

DESIGN GUIDE

Designing for 3D Printing: SLA, SLS, MJF, FDM, and DMLS

How to choose among SLA, SLS, MJF, FDM, and DMLS, with minimum wall and feature sizes, overhang and orientation rules, drain holes, and achievable tolerances.

3D printing builds a part layer by layer, so the design rules are unlike any subtractive or molded process: there is no tool to reach a feature and no cavity to fill, but every layer has to be supported by the one beneath it, and each technology has its own smallest feature it can resolve. Choosing the right process is half the design decision.

This guide covers the rules that matter across the five technologies we run, with the numbers we use. Each is the same check the FabWorks DFM engine applies to your model before you quote, grounded in the ISO/ASTM 52900 and 52911 additive standards and published machine data. Where a rule depends on the technology, we say so.

The five technologies

Technology	Material class	Resolution	Typical use
SLA / MSLA (resin)	Photopolymer resin	Finest detail, smooth surface	Concept models, fit checks, masters
SLS (nylon powder)	PA11, PA12, filled nylons	Isotropic, no supports	Functional plastic parts, living hinges
MJF (nylon powder)	PA12, PA11	Fine, consistent	Functional parts, small runs
FDM (filament)	ABS, PC, nylon, PLA, PETG	Coarsest, layer lines visible	Jigs, rugged prototypes, large

Technology	Material class	Resolution	Typical use
			parts
DMLS / SLM (metal)	316L, AISi10Mg, titanium, tool steel	Fine, needs support and machining	Metal prototypes, complex metal parts

Minimum wall and feature size

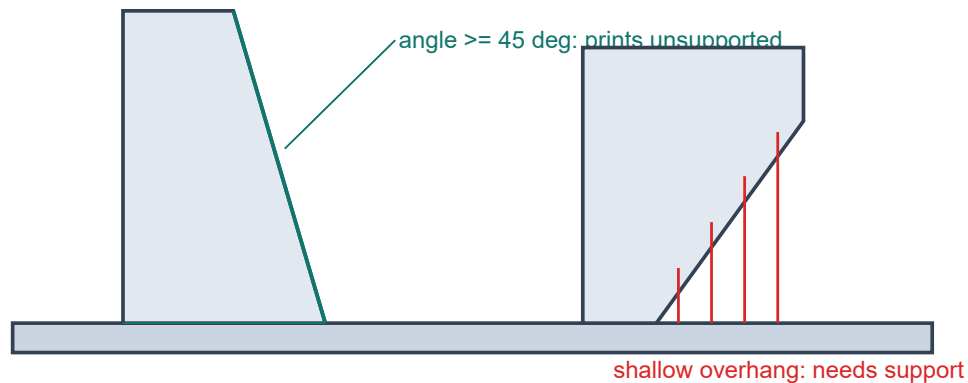
A wall thinner than the process can resolve prints incompletely or warps off the plate. Powder processes need a thicker minimum than resin because loose powder has to support the melt.

Technology	Min wall (recommended)	Min positive feature
SLA	0.8 mm (0.031 in)	0.3 to 0.5 mm (0.012 to 0.020 in)
SLS	1.0 to 1.2 mm (0.040 to 0.047 in)	0.8 to 1.0 mm (0.031 to 0.040 in)
MJF	0.8 to 1.0 mm (0.031 to 0.040 in)	0.5 to 0.8 mm (0.020 to 0.031 in)
FDM	1.0 to 1.2 mm (0.040 to 0.047 in)	0.8 to 1.0 mm (0.031 to 0.040 in)
DMLS metal	0.5 mm (0.020 in)	0.3 to 0.5 mm (0.012 to 0.020 in)

Overhangs, supports, and orientation

On resin, metal, and filament processes, any surface shallower than about 45 degrees from horizontal needs support structures, which leave witness marks where they attach and cost time to remove. SLS and MJF are self-supporting because the surrounding powder holds the part, so they carry no support marks at all.

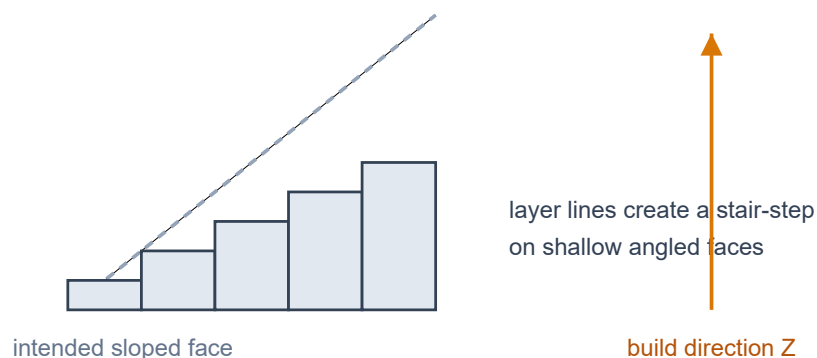
Overhang angle and support



Faces steeper than about 45 degrees print unsupported; shallower faces need support marks.

Orientation decides which faces stair-step, where supports land, and which axis is strongest, since a printed part is weakest across the layer lines. Tell us the critical face or the load direction and we orient to suit.

Layer orientation and stair-stepping



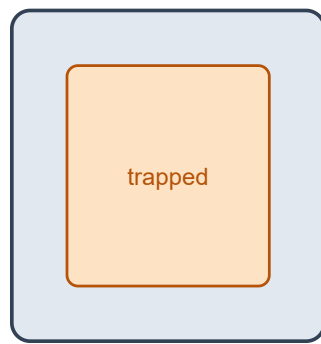
Orientation sets which faces stair-step, where supports land, and the strong axis.

On FDM, a horizontal gap can bridge only a limited span, about 5 to 10 mm (0.2 to 0.4 in), before it sags; longer spans need support or a redesign. Metal printing bridges only about 2 mm (0.08 in) unsupported.

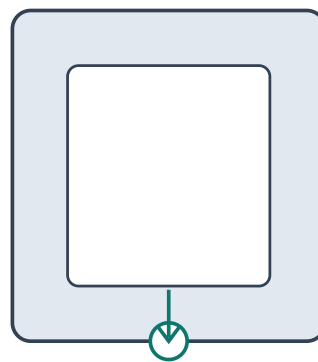
Holes, hollows, and trapped material

Printed holes shrink slightly and are best reamed if they must be precise. As a screening minimum, keep holes at or above 0.5 to 1 mm (0.020 to 0.040 in) on resin, MJF, and metal, and 1.5 to 2 mm (0.060 to 0.080 in) on SLS.

Hollow parts need an escape hole



sealed void: fails



drain hole: clears

Powder or resin in a sealed cavity cannot escape. Add a drain hole (2 to 4 mm) per void.

Hollowing a part saves material and time, but a fully enclosed cavity traps liquid resin or loose powder that can never come out. Add at least one drain or escape hole, about 2 to 4 mm (0.08 to 0.16 in), to every enclosed void.

Accuracy and finish

Technology	Typical tolerance
SLA	+/- 0.1 to 0.15 mm (0.004 to 0.006 in)
SLS / MJF	+/- 0.3 mm (0.012 in)
FDM	+/- 0.3 to 0.4 mm (0.012 to 0.016 in)
DMLS metal	+/- 0.1 to 0.2 mm (0.004 to 0.008 in), machined where tighter

Additive tolerances are looser than machining and scale with part size, so call out only the critical dimensions and plan to machine metal features that need a tight fit. Finishes range from as-printed through bead blast, dye, sanding, and paint; metal parts are typically bead blasted or machined on functional faces.

The DFM checklist

Before you send a model to quote, confirm:

1. Technology chosen for the material, detail, and strength the part needs.
2. Walls and features at or above the process minimum.
3. Overhangs above 45 degrees, or supports accepted on resin, metal, and filament.
4. The part oriented for the critical face and load direction (tell us which).
5. Enclosed cavities given a drain or escape hole.
6. Tight tolerances called out only where needed, metal fits planned for machining.

How FabWorks checks this for you

When you upload a model, our DFM engine measures wall thickness, feature size, overhangs, hole sizes, and enclosed voids against the technology and material you choose, before you quote. It flags a wall too thin to build, an unsupported overhang, or a sealed cavity that will trap powder, and explains the fix. Each result is grounded in the additive design standards behind it. The outcome is a part that builds cleanly and comes off the machine right.