed parts, since their tolerances are not as tight as mass produced carriages and extruders. This will be far less of an issue if using an e3D Titan extruder with an e3D hotend on a $3,000+ machine than on a $300 stock Prusa i3 with printed parts.

This is because more expensive machines usually have higher end parts holding everything together. Lower end machines will use printed parts with minor tolerance issues, and may even use zip ties to hold belts and bearings in place. These zip ties can become stretched over time and results in a carriage that is not harnessed tight.

Frequently check to see if you can easily move or rattle your hotend/extruder setup. Lower the print temperature to room temperature and literally grab the hotend and try and move it around. Don't be rough, just give it a little rattle. If you see the

carriage or hotend rattle, you will need to fix that in order to get rid of this problem.

If you can, you will likely need to tighten the screws/bolts that hold the extruder or hotend to the carriage. If tightening doesn't fix the problem, check to see if your printed parts are worn out or off tolerance. If they are, you will need to reprint them. If not, you can move on to the next solution

Replace old/worn out bearings and make sure they are harnessed tight

If your carriage is rattling when you manually try to move it, but your entire extruder carriage and hotend are tightly set up, your bearings may have become loose over time. This is far more common with plastic bearings than with metal ones, as you can imagine.

When doing the step above – rattling the hotend when at room temperature – you should be able to see if there is a small gap in the bearing.

I didn't think this could happen until I replaced a 2 year old printer's bearings who was experiencing consistent Z-axis wobble, and it fixed the issue instantaneously.

If you are using a less expensive machine that uses zip ties or something similar to hold the bearings in place, you will need to replace these with new zip ties, or find a part online to print that will hold the bearings tight.

Ensure your hotend and nozzle is set up properly and tight

This process is explored in the "Built up Material in Nozzle" chapter, but can also show symptoms in a Z-axis wobble. Every hotend setup needs to be assembled in a slightly different fashion, but nearly all of them require you to not over tighten.

When the hotend heats, the metal expands and can cause your once tight nozzle/heater block to actually have minor gaps. This can lead to rattling of the heater block throughout the print, causing an ugly wobble in your print.

If you notice that your heater block is loose when hot, or that you constantly have to brush off the nozzle from excess material, you will likely need to tighten these parts.

I always suggest doing the final tightening of your nozzle and heater block when heated to 240°C using proper gloves and tools. Remember that you have a high chance of burning yourself, so only do this with extreme caution. If you tighten the hotend/nozzle when at room temperature, you will find it won't be tight at 240°C.

You also want to make sure to not over tighten anything. I have broken quite a few heater blocks, nozzles, and heat break barrels due to over tightening. These parts, especially when hot, can easily snap under pressure. This hasn't happened to me in a while, but would occur when I didn't understand how careful I should be. When you are doing this, make sure to only tighten until you know that the nozzle and

heat block are not loose and will not unscrew during the print – don't muscle it.

If you still are experiencing your hotend is rattling and you have made sure it's harness is tight, you will likely need to upgrade or replace your nozzle, heat block, or entire hotend. Poorly made parts will not have tight tolerances and can lead to these gaps in your threads.

Make sure the build plate is harnessed tight

Just as with the extruder carriage and bearings, you will need to make sure your print bed does not have any rattling in it. This will NOT be an issue with CoreXY machines, since the bed only moves up and down.

When using a Cartesian machine, free-play or rattling in the build plate will result in Z-wobble. When not printing, and with the bed at room temperature, give it a good rattle up and down. The print bed should not have any movement other than what comes from the whole machine moving. If the print bed has some free play in the bearings or harness, this will need to be fixed.

On one of my inexpensive DIY machines, the build plate harnesses are attached to the bearings via zip-ties. These zip ties seem to stretch over about a month or two of printing, and so I will cut them off and replace them as needed.

Just as with the Carriage, you will need to replace any plastic bearings that have become worn out over time.

Finally, make sure that the parts that are connecting the bearings to the build plate are securely tightened and up to tolerance. Take your glass or other print bed off, and then tighten all of the screws that are connecting everything. These, as with all other screws, will loosen over time.

Tighten all belts

Other than confirming all harnesses are tight and that there is zero rattling on the extruder and build plate, the next most common reason for Z-wobble is a loose belt.

As explained elsewhere in this book, it is possible to over tighten a belt, but it is pretty difficult to do so on low end machines where the belt is just held together via zip ties. Both the X and Y axis belts should be very springy to the touch with zero-droop.

If there is any droop in your belt, you will need to tighten. For low end, non-upgraded machines, cut the zip tie that is holding the belt together, grab some pliers, and pull tight as you put on a new zip tie. Make sure the belt is tighter than it was and that the zip tie is pinching everything so that the belt won't slip.



Even better than doing this would be to print a manual way to tighten your belts. There is likely a file on Thingiverse for your specific machine setup, you would just need to search. I have added such a way to tighten the X-axis belt on my CR-10 by a model designed by donnyb99 on that site. I disassembled the X-axis belt and where the bearing is connected, and then added this printed part. I can now easily tighten the X-axis belt via a simple turn on the knob.

Be careful when adding one of these, since you will now be able to over tighten, which I had mentioned is difficult to do without this. Just turn the knob until the belt is very springy to the touch. There is no real scientific way to do this, you just want to make sure there is zero droop whatsoever.

Check for wobbly or bent rods

Using thin threaded M5 or M6 Z-rods instead of thicker M8 or M10 leadscrew will lead to a wobbly or bent rod over time. It is a major upgrade when you have a thick M10 leadscrew that is connected on the top and bottom of the frame of your printer, since you will experience far less z-wobble on a machine that will last over a longer period of time.

If you have these thinner threaded rods, just move your printer up and down in the Z-axis. You will actually be able to see this problem easily since the rods will wobble back and forth. This is exactly what happened on my DIY machine and I fixed it by upgrading to M8 leadscrews (just make sure you print all the parts you need before disassembling and upgrading). To get a full tutorial on this, please see my YouTube video titled "Upgrading your 3D Printer to 8mm Leadscrews" by The 3D Print General.

There are some models on Thingiverse and elsewhere that help with this wobble in your threaded Z-axis rods, such as the "anti wobble coupling" by toolson, and they actually work quite well. While this is true, they won't be able to fix an actual bent

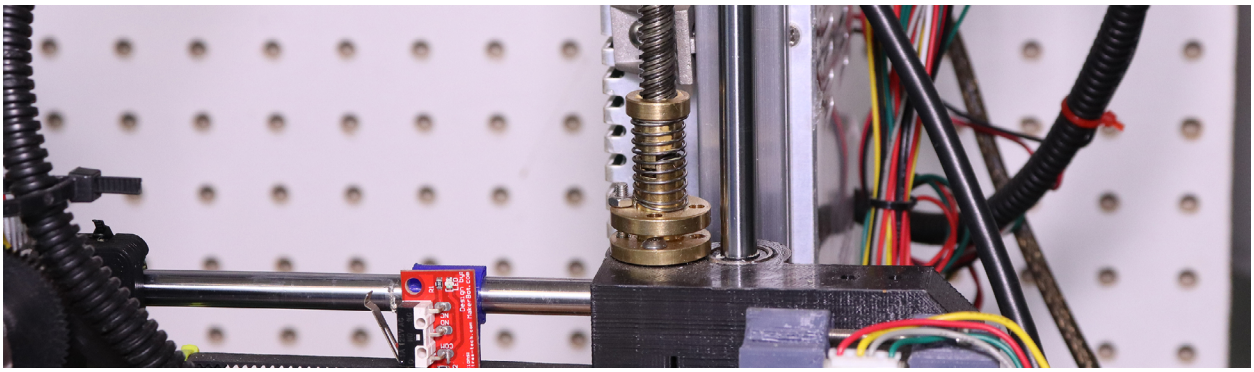
rod and nothing will be quite as good as actually upgrading to thicker leadscrews.

This is not common at all unless you frequently transport your machine. An actual bend in your threaded Z-axis rod will require you to replace it. I have only had to do this on one machine, one time in my history of printing, and it was on a thin threaded Z-rod.

While I have only had one fully bent rod, I have had plenty of ones that will wobble when moving up and down, and this is anything but ideal.

Add an anti-backlash spring loaded nut

These are normally only meant for thicker leadscrews, and they help quite a lot to prevent any backlash when moving up and down in the z-axis.



This is slightly confusing as to explain, but with these springs and added nuts, you can expect a lot less rattling and prints looking as if they have Z-wobble. Please take note of the pitch of your leadscrew since you will need to make sure your anti-backlash parts are the same. The majority of 8mm leadscrews have a pitch of 2, but you will just need to confirm with your printer specs (or with the part you buy online when upgrading).

Also make sure you are able to actually use these on the printer you are adding them too. I bought a set that were too thick to add to the CR-10, and their holes did not line up, having me return the set. You also want to make sure you are able to get full use out of the Z-height on your machine. As you can tell from the photo of my anti-backlash nuts above, I am losing about 20mm of height on my machine since the Z-axis motors are on the top. I have added extenders to make up for this, but keep in mind where you add these and if you are reducing your max Z-height for prints.

Lubricate guide rods

Your X and Y axis guide rods should be smooth enough so that both carriages can move around freely. Your Z axis guide rods (the ones that are not threaded) should also be smooth enough for the carriage to move in the Z-direction without any skipping or any bearings getting stuck.

Most printers use self-lubricating bearings, but even these will require lubrication after frequent printing. If your bearings are getting stuck or having trouble moving during the print, you can experience some Z-axis wobble.



Get some white lithium grease apply with a rag to these non-threaded guide rods. Move the carriage around on all axis's so that it spreads across your bearings. This should help with the issue.

Along with the smooth rods, you should also add a bit of lubrication to the threaded rods/leadscrews. You want to make sure these aren't entirely dry to the touch.

Make sure the bed can move back and forth smoothly

As mentioned in the "Unlevelled Build Plate" and "Layer Shifts" chapters, if you over tighten one or multiple corners, you will have a lot of difficulty moving the bed back and forth (of course only on Cartesian machines). This difficulty to move may lead to stepper motor skips which cause layer shifts, but can also cause some Z-wobble. Follow the steps in those two chapters to help remove this problem.

Is your part too tall and skinny?

This problem will not occur quite as often on a CoreXY machine, but on Cartesian 3D printers where the build plate is being moved back and forth, a tall skinny print may end up wobbling. You can use as much brim as you like with the perfect bed adhesion, if you are printing a tall and skinny part, it will likely wobble as the bed is moved back and forth.

This wobble will make the top of your print have this Z-wobble, while the bottom of

your print looks just fine.

To be honest – there is no perfect way around this. You can manually design some anchors onto your print so that as it gets taller, it is held further in place, but just printing a very tall skinny part as is will likely result in this Z-wobble.

If I am unable to design in some further anchors, I am often forced to cut extremely thin display parts in two, to be glued together post printing.

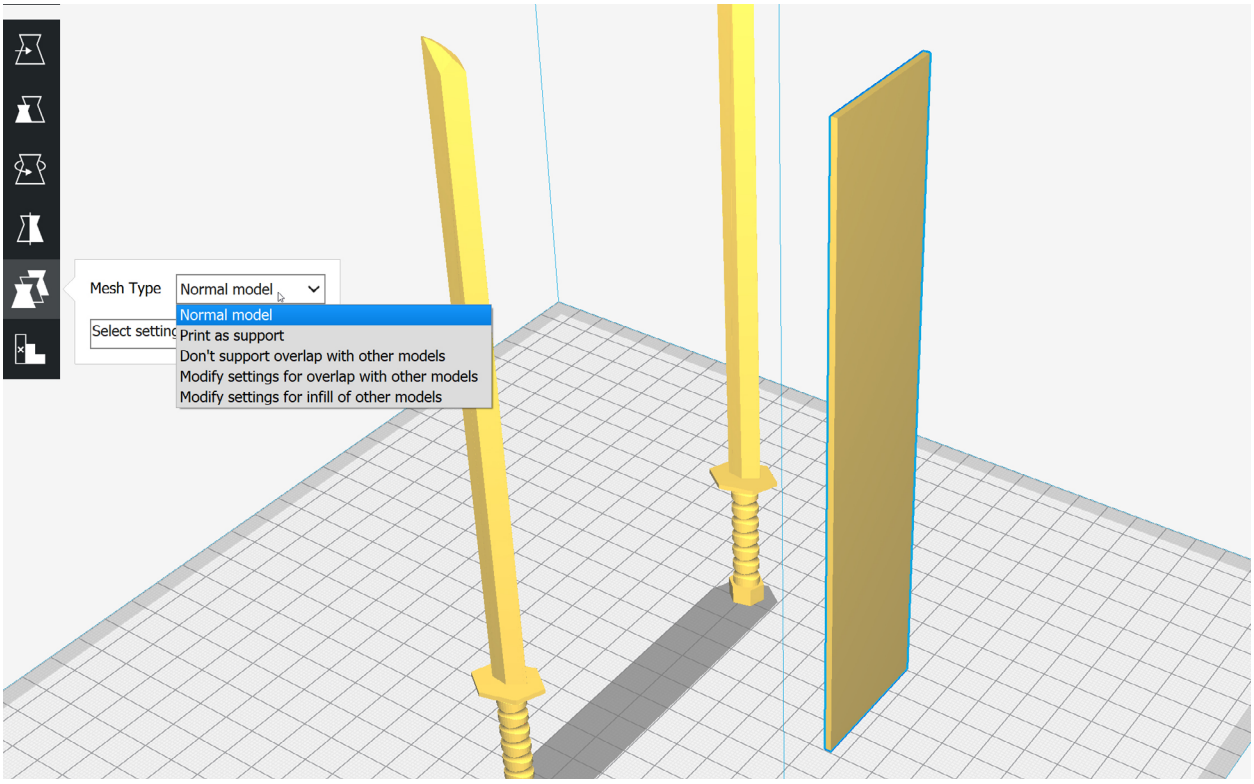
Below is a summary of how to add a very simple anchor to your part via Cura (as explained in the "Parts Being Knocked Over" chapter)

Anchoring Parts:

I am positive there are better ways to do this, but the easiest way would be to anchor your print in Cura. I have rarely done this, but it definitely helps with a tall, skinny print wobbling back and forth. This is also explained in the "Parts Being Knocked Over" chapter.

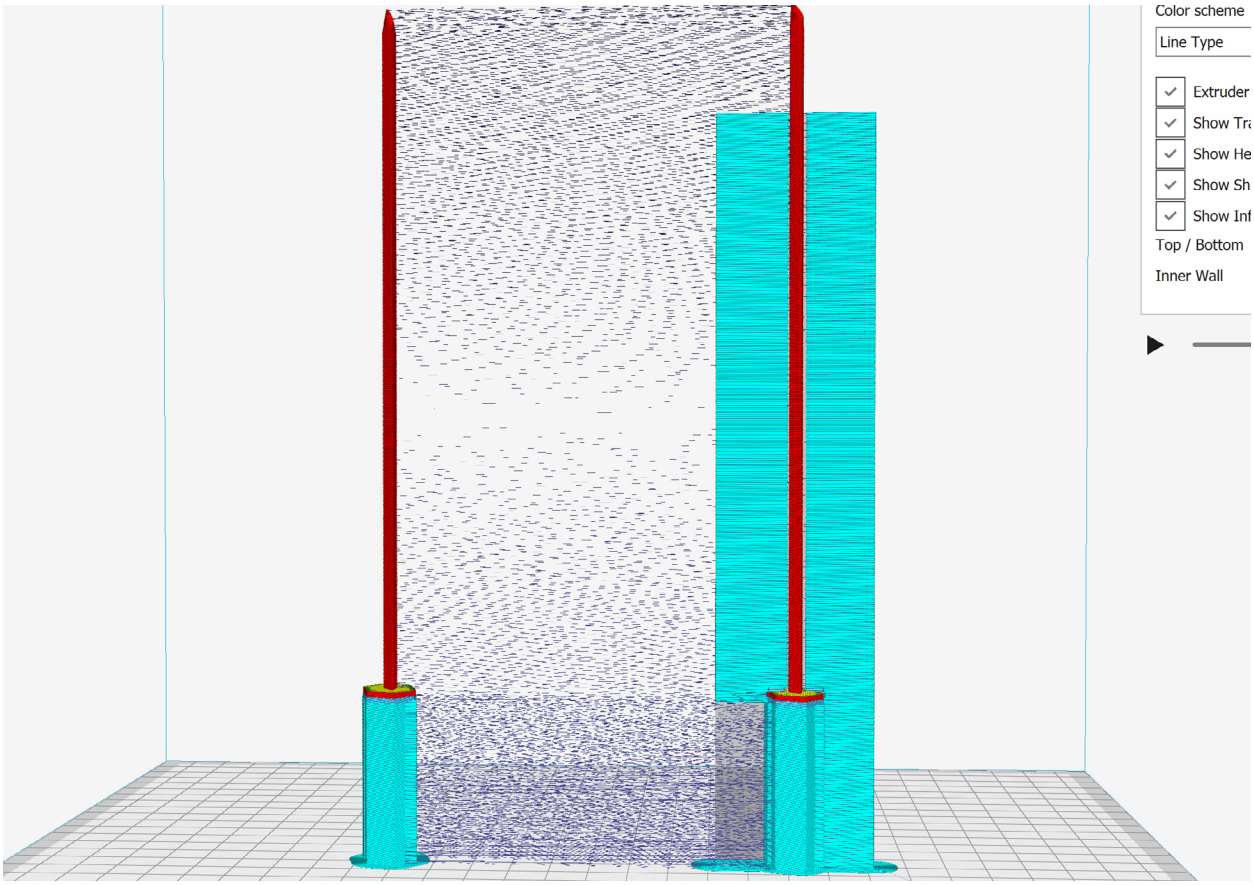
Below is an example of two skinny swords from a Deadpool print that I made for my YouTube channel. When not adding any anchors, my Cartesian machine would wobble the build plate back and forth and cause the top half of these swords to look extremely ugly (if they didn't get knocked off).

Cura now allows you to bring in a second model that intersects with your main print. They also allow you to print a part entirely as support. This means you can drag in a second object that acts only as support for your main structure.



This rectangle in the example above is thin, so that it won't take up too much material, yet will extend the anchoring for the sword (I added a second sword to compare how it will slice). After bringing in a shape that will work for your model, you can choose the model and click (Print as support)

After turning the shape into "Print as support", you can then drag it over your tall, skinny print.



As you can see in "Layer Mode", this entire shape is now support structure that can help to anchor your tall skinny print to help prevent this wobbling back and forth.

As mentioned – there are many other ways to do this, this is just the simplest way since it allows you to do this right in your slicing software.

Summary of Fixes and Precautions

- Tighten all of the bolts and screws that connect your extruder and hotend to your printer.
- Check to see if you need to replace printed parts on your extruder carriage.
- Reduce all rattling in the X-axis carriage.

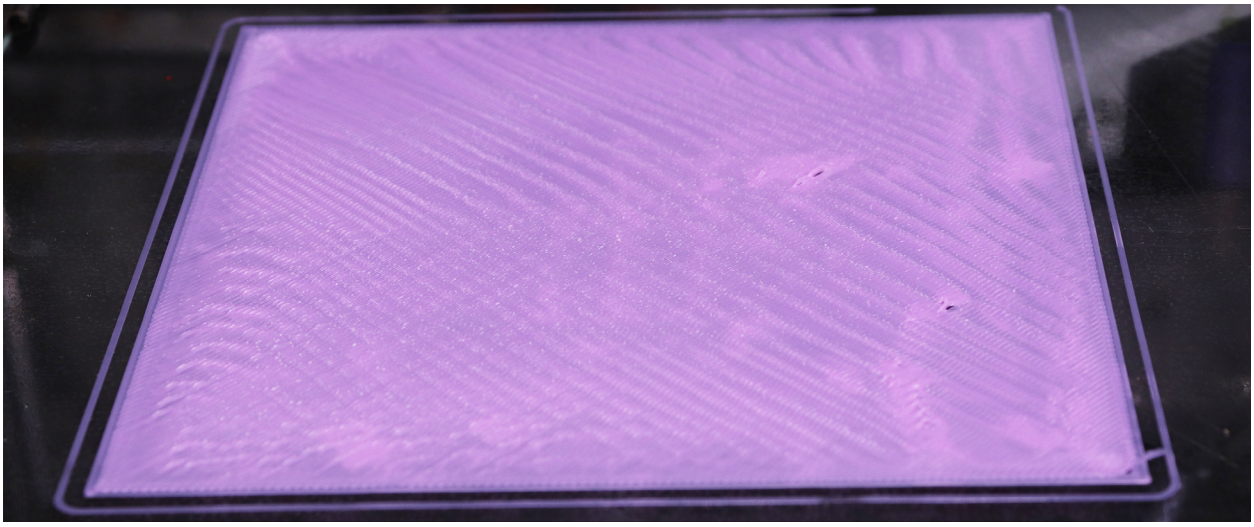
- Replace old or worn out bearings.
- Tighten your hotend setup while heated to 240°C, making sure to not over tighten.
- Remove all wobble in your print bed including replacing any bearings and tightening all harnesses.
- Tighten both the X and Y-axis belts. There should be zero droop.
- If using a thin threaded Z-axis rod, print a part that can help guide and reduce wobble.
- If possible, upgrade to a thick M8 or M10 threaded Z- axis rod.
- Replace any physically bent Z-axis rod.
- Lubricate all axis's guide and threaded rods if dry .
- Confirm the build plate can move smoothly.
- If your part is just too tall and skinny to not wobble – you may need to cut in half or manually design in anchors.

Z- Height Calibration

The Z-height calibration refers to honing in how far the nozzle is from the platform. This is determined on most machines by an adjustable screw that will run into an endstop. This is by far the most common mistake when starting a new print, but also the quickest to spot.

Along with many other printing failures, a combination of mechanical issues and slicer software tweaks can be the culprit.

Too Close to the Build Plate



Having a nozzle start too close to the build plate can be very problematic for your machine. Not only can it prevent material from coming out – often causing a nozzle clog – it can actually damage your printer.

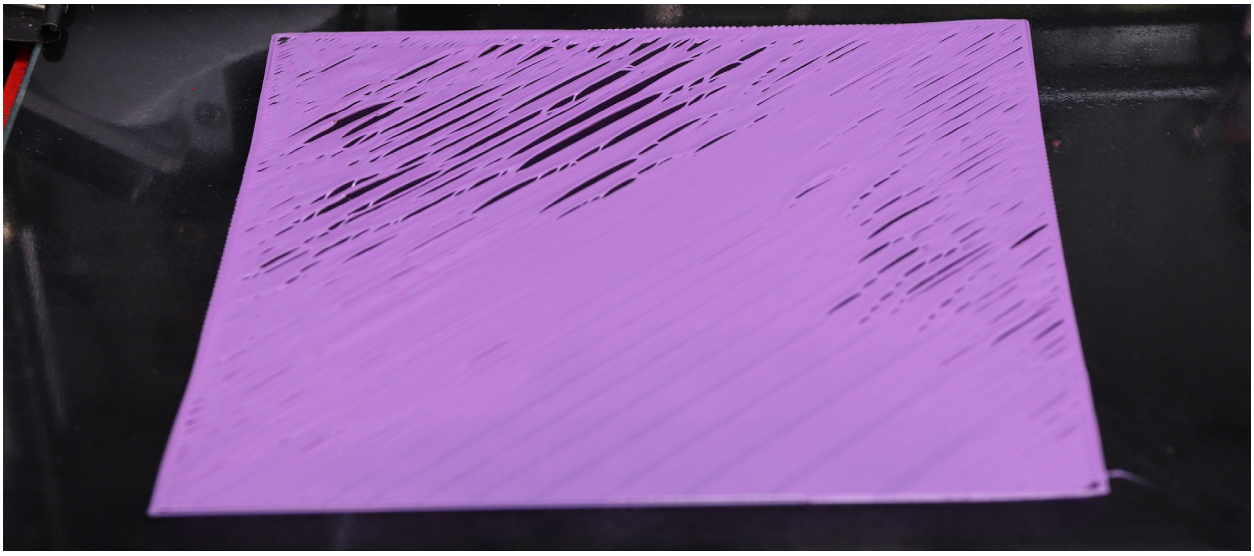
When a printer starts to close to the bed it will grind away. This can cause your printer bed to be damaged by serious scratches and can actually degrade your nozzle. This is just another reason it is essential to watch your first layer printing before leaving your machine unattended.

Over dozens of prints where some started too close to the bed, you can easily see this degradation in the picture.

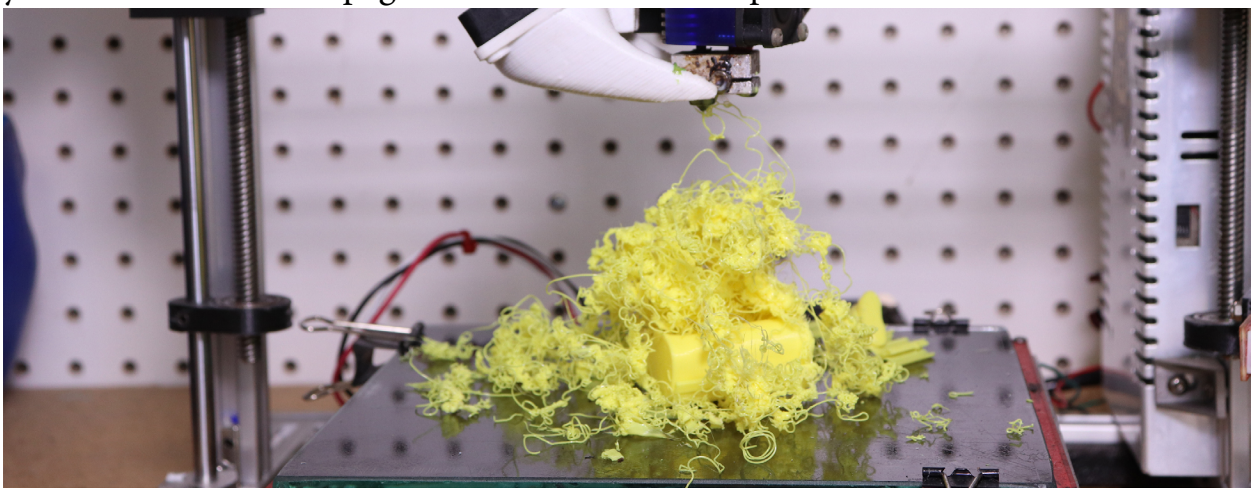


This will cause you to have to purchase a new nozzle and possibly a new build plate. If you are using a PEI surface, this can be an expensive fix.

Too Far from the Build Plate



If you were to let your printer go when the nozzle is too far from the build plate, the headaches involved can vary depending on the distance. If only slightly too high, your part may be more susceptible to warping and will cause gaps on the bottom layer of your print. A little further and your print may get knocked off, and very far you will be left with a spaghetti monster to clean up.



Generally, having parts start too far from the build plate will not cause as much damage to your machine, but you will be left with wasted material and a cleanup on your hands. There have been rare cases in which this mess of material engulfs the hotend as it continues to print and once cooled leads to a hotend being submerged in a solid block of plastic. The cleanup from this can be so extreme that it may warrant or require replacing hot end components. If this happens to you, check the "Bed Adhesion" chapter for cleanup.

Mechanically adjusting Z-height (printer too far/ close from bed)

Most machines come with an adjustable screw or lever that will run into your Z-endstop when finding home. If your printer does not have an adjustable Z-height, I highly suggest printing one.

Files for adjustable Z-height calibrations can be found on websites such as Thingiverse, you will just have to find the correct one for your printer.

Being able to easily adjust the Z-height is key, because over multiple prints these mechanical machines are very susceptible to slight variations. You will want this screw to have a spring keeping pressure on it in order to prevent as many minor turns as possible throughout the rattling of the printer.

You will want your nozzle to be in a starting position where you can slide a piece of paper under it, but so that it has some resistance. This distance will be tweaked depending on the initial layer height, as mentioned later in this section.

If this initial height is good, but the rest of your bed is not, you will want to read from the "Unleveled Build Plate" chapter of this book.

You will want to make sure to auto home, then disable steppers, and check this before starting a print because there are a lot of problems that can occur from letting your printer continue at the wrong Z- height.

Initial layer height

The height of your first layer will drastically determine minor variations in your Z-height.

This may seem intuitive, but it is very noticeable when printing different degrees of quality parts. Printing in draft quality, or having your initial layer height be close to 0.4mm, will allow you to start prints much more simply than if you were to only be printing in very fine quality.

This mean that changing your nozzle diameter can change your Z-height calibration. Using a very fine 0.25mm nozzle will cause your printer to have to be much closer on the initial layer height than a printer with a 0.6mm diameter nozzle.

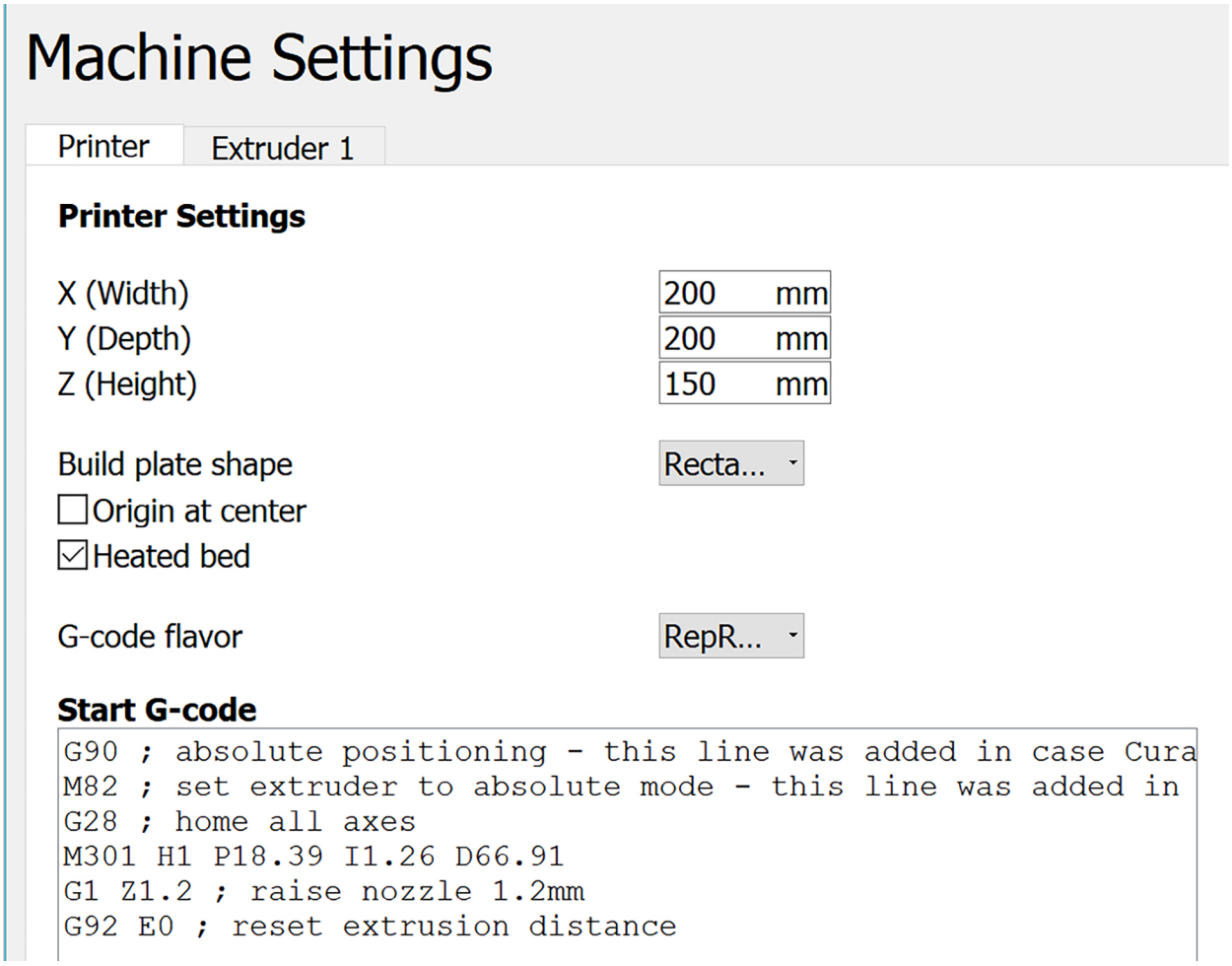
Variations based on material/temperature

As with all materials, glass and metal expands when heated. This means that you should always be checking your Z-offset with the nozzle and bed heated to your desired temperatures. This also unfortunately means that you may have to tweak your Z-height based on what material you are printing with.

Changing the Z-height in the "Start G-Code" by adding a Z-offset

Along with mechanically changing the Z-height, you can also change it by adding a positive or negative Z-offset to your start G-Code. This is ideal for when you have specific settings based on what layer height, nozzle you are using, and material being extruded since you have a saved profile for each. Having a Z-offset for each material profile you have is beneficial because a heated build plate to 110°C will be closer to your nozzle than one that is heated to 50°C, due to the expansion of your bed.

Right under the section in your start G-Code of your Z axis finding home (G28 Z0), you will want to add your Z-offset with G1 Z<position> as shown below:



In the above example, the nozzle will raise by 1.2mm after finding home and before starting the print. So, if you found the perfect Z-height when using PLA, but you have noticed you have to raise your nozzle roughly 1.2mm every time you switch to ABS, you could instead add this to your ABS profile when slicing parts. This should reduce the amount of times you have to mechanically adjust your Z-height, but will require you to tweak all of your profiles based on initial layer height, material being used, and nozzle diameter.

Here is a photo of what your print should look like when at the proper Z-Height:



Summary of Fixes and Precautions

- Mechanically adjust where your extruder runs into the Z-Endstop.
- Print additional parts if your printer does not have an adjustable Z-height option.
- Make sure to auto home before starting a print to make sure your nozzle is not too close or too far when starting a print.
- Increase the initial layer height in order to have an easier time honing in on the correct Z-Height.
- Small layer heights and nozzle diameters can lead to a lot of headache honing in proper Z-height.
- Recognize that different temperatures can lead to different Z-heights due to the expansion of your build plate.
- Create unique profiles with Z-offsets in the start G-Code to reduce the amount of mechanical adjust required.

Tips if Still Not Working

While I tried my hardest for this book to be all inclusive for every printing error, there may be a unique situation not covered that you can experience. Below are some good solutions if you can't fix your problem with any of the remedies described earlier.

Turn off machine and power supply for 10 seconds and turn back on

This used to be a running joke at our facility because it was surprising how many times it fixed a problem. I would be frustrated for a half hour and someone would yell out "Did you turn it off and back on?" We would then laugh ironically as it actually worked.

For most problems it doesn't hurt to turn off your machine and power supply for 10 seconds and turn them back on.

Check frame for sturdiness and loose bolts

If you have been printing for a while without confirming all bolts on your machine are tight, or you have an acrylic frame, you can experience minor mechanical shifts resulting in ugly or failed prints.

Periodically examine your frame for any bends or loose bolts and fix or tighten as needed. Review the "Mandatory Maintenance" chapter in this book for proper precautions.

Flash firmware

This is another thing that is confusing but would work more often than I would think. Sometimes re-flashing the firmware onto a machine that was giving very strange failures would fix the problem. This isn't possible with a machine that hasn't been bootloaded (CR-10 from the factory as an example), but the vast majority of machines should be bootloaded with Marlin to allow you to re-flash your machine. To learn how to bootload, visit TH3D's YouTube channel and website.

I go over how to flash your firmware in the "Over and Under Extrusion" chapter, and elsewhere in this book.

Re-slice G-Code

There is a chance your G-Code itself can be corrupted and you can fix your error just by re-slicing and exporting a new G-Code.

Switch filament manufacturers

The quality of your filament matters. You could spend a full week trying out all of the solutions described in this book and still not be able to achieve a successful print if you are using subpar filament.

I don't even bother with filaments that do not have high reviews and aren't made by well-known manufacturers, since their quality control is frequently very poor. Always read reviews, or check the "Resources" chapter for some of my personal favorite manufacturers.

Search online and on YouTube

If you are having a problem, it is likely someone else somewhere has experienced it. If you can search for your specific problem on Google to find a forum or a thread somewhere where someone has successfully fixed your issue. You can also search on YouTube.

I suggest subscribing to Thomas Sanladerer's YouTube channel since he covers an immense amount of 3D printing tutorials, and continually comes out with new videos. He has personally taught me a lot of what I know from these videos.

There is also the 3D printing Facebook group. There are currently over 72,000 members, and most are extremely helpful in diagnosing people's prints. I frequently respond to individual's problems, and love to read other's responses to help grow my own knowledge base.

Send me an email

If you purchased this book and are still experiencing issues, feel free to shoot me an email anytime at Sean@3DPrintGeneral.com

Limitations Involved with 3D Printing

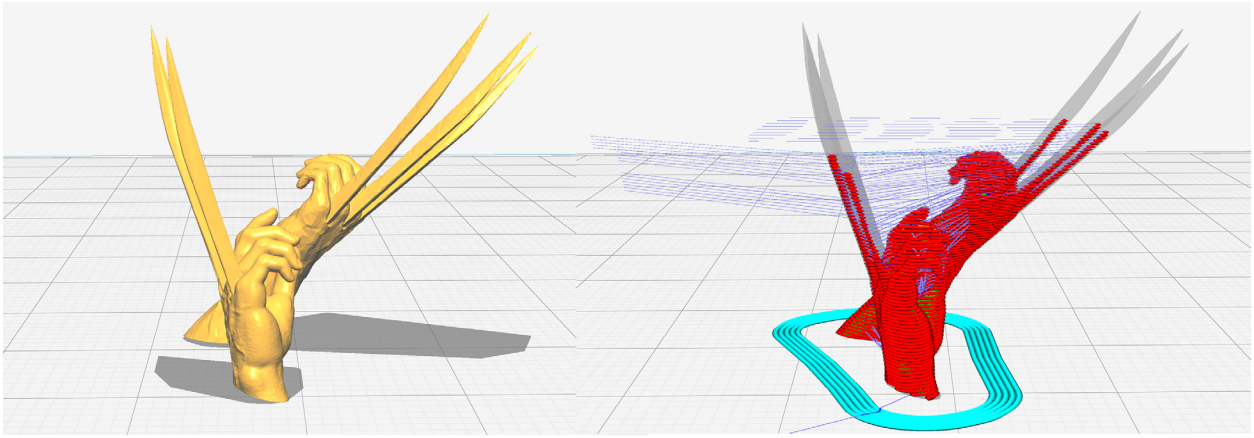
3D printing is not a perfect end-all solution for manufacturing parts, especially with today's technology. The RepRap project started as a dive into the unknown. Problems were addressed as they came and prints were considered a success if they were useable at all. These early prints would be considered unacceptable in today's terms.

I compare today's 3D printing market to that of using email in the mid 1990's. Most machines are not plug-and-play and require a bit of knowledge before you can even have one successful print.

Almost all 3D printing manufacturers, especially those on Kickstarter, will brag how their printer can be instantly used by anyone for high quality parts, but there really isn't a desktop machine like this. Every printer will have its own limitations and difficulties.

Unprintable models

I go over this in further detail in the "Model Errors" chapter, but the models you can print are limited to the capabilities of your machine, and 3D printing in general.



Walls and details that are thinner than your nozzle diameter will not be shown on your final print. If you need text to show up in which the thickness of each letter is less than 0.1mm, you may not be able to find a desktop 3D printer on the market that can achieve this detail.

There are also issues involved with the need of support material. Since gravity will always be a factor, FDM printing will have problems printing clean overhangs. Not only will cleanliness of the underside of overhangs be effected, but you will not be

able to print specific models since you cannot remove the support material.

You essentially will not be able to print any object entirely hollow unless it is a sphere or comes to a rounded top. If you want to print a giant rectangular cube, it will require at least 10% infill in order to have the top surface print cleanly. A true hollow rectangle would require support material, but there wouldn't be a way to remove it, meaning you cannot print one.

Active patents that prevent innovation

One of the things that prevented 3D printing from reaching the masses sooner were a few integral patents. As with many blossoming industries, unused or all-encompassing patents that have a long life span can prevent people from innovating out of fear of being sued. Now that 3D printing has grown into such a lucrative market, there are countless patents that currently exist.

Some of these patents prevent innovation to enclosed build environments, some to build plates, and others that encompass entire sections or methods of printing. There may be innovations to these patents that are not currently even explored.

Print speed limitations

Barring a major change in the process used in FDM printing, there will always be a limitation to just how fast a part can be printed. The better the tolerances, the longer the print - there is no way around that.

Speed is also limited by the material you are using and the size of your printer frame. Even if you had an extruder with an excessive amount of torque that grasps tightly onto the filament, there are limitations involved with melting each individual material composition. The filament needs to melt to a good viscosity before it can be pushed out of the small nozzle head, and this cannot be done instantly. Using a Volcano hotend will help, but there will always be this limitation.

The larger your printer, the greater the speeds you will actually be able to achieve as well. This is because your printer's top speed is limited to the space required to accelerate and decelerate your extruder. There is a limit to any machine as to what speed it can actually physically ever get to due to this space needed to accelerate and decelerate.

If you require speed when 3D printing, you may want to consider an alternative 3D printing technology such as MJF or DLP over FDM. While significantly more expensive, these technologies were developed to significantly speed up printing and alleviate some other process limitations found in FDM.

Upgrades and Purchasing a New Printer

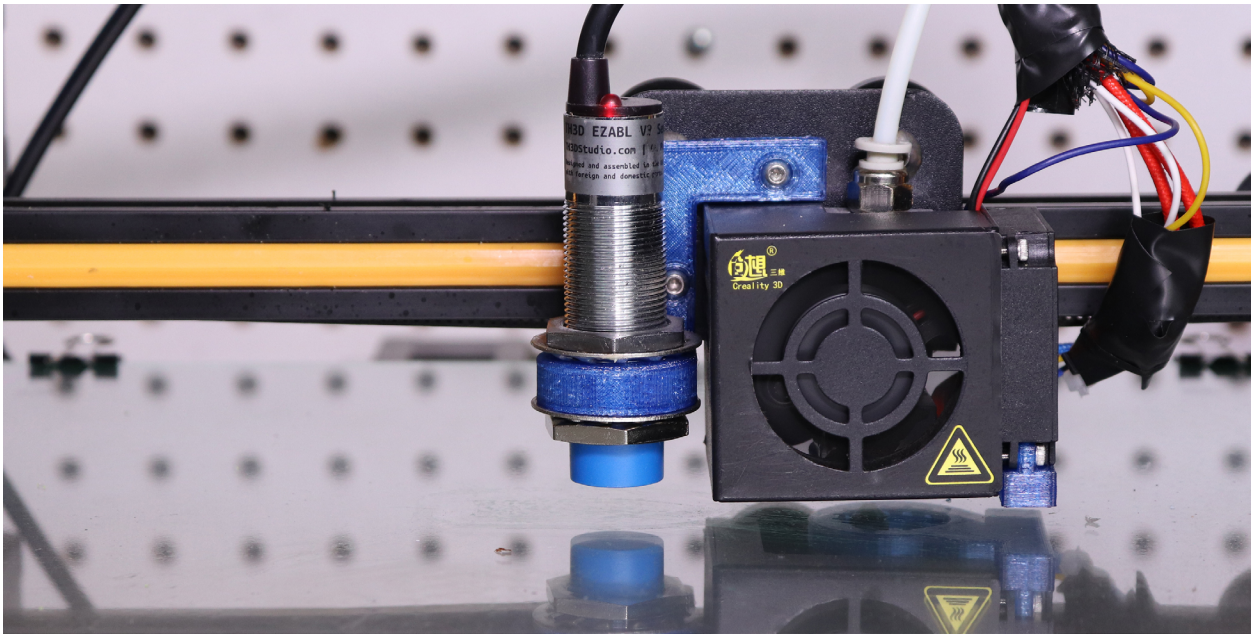
3D printing technology is rapidly improving and even purchasing this book just two years from publication can have many of these failures fixed on stock new machines.

These are parts that you may want to think about upgrading on your machine, or what you should look for when purchasing a new printer.

Auto bed leveling/auto bed tramming

Five years ago the vast majority of manufacturers that offered an auto bed leveling system did not live up to their marketing. They were too mechanical and overall did not work much better than just adding a few extra end stops.

In 2018/2019 you can now purchase 3rd party auto bed leveling lasers, or buy a machine that comes stock with one. Many of these work quite well and can save you an endless amount of headaches that I describe in the "Unlevelled Build Plate" chapter.



When using one of these sensors, your extruder will travel to different sections of your build plate at the start of a new print. The sensor will be able to tell variations in how level your bed is and will change your printer firmware to automatically tilt your model accordingly to compensate. These don't actually "level" the build plate,

but rather make minor variations in your initial z-height.

I personally use the EZABL by TH3D Studios, which can be purchased and added to your machine for roughly $60 at TH3DStudio.com. This was a great upgrade since I always have a level build plate now.

Printers such as the Lulzbot TAZ6 and Lulzbot Mini come stock with these sensors, but you can purchase one to add to your machine online.

These sensors will be mounted to your carriage, and you can find a model for your printer setup on Thingiverse. You should watch videos on TH3D's YouTube channel, since Timothy covers a lot of issues and setup guides for adding this to your machine. I also have a quick video where I go over this on my YouTube channel titled "Paid Upgrades for the CR-10".

Get thick threaded Z-axis rods

A great improvement to think of when buying your next printer is to see how thick the threaded Z-axis rod is. The thicker the rod and the better it is housed into the frame, the longer your printer will last without having tolerance issues.

Thick M10 threaded Z-axis rods on a printer usually means the printer will come with a higher price tag, but it is definitely a great upgrade. I have a video in which I go over this exact process titled "Upgrading your 3D Printer to 8mm Leadscrews" on my YouTube channel.

Upgrade hotend

Many high end printers come stock with a well-made hotend. There are quite a few options out there, but I always suggest buying from a reputable manufacturer - do not try and save money by buying cheap knockoffs.

I have personally standardized to E3D hotends (and Micro-Swiss for my CR-10). E3D has a wide array of nozzle diameter choices, are well- made, and seem to experience minimal amount heat creep. And if you do experience a nozzle clog, they are easy to disassemble. You can see all of these options over at MatterHackers.com

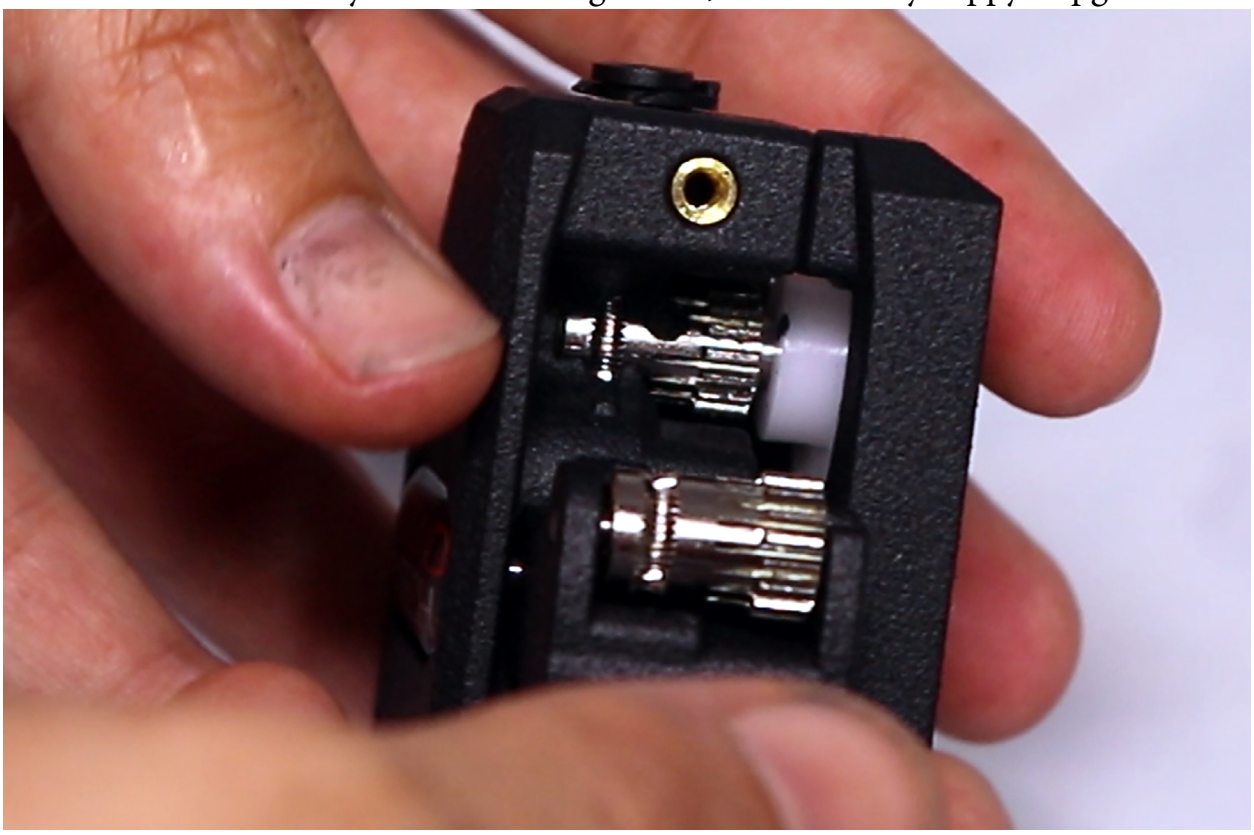
Replacing a poorly made or very old hotend can drastically reduce failures, increase the amount of materials you are able to print, and improve the quality of your parts.

Well-made geared extruder

A stock direct drive extruder without any added gear ratio cannot print nearly as fast as a well-made geared extruder. You can print these parts, but there are manufacturers that produce these extruders with some added benefits to the tolerance and quality.

For instance, E3D also makes their Titan Extruder. This is a well-built, light, geared extruder.

My personal favorite is has become the Bondtech BMG dual drive extruder. This extruder not only has a 3-1 gear ratio, it also has two sets of teeth gripping onto the filament, rather than just one pushing against a bearing as with all other extruders. If you can't print with one of these extruders, you likely can't print with any extruder. Unfortunately it costs $100 right now, but I am very happy I upgraded.



These can be used with a Bowden setup with an extra part, or can be used on direct drive machines with an E3D barrel without any extra parts needed. I bought mine over at MatterHackers.com.

Cartesian vs CoreXY vs Delta

There are benefits to each of these types of printers, but the vast majority of machines you will see on the market are Cartesian, with CoreXY being the second most common.

Delta printers seem to be able to print the fastest, but are difficult to calibrate and must be tall if you want to print wide objects. Personally, I do not have extensive experience with delta machines.

CoreXY printers have been growing in popularity and they seem to have added sturdiness. I personally like CoreXY machines the most for quality, but have a bit more experience on Cartesian models. Printers such as the Zortrax M200 and Makerbot Replicator 2 are CoreXY machines.

CoreXY machines have the benefit of not moving the build plate back and forth. This constant moving of the build plate will not only have extra wear and tear on your printer, but can result in parts getting knocked over easier with an increased chance of ghosting.

Enclosed machine

I go over this added benefit in the "Warping" chapter in this book, but it is highly beneficial to purchase a machine that has its build area at least somewhat enclosed. This way you do not have to worry as much when wanting to print in materials that have a high shrinkage rate. You just need to make sure your board is not enclosed with the printer, since you don't want that overheating.

Metal Frame

If you haven't noticed already, I highly suggest purchasing a printer with a strong metal frame. Acrylic framed printers will not hold up over time and, in my opinion, it is entirely worth upgrading to metal.

I will not bother with acrylic machines or any printer that is not sturdy.

24V instead of 12V

Most inexpensive 3D printers come with a 12V power supply and output. Buying a printer that is 24V will come with a few added benefits. These include a shorter time required to heat your build plate, more torque to your stepper motors (reduces extruder stepper skipping and allows for faster prints), and results in less noise produced by your stepper motors.

If you are building a printer from scratch, just make sure any upgrades you purchase are rated for 24V. If you are converting your printer to 24V, you will need to change your hot end heater cartridge, your heated build plate (or at least the wiring to it if your build plate can handle 24v), and you must remove a specific diode on a RAMPS board.

Keep in mind a 24V machine is likely going to be more expensive than their 12V counterparts.

1.75mm vs 3.00mm Filament

This debate has been going on for years and there is still no specific answer. I have extensive experience using both, and personally I like to use 1.75mm more.

3.00mm filament (more accurately it is 2.85mm filament) has a tighter tolerance in its diameter than 1.75mm filament does. This should theoretically mean parts should come out cleaner and you should experience less nozzle clogs, but I do not notice much of a difference in this vs 1.75mm printers. You can print parts a bit faster though because less torque is required on your extruder stepper since less distance is required to extrude the same amount of material. This means your

extruder can turn slower when dealing with 3.00mm filament.

The biggest issue with 3.00mm filament is with its want to curl back onto its spool. When you are near the end of a 3.00mm PLA spool, you can experience an extensive amount of breaks in the filament due to want to curl back into a circle.

This is far more annoying than you can imagine if you haven't dealt with it yourself. This factor alone has made me prefer 1.75mm filament, since it is far easier to manipulate and you don't ever have to throw away spools that still have 30 grams of material on them. I see no quality differences with 1.75mm and the headaches involved with using it are far less.

Resources

Below are some great resources you can use to help you in your 3D printing exploration.

Video Tips and Tutorials

The 3D Print General (Myself) YouTube Channel

Thomas Sanladerer's YouTube Channel

3D Maker Noob YouTube Channel

Joseph Prusa YouTube Channel

Maker's Muse YouTube Channel

Make Anything YouTube Channel

3D Print Nerd YouTube Channel

Other Resources

Facebook 3D Printing Group (very helpful group of people)

TH3DStudio.com

Free 3D Models For Download

Thingiverse.com

MyMiniFactory.com

GrabCAD.com

Free Software

Cura for slicing parts

Tinkercad.com for editing .stl files

Repetier Host for directly connecting to printer

MeshMixer for editing model

Places to Purchase Printer Parts

3DPrintGeneral.com – where I link to many of the parts I use

MatterHackers.com – my new favorite for finding reputable parts

Amazon.com

Fillastruder.com for E3D parts in the United States

Newegg.com

DigiKey.com
Mouser.com

Reputable PLA ABS and PETG Manufacturers

Hatchbox
AIO Robotics
Proto-pasta
Esun
IC3D
Fiberlogy

Reputable Unique Material Manufactuers

3DX Tech
taulman3D
MatterHackers
NinjaTek
Proto-Pasta
Fiberlogy
ColorFabb

My Favorite 3D Printers (Pre-Built)

My Favorite 3D Printers (Pre-Built)
$ - CR-10 (best price for what you get)
$$ - Prusa i3 MK3 (highest praise in the community)
$$$ - Lulzbot TAZ 6 (personal all around favorite)
$$$ - Zortrax M200 (great surface quality)
$$$$ - Ultimaker 3 (easy to use)
$$$$$ - Raise3D (large, enclosed printer)

Places to Order Prints

SD3D.com
Shapeways.com
ProtoLabs.com

About the Author



Sean Aranda began his 3D printing explorations by becoming the Operations Manager for SD3D in early 2015. Along with adding innovations to the 3D printing industry, SD3D is a production service provider for everyone from inventors to large businesses. With over a dozen FDM machines and a few hundred clients, Sean was able to amass well over 50,000 hours of successful printing experience while with the company.

Sean was also able to receive extensive experience with material variations and profile settings required for a successful print, since SD3D offers over 16 different types of FDM 3D printing materials.

While he had a very basic background before starting at SD3D, Sean's knowledge of printing failures grew rapidly as he helped all clients get a usable 3D print within the promised timeframe. Whenever a printer failed or had a malfunction, Sean was in charge of all maintenance and repairs.

Sean has also authored a much smaller graphic handbook for 3D printing called The A-Z 3D Printing Handbook, which has amassed well over 4,000 downloads. What you are reading now is the second edition of this book, the first selling thousands of copies helping the community to achieve consistent, successful prints.

Since releasing the first edition of this book, Sean has been growing a 3D printing channel titled The 3D Print General where he goes over fun prints and in-depth tutorials. In these two years of videos he has easily added another 25,000 hours of successful printing. These videos are meant to fill in the gaps from this book and to show first hand examples of how you can fix your 3D printing failures. He also offers giveaways and unique designs on his Patreon.

A lot of what you are reading in this book is from pure trial and error over the course of two years of full time manufacturing across dozens of machines, and from countless hours for his YouTube channel. But it definitely wasn't all trial and error, the 3D printing expert, co-founder of SD3D, and editor of this book, David Feeney was always around to educate.

About the Editor



One of the few people you will meet with a drastic amount of more 3D printing experience than Sean is David Feeney. David is an experienced manufacturing engineer, project manager and entrepreneur. With six years of direct experience in the automotive and industrial equipment industries as a manufacturing engineer and project manager, David understands the needs of the client extremely well.

Prior to co-founding SD3D, David worked at Aptera Motors as a project manager for over a dozen electrical and mechanical engineers. David later worked as a manufacturing engineer for the Large Compressor Set and Mechanical Drive (LCSMD) division for Solar Turbines. In this role David was solely responsible for fixing any design or manufacturing issues prior to hitting the assembly line on over $100 million worth of equipment annually. David holds a bachelor's of science degree in mechanical engineering from San Diego State University and is EIT accredited with the California Board for Professional Engineers.

Mr. Feeney is the head of the technology upgrades at SD3D and is utilizing his hundreds of thousands of hours of 3D printing experience by helping to advance the entire industry. Visit SD3D.com now to find out more.

Whether you are new to 3D printing, or you have dozens of successful prints under your belt, this book is going to help you!

Sean Aranda and David Feeney have hundreds of thousands of successful 3d printing hours combined, so let them help you achieve consistent clean prints.