



Essential Guide to Perfect Prints

Welcome to Better 3D Printing!

At **what2print**, we know that choosing the right filament is only half the battle. The other half is dialing in your printer to get that perfect, flawless result. This guide is designed to be your go-to resource, whether you're a beginner struggling with your first layer or an expert looking to master a new material.

The 5 Universal Truths of 3D Printing

Before diving into specific filaments, remember these five rules. 90% of printing problems can be solved by mastering them.

- **A Leveled Bed is a Happy Bed:** The single most important step for a successful print. Your nozzle needs to be the perfect distance from the build plate across its entire surface. Too close, and the filament can't extrude. Too far, and it won't stick.
 - **Tip:** Print a "bed level test" model that prints squares in each corner and the center. Adjust as it prints until all squares look perfect.
- **The First Layer is Everything:** Your first layer is the foundation of your entire model. It needs to be "squished" onto the bed just right. It should look clean, uniform, and be firmly attached. If your first layer fails, the rest of the print will too.
 - **Tip:** Slow down your first layer speed in your slicer to 15-20 mm/s to give the filament plenty of time to adhere.
- **Keep it Clean:** A clean build plate ensures good adhesion. A clean nozzle prevents clogs and artifacts.
 - **Build Plate:** Wipe it down with Isopropyl Alcohol (IPA) before every print.
 - **Nozzle:** Use a brass wire brush to gently clean any built-up plastic off the nozzle while it's hot.
- **Keep Your Filament Dry:** Most filaments are hygroscopic, meaning they absorb moisture from the air. Wet filament leads to stringing, weak prints, and a crackling/popping sound during extrusion.
 - **Tip:** Store your filament in an airtight container or bag with desiccant packets. For filament that's already wet, use a dedicated filament dryer or a food dehydrator.
- **Calibrate, Calibrate, Calibrate:** Your printer isn't perfect out of the box. Calibrating your **E-steps** (ensuring the extruder pushes the right amount of filament) and **Flow Rate** (fine-tuning that amount for a specific filament) is key to dimensional accuracy and quality.

Filament-Specific Printing Tips

Every material has its own personality. Here's how to handle the most popular ones.

1. PLA (Polylactic Acid)

The beginner's best friend. Easy to print, low-odor, and available in countless colors.

Setting	Recommended Value	Why?
Nozzle Temp	190°C - 220°C	PLA has a low melting point. A temp tower will find your filament's sweet spot.
Bed Temp	50°C - 60°C (or	Helps with

	none)	adhesion and prevents slight warping on large prints.
Part Cooling Fan	100% (after 1st layer)	PLA loves cooling. It helps solidify overhangs and bridges for sharp details.
Enclosure?		

**Not
needed**

**An
enclosure
can
actually
cause
heat
creep and
clogs with
PLA**

Common Issues & Fixes:

- **Stringing:** Lower nozzle temperature slightly or increase retraction distance/speed.
- **Brittle Prints:** Your filament might be old or "wet." Try drying it.
- **Clogged Nozzle:** Often caused by "heat creep." Ensure your hotend's cooling fan is working properly.

2. PETG (Polyethylene Terephthalate Glycol)

The best of both worlds. Stronger and more heat-resistant than PLA, but easier to print than ABS.

Setting	Recommended Value	Why?
Nozzle Temp	230°C - 250°C	Needs more heat than PLA to melt and bond properly.
Bed Temp	70°C - 85°C	Essential for bed adhesion. PETG can stick too well to

		some surfaces.
Part Cooling Fan	20% - 50%	Too much cooling can cause weak layer bonds. Use less fan than with PLA.
Enclosure?	Recommended, not required	Helps maintain stable temps, reducing

		warping and improving layer adhesion
--	--	---

Common Issues & Fixes:

- **Awful Stringing/Oozing:** This is PETG's main weakness. **DRY YOUR FILAMENT!** Also, tune your retraction settings and consider enabling "coasting" in your slicer.
- **Sticking Too Well to Bed:** Use a release agent like Windex or a glue stick on your build plate.
- **Blobs on Print:** Caused by oozing. In addition to the stringing fixes, ensure your nozzle is clean.

3. ABS (Acrylonitrile Butadiene Styrene)

The strong, heat-resistant choice for functional parts. Can be challenging to print.

Setting	Recommended Value	Why?
Nozzle Temp	240°C - 260°C	High temperature is needed for its strength and properties.
Bed Temp	90°C - 110°C	Absolutely critical to

		prevent the print from warping and detaching .
Part Cooling Fan	0% - 10% (OFF)	Cooling is the enemy of ABS. It causes layer splitting and warping.
Enclosure?	REQUIRED	

**Traps
heat to
create a
stable,
warm
environm
ent,
preventin
g drafts
and
warping**

Common Issues & Fixes:

- **Warping & Layer Splitting:** The #1 issue. Use an enclosure. Print with a wide brim or raft.
- **Bad Smell / Fumes:** ABS releases fumes (VOCs) when printed. Print in a well-ventilated area and use an enclosure to help contain them.
- **Poor Bed Adhesion:** High bed temp is a must. Use an "ABS slurry" or glue stick on the bed.

4. TPU (Thermoplastic Polyurethane)

The flexible, rubber-like filament. Perfect for phone cases, vibration dampeners, and RC car tires.

Setting	Recommended Value	Why?
Print Speed	SLOW: 15 - 30 mm/s	This is the most important setting. Printing too fast will cause the flexible filament to bunch up and

		jam in the extruder.
Nozzle Temp	220°C - 240°C	Similar to PETG, needs a good amount of heat.
Bed Temp	40°C - 60°C	Helps the first layer stick down securely.
Retraction	Turn OFF or use very low values	Retracting a "wet noodle"

of
filament
is very
difficult
and a
primary
cause of
jams

Common Issues & Fixes:

- **Extruder Jamming: Slow down your print speed.** If you have a Bowden tube setup, this is more common. A direct drive extruder is highly recommended for flexibles.
- **Heavy Stringing:** A natural side effect of turning off retraction. You can clean it up post-print with flush cutters or a quick blast from a heat gun.
- **Poor Overhangs/Bridging:** TPU doesn't bridge well. Orient your models to minimize the need for steep overhangs and long bridges

What2Print Troubleshooting Quick Guide

The Problem	What It Looks Like	Top 3 Things to Check
Print Won't Stick	Filament spaghetti on the first layer; corners of the print lifting up.	1. Bed Level: Re-level your bed. Your nozzle is likely too far away. 2. Bed Temp: Increase bed temperature by 5°C. 3. Clean Bed: Clean your build plate with Isopropyl Alcohol.
Stringing / Oozing	Fine, spiderweb-like strings between	1. Dry Filament: Your filament has absorbed moisture. Dry it out! 2. Temperature: Lower your nozzle temperature by 5°C. 3. Retraction: Increase retraction distance or speed in your slicer.

separate
parts of
the
model.

Layer Separation

Gaps or
cracks
appearing
between
layers,
especially
on tall
prints.

1. **Nozzle Temp:** Increase nozzle temperature by 5-10°C for better layer bonding.
2. **Cooling Fan:** Decrease your cooling fan speed (or turn it off for ABS).
3. **Under-extrusion:** Check for a clog or calibrate E-steps.

Under-extrusion

Thin,
weak, or
missing
layers.

1. **Clogged Nozzle:** Perform a "cold pull" or use a nozzle cleaning needle.
2. **E-Steps:** Calibrate your extruder's E-steps.
3. **Tangled Spool:** Check that your filament spool can unroll freely.

**The print
looks and
feels
snappy**

[Page 6: Final Thoughts & Call to Action]

Practice Makes Perfect!

3D printing is a journey of constant learning and experimentation. Don't be discouraged if a print fails! Every failure is a lesson that makes you a better printer. Use this guide as a starting point, but don't be afraid to tweak and tune your settings to find what works best for your specific printer and filament.

The Team at What2Print Want More?

For in-depth filament reviews, project ideas, and advanced tutorials, visit us at:

what2print.com