

Cura settings for Ender 3 and other Creality printers

The extrusion multiplier, which is called “flow rate” in [Ultimaker’s Cura](#), specifies the rate at which your printer will extrude material. Based on the value for this setting, Cura automatically calculates how fast to move the extruder motor for certain print speeds or filament diameters.

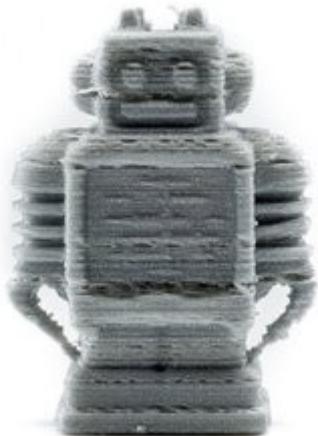
By default, the flow rate is 100%, however not all 3D printers or filaments are created equal. You may need to adjust this value to print successfully with your specific machine and materials. Most materials use a flow rate between 90% and 110%, but you can increase this value above or below these percentages.

Here’s how to locate the flow rate:

1. Once you have Cura open, open custom settings.
2. Right-click and select “Configure Setting Visibility...”
3. Select “Check All” and return to your (now expanded) set of settings.
4. Under “Material Settings”, you’ll find “Flow Rate” and “Initial Flow Rate”.

In the rest of this article, we’ll dive into how to adjust these settings to rid your prints of five different issues.

Under-Extrusion

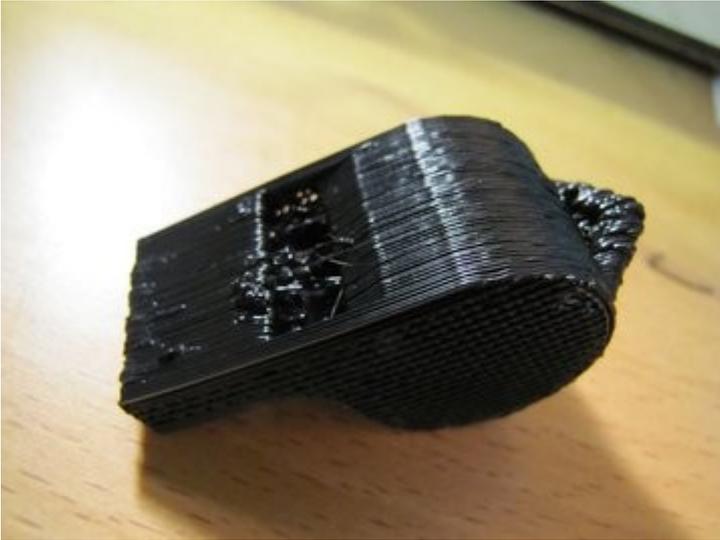


[Under-extrusion](#) can be directly linked to an inadequate flow rate. If you’re experiencing under-extrusion in your prints, try increasing the setting by 5% increments until you see some improvement.

Just know that you shouldn’t rely completely on flow rate. Sometimes under-extrusion can be caused by low print temperature, fast print speeds, or [clogged nozzles](#). If you increase flow rate to over 115% with no improvement in your prints, you should revert to other fixes.

Values that are too high can force too much material through the nozzle at a time. This results in a nasty nozzle to [clean](#) or replace.

Over-Extrusion



If you're having the opposite problem – that is, [over-extrusion](#) – you should probably lower your printer's flow rate. Do so in 5% increments, and check for print quality improvements.

Like with under-extrusion, flow rate adjustment isn't the only fix for over-extrusion. If you find that you keep lowering your printer's flow rate with no print improvement, lower instead print temperature or the filament diameter input in Cura.

Poor Bridging



Poor, droopy bridges can also be corrected by adjusting flow rate in Cura. If filament is being extruded unevenly or at an excessive rate, material flowing from the nozzle will fall into a miserable mess.

Lower your printer's flow rate in 5% increments and keep a lookout for improvements. If you start seeing some under-extrusion, bring the rate back up and use some of our [other bridging fixes](#).

Layer Delamination



Layer delamination is a pesky 3D printing issue, but proper calibration of flow rate in Cura can be an easy fix.

Try increasing your printer's flow rate in 5% increments and check for better print quality. Don't increase the rate too much, though. This can result in over-extrusion and even nozzle clogging.

If increasing the flow rate doesn't work to fix layer separation, try applying some of our [other delamination tips and tricks](#).

Poor Bed Adhesion



Prints popping off the plate is annoying; warping, equally so. To [improve print bed adhesion](#), you can try increasing flow rate in 5% increments. By increasing the amount of plastic extruded, the first layer should have a greater surface area with which to adhere to the print surface.

However, don't get rid of a print issue at the cost of getting another. Increasing flow rate too much can result in over-extrusion and nozzle clogging.

If inadequate adhesion continues in your prints, read about our [easy fixes for warping in PLA, PETG, and ABS](#).

Print Speed



Print speed is the main speed setting that will influence your 3D prints. As the name suggests, print speed determines how fast the motors of your printer move. This includes the motors controlling the X and Y axis, as well as the extruder motor(s).

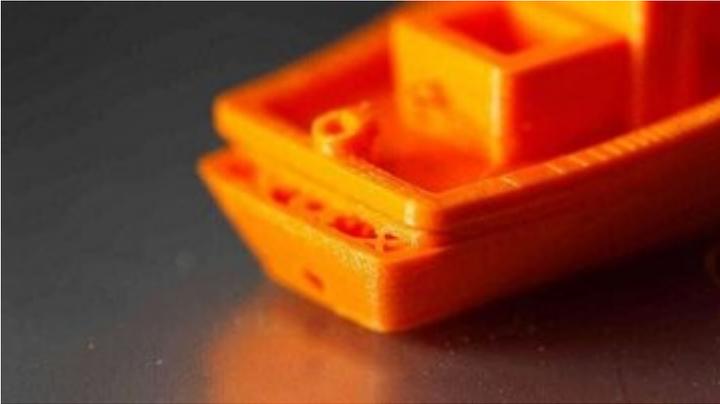
To test for print speed, download the print speed test model above. In the link, you will find instructions on how to create the settings for this print. Essentially, it will print the same shape at gradually increasing speeds, allowing you to visually identify the optimal setting.

Too slow of a print speed may cause print deformation due to the nozzle sitting on the plastic for too long. Too fast and there may be other overheating artifacts caused by insufficient cooling, as well as [ringing](#), [under extrusion](#), and [weak layer adhesion](#). The sweet spot should be as fast as your printer can get without sacrificing too much print quality. Decide how much print quality is appropriate for your use case.

To better refine print speed, this setting is commonly split into many more **secondary settings**:

- **Outer wall/shell speed:** This setting adjusts how fast the outermost perimeter of the model is printed. It is typically decreased slightly to improve surface quality.
- **Inner wall/shell speed:** This setting adjusts how fast the inner perimeter(s) of the model is/are printed. It is typically the same as the overall print speed to decrease print times while maintaining print strength.
- **Infill speed:** This setting adjusts how fast the infill of the model is printed. It's typically the same as the overall print speed to decrease print times while maintaining print strength.
- **Top/bottom speed:** This setting adjusts how fast the tops and bottoms of the model are printed. It's typically decreased slightly to improve surface quality.

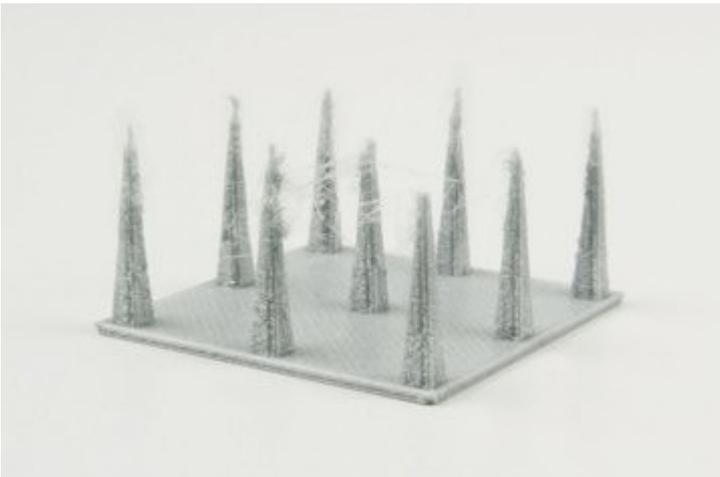
Travel Speed



Travel speed adjusts how fast the printer's print head moves when it's not extruding plastic. Increasing travel speed can save significant amounts of print time, but increasing it too much may lead to [ringing/ghosting artifacts](#) or even misaligned layers (and thus print failure).

To determine the optimal travel speed for your printer, print out the test print (above) at various travel speeds, starting at 100 mm/s and adjusting by 5 mm/s increments. Keep bumping up the speed if the surface quality is acceptable, and decrease if print quality suffers. Look out for defects like misaligned layers (above).

Retraction Speed



[Retraction speed](#) adjusts how fast the printer pulls filament back (a retraction) before travel moves. This setting is critical for reducing [stringing](#) and improving print quality. Too slow, and you may be left with unsightly strings and blobs on your prints. Too fast, and you may experience the symptoms of [filament grinding](#), which also include [stringing](#) and [blobbing](#).

To determine the optimal retraction speed for your printer, print out the retraction speed test (above) at various retraction speeds, starting at 25 mm/s and adjusting by 5 mm/s increments. Look out for web-or-tree-like strings stretching between the spikes on the

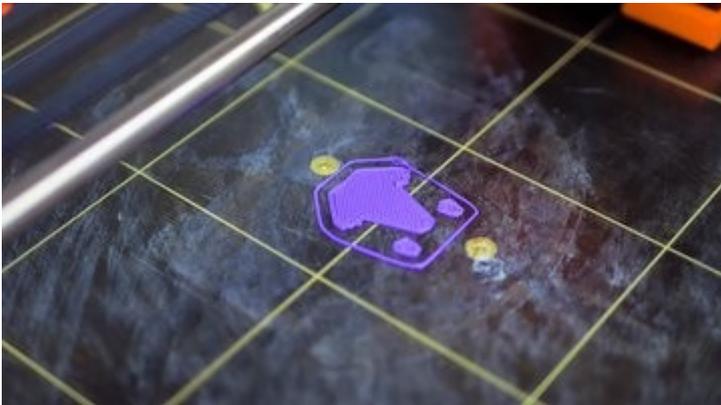
model. The perfect retraction speed should be the highest value that minimizes these artifacts without jeopardizing filament flow.

To better refine retraction speed, this setting is commonly split into two more **secondary settings**:

- **Retraction Retract Speed:** This setting adjusts how fast the retract (the actual pulling back of the filament) occurs. This is usually the same as your overall retraction speed, and tuning it follows the same process.
- **Retraction Prime Speed:** This setting adjusts how fast the filament is reinserted after the initial retract, in preparation for further printing. Increasing this speed will result in reduced print times and less blobbing due to extended exposure of the plastic to heat. Increasing it too much, however, may also result in insufficient heating of the freshly-reinserted filament before printing resumes. For most users, keeping this the same as the general retraction speed setting should work well.

For more on reducing stringing, check out our [dedicated guide to retraction speed](#).

Shell, Infill, and Layer Height

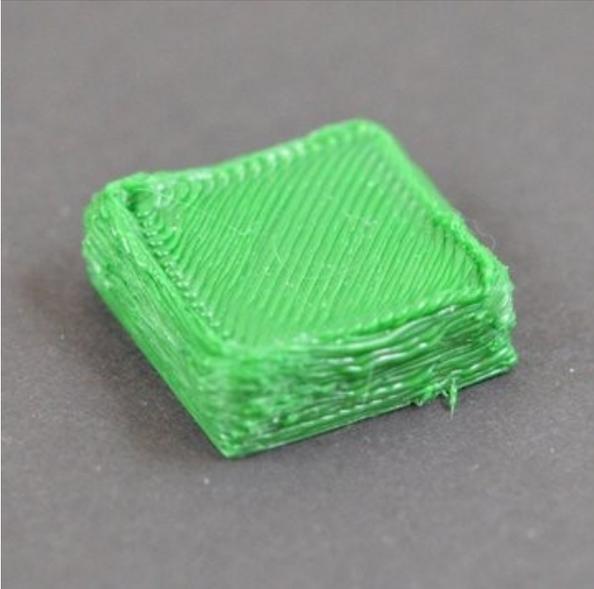


Your 3D printer's speed isn't affected only by "speed" specifically. It is also significantly affected by how much plastic, and in what thickness, each layer of the 3D print is extruded. These settings are incredibly detailed, so we will only go through general information about how each affects print speed. For more, we have specific guides to 3D printing [shells](#), [infill](#), and [layer height](#).

- **Shells:** The shells setting indicates how many perimeters of plastic are extruded to form the outline of your part. Increasing the shell count increases part strength, but also print times.

Infill: Infill is the internal structure generated to save material when printing the interior of 3D prints. The infill pattern will marginally affect print speeds, while infill density will alter print times drastically. A higher infill density will increase part strength but also print time.

Temperature



One of the most important settings is temperature. An incorrect temperature setting can cause [over or under-extrusion](#), curling, and [blobs or zits](#). Most PLA filaments require at least 180 °C for smooth extrusion and can stretch up to 220 °C before over-extrusion becomes a problem.

Ideal temperatures vary between materials (ABS: 220-250 °C, PETG: 220-245 °C), but the brand of filament can also make a difference. When trialing a new filament, it's a great idea to [print a temperature tower](#). In general, we find 200 °C a great average temperature for PLA.

Remember: only change one setting at a time when calibrating your printer.

Summary:

- **PLA:** 200 °C
- **ABS:** 230 °C
- **PETG:** 240 °C

The bed temperature also depends greatly on your material. PLA doesn't actually require heating (but still benefits from it), especially if you use something for bed adhesion, like blue painters tape, hairspray, or glue. Ideally, your bed temperature should always be higher than the ambient room temperature.

Bed temperature for ABS should be significantly higher than for PLA, around 110 °C. If you'll be [using your Ender 3 for ABS](#), then you'll also probably need to [build an enclosure](#) to eliminate excess cooling and prevent warping and layer splitting.

Summary:

- **PLA:** 50 °C
- **ABS:** 110 °C
- **PETG:** 70 °C

Speed



Finding your perfect print speed is all about balancing the total time per job with print quality. As your print speed increases, unfortunately, your print quality drops. This is because a slower-moving hot end on finer details is much less likely to mess something up. On the other hand, nobody wants to wait two weeks for a single print.

On average, most users print PLA between 45 mm/s and 65 mm/s. You can increase this speed by utilizing OctoPrint and [alternative firmware such as Klipper](#) or [Marlin](#). We'd recommend a starting speed of 60 mm/s, and decreasing the setting for prints requiring a greater level of detail.

Flexible materials must be printed significantly slower. A print speed of 20 mm/s to 40 mm/s should drastically heighten your chances of success.

PETG is slightly more stringy than PLA, and to combat this, you may want to drop your print speed by 20 mm/s or so. 30 to 55 mm/s works well. It's best to start on the lower end of the scale and work up.

ABS requires roughly the same print speed as PLA. 45 mm/s to 65 mm/s is ideal.

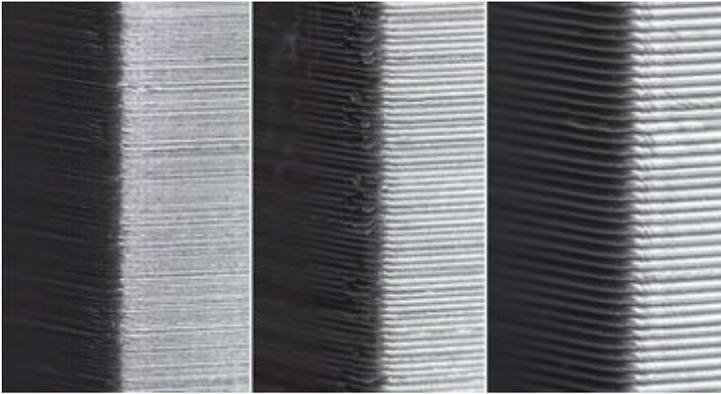
For larger prints that require less detail, you can raise the print speed on your Ender 3 up to 120 mm/s.

Travel speed can be increased to 150 mm/s without any issue.

Summary:

- **PLA:** 60 mm/s
- **Flexible materials (e.g. TPU):** 30 mm/s
- **PETG:** 40 mm/s
- **ABS:** 60 mm/s
- **Travel Speed:** 150 mm/s

Layer Height



[Layer height could be renamed detail](#), and the outcome would be the same. Halving the layer height doubles the print resolution. By halving the layer height, you're fitting twice as many layers into a single print.

The inverse to greater resolution is significantly slower print times. Finding the perfect combination of print speed vs layer height is essential to high-quality prints in lower time frames.

Layer height can be broken down into "magic numbers". Magic numbers are essentially layer heights that work best on the Ender 3:

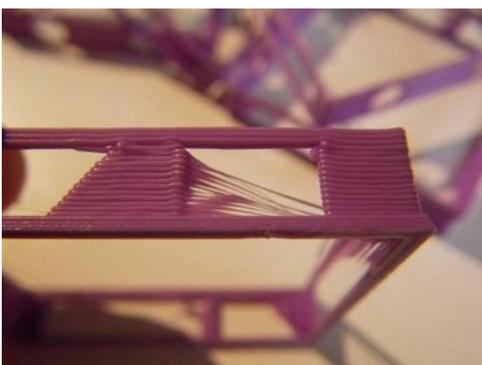
- 0.24 mm
- 0.20 mm
- 0.16 mm
- 0.12 mm
- 0.08 mm

You'll notice that the magic numbers are all multiples of 0.04 mm. Simply put, this is because [the Z-axis stepper motor on the Ender 3 will raise the hot end 0.04 mm per step](#).

Summary:

- **Fine detail, slow:** 0.12 mm
- **Coarse detail, fast:** 0.20 mm

Retraction



Retraction is your best weapon to combat oozing and stringing in prints. Retraction is broken down into two main settings:

- Retraction distance
- Retraction speed

Essentially, retraction is a tool that reverses the extruder motor to relieve pressure on the nozzle during travel. Hopefully, without pressure on the nozzle, no excess material should be extruded.

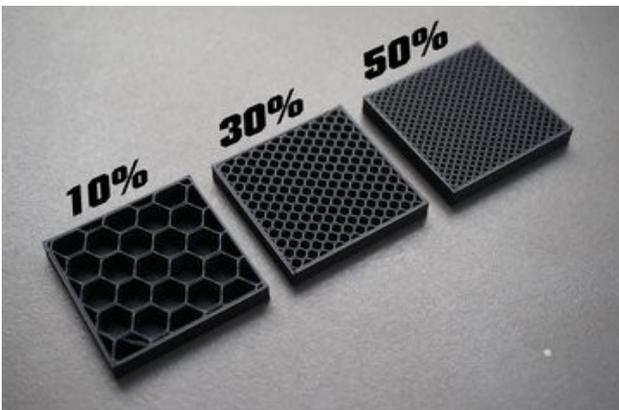
For a while, the perfect setting was hard to find, and the Ender 3 became known for terrible stringing. That's far in the past now, and online users have what appears to be the perfect retraction setting for PLA, a distance of 6 mm at a speed of 25 mm per second.

PETG prefers a shorter retraction distance, ideally around 4 mm. ABS benefits from a 6-mm retraction distance, but a faster retraction at 40 mm per second. It's a good idea to play with these settings to find the best ones for you, to do this you can find [retraction test prints on Thingiverse](#).

Summary:

- **PLA:** 6 mm @ 25 mm/s
- **PETG:** 4 mm @ 25 mm/s
- **ABS:** 6 mm @ 40 mm/s

Infill



Infill affects both the strength and weight of your print. There is no 'perfect' setting for infill, as it purely depends on the purpose of the print. For lightweight, decorative prints, you can drop your infill down to 5% or 10%. This will decrease print time and make the print significantly lighter.

For functional prints, you'll want a higher infill setting to increase strength and durability. Anywhere from 20% to 60% is ideal. Some prints may even require 100% infill.

Summary:

- **Decorative prints:** 10%
- **Standard prints:** 20%
- **Functional prints:** 40%

Initial Layer



Ensuring your first layer sticks to the print bed effectively is paramount to good print quality. Poor bed adhesion will cause a print to fail almost every time. As such, you should alter your initial layer settings to ensure the print won't delaminate during the print, which wastes significant amounts of time and filament.

- **Initial layer thickness** can be increased to 0.20 mm or 0.24 mm to give you a thicker first layer and improve bed adhesion.
- **Initial layer speed** should be slightly slower than your regular print speed. A slower speed gives the first layers more time to bond together and adhere to the print bed. Decreasing the initial layer speed shouldn't drastically affect total print time, as very few layers are being slowed down. We find 30 mm per second to be ideal for initial layer speed. Reduce this number if you're having print adhesion problems.
- **Number of slower layers** is the number of layers that you print at your initial layer speed. 5 should be plenty.
- **Initial fan speed** should be set to 0%. Excessive cooling of the earlier layers can cause your print to slightly shrink or warp and consequently pop off of the print bed.