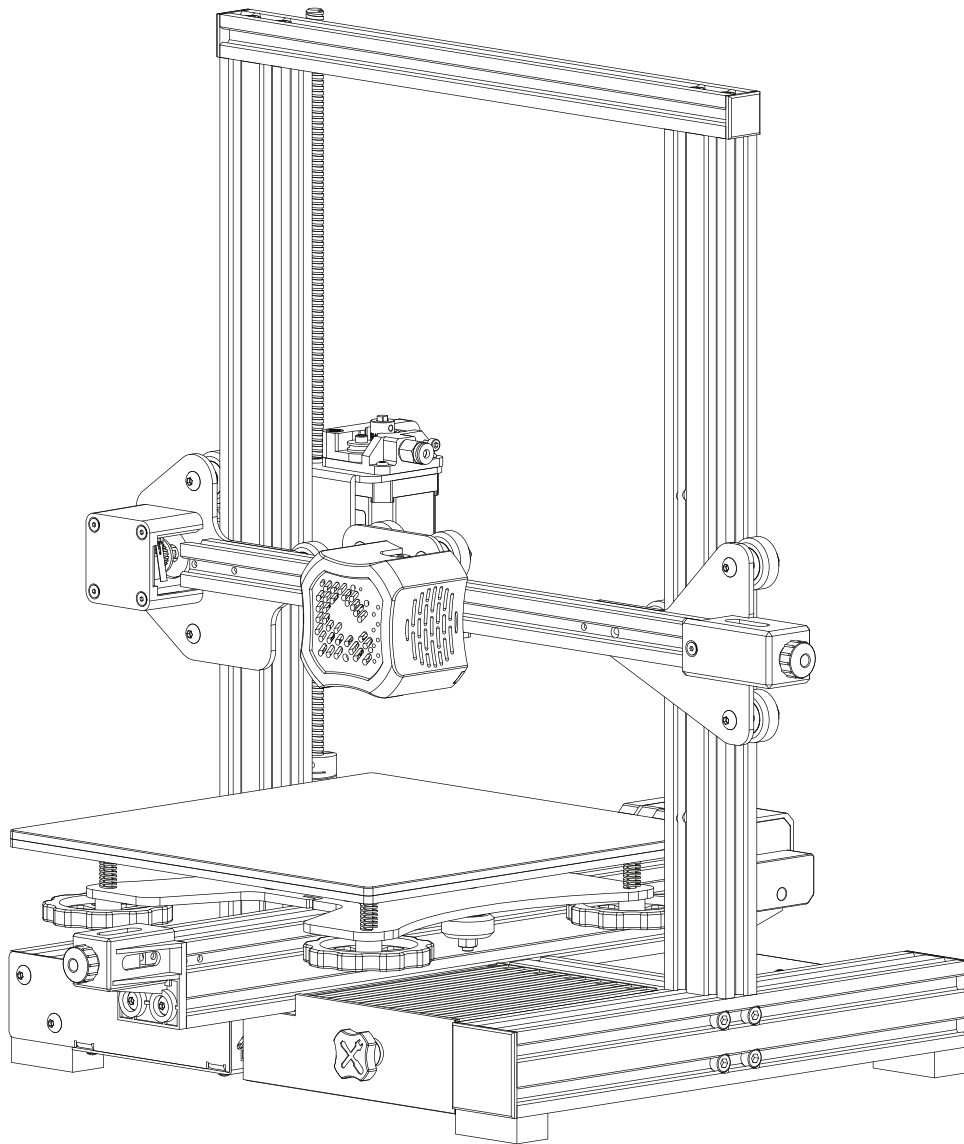


3D PRINTING QUICK START GUIDE



v1.0

TABLE OF CONTENTS

Where the heck am I?

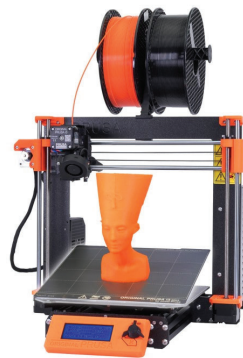
TABLE OF CONTENTS	_____	1
3D PRINTERS	_____	2
PRINTER CALIBRATION	_____	3-5
UPGRADES	_____	6
TIME TO PRINT	_____	7
MATERIALS	_____	8
SLICING	_____	9-10
GOING FURTHER	_____	11

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 the basic price range.



Machine

Ender 3 v2

Prusa i3 MK3S+

Bambu Labs X1 Carbon

Overview

The Ender 3 v2 is one of the most affordable, popular, and supported 3D Printers available. They can usually be found for cheap, but there are some drawbacks with the price. You may want to consider upgrades to make the Ender 3 v2 more reliable and capable of better prints.

The Prusa i3 MK3S+ is a reliable workhorse developed by industry veteran Josef Prusa. It is an entirely open source platform that features mesh-based bed leveling, a reliable filament runout system, and a sturdy and modular design.

The X1 Carbon sets a new standard. It features technology unused in any other printer, like LiDAR failure detection, truly automatic bed leveling, and resonance calibration, all in conjunction with a suite of other quality improvements. For some of the fastest and best looking prints you can make.

Build Volume

L 220mm
W 220mm
H 250mm

L 250mm
W 210mm
H 210mm

L 256mm
W 256mm
H 256mm

Average Print Speed

60mm/s

80mm/s

200mm/s

Price

\$210

\$650

\$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?

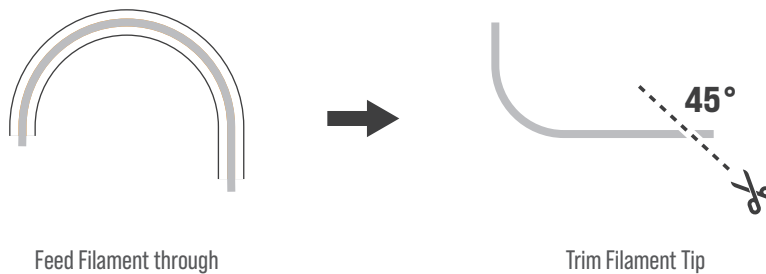
How Do I Calibrate My Machine?

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.

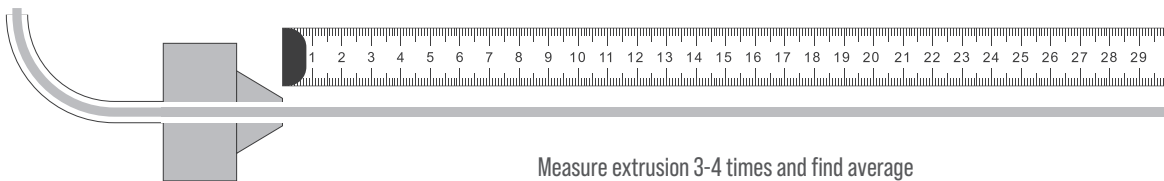
NOTE: This guide uses PLA+, but the instructions apply in general for most common materials.

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

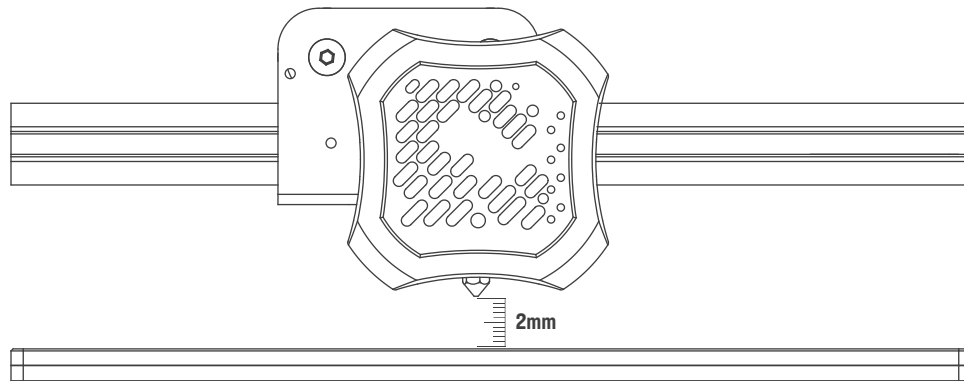
$$97 \text{ steps} \quad \times \quad 100\text{mm} \quad / \quad 103\text{mm} \quad = \quad 94.175$$

PRINTER CALIBRATION

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

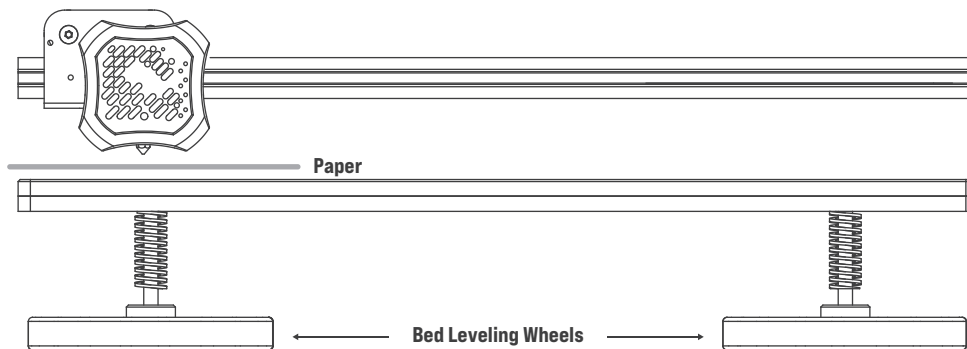
Leveling the Printing Bed

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

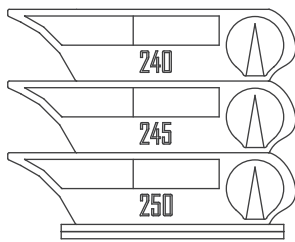
Print a test layer to confirm the bed is correctly leveled. Many files can be found at all 3D file repos, but you want to find one that reaches the entire bed to get an accurate reading on the bed's level. For the most accurate test, find a file specifically made for the printer you are using.

PRINTER CALIBRATION

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

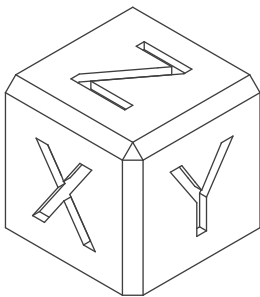
Benchmarking Test Prints

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.



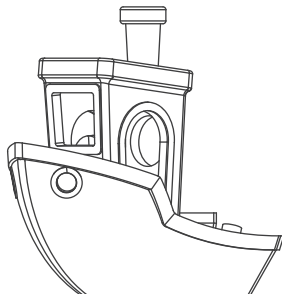
Temperature Towers

The Temperature Tower is a way to test stringing, overhangs, and bridging at different temperatures, all in one part. This is accomplished by setting Gcode 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 20mm³ 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 'secretly' 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.

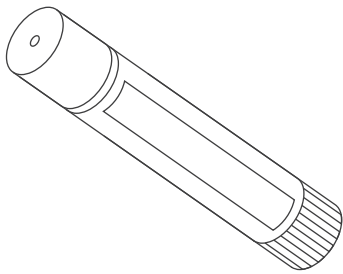
Making a test print gives you the chance to ensure your printer is set up to produce ideal parts. Even doing one of these 3 test prints can save you **HOURS** of time – and the cost of wasted filament.

UPGRADES

I want to get more from and improve my machine

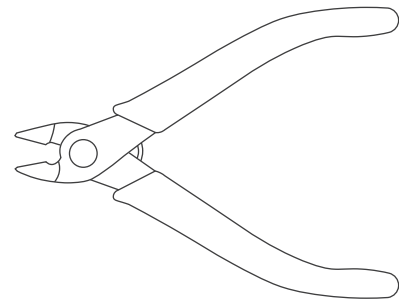
Upgrading Your Setup

With any setup, you will eventually want to optimize it to fit your unique needs and style. There is no one ultimate upgrade, so instead we list some essentials for workflow quality of life.



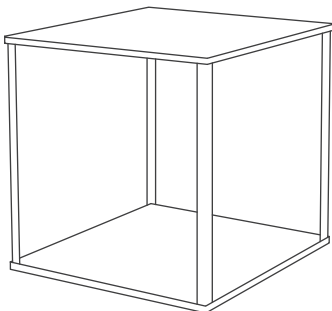
GLUE STICK

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.



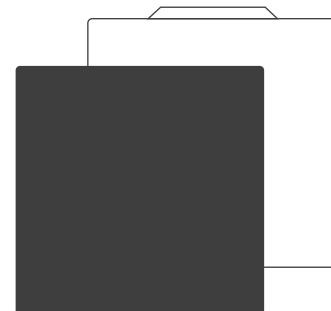
MODEL NIPPERS

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.



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.



REMOVABLE PRINT BEDS

Having a removable print bed can help with part removal and ensure no damage to your printed parts from wrenching them off your machine. Removing the bed and applying a bend will help parts pop off with ease.

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

Watch as the Slicer takes your 3D file and makes 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 are the lifeblood of future successes.

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
PLA	Polyactic Acid is the most widely used material in FDM 3D printing and is known for its accessibility. It's easy to run, cost-effective, and suitable for all kinds of prototyping.	<ul style="list-style-type: none"> ◦ Available Anywhere ◦ Cost-Effective ◦ Low Barrier To Entry 	<ul style="list-style-type: none"> ◦ Warps in hot conditions ◦ Lower strength
PLA+	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"> ◦ Greater Strength ◦ Greater Elasticity ◦ Ductile 	<ul style="list-style-type: none"> ◦ More Expensive ◦ Needs Higher Temp ◦ Warps In Heat
PETG	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"> ◦ Greater Impact Strength ◦ Resists Heat ◦ Water Resistant 	<ul style="list-style-type: none"> ◦ Prone To Surface Defects ◦ Expensive ◦ Shatters Under Shock
NYLON	Nylon is a generic term for synthetic polymade 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"> ◦ High impact strength ◦ High elastic strength ◦ Resists Heat 	<ul style="list-style-type: none"> ◦ Absorbs Water ◦ Prone to Warp ◦ Very Expensive
TPU	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"> ◦ Highly Elastic ◦ Oil Resistant ◦ Good Shock Absorption ◦ Good Sealing Capabilities 	<ul style="list-style-type: none"> ◦ Tricky To Run ◦ Limited Application ◦ Requires Direct Drive

MAKERS CHOICE

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.

SLICING



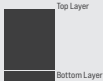
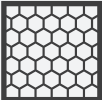



The software and the settings

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
Quality	Quality 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
Walls	Walls 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.	
Top/Bottom	Top/Bottom 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.	
Infill	Infill 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.	
Material	Material refer 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.	
Cooling	Cooling 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.	
Travel	Travel refers to when the printhead is moving without printing. These settings are crucial for reducing the stringing effect. Retraction is a feature that pulls back filament to prevent unnecessary extrusion when the printhead navigates over areas not meant for printing. A good Retraction Distance would be 5mm and a Retraction Speed of 45mm/s is recommended. When the travel distance is short, a Retraction Minimum Travel of 1.5mm is useful. Combing 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

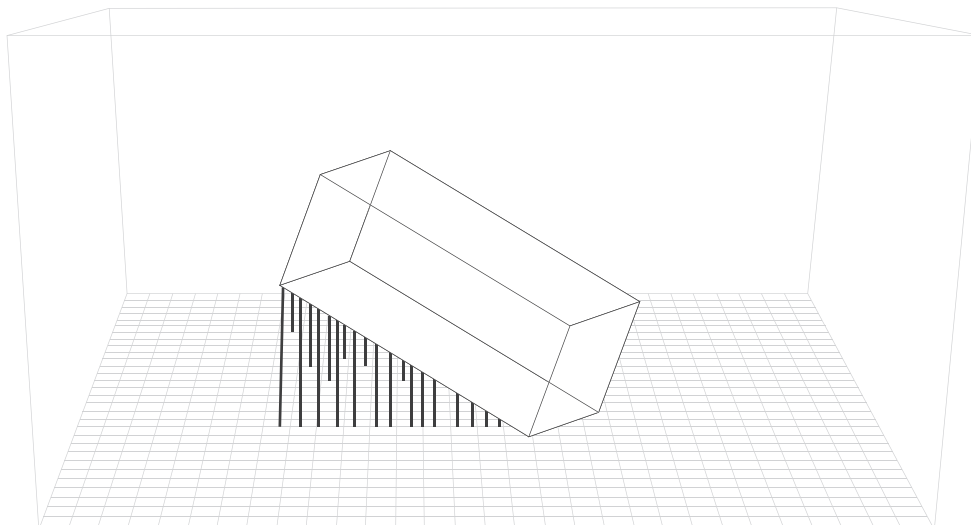
These settings can be adjusted for optimal speed without compromising quality.

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.

[3D Printing Failures: 2022 Edition](#)

This book should help you to diagnose and fix any 3D printing issue you have.

[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.

[Cybernaut Kit](#)

The perfect kit that has everything you need to print with confidence!