3DWB
WORKBENCH

3D PRINTING HACK PACK

8 Slicer Tricks. Verified. Ready to Print.

8	2,111	Verified
Hacks	Saves on Hack #1	All Settings

TOPICS:

Scarf Joint Seam · Thread Tolerances · Elephant Foot · Ironing · Layer Strength · Fuzzy Skin · Cold Pull · Anti-Warping

3DWB
WORKBENCH

ABOUT THE CREATOR

Juri Pranjic

@3dworkbench

Juri is a 3D designer based in Switzerland. Online he is known as @3dworkbench, a community of over 85,000 makers built around functional 3D printing.

His work focuses on objects that solve real problems: parts that fit precisely, hold up over time, and look intentional. Kitchen organizers, cable management systems, custom details for his daughter's toy wagon.

Juri runs two brands alongside a full-time job. 3dworkbench covers game props, replicas and sci-fi models. FORMAstudios focuses on minimalist home decor. Everything he shares has been tested under real constraints.

The hacks in this guide come from his most-saved posts across 34 analysed uploads. Save counts do not lie: people bookmark things they actually plan to use.

85K+	1.409	34	2.111
Followers	Posts	Posts analysed	Top save count

CONTENTS

8 Hacks.

01 **Eliminate the Z-Seam with Scarf Joint** — 2,111 Saves

02 **Thread Tolerances** — 580 Saves

03 **Elephant Foot Fix** — 445 Saves

04 **Ironing for Smooth Top Surfaces** — 390 Saves

05 **Layer Orientation = Strength** — 620 Saves

06 **Fuzzy Skin for Texture & Grip** — 355 Saves

07 **Cold Pull — Clean Your Nozzle** — 290 Saves

08 **Stop Warping For Good** — 480 Saves

Eliminate the Z-Seam with Scarf Joint

01

One setting in your filament profile. Set it once, forget it.

THE PROBLEM

Every layer has a start point and an end point. The slicer stacks them in the same spot by default, which leaves a visible line running up the side of your print. On round or organic shapes you cannot paint it away because there is no corner to hide it in.

THE FIX

Enable Scarf Joint Seam in your filament profile (Bambu Studio v1.9 or later). Instead of a hard stop and start, the slicer feathers the transition across a short distance. The seam blends into the wall. Set it once per filament and it applies to every print automatically.

SLICER SETTINGS

Scarf Seam Type	Contour / Contour and Hole
Smart Scarf Seam	Enabled
Scarf Joint Flow Ratio	0.8
Wipe on Loop	Disabled — conflicts with Scarf
Wipe on External Loops	Disabled

PRO TIP

Turn off Wipe on Loop and Wipe on External Loops when using Scarf. Running both at the same time makes the result worse, not better. Scarf is most effective on round and curved surfaces.

Thread Tolerances

Printed threads that actually fit. No guessing, no reprints.

02

THE PROBLEM

FDM printers lay down slightly more material than the path width. That is usually fine, but for threaded holes and internal geometry it means things come out too tight. The fix is straightforward once you know it.

THE FIX

Leave the screw at nominal size. It prints correctly as-is. Enlarge the hole or nut slightly to compensate for the extra material. Start with +0.15 mm on the female part, print a quick test, and adjust from there. Once you find your number it works on every print.

SLICER SETTINGS

XY Comp. (female / hole)	+0.1 to +0.2 mm — calibrate per printer
Screw (male)	Nominal size, no compensation
Thread Profile	ISO 60° — most printable
Min Thread Pitch	1.0 mm for reliability
Screw Infill	40%+ triangles for strength

PRO TIP

Your offset depends on your printer, nozzle size and filament brand. A short M5 bolt and matching nut takes about 5 minutes to print. Find your number, write it down somewhere you will see it. Threads are one of those things you calibrate once and never think about again.

Elephant Foot Fix

03

First layers that are accurate, not squished.

THE PROBLEM

The first layer gets pressed harder into the bed than the rest, which makes it spread slightly wider. On most parts this is harmless. On anything that needs to fit precisely with another part, that small flare at the base causes problems.

THE FIX

Start with your Z-offset and bed leveling. Too close to the bed is almost always the root cause. If things still look off after that, enable Elephant Foot Compensation in the slicer. The CAD approach also works well: a 0.3 mm chamfer on the bottom edge of your model compensates optically and does not need any slicer adjustment.

SLICER SETTINGS

Elephant Foot Comp.	0.1–0.2 mm (Bambu / PrusaSlicer)
Initial Layer Horiz. Exp.	-0.1 mm (Cura)
First Layer Height	0.2 mm
First Layer Speed	max. 30 mm/s
Bed Temp	Keep consistent throughout

PRO TIP

A small chamfer on the bottom edge of a model is good practice regardless of elephant foot. It reduces stress concentration, helps parts sit flat on surfaces, and makes the base look clean. Worth adding to your default CAD template.

Ironing for Smooth Top Surfaces

Top surfaces with no ridges. No sanding required.

04

THE PROBLEM

A standard top layer is laid down in rows. Each row sits slightly higher than the last, leaving small ridges across the surface. Under a light or in photos these ridges are clearly visible.

THE FIX

Ironing adds a second pass after the top layer is finished. The nozzle moves slowly across the surface with almost no material coming out. The heat from the nozzle softens the ridges and flattens them. The result is noticeably smoother than a standard top layer.

SLICER SETTINGS

Ironing Type	Topmost Surface
Ironing Flow	Use slicer default. Start at 15%, increase in 5% steps until surface looks right
Ironing Speed	Start: 40 mm/s. Reduce speed before increasing flow
Line Width	0.1 mm — finer is smoother

PRO TIP

Ironing works on flat horizontal surfaces. On angled or curved top faces the result is less predictable and sometimes makes things worse. Use the topmost surface setting so it only applies where it actually helps.

Layer Orientation = Strength

05

The same part, printed two ways, can have completely different strength.

THE PROBLEM

FDM parts are not equally strong in all directions. The bond between layers is weaker than the material within a layer. A part that handles horizontal stress well can snap easily under vertical load if the layers run the wrong way.

THE FIX

Before you hit slice, think about where the stress will be. If a part gets pulled in one direction, you want the layer lines running across that direction, not parallel to it. A hook printed lying flat breaks easily. The same hook printed upright is much stronger.

SLICER SETTINGS

Infill Pattern	Gyroid / 3D Honeycomb
Infill	40%+ for load-bearing parts
Walls / Perimeter	4–6 for structural parts
Material	PETG > PLA for impact resistance

PRO TIP

Orientation matters more than infill percentage for most functional parts. It takes about 30 seconds to think through before slicing and can make a significant difference in how long something lasts.

Fuzzy Skin for Texture & Grip

06

A textured surface finish built directly into your print.

THE PROBLEM

A standard outer wall is smooth and slightly shiny. That works fine for many parts but for anything you hold or handle regularly it can feel slippery and cheap.

THE FIX

Fuzzy Skin moves the nozzle in small random deviations as it prints the outer wall. The result is a matte, slightly rough surface that looks and feels intentional. Point Distance controls how often the deviation happens. A smaller value means more frequent changes, producing a finer texture. A larger value creates a coarser, more pronounced surface. Skin Thickness controls how far the nozzle moves from the original path. A lower value gives a subtle effect you mostly feel rather than see. A higher value creates visible texture that reads clearly across the room.

SLICER SETTINGS

Fuzzy Skin	Enabled — outer walls only
Skin Thickness	0.3 mm — subtle and premium
Point Distance	0.8 mm — finer texture
Apply to	Outer contour only, not top/bottom

PRO TIP

Start with the default values and print a small test block before applying it to a full model. The interaction between point distance and skin thickness is easier to understand once you can hold both versions in your hand.

Cold Pull — Clean Your Nozzle

A simple way to clean your nozzle without taking anything apart.

07

THE PROBLEM

Nozzles do not always clog completely. More often there is a partial blockage from burnt filament or leftover material from a previous colour or type. The print keeps going but flow is inconsistent and quality drops.

THE FIX

Heat the nozzle to printing temperature and push filament through by hand to clear any loose material. Then let it cool to the pull temperature for your filament type. Once it reaches that temperature, pull the filament out in one steady motion. The tip will have picked up debris from inside the nozzle. Repeat until the pulled piece comes out clean with no dark spots.

SLICER SETTINGS

Pull Temp — PLA	90°C
Pull Temp — PETG	125–135°C
Pull Temp — ABS	140–160°C
Best cleaning filament	Nylon — grips residue better than PLA

PRO TIP

A cold pull is worth doing when you switch between different filament types, particularly when going from a high-temperature material to a lower one. Same-material colour swaps usually do not need it. The whole process takes about five minutes.

Stop Warping For Good

Why prints warp and what actually fixes it.

08

THE PROBLEM

As a print cools, the material contracts. The outside cools faster than the inside, which creates internal stress that pulls the corners of the part upward. On large flat prints this shows up as lifted edges or a part that pops off the bed partway through.

THE FIX

Gleichzeitig auf mehreren Ebenen ansetzen: Bett-Haftung + Gehäusetemperatur + Teildesign + konstante Betttemperatur durch den gesamten Druck. Nur eine Massnahme allein reicht selten.

SLICER SETTINGS

Brim Width	8–12 mm for large flat parts
Bed Temp	Consistent — do not drop after layer 1
Enclosure	Closed — thermal stability is everything
Draft Shield	Enable for tall thin parts

PRO TIP

Sharp corners at the base of a print concentrate thermal stress in one spot, which is often where lifting starts. A small fillet of 1 to 2 mm on the bottom edges distributes that stress more evenly. It is a small change in CAD that makes a noticeable difference on large prints.

You've got the hacks. Now ship the prints.

WHAT'S NEXT

PETG Crystal Clear — Free Guide

Comment CLEAR on my next post — I will send you the complete PETG transparency guide straight to your DMs via ManyChat.

@3dworkbench

3DWB
WORKBENCH