
3D Printing:
Safety, Fundamentals, and Troubleshooting
Safety
Printer Fundamentals
Troubleshooting
Resources

Spot the Hazard Before You Start



What do you notice?

1. What stands out?
2. What might be hazardous?
3. What could change the risk?

What could change the risk?

- Where the cord is routed
- Whether people walk through the area
- The condition of the cord
- Whether liquids or sharp objects are nearby
- Whether the workspace is cluttered

Keep cords clear, dry, and out of walkways. Clean up clutter and report hazards.

Spot the Hazard Before You Start



What do you notice?

1. What stands out?
2. What might be hazardous?
3. What can't you tell from the photo?
4. Would you touch it? Why or why not?

Do not touch exposed wiring.
Stop, report it, and get help.

Create the Right Environment for Safe 3D Printing

Good ventilation, approved materials, and added filtration help keep our lab safer.

Keep the room well ventilated.



- 3D printing can release ultrafine particles and VOCs into the air.
- Keep the room well ventilated with building ventilation, air exchange, or exhaust systems.
- Good room ventilation helps keep the air cleaner for everyone in the lab.

Use PLA or PETG only.

APPROVED FOR OUR LABS

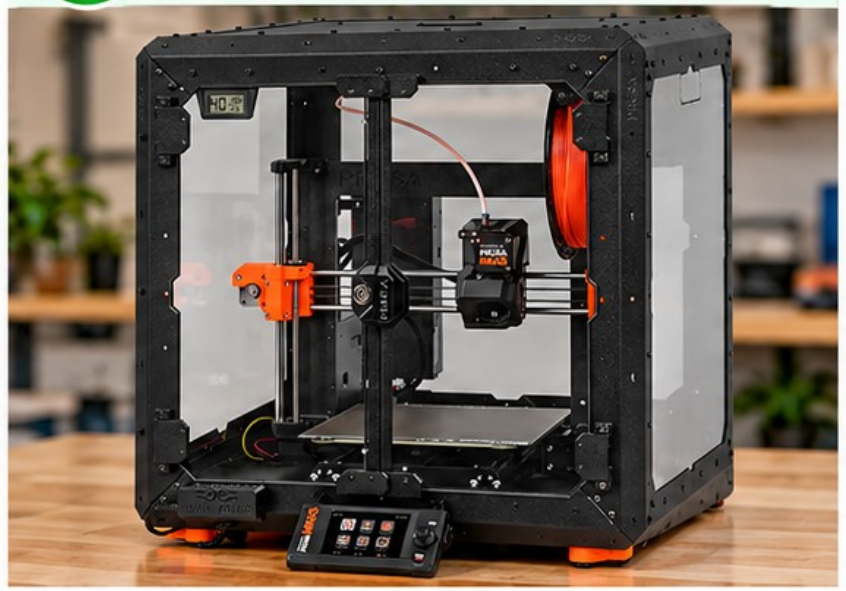


NOT ALLOWED IN OUR LABS



Do not use ABS or other high-temperature filaments in our lab.

Advanced filtration helps.



- Enclosed printers with filtration systems (e.g., carbon and HEPA filters) can help reduce ultrafine particles and VOCs at or near the printer.
- Advanced filtration is a great added layer of protection.
- Filtration does not replace good room ventilation. Always keep the room well ventilated.

GOOD VENTILATION
Helps reduce buildup of ultrafine particles and VOCs.

PLA / PETG ONLY
Approved materials for our labs.

NO ABS / HIGH-TEMP FILAMENTS
Not allowed in our labs.

ADVANCED FILTRATION HELPS
A valuable extra layer of protection.

Lockout/Tagout (LOTO) Before Servicing the 3D Printer

1 Turn the 3D printer off and unplug it.



2 Begin working on the 3D printer.



3 Another person sees the 3D printer is off and might plug it back in.



4 The plug is locked out with a tag so the printer cannot be unexpectedly energized!



Lockout/Tagout (LOTO) prevents unexpected energization.

Even if the machine is off, someone could plug it back in. Use a lockout device and tag whenever you service or maintain equipment.

PREVENTS PLUG IN
Stops unexpected power.

KEEPS PEOPLE SAFE
Reduces the risk of injury.

SUPPORTS A SAFER LAB
Builds good habits and clear communication.

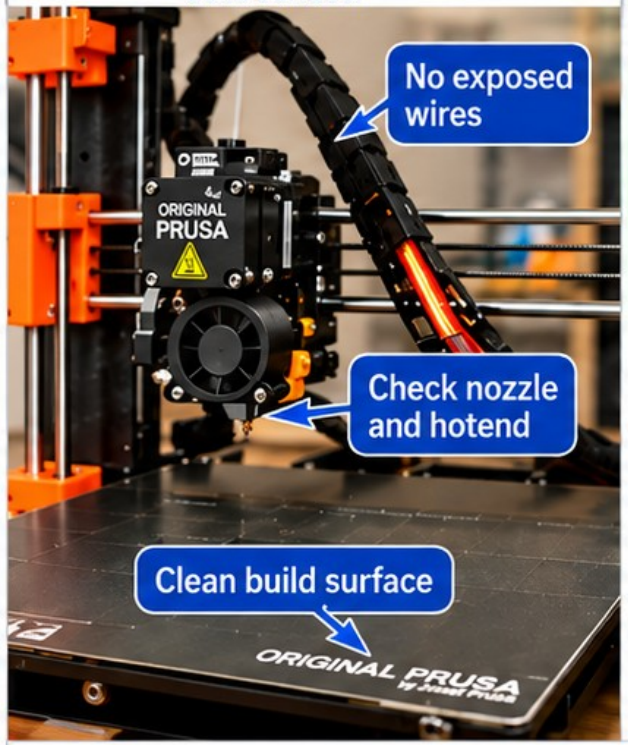


Start the Print Right

Inspect • Watch the First Layer • Monitor Running Prints • Operate Safely

1 Inspect before you print.

Check for damage, loose parts, exposed wires, and a clean build surface.



2 Use clean, undamaged filament.

Keep filament dry and free of contamination.



3 Watch the first layer directly.

A student or teacher must watch the printer in person during the first layer to make sure the print starts correctly.

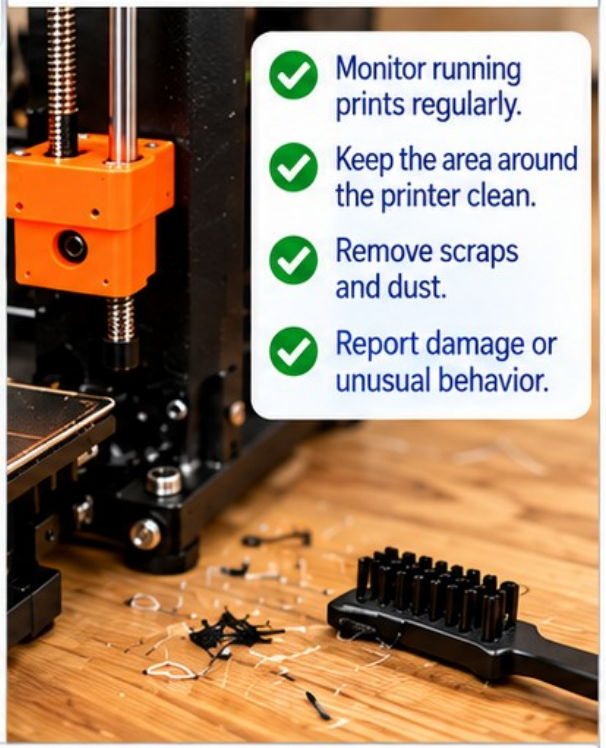


A good first layer is the difference between a successful print and a failed print.

- Watch the first layer in person.
- Check that the filament is sticking properly.
- Stay nearby for the first several layers.
- Stop the print if the first layer is failing.

4 Monitor running prints and keep the printer clean.

Once the print is going well, continue to check on it regularly.



- Monitor running prints regularly.
- Keep the area around the printer clean.
- Remove scraps and dust.
- Report damage or unusual behavior.

If a print is failing or turns into a "spaghetti" mess, stop the print.

If you notice unusual noises, smoke, burning smells, or exposed wiring: turn off the printer, unplug it, and notify the instructor.

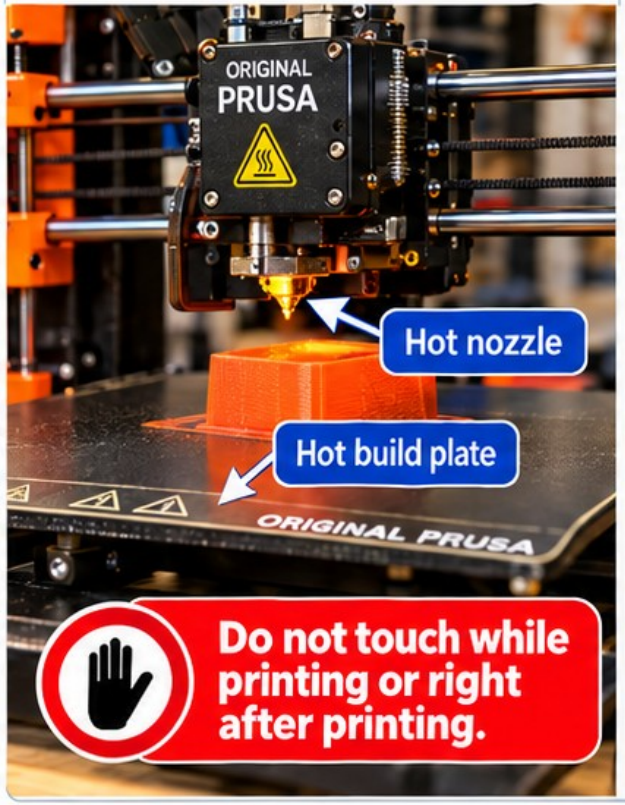
Running prints should always be monitored.
Watch the first layer directly.
After that, continue checking on the print regularly.

Know the Hazards Before You Touch

Hot Surfaces • Moving Parts • Safe Print Removal • Stop for Hazards

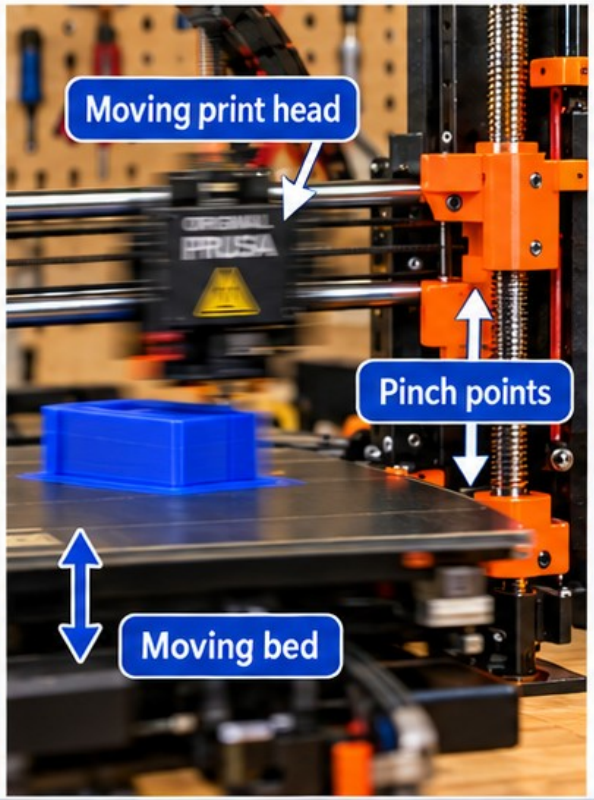
1 Hot nozzle and heated bed.

The nozzle and build plate get very hot during printing.



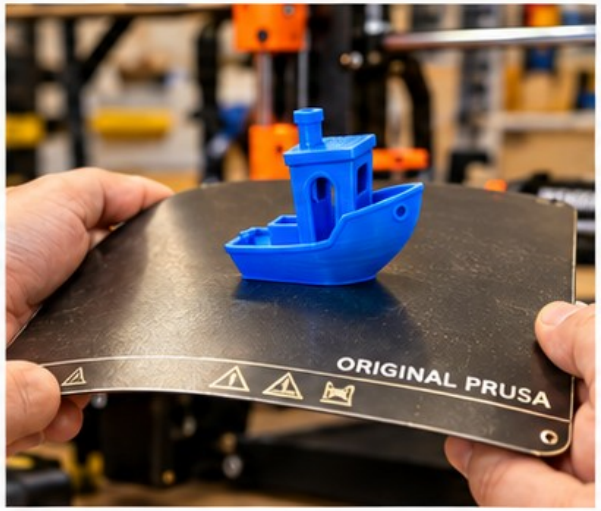
2 Keep clear of moving parts.

Keep hands, hair, sleeves, and tools away from moving parts.



3 Remove prints safely.

Let the plate cool, remove the sheet, and flex it to release the print.



- Wait until the printer and sheet are cool.
- Use the flexible sheet to release parts.
- Do not force parts off the bed with sharp tools.

4 Stop if a print fails.

Do not pull filament off a hot nozzle or heater block.



- Stop the print.
- If filament is building up around the hotend, get the instructor.
- Do not touch damaged or overheated parts.

Stopping a print does not mean it is safe to touch.
The nozzle and bed may stay hot for several minutes.

Wait for hot parts to cool before touching or removing prints.

If you see smoke, burning smells, exposed wiring, or a damaged hotend, turn off the printer, unplug it, and notify the instructor.

From Safety to Success

Now that we know how to stay safe, let's learn what the printer is, how it works, and how to set it up for a successful print.

Quick Safety Recap

Safe environment
Keep the area clear and free of hazards.

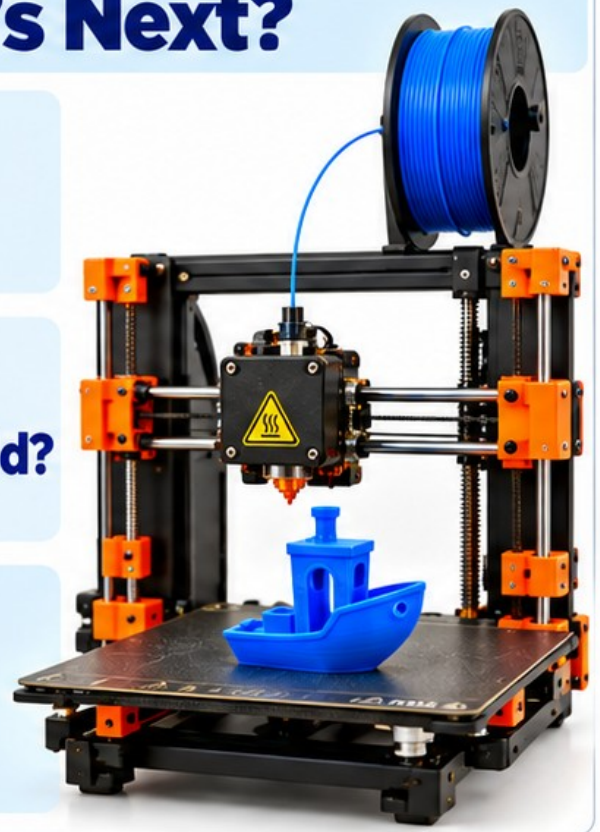
Safe operation
Keep hands, hair, and tools away from hot parts.

Monitor the printer
Watch the print and look for problems.

Next: understand the machine

What's Next?

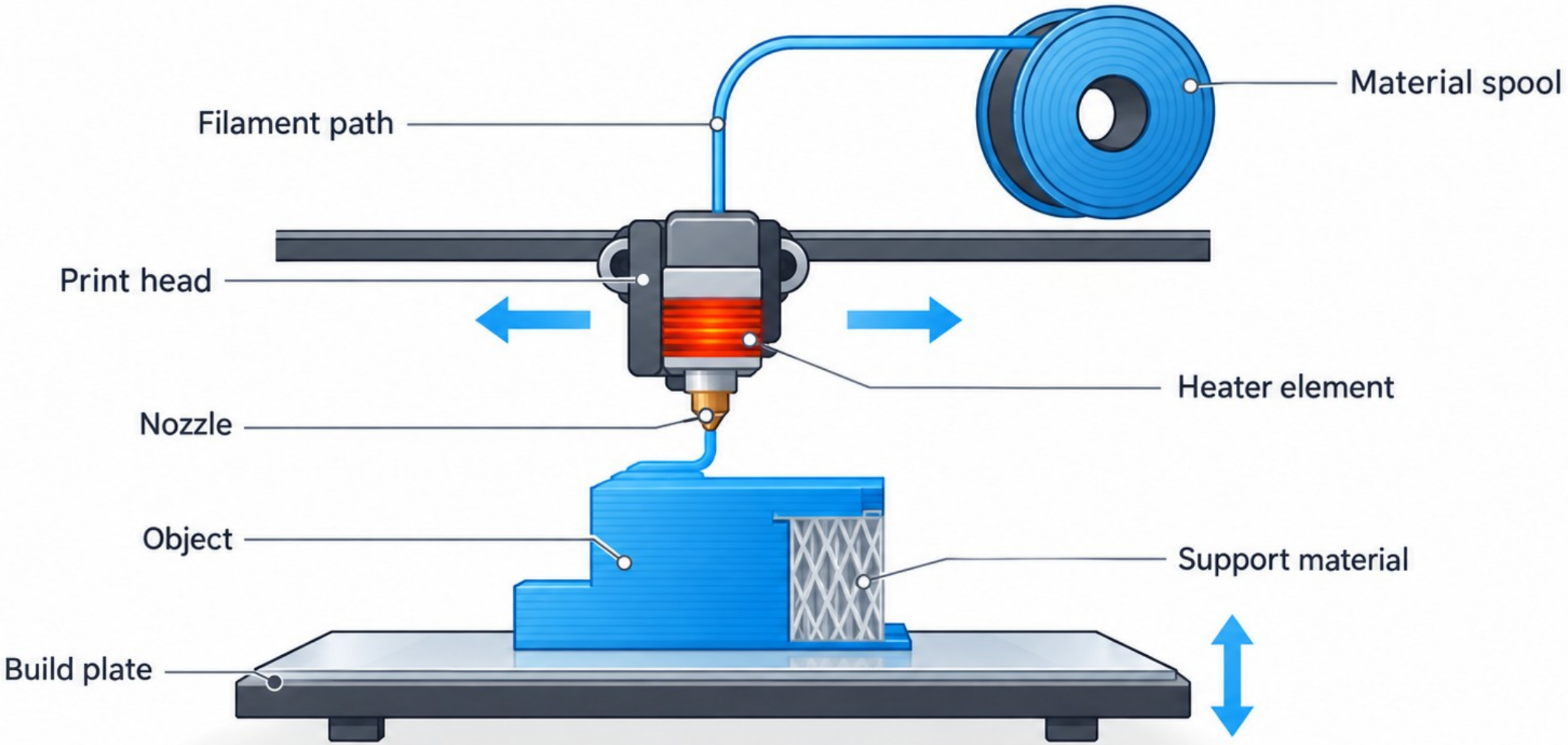
1. What is a 3D printer?
2. How does it move and build?
3. How do we set it up for success?



Safety comes first. Understanding the printer helps us print successfully.

What Is a 3D Printer?

A machine that melts plastic and builds an object one layer at a time.



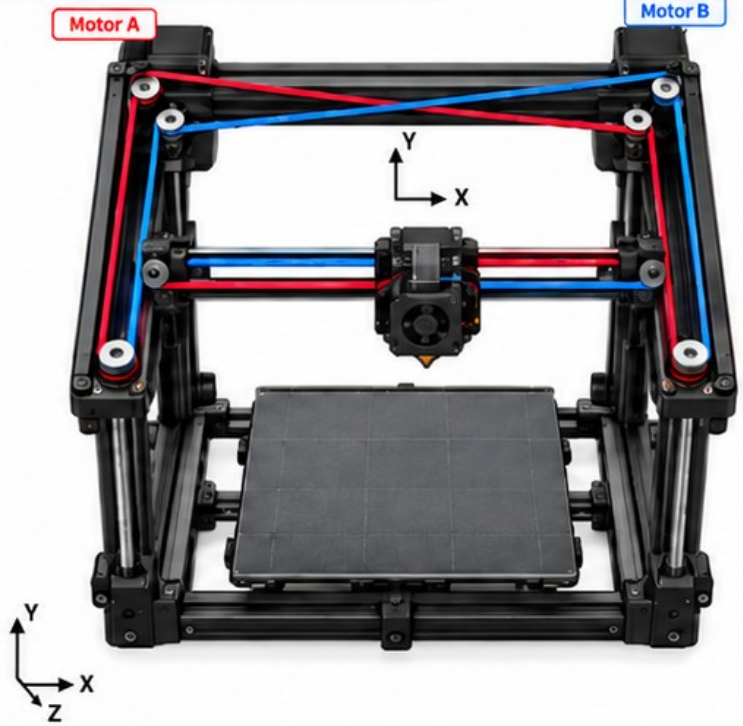
How Does a 3D Printer Move?

3D printer kinematics

Core One = CoreXY

CoreXY: named for coordinated X-Y belt motion.

CoreXY Kinematic Diagram



Motor A (Belt A)
Drives the red belt loop.



Motor B (Belt B)
Drives the blue belt loop.



X Movement (ΔX)
Both motors rotate in the same direction.



Y Movement (ΔY)
Motors rotate in opposite directions.

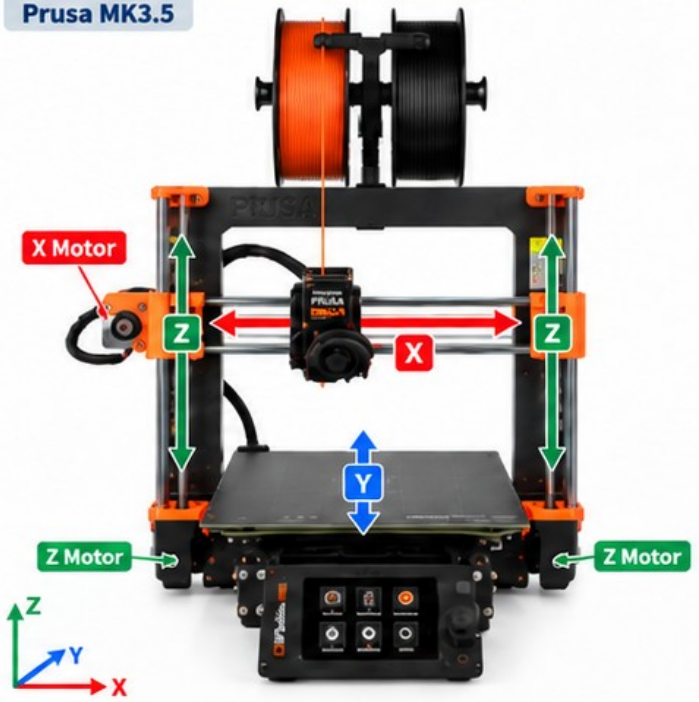


MK3.5 = Cartesian

Cartesian: named for the X-Y-Z coordinate system.

Cartesian Kinematic Diagram

Prusa MK3.5



X Motor
Moves the print head left and right along the X-axis using the belt.



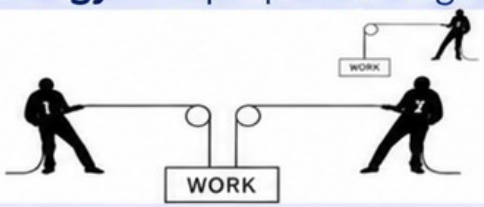
Y Motor
Moves the heat bed forward and backward along the Y-axis.



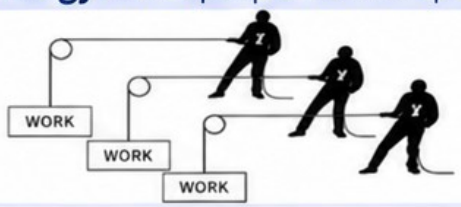
Z Motor
Raises and lowers the X-gantry together along the Z-axis.



CoreXY Analogy: Two people work together



Cartesian Analogy: Three people work independently



Set the Printer Up for Success

Before we print, we focus on the baseline processes that have the biggest impact on success.

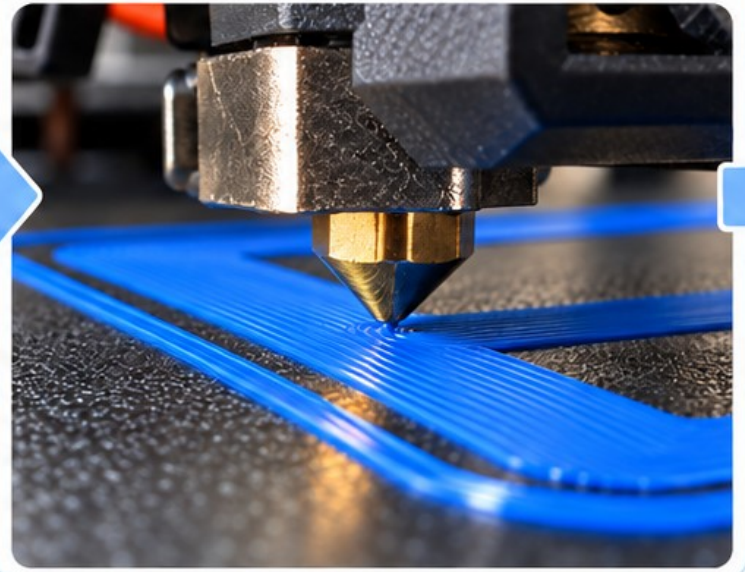
1 Maintain the print bed

A clean, well-maintained print surface gives the print a strong start.



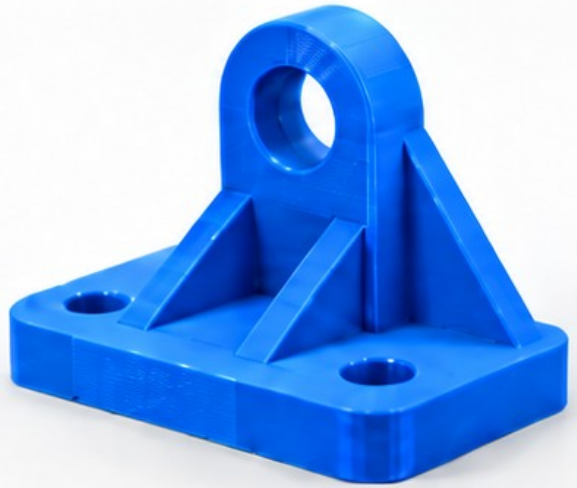
2 First-layer adhesion

A strong first layer is one of the best predictors of a successful print.



3 Strong design choices

Thoughtful features, realistic overhangs, and print-friendly design choices lead to better prints.



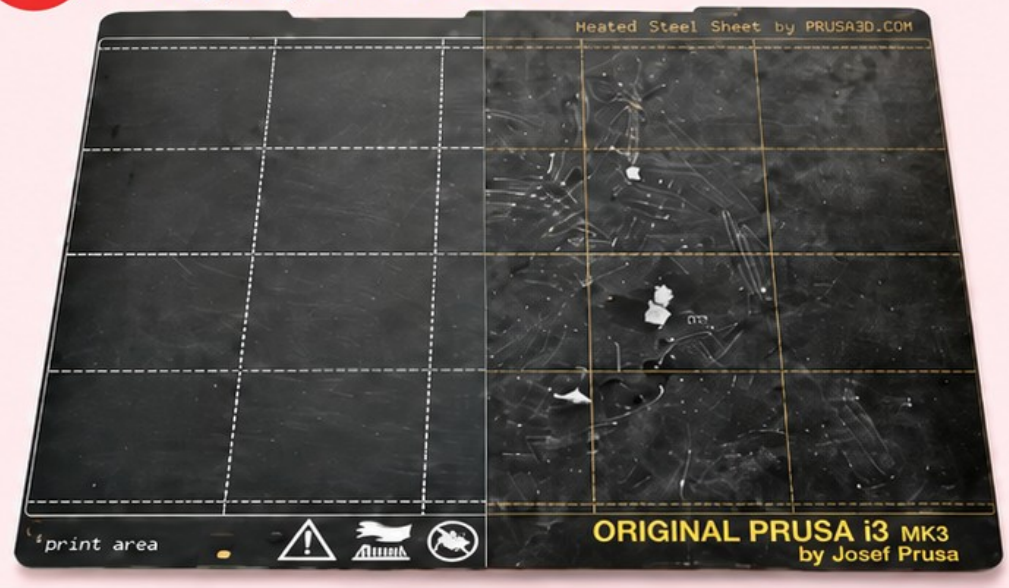
Clean sheet. Strong first layer. Strong design. Better prints.

Build Sheet Readiness

Clean, dry, and free of oils for reliable adhesion

Not Ready to Print

Dirty, oily, or wet surface



Ready to Print

Clean, dry, and fingerprint-free



1 Remove the Sheet



Carefully remove the build sheet using the tabs or sides.

2 Wash with Dish Soap



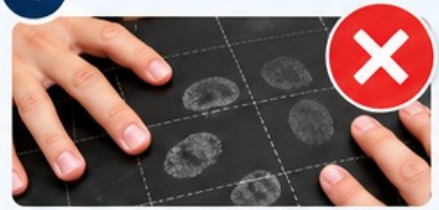
Use dish soap and warm water. Scrub thoroughly in small circles, especially textured plates.

3 Rinse and Dry



Rinse completely and dry with a clean paper towel or let it air dry.

4 Avoid Touching



Handle by the tabs or sides. Avoid touching the print area to prevent oils from your fingers.

5 Reinstall and Print



Place the sheet back on the bed. It's now ready for reliable first-layer adhesion.

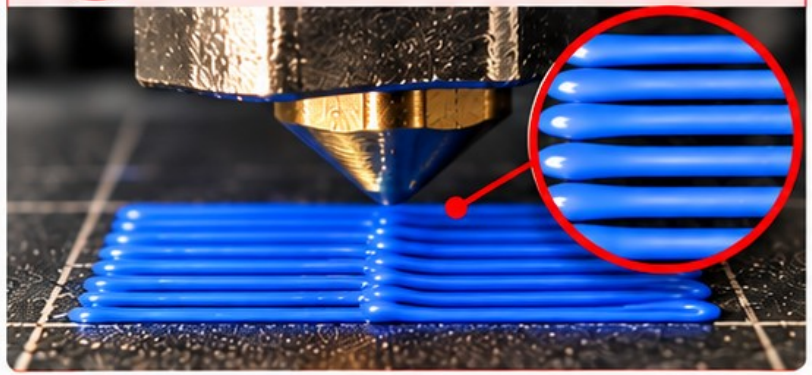
Tip: Isopropyl alcohol does not do a great job of cleaning oils.
For best results, always use dish soap and water, especially for textured plates.

First-Layer Calibration

After your build sheet is clean and dry, dial in the correct nozzle height for a reliable first layer.

Too High

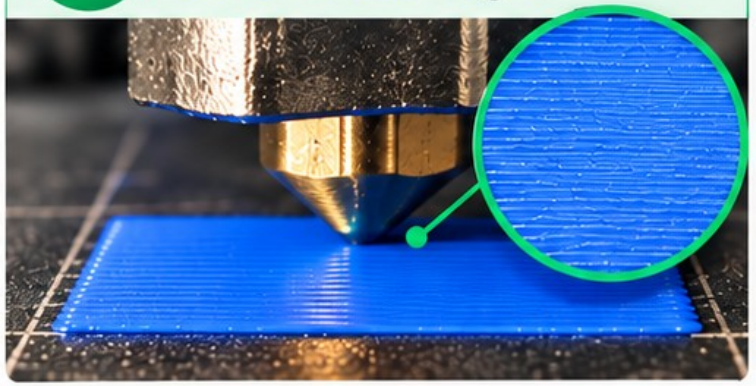
Nozzle is too far from the bed.



- Lines are rounded and don't fully touch.
- Visible gaps between lines.
- Weak adhesion — the print can be easily scraped off with a fingernail.

Just Right

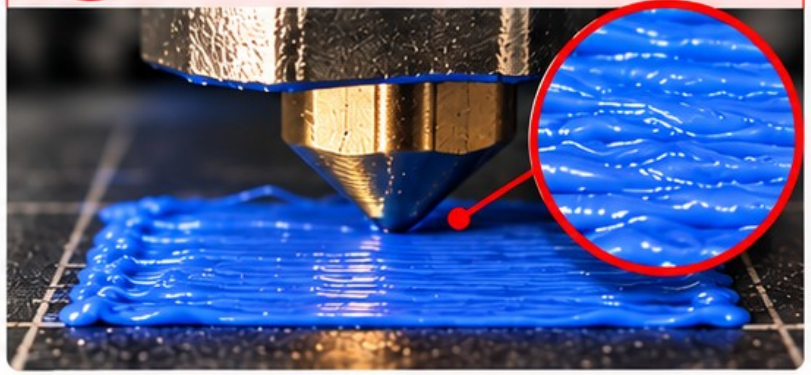
Correct nozzle height.



- Lines are evenly squished and fully connected.
- Smooth, consistent surface.
- Good adhesion — you can't easily scrape it off with a fingernail.
- No high or low spots.

Too Low

Nozzle is too close to the bed.

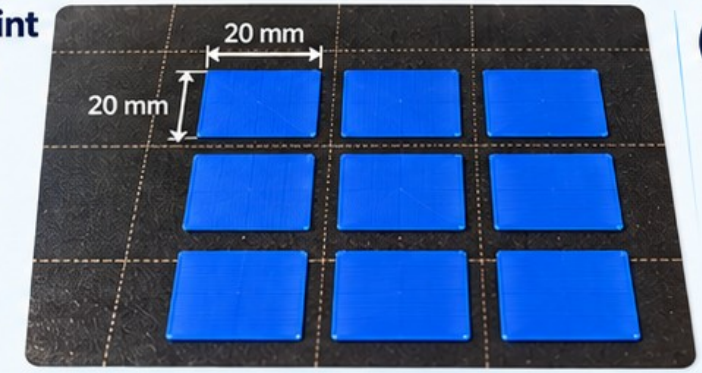


- Lines are over-squished, with extra plastic pushed toward the edges.
- Surface looks rough or glossy and can appear slightly translucent from over-compression.
- You may see ridges at the edges and a thinner, more transparent look in the middle.

1 Run a First-Layer Test Print

I like to make 9, 20 mm squares as a single layer with a 0.25 mm or greater first layer height. Thicker first layers are less sensitive and easier to maintain, especially on larger printers where first layer consistency can be a challenge.

You can model this in your CAD software or copy and paste within the slicer.



2 Inspect Across the Bed

Check multiple locations—especially the edges and the center—to make sure the height is consistent.

3 Adjust in Small Steps

Make small Z-offset or Live Z adjustments. Then rerun the test—live changes do not show the full result immediately because nozzle pressure needs time to stabilize.

4 Recheck as Needed

Once you think the height looks good, run the same test again and inspect it. Rechecking matters because extrusion pressure stabilizes over time.

DESIGN RULES FOR FDM 3D PRINTING

Assumes a 0.45 mm extrusion width with 0.2 mm layer height.

Values are starting points. Always test on your printer and material.

Supported Walls

Walls attached on at least two sides are stronger and can be thinner.



~0.9 mm
(2 perimeters)

Unsupported Walls

Freestanding walls should be thicker for stiffness and print reliability.



≥ 1.35 mm
(3 perimeters)

Overhangs / Support Angle

Use 45° from vertical as a conservative starting point. Steeper overhangs may need support depending on cooling and material.



≤ 45°

Bridges

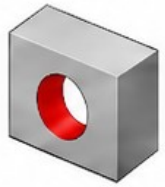
Short horizontal spans can often print without support. Longer bridges depend on cooling, speed, and material.



~10–20 mm
(starting range)

Holes

Printed holes usually come out undersized. Model them slightly larger and test critical fits.



Add +0.2–0.4 mm to Ø

Clearance / Moving Parts

Leave clearance between moving parts so they do not fuse together.



~0.3 mm per side

Escape Holes / Support Removal

For enclosed cavities, add openings so trapped support can be removed.



≥ 5 mm

Embossed & Engraved Details

Raised or recessed details need enough stroke width and depth to remain legible.



≥ 0.6 mm stroke, ≥ 0.4 mm depth

Minimum Features

Ribs, posts, and small protrusions should be large enough to print consistently.



≥ 1–2 mm

Pin / Post Diameter

Vertical pins and posts need enough diameter for strength and printability.



≥ 3 mm

Text / Details

Text on flat surfaces should be tall enough and thick enough to read clearly.



≥ 1 mm text height

Tolerance / Accuracy

FDM printers have dimensional variation. Use test prints for critical fits and tolerances.



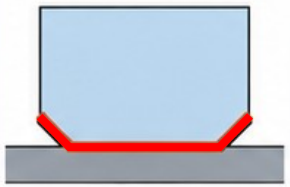
Typical: ±0.1–0.3 mm

Fillets (Inside Corners)



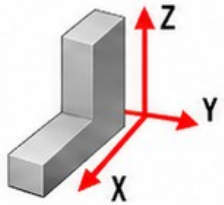
Add fillets (≥ 1 mm radius) to reduce stress concentration and improve strength.

Chamfers (Outside Corners)



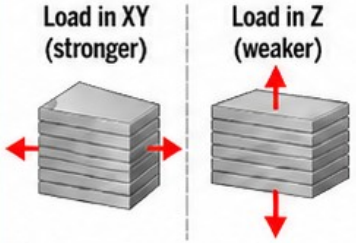
A 0.3–0.5 mm chamfer at 45° is a reasonable starting point for elephant's-foot compensation.

Orientation



Orient parts to minimize supports, improve bed contact, and align strength with expected loads.

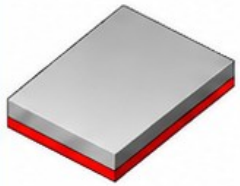
Layer Strength



Stronger:
Loads along the layer lines (XY) are strongest.

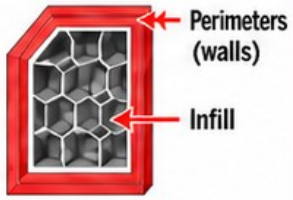
Weaker:
Loads that separate the layers (Z) are weaker, especially in tension.

Large Flat Surfaces



Large, unsupported flat areas can warp. Use brims, chamfers, or ribs to improve bed adhesion.

Perimeters / Strength



More perimeters (outer walls) increase strength and stiffness more than higher infill density, especially around outside walls and holes.

Tall, Narrow Columns



Avoid very tall, slender columns.

Very tall, slender features can wobble, flex, or fail during printing. Reduce height, increase thickness, or add bracing.

Threaded Features



Print threads ≥ 8 mm major diameter.

Larger printed threads work best. For smaller or high-strength threads, use inserts or post-processing.

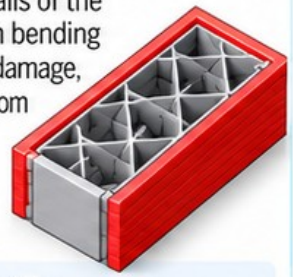
PERIMETERS vs. INFILL IN FDM PRINTING

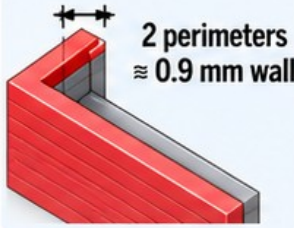
Assumes a 0.45 mm extrusion width with 0.2 mm layer height.

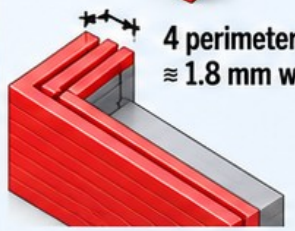
Strength depends on geometry, print orientation, material, and load direction. Values are starting points.

1. WHAT ARE PERIMETERS?

Perimeters (shells) are the outer walls of the print. They carry most of the load in bending and resist impact, wear, and surface damage, because the outer material is farthest from the neutral axis.

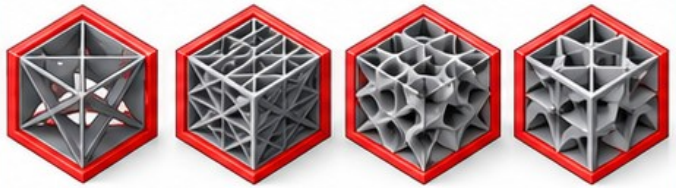






2. WHAT IS INFILL?

Infill supports the inside of the part. It adds strength, helps support top layers, but also adds weight, material, and print time.



Grid
Fast and simple. Good for general use.

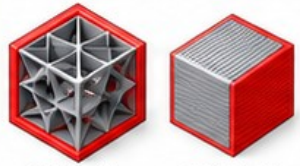
Rectilinear
Alternating lines. Stiff in X/Y directions.

Gyroid
Continuous 3D structure. Strong in multiple directions.

Cubic
Very strong, but uses more material and takes longer.

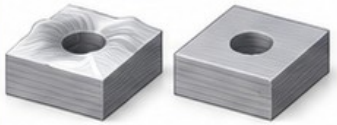
3. COMMON STUDENT MYTHS (AND THE FACTS)

"100% infill always makes a part strongest."



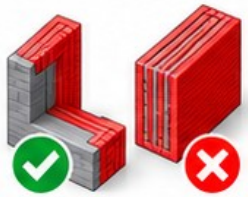
More infill can increase strength in some cases, but perimeters often increase bending strength more efficiently. 100% infill makes parts heavier, uses more material, and takes longer to print.

"Infill doesn't matter as long as I have perimeters."



Infill supports the top and bottom layers. Too little infill can cause sagging, poor surface quality, and weak internal structure, even with many perimeters.

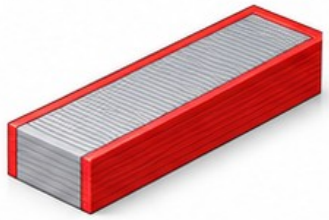
"More perimeters are always better."



After a certain point, additional perimeters give diminishing returns, add weight and print time, and can cause fit issues in tight spaces. Use enough for your load and geometry.

4. PERIMETERS + INFILL: EXAMPLE

For many real parts, stronger outer walls with moderate infill give a better strength-to-weight balance than 100% infill.



2 perimeters + 100% infill

- Heavier
- Longer print time
- Can be more brittle



4 perimeters + 20% infill

- Higher strength-to-weight
- Less material and faster
- Good for most functional parts

5. HOW EACH SETTING AFFECTS THE PRINT

Increasing perimeters



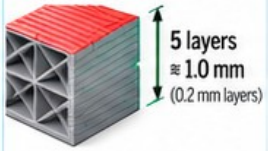
- Thicker outer walls
- Higher bending strength and stiffness
- Better wear resistance
- Improved hole and edge strength

Increasing infill (density or pattern)



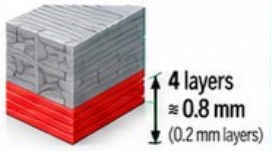
- Stronger internal structure
- Supports top layers
- Increases weight, material, and print time

More top solid layers



- Stronger, smoother top surfaces
- Prevents sagging over infill
- Important for bridging and enclosed cavities

More bottom solid layers



- Stronger bottom surface
- Better dimensional accuracy on the first layer
- Works with a small bottom chamfer to reduce elephant's foot

6. GOOD STARTING POINTS

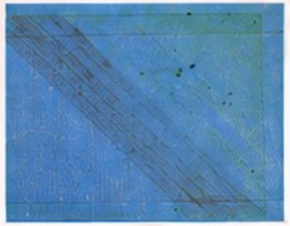
 **Visual models (low load)**	2 perimeters 10–15% infill 3–5 top / 3–5 bottom layers
 **General functional parts**	3–4 perimeters 15–25% infill 4–6 top / 4–6 bottom layers
 **Higher-load parts**	4–6 perimeters 25–40% infill 5–8 top / 5–8 bottom layers
 **Only use 100% infill when a specific need justifies it.**	e.g., small solid parts, compression loads, threads (use printed inserts if possible).

FDM 3D PRINTING: COMMON PROBLEMS, CAUSES, AND FIXES

Look at the print. Identify the problem.
Check the most likely causes. Adjust one thing at a time.

FIRST LAYER & ADHESION PROBLEMS

The print doesn't start correctly or doesn't stay on the bed.



Not extruding at start of print

Most likely causes:

- Nozzle is too close to the bed (first layer starts too low)
- Inccrrect start G-code
- Extruder not pushing



Not sticking to the bed

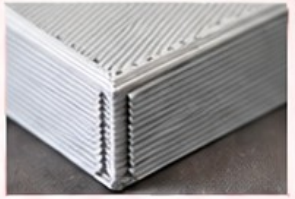
Most likely causes:

- Z-offset too high
- Bed temperature too low
- Poor adhesion settings



Warping

- Bed temperature too low
- Poor adhesion
- High ambient drafts
- No enclosure (for ABS/ASA)



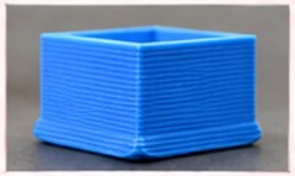
Gaps in floor corners

- Z-offset too high
- Low bed temperature
- Poor bed adhesion
- Too much part cooling



Poor first layer (gaps)

- Z-offset too high/low
- Inconsistent extrusion
- Dirty build plate
- Cold bed temperature



Elephant's foot (bottom bulge)

- Nozzle too close to bed
- Bed temperatine too high
- Low bed adhesion
- Poor model adjustments

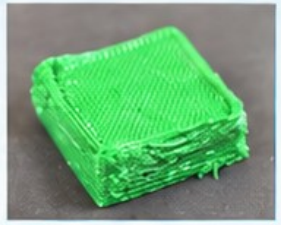
EXTRUSION & FLOW PROBLEMS

The printer isn't extruding the right amount of plastic.



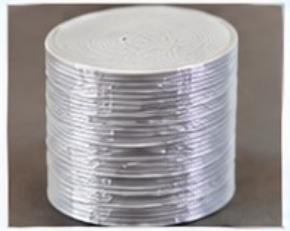
Under-extrusion

- Filament not feeding consistently
- Low nozzle temperature
- Partial clog
- Print speed too high



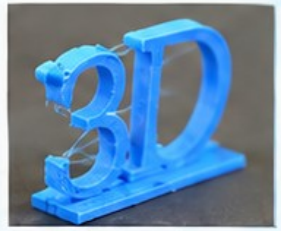
Over-extrusion

- Extruding too much plastic
- Flow/EM multipiler too high
- Nozzle temperature too high
- Print speed too slow



Inconsistent extrusion

- Filament diameter varies
- Extruder tension inconsistent
- Temperature too low/high
- Moisture in filament



Stringing or oozing

- Retraction too low
- Nozzle temperature too high
- Travel speed too low
- Moist filament



Gaps in top layers

- Not enough top layers
- Infill not supporting top layers
- Under-extrusion
- Print speed too fast / too cool



Layer separation (weak layer bonding)

- Nozzle temperature too low
- Poor part cooling
- Travel speed too low
- Moist filament

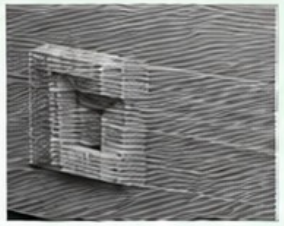
MECHANICAL & MOTION PROBLEMS

Layers don't line up or show artifacts from movement.



Layer shifting

- Loose belts or pulleys
- High print speed/acceleration
- Mechanical obstruction
- Unstable printer or surface



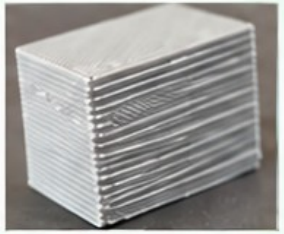
Vibrations and ringing

- High print speed/acceleration
- Loose belts or frame
- Heavy print head
- Resonance (sturdy surface)



Overheating (soft, deformed)

- Nozzle temperature too high
- Insufficient part cooling
- Printing too slowly
- Small features heat up



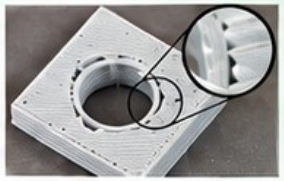
Lines on the side of print

- Z-axis lead screw debris
- Loose bolts or pulleys
- Vibration / wobble
- Inconsistent extrusion



Poor bridging

- Insufficient part cooling
- Printing too hot
- Bridge speed not tuned
- Bridge too long



Gaps between infill and walls

- Under-extrusion
- Infill overlap too low
- Inconsistent extrusion width
- Slicer settings (infill-to-wall gap)

DESIGN & GEOMETRY PROBLEMS

The model prints, but the result isn't as intended.



Small features not printed

- Feature smaller than nozzle
- Incorrect print orientation
- Insufficient cooling
- Slicer minimum feature size



Gaps in thin walls

- Wall thickness too thin
- Too few perimeters
- Under-extrusion
- Nozzle diameter too large for the feature



Poor surface above supports

- Insufficient support density
- Improper support settings
- Large layer height
- Inadequate interface layers



Dimensional accuracy issues

- Extrusion not calibrated
- Material shrinkage
- Incorrect flow rate
- Slicer tolerance/scale settings

KEY TROUBLESHOOTING PRINCIPLES

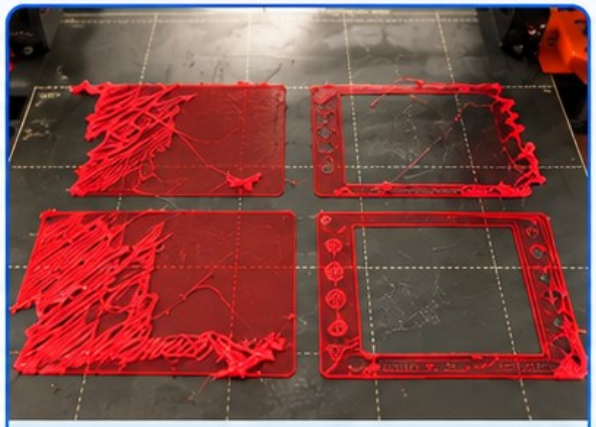
1. Identify the problem by how it looks.
2. Check the most likely causes.
3. Change one setting at a time.
4. Print a small test to verify the change.
5. If issues continue, check filament, nozzle, and mechanicals.

3D Printing Troubleshooting

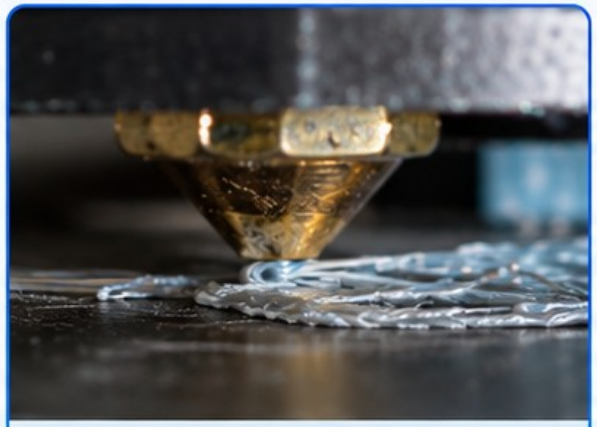
Look at the print evidence. What may have happened, and what would you check first?



Case 1



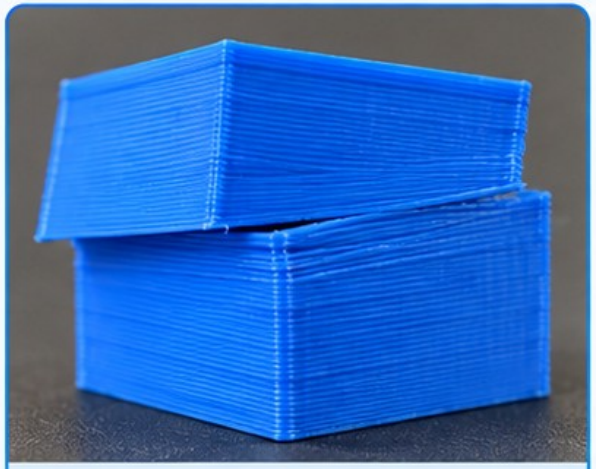
Case 2



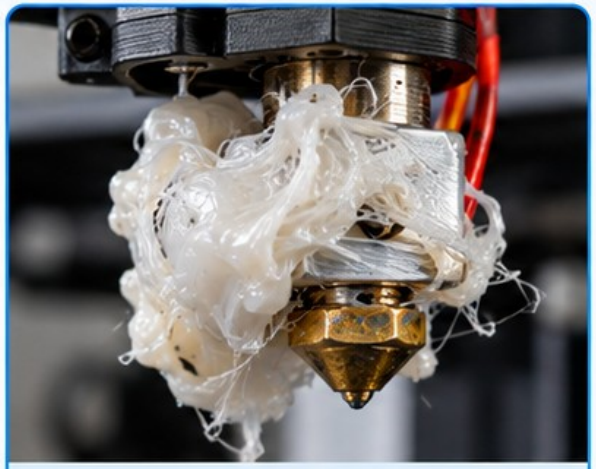
Case 3



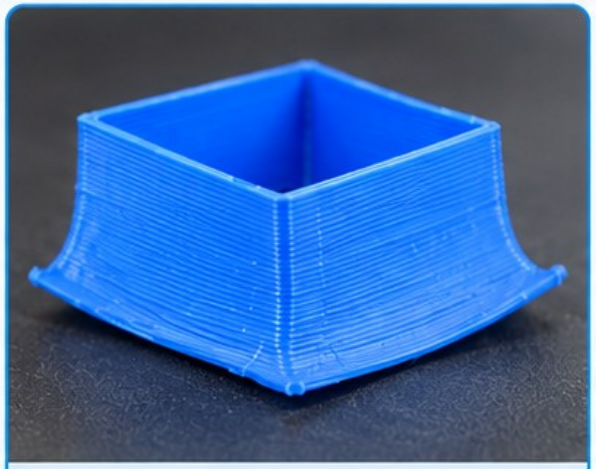
Case 4



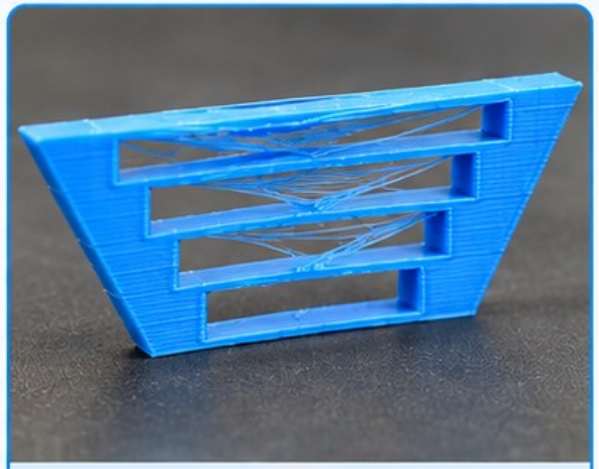
Case 5



Case 6



Case 7

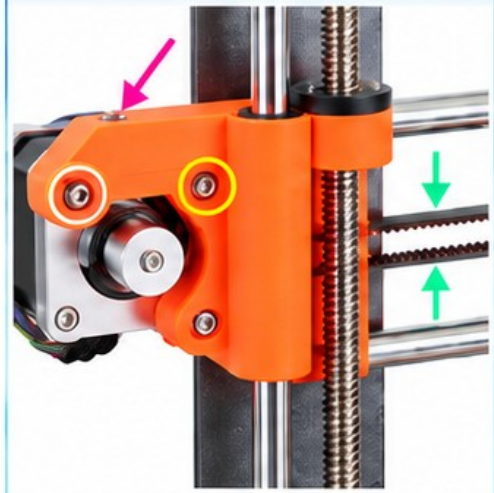


Case 8

FDM 3D PRINTER MECHANICAL MAINTENANCE & TROUBLESHOOTING

Common motion-system issues to inspect first on FDM 3D printers.

1 BELT TENSION + PULLEY SECURITY



- Loose belts can cause layer shifts, ringing, or inaccurate motion.
- Old belts may lose flexibility or need replacement.
- Check pulley grub screws and belt adjustment screws.
- Collisions or students bumping the printer can also lead to layer shifts.

2 CLEAN LINEAR RODS



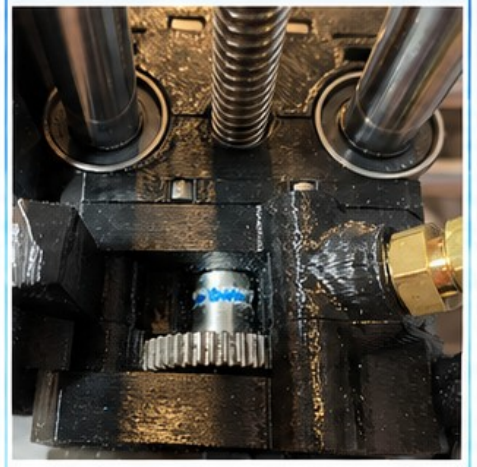
- Dirty or dry rods can cause rough motion, squeaking, banding, or drag.
- Wipe rods clean with a lint-free cloth.
- Apply a light, approved lubricant if appropriate (for your printer type).
- Clean rods periodically to maintain smooth, consistent motion.

3 Z-AXIS ASSEMBLY MAINTENANCE



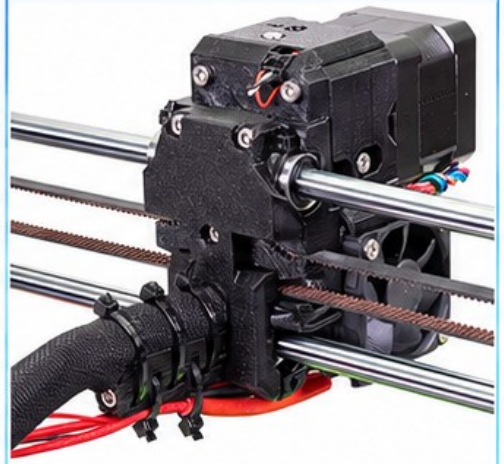
- Check and tighten the top Z-axis bolts and brackets.
- Clean the vertical smooth rod.
- Inspect the lead screw and rod for smooth, straight motion.
- Wobble or drag can reduce layer consistency.

4 EXTRUDER GEARS + FILAMENT GRIP



- Hobbed gear teeth can get packed with filament dust or debris, reducing grip.
- Slipping or clicking can cause under-extrusion and print quality issues.
- Weak or inconsistent filament feed often indicates dirty or worn gears.
- Clean debris from the gears so they can grip filament properly.

5 LINEAR BEARINGS: CLEAN / INSPECT / REPLACE



- Dirty or worn linear bearings can cause rough movement, play, or noise.
- Check for smooth motion and signs of wear.
- Replace worn or damaged bearings as needed (for your printer type).
- Make sure the carriage moves smoothly with no binding or excessive play.

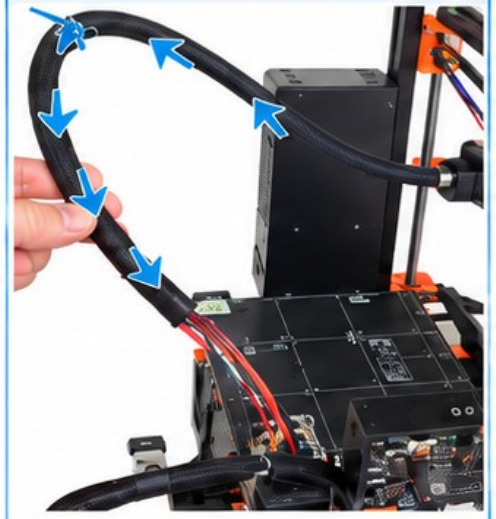
Power down before adjusting belts, rods, or motion hardware. Check one thing at a time and retest.

3D PRINTER ELECTRICAL MAINTENANCE & TROUBLESHOOTING

Focus areas, common issues, and how to keep your printer reliable and safe.

1 CABLE TO PRINT HEAD

INSPECT / ROUTE / REPLACE



The moving cable bundle to the print head flexes thousands of times and often wears where the wires meet the connectors.

LOOK FOR

- Frayed sleeving, exposed wires
- Broken or intermittent wires
- Loose connectors at the head

FIX

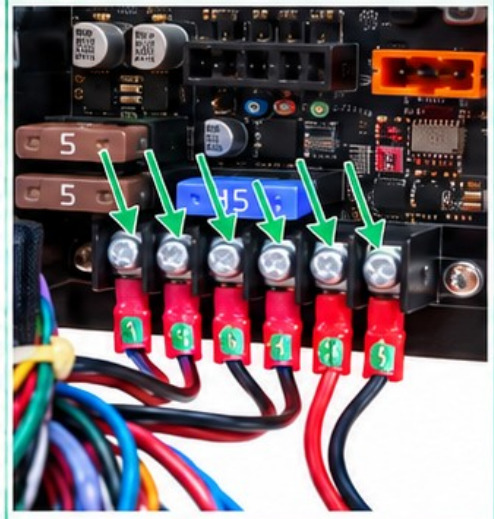
- Re-sleeve or replace damaged cable
- Ensure strain relief and smooth routing (no sharp bends)
- Replace loose or damaged connectors

PREVENTS

- Intermittent failures
- Nozzle/bed/heater/thermistor errors

2 POWER DISTRIBUTION BOARD

CHECK CONNECTIONS



The main power connections must be secure and free of corrosion. Vibration and heat can loosen terminals over time.

LOOK FOR

- Loose screw terminals
- Discolored, overheated, or corroded connectors
- Wires pulled from crimp terminals

FIX

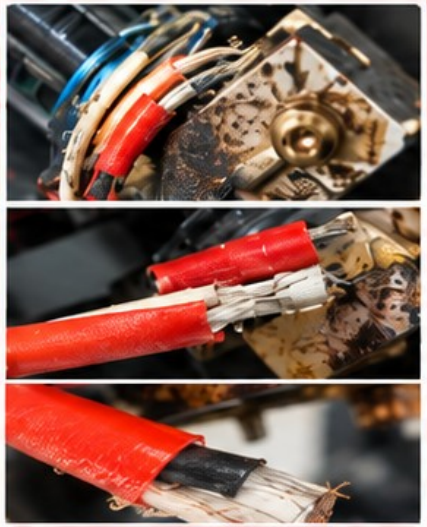
- Tighten screws (while powered off)
- Replace damaged crimp connectors
- Check for correct fuse ratings

PREVENTS

- Intermittent power loss
- Overheating and potential fire hazard

3 HOTEND WIRING

KEEP CLEAN AND INTACT



Heater and thermistor wires near the nozzle see high heat and constant motion. Damaged wires are a common cause of thermal runaway errors.

LOOK FOR

- Frayed, cut, or heat-damaged wires
- Melted insulation or exposed strands
- Wires contacting the heater block or nozzle
- Loose set screws or connector pins

FIX

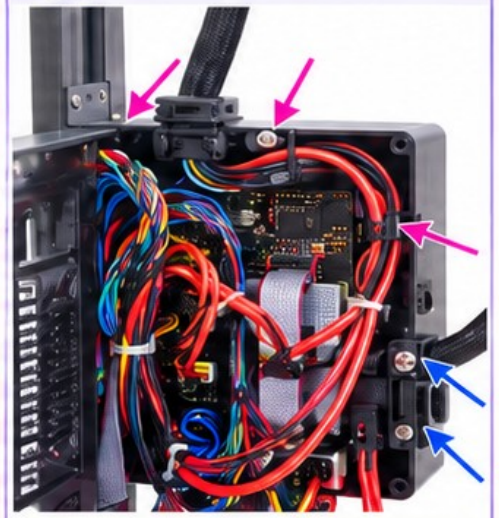
- Clean any melted filament and debris
- Replace damaged heater/thermistor wires
- Ensure proper strain relief and routing

PREVENTS

- Thermal runaway errors
- Erratic temperature readings

4 CONTROL BOX WIRING

ORGANIZE / SECURE / INSPECT



Messy or unsecured wiring can lead to snagging, intermittent connections, and difficult troubleshooting.

LOOK FOR

- Loose or tangled wires
- Cables not secured with strain relief or zip ties
- Wires near moving parts or hot surfaces
- Dust or debris inside the enclosure

FIX

- Route and bundle cables cleanly
- Use zip ties and strain reliefs (as shown)
- Keep high-current wires (red/black) separate from signal wires
- Remove dust and check for any damaged insulation

PREVENTS

- Intermittent errors

5 CONNECTORS & CRIMPS

INSPECT FOR WEAR / REPLACE



Worn or loose connector housings and crimp pins can cause intermittent electrical connections. Inspect for bent or loose pins and replace worn connectors as needed.

LOOK FOR

- Cracked or loose connector housings
- Damaged or loose crimp pins
- Bent, pushed out, or dirty pins
- Intermittent connection at small plugs

FIX

- Replace worn or damaged connectors
- Re-crimp pins or replace crimp terminals
- Ensure pins are fully seated and secure
- Use proper crimping tools and technique

PREVENTS

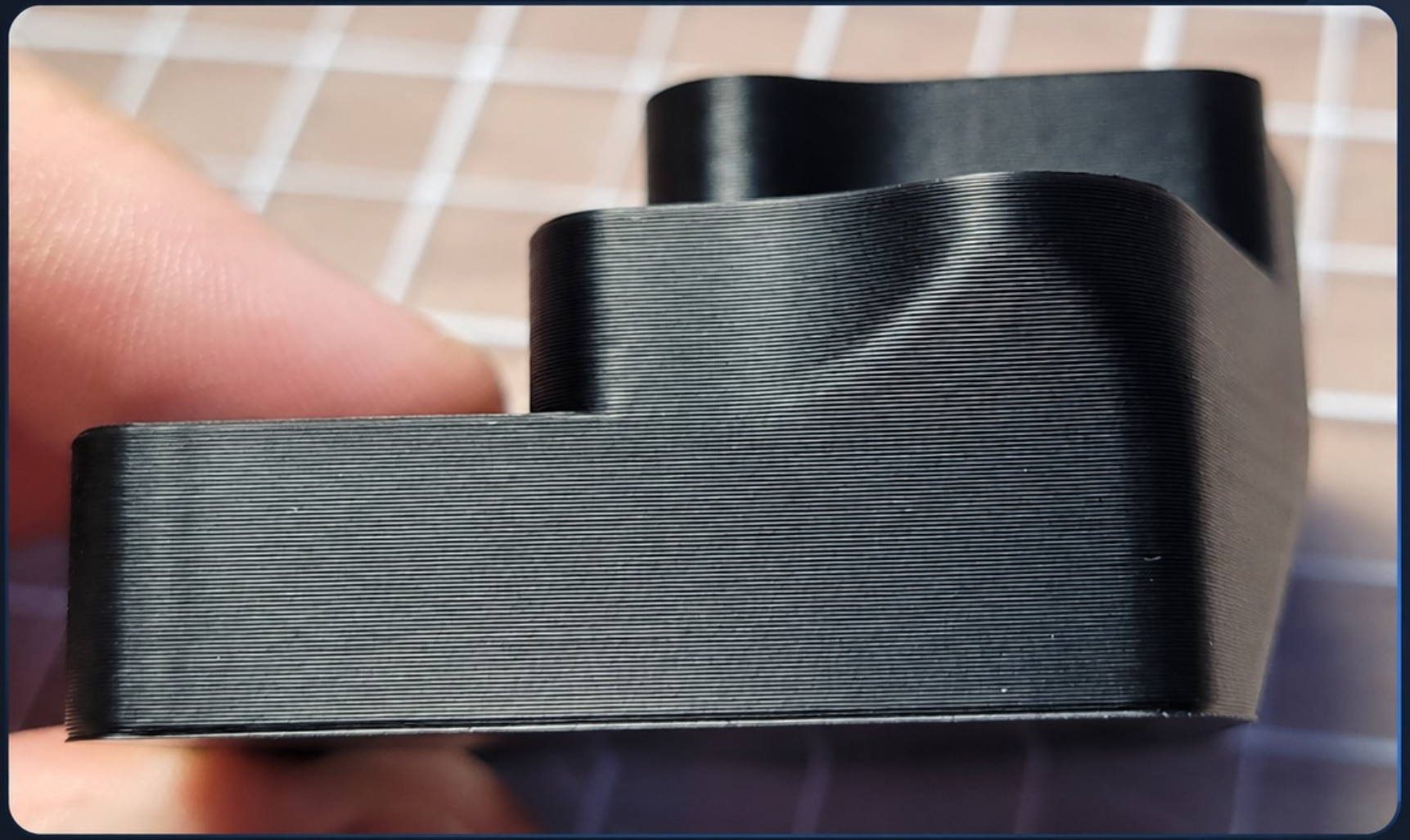
- Intermittent heater/thermistor faults
- Thermal runaway errors


PRUSA
PRUSA
FINAL APPROACH
Safety → Maintenance → Troubleshooting → Resources

This is what your prints can look like!

Clean, crisp, and beautifully made.

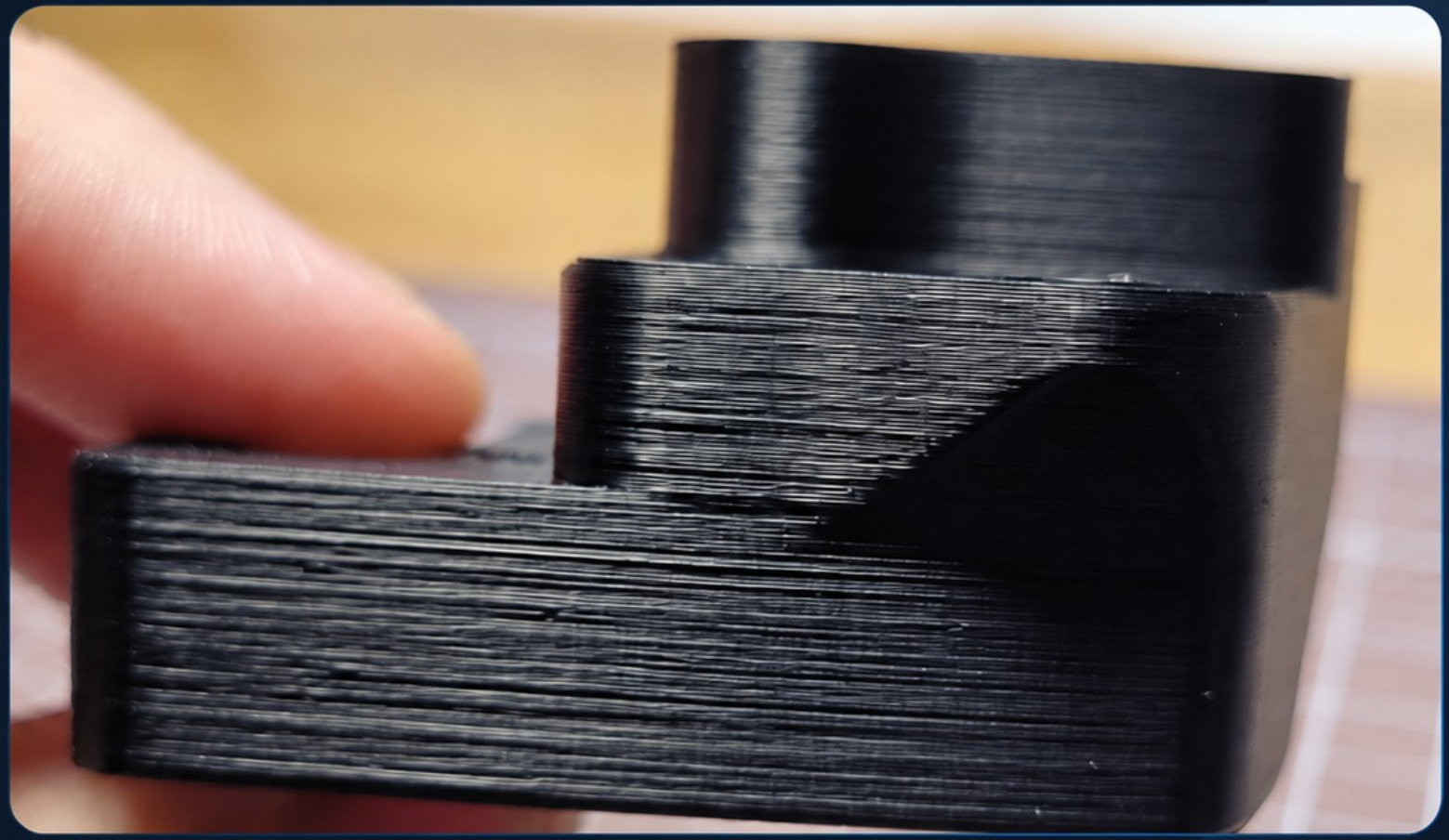
- Clean surfaces
- Crisp edges
- Consistent layers



Or, wait...

A different angle reveals a different story.

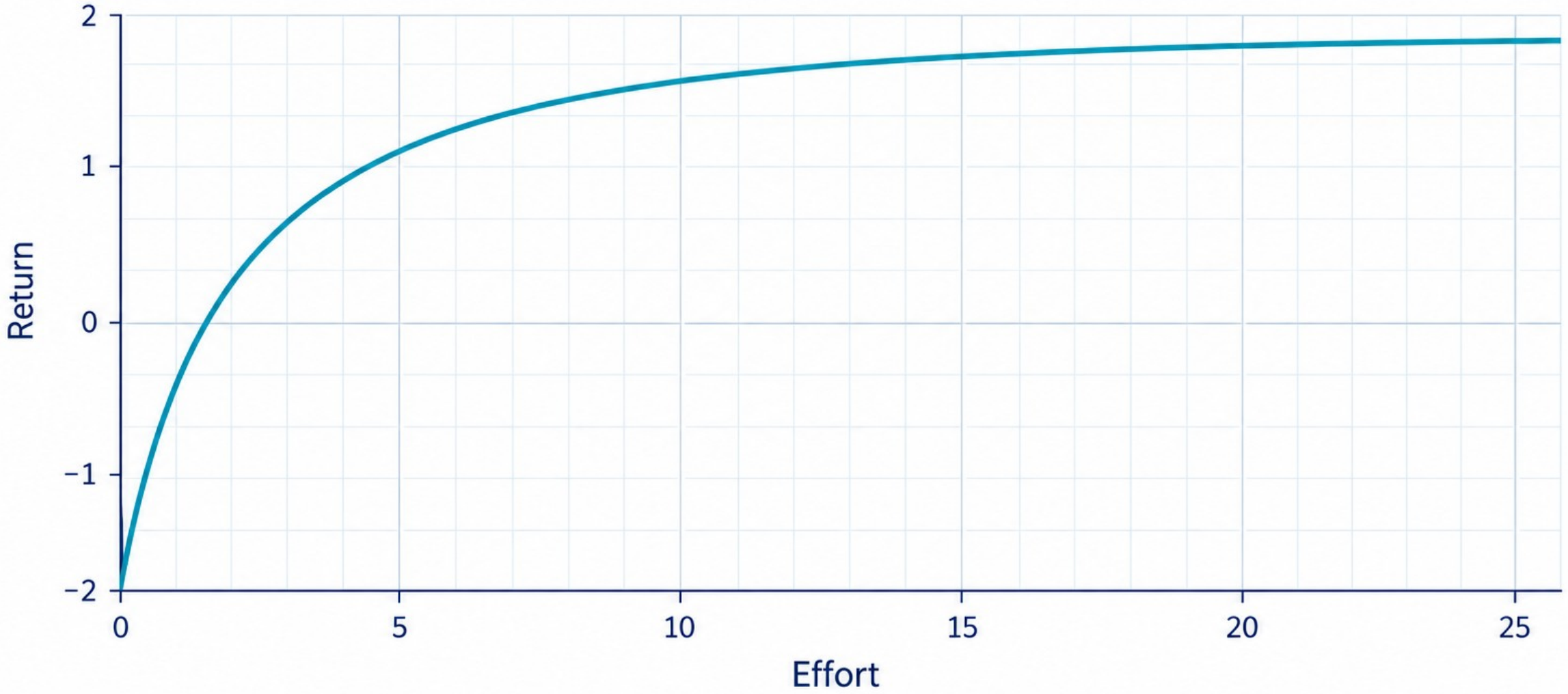
- Surface variation
- Visible layer lines
- Don't let great get in the way of good.



Printed part = design intent + process error + material variation.

Our goal is to reduce error until the part is fit for purpose.

The Law of Diminishing Returns



References and Deeper Learning

01

Ellis’ Print Tuning Guide

https://ellis3dp.com/Print-Tuning-Guide/

02

Print Quality Troubleshooting — Simplify3D

https://www.simplify3d.com/resources/print-quality-troubleshooting/

03

How to Make (Almost) Anything

https://fab.cba.mit.edu/classes/

04

How to Design (Almost) Anything

https://fab.cba.mit.edu/classes/

05

How to Make Something That Makes (Almost) Anything

https://fab.cba.mit.edu/classes/