

Most CNC machining cost and quality problems are created before a machine is programmed. These ten principles cover the specific design decisions — tolerances, setups, corner geometry, wall thickness, holes, undercuts, surface finish, materials, datums, and threading — that consistently affect machinability, repeatability, and total production cost. A tolerance change at the drawing stage costs nothing. The same change after first article costs significantly more.

1

Principle 1

Specify Only the Tolerances the Application Requires

A reasonable general tolerance for CNC machined features is $\pm 0.005"$ (± 0.127 mm) for most non-critical dimensions — a starting point consistent with standard commercial CNC practice. Every tolerance tighter than this range may require different process approaches: different workholding strategies, tighter thermal control, alternative finishing sequences, or more intensive inspection.

Before specifying a tight tolerance, confirm the functional consequence of being at the edge of that tolerance band. If the part still functions at the limit, the tolerance may be unnecessarily tight.

"If you cannot state the functional consequence of being at the limit of the tolerance, question whether the tolerance is necessary."

GD&T is most useful where functional feature relationships, datum structures, orientation, and location all need to be controlled simultaneously. It communicates functional intent more precisely than coordinate tolerancing in those situations — but it is not categorically superior for every part.

2

Principle 2

Design for the Fewest Possible Setups

Each time a part is repositioned in a machine, variation can accumulate and labor cost increases. Features reachable from the same direction should be grouped to complete in one setup where possible.

5-axis machining can complete complex geometry in fewer setups than 3-axis machining. The specific reduction depends on part geometry, but consolidating setups through machine capability is a legitimate cost-reduction strategy worth evaluating during design.

When multiple setups are unavoidable, design clear reference surfaces into the part that support consistent repositioning. A part re-established consistently from controlled locating features is substantially easier to hold to tolerance across operations.

At tight tolerances, reducing setups may become less about machine capability and more about controlling dimensional risk.

3

Principle 3

Specify Internal Corner Radii Larger Than the Tool Radius

Internal corners on machined parts cannot be perfectly sharp. A rotating end mill leaves a minimum radius equal to its own radius. However, specifying a corner radius exactly equal to the tool radius forces the cutter into full radial engagement — increased cutting forces, heat generation, and deflection risk, and the least efficient cutting condition.

A practical starting point: specify internal corner radii at approximately 1.25× to 1.5× the cutter radius; larger radii are preferred where geometry allows. For a pocket machined with a 1/4" (0.250") end mill, the cutter radius is 0.125". Specifying a corner radius of 0.156"–0.188" allows the tool to clear the corner cleanly.

End mills are available from below 1/8" for fine detail work through 1" and larger for heavy roughing. Present the tool range to your machining partner early if internal geometry is complex.

4

Principle 4

Observe Minimum Wall Thickness Guidelines

Thin walls flex under cutting forces, causing chatter, dimensional variation, and potential damage. The following are starting-point guidelines — actual minimums vary with wall height, material condition, geometry, cutter engagement, and tolerance required:

- Aluminum: 0.025"–0.040" for short, well-supported walls; progressively thicker for taller unsupported sections
- Steel: 0.050"–0.060" as a reasonable starting point for most geometries
- Titanium and nickel alloys: Higher minimums apply due to substantially greater cutting forces

The ratio of unsupported height to wall thickness matters as much as the absolute dimension. Minor geometric changes can sometimes improve stability without changing the functional envelope.

5

Principle 5

Consider Hole Depth-to-Diameter Ratios in Tooling Selection

As a planning reference for conventional tooling, hole depths of approximately 3× to 5× the drill diameter are typical before chip evacuation and heat management become more demanding. The appropriate method depends on the drill type, material, diameter, coolant delivery, and chip formation characteristics — not on a single depth threshold.

Modern carbide drills with through-tool coolant can often be run continuously at substantially greater depth-to-diameter ratios. Specialized gun drills are designed for very deep holes — 10× diameter and beyond — in suitable applications.

Where possible, specify through-holes rather than blind holes. Through-holes are easier to drill, easier to inspect, and eliminate blind-hole depth tolerance as a variable.

6

Principle 6

Avoid Undercuts Unless Functionally Required

An undercut is any feature that cannot be reached by a standard end mill moving axially. Undercuts require alternative tooling: T-slot cutters, lollipop (back-turning) cutters, form tools, or additional setups repositioning the part to access the feature from a different angle.

Each of these adds cost disproportionate to the feature's typical size. Where an undercut is genuinely required, specify it explicitly and verify the geometry is compatible with available standard tooling. Custom form tools add lead time and cost.

In many cases, an undercut is present because a design was carried over from a casting or a different manufacturing process. Review whether the feature is still functionally required in the machined version.

7

Principle 7

Specify Surface Finish by Functional Requirement

Surface finish specifications affect cost. Finer finishes require additional passes, slower feeds and speeds, or secondary operations. The following Ra values (microinches) are directional manufacturing benchmarks, not application prescriptions:

- 125 Ra: A commonly achievable reference finish from conventional milling or turning
- 63 Ra: Improved finish often achievable with careful finish passes; a useful benchmark for many interface surfaces
- 32 Ra: Fine finish, often achievable by conventional finish turning or milling under appropriate conditions
- 16 Ra and below: Frequently requires additional operations such as grinding, lapping, or precision hard turning

Apply surface finish callouts to specific features based on confirmed functional requirements, not as a blanket specification in the title block.

8

Principle 8

Consider Machinability Alongside Function in Material Selection

Material choice drives cycle time, tooling cost, and surface finish achievability. When two materials can satisfy the functional requirement, machinability is a real cost consideration:

- Aluminum 6061-T6: Among the most machinable common alloys. Fast cycle times, good surface finish
- Stainless 303 vs. 316: 303 machines significantly better, but the sulfur addition that improves machinability reduces corrosion resistance. If corrosion resistance is the primary requirement, 316 may be the correct choice
- 4140 steel: Machinable in commonly machined conditions; machinability changes materially at higher hardness
- Titanium Ti-6Al-4V and nickel superalloys (Inconel, Hastelloy, Waspaloy): Substantially higher machining costs. Understanding the implications early avoids cost surprises

9

Principle 9

Define Datum Features Based on Function and Accessibility

Datum features are the reference framework for fixturing during machining and measurement during inspection. Effective datum features are not limited to flat surfaces — they may be flat surfaces, cylindrical features (bores or pins), slots, or patterns, each establishing a datum through physical contact or derived geometry such as an axis or plane.

The primary driver of datum selection should be functional design intent. Accessibility for fixturing and inspection is a secondary but important practical consideration. When possible, machine datum features and the features referenced to them in the same setup.

A brief conversation with your machining partner about datum accessibility early in design often prevents problems at first article.

10

Principle 10

Use Standard Thread Forms and Specify Inserts for High-Cycle Assemblies

Non-standard thread forms and pitches require custom taps and inspection gauges and complicate lead times. Standard UNC, UNF, and metric thread forms use off-the-shelf tooling and standard gauges.

- Coarse threads (UNC): Preferred for rapid assembly, tolerance of handling damage, use in soft materials, and resistance to cross-threading
- Fine threads (UNF): Offer tighter preload control; due to smaller helix angle, can be more resistant to vibrational loosening in some applications

In aluminum or soft materials subject to repeated assembly, specify thread inserts (Helicoil, Keensert, or equivalent). A damaged insert can often be replaced without scrapping or reworking the parent component.

Thread engagement depth is application-dependent. A common starting-point guideline for steel fasteners into aluminum is 1.5× to 2× the thread diameter, but appropriate engagement depends on alloy, thread class, loading, and fastener analysis.

About Borg Design

Borg Design, Inc. is a fourth-generation precision CNC machining company in Hudson, Massachusetts, founded in 1945. Registered mechanical engineers on staff. DFM and SolidWorks engineering support. In-house CMM inspection. Prototype through production across aerospace, defense, medical device, industrial, and automotive programs. 58,000 sq. ft. | 50+ precision machines | CAGE Code: 1V3P1