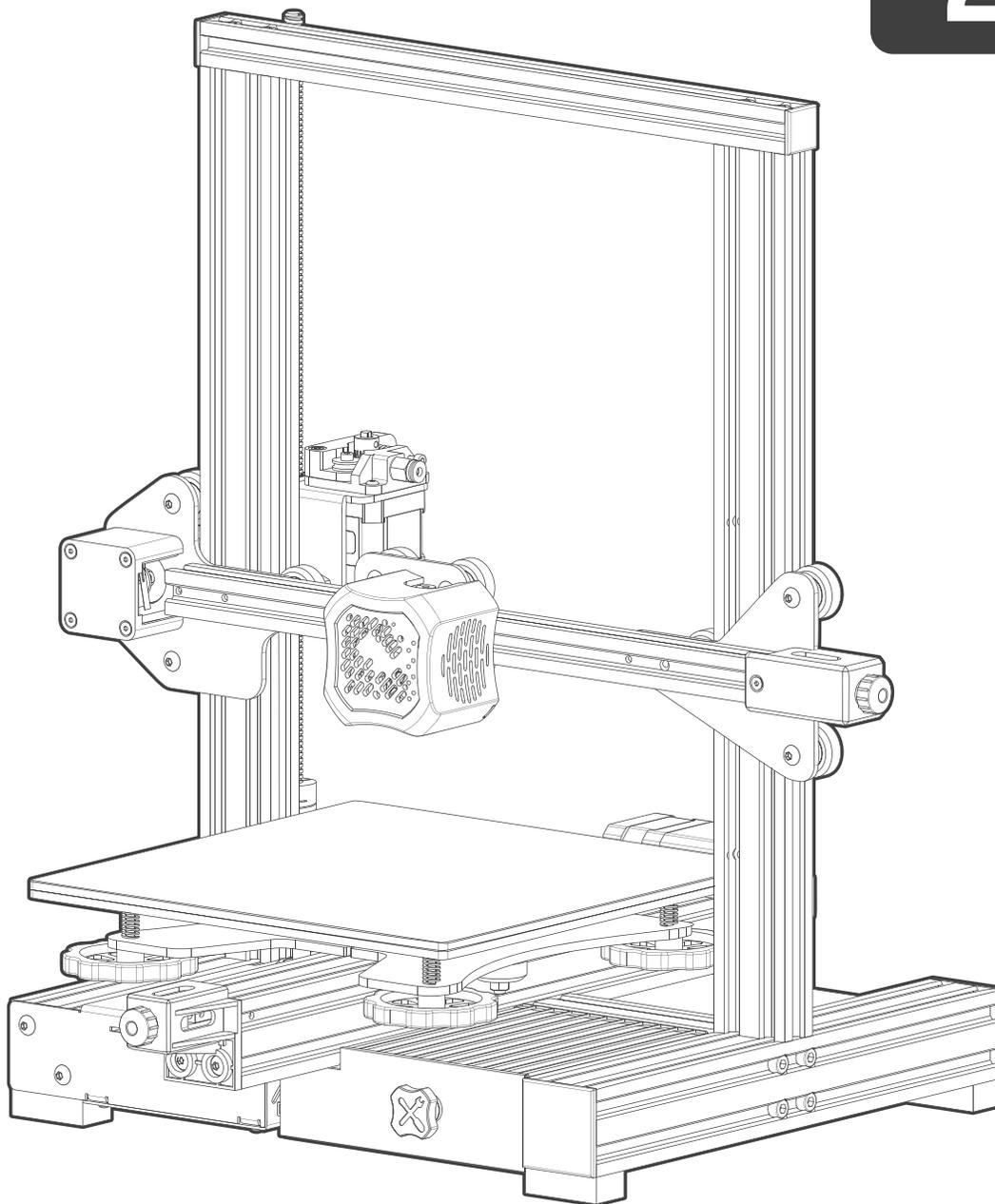


# QUICK START GUIDE

## 3D PRINTING (FDM)

2024



OVER 100,000+ DOWNLOADS

# TABLE OF CONTENTS

Where the heck am I?

<b>TABLE OF CONTENTS</b>	_____	<b>1</b>
<b>3D PRINTERS</b>	_____	<b>2</b>
<b>PRINTER CALIBRATION</b>	_____	<b>3-5</b>
<b>UPGRADES</b>	_____	<b>6-7</b>
<b>TIME TO PRINT</b>	_____	<b>8</b>
<b>MATERIALS</b>	_____	<b>9</b>
<b>SLICER SOFTWARE</b>	_____	<b>10</b>
<b>SLICING</b>	_____	<b>11-12</b>
<b>GOING FURTHER</b>	_____	<b>13</b>

# 3D PRINTERS

Which 3D printer and workflow should I choose?

## Which Printer Should I Get?

Figuring out where to start with 3D printing can be daunting. Confusion abounds concerning material choices, slicer settings, and even which printer to choose in the first place. Our quick start guide aims to help beginners answer these questions and start printing. Below are our recommendations for your first printer based on your budget.



### Machine

**Ender 3 v3 SE**

**Sovol SV07 Plus**

**Bambu Labs X1 Carbon**

### Overview

The Ender 3 V3 SE, an upgrade in the popular Ender series, offers improved features at an affordable price. While it's a solid choice for entry-level 3D printing, investing in a few upgrades can significantly enhance its reliability and print quality, making it a more versatile choice for both newcomers and seasoned enthusiasts.

The Sovol SV07 Plus is a robust and adaptable 3D printer known for its open-source flexibility. It features advanced mesh-based bed leveling and an efficient filament detection system, ensuring consistent printing results. With its sturdy, modular design, the Sovol caters to both hobbyists and professionals, offering reliability and ease of use.

The X1 Carbon 3D printer revolutionizes printing with LiDAR failure detection, automatic bed leveling, and resonance calibration, ensuring high-quality, fast prints with minimal setup. Its advanced features make it ideal for superior printing. The X1D1 complements this by offering ease of use, designed to work perfectly out of the box, simplifying 3D printing for all users.

### Build Volume

L 220mm  
W 220mm  
H 250mm

L 300mm  
W 300mm  
H 350mm

L 256mm  
W 256mm  
H 256mm

### Average Print Speed

60mm/s

80mm/s

200mm/s

### Price

**\$220**

**\$470**

**\$1200**

Regardless of machine, these are all capable of taking your idea from a 3D design to a physical object at home.

# PRINTER CALIBRATION

How do I make and maintain consistent, high quality prints with my printer?

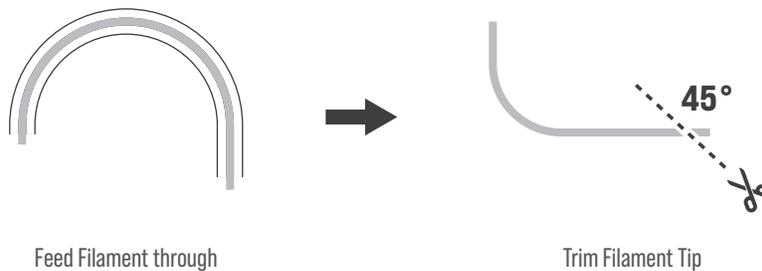
## How Do I Calibrate My Machine?

In 3D printing, calibration and quality control are essential components that can make or break your 3D printing experience. Here are step by step instructions to get your 3D printer to produce consistent and accurate prints.

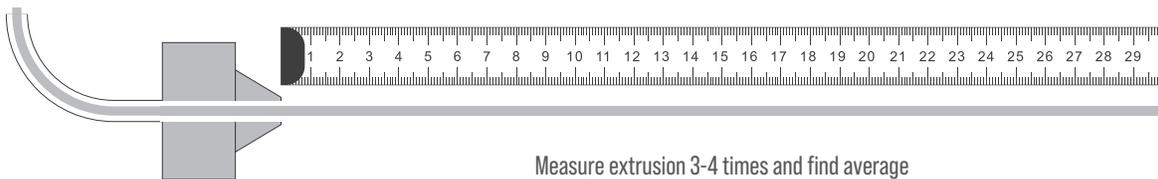
**NOTE:** This guide uses PLA+, but the instructions apply in general for most common materials.

### Calibrating your Extruder

To start, turn on your printer and set the hot end temperature to **210° - 220° C**. Feed the filament through your extruder until it reaches the end of the bowden tube. Trim the fed filament to **45°** angle to assist with feeding later.



Use your printer control panel to extrude **100mm** of filament. The exact process will vary depending on the printer. Measure the distance from the end of the extruded filament to the end of the bowden tube. Repeat **3-4** times and average the results.



Locate your printer's current **E-step** value in the printer's control settings. Calculate the new E-step value by multiplying the current E-step value by the expected extrusion length, and then divide by the actual extrusion length. Enter this new e-step value into your printer's settings and save it.

Formula

$$\text{Current E-step Value} \times \text{Expected Extrusion} / \text{Amount Extruded} = \text{New E-step Value}$$

Example

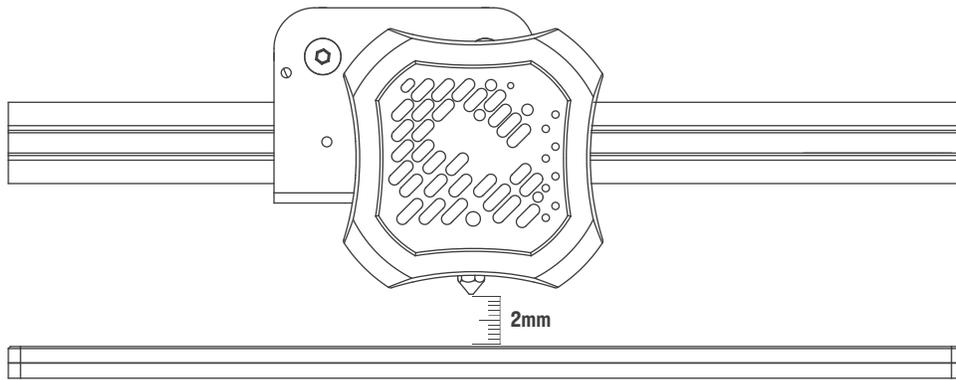
$$97 \text{ steps} \quad \times \quad 100\text{mm} \quad / \quad 103\text{mm} \quad = \quad 94.175$$

# PRINTER CALIBRATION

How do I make and maintain consistent, high quality prints with my printer?

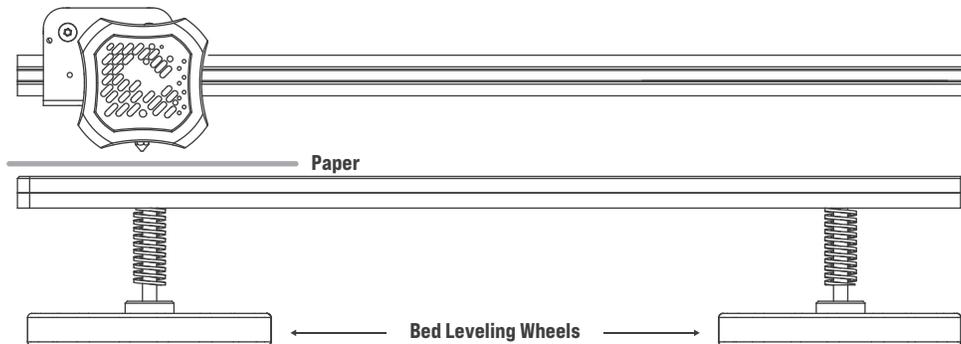
## Leveling the Printing Bed

Use the printer's controls to move the hot end above each bed leveling wheel, or manually move it after disabling the X/Y stepper motors. Use the wheels to bring the bed close to the nozzle (~2mm).



Bring bed to 2mm from nozzle

Using a piece of paper or a feeler gauge (0.2mm), move from corner to corner adjusting the bed height using the bed leveling wheels so the nozzle just touches the paper/gauge. Repeat for each wheel; this may require several attempts to get right.



Adjust bed leveling wheels until every corner touches the paper/gauge

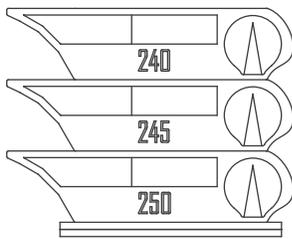
Print a test layer to confirm the bed is correctly leveled. Many files can be found at all 3D file repos, but you want to find one that reaches the entire bed to get an accurate reading on the bed's level. For the most accurate test, find a file specifically made for the printer you are using.

# PRINTER CALIBRATION

How do I make and maintain consistent, high quality prints with my printer?

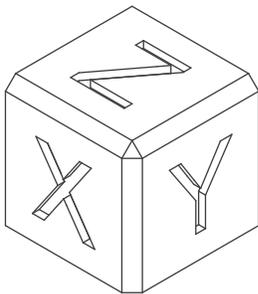
## Benchmarking Test Prints

A test print can be used to check if everything is functioning properly with your printer. Here are the most common test prints with explanations of their purposes.



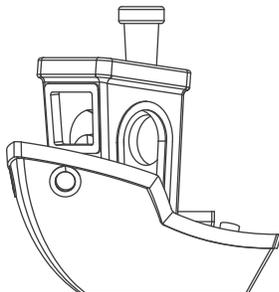
### Temperature Towers

The Temperature Tower is a way to test stringing, overhangs, and bridging at different temperatures, all in one part. This is accomplished by setting G-Code parameters to change the nozzle temperature across set distances. These references help you find the ideal nozzle temperature for your machine.



### Calibration Cubes

The Calibration Cube is one of the most popular test items used for 3D printer calibration. The design is usually a simple square measuring 20mm<sup>3</sup> that is printed in each corner and the center of the build plate. This file is best for measuring the dimensional accuracy of the 3 Cartesian axes (X,Y, and Z) and can assist you in locating any problems with your extrusion rate, bed leveling, or other factors that might affect the print's accuracy.



### Benchy

A Benchy is an unassuming tugboat model that functions as a torture test for your printer. It features difficult bridges, overhangs, towers, text, archways, and circles. It's a fairly short print, coming in at, or under, two hours. It is hard to print a perfect Benchy, but because of this, you can get very close to ideal prints of other parts if you tune your printer based on Benchy results. For most users, Benchy has become the go to print to test new printers.

Making a test print gives you the chance to ensure your printer is set up to produce ideal parts. Even doing one of these 3 test prints can save you **HOURS** of time – and the cost of wasted filament.

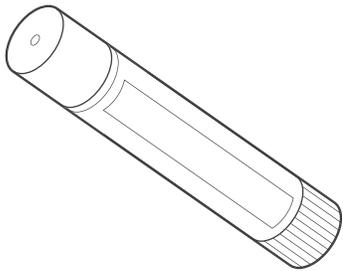
# UPGRADES

I want to get more from and improve my machine

## Upgrading Your Tools

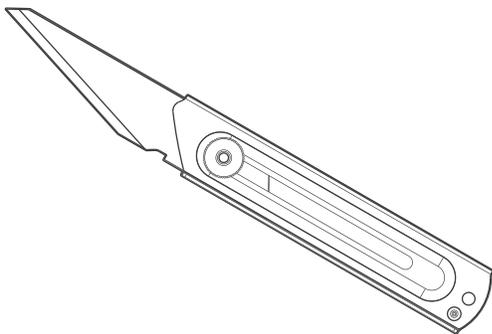
These simple tools can make a world of difference when working with your 3D Printer.

With any setup, you will eventually want to optimize it to fit your unique needs and style.



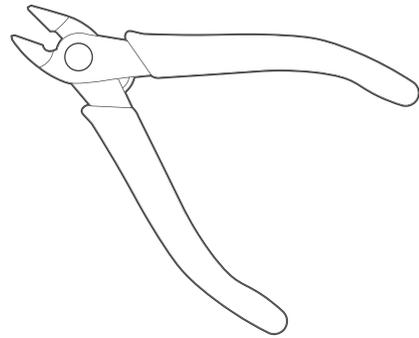
### GLUE STICK/ADHESIVE

While odd at first glance, a gluestick is often an invaluable tool for 3D printing. If you ever face adhesion issues that cleaning the build surface doesn't fix, rubbing a gluestick over it may sort you.



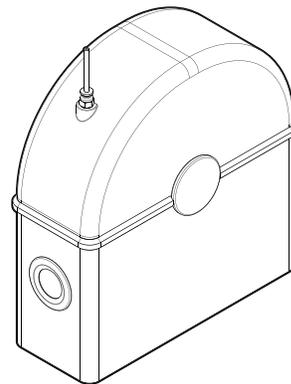
### CRAFT BLADE

Precision cutting tools are crucial for 3D printing, offering sharp, accurate cuts for refining and detailing printed objects. Ergonomically designed, they are essential for achieving a professional finish in 3D printed projects.



### MODEL NIPPERS

You'll usually find cheap nippers with just about any printer you buy, but it may be worth investing in some quality model nippers if you're going to be printing more than a handful of parts. Nippers remove defects, supports, and much more.



### FILAMENT DRYER

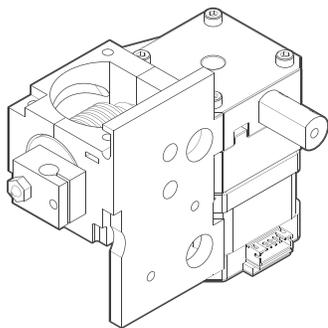
A filament dryer is now necessary for 3D printing, maintaining filament quality by removing moisture. It prevents printing issues and is especially useful in humid conditions, ensuring consistent, high-quality prints.

# UPGRADES

I want to get more from and improve my machine

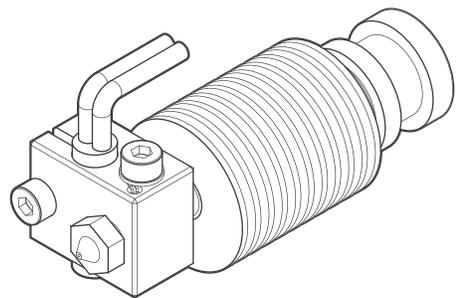
## Upgrading Your Features

Consider upgrading the following components for your printer. For lower budget 3D printers these can easily increase the types of materials you can work with as well as vastly improving the reliability of your prints.



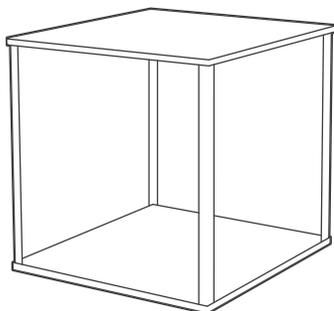
### DIRECT DRIVE

A direct drive extruder improves 3D printing with precise filament control, reducing retraction issues and enhancing print quality, especially for flexible materials. It's crucial for finer details and smoother finishes in prints.



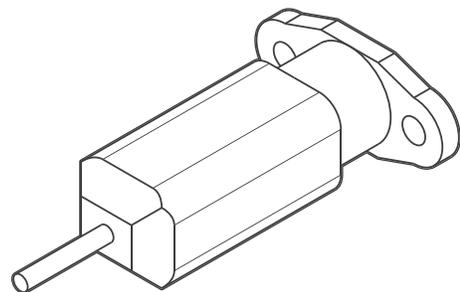
### ALL METAL HOTEND

An all metal hotend enables high-temperature printing, expanding material options for 3D printers. Its durability enhances print quality and reliability, especially with high-temperature filaments.



### PRINTER ENCLOSURE

Enclosures help trap heat generated by the printer, reducing part warping and keeping your room a little cooler, quieter, and cleaner. It can also reduce the fumes produced by some materials.



### AUTO LEVELING SENSOR

An auto bed leveler streamlines 3D printing by automatically calibrating the print bed, ensuring consistent, high-quality prints and reducing manual setup time.

# TIME TO PRINT

A checklist to help your workflow

## The 3D Printing Workflow Checklist

Now that you have chosen and calibrated your machine, its time to print something!

The most common 3D printing workflow is as follows:

**Choose a project**

With the machine squared away, decide what you want to build. Come up with some ideas, take a look around online, and pick what you like.

**Choose a material**

Different parts have different objectives, and different objectives have different material requirements. Decide which material suits your goals.

**Upload your project file to a slicer**

Reference the **SLICING** section of this guide.

**Slice/Export your file**

Use the Slicer software to prepare your 3D file and make it compatible with your machine.

**Send file to your 3D printer**

Import the sliced file into your machine and get it ready to start.

**Watch it print**

Finally through the setup, you can build your part and watch it go. A few hours later, you'll be able to pry it off the build plate.

**Remove file from print bed/Clean for future printing**

Going over your part and scanning for quality issues will save you the trouble of unreliable parts in the future. Save yourself some headache and double check your work.

**Enjoy**

Remember its only just begun. Upgrading and tinkering with your machine is the lifeblood of future success.

If you get lost or confused on what to do next during your project, reference this checklist again to ensure you are on the right path to a quality end product.

# MATERIALS

Which material should I be using and what are their differences?

## Choosing a Material

Now that your machine is calibrated, its time to decide what material to print with. Here are the most common hobby materials, along with the pros and cons of each.

	Overview	Pros	Cons
<b>PLA</b>	Polyactic Acid is the most widely used material in FDM 3D printing and is known for its accessibility. It is easy to run, cost-effective, and suitable for all kinds of prototyping.	<ul style="list-style-type: none"> <li>Available Anywhere</li> <li>Cost-Effective</li> <li>Low Barrier To Entry</li> </ul>	<ul style="list-style-type: none"> <li>Warp in hot conditions</li> <li>Lower strength</li> </ul>
<b>MAKERS CHOICE</b> <b>PLA+</b> <b>PLA Pro</b>	Polyactic Acid + supersedes PLA in some technical spheres, as it is an improvement of the formula, which offers greater strength and shock resistance than regular PLA.	<ul style="list-style-type: none"> <li>Greater Strength</li> <li>Greater Elasticity</li> <li>Ductile</li> </ul>	<ul style="list-style-type: none"> <li>More Expensive</li> <li>Needs Higher Temp</li> <li>Warp In Heat</li> </ul>
<b>PETG</b>	Polyethylene Terephthalate Glycol is a commonly used material that is known for its strength and ease of use. Suitable for long term fixtures, PETG is capable of greater continued use and wear than PLA/+, but fails abruptly under significant force.	<ul style="list-style-type: none"> <li>Greater Impact Strength</li> <li>Resists Heat</li> <li>Water Resistant</li> </ul>	<ul style="list-style-type: none"> <li>Prone To Surface Defects</li> <li>Expensive</li> <li>Shatters Under Shock</li> </ul>
<b>NYLON</b>	Nylon is a generic term for synthetic polyamides and can be found in most anything you use from day to day. They are useful allrounders that can be tailored to the strengths needed by a project by various infills like glass or carbon fiber, but are difficult to run and remain expensive.	<ul style="list-style-type: none"> <li>High impact strength</li> <li>High elastic strength</li> <li>Resists Heat</li> </ul>	<ul style="list-style-type: none"> <li>Absorbs Water</li> <li>Prone to Warp</li> <li>Very Expensive</li> </ul>
<b>TPU</b>	Thermoplastic Polyurethane are good for parts that need to be flexible, comfortable, and oil resistant, though they can be tricky to run reliably. TPU should be used only for the specialized tasks it is suited for. Printing it requires a direct drive extrusion system.	<ul style="list-style-type: none"> <li>Highly Elastic</li> <li>Oil Resistant</li> <li>Good Shock Absorption</li> <li>Good Sealing Capabilities</li> </ul>	<ul style="list-style-type: none"> <li>Tricky To Run</li> <li>Limited Application</li> <li>Requires Direct Drive</li> </ul>

As the world of 3D printing expands, so will the materials used. Breakthroughs such as PLA+ and materials that give finishes while printing are just the start of what is possible with 3D printing. Experimenting with a variety of materials is a great way to experience all there is in the space and learn new technologies.

# SLICER SOFTWARE

The software

## Choosing a Slicer

In 3D printing, the right slicer software is vital for quality results. Here are three top options to consider, tailored to various needs and skill levels.



### Ultimaker Cura

Ultimaker Cura remains a top choice in 3D printing software due to its widespread use and user-friendly design. Most online tutorials and guides use Cura's naming conventions, making it a go-to resource for many in the 3D printing community. It supports a variety of 3D printers and offers customizable settings, ensuring optimal print quality. Regular updates keep it aligned with the latest 3D printing advancements, making it suitable for both novices and experienced users.



### Orca Slicer

Orca Slicer is a progressive choice for 3D printing enthusiasts, especially power users. It is a derivative of Super Slicer, which itself is based on Prusa Slicer, and is an open-source fork of Bambu Slicer. While still in beta, Orca Slicer is known for producing faster prints. Its development focuses on providing advanced features and efficiencies, appealing to users who seek more control and optimization in their slicing software.



### Bambu Slicer

Bambu Slicer is a proprietary software specifically designed for Bambu printers. It unlocks the full feature set of these printers. Although it offers specialized functions for Bambu printer users, those without specific needs for Bambu Slicer's unique features might consider alternatives like Orca Slicer. This provides more flexibility and avoids reliance on proprietary software, especially for users keen on customization and privacy.

Ultimately, the choice of slicer software largely depends on individual needs and preferences. Regardless of the option you choose, each slicer offers its unique strengths and capabilities, ensuring a satisfactory 3D printing experience.

# SLICING

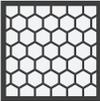
And the settings

## Setting Up a Slicer

Apart from hardware calibration, the slicer is where all of the magic happens. From determining nozzle temperature, infill pattern and density, down to support structures; if it has to do with how your part is built, it happens here.

We'll take this one step at a time by listing a setting, explaining what it does, and what our recommended values are.

**NOTE:** This guide uses CURA SLICER, but the instructions are a general reference for most common slicer softwares.

	Overview	Example
<b>Quality</b>	<b>Quality</b> refers to layer height and line width. Layer height can be as low as 0.1mm for exceedingly fine detail, or as high as 1mm for fast, rough prototypes. A good rule of thumb is 25-75% of your nozzle's width.	 High Medium Low
<b>Walls</b>	<b>Walls</b> refers to a suite of settings that will determine how thick the outer walls of your part are, as well as various quality settings in relation to them. It is recommended to use a wall count of at least 5 for accessories/peripherals, and all walls for receivers and other structurally-important parts.	
<b>Top/Bottom</b>	<b>Top/Bottom</b> of a 3D print affects surface quality more than structural integrity. Key settings include top/bottom thickness, measured in multiples of layer height. It's suggested to maintain a minimum top thickness of 1mm and a bottom thickness of at least 4x the layer height.	
<b>Infill</b>	<b>Infill</b> forms a 3D print's internal structure, improving efficiency over solid prints. Its density and pattern affect strength, weight, and printing time. Select infill patterns like concentric, cubic, or lines based on needs for flexibility, strength, or speed. For parts receiving little or indirect force, lower infill (30% - 50%) is acceptable.	
<b>Material</b>	<b>Material</b> refers to settings that guide temperature and flow. The nozzle's printing temperature, typically around 210°C, melts the filament for extrusion. Build plate temperature, about 55°C, promotes adhesion and reduces warping. Both temperatures can vary based on material and manufacturer's recommendations.	
<b>Cooling</b>	<b>Cooling</b> settings enhance surface quality but need to be managed to avoid warping or poor bed adhesion. You should enable Print Cooling and set fan speed to 100%, typically without adjustment unless testing a new printer or material. The Regular Fan Speed at Height setting turns the fan on at a set height, ideally off for the first layer (0.7mm) to avoid poor bed adhesion or warping. The Minimum Layer Time and Lift Head settings, which determine waiting time before moving to the next layer and raising the printhead respectively, should be set to 3 seconds and enabled. These ensure no warping or delamination for the shorter layers.	
<b>Travel</b>	<b>Travel</b> refers to when the printhead is moving without printing. These settings are crucial for reducing the stringing affect. Retraction is a feature that pulls back filament to prevent unnecessary extrusion when the printhead navigates over areas not meant for printing. A good <b>Retraction Distance</b> would be 5mm and a <b>Retraction Speed</b> of 45mm/s is recommended. When the travel distance is short, a <b>Retraction Minimum Travel</b> of 1.5mm is useful. <b>Combing</b> is another feature that keeps the tool path within the print boundaries, reducing stringing on external faces, and should be set to "All".	

# SLICING

More advanced settings

## Build Plate Adhesion

**Build Plate Adhesion** refers to settings that improve print stability on the build plate. 3 methods are Brim, Skirt, and Raft:

- **Brim:** Additional concentric rings around the print's first layer, aiding adhesion for prints with small "footprints" or low surface contact.
- **Skirt:** A separate outline of the print that ensures smooth material flow before the first layer and allows last-minute bed leveling adjustments. Usually generated automatically by slicers.
- **Raft:** An entire part on its own, upon which your model is built. When printing rafts, slicers generally attempt to save material by putting space between adjacent lines. This is the no-holds-barred approach to bed adhesion because the bottom surface area is printed and extended, and your print is actually printed on top of this material.

## Speed

**Speed** settings control printer part movements, affecting print quality and time. Key settings include:

- **Print Speed** (printhead movement while extruding): 45mm/s
- **Infill Speed** (same as Print Speed)
- **Wall Speed** (affects visual quality, slower): 22.5mm/s
- **Outer Wall Speed** (same as Wall Speed)
- **Inner Wall Speed** (between Print and Wall Speed): 35mm/s
- **Top/Bottom Speed** (for first and final layers, slower): 22.5mm/s
- **Support Speed** (for removable support structures): half-speed
- **Travel Speed** (printhead movement without extruding, faster): 150mm/s
- **Initial Layer Speed** (crucial first layer): 20mm/s
- **Z-Hop Speed** (printhead vertical movement to avoid built structures): 3mm/s

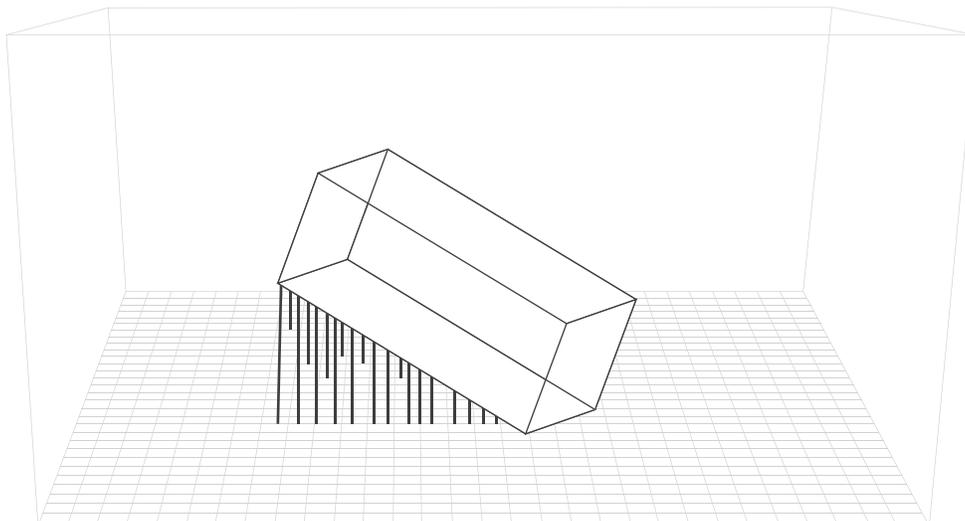
These settings can be adjusted for optimal speed without compromising quality.

## Support

**Support** refers to settings for printing of complex parts with overhangs by creating temporary structures. Key settings include:

- **Generate Support:** On for parts needing support
- **Support Placement:** Typically set to Everywhere to allow support both on part and bed
- **Support Overhang Angle:** 40%-60%, recommended 55% for PLA
- **Support Pattern:** Choose from Lines, Grid, or Zig Zag depending on preference and needs
- **Support Density:** Keep between 10-14% to ensure support strength without waste
- **Support Z distance:** Set to 0.2mm or 0.4mm, a multiple of layer height
- **Support Interface:** Enable it for a buffer zone between part and support. Set Interface Thickness to 1.0mm, Interface Resolution to 0.25mm, Interface Density to 90.0%, and Interface Pattern to Concentric.

## Example



# GOING FURTHER

Where do I go now?

## Never Stop Learning

This guide is meant to get you started in the world of 3D printing, but it is by no means exhaustive. One of the best things about 3D printing is its online, community quality. Here are some of the alternative guides, tutorials, and starter kits we recommend to continue your journey in the space.

### [Creality's NEW Ender 3 v2 3D Printer! Unbox! Build! First Print!](#)

A thorough look into the Ender 3 v2. This is a quick look at all the stuff that comes in the box, along with a **VERY DETAILED** assembly guide. Then of course, there's a file printed with time-lapse.

### [Why is my Z-Axis so high?](#)

A small guide to a very common issue that happens with the Ender 3.

### [Mastering Fusion 360](#)

Tired of printing other peoples projects? Learn how to make your own.

### [Harlot v3 Kit](#)

Everyone's first 3D printed project (at least it should be).

### [PBS Kit](#)

A great build for more reliable kit projects.

### [Free and Open Source Software](#)

A guide to the culture and foundations of the open sharing of CAD files online