

DFM DESIGN GUIDE

Plastic Injection Molding — quick reference for design engineers

WALL THICKNESS

- Uniform walls 1.5–3.5 mm for most engineering plastics; avoid abrupt transitions (3:1 runout).
- Thick sections sink — core them out.
- Rib thickness $\leq$ 60% of adjoining wall.

DRAFT ANGLE

- 0.5–1° minimum on smooth faces; 1–2° minimum with VDI/Mold-Tech texture.
- Deep cores for box parts: add draft + texture to fight vacuum during ejection.

TOLERANCES

- Standard: ± 0.1 mm Fine: ± 0.05 mm Tight (small features): ± 0.02 mm.
- Apply tight tolerances to functional interfaces only — every callout adds inspection cost.

GATES, WELD LINES & COSMETICS

- Gate on non-cosmetic faces where possible; discuss gate location before tooling starts.
- Mark cosmetic faces A / B / C and define the inspection angle and light source.
- Text: engraved 0.25–0.4 mm deep with 2° side draft per stroke.

UNDERCUTS & THREADS

- Sliders/lifters add tooling cost and maintenance — confirm necessity early.
- Prefer metal inserts (heat-stake / ultrasonic) over molded-in threads in hard plastics.

MATERIAL SELECTION QUICK LIST

- Housing / cosmetics: ABS, PC/ABS • Tough & transparent: PC • Gears & sliding: POM
- Structural / metal replacement: PA6-GF30, PEEK • Medical: PC, PEEK, PSU (USP Class VI options)

VOLUMES

- Prototyping: CNC (3–10 days) • $\leq$ 1,000 pcs: bridge tooling • 5,000+: production mold.
- Design tooling steel & hot runner for the real annual volume — honesty is the cheapest cost-down.