MechPart Pro

B2B Precision Manufacturing · Technical White Paper

ENGINEERING

Design for Manufacturability: A Practical Handbook

Reduce cost and lead time before the first chip is cut — wall thickness, radii, tolerancing strategy, feature consolidation and process-specific rules across CNC, sheet metal, casting and molding.

What's inside

1. Why DFM pays

Most of a part's cost is locked in at the design stage. The geometry decides how many setups it needs, which tools can reach each feature, how slowly the machine must run and how much hand-finishing is required. Design-for-manufacturability (DFM) shapes a part so it is easy to hold, cut and inspect without compromising function — and small changes often yield large savings.

2. Universal rules

Feature	Rule of thumb	Why
Internal corner radius	>= 1/3 of cavity depth	Lets a rigid standard tool reach it
Pocket depth	<= ~4x tool diameter	Limits tool deflection and chatter
Wall thickness (metal)	>= 0.8 mm	Resists vibration and distortion
Wall thickness (plastic)	>= 1.5 mm	Compensates low rigidity
Hole depth	<= ~4-6x diameter	Avoids special deep-hole tooling
Thread depth	~1.5-2x diameter	Full strength, less tap breakage
General tolerance	ISO 2768-m unless function needs tighter	Tight callouts only where they matter

3. Tolerancing strategy

A general tolerance such as ISO 2768 medium is effectively free because the machine holds it naturally. Every callout tighter than that adds inspection, slower finishing passes and scrap risk. Reserve tight tolerances for the few features that mate or seal, and dimension from a single datum to control stack-up across an assembly.

- Tighten only mating, sealing and bearing features.
- Prefer GD&T; position over coordinate tolerances for holes.
- Dimension from one datum edge, not chained feature-to-feature.
- State surface finish only where it functions.

4. Process-specific guidance

CNC machining

- Maximise and standardise internal radii; minimise the number of setups.
- Avoid deep narrow pockets, thin tall walls and inaccessible undercuts.
- Engrave (recess) rather than emboss text; choose a machinable alloy.

Sheet metal

- Inside bend radius >= material thickness; standardise one radius per part.
- Keep holes 2-3x thickness away from bends; add bend reliefs.
- Minimum flange ~4x thickness so it seats on the brake die.

Injection molding & casting

- Keep walls uniform; core out thick sections and replace with ribs.
- Add draft (>= 1 degree) and fillets at junctions.
- Ribs ~50-60% of wall thickness to avoid sink marks.

5. A practical DFM checklist

- Walls within the process minimum, and uniform where molded or cast.
- Internal radii generous and consistent; pockets not excessively deep.
- Holes standard-size and reasonable depth; threads to sensible depth.
- Tolerances at ISO 2768-m except where function demands tighter.
- Surface finish specified only on functional faces.
- Material chosen for both function and manufacturability.
- A complete 3D model plus a 2D drawing of the controlled dimensions.

MechPart Pro provides free DFM feedback with every quote across CNC, sheet metal, casting, forging and molding — flagging thin walls, hard-to-reach features and over-tight tolerances before production. Upload your model at mechpart.io/?page=quote.