The 10-Point CNC DFM Checklist

A free pre-quote checklist from Global Precision Works. Run it on any machined part before you release the drawing. Design locks in 70–80% of a part's cost before a supplier ever quotes it — so the cheapest changes are the ones you make on the model, not at the negotiating table.

The 70–80% figure is the founding premise of Design for Manufacture & Assembly (Boothroyd & Dewhurst), corroborated by peer-reviewed cost-engineering research; tolerance figures from machining cost-engineering references. Scope: parts machined from wrought stock.

THE CHECKLIST

- ☐ **1 Default the tolerances**
  Whole part at ±0.005″ / ISO 2768-m, with only functional features called tighter.
- ☐ **2 Count the tight callouts**
  Justify each sub-±0.001″ dimension by a fit, seal, or location — or loosen it.
- ☐ **3 Radius the internal corners**
  Inside corners radiused to fit a real tool, not drawn sharp (a round end mill can't cut a sharp corner).
- ☐ **4 Check pocket depth**
  No cavity deeper than ~3–4× the tool diameter that has to reach the bottom.
- ☐ **5 Count the setups**
  Fewer orientations = lower cost. Group features onto fewer faces where function allows.
- ☐ **6 Check wall thickness & spacing**
  No wall — or web between a hole and a nearby edge or feature — thin enough to chatter, deflect, or break out under cutting load. Add ribs for stiffness.
- ☐ **7 Standardize holes & threads**
  Standard drill sizes and thread series; tap no deeper than the thread engagement the bolted joint requires.
- ☐ **8 Right-size the material**
  Cheapest alloy that meets the requirement; starting stock close to the finished envelope.
- ☐ **9 Specify finish only where it matters**
  Fine surface finish called out on functional/cosmetic faces only, not blanket across the part.
- ☐ **10 Confirm datums & GD&T are defined by function**
  Datums and tolerances reflect how the part mounts and functions — and reference features the shop can actually fixture and inspect from.

TOLERANCE CHEAT-SHEET — A TOLERANCE IS A BUDGET

TOLERANCE BAND	RELATIVE COST	WHEN TO SPEND IT
±0.005″ (0.13 mm) — standard	**1× (no premium)**	General / non-critical
±0.001″ (0.025 mm) — precision	**~2–4×**	Locating, light press fits
±0.0005″ (0.013 mm) — tight	**~5× and up**	Bearing seats, sealing faces

Send us your print. Free DFM review before we quote.

An engineer reviews your model for tolerance risk, setup count, material fit, and cost-driving geometry — then returns a line-item quote you can compare directly against your current supplier.

Request a Quote →

Global Precision Works — precision machining & assembly, Monterrey, Mexico **Quote:** gpw-solutions.com/request-a-quote **Email:** sales@gpw-solutions.com

Call: +1 (469) 949-6321