Quick 3D
Your Single Manufacturing Platform

FREE ENGINEERING GUIDE

Design for 3D Printing Guide

Practical rules for walls, holes, snap-fits and file preparation — so your parts print right first time, cost less and are quoted accurately.

Inside this guide

1. Wall thickness by process
2. Seven tips for better walls and features
3. Designing snap-fits that don't snap
4. Preparing and exporting STL and STEP files
5. Common file problems and fixes
6. Pre-submission checklist

Quick 3D · 36, Jalan Serendah 26/41, Sekitar 26, 40400 Shah Alam, Selangor, Malaysia · +60 11-5629 2133 · sales@quick-3d.com · www.quick-3d.com

1. Wall thickness by process

Wall thickness is the most common reason a 3D printed part fails — or costs more than it should. Too thin and it breaks or won't print; too thick and it warps, wastes material and takes longer.

A **supported** wall is connected to other walls on two or more sides, like the side of a box. An **unsupported** wall is attached on one side only, like a free-standing fin or tab — it needs to be thicker.

Process	Thin-wall starting point	Comfortable for functional parts	Watch out for
FDM	About 1 mm	1.5–3 mm	Walls are built from extruded lines, so thickness works best in multiples of the line width. Strength is lowest between layers.
SLA	About 0.5 mm supported, 1 mm unsupported	1–3 mm	Very thin walls can warp during curing; hollow large solid sections and add drain holes.
SLS / MJF	About 1 mm	1.5–3 mm	Thick solid blocks can warp and cost more; hollow them and add escape holes for powder.
Metal	About 1 mm	Depends on geometry	Thin walls must also survive heat treatment and support removal.

These are conservative starting points that print reliably — not hard limits. The right value also depends on the material, part size and what the part has to do.

> **Design to the process accuracy.** Quick 3D's prototyping and industrial FDM services achieve a dimensional accuracy of ±0.3 mm at layer heights of 50–300 µm, with build volumes up to 500 × 500 × 500 mm. Size gaps, holes and clearances with that in mind.

2. Seven tips for better walls and features

1. **Keep walls as uniform as you can.** Even thickness cools and shrinks evenly, which reduces warping.
2. **Blend thickness changes gradually.** Use tapers and fillets instead of sudden steps from thin to thick.
3. **Use ribs instead of thick walls.** A thinner wall with ribs is often stiffer, lighter and cheaper than one thick slab.
4. **Hollow large solid parts.** For SLA, SLS and MJF add drain or escape holes; for FDM, lower infill does a similar job.
5. **Thicken unsupported features.** Free-standing fins, tabs and pins need more material than supported walls.
6. **Size small holes generously.** Small holes can print slightly undersize; for precise fits, design them to be drilled or reamed afterwards.
7. **Make text and fine details bold enough.** Embossed or engraved text needs enough width and depth to stay readable after finishing.

Thinning over-built walls, adding ribs and hollowing large sections are some of the easiest ways to cut cost without losing strength.

3. Designing snap-fits that don't snap

A snap-fit is a joint where one part flexes as it is pushed over another, then springs back into a recess to lock in place. Most failures happen at the root of the beam.

Choose a material that can bend

Best: PA11 and PA12 nylon (SLS, MJF) — PA11 is the most forgiving for repeated use — and polypropylene on SLS. Good: tough FDM nylon, PETG and ABS. In SLA use tough or durable resins; standard rigid resins are too brittle.

Shape the beam to spread strain

Make the beam long, taper it from a thicker root to a thinner tip, add a fillet where it meets the wall, and keep the hook shallow.

Set the angles

A gentle lead-in angle makes assembly easy. A steep retention face makes a joint that is hard to open; a sloped face lets it be released.

Print in the right orientation

In FDM, orient the beam so it bends along the layers — usually lying flat. MJF parts have uniform properties in every direction, which suits snap-fits.

Then test. Allow clearance for print accuracy, and print a small test of just the joint before the full part.

4. Preparing and exporting your files

Format	What it is	When to use it
STEP (.step / .stp)	An exact solid model with true curves, faces and edges	Best for quoting and engineering parts — we can measure features, make small edits or machine critical surfaces.
STL (.stl)	A mesh of triangles approximating the surface	Fine for most printing. Export at a good resolution so curves stay smooth.
3MF (.3mf)	A modern mesh format that can carry units, colours and materials	Useful for multi-colour or multi-material parts.
OBJ (.obj)	A mesh format that can carry colour and texture	Common for scans and full-colour models.
2D drawing (.pdf)	Dimensions, tolerances, threads and notes	Send alongside the 3D file for tight tolerances, threads or a specific finish.

Export checklist

- **Units in millimetres.** STL files don't store units — a part designed in inches can arrive 25.4 times too small.
- **Fine STL resolution.** Too coarse makes curves faceted; excessively fine just makes huge files.
- **Binary STL.** Much smaller than ASCII with the same geometry.
- **One part per file, clearly named** — for example bracket-v3.step.
- **Remove what shouldn't print:** reference geometry, construction bodies, screws and other hardware.
- **Reopen the exported file** in a viewer to check size, smoothness and that nothing is missing.

5. Common file problems and fixes

Problem	What you see	Fix
Faceting	Curves look like flat segments	Re-export the STL at a finer resolution, or send the STEP file
Wrong units	Part is tiny or huge	Export in millimetres, or tell us the units used
Holes in the mesh (non-manifold)	Gaps, missing faces or viewer errors	Fix the model so every body is a closed solid, then re-export
Overlapping bodies	Several shells inside each other	Combine into one solid before export, or export separately
Walls too thin	Features missing or fragile	Thicken walls — see section 1
Threads lost	Cosmetic threads don't appear in an STL	Call out threads on a drawing; small threads are usually tapped or fitted with inserts after printing

6. Pre-submission checklist

- ☐ Walls meet the starting thickness for the process; unsupported features thickened
- ☐ Thickness changes blended; large solids hollowed with drain or escape holes
- ☐ Clearances and holes sized for print accuracy; precise holes marked for drilling or reaming
- ☐ Snap-fits in a flexible material, with tapered beams and root fillets
- ☐ STEP file attached (plus STL if already checked), exported in millimetres
- ☐ Drawing attached for critical dimensions, tolerances and threads
- ☐ Quantity, material or requirements, finish and required date included

Get a printability review with your quote

Send your file to sales@quick-3d.com, WhatsApp +60 11-5629 2133, or visit quick-3d.com/contact-us. We review wall thickness and other printability issues before printing, and our DfAM team can optimise your design for strength, weight and cost. No 3D file? We can turn a sketch, drawing or physical part into CAD.

These are general design guidelines; results depend on material, geometry and process settings. © 2026 Quick 3D (Quick ED Solutions & Services Sdn. Bhd.)