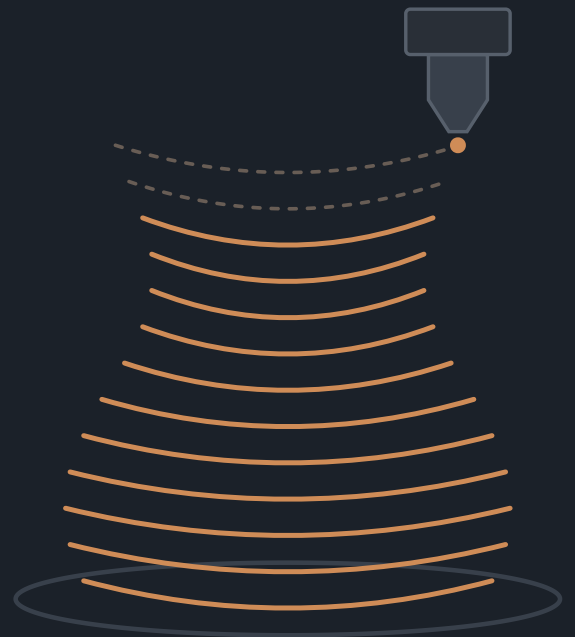


3D printing, from zero

A beginner's guide to printers, materials, settings, software and file formats, with the mistakes worth avoiding and the reasons behind them.



Edition 1.2 · October 2026 · fact-checked 11 October 2026

Written for 3D artists and designers starting to print.

Temperatures and prices are typical ranges; your printer's and material maker's values come first.

What is in this guide

Read chapters 1 to 3 before buying anything. Chapters 4 to 6 are what you will come back to while printing; 7 to 9 connect printing to the software you already use.

01

How 3D printing works

From model to object: the five steps every print goes through.

02

Printer types

Filament (FDM), resin (MSLA), powder (SLS/MJF) and how they differ.

03

Choosing a first printer

The features that matter, and what to ignore.

04

Slicer settings explained

Layers, walls, infill, supports, adhesion, speed, temperature.

05

Designing for printing

Overhangs, bridges, wall thickness, tolerances, orientation, strength.

06

Materials

Filaments and resins compared: temperatures, strength, use, care.

07

Software

Modelling, repair, slicers, resin tools, printer firmware, model libraries.

08

File formats

STL, OBJ, 3MF, STEP, G-code and the unit trap.

09

From 3ds Max to a print

A checklist for artists exporting their own models.

10

Finishing

Support removal, sanding, priming, painting, inserts, smoothing.

11

When prints fail

The common faults, their causes and fixes.

12

Safety, care and calibration

Fumes, resin handling, fire, drying filament, a calibration order.

13

Glossary

The terms you will meet in forums and slicers.

Two rules of thumb before you start

- **Filament for objects, resin for detail.** Most people should start with a filament (FDM) printer; add resin only if you need miniatures, jewellery-level detail or very smooth surfaces.
- **Print other people's tested models first.** A known-good file from a model library isolates printer problems from design problems. Print your own designs once the printer reliably prints theirs.

How 3D printing works

A 3D printer builds an object by adding material one thin layer at a time. Every desktop process follows the same chain: a closed 3D model is cut into layers by a program called a slicer, which writes instructions the printer follows.



The slicer is where most quality settings are chosen.

What each step really does

- 1 **Model.** Any 3D program can be the source, but the printer needs a *closed* surface ("watertight" or "manifold"): a solid with a clear inside and outside, no holes, no loose faces, no zero-thickness planes. Render-only models often fail here.
- 2 **Export.** The model is saved as a triangle mesh, usually 3MF or STL. Curves become flat facets, so export with enough resolution: about 0.01 mm deviation is plenty for filament prints; go finer for resin.
- 3 **Slice.** The slicer cuts the mesh into horizontal layers, plans the nozzle's paths or the light's pattern for each layer, adds supports where needed, and writes a printer-specific instruction file (G-code for filament printers, an image stack for resin).
- 4 **Print.** The printer works through the layers. Watch the first layer of every print: it is the most common point of failure.
- 5 **Finish.** Remove supports, clean, cure (resin), sand, prime and paint as needed.

Resolution is not one number

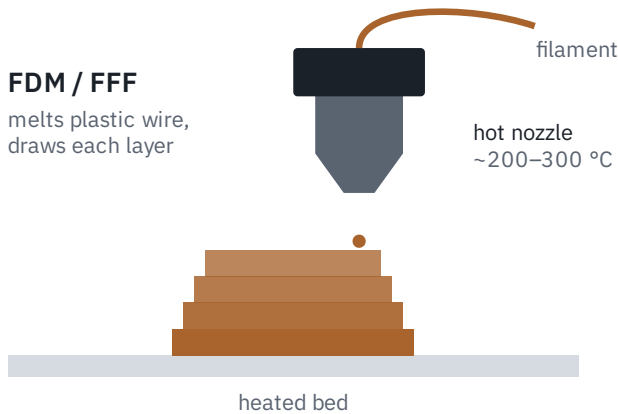
Printers are precise in two different ways, and they rarely match:

- **Layer height (Z).** How thick each layer is: typically 0.1–0.3 mm on filament printers, 0.025–0.1 mm on resin printers. Thinner layers mean smoother slopes and longer prints.
- **Detail in the layer (XY).** On filament printers, set by the nozzle width (0.4 mm standard): nothing narrower than about the nozzle width prints cleanly. On resin printers, set by the screen's pixel size (about 0.015–0.05 mm on current consumer printers), so fine detail is far easier.

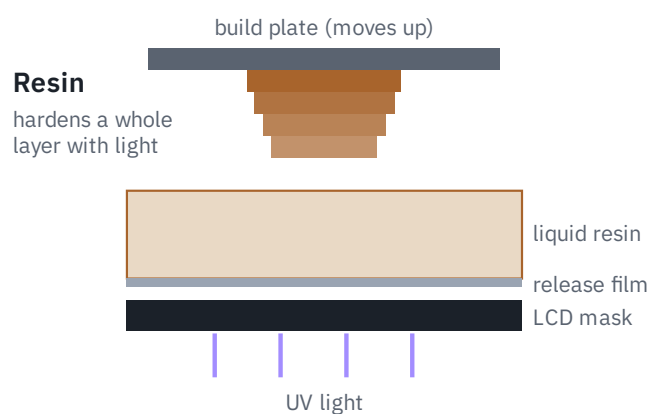
Why this matters to a 3D artist: a model can look perfect on screen and still be unprintable, because a feature much thinner than the nozzle disappears, and a surface at a shallow angle shows each layer as a visible step. Chapter 5 covers the rules.

Printer types

Three families cover almost everything you will meet. Two fit on a desk; the third you usually order from a print service.



Filament: a heated nozzle draws each layer with molten plastic.



Resin (MSLA): a screen lets UV light through in the shape of each layer; the part grows upside down.

TYPE	HOW IT WORKS	DETAIL	STRENGTHS	WEAKNESSES
FDM / FFF filament	Melts a plastic filament and lays it down line by line.	Layers 0.1–0.3 mm; features $\geq$ nozzle width (0.4 mm)	Cheap to run, many strong materials, large parts, multi-colour, low mess.	Visible layer lines; weak between layers; fine detail limited.
Resin SLA, MSLA, DLP	UV light hardens liquid resin layer by layer: a laser traces it (SLA), or a screen (MSLA) or projector (DLP) exposes it all at once.	Layers 0.025–0.1 mm; pixels about 0.015–0.05 mm	Excellent detail and smooth surfaces; on MSLA and DLP, many small parts print about as fast as one.	Hazardous, messy liquid; washing and curing; brittle standard resins; smaller build volume.
Powder SLS, MJF	A laser (SLS) or fusing agent plus heat (MJF) fuses plastic powder, usually nylon; unfused powder supports the part.	Layers about 0.08–0.12 mm	No supports, complex interlocking parts, strong nylon, strength more even in all directions than FDM.	Industrial or costly benchtop machines; most people order from a service. Grainy surface.
Others	Material jetting (full colour, e.g. PolyJet); binder jetting (full-colour sandstone, and metal parts sintered in a furnace afterwards); metal laser powder bed fusion (DMLS/SLM), which melts metal powder.	varies	Full colour, metals, engineering parts.	Mostly through service bureaus; expensive.

Inside an FDM printer

FEATURE	OPTIONS	WHAT IT MEANS FOR YOU
Motion system	Bed-slinger (the bed moves front–back), CoreXY (head moves in X and Y, bed only drops), delta (three arms).	CoreXY is faster and often enclosed; bed-slingers are usually cheaper and open. Deltas are now niche.

FEATURE	OPTIONS	WHAT IT MEANS FOR YOU
Extruder	Direct drive (motor on the print head) or Bowden (motor on the frame, pushes through a tube).	Direct drive handles flexible filament (TPU) and retracts more precisely; most popular current printers use it.
Enclosure	Open frame or enclosed box.	An enclosure is recommended for ABS, ASA, nylon and PC (they warp in drafts) and helps contain fumes. PLA can clog in a hot chamber: open the vent, lid or door as the maker advises.
Bed levelling	Manual knobs, or automatic probing.	Automatic levelling plus automatic first-layer height takes the guesswork out of the first layer, the most common beginner problem.
Multi-material	Filament changers (one nozzle, several spools), tool changers or dual nozzles.	Changers make multi-colour easy but waste filament at each colour change ("purge"); tool changers waste far less.
Build surface	Textured or smooth PEI spring steel, glass, special coatings.	Flexible steel sheets let you pop parts off by bending. Textured PEI suits most materials without glue.

03 • BUYING

Choosing a first printer

The best first printer is the one that works out of the box, so you learn to design and print rather than to repair a machine. In 2026 that is easy to find.

What matters, in order

- 1 Reliability out of the box.** Automatic bed levelling, automatic first-layer calibration, good default slicer profiles. Read recent reviews, not specifications.
- 2 Ecosystem.** A maintained slicer with tuned profiles, available spare parts (nozzles, belts, build plates), an active user community.
- 3 Enclosure,** if you will print ABS, ASA, nylon, or want fumes contained and a quieter room.
- 4 Build volume.** About 250 × 250 × 250 mm covers most things; large models can be printed in parts and joined (chapter 5).
- 5 Multi-colour,** only if you need it. It adds cost, purge waste and maintenance. It is excellent for signs, labels, two-colour dice and figurines.
- 6 Openness.** Some brands push you to their cloud and their slicer; others are fully open. Check you can print offline over your local network or from a USB stick or SD card.

Price tiers (approximate, US, October 2026)

TIER	TYPICAL PRICE	WHAT YOU GET
Entry filament	US\$200–400	Open frame, mostly bed-slingers, auto levelling, good for PLA, PETG and TPU. Often a small multi-colour option.
Mid filament	US\$350–1,300	Enclosed CoreXY, fast, prints ABS/ASA, cameras and failure detection, multi-colour systems.
Prosumer filament	US\$1,200+	Larger volumes, tool changers or dual nozzles (now also on some mid-tier models), engineering materials (nylon, PC, carbon-fibre filled).
Entry resin	US\$140–650	MSLA with a high-resolution monochrome screen. Add a wash and cure station.
Professional resin	US\$2,500+	Engineered resins, tuned workflow and very consistent results (for example Formlabs Form 4).

Well-known brands: Bambu Lab, Prusa Research, Creality, Elegoo, Anycubic, Qidi, Sovol and Snapmaker for filament; Elegoo, Anycubic, Phrozen and Creality for consumer resin, Formlabs for professional resin. Prices move often and models change yearly: check current prices and reviews.

Budget the extras

Filament printer

- 3–5 spools to start (PLA, PETG), about US\$12–30 per kg
- A filament dryer or a sealed box with desiccant
- Spare nozzles (0.4 mm), a second build plate
- Flush cutters, deburring tool, scraper, isopropyl alcohol

Resin printer

- Wash and cure station (or tubs plus a UV lamp)
- Nitrile gloves, safety glasses, paper towels, silicone mat
- Isopropyl alcohol (litres of it) or water for water-washable resin
- Ventilation or an air purifier, spare release film (FEP or similar)

Recommendation for a 3D artist starting out: an enclosed CoreXY filament printer from a brand with a mature slicer. Add a resin printer later if you want miniatures, jewellery masters or presentation-grade detail in small models.

04 • SLICER

Slicer settings explained

Slicers offer hundreds of settings, but the defaults for your printer and filament are good. These are the ones worth understanding first, with the values most people use on a 0.4 mm nozzle.

SETTING	WHAT IT DOES	TYPICAL	WHEN TO CHANGE IT
Layer height	Thickness of each layer.	0.2 mm (0.08–0.28)	Thinner for smooth curves and miniatures; thicker for fast, large parts. Keep it at most about 75–80 % of the nozzle diameter (about 0.3 mm).
Line width	Width of each extruded line.	0.42–0.45 mm	Slightly wider than the nozzle gives stronger walls.
Walls (perimeters)	Number of outline loops around each layer.	2–3	4–6 for strength. Walls add more strength than infill does.
Top / bottom layers	Solid layers closing the top and bottom.	~0.6–1 mm each	More on flat tops at low infill, to avoid "pillowing".
Infill density	How full the inside is.	10–20 %	15–40 % for functional parts; 100 % only for small, heavy or balanced parts (dice).
Infill pattern	Shape of the inside lattice.	Grid, gyroid, cubic	Gyroid and cubic are strong in every direction; lightning is fast and only holds up the top.
Supports	Temporary structure under overhangs.	Off, or tree supports	Needed for overhangs steeper than about 45° from vertical; tree supports use less material and usually come off more cleanly on organic shapes.
Bed adhesion	Skirt, brim or raft around or under the part.	None, or auto brim	Brim for small footprints, tall thin parts and warping materials.
Nozzle temperature	How hot the filament is melted.	per material	Hotter: better layer bonding, more stringing. Cooler: crisper detail, weaker bonding.
Bed temperature	Keeps the first layers stuck.	per material	Raise slightly if corners lift.

SETTING	WHAT IT DOES	TYPICAL	WHEN TO CHANGE IT
Cooling fan	Cools each layer as it is laid.	PLA 100 %, ABS about 10–30 %	More for overhangs and bridges; less for strength and warping materials.
Speed	How fast the head moves.	printer profile	Slow outer walls for quality; the hot end's melt rate (max volumetric flow) is often the real limit.
Retraction	Pulls filament back before travel moves.	~0.5–1 mm direct drive	Raise if stringing; too much can cause jams.
Seam position	Where each layer's loop starts and ends.	Aligned or rear	Hide it on a sharp corner or the back of the model.

Useful tools inside the slicer

- **Variable layer height:** thin layers only where the model curves, thick ones elsewhere.
- **Modifiers:** boxes or shapes that change settings in one area, for example more walls around a screw hole.
- **Support painting / blockers:** paint where supports are allowed or forbidden.
- **Ironing:** the hot nozzle passes again over flat tops to make them smooth.
- **Fuzzy skin:** deliberate random texture on outer walls, which hides small flaws and suits grips and handles.
- **Cut / split:** divide a model that is too big; PrusaSlicer, Bambu Studio and OrcaSlicer can add connectors (plugs, dowels) to align the parts.
- **Preview:** always scroll through the sliced layers before printing. It shows missing walls, floating parts and unsupported areas before they waste hours.

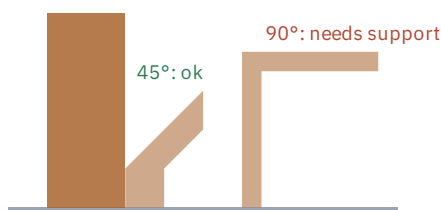
Infill is not strength. Two extra walls often add more strength than raising infill from 20 % to 30 %, for less plastic. For strong parts: more walls, thicker top and bottom, sensible orientation (chapter 5), then infill.

05 • DESIGN

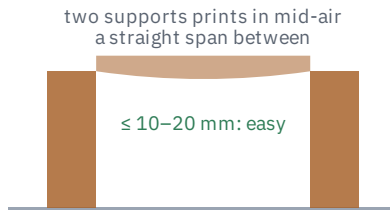
Designing for printing

The same few physical rules decide whether a model prints. Learn them once and most of your designs will print first time.

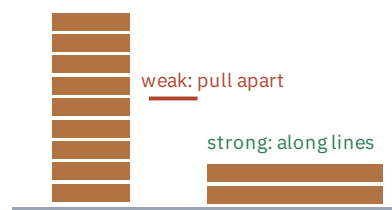
Overhangs



Bridges



Strength follows the layers



At 45°, each 0.2 mm layer steps out by 0.2 mm, so about half of each 0.4 mm line still rests on the layer below; much steeper and it starts to droop. A horizontal span supported at both ends can bridge. Parts break most easily between layers.

RULE	FILAMENT (0.4 MM NOZZLE)	RESIN
Overhang without support	Up to about 45° from vertical (50–60° with good cooling; more on printers with strong all-round cooling).	Islands and steep overhangs always need supports.
Bridge	10–20 mm is easy; about 50 mm is a practical limit with tuned bridge settings.	Short spans print; support longer ones.

RULE	FILAMENT (0.4 MM NOZZLE)	RESIN
Minimum wall	0.8 mm (two lines); 1.2 mm+ for strength.	0.8 mm or more for sturdy parts; 0.2–0.6 mm is the technical minimum, by printer (thin walls warp).
Smallest detail	About the nozzle width (0.4 mm) in XY.	About 0.1–0.2 mm.
Embossed / engraved text	Strokes at least two lines wide (about 0.8 mm) and 0.6 mm or more deep or high; avoid hairline fonts.	Strokes of 0.3–0.4 mm work.
Clearance between moving parts	At least 0.3 mm (sliding and print-in-place hinges); about 0.1 mm for a press fit, tested on your printer.	At least 0.4–0.5 mm for moving parts (less may fuse); 0.1–0.2 mm only for press fits.
Holes	Print slightly small: model them a little larger, use the slicer's hole compensation, or drill to size. Horizontal holes: use a teardrop or flat top.	Print close to size; resin shrinks slightly.
Bottom edges	Add a 0.5 mm chamfer: hides the "elephant's foot" bulge of the first layers.	Angle parts on supports instead.
Hollow parts	Not needed: the slicer adds infill automatically.	Hollow with walls of about 2 mm and add at least two holes of 3.5 mm or more (one near the build plate), or suction can fail the print and trapped resin can crack the part.

Orientation decides more than anything

- **Strength:** put the direction that will be pulled or bent along the layers, not across them. A hook whose layers cross its bend snaps easily; turned so the layers follow the bend, it holds far more.
- **Surface:** the side facing the bed is flat and smooth (or textured by the plate); supported sides are rough. Put the visible face up or vertical.
- **Accuracy:** circles are most accurate when they lie flat (in XY); vertical circles get a stepped top.
- **Supports:** rotate to minimise overhangs, but never at the cost of strength where it matters.

Large models and assemblies

Split anything larger than the build volume, and anything that would need many supports, into parts. Add alignment pins or keys (with 0.2 mm clearance), wide, flat glue faces, and hide the joints along natural lines. Threaded heat-set brass inserts make strong, reusable screw points in plastic.

A precise mesh still needs printable geometry. Architectural models are a classic trap: 1:200 railings, mullions and window frames become thinner than the nozzle. Thicken them to at least 0.8 mm at print scale, or print the model larger.

06 • MATERIALS

Materials

Start with PLA. Learn PETG next. Everything else is for specific needs. Temperatures are typical ranges: the value printed on your spool comes first.

Filaments

MATERIAL	EASE	NOZZLE °C	BED °C	ENCLOSURE	GOOD FOR	WATCH OUT FOR
PLA	●●●●●	190–220	45–65	No	Models, prototypes, décor, architectural models	Softens at ~55–60 °C (a hot car). Brittle compared with PETG. Variants: matte, silk, wood-filled, PLA+.

MATERIAL	EASE	NOZZLE °C	BED °C	ENCLOSURE	GOOD FOR	WATCH OUT FOR
PETG	●●●●○	230–250	70–90	No	Functional parts, outdoors in shade, containers	Strings; sticks very hard to smooth PEI (use glue stick as a release layer). Tougher, less stiff than PLA; softens around 70–80 °C.
ABS	●●○○○	240–270	90–110	Yes	Heat-resistant parts, smoothing with acetone vapour	Warping and cracks in drafts; styrene fumes: ventilate or filter.
ASA	●●○○○	240–260	90–110	Yes	Outdoor parts (UV-stable), car parts	Like ABS: warping, fumes; also smooths with acetone.
TPU	●●○○○	210–240	30–60	No	Flexible parts: seals, phone cases, tyres, feet	Print slowly; direct-drive extruder strongly preferred. Hardness given as Shore (95A common).
Nylon (PA)	●○○○○	240–290	80–110	Yes	Gears, hinges, tough wear parts	Absorbs moisture quickly; dry before and during printing. Warps; usually needs glue stick and an all-metal hot end.
PC	●○○○○	260–310	100–120	Yes	High heat and impact resistance	Needs a hot, enclosed printer; warps strongly.
CF / GF filled	●●●○○	as base	as base	as base	Stiff, matte, dimensionally stable parts	Fibres wear brass nozzles quickly; use hardened steel (it may need about 5 °C more). Often more brittle than the plain plastic.
PVA / BVOH	●●○○○	190–220	50–60	No	Soluble supports for PLA on multi-material printers	Dissolves in water, so very moisture-sensitive; expensive.

Diameter

Almost every printer uses **1.75 mm** filament; some older machines and UltiMaker's S series use 2.85 mm. They are not interchangeable.

Moisture

All filaments absorb water from the air. Wet filament pops, strings and prints weak. PLA and ABS tolerate it best; nylon, PC, PVA/BVOH and TPU worst. Dry before printing, roughly PLA 45–55 °C, PETG 55–65, TPU 60–70, ABS/ASA 75–85, nylon and PC 75–90, for 4–12 hours (the spool's datasheet comes first), and store sealed with desiccant.

Build plate and material

Textured PEI suits nearly everything. Smooth PEI gives smooth bottoms but PETG can bond too strongly: use a glue stick layer. Wipe plates with isopropyl alcohol between prints and wash them with dish soap and water now and then; a greasy plate (fingerprints) is the most common reason prints come loose. Never use acetone on textured PEI.

Colour and finish

Matte filaments hide layer lines best, especially in darker colours; glossy ones show them most. Silk can hide fine lines but shows seams and flaws on flat areas; translucent shows the infill inside.

Resins

TYPE	CHARACTER	USE
Standard	Detailed, hard, brittle.	Miniatures, display models.
ABS-like / tough	Less brittle, slightly flexible.	Functional prototypes, snap-fits, assemblies.
Water-washable	Cleaned with water instead of alcohol.	Convenience. Still hazardous: never pour wash water down the drain uncured.
Flexible / elastic	Rubber-like.	Gaskets, grips, soft parts.
Castable	Burns out cleanly.	Jewellery and lost-wax casting masters.
High-temperature, engineering	Heat or chemical resistance.	Moulds, jigs, technical parts.

Software

You need two kinds of program: one to make the model and one to slice it. Everything else is optional.

Making models

PROGRAM	KIND	BEST FOR
Tinkercad	Browser, free	Absolute beginners, simple parts from boxes and cylinders.
Fusion	Parametric CAD	Mechanical parts with exact dimensions. Free for non-commercial personal use, with limits (for example 10 editable documents).
Onshape	Browser CAD	Parametric CAD anywhere; the free plan is for non-commercial use and makes documents public.
FreeCAD	Open-source CAD	Parametric CAD without licences; steeper learning curve.
Rhino + Grasshopper	NURBS, visual scripting	Architecture, jewellery, parametric forms.
SketchUp	Surface modelling	Architectural massing; needs care to export closed solids.
Blender	Mesh, sculpt, free	Organic models. The free 3D-Print Toolbox extension (installed from Blender's extensions, no longer bundled) checks printability.
3ds Max	Mesh, poly modelling	Your existing models; see chapter 9 for the export checklist.
ZBrush, Nomad Sculpt	Digital sculpting	Miniatures and figures; very high polygon counts (decimate before export).

Slicers

SLICER	PRINTERS	NOTES
Bambu Studio	Bambu Lab, others	Polished, multi-colour painting, cloud and local printing.
OrcaSlicer	Most brands	Open source; began as a fork of Bambu Studio. Built-in calibration tests. A strong default choice.
PrusaSlicer	Prusa, many others	Open source, excellent defaults; also slices for Prusa's resin printer.
UltiMaker Cura	Very broad	One of the oldest open-source slicers, with plug-ins.
Manufacturer slicers	Their own printers	Creality Print, Elegoo and Anycubic tools: tuned profiles for those machines.
Lychee Slicer, CHITUBOX	Resin	Supports, hollowing, drain holes, island detection.
UVtools	Resin (files)	Inspects and repairs sliced resin files; finds islands and suction cups.

Checking and repairing meshes

- **Your slicer:** most repair small mesh errors on import and warn you.
- **Blender 3D-Print Toolbox** (free extension): checks thickness, overhangs, non-manifold edges.
- **Meshmixer:** no longer developed, but still a free download with tools for supports and repairs.
- **Autodesk Netfabb** (now sold as Fusion with Netfabb): professional, paid repair and preparation.

Printer firmware and remote control

- **Marlin:** classic firmware on many printers; recent versions also have input shaping.
- **Klipper:** moves computation to a small computer; faster printing, input shaping.
- **OctoPrint, Mainsail, Fluidd:** web interfaces to watch and control printers.
- Many current printers include their own app, camera and failure detection.

Where to find models

Printables, MakerWorld, Thingiverse, Thangs, MyMiniFactory and Cults3D. Check the licence before selling prints of other people's designs: many are free for personal use only.

08 • FILES

File formats

Use **3MF** when you can. STL still works everywhere, but it forgets units, colours and settings.

FORMAT	WHAT IT STORES	UNITS	USE IT FOR
3MF	Triangle meshes in a zip package, plus units, colours, materials, several objects and parts, and slicer settings.	Yes	The modern default. Multi-colour and multi-part models; sharing a ready-to-print project.
STL	Triangles only (binary or text). No colour, no units, no parts.	None (assumed mm)	Universal exchange. Check the scale on import.
OBJ	Polygons, UVs; colours and textures via a separate MTL file.	None	Coloured meshes and textured models; full-colour print services.
STEP	Exact CAD geometry (curves and surfaces, not triangles).	Yes	Engineering exchange. PrusaSlicer, Bambu Studio and OrcaSlicer import it and turn it into triangles at the resolution you choose (free Cura does not).
PLY, AMF	Meshes with vertex colour (PLY); AMF is an older multi-material format.	Varies	3D scans (PLY); AMF is rarely used now.
G-code	Machine instructions: moves, temperatures, fan speeds. Also .bgcode (Prusa binary) and .gcode.3mf (Bambu).	Printer	Sending to a printer. It is specific to one printer, nozzle and material: never reuse G-code across printers.
Resin print files	Layer images plus exposure settings: .ctb, .pwm, .x, .y, .z, .g and others.	Printer	One specific resin printer model.

The unit trap

STL and OBJ store bare numbers. Slicers assume they are millimetres. A model built in centimetres arrives ten times too small; one built in inches 25.4 times too small; one in metres a thousand times too small and nearly invisible. If a model appears tiny or huge in the slicer, the units are wrong, not the model. Fix it at export (chapter 9), or scale by the right factor in the slicer: never by eye.

What makes a mesh printable

- **Closed (watertight):** every edge is shared by exactly two faces, so there are no holes or open borders.
- **Manifold:** no edge shared by three or more faces, no faces touching at a single point, no internal duplicate faces.
- **Consistent normals:** every face points outwards.
- **No self-intersections:** parts may overlap as separate shells (slicers merge them), but a single shell should not pass through itself.
- **Real thickness:** a single plane (a leaf, a sheet of glass) has no volume and prints nothing. Give it thickness.

From 3ds Max to a print

A render-ready scene and a print-ready object are different things. This checklist turns one into the other.

- 1 Set units to millimetres.** Customize > Units Setup > System Unit Setup: millimetres, set before you model (Autodesk warns against changing it in an existing scene; there, know the factor). Model at real size, or at the final print scale for architectural models.
- 2 One closed object per part.** Convert to Editable Poly; weld coincident vertices; cap open borders; delete hidden internal faces.
- 3 Check with the STL Check modifier.** It highlights open edges, double faces, spikes and multiple edges. Fix until it reports no errors.
- 4 Give thickness to surfaces.** Planes, glass and thin shells need a Shell modifier with a printable thickness (0.8 mm or more on filament).
- 5 Apply modifiers you want printed.** Collapse the stack, or check the face count after export, so TurboSmooth, Chamfer and Shell are in the file. Mind the polygon count: more triangles slow the slicer, and some print services cap uploads at 1 million.
- 6 Export.** File > Export > STL (binary) or OBJ; export selected objects only. Check the face count and the object's size in the slicer.
- 7 Check in the slicer.** Correct size? Any repair warning? Scroll the sliced preview for missing walls and floating islands.

Typical surprises

WHAT YOU SEE	WHY	FIX
Model is tiny or huge in the slicer	Unit mismatch.	Export in mm, or scale by exactly 10, 25.4 or 1000.
Parts of the model missing after slicing	Open mesh, flipped normals, or zero-thickness surfaces.	STL Check; cap holes; unify normals; add Shell.
Faceted curves	Too few segments at export or in the model.	Raise segments or add TurboSmooth before export.
Thin details vanish	Thinner than the nozzle at print scale.	Thicken to 0.8 mm+, print larger, or use resin.

Finishing

Most prints need only support removal. Presentation pieces benefit from the full sequence below.

Filament prints

- 1 Remove supports** with flush cutters and pliers; a deburring tool cleans edges and holes.
- 2 Sand** from coarse to fine (about 120 → 220 → 400 → 800 grit), wet if you can. Work lightly: friction can soften PLA, which softens at about 55–60 °C.
- 3 Fill and prime** with filler primer, sand again; repeat until layer lines disappear.

Resin prints

- 1 Wash** in isopropyl alcohol (or water for water-washable resin): for the time the resin maker gives (often only a few minutes for hobby resins), ideally in two baths, dirty then clean. Don't over-soak: some resins swell or turn brittle.
- 2 Dry** completely before curing (Formlabs suggests at least 30 minutes after alcohol); leftover alcohol or resin can leave a white film.

- 4 **Paint** with acrylics or spray paint; finish with clear coat.
- 5 **Smooth** ABS and ASA with acetone vapour (outdoors or with proper ventilation, away from flames; the vapour is heavier than air and collects low).
- 6 **Insert threads:** press heat-set brass inserts with a soldering iron set about 10–20 °C above the filament's printing temperature.
- 3 **Remove supports:** many hobbyists do it while the part is still slightly soft; Formlabs recommends after curing so parts don't warp.
- 4 **Cure** under UV light for the time the resin maker gives; heavy over-curing can make some resins yellow or brittle.
- 5 **Sand and prime** where supports touched, then paint as for filament prints.

Glue: cyanoacrylate (superglue) for PLA, PETG and resin (sand smooth faces lightly first); plastic weld or ABS slurry (ABS dissolved in acetone) for ABS; two-part epoxy for large or stressed joints and to fill gaps.

11 • TROUBLESHOOTING

When prints fail

Change one thing at a time and keep notes. Many problems come from the first layer, wet filament or wrong temperatures.

Filament

PROBLEM	LIKELY CAUSE	FIX
First layer doesn't stick	Nozzle too high, dirty plate, bed too cold.	Re-run first-layer calibration; wash plate with dish soap; raise bed temperature 5 °C; slow the first layer.
Corners lift (warping)	Plastic shrinks as it cools; drafts.	Brim, clean plate, no drafts, rounded corners in the design; for ABS/ASA an enclosure and less cooling.
Stringing / hairs	Wet filament, too hot, too little retraction.	Dry the filament; lower temperature 5–10 °C; increase retraction a little.
Gaps, thin walls (under-extrusion)	Partial clog, wet filament, too fast or too cold for the hot end.	Clean or replace nozzle; dry filament; lower speed or raise temperature.
Elephant's foot	First layers squashed outwards.	Elephant-foot compensation in the slicer; slightly higher first layer; chamfer the bottom edge.
Layer shift	Head hit something; loose belt; too fast.	Check belts and pulleys; clear obstructions; reduce speed or acceleration.
Ripples after corners (ringing)	Frame vibration at speed.	Input-shaping calibration; lower acceleration; firm surface for the printer.
Droopy overhangs	Too steep, too hot, too little cooling.	Supports, more fan, lower temperature, reorient.
Spaghetti	Part came loose or collapsed mid-print.	Better adhesion (brim), supports, and turn on failure detection if your printer has it.
Zits and blobs	Seam and pressure at loop starts.	Align the seam to a corner; calibrate pressure advance; enable wipe.
Cracks between layers	Too cold, too much cooling (ABS/ASA).	Raise temperature; reduce fan; use an enclosure.

Resin

PROBLEM	LIKELY CAUSE	FIX
Nothing on the build plate	Plate not levelled; bottom exposure too short; cold resin.	Clean and level the plate; increase bottom exposure; keep resin and room at about 25 °C, not below 20 °C (check the resin's datasheet).

PROBLEM	LIKELY CAUSE	FIX
Part stuck to the film	Supports too weak; plate adhesion lost.	Stronger supports; check exposure; clear cured bits from the vat (plastic scraper or the printer's tank-cleaning function).
Layers separate	Exposure too short; lift too fast.	Run an exposure test; slow the lift.
Holes or cracks in hollow parts	Suction ("cupping") and trapped resin.	Add drain holes, one as close to the build plate as possible; reorient.
Missing small parts	Unsupported islands.	Island detection in the slicer; add supports.

12 • CARE

Safety, care and calibration

Safety first

- **Fumes and particles.** All filament printing releases ultrafine particles and some vapours; ABS, ASA and nylon release the most, and ABS and ASA give off styrene. Print in a ventilated room, ideally with an enclosure and filter, and not where you sleep.
- **Resin is a skin sensitiser.** Always wear nitrile (not latex) gloves and eye protection; wash resin off skin with soap and water, never alcohol; ventilate; keep it away from food, children and pets. Allergies can build up over time.
- **Waste.** Cure leftover resin, supports and resin-soaked paper towels in sunlight or UV before binning them. Never pour liquid resin, or wash liquid containing it, down a drain or in the bin: used alcohol goes to hazardous-waste collection (check local rules). Isopropyl alcohol is flammable.
- **Fire.** Hot ends reach 290–350 °C. Use a printer with thermal-runaway protection (a firmware safety cut-out; check yours has it), fit a smoke detector nearby, and don't leave early prints unattended.
- **Burns and pinches.** Keep hands away from the hot end, the heated bed and moving belts.

Routine care

- Keep the build plate free of fingerprints; clean it as its maker advises (IPA, or dish soap and water).
- Store filament sealed; dry it before printing if it pops or strings.
- Replace the nozzle when prints degrade; use hardened steel for filled filaments.
- Check belt tension and keep rods or rails clean.
- Resin: filter the resin after a failed print or before storing it, inspect the film for scratches, replace it when cloudy or damaged.
- Update firmware and slicer profiles.

Calibration, in this order

- 1 First layer / Z offset
- 2 Temperature tower (best temperature per filament)
- 3 Maximum volumetric speed (how fast the material can melt)
- 4 Pressure advance (sharp corners, no blobs)
- 5 Flow rate (wall thickness matches the slicer)
- 6 Retraction (stringing)
- 7 Dimensional check: a tolerance test or a 20 mm test cube

Steps 2 to 7 follow the order in OrcaSlicer's calibration guide, which builds each test print for you. Many current printers set the first layer automatically, some also pressure advance; input shaping is calibrated at the factory (Prusa) or by the printer itself (Bambu Lab).

Printing balanced parts (dice, wheels, tops)

A perfectly balanced model only stays balanced if it prints evenly: use 100 % infill, place the seam on an edge, and keep the whole part in one material. Sparse infill is rarely symmetrical, so one side can end up heavier. A salt-water float test often fails, because solid PLA is about as dense as saturated salt water; only counting many rolls shows whether a die is fair.

Glossary

Bed-slinger	A printer whose bed moves back and forth (Y) while the head moves sideways (X).
Bowden	An extruder mounted on the frame, pushing filament through a tube to the hot end.
Bridge	A horizontal span printed in the air between two parts of the print.
Brim / skirt / raft	Extra lines around (brim) or near (skirt) the part, or a base under it (raft), for adhesion or priming.
CoreXY	A motion system where two motors together move the head in X and Y; the bed usually moves only up and down.
Direct drive	An extruder motor mounted on the print head, right above the nozzle.
Elephant's foot	The bulge at the bottom of a print, caused by the squashed first layers.
FDM / FFF	Fused deposition modelling / fused filament fabrication: printing with melted filament.
FEP	The transparent non-stick film at the bottom of a resin vat (some newer vats use nFEP or ACF film instead).
G-code	The text instructions a filament printer follows: moves, temperatures, fan speeds.
Hot end	The heated part of the print head: heater, thermistor, heat break and nozzle.
Infill	The internal lattice of a print.
Input shaping	Firmware that cancels most of the printer's vibrations so it can print fast with fewer ripples.
Island	In resin printing, a part of a layer not connected to anything below; it usually needs a support.
Manifold / watertight	A closed mesh with a clear inside and outside, printable as a solid.
MSLA	Masked stereolithography: resin cured through an LCD screen, one whole layer at a time.
Overhang	A surface leaning out beyond the layer below.
PEI	A plastic build surface most filaments stick to when hot and release when cool.
Perimeters / walls	The outline loops of each layer.
Pressure advance	Compensates for pressure in the nozzle so corners are sharp and lines even.
Purge / prime tower	Waste printed when changing colour or material, to clear the old one from the nozzle.
Retraction	Pulling filament back before a travel move to stop oozing.
Seam	The visible point where each layer's outer loop starts and ends.
Slicer	Software that turns a 3D model into layer-by-layer printer instructions.
Stringing	Thin hairs of plastic left across gaps during travel moves.
Supports	Temporary structures printed under overhangs, removed afterwards.
Volumetric flow	How much plastic (mm ³ /s) the hot end can melt: often the real limit on print speed.
Z offset	The fine adjustment of the nozzle's height above the bed for the first layer.

Where to go next: your printer maker's knowledge base (Prusa's and Bambu Lab's are excellent even for other brands), the slicer's built-in calibration tests, and the communities on Printables and r/3Dprinting.