

# Bay Engineering – Tolerance Guidelines for CNC Machining

These guidelines help customers choose practical, cost-effective tolerances for CNC machined parts. Tighter tolerances are achievable, but they typically increase machining time, inspection effort, and cost. If your drawing specifies different requirements, the drawing takes precedence.

## General tolerances (typical)

Unless otherwise specified, the tolerances below are typical starting points for CNC milling/turning of stable materials. Actual capability depends on part size, geometry, datum scheme, material, heat treatment, and finishing operations.

Dimension range (mm)	Typical linear tolerance
0–10	±0.05 mm
>10–50	±0.10 mm
>50–150	±0.20 mm
>150–300	±0.30 mm
>300 (large parts)	By agreement / assessment

**Guidance:** As a rule of thumb, specifying tighter than ±0.05 mm on general features, or requiring tight tolerance across multiple features without clear datums, will usually add cost. Where only certain features are critical, tolerancing those features only is often the most economical approach.

## Feature-specific guidance

- **Holes (drilled):** Diameter is typically less accurate than reamed/ground bores. If a close fit is required, specify a ream/boring operation and the target tolerance.
- **Holes (reamed/precision bored):** Commonly used for dowels/bearing fits; capability depends on diameter/depth and material.
- **Threads:** Standard threads are produced to the relevant class (e.g., metric 6H internal / 6g external unless otherwise stated). Call out thread class and any special requirements (depth, relief, gauging).
- **Slots & pockets:** Internal corner radii are limited by cutter diameter. Specify minimum internal radius or allow a tool radius where possible.
- **Sharp edges:** Unless specified, edges may be broken/deburred (e.g., 0.2–0.5 mm). If a sharp edge is functional, call it out.

## Geometric tolerancing (GD&T) and datums

If you require tight control of form, orientation, or location, GD&T is often clearer than tightening multiple  $\pm$  tolerances. A well-defined datum scheme also reduces inspection ambiguity.

- **Datums:** Identify functional datum faces/features (A/B/C) and ensure they are machinable and inspectable.
- **True position:** Use position tolerances for hole patterns rather than chaining linear tolerances.
- **Runout/concentricity:** For turned parts, specify runout to a datum axis when controlling rotating features.
- **Inspection method:** If a feature requires a specific gage, CMM strategy, or functional measurement, state it on the drawing.

## Surface finish and secondary processes

- **Surface roughness (Ra):** If finish is critical, specify the Ra value and the surfaces it applies to.
- **Anodising/plating/painting:** Coatings add thickness and can affect fits. If a fit is critical, state whether dimensions are **pre-** or **post-**finish.
- **Heat treatment:** Can introduce distortion. For tight tolerances after heat treat, allow for finish machining/grinding.
- **Grinding/honing:** Typically used where very tight tolerances or fine finishes are required—please highlight these features when quoting.

## What to include with your RFQ

- 3D CAD model and/or dimensioned drawing (PDF preferred for drawings)
- Quantity (prototype, batch size, annual demand if applicable)
- Material grade/specification and any certification requirements
- Surface finish/coating requirements and whether dimensions are pre/post finish
- Any critical-to-function features (fits, sealing surfaces, bearing seats, etc.)

**Important:** These values are guidelines only. If you have a specific standard you need us to work to (e.g., ISO 2768, customer-specific tolerances, or inspection reporting), please state this at enquiry stage.