Maximizing Your Ender 3 Print Speed: A Comprehensive Guide

Tasos Polygenis
05 Jun, 2023

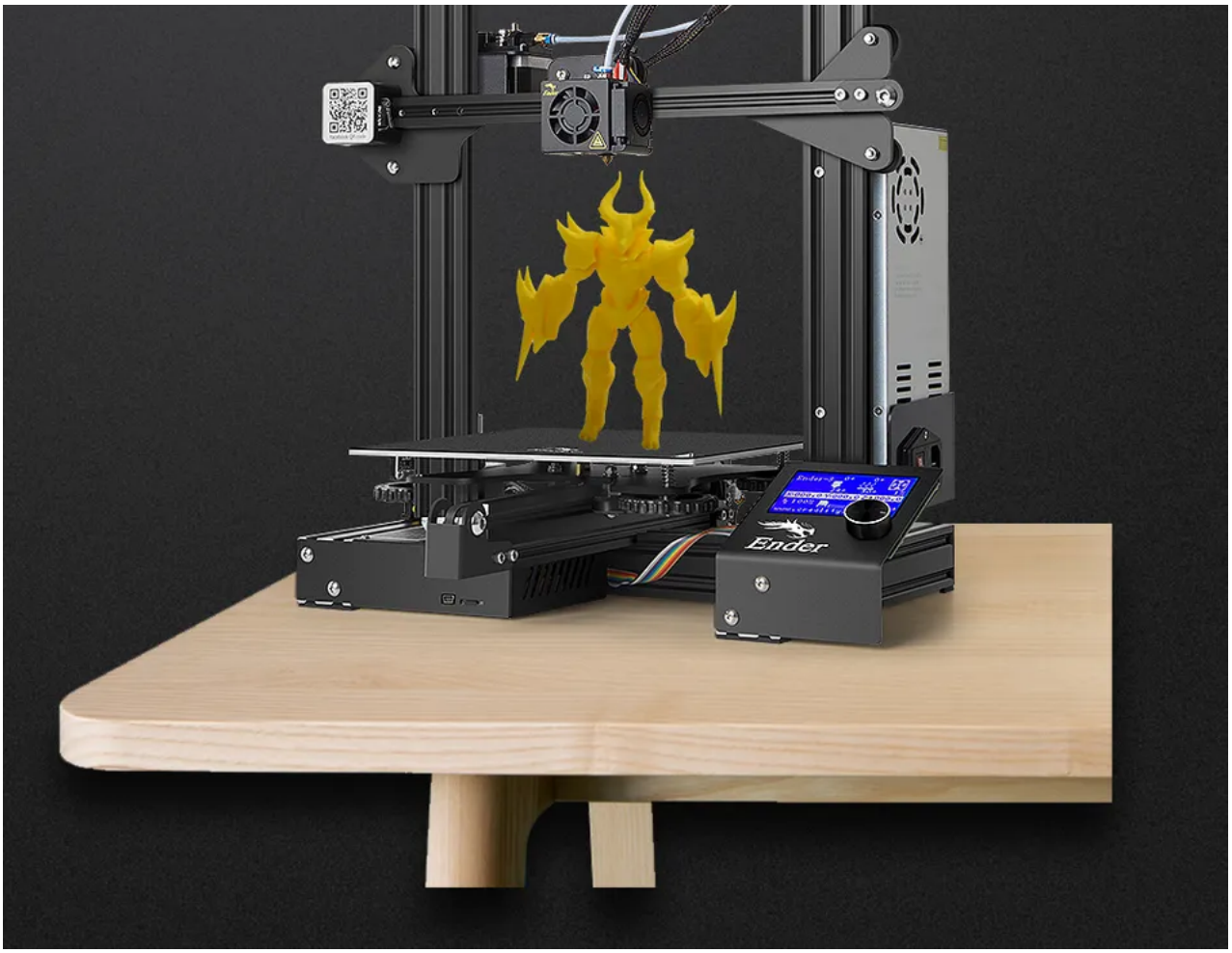
The Creality Ender 3

3D printing, also known as additive manufacturing, is a transformative technology that enables the creation of three-dimensional objects from digital designs. This technology works by laying down successive layers of material until the entire object is created. Among various types of 3D printing technologies, Fused Deposition Modeling (FDM) is one of the most commonly used, especially in consumer-grade 3D printers.

Print speed, a critical element in the 3D printing process, refers to the rate at which the printer head moves while depositing the filament. It is typically specified in millimeters per second (mm/s). Along with other factors, this ultimately determines the time it takes a 3D printer to produce a physical object from a digital design. Confusingly, some people conflate print speed with printing time — i.e. the time it takes to finish a print. While the two are related, they are not the same. Case in point: small parts can have a very short printing time even when printed at slow speeds, simply because they are small.

The Creality Ender 3, one of the most popular entry-level FDM printers, has a listed top speed of 180 mm/s. But does the stated Ender 3 print speed reflect reality, or does it make more sense to print at the recommended rate of below 60 mm/s? In this beginner-friendly guide, we look at ways to maximize your Ender 3 print speed — via material, hardware, and software choices — while maintaining acceptable print quality.

About the Ender 3



Printing a model on the Ender 3 (Credit: Creality)

The Ender 3 is a popular budget-friendly 3D printer developed by Creality. Known for its reasonable print quality, reliability, and upgradeability, as well as its exceptionally low price, the Ender 3 has — along with the very similar Ender 3 Pro and Ender 3 V2 — established a strong presence in the 3D printing community. This printer uses FDM technology and provides a print volume of 220 x 220 x 250 mm, catering to a wide range of 3D printing needs.

According to Creality, the "normal" print speed for the Ender 3 ranges between 30–60 mm/s. However, the manufacturer says the printer is

Creality Ender 3	180
Original Prusa i3 MK3S+	200+
Anycubic Kobra	180

Running the printer at its top speed isn't usually recommended due to potential quality issues. As such, the balance between speed and print quality becomes an important factor for Ender 3 users. Fine-tuning the printer's settings to get the right balance can help users optimize their print results without compromising the speed too much.

Remember, finding the best Ender 3 print speed is more than just changing a setting. It's about understanding the interplay between hardware and software, the filament being used, and the nature of the 3D model you're trying to print. Only with this understanding can you unlock the full potential of your Ender 3. That being said, many users have, with careful preparation, enjoyed (relatively) successful printing around the top speed of 180 mm/s.

Print Speed and Quality

When it comes to 3D printing, there's an intrinsic connection between print speed and quality. Understanding this connection is essential for optimizing the performance of your Ender 3 printer. It isn't just about

need to produce prototypes rapidly. It's also advantageous when printing larger models, as these can take a substantial amount of time to print at slower speeds.

On the other hand, print speed influences the mechanical properties of the printed object. When the print head moves faster, the filament has less time to bond to the previous layer, which might lead to less precise prints.[1] This is particularly noticeable in prints with intricate details or overhangs. In such cases, slower print speeds may be necessary to ensure adequate cooling and proper adhesion. However, some researchers have found that fast printing speeds can actually improve the tensile strength of printed parts.[2]

Moreover, print speed also impacts the thermal properties of the print. Faster speeds might not allow enough time for each layer to cool and harden before the next one is deposited. This could result in warping or other deformities in the printed object, especially when using materials that contract significantly upon cooling, such as ABS.

Note that 3D printer manufacturers often differentiate a machine's maximum print speed from its maximum travel speed. Why? Because the rate at which the printer head can physically move (or "travel") typically exceeds the rate at which it can move while successfully depositing material.

Recommended reading: ***3D print speed: What it is and why it matters***

Factors Affecting Ender 3 Print Speed

The Ender 3's hardware determines its maximum print speed (Credit: Creality)

When operating an Ender 3 printer — or any FDM machine, for that matter — a few different factors determine the maximum viable print speed. Understanding these factors can help you boost printing speeds without compromising the printed parts.

Part geometry

One of the primary factors affecting print speed is the type of object being printed. Objects with intricate details and complex geometries often necessitate a slower print speed to ensure precision and accuracy. In

different materials require different print speeds and print temperatures.

For example, flexible filaments like TPU will not print successfully at very high speeds, as the added pressure required can cause the filament to coil up and get stuck in the extruder. But rigid filaments like PLA and PETG can be pushed through the extruder at relatively fast speeds. The table below shows typical printing speeds for a few common materials. However, for certain parts, the max speed can be much higher.

Material	Typical print speed (mm/s)
PLA	60
ABS	60
PETG	40
TPU	20

Hardware

The mechanical condition and calibration of the printer can help ensure that the Ender 3 is reaching top speeds. Regular maintenance, such as cleaning the nozzle, lubricating the moving parts, and ensuring the belts

to adhere to the bed, while a surface with good adhesion for the chosen printing material allows the first layer to be deposited faster without risk of detachment. Using the right bed temperature will also help with adhesion.

Lastly, the printer's motion system, which includes the motors, belts, and pulleys, can impact the achievable print speed. A well-maintained and calibrated motion system ensures accurate and fast movements of the print head, allowing for higher print speeds without loss of accuracy or precision. Regular maintenance of these components is essential to maintain optimal print speed capabilities.

More advanced users can consider improving the hardware of their Ender 3 via mods and upgrades (discussed in the next section).

Software

It goes without saying that the print speed setting in the slicer software — Cura, for example — determines the print speed. However, many beginners are surprised to find out that their printer isn't always printing at the specified print speed.

Why is this? In short, some slicer settings override others because they are more specific than the overall print speed. For example, while the overall print speed may be set to a high value, other parameters such as infill print speed and acceleration may be set to a lower value, effectively nullifying the overall speed during certain stages of the build.

Optimizing Your Ender 3 for Faster Print Speeds

Printer Calibration

A crucial step to enhancing print speed is ensuring your Ender 3 is correctly calibrated. Calibration is the process of fine-tuning the printer's settings and physical components to ensure accurate, high-quality prints. When a 3D printer is well-calibrated, it performs optimally and reduces the likelihood of print errors, which are more likely to occur at high printing speeds.

Basic calibration of the Ender 3 involves several procedures, including bed leveling, extruder calibration, and PID tuning.

Bed Leveling

Bed leveling is the process of adjusting the print bed to ensure it is perfectly parallel with the movement of the extruder. An uneven print bed can lead to print defects, such as poor first layer adhesion, which may ultimately slow down the print process due to repeated print failures.

Bed leveling is typically achieved through the printer's leveling knobs and a piece of paper or a feeler gauge, though it is also possible to install additional bed leveling hardware to make this process more accurate.

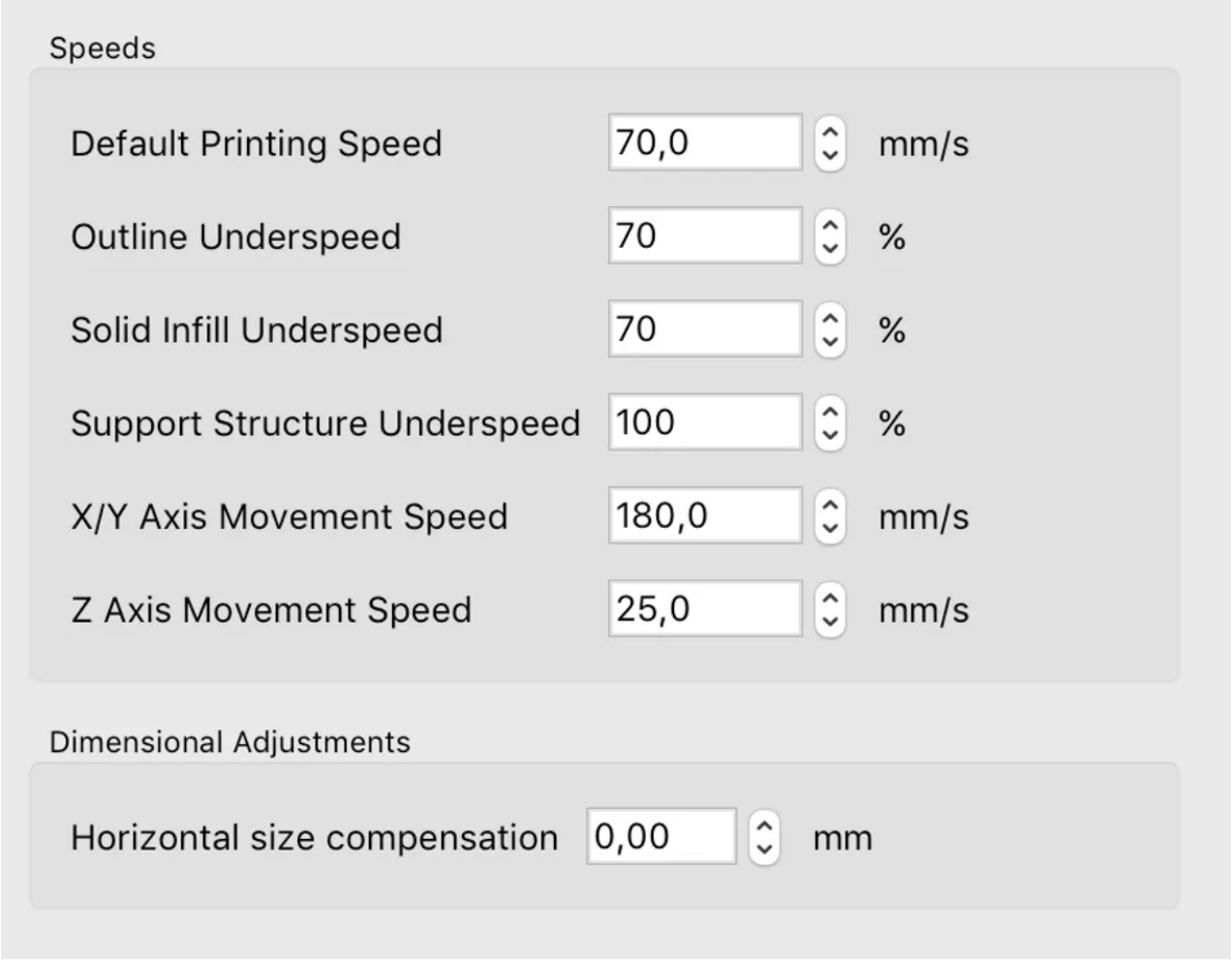
Recommended reading: ***Ender 3 Pro Bed Leveling: A Comprehensive Guide***

Extruder Calibration

Extruder calibration ensures that the correct amount of filament is extruded during the print process. Incorrect extrusion can lead to problems

PID tuning is the process of fine-tuning the printer's temperature control system, known as the PID controller. A well-tuned PID controller ensures that the printer's hot end and bed quickly reach their target temperatures and maintain them consistently throughout the print process. Poor temperature control can lead to print defects and inconsistencies, slowing down the print process.

Slicer Settings



Cura speed settings (Credit: Ultimaker)

Print Speed

Obviously, the most crucial slicer setting is print speed itself. But while it may seem logical to simply increase the print speed to achieve faster prints, the relationship isn't linear. There are limits to how fast the extruder can melt filament and how quickly the motors can move without losing precision.

Acceleration

Acceleration and jerk settings also directly affect print speed. Acceleration determines how quickly the printer reaches its maximum speed or changes speed during printing moves. Higher acceleration values reduce the time spent on these movements and increase the effective print speed. Faster acceleration enables quicker travel moves between print areas, resulting in reduced non-printing time.

Jerk

Jerk refers to an instantaneous change in velocity or direction.[3] It controls how quickly the printer changes direction at sharp corners or curves. Higher jerk values allow for faster direction changes, reducing the time required for these movements and increasing print speed. Faster jerk settings help the printer maintain higher speeds during intricate parts of the print, resulting in reduced print time.

Temperature

Adjusting the temperature settings can also influence print speed. Printing at a higher temperature allows for faster extrusion as the filament melts more quickly. However, different filaments have different optimal

the extruder, but also one that doesn't need intense cooling, which ultimately slows the process down.

Though it has its limitations, ABS is a great 3D printing material for fast printing. It doesn't easily get stuck in direct drive or Bowden extruders, and it cools down quickly, which makes parts less likely to warp or collapse mid-print.

Recommended reading: ***Comparison of PLA, ABS, and PETG Filaments for 3D Printing***

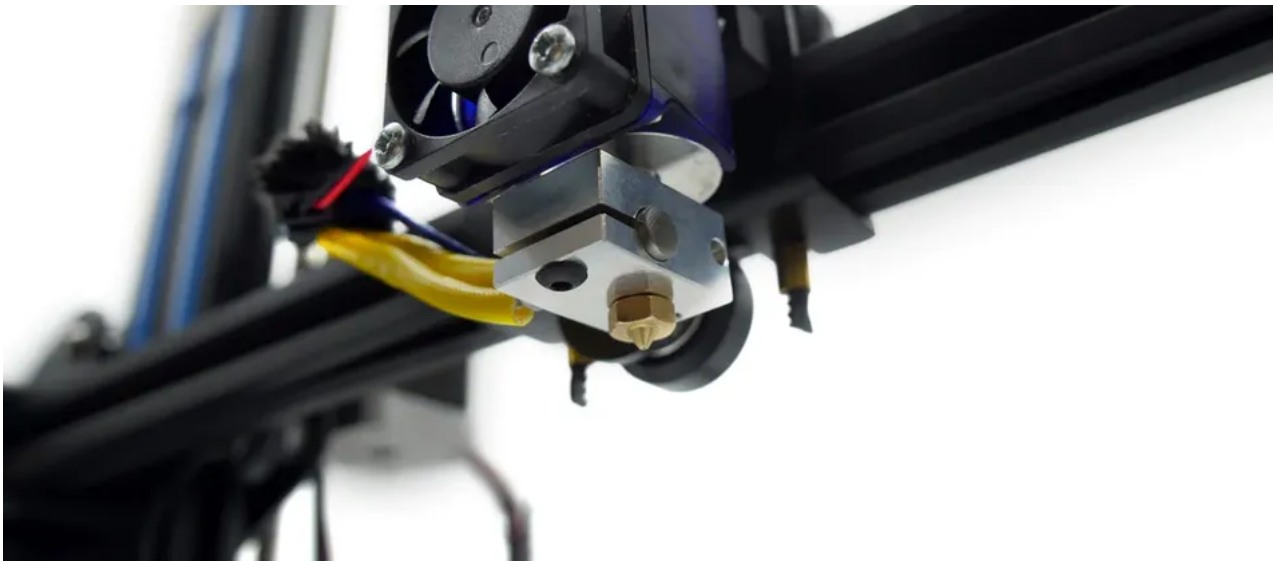
Firmware

Many Ender 3 users find they can achieve faster printing speeds by replacing the printer's default Marlin firmware with Klipper firmware. Klipper provides advanced motion kinematics and additional tools that can help Ender 3 users get faster speeds out of their hardware, such as improved control over acceleration. Below is a general guide on how to install Klipper on an Ender 3.

1. Obtain a Raspberry Pi or another supported microcontroller board.
2. Install Klipper-compatible firmware on the microcontroller board. For a Raspberry Pi, you can use a distribution like OctoPi, which includes the necessary software.
3. Connect the microcontroller to your Ender 3's control board using appropriate cables (e.g., USB or serial).
4. Install the necessary software on the microcontroller. For a Raspberry Pi running OctoPi, follow the installation instructions provided by the OctoPrint project.

7. Within the printer.cfg file, you'll need to define settings specific to your Ender 3. This includes details like bed size, steps per millimeter for each axis, extruder settings, and other parameters.
8. You may refer to the Klipper documentation or community forums for guidance on the specific settings to use for an Ender 3 printer.
9. After making the necessary configuration changes, save the printer.cfg file.
10. Restart the Raspberry Pi or microcontroller board to apply the changes.
11. Once the microcontroller restarts, you can connect to Klipper via OctoPrint or another compatible interface. Open the web interface or use a terminal-based program to establish a connection.
12. From the interface, you can upload and manage G-code files, send commands, and control your Ender 3 printer.

Extruder and Hot End



Installing the E3D V6 hot end can increase print speed

The combination of an upgraded hot end and extruder can enhance the printing speed of an Ender 3 in the following ways. An E3D V6 all-metal hot end, for example, with its lightweight design and efficient thermal properties, enables rapid heating and cooling, reducing the time required for temperature changes during the printing process. Its optimized heat dissipation maintains stable temperature control, allowing for faster print speeds without compromising print quality.

Additionally, a new extruder like the Bondtech BMG, equipped with a dual-drive gear system, provides superior filament grip and feeding performance. The dual-drive mechanism ensures consistent and reliable filament feed, preventing slippage even at higher speeds. This enhanced grip allows for faster filament flow, enabling higher printing speeds without compromising accuracy or print quality.

Overall, the combined hardware upgrade of a V6 hot end and BMG extruder reduces the likelihood of various extrusion-related issues that can hinder print speeds. The V6 hot end's efficient thermal performance minimizes

Print Times

As we have discussed, the print speed of the Ender 3 is not the only factor that determines how long your print job will take. For example, builds will naturally take a long time if parts are overly complex, incorrectly oriented, or feature very thin layers — even if your printer head is moving at lightning speed. This section looks at ways to shorten print times without actually altering the printing speed.

Efficient Print Planning

Print planning — planning of the layout, order, and orientation of your print jobs — can shorten print times. The optimal planning strategy can vary depending on the specific requirements of your print.

One aspect to consider is the placement and orientation of parts on the print bed. The orientation of a print can impact not only the strength and aesthetics of the final product but also the print time. For instance, orienting a part to minimize the height can reduce the total number of layers needed and thereby decrease print time. However, this orientation may require support structures that, while ensuring print integrity, will add to the print time and material usage. In some cases, a part may be designed to print in a specific orientation, and changing this can adversely impact the part's function.

Batch printing is another tactic that can be used to optimize print times. If multiple copies of the same part need to be printed, it might be faster to print them all at once rather than individually, depending on the specific 3D printer and part geometry. For example, when printing multiple parts simultaneously, the printer's hot end does not have to travel back and forth

reduce the non-print movements and hence decrease the print time. 3D slicing software typically optimizes this to an extent, but manually optimizing the print sequence can sometimes yield better results.

Extrude More Material

Layer height and nozzle size are tied to the speed of 3D printing, with an intricate interplay between them. A crucial element of understanding this relationship begins with the nozzle size of the 3D printer. The Ender 3 typically comes with a nozzle size of 0.4mm. The nozzle size governs the width and height of the filament line that is extruded, with larger nozzles capable of extruding more filament in a given time period than smaller ones.

A larger nozzle, such as a 0.6 mm or 0.8 mm, can significantly decrease print times. This is due to the increased amount of filament that can be deposited. For example, a 0.8mm nozzle can theoretically print almost four times faster than a 0.4 mm nozzle, given that it can extrude four times the volume of filament in the same time period. This is useful for large, less detailed prints where the larger layer lines won't detract from the overall aesthetic or function of the printed part.

Layer height also plays a vital role in the print speed equation. Higher layer heights mean fewer total layers, resulting in faster prints. For instance, a print with a layer height of 0.3 mm will have half as many layers as the same print at a layer height of 0.15 mm, thereby reducing the print time by roughly half. However, similar to the trade-off with larger nozzles, a larger layer height can lead to a decrease in the level of detail and a rougher surface finish.

Reduce Infill

The infill density and pattern also significantly influence print time. Infill is the internal structure of the print, and a higher infill density means more material is used, which increases print time. Reducing infill density can therefore speed up prints.

However, this might affect the strength and weight of the printed part. Additionally, different infill patterns can be faster to print; for example, a rectilinear infill is often quicker to print than a honeycomb pattern, but it may not provide the same strength.

Reduce Supports

Supports also add to the print time, as they require additional material and often necessitate slower print speeds to ensure accuracy. Of course, supports play a vital role in preventing print failure, but it is sometimes possible to eliminate them by re-designing the part to remove overhangs or simply reorienting the part on the print bed.

Conclusion: Balancing Speed and Quality in Your Ender 3 Prints

Balancing speed and quality is one of the main challenges of 3D printing. While increasing print speed is often desirable to reduce production time, it's important to recognize that speed enhancements can also lead to a reduction in the overall print quality. Factors such as the layer adhesion, precision, and surface smoothness can suffer when trying to push the printer beyond its intended limits.

Frequently Asked Questions (FAQs)

Q: Is it safe to continuously print at high speeds with my Ender 3?

The safety of high-speed printing largely depends on the specific configurations and modifications made to the printer. While the Ender 3 is generally capable of high-speed printing, it's critical to consider factors such as the printer's cooling system and motor capabilities. Inadequate cooling at high speeds can lead to heat creep, which can result in clogs or, in extreme cases, damage to the printer. Similarly, continuously running the motors at high speeds can lead to excess heat and wear, reducing their lifespan. Regular maintenance and monitoring can help ensure safe operation.

Q: How can I tell if my print speed is too high?

There are several signs that your print speed might be too high. These include poor layer adhesion, which can lead to weak prints or print failures, visible vibrations or wobbling during printing, under-extrusion or thin layers, and overall poor print quality. If you're noticing any of these issues, it might be worth reducing your print speed and seeing if the quality improves.

Q: Can I use different print speeds for different parts of my print?

Yes, using different print speeds for different parts of your print is a technique for balancing speed and quality. For example, you might choose to print the outermost layer at a slower speed to ensure a smooth finish, while the infill can be printed at a higher speed since it's not

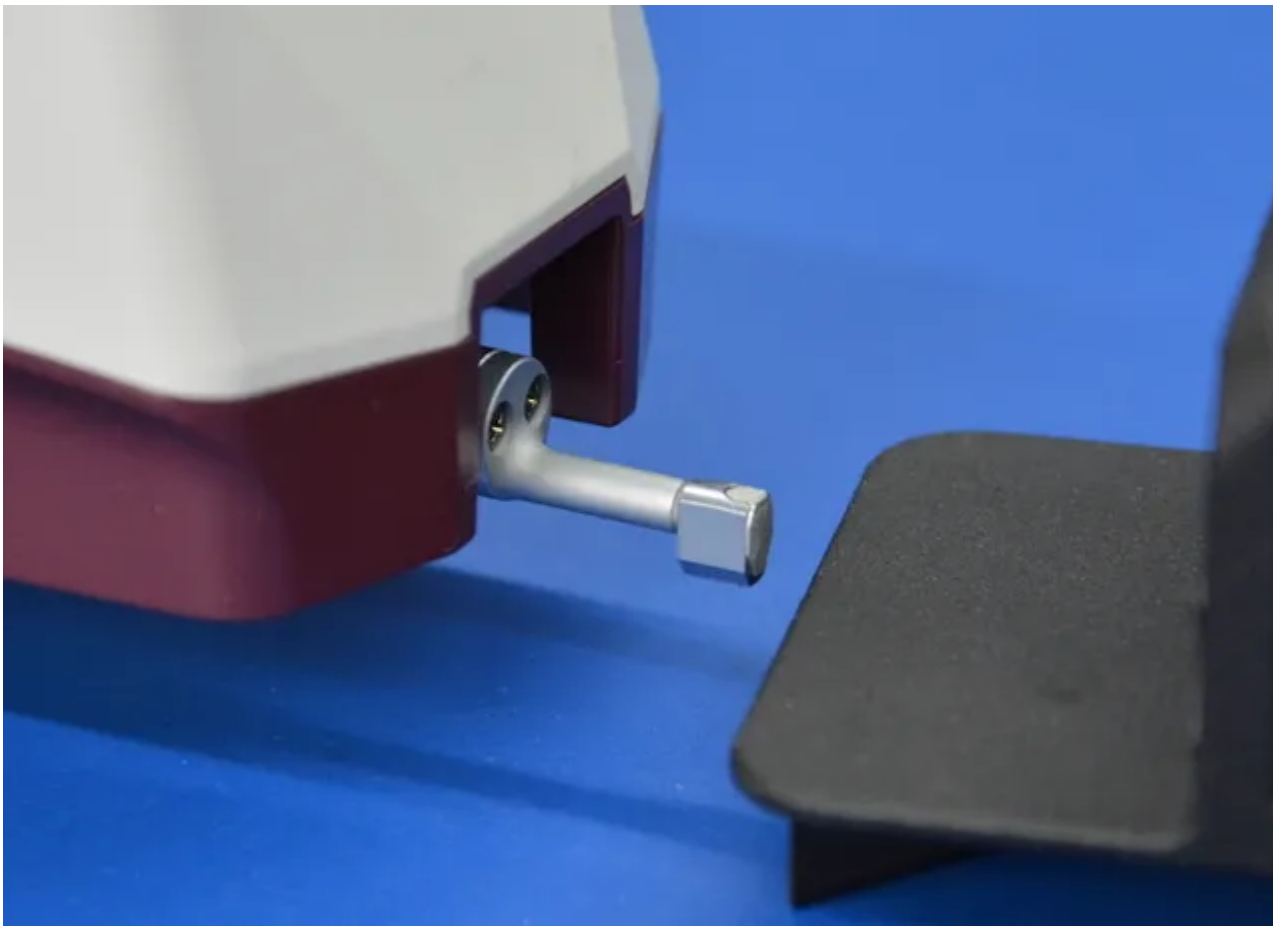

ASA vs PLA: A Comprehensive Comparison and Guide
13 minutes read


ASA vs PETG: A Comprehensive Comparison and Guide
13 minutes read

Surface Roughness in 3D Printing

Nikolaus Mroncz from Xometry Europe
less than a minute read

deposition modeling parameters for improved PLA and ABS 3D printed structures. International Journ L aterials and Manufacture. 2020 Sep 1;3(3):284-97.



This article explains what roughness is in 3D printing, provides surface roughness measurements for SLS, MJF, FDM, DMLS, Carbon DLS, and Polyjet 3D prints and highlights the benefits of surface treatments to achieve a smooth surface for 3D printed parts.

design of the part to be printed and the position and orientation of the part during build also influence the surface quality.

Before choosing an additive manufacturing process, one needs to consider the roughness (measured in most cases in Ra) of the 3D printed parts. For the technologies that use support structures, like fused deposition modeling (FDM), special care needs to be taken since there is a contact point between the support structures and the actual part which has a

CONTINUE READING →

HDPE 3D printing: applications, advantages, alternatives

Benedict O'Neill
8 minutes read

Industrializing 3D printing

Zach Winn
5 minutes read

3D printing is great, but not for everyone

Marianne Vang Ryde from Technical University of Denmark
6 minutes read

Semiconductors

3D Printing

Sensors

RESOURCES

Submit Content

Writing Guide

Sitemap

Cookie Settings