

DESIGN GUIDE

Designing for Injection Molding: A DFM Guide

Wall thickness, draft, ribs and bosses, undercuts, gates, and tolerances for injection-molded parts, keyed to resin, with the numbers we use.

Injection molding rewards discipline at the design stage. A molded part is made hundreds or thousands of times from one tool, so a wall that sinks, a rib that warps, or an undercut that forces a side action is a defect or a cost that repeats on every shot. Get the geometry right and the tool is simpler, the cycle is faster, and the parts are consistent.

This guide covers the design rules that most affect moldability, with the numbers we use. Each is the same check the FabWorks DFM engine runs on your model before you quote, and each is grounded in published polymer and tooling references. Molding rules depend heavily on the resin, so the values below are keyed to material where it matters.

Wall thickness

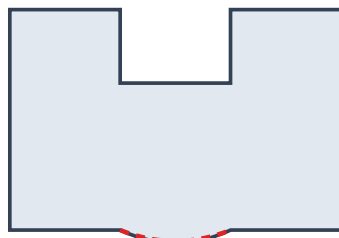
Wall thickness sets fill, cooling, and cycle time. Too thin and the cavity will not fill (short shot). Too thick and the part sinks, warps, and traps voids as the core cools slower than the skin. Design to a resin-appropriate nominal wall and hold it as uniform as possible.

Resin	Typical wall range	Practical minimum
ABS	1.2 to 3.5 mm (0.047 to 0.138 in)	1.2 mm (0.047 in)
Polycarbonate (PC)	1.0 to 4.0 mm (0.039 to 0.157 in)	1.0 mm (0.040 in)
Polypropylene (PP)	0.8 to 3.8 mm (0.031 to 0.150 in)	0.8 mm (0.031 in)
Nylon (PA6, PA66)	0.8 to 2.9 mm (0.031 to 0.114 in)	0.8 mm (0.031 in)

Resin	Typical wall range	Practical minimum
POM (acetal)	0.8 to 3.0 mm (0.031 to 0.118 in)	0.8 mm (0.031 in)
PEEK	1.0 to 3.0 mm (0.039 to 0.118 in)	1.0 mm (0.040 in)

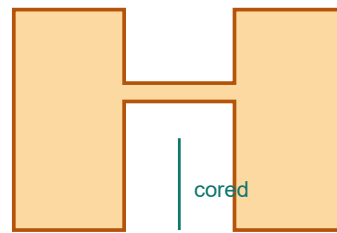
Wall uniformity and sink

thick section: sinks



sink mark

cored out: uniform wall



cored

Keep thick-to-thin transitions under about 3 to 1; core out heavy masses. Schematic, not to scale.

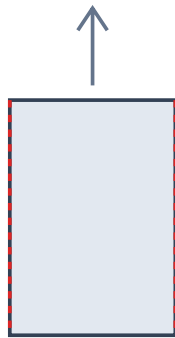
Keep wall transitions gradual. A thick-to-thin ratio beyond about 3 to 1 causes differential cooling, which shows up as sink and warp. Where a thick section is unavoidable, core it out rather than leaving a solid mass.

Draft angle

Every face parallel to the direction the part is ejected needs draft, or the part drags on the tool and scuffs, sticks, or ejector-pins through. The rougher the texture, the more draft it needs.

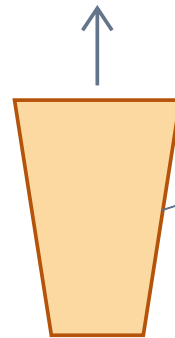
Draft and ejection

0 degrees: drags on tool



ejection

with draft: releases cleanly



ejection

Add about 1 degree per 25 mm of draw depth, more for textured surfaces. Schematic, not to scale.

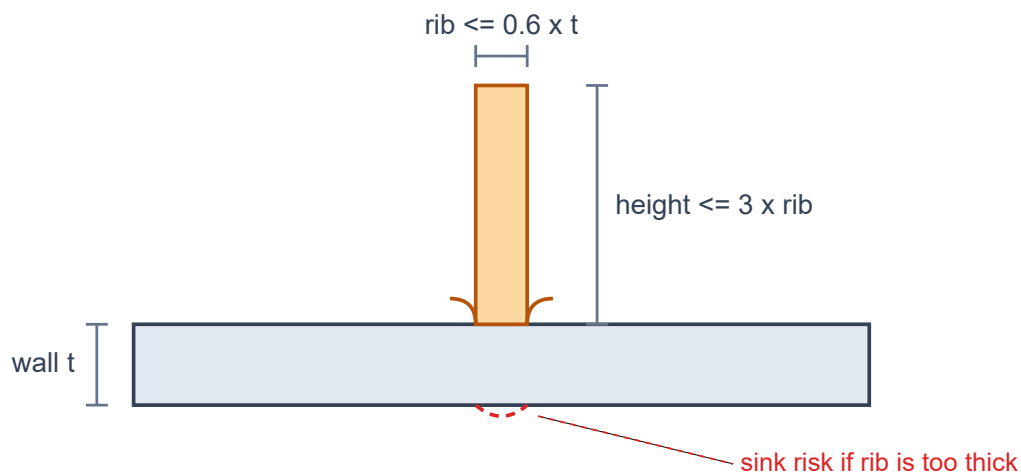
Surface	Recommended draft
Smooth, polished (SPI A)	0.5 to 1 degree
Semi-gloss (SPI B)	1 to 2 degrees
Matte (SPI C)	1.5 to 3 degrees
Light texture	3 to 5 degrees
Heavy texture	5 to 7 degrees

Rule of thumb: add roughly 1 degree of draft for every 25 mm (1 in) of draw depth, and add draft for textured surfaces on top of that. Draft is nearly free at the design stage and expensive to add after the tool is cut.

Ribs and bosses

Ribs add stiffness without thickening the wall, but a rib that is too thick pulls a sink mark on the opposite face.

Rib proportions and sink risk



Schematic, not to scale.

- Rib thickness: keep at or below 60 percent of the nominal wall (for stiffer resins such as PP and PA6, closer to 50 percent).
- Rib height: up to about 3 times the rib thickness.
- Boss outer wall: keep at or above 40 percent of the boss outside diameter, and connect bosses to a wall or rib rather than leaving them standing alone.

Corners, sink, and stress

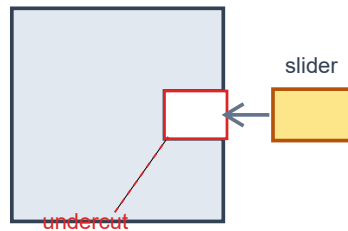
- Add a radius to every internal corner. Sharp internal corners concentrate stress and cause cracking; a radius of about half the wall thickness is a good default.
- Avoid sharp external corners; a small radius (0.5 mm / 0.020 in and up) improves flow and durability.
- Watch for sink opposite thick features. A local section thicker than about 4 mm (0.16 in) in ABS (higher in some resins) risks a visible sink mark.

Undercuts

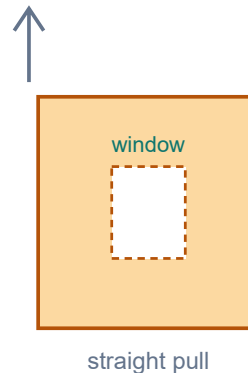
Any undercut prevents the part from pulling straight off the core, so it forces a side action (a slider or lifter) in the tool, which adds cost and cycle time.

Undercuts and side actions

undercut: needs a side action



pass-through window: straight pull



Design undercuts out where you can; flag the rest for tooling. Schematic, not to scale.

Design undercuts out where you can, for example by adding a pass-through window under a snap hook so the tool can form it on a straight pull. Where an undercut is functional, flag it early so the tool is quoted correctly.

Gates, weld lines, and fill

- Fill path: a single gate can reliably fill roughly 100 mm (4 in) of flow length on a cold runner, more on a hot runner. Long, thin, or complex parts may need multiple gates.
- Weld lines: where two flow fronts meet (around a hole or boss) the part is weaker and may show a witness line. Position holes and gates so weld lines fall away from load-bearing or cosmetic areas.
- Gate location is a cosmetic and structural decision; tell us the show surface and the load path and we will place the gate to suit.

Tolerances

Molded tolerances are driven by shrinkage, which varies by resin, so they are looser than machined tolerances and resin-dependent. Call out tight tolerances only where a part mates or seals.

Resin	Achievable on a small feature
ABS	+/- 0.05 to 0.1 mm (0.002 to 0.004 in)
PP	+/- 0.075 to 0.15 mm (0.003 to 0.006 in)
Nylon (PA6)	+/- 0.1 to 0.2 mm (0.004 to 0.008 in)

Glass-filled grades shrink less but warp anisotropically, so treat their tolerances as grade-specific rather than assuming they are simply tighter.

The DFM checklist

Before you send a model to quote, confirm:

1. Nominal wall within the resin range, held uniform, thick-to-thin transitions kept under 3 to 1.
2. Draft on every ejection face, matched to the surface finish and draw depth.
3. Ribs at or below 60 percent of wall, bosses connected and correctly proportioned.
4. Internal corners radiused; thick masses cored out to prevent sink.
5. Undercuts removed where possible, remaining ones flagged for side actions.
6. Flow length within single-gate reach or multi-gating planned; weld lines steered away from critical areas.
7. Tight tolerances only on mating and sealing features.

How FabWorks checks this for you

When you upload a model, our DFM engine measures wall thickness and uniformity, checks draft against the intended finish, evaluates rib and boss proportions, detects undercuts and sink risk, and estimates fill and weld-line behavior, per resin, before you receive a quote. It tells you where a wall will sink, where a face lacks draft, or where an undercut will drive tooling cost, and why. Each result is grounded in the published reference behind it, so the guidance is traceable. The outcome is a part designed to mold cleanly, so tooling quotes faster and first-article parts come back consistent.