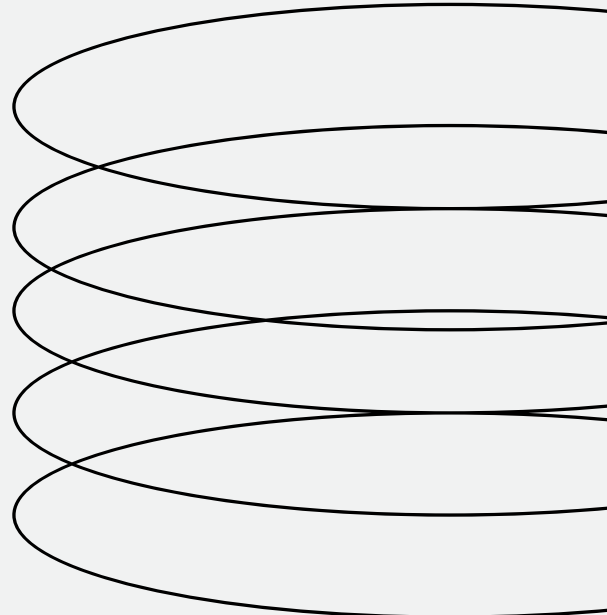




Troubleshooting Your Ender 3D Printer



RUNNING INTO PRINTER ISSUES?
This guide contains tips on how to solve the most common 3D printing problems





Getting Started

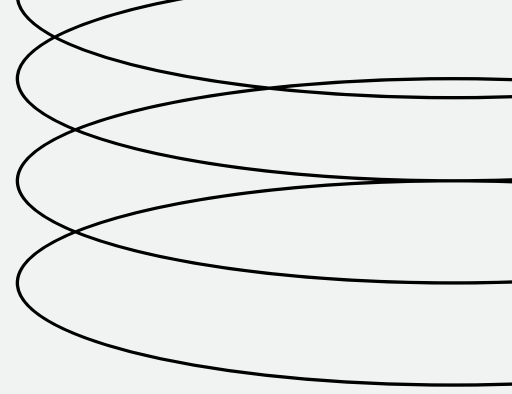
HOW TO USE THIS GUIDE

Welcome! This document describes how to solve many common printer problems with the Ender family of 3D printers. The guide focuses on several different issues that may impact your prints, including bed leveling, stringing, print curling, and general print quality issues. Printing hobbyists say that these are the most significant and common issues that they deal with. Note that although I focus on Ender printers in this guide, the solutions and techniques are applicable to almost any filament-based 3D printer.

Throughout the document, I will assume you understand the basics of 3D printing and are reading the guide to create higher quality and more reliable prints. But be aware that many of the issues that I cover can have complex solutions that may not work every time. It's important to not get discouraged. The 3D printing hobby involves frequent tinkering and monitoring to ensure your prints look their best. Even minor environmental factors such as temperature or air pressure can cause your prints to fail, so there is no way to completely guarantee a successful print. Still, by following this guide, you should be able to combat the most common print issues and generally create more detailed and reliable prints.

Table of Contents

A BRIEF RUNDOWN ON WHERE TO START



1. BED LEVELING

This section will cover the best methods to level your print bed. When the bed is leveled correctly, the print nozzle will be a consistent distance from the build plate across its entire surface. This is the most common issue that many people encounter. Leveling your bed will help your prints stick to the build plate and improve the quality of the first print layer.

2. STRINGING

Stringing occurs when your printer's nozzle oozes small amounts of excess filament as it moves over an open area. This excess filament travels with the nozzles and leaves tiny threads of plastic that connect to different sections of your print. Stringing can be removed post-print, but it can leave unseemly burrs behind and wastes your time. This section will discuss ways to reduce or eliminate stringing on your prints.

3. PRINT WARPING OR CURLING

In this section, I will discuss ways to minimize print warping. When a print warps, parts of the print bend or curl upward, causing permanent deformation. This can make your print unusable or cause the print to separate from the build plate.

4. PRINT QUALITY ISSUES

Improving print quality is a broad topic, and solving any of these printer issues will have an effect on your print quality. Still, this section will discuss several simple ways you can improve print quality without needing to completely overhaul your printer setup.

5. PRINT CALIBRATION

Whenever you make any major change to your printer's settings, it's important to calibrate your printer to ensure that the printer is tuned correctly and that it will print effectively. Print calibration typically entails printing a small benchmark print to judge the performance of your printer. This confirms that the printer is functioning correctly and able to tackle any project that you may start.

Bed Leveling

Bed leveling is a vital step to using your 3D printer. Many people think that you only need to level your bed when you first set up the printer, but this isn't true. It's very important to regularly recalibrate your bed leveling. Many people recommend leveling after every few prints, but some people level before every print! Ultimately, how often you level your bed is up to you, but it is important to consider that over time, the print bed will become unlevelled, even if it's only by a small amount. Regularly leveling ensures that your prints have strong adhesion to the build plate, leading to more successful prints. Read on for some general tips on bed leveling, followed by a full description of the best way to level your Ender print bed.

BED LEVELING TIPS

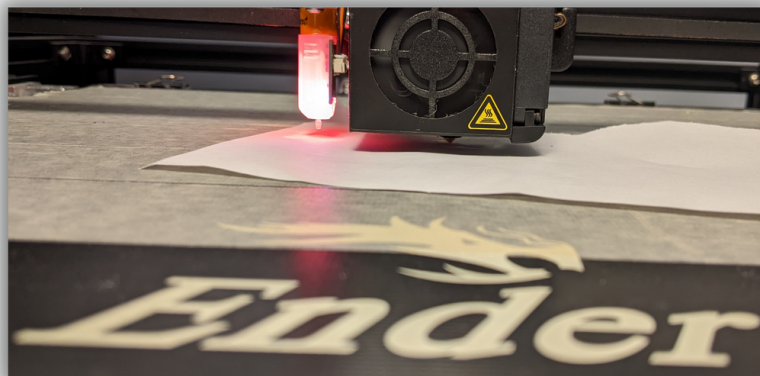
- Heat your printer up to printing temperatures before leveling. Your print bed and nozzle will expand slightly as they heat up, which will affect the distances between the nozzle and bed. **Be careful handling the printer once it is up to temperature!** Keep in mind that if you switch filament to a different material with different print temperatures, you may need to re-level your bed to account for the new temperatures.
- Gather your tools. You don't need any special equipment to level an Ender print bed, but you will need to measure the distance between your nozzle and build plate. A common rule of thumb states that the ideal distance is the thickness of a normal piece of paper. You can use paper to check this height, or use a thickness gauge at the thinnest tab instead. I will assume you are using paper for this guide, but thickness gauges are more consistent and are a good investment if you will be leveling often.
- Be careful with the adjustment knobs. Leveling involves moving your print bed on the scale of millimeters using the adjustment knobs under the bed. At this scale, even very minor adjustments to one corner can affect other parts of the bed. You don't need to turn them very far in order to have an effect. Turn each knob gently and slowly. Large turns will negate any fine adjustments you've made to other parts of the bed and you will need to start over. Even so, you will want to double check your leveling across the bed to confirm the distances are correct.

OTHER RESOURCES ABOUT BED LEVELING

- [3D Printer Bed Leveling: Easy Step-by-Step Guide](#)
- [Ender 3: How to level the bed and keep it leveled!](#)
- [How Often Should You Level a 3D Printer Bed?](#)
- [Creality Ender 3 - Easy way to level your bed video](#)
- [3D print bed leveling for the Ender 5 video](#)
- [3DBenchy calibration print](#)

Bed Leveling Step by Step

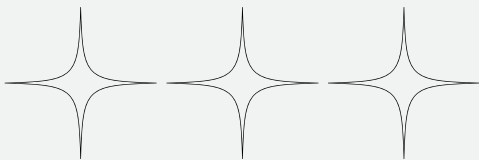
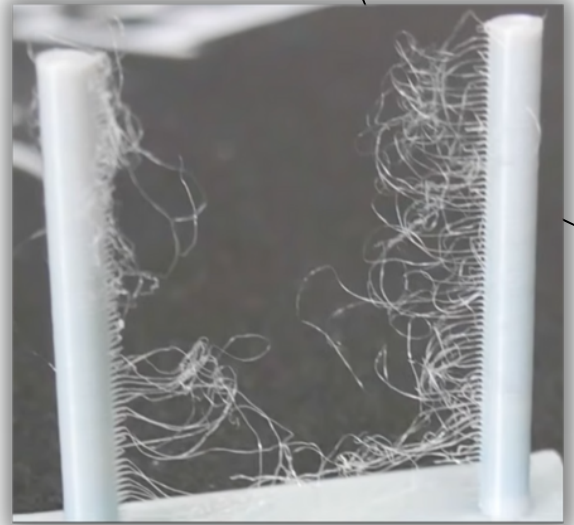
1. **Heat the print bed and nozzle to printing temperatures.** If you are printing with PLA, this will probably be about 200 degrees for the nozzle and 60 for the bed. This can be done within the settings menu of your printer. **Remember to not touch the print nozzle once it is up to temperature!**
2. **Position the nozzle at the print origin.** Your printer interface will have a "home" button in the leveling menu that places the nozzle at the origin coordinates (0,0,0). This will position the nozzle at the height where it will print the first layer. When leveling, you will want this height to be consistent all across the build plate surface.
3. **Adjust the distance between the nozzle and print bed.** Slide your paper between the nozzle and the print bed, then adjust the z height until the paper fits snugly. You should be able to adjust the center z-height in your software settings on most Ender printers, but some may need you to use an adjustment knob under the bed. Adjust the height until the paper can barely slide under the nozzle.
4. **Once the center of the bed is leveled, move the nozzle to a corner of the build plate and level the corner.** You can do this manually if you disconnect the nozzle motors, but many printers will let you move the nozzle to this position in the leveling menu. Once your nozzle is in position, check the gap using your paper. You will again adjust this distance, but instead of using the printer software, there will be a knob underneath the corner of the bed. Slowly turn this knob to manually adjust the bed height. Once you've finished one corner, repeat for each other corner.
5. **Double check your leveling.** Once you have finished leveling, move the nozzle back to the first corner and check the leveling at each corner again. Adjust as needed.
6. **Run a test print to confirm your leveling.** Once you have completed your bed leveling, it is wise to run a small calibration print to confirm that your leveling will lead to good prints. See Section 5 on Print Calibration for specifics about this task. Once you have ensured your leveling is successful, you are ready to move on to other projects!



A piece of paper snugly inserted between an Ender 5 nozzle and build plate

Stringing

Stringing is one of the most common issues that plagues filament-based 3D printers. On a basic level, stringing occurs when a printer nozzle oozes small amounts of filament as it travels over an open unsupported area of the print. This filament can attach to a print's edges and cause fiber-like protrusions that mar the surface of a print. Stringing is often easy to remove when post-processing your prints, but severe stringing can be a major hassle and may ruin fine surface details. There are several causes to stringing in filament 3D printing, so it can be challenging to isolate the precise issue. Thankfully, however, the most common causes are easily fixable. The following sections will discuss these causes and how you can reduce or eliminate stringing in your prints.



RESOURCES ABOUT STRINGING

- [3D Printer Stringing: 5 Simple Solutions](#)
- [5 Ways to Fix Stringing & Oozing in Your 3D Prints](#)
- [How to Avoid 3D Print Stringing: The Ultimate Guide](#)
- [Cura: Combing Mode - Simply Explained](#)
- [Creality Ender 3 V2 - Stop Stringing, Improve Print Quality & Time With These 2 Settings video](#)

Common Stringing Causes

- **Retraction Settings:** Poor retraction settings is the primary cause of stringing issues. When retraction is enabled in your slicer, the filament is retracted slightly whenever the nozzle moves over an empty space. This keeps filament from oozing out of the nozzle and can stop stringing entirely. Check your retraction settings in your slicing software to ensure you have retraction enabled. You may need to adjust your retraction speed and distance to hone in on what settings work best for your printer.
- **Temperature:** Stringing can also be caused by an excessively high nozzle temperature. If the temperature is too hot for your material, it will melt extra filament and cause stringing. Learn the proper nozzle temperature for the material you are using and always keep within those parameters.
- **Print Speed:** It is important to find the ideal print speed for each project you work on. Printing too slow gives filament more time to ooze out, but printing too fast can reduce your print quality. Practice with different speeds to see what results in the least stringing on your machine.

Combing: An Easy Fix for Stringing

The previous section lays out several solutions that can help minimize stringing, but sometimes these fixes don't always work. Depending on the geometry of your print, there is another process you can enable in your print slicer software that will almost completely eliminate stringing. This process is known as "combing". You should be able to search your slicer for "combing mode" and activate it before sending your design to your printer.

When you activate combing mode, the software limits the movements of your nozzle so it will only pass above printed areas. Without being able to move over empty air, there will be no way for stringing to occur on the exterior print surface. The nozzle may still ooze and string slightly, but it will be completely contained to the inside area of the print. While this sounds like a perfect solution, there are some drawbacks. Some print geometries may be too complicated for combing to work effectively. The print time can also significantly increase due to the limitations on the nozzle's movements. Still, combing can be a powerful tool to reduce stringing if you really need it.

Print Warping or Curling

Print warping is another common problem on filament 3D printers. When a print warps, part of the print curls upward. Warping is caused by the large temperature ranges between different layers of a print. Filament shrinks slightly as it cools, and this effect can add up over several layers, resulting in a print warping off the build plate. Warping is particularly common at sharp corners where two print lines meet up. Warping can be particularly nefarious since you often won't have any idea it is happening until you are well into the middle of a print. Once a print warps, there is very little you can do to fix it, as the material has already compressed and cooled into its final position. Thankfully, warping can be relatively straightforward to minimize, but it's important to note that your choice of filament material can affect this. The following sections will explain how to reduce warping and share some other resources for further reading.

HOW TO REDUCE WARPING

- **Increase bed heat:** All Ender printers are equipped with a heated bed, and you should always be using it. Different filaments need different bed heats, so look up an appropriate range for your material. If you aren't already at the top of this range, try turning up your bed heat by 5 degrees or so to keep the filament warm and reduce warping.
- **Improve bed adhesion:** Increasing the adhesion forces between your print and the bed is another great way to reduce warping. Many people place an even layer of masking tape on the build plate. The tape will increase the surface roughness, which allows filament to better adhere to the bed. Make sure the tape is as flat as possible, and relevel your bed if you do this. You can also use a glue stick or a layer of hairspray on your bed to improve adhesion.
- **Use a raft:** A raft is a bed of material that is printed onto the build plate. Your main print is then built on top of the raft. Rafts increase the surface area that touches the bed, which improves bed adhesion. Rafts can be easily added in your slicing software.



RESOURCES ABOUT WARPING

- [3D Print Warping: PLA, PETG, ABS - 3 Easy Fixes](#)
- [3D Prints Warping or Curling? - Why it Happens and How to Prevent It](#)
- [3D Printing Raft: When Should You Use It?](#)
- [Understand why 3D prints warp - simple explanation with experiment video](#)
- [3D Printer Troubleshooting Guide: Warping video](#)

Improving Print Quality

There are many ways to improve the quality of your 3D prints. Indeed, all of the tips already mentioned in this guide will have some effect on the overall results of your prints. Still, there are several specific changes you can make to fine-tune your machine and create higher-quality prints. The following sections will describe some of these changes and how they will improve your printing.

METHODS TO IMPROVE PRINT QUALITY

- **Manage your print speed:** Slowing down the movement speed of your print head will generally increase the surface quality of your prints. When the printer moves faster, tiny vibrations are sent through the machine that can lead to a rougher surface finish. Slowing down your printer will reduce these vibrations. Your print time can dramatically increase on larger projects, however. 50 mm/s is a good speed for high quality results that won't take too long to print.
- **Respect your filament:** Be aware of the ideal conditions for using and storing your filament. Most filaments absorb water from the air over time. This water then evaporates as the filament heats up in your printer, causing extrusion issues. Make sure you keep your filament in a dry environment as much as possible. Also remember that filament can be very susceptible to environmental temperatures. If the temperature gets colder, your filament can become brittle and hard to use. You may want to consider building a filament box to store your filament in to keep it dry and temperature controlled. A link to building a filament box is included below.
- **Use supports wisely:** Supports are a necessary evil when 3D printing. Many prints rely on supports when building complex or free-floating areas, but every print area that touches the supports is at risk for poor surface finishes. You can minimize this by orienting your prints such that the supports are on unimportant or hard-to-see areas as much as possible. You may also consider slicing your prints into separate components that don't need as many supports. Always be thinking about the best way to orient a print such that the supports are minimally invasive while still providing the necessary structure for a successful print.

OTHER RESOURCES ABOUT IMPROVING QUALITY

- [3D Printing Tips: How to Improve Print Quality](#)
- [The Best 3D Print Speed Settings for PLA & More](#)
- [Filament Storage for 3D Printing: Your How-To Guide](#)
- [Top 3 DIY Filament Dry Box Projects You Can Make at Home](#)
- [How to 3D Print Support Structures Properly](#)

Print Calibration

It is important to calibrate your printer whenever you level the bed or make any other major change to your settings. Print calibration ensures that your bed is leveled accurately and that any projects you work on will print with consistently high quality. To calibrate your printer, you must select a benchmark model to print each time you adjust your printer settings. The benchmark model can then be compared to previous models to identify any issues that need to be adjusted before you print a bigger or more complex project. There are many different kinds of benchmark models, and this section will cover some of the more common ones and discuss why you might use each type.

A BENCHMARK FOR EVERY NEED

- **Ensuring a smooth first layer:** Laying down a smooth first layer of a print is key for strong bed adhesion, and you can test how well your printer handles first layers with a calibration print. These calibration models will print a single layer of filament at different spots all across your bed. If any of these areas aren't leveled properly, the filament won't stick and you will know that you need to relevel that area. You can find many examples of first layer calibration prints at thingiverse.com, but make sure that you scale the file in your slicer so that it covers most of your bed area!
- **Testing the ideal temperature:** Temp towers, or temperature towers, are another useful tool, particularly if you are using a new brand of filament and you want to identify the perfect print temperature. A temp tower is a model that prints at varying heat as the print progresses, showing how the filament reacts at each temperature. Some slicers do not allow you to set temperature changes mid-print, but Cura does. You will need to set the temperature to change at specific layer heights. [This tower](#) from thingiverse is a popular option, and there are instructions for setting the temperature changes in the comments.
- **Confirming the overall performance of your printer:** If you want to confirm that your printer settings are dialed in and optimized properly, consider printing what is commonly known as a "torture test". These prints use complex geometries and intricate details to really test the optimization of your printer. The most common torture test is a small boat model called "[3DBenchy](#)", or Benchy for short. This model was designed for an ideal balance of challenging print structures and low print time and material used. Many people keep their Benchys to compare different filaments and print settings.

OTHER RESOURCES ABOUT PRINT CALIBRATION

- [How to Calibrate Your 3D Printer](#)
- [How to Use a Temp Tower in Cura](#)
- [Teaching Tech 3D Printer Calibration](#)

Conclusions

Thank you for reading this guide on how to solve common 3D printer issues on Ender printers. I hope you found it informative and helpful. The problems that I have discussed were the most common issues found from a survey I conducted among 3D printing hobbyists online. Of course, there are many other issues that you may encounter. Thankfully, there is a vast online support community for 3D printing and even Ender printing specifically. I encourage you to search online for any problems you may have. The odds are high that someone else has had the same problem and posted a solution. In particular, you may find lots of help on the Reddit website. Reddit has communities devoted to 3D printing (r/3Dprinting), solving print issues (r/FixMyPrint), and specific Ender printers (r/Ender3, r/Ender5). These resources are great places to ask for assistance, connect with the community, or even find inspiration for future prints. The 3D printing hobby is a rich and rewarding one, and the more effort you put into your prints, the more satisfied you will be. With a little bit of care and some time dedicated to optimizing your printer, you can create amazing results on almost any printer.

