Design for Manufacturing (DFM) checklist

43 checks for moulded, sheet metal and machined parts, and for assembly. Numbers are published starting points; your supplier's limits win.

Skill-Lync · skill-lync.com/tools/dfm-checklist Version 16 Nov 2026

Part / assembly:	Reviewed by:	Date:

1. Every part, before the drawing is released

Checks that apply whatever the process. Most cost is locked in by these decisions, long before anyone cuts metal.

- [ ] The process suits the quantity: tooled processes (moulding, die casting, stamping) for volume; machining, 3D printing or laser cutting and bending for prototypes and small batches.
  A mould or die is a large one-time cost that only pays back when it is spread over many parts.
- [ ] Tight tolerances only on functional features; everything else falls under a general tolerance note (for example ISO 2768-1 class m).
  Every tight tolerance adds inspection and often an extra operation. Most features never needed it.
- [ ] Datums and GD&T callouts are on the features that locate and mate, and they match how the part is fixtured and inspected.
- [ ] Standard materials and stock sizes (sheet gauges, bar and plate sizes, common resin grades) that suppliers keep in stock.
- [ ] One fastener type and size wherever possible, from the standard catalogue, not a special.
- [ ] The supplier or toolmaker has done a DFM review of the 3D model before the drawing is frozen.
  Shops publish their own limits; theirs beat any generic rule, including the ones on this page.

2. Injection-moulded plastic parts

Most moulding defects (sink marks, warp, short shots, parts stuck in the tool) trace back to wall thickness, draft and thick spots.

- [ ] Nominal wall is uniform and inside the resin's range: ABS 1.14–3.56 mm, polycarbonate 1.02–3.81 mm, polypropylene 0.89–3.81 mm, nylon 0.76–2.92 mm. [3]
- [ ] Where the wall must change, it tapers over a length of at least 3× the change in thickness, never a step. [7, 6]
- [ ] Every face parallel to the mould opening has draft: 0.5° minimum, 2° works for most parts, 3° or more for light texture and shut-offs. [3]
- [ ] Textured faces get extra draft: start from 1° and add 1° for every 0.025 mm (0.001 in) of texture depth. [6]
- [ ] Ribs are no thicker than 60% of the wall they stand on (40% on glossy show surfaces), no taller than 3× their own thickness, and spaced at least 2.5–3× the wall apart. [7, 5]
- [ ] No sharp corners: inside radius at least 0.5× the wall (1× where the corner carries load), outside radius = inside radius + wall. [5, 6]
  Stress concentration rises steeply below an inside radius of half the wall; the matching outside radius keeps the wall constant round the corner.

- [ ] Screw bosses: wall 60–70% of the adjacent wall or less, outside diameter about 2.0–2.4× the screw or insert diameter, tied to walls with ribs or gussets instead of merged into them. [4, 5, 6]
- [ ] No undercuts unless they earn their cost: every side action or lifter adds tooling cost and lead time. Check whether a pass-through core or a snap-fit redesign removes it.
- [ ] Tolerances allow for shrinkage: about ±0.08 mm plus at least ±0.002 mm per mm of feature length (±0.28 mm on a 100 mm dimension). [3]
- [ ] Show surfaces are marked on the drawing so the toolmaker keeps gates, ejector marks and knit lines off them.

3. Sheet metal parts

Sheet metal goes wrong at the bends: flanges too short for the press brake, holes that stretch into ovals, corners that tear.

- [ ] One sheet thickness for the whole part, in a stock gauge.
- [ ] One inside bend radius for every bend, picked from the shop's tooling (Protolabs' standard set is 0.76, 1.52, 2.29 and 3.05 mm). Safe minimums are roughly 0.5–1× thickness on formable alloys, larger on hard tempers and across the grain. [1, 9]
- [ ] Flanges are long enough for the press brake to grip: between 2.5× and 4× thickness plus the bend radius, depending on material and gauge (use the calculator below). [2]
- [ ] Holes and slots are at least as wide as the sheet is thick. [1]
- [ ] Holes sit at least 1.57 mm from an edge in sheet up to 0.91 mm thick, and at least 3.18 mm in thicker sheet. [1]
- [ ] Holes, slots and other features keep clear of bends by the same multiplier as the flange length; anything within 4× thickness of a bend is at risk of distorting. [2, 1]
- [ ] A bend relief is cut wherever a bend ends against flat material, so the corner does not tear (ask the shop for its relief size). [9]
- [ ] Notches are at least the thickness (and at least 1.02 mm) wide, tabs at least 2× the thickness (and at least 3.2 mm) wide; neither longer than 5× its width. [1]
- [ ] Slots stay below a 5:1 length-to-width ratio for laser cutting and 3:1 for punching. [9]
- [ ] Hems have an inside diameter of at least the thickness and a return length of about 6× the thickness. [1]
- [ ] Tolerances respect bending: ±1° on bend angles, and dimensions that cross bends loosen with every bend (in sheet under 3.3 mm: ±0.25 mm to a formed feature, ±0.38 mm across 2 bends, ±0.76 mm across 3 or more). [1]

4. CNC machined parts

Machining cost is driven by the number of setups, the number of tools, and how long and thin those tools have to be.

- [ ] Inside vertical corners have a radius of at least ⅓ of the pocket depth. A sharp inside corner cannot be milled. [8]
- [ ] Pockets are no deeper than 4× their width. [8]
- [ ] Walls are at least 0.8 mm thick in metal and 1.5 mm in plastic (0.5 mm and 1.0 mm are possible, at a price). [8]

☐ Holes use standard drill sizes and are no deeper than 4× the diameter (10× is common, beyond that needs special tools). [8]	
☐ Threads are M6 or larger where possible, with a thread length of 1.5–3× the diameter. [8]	
☐ Pocket floors use a standard radius (0.5 mm, 1 mm) or none. [8]	
☐ Features are reachable from as few directions as possible, ideally one or two setups.	
☐ Standard tolerance is about ±0.1 mm; ±0.02 mm is possible but only where the function needs it. [8]	
☐ Text is engraved rather than embossed, at least 20 pt. [8]	

5. Assembly (DFA)

Design for assembly cuts parts, fasteners and handling time. Boothroyd and Dewhurst's three questions are the core of it.

☐ Each part passes at least one test, or it is a candidate to merge or delete: (1) it moves relative to the parts already assembled; (2) it must be a different material or isolated from them; (3) it must be separate so other parts can be assembled or serviced. [10]	
☐ Fasteners are minimised: snap fits, press fits or features moulded into the part replace screws where the joint does not need to come apart.	
☐ The product builds up on one base part, with parts added from one direction (top-down).	
☐ Parts guide themselves into place: chamfers, lead-ins and locating features, no holding while fastening.	
☐ Parts are symmetric, or obviously asymmetric, so they cannot go in the wrong way round (mistake-proofing).	
☐ Parts are easy to grip and do not tangle, nest or stick together; flexible parts and loose cables are avoided where possible.	
☐ There is room for hands and tools at every joint, and service parts come out without dismantling everything else.	

Sources (checked 16 Nov 2026)

1. Protolabs, Design Guidelines for Sheet Metal Fabrication — https://protolabs.com/services/sheet-metal-fabrication/design-guidelines
2. Protolabs, Design for Sheet Metal Fabrication Toolkit (flange and feature-to-bend tables) — https://www.protolabs.com/resources/design-for-sheet-metal-fabrication-toolkit/
3. Protolabs, Injection Molding Design Guidelines — https://protolabs.com/services/injection-molding/plastic-injection-molding/design-guidelines
4. Protolabs, Design Better Screw Bosses on Molded Parts — https://protolabs.com/resources/blog/design-better-screw-bosses-on-molded-parts
5. Stratasys Direct, Injection Molding Design Guidelines (2017) — https://stratasys.com/siteassets/sdm/resources/design-guidelines/injection-molding/injection_molding_design_guidelines_2017.pdf
6. Fathom, Design Guidelines for Injection Molding (rev. 05152025) — https://fathommfg.com/app/uploads/2025/12/Design-Guidelines-for-Injection-Molding.pdf
7. Xometry, Plastic Ribs for Injection Molding Design — https://www.xometry.com/resources/injection-molding/plastic-ribs-for-injection-molding-design/
8. Hubs, How to design parts for CNC machining — https://www.hubs.com/knowledge-base/how-design-parts-cnc-machining/
9. Fabcon, Sheet Metal DFM Guide: Rules, Failure Modes and Review — https://fabcon.com/articles/?p=469
10. Boothroyd, Dewhurst and Knight, Product Design for Manufacture and Assembly (3rd ed., CRC Press, 2011)

Interactive version with calculators for flange length, rib and boss sizes, draft and CNC corner radii: skill-lync.com/tools/dfm-checklist. Learn DFM and DFA on industry projects: skill-lync.com/medical-technology-courses/design-manufacturability-dfm