



CNC MACHINING

DESIGN GUIDE

Contents

1. Introduction	1
2. CNC Machining Fundamentals	1
2.1 What Is CNC Machining	1
2.2 CNC Milling	2
2.3 CNC Turning	8
3. Materials for CNC Machining	13
3.1 Key Factors in Material Selection	13
3.2 Common Materials for CNC Machining	14
3.3 Material Selection Guidelines	16
4. Design for CNC Machining	17
4.1 Pockets and Cavities	17
4.2 Inner Radii and Edge Features	18
4.3 Wall Thickness	19
4.4 Hole Design	20
4.5 Thread Design	21
4.6 Tool Accessibility	22
4.7 Setup Reduction	23
4.8 Tolerances and Surface Roughness	24
5. Supplier Selection Guide	25
5.1 Costs and Influencing Factors	25
5.2 CNC Machining by Production Volume	30
5.3 CNC Supplier Evaluation	31
6. Conclusion	33

1. Introduction

CNC machining is one of the most widely used manufacturing processes, particularly for metal and engineering plastic parts. From prototype validation and low-volume production to the repeat production of precision components, CNC machining is widely applied in the automotive, consumer electronics, medical device, aerospace, and industrial equipment sectors.

However, the success of a **CNC machining** project does not depend on machine accuracy alone. Part geometry, material selection, machining method, tolerances, surface requirements, workholding strategy, and inspection standards all affect final part quality, manufacturing cost, and lead time.

First Mold draws on years of experience in prototyping and production machining of metal and engineering plastic parts to prepare this guide for procurement professionals, product designers, engineers, and project managers. It is intended to support more informed decisions during design, sourcing, and production and to help convert product designs into conforming finished parts more efficiently.



2. CNC Machining Fundamentals

2.1 What Is CNC Machining

CNC stands for Computer Numerical Control. It is a subtractive manufacturing process in which pre-programmed instructions control the movement of the cutting tool or workpiece. Starting from plate, bar, block, casting, or forging stock, excess material is removed through milling, turning, drilling, boring, reaming, or tapping to produce metal or engineering plastic parts with the specified geometry, dimensions, and surface quality.



Figure 1 : Typical CNC Machining Process

Although CNC machining is highly automated, the final result still depends on the machining program, machine condition, cutting tools, workholding, material properties, and inspection method. Consequently, the same design file may produce different results in accuracy, surface quality, cost, and lead time when manufactured by different suppliers.

2.2 CNC Milling

CNC milling is one of the most common CNC machining processes. It is suitable for producing planes, holes, slots, pockets, external profiles, and complex surfaces. Unlike turning, in which cutting is primarily achieved by rotating the workpiece, milling generally removes material from a stationary workpiece with a rotating cutting tool.

Depending on machine configuration, machining centers may be vertical, horizontal, or gantry-type. These machines differ in travel, spindle orientation, rigidity, chip evacuation, and the range of part sizes they can accommodate.

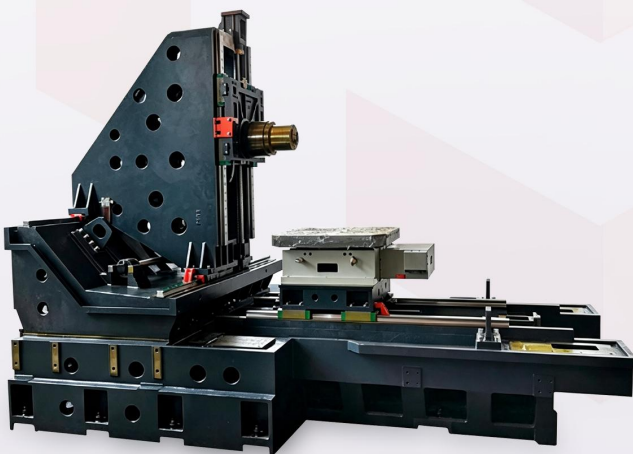


Figure 2 : Horizontal Machining Center

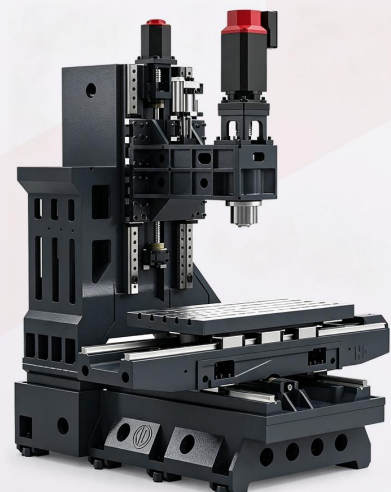


Figure 3 : Vertical Machining Center

2.2.1 Working Principle

During CNC milling, the workpiece is secured with a vise, clamps, a zero-point clamping system, or a dedicated fixture. The cutting tool is mounted in the spindle and rotates at high speed. The machine reads a program generated by a CAM system and controls its motion axes along the prescribed toolpath, allowing the cutting edges to engage the workpiece continuously and remove material.

A typical milling process generally includes roughing, semi-finishing, and finishing.

- Roughing rapidly removes most of the excess material.
- Semi-finishing leaves a consistent allowance for finishing.
- Finishing produces the final dimensions, geometric accuracy, and surface quality.

Some operations may be combined for simple parts or less demanding requirements. High-precision parts, deep cavities, thin walls, or complex surfaces may require additional steps such as stress relief, secondary setups, local finishing, or compensation based on inspection results.

Tool selection and cutting parameters jointly affect material removal efficiency, tool life, machining stability, and final surface quality. Common milling parameters include cutting speed, spindle speed, feed per tooth, table feed rate, axial depth of cut, and radial width of cut. Table feed rate can be expressed as:

$$V_f = Z_n \times n \times f_z$$

Where:

- V_f = Table feed rate (mm/min)
- Z_n = Number of cutting teeth
- n = Spindle speed (r/min)
- f_z = Feed per tooth (mm/tooth)



2.2.2 Common Milling Operations

CNC milling can produce different part features by changing the tool type, cutting direction, and toolpath. The most common representative operations include face milling, profile milling, slot milling, pocket milling, and surface milling.

- **Face milling** uses a face mill to machine relatively large flat surfaces, such as datum surfaces, assembly surfaces, and sealing faces. The resulting flatness and surface quality can affect the positional accuracy of features machined in subsequent setups.

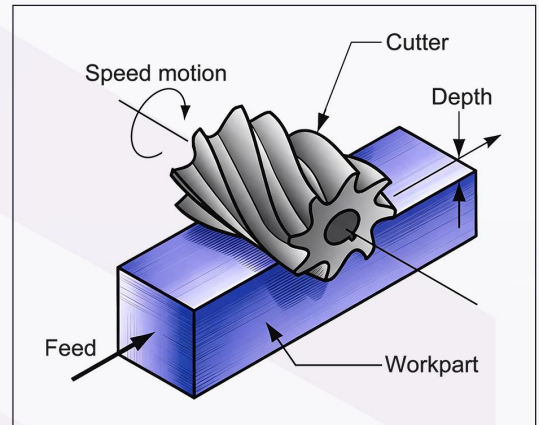


Figure 4 : Face Milling

- **Profile milling** follows the internal or external boundary of a part to produce sidewalls, steps, and irregular profiles. Parts with tighter dimensional or surface requirements are typically roughed with an appropriate allowance and then finished to specification.

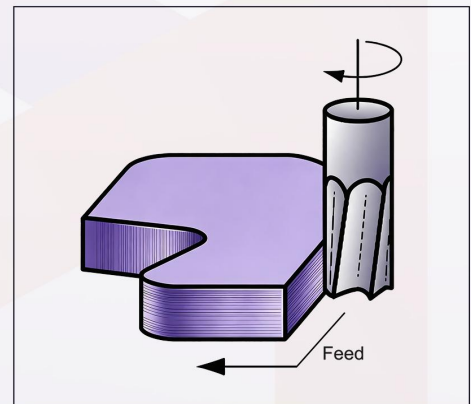


Figure 5 : Profile Milling

- **Slot milling** produces straight slots, keyways, and other narrow features. Tool selection and machining stability depend on slot width, depth, bottom geometry, and chip evacuation. Deep slots require greater tool overhang, increasing the risk of deflection and vibration.

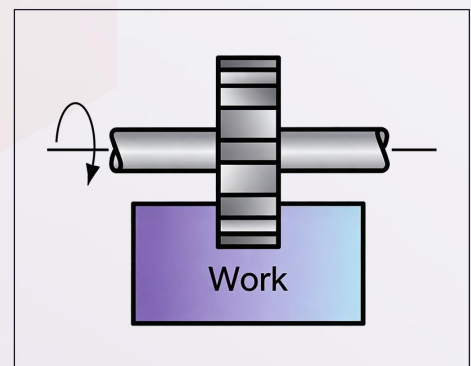


Figure 6 : Slot Milling

- **Pocket milling** removes material in layers to form pockets, recesses, and mold cavities, typically through roughing, corner cleanup, and finishing. Deep pockets restrict tool access, holder clearance, chip evacuation, and cooling, requiring careful coordination of tool diameter, overhang, sidewall requirements, and toolpath.

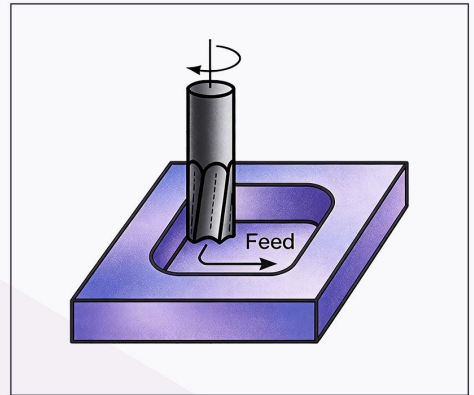


Figure 7 : Pocket Milling

- **Surface milling** produces mold cores, cavities, blades, and other three-dimensional contours, typically using ball-nose or bull-nose end mills. Surface quality depends on tool diameter, stepover, cutting direction, and tool orientation. A smaller stepover reduces cusp height but increases machining time.

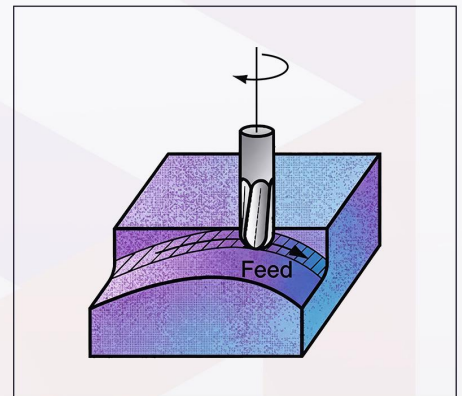


Figure 8 : Surface Milling

In addition to these common milling operations, CNC machining centers can perform drilling, boring, reaming, and tapping according to part features and production requirements. Combining operations appropriately can reduce process transfers and repeated setups while improving machining efficiency and positional accuracy between features.

2.2.3 3-Axis, 4-Axis, and 5-Axis CNC Milling

The axis count of a CNC milling machine refers to the number of controlled motion axes. A 3-axis machine has three linear axes: X, Y, and Z. A 4-axis machine typically adds one rotary axis, while a 5-axis machine combines three linear axes with two rotary axes. Rotary axes change the relative orientation of the tool and workpiece, improving tool accessibility and reducing repeated setups.

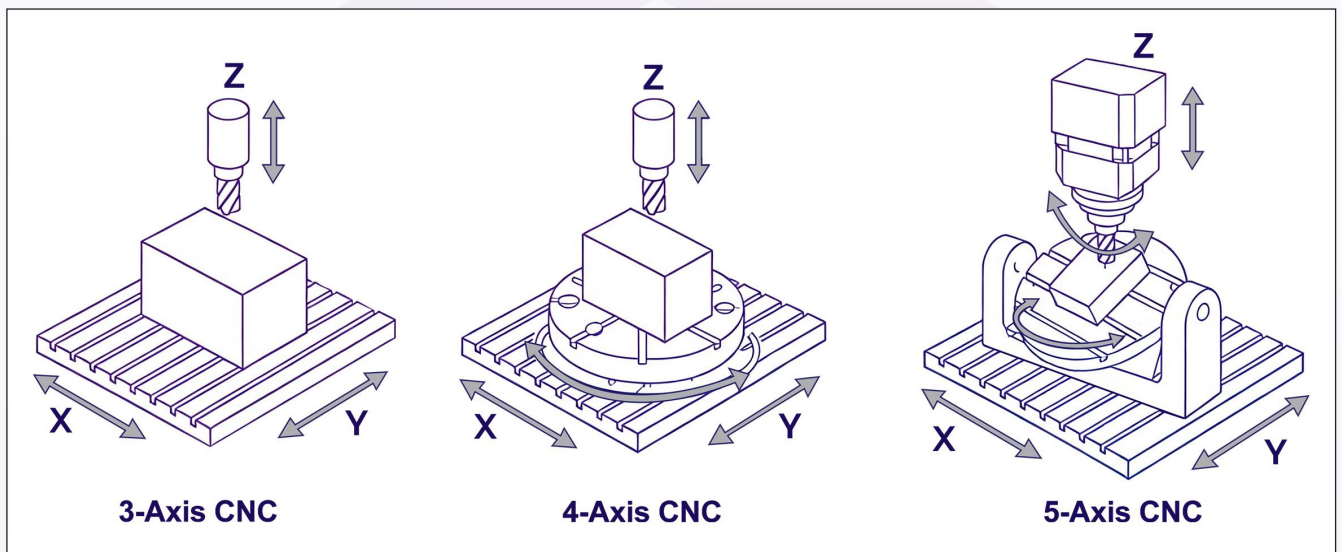


Figure 9 : 3-Axis vs. 4-Axis vs. 5-Axis CNC

• 3-Axis CNC Milling

In 3-axis machining, the tool approaches the workpiece from a fixed orientation during each setup. It is suitable for plates, brackets, housings, cavities, and general mold components with features concentrated in a limited number of directions. Programming and workholding are relatively straightforward, often making it the most economical option for simpler parts. Multi-sided parts generally require additional setups, increasing handling time and the risk of positioning errors.

• 4-Axis CNC Milling

4-axis machining adds one rotary axis to the three linear axes and supports indexed or simultaneous motion. It is suitable for holes, slots, planes, and certain helical features distributed around a common axis. Rotation reduces repeated setups and improves positional consistency between side features. However, tool accessibility remains limited by the single rotary axis.

• 5-Axis CNC Milling

A 5-axis machine uses two rotary axes to adjust the relative orientation of the tool and workpiece. It is suitable for multi-sided parts, inclined features, complex surfaces, and features requiring tight positional accuracy across different surfaces.

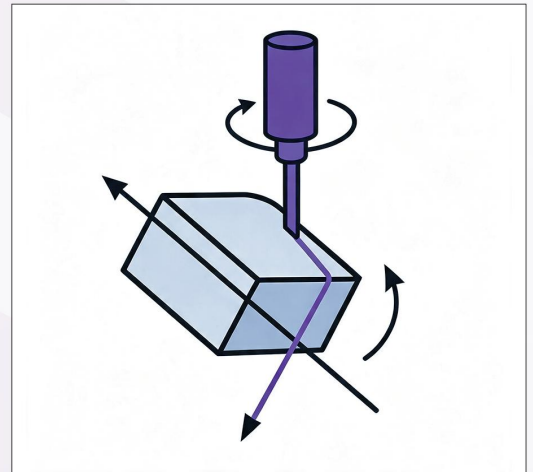


Figure 10 : 3+2 Positioning

5-axis machining includes 3+2 positional machining and simultaneous 5-axis machining. In 3+2 machining, the rotary axes position the tool or workpiece at a specified angle and remain fixed during 3-axis cutting. In simultaneous 5-axis machining, all five axes move in coordination during cutting. Both methods can reduce setups and improve tool accessibility, but require more advanced programming, simulation, collision checking, and machine calibration.

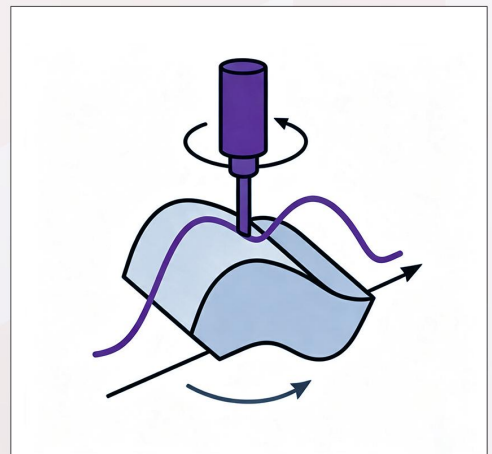


Figure 11 : Simultaneous 5-Axis

Table 1. Comparison of 3-Axis, 4-Axis, and 5-Axis CNC Milling

Comparison	3-Axis CNC Milling	4-Axis CNC Milling	5-Axis CNC Milling
Axis Configuration	X, Y, and Z	X, Y, and Z + 1 rotary axis	X, Y, and Z + 2 rotary axes
Machining Method	Fixed orientation per setup	Indexed or simultaneous 4-axis	3+2 positional or simultaneous 5-axis
Suitable Features	Features concentrated in a few directions	Features arranged around a common axis	Multi-sided, inclined, or complex features
Programming Complexity	Relatively low	Moderate	Relatively high
Main Advantage	Simple and cost-effective	Reduces setups for circumferential features	Greater accessibility with fewer setups
Main Limitation	Multi-sided parts require repositioning	Access is limited by one rotary axis	Higher programming and calibration requirements

2.3 CNC Turning

2.3.1 Working Principle

The basic principle of CNC turning is to rotate the workpiece about the spindle centerline while controlling the cutting tool in the axial and radial directions. As the cutting edge engages the rotating workpiece, material is removed as chips to form the required dimensions and profile.

Z-axis movement at a constant X-position produces cylindrical surfaces, while X-axis movement toward the spindle centerline is used for facing, grooving, and parting-off. Coordinated X- and Z-axis motion produces tapers, arcs, shoulders, and threads. Advanced turning centers may include additional linear or rotary axes.

On a typical 2-axis CNC lathe, tool motion is controlled primarily along the X- and Z-axes:

- Z-axis: Parallel to the spindle centerline; controls axial position and machining length.
- X-axis: Perpendicular to the spindle centerline; controls diameter and radial depth of cut.

The main turning parameters are spindle speed, cutting speed, feed, and depth of cut. Their selection depends on the workpiece and tool materials, part rigidity, and surface requirements. Excessive cutting loads may cause rapid tool wear, workpiece deformation, or chatter, whereas overly conservative parameters can increase machining time and impair chip formation.

Common CNC Turning Formulas:

Feed Rate: $V_f = f_n$

Theoretical Machining Time: $T = \frac{L}{f_n}$

Radial Depth of Cut: $a_p = \frac{D_o - D_f}{2}$

Material Removal Rate: $Q = 1000 V_c f a_p$

Where:

V_f : Feed Rate (mm/min)

f_n : Feed per Revolution (mm/rev)

T : Theoretical machining time (min)

L : Machining length / stroke (mm)

a_p : Depth of cut (mm)

D_o : Initial diameter (mm)

D_f : Final diameter (mm)

V_c : Cutting speed (m/min)

Q : Material removal rate (mm³/min)

2.3.2 Common Turning Operations

CNC turning is not a single cutting action. A complete part generally requires a combination of operations to produce external diameters, faces, internal bores, grooves, and threads. The following are the most common representative turning operations.

• External Turning

External turning removes material from the outside diameter of a workpiece. The tool generally travels parallel to the spindle axis until the blank reaches the specified outside diameter, length, and surface requirements.

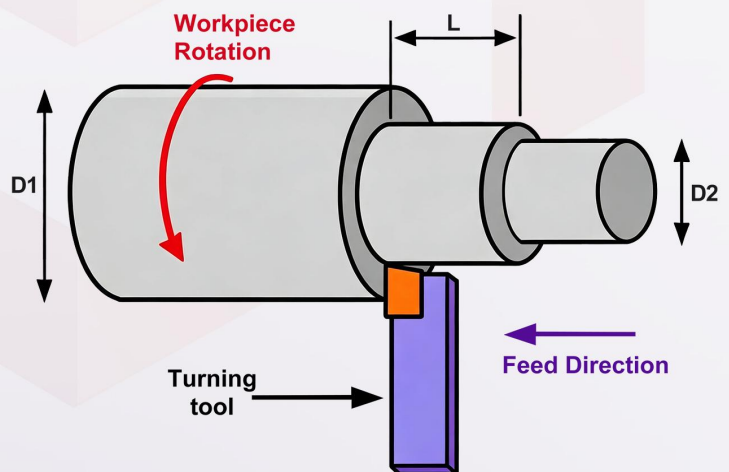


Figure 12 : External Turning

Movement parallel to the axis produces cylindrical surfaces. By changing the toolpath or feed direction, shoulders, tapers, and certain curved profiles can also be produced. According to the amount of material to be removed and the final requirements, external turning generally includes rough turning and finish turning. Rough turning removes the machining allowance, while finish turning produces the final dimensions and surface.

• Facing

Facing produces a plane perpendicular to the workpiece axis of rotation. The tool moves from the outside diameter toward the center, or from the center toward the outside diameter, progressively removing material from the end of the workpiece.

This operation is generally used to establish the overall part length, create an axial machining datum, or produce an assembly or contact face. Face flatness and perpendicularity may affect the positional relationship of subsequent features, so a datum face is commonly machined before other turning operations.

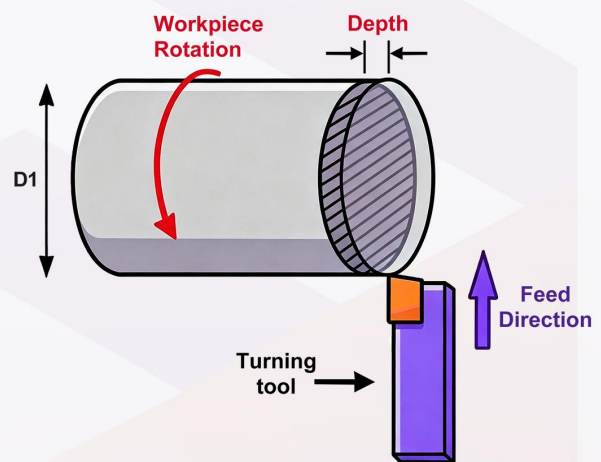


Figure 13 : Facing

• Drilling

Drilling produces an initial hole along the rotational center of the workpiece. The drill generally feeds along the spindle axis into the rotating workpiece. Because the hole position is defined by the spindle center, a conventional CNC lathe primarily produces holes concentric with the workpiece centerline.

Drilling may be used as a standalone operation or to create an initial hole for subsequent boring, reaming, or internal threading. For deeper holes, drill drift, chip evacuation, and cooling must be controlled to prevent diameter error, surface damage, or tool overheating.

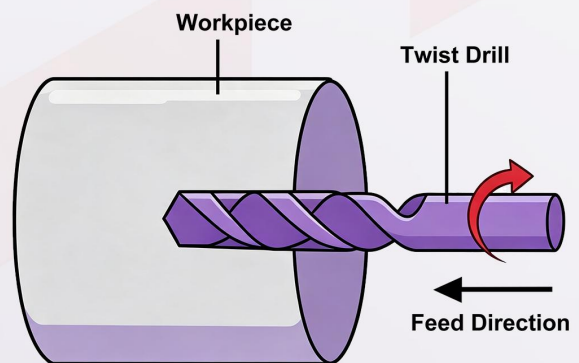


Figure 14 : Drilling

• Boring

Boring uses a single-point cutting tool to enlarge or finish an existing hole. It is primarily used to control internal diameter, coaxiality, cylindricity, and bore surface quality. Compared with drilling, boring can further correct hole position and size, but it cannot produce an initial hole in solid material.

During internal machining, the boring bar must enter the workpiece. A small bore or high depth-to-diameter ratio restricts the available boring-bar diameter and rigidity, increasing the risk of deflection and chatter. Deep-hole boring therefore requires consideration of bore diameter, depth, tool overhang, and chip evacuation.

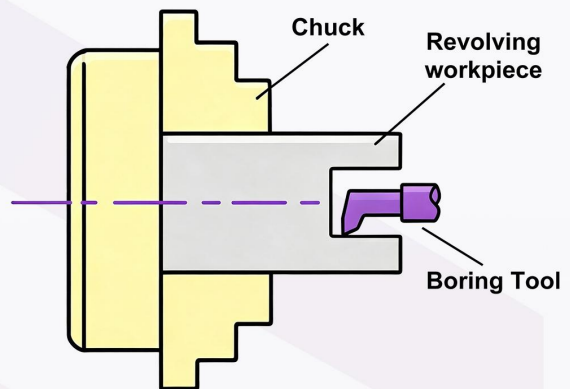


Figure 15 : Boring

• Grooving and Parting-Off

Grooving produces an annular groove of specified width and depth on an outside diameter or internal bore, such as an O-ring groove, retaining-ring groove, or relief groove. A narrow cutting tool feeds radially into the rotating workpiece and may also move slightly in the axial direction to increase groove width.

Parting-off uses a tool and feed method similar to grooving, but the tool continues toward the workpiece center until the finished part is separated from the bar stock. As the tool approaches the center, cutting speed decreases and clearance on both sides of the tool becomes more limited. Chip evacuation, tool rigidity, and workpiece support therefore directly affect process stability.

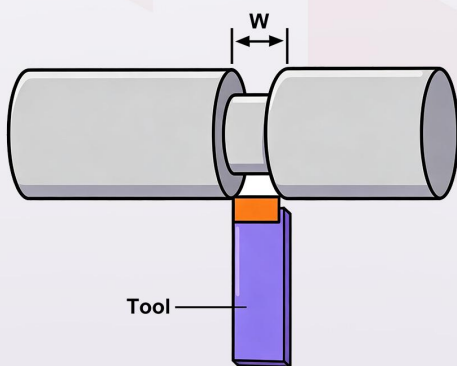


Figure 16 : Grooving

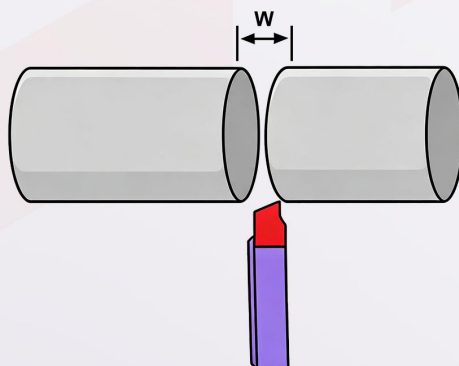


Figure 17 : Parting-Off

• Thread Turning

Thread turning precisely coordinates spindle rotation with the axial feed of the cutting tool to produce a helical groove on an outside diameter or internal bore. The axial travel per revolution determines the thread lead, while the tool geometry determines the thread form.

Threads generally require multiple passes to reach the specified depth. The thread standard, nominal diameter, pitch, direction, tolerance class, and runout space should be defined before machining. Threads close to a shoulder or the bottom of a blind hole also require an appropriate relief groove to prevent interference between the tool and adjacent geometry.

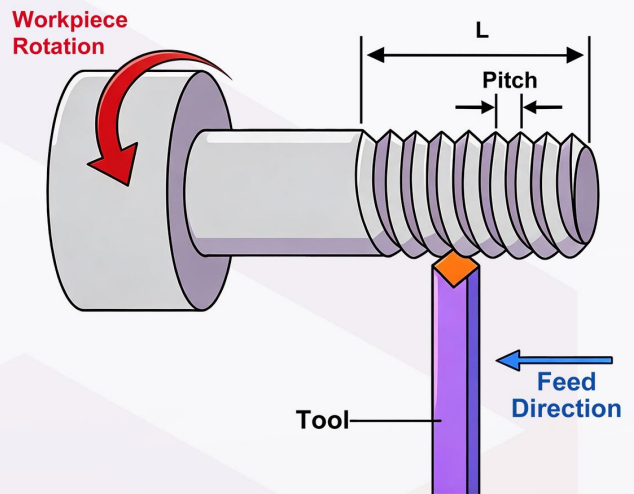
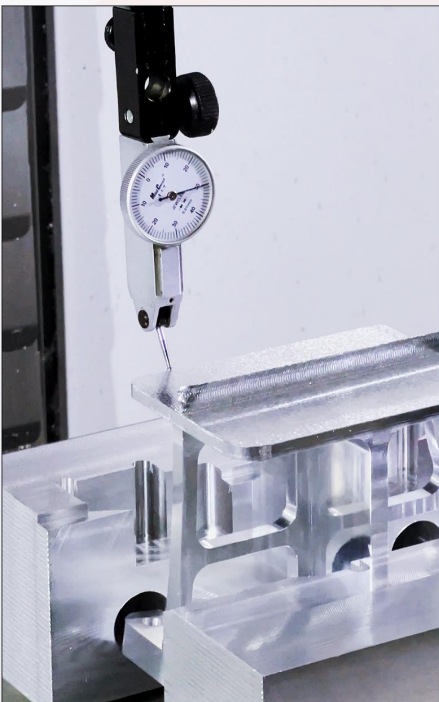


Figure 18 : Thread Turning



In addition to the operations above, CNC lathes may perform taper turning, contour turning, reaming, knurling, and tapping according to part geometry and production requirements. Turning centers with live tooling can also produce off-center holes, cross holes, flats, slots, and other non-rotational features. The process combination should be determined according to part geometry, dimensional tolerances, surface requirements, and production quantity.

3. Materials for CNC Machining

Material selection affects not only the service performance of CNC-machined parts but also machining efficiency, dimensional stability, surface quality, tool life, and overall manufacturing cost. The selected material must therefore meet the functional requirements of the part and be compatible with actual machining conditions.

3.1 Key Factors in Material Selection

The following factors should be considered when selecting a material for CNC machining:

- **Mechanical properties:** Material strength, stiffness, hardness, toughness, fatigue performance, and wear resistance should match the loads applied to the part.
- **Service environment:** Temperature, humidity, chemical exposure, corrosion, and outdoor conditions may limit the range of suitable materials.
- **Weight:** Aluminum alloys, titanium alloys, and engineering plastics may be considered when weight reduction is required.
- **Machinability:** Material hardness, toughness, thermal conductivity, chip formation, and work-hardening behavior affect cutting speed, tool wear, and machining time.
- **Dimensional stability:** Thermal expansion, moisture absorption, residual stress, and insufficient material stiffness can cause dimensional changes during or after machining.
- **Surface treatment and secondary processing:** The material should be compatible with the required surface quality and processes such as anodizing, plating, passivation, heat treatment, coating, or polishing.
- **Overall cost:** In addition to material price, stock availability, machining time, tool consumption, secondary processing, and scrap risk should be considered.

Material hardness is not the only basis for tool selection. Tool material, coating, geometry, flute count, and cutting parameters should be selected collectively according to the workpiece material and machined features.

3.2 Common Materials for CNC Machining

3.2.1 Metals

- **Aluminum** alloys have low density, a favorable strength-to-weight ratio, corrosion resistance, and good machinability. They are widely used for housings, brackets, fixtures, heat-dissipation components, robotic parts, and aerospace structures.
6061 is a common general-purpose aluminum alloy that provides a good balance of strength, corrosion resistance, machinability, and cost. 7075 offers higher strength but generally costs more and typically provides lower corrosion resistance and weldability than 6061.
- **Stainless** steel provides good corrosion resistance, strength, and heat resistance. Rigid workholding, wear-resistant carbide tools, suitable tool coatings, and continuous stable feed are generally required to control tool wear and maintain surface quality. Common grades include 304, 316, and 17-4 PH.
- **Copper** and copper alloys are generally selected for electrical conductivity, thermal conductivity, corrosion resistance, or low-friction performance. Typical applications include electrical components, heat sinks, electrodes, fluid fittings, bushings, and bearings.
Pure copper is soft and highly ductile, making it prone to material adhesion, burr formation, and surface smearing during machining. Brass is generally easier to machine than pure copper, while bronze offers better wear and friction properties and is suitable for sliding components.
- **Titanium** alloys provide a high strength-to-weight ratio, good corrosion resistance, and biocompatibility.
Ti-6Al-4V is a widely used titanium alloy grade for aerospace, medical, high-performance automotive, and chemical-processing equipment.

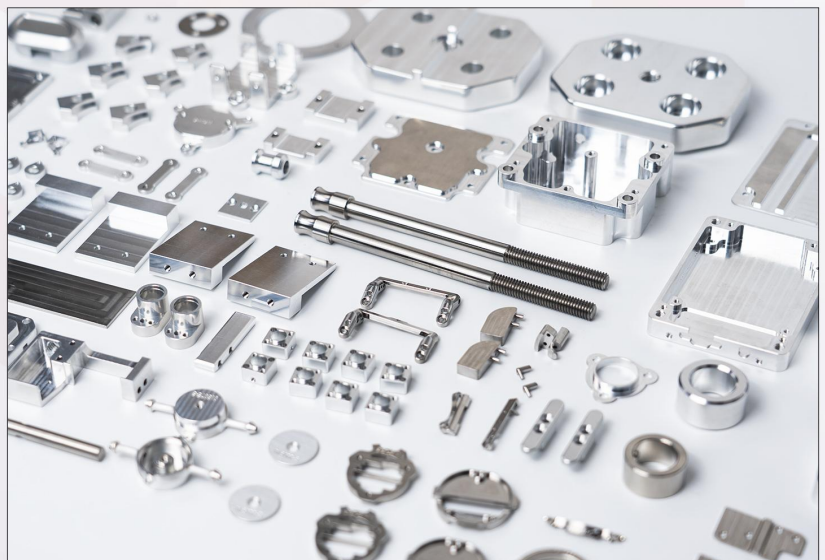


Figure 19 : First Mold Representative Metal Parts

3.2.2 Plastics

- **ABS** provides good toughness, impact resistance, and a degree of dimensional stability at a relatively low cost. It is commonly used for housings, brackets, control panels, fixtures, and functional prototypes. Its heat and weather resistance are limited, so the actual service environment must be verified for end-use parts.
- **POM** also known as polyoxymethylene or acetal, provides good stiffness, wear resistance, low friction, and relatively stable dimensions. Its chip formation is generally stable, making it suitable for plastic parts with tighter dimensional stability requirements, including gears, slides, valve components, fixtures, and precision mechanical components.
- **Nylon** provides good toughness, wear resistance, and low friction, making it suitable for gears, rollers, bearings, bushings, and other sliding parts. Its elasticity may allow the material to shift or deform under cutting or clamping forces. Nylon also absorbs moisture, which can change its dimensions and mechanical properties. Parts with tight tolerances should be machined and inspected under a defined humidity condition or material conditioning state.
- **PC** provides good impact resistance, transparency, and moderate heat resistance. It is commonly used for guards, viewing windows, lighting components, housings, and functional prototypes. Sharp tools and controlled cutting temperatures are required to reduce melting, burrs, and surface damage. Transparent surfaces generally retain tool marks after CNC machining and require polishing when higher optical clarity is needed.



Figure 20 : First Mold Representative Plastic Parts

- **PEEK** provides good heat resistance, mechanical strength, chemical resistance, wear resistance, and dimensional stability. It is commonly used for aerospace, medical, semiconductor, electrical-insulation, and high-temperature mechanical components.

3.3 Material Selection Guidelines

Table 2. CNC Machining Material Selection Guide

Performance Requirement	Recommended Materials	Typical Applications	Key Selection Considerations
Lightweight	6061 and 7075 aluminum alloys	Housings, brackets, and structural components	Strength, weight, surface finishing, and deformation risk
High Strength	Alloy steel, stainless steel, and titanium alloys	Shafts, connectors, and load-bearing components	Load, heat treatment, corrosion resistance, and machining difficulty
Corrosion Resistance	Stainless steel and titanium alloys	Medical, food-processing, and outdoor equipment components	Corrosive media, service environment, and material cost
Electrical or Thermal Conductivity	Copper, copper alloys, and aluminum alloys	Electrodes, electrical components, and heat-dissipation components	Electrical conductivity, thermal conductivity, burr formation, and surface quality
Wear Resistance and Low Friction	POM, PA, and PEEK	Gears, bushings, and guide rails	Friction, moisture absorption, temperature resistance, and dimensional stability
Optical Clarity	PC and PMMA	Transparent housings, viewing windows, and optical components	Clarity, impact resistance, tool marks, and polishing requirements

Get Custom Quote

4. Design for CNC Machining

4.1 Pockets and Cavities

Pockets and cavities are typically machined by progressively removing material with an end mill. As machining depth increases, greater tool overhang is required, reducing tool rigidity and increasing the risk of vibration, deflection, tool wear, and inconsistent surface quality. Chip evacuation and coolant delivery also become more difficult inside deep cavities, potentially reducing machining efficiency.

Design considerations include:

- Cavity depth should generally not exceed four times the minimum effective cavity width.
- Deeper or narrower features should be evaluated based on tool diameter, tool overhang, workpiece material, and surface requirements.
- Provide sufficient clearance for tool movement, coolant delivery, and chip evacuation.
- Avoid unnecessarily deep cavities and narrow slots, particularly features with a small width relative to their depth.
- Use larger internal corner radii in deep cavities to allow the use of larger, more rigid cutting tools.

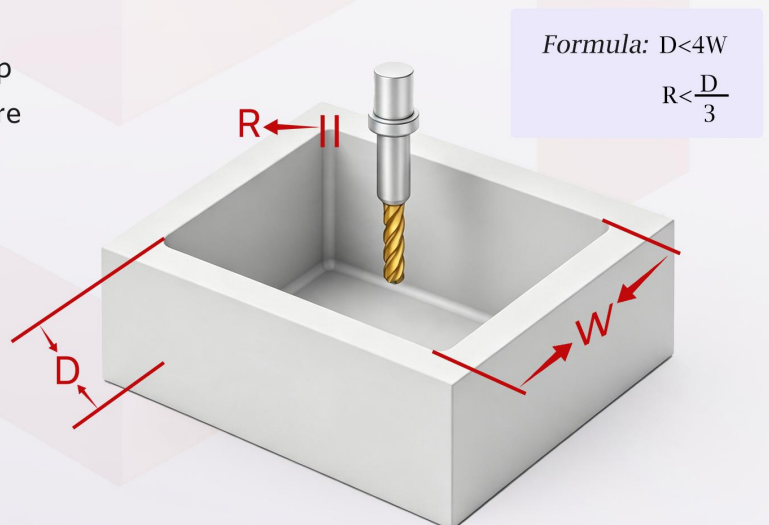


Figure 21 : Deep Pocket Design Points

4.2 Inner Radii and Edge Features

CNC milling uses rotating tools, so a vertical internal corner in a pocket cannot be machined as a perfectly sharp right angle. An internal radius corresponding to the tool radius must remain.

Smaller internal corner radii require smaller-diameter tools and generally increase machining time and the risk of tool breakage. Consider the following during design:

- Use larger internal corner radii wherever possible to allow larger, more rigid tools.
- The internal corner radius should not be exactly equal to the tool radius. Otherwise, cutting engagement increases suddenly as the tool enters the corner, which can cause chatter and surface-quality problems.
- Specify a radius slightly larger than a standard tool radius to provide clearance for tool movement.
- Internal corner radii in deep pockets should increase appropriately with pocket depth.
- A bottom fillet may be added to a pocket, but avoid requiring both very small sidewall radii and very small bottom radii.

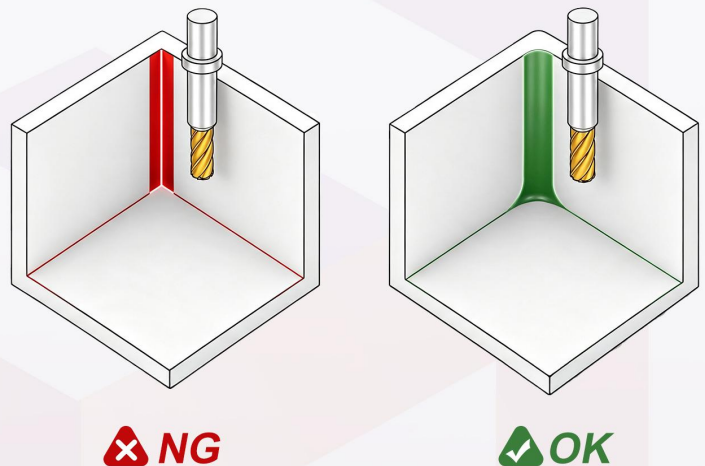


Figure 22 : Incorrect Internal Radii vs. Correct Internal Radii

External radii are generally easy to machine. However, radii required on both the top and bottom surfaces may require the part to be flipped or an additional machining direction. If an external radius has no defined functional or safety purpose, a chamfer may be considered instead.

Chamfers are commonly used to remove sharp edges, assist assembly, and improve hole-edge condition. Standard angles and dimensions, such as a 45° chamfer, should be preferred. Chamfers in different directions may require additional setups. Drawings should therefore distinguish functional chamfers from general deburring requirements and avoid applying a tight chamfer dimension uniformly to every edge.

4.3 Wall Thickness

Wall thickness directly affects part rigidity during cutting and clamping. Insufficient wall thickness may allow cutting forces to cause vibration and elastic deflection, while clamping forces may produce localized deformation. These effects can reduce dimensional accuracy and surface quality.

Consider the following during design:

- Metal thin walls may initially be designed with a thickness of 0.5–0.8 mm or greater.
- Plastic thin walls may initially be designed with a thickness of 0.8–1.2 mm or greater.
- Thin-wall height should preferably be limited to approximately five times the wall thickness.
- Maintain uniform wall thickness where possible and avoid abrupt transitions between thick and thin sections.
- For tall, thin structures, consider increasing the wall thickness, reducing the height, or adding reinforcing features to improve rigidity.

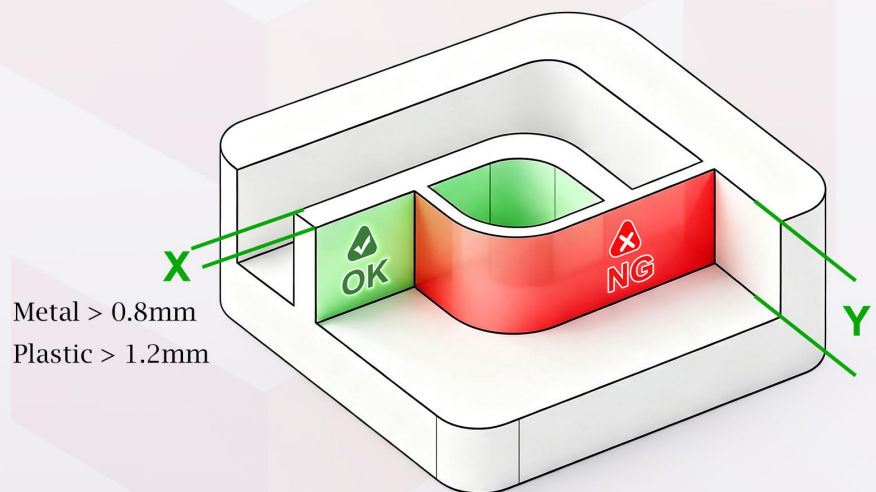


Figure 23 : Wall Thickness Design

4.4 Hole Design

Standard hole diameters generally correspond to commonly available drills and reamers, making them easier to machine and inspect. Hole design should consider diameter, depth, position, machining direction, entry surface, and bottom geometry.

As hole depth increases, drill drift, diameter variation, degraded surface quality, or tool breakage may occur. Therefore:

- Use standard drill diameters wherever possible.
- Minimize the hole depth-to-diameter ratio.
- For deep holes, specify the functional depth that actually requires control rather than applying the same accuracy requirement to the entire depth.
- Avoid starting a hole directly on an inclined or curved surface. If unavoidable, add a flat or counterbore as the tool-entry area.
- For intersecting holes, consider whether exit burrs can be removed.

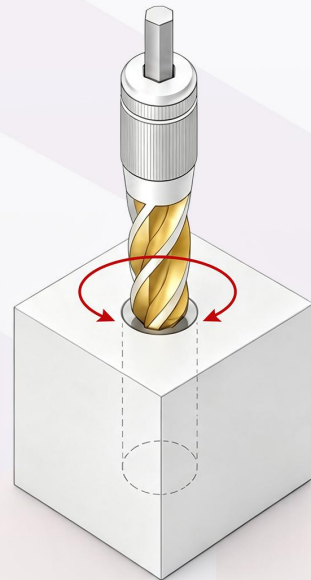


Figure 24 : Standard Hole Diameters



Figure 25 : Avoid Flat-Bottom Blind Holes

4.5 Thread Design

Standard thread specifications should be preferred so that standard taps, thread mills, and inspection tools can be used. Consider the following during design:

- Avoid unnecessarily deep threaded holes. Once thread engagement exceeds the effective load-bearing requirement, additional length generally provides little further increase in joint strength.
- For a threaded blind hole, the drilled depth must exceed the effective thread depth to provide clearance for the drill point and incomplete tap threads.
- Provide the required tool runout space at the end of the thread.
- Threaded holes near a thin-wall edge may weaken the local structure; sufficient wall thickness should be retained.
- The drawing should specify the thread standard, size, tolerance class, and effective thread depth.

Internal threading is generally more restricted by access and depth than external threading. For threads that are difficult to access or oriented in an unusual direction, assess whether a threaded insert, through hole, or alternative fastening structure can be used.

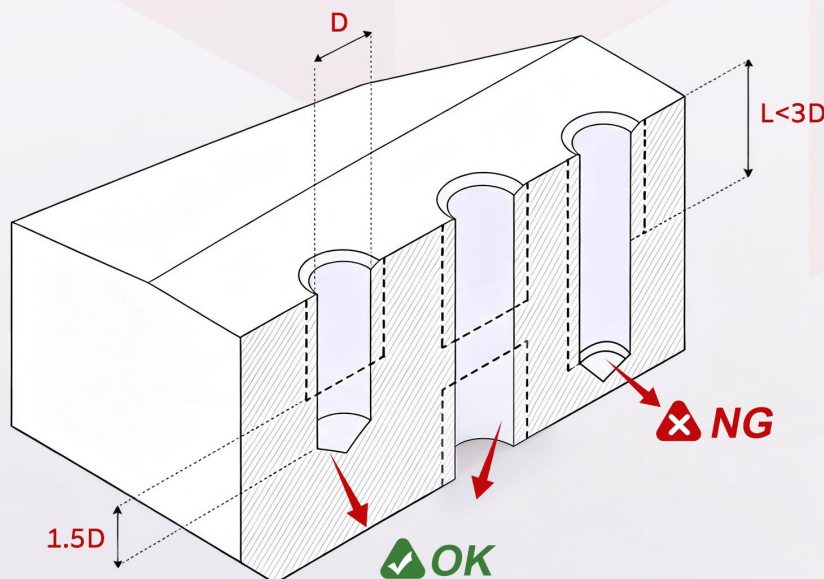


Figure 26 : Threaded Hole Design Comparison

4.6 Tool Accessibility

Tool accessibility determines whether a feature can be machined with standard tools and machining directions. Undercuts, deep cavities, and obstructed surfaces may require specialized tools, multi-axis machining, or additional setups.

Design considerations include:

- Ensure sufficient clearance for tool entry, cutting, and retraction.
- Check for interference with the tool, holder, or spindle.
- Replace non-essential undercuts with open or through-slots.
- Prefer standard dimensions for T-slots and dovetail grooves.
- Avoid obstructed features in deep or narrow areas.
- Consider splitting the part if access is impractical or costly.

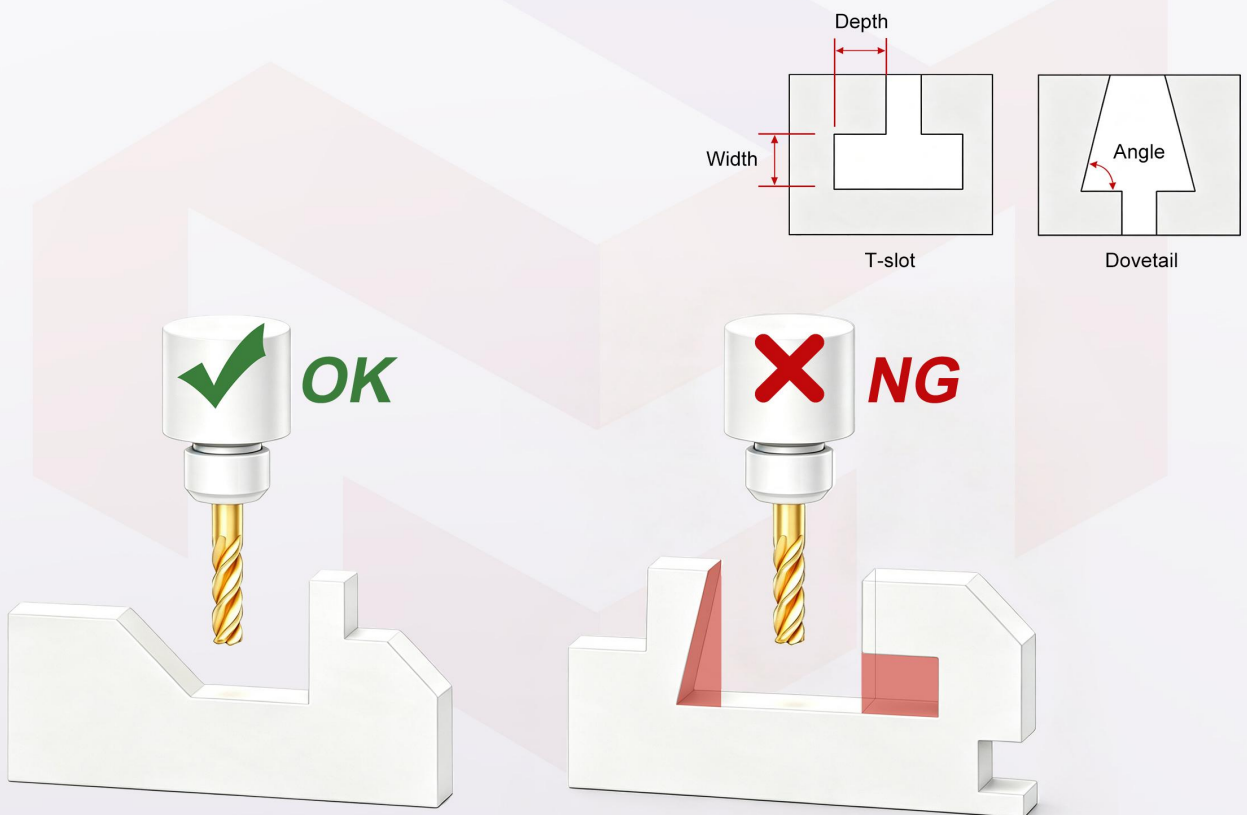


Figure 27: Open Tool Access vs. Undercut Features

4.7 Setup Reduction

Multiple machining directions often require additional setups, increasing machining time and the risk of datum-transfer errors. Where practical, related features with tight positional or geometric tolerances should be machined from the same datum and setup.

Design considerations include:

- Align feature directions to reduce setups.
- Provide stable locating and clamping surfaces.
- Avoid clamping positions that obstruct machining or deform the part.
- Use consistent datums for related critical features.
- Add machining tabs or clamping stock where necessary.

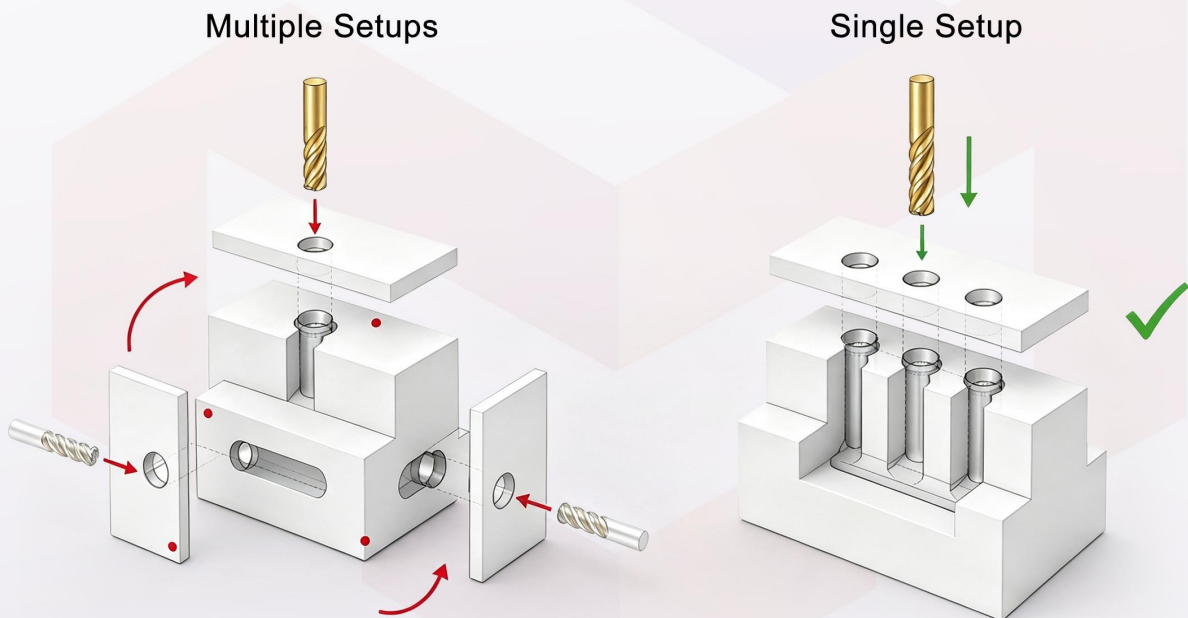


Figure 28 : Multiple Setups vs. Single-Setup Machining

4.8 Tolerances and Surface Roughness

Tolerances define the permissible limits of dimensional variation. Achievable tolerances depend on nominal size, geometry, material, and machining conditions. Unless otherwise specified, general tolerances may follow ISO 2768-m or the manufacturer's standard. First Mold supports CNC-machined parts with tolerances of ± 0.05 mm (± 0.002 in), with project-specific requirements evaluated through engineering review.

Tight tolerances and stringent surface finish requirements increase machining, inspection, and rework demands and should therefore be specified only where functionally necessary. Drawing considerations include:

- Apply tight tolerances only to functional features.
- Use general tolerances for noncritical dimensions.
- Establish clear datums for geometric tolerances.
- Avoid duplicate or conflicting dimensions.
- Specify surface roughness according to contact, sealing, friction, or appearance requirements.
- Ensure that all requirements can be practically inspected.

Dimensional tolerances and surface roughness should be specified separately. For post-processed parts, indicate whether dimensions apply before or after treatment, and account for coating thickness, distortion, and finishing allowance.

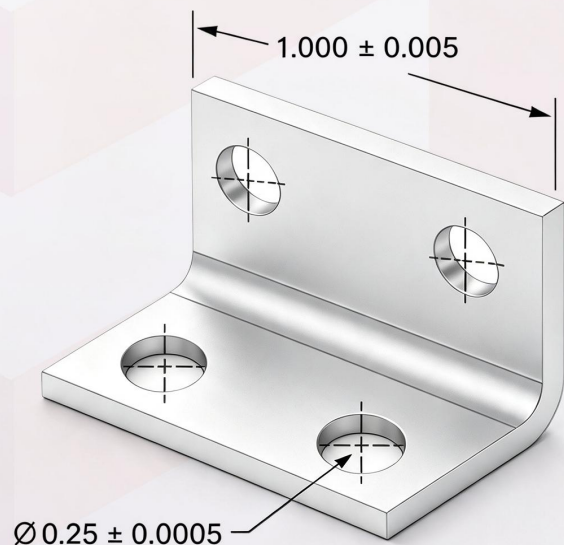


Figure 29 : CNC Tolerancing

5. Supplier Selection Guide

CNC machining planning must balance part performance, cost, production efficiency, and quality consistency. Higher axis capability or tighter tolerances do not necessarily provide a better solution.

5.1 Costs and Influencing Factors

5.1.1 Material and Stock

Material cost depends not only on unit price but also on purchasing specifications, stock form, material utilization, and machinability.

The main factors include:

- Material grade, condition, and sourcing difficulty
- Stock form, such as plate, bar, forging, or casting
- Difference between stock dimensions and the finished-part profile
- Material removal volume and utilization rate
- Cutting performance, tool wear, and cooling requirements

Provided that part-performance requirements are met, common material grades and standard stock close to the finished dimensions can generally reduce material waste and machining time. Although titanium alloys, high-temperature alloys, and certain high-hardness materials may be used in limited quantities, their lower cutting efficiency and higher tool consumption can still increase costs significantly.

5.1.2 Programming and Machining Preparation

Complex surfaces, multi-directional features, and tight relative-position requirements generally require more CAM programming, process planning, and toolpath verification. Preparatory work primarily includes:

- Datum and process planning
- CAM programming and toolpath optimization
- Selection of tools, cutting parameters, and machining sequence
- Fixture design and manufacturing
- First-article setup and program verification

For a one-off prototype, preparation costs must be absorbed by a small number of parts and therefore represent a relatively high proportion of the unit price. As order quantity increases, these costs can be distributed across more parts, although dedicated fixtures and standardized processes may require further investment.

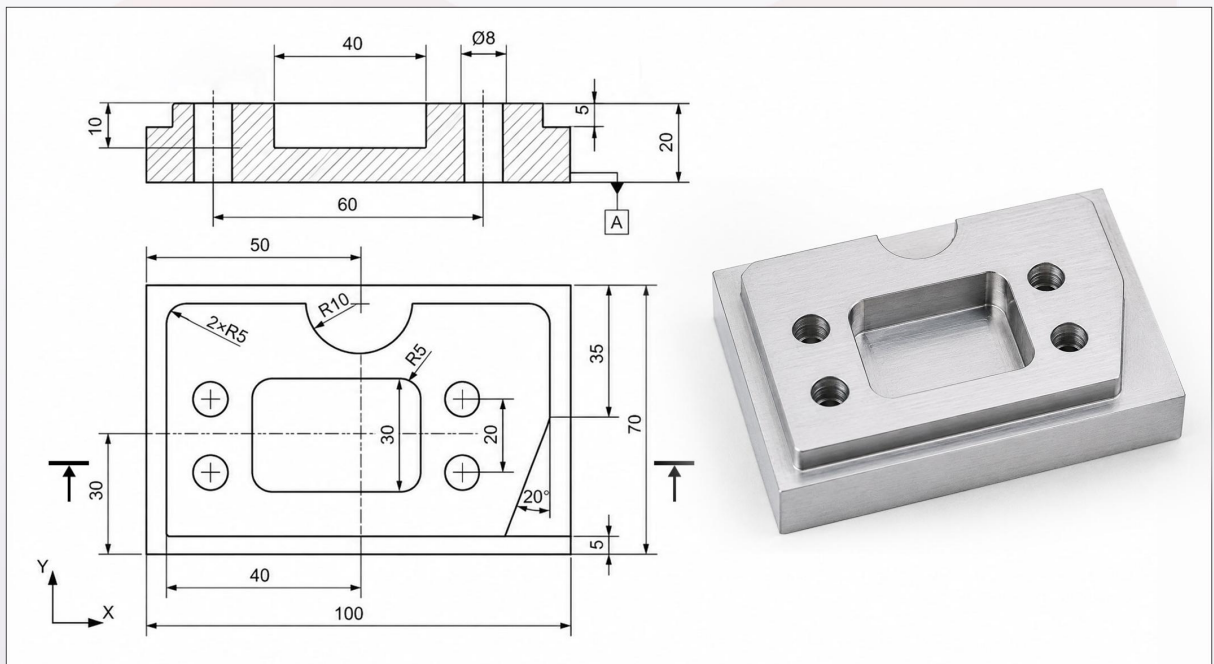


Figure 30 : CNC Milling Programming Example

5.1.3 Machine Selection and Machining Time

Machine occupancy time is generally a major component of CNC machining cost. Machine cost per part can be expressed in simplified form as:

$$C_{machine,unit} = R_{machine} \times T_{machine}$$

Where:

$C_{machine,unit}$: Machine cost per part

$R_{machine}$: Machine hourly rate

$T_{machine}$: Machine occupancy time per part



Machine occupancy time includes cutting and auxiliary operations such as loading, unloading, tool changes, part repositioning, alignment, and in-process measurement. It generally increases with material-removal volume, toolpath length, finishing requirements, setup count, limited tool access, and secondary processes such as EDM or grinding.

Machine selection also affects cost. 3-axis machining is generally more economical for parts with features concentrated on a few accessible faces. 4-axis machining suits circumferential features, while 5-axis machining is better for complex surfaces and multi-directional features. For parts combining turned and milled features, mill-turn machining can reduce setups and transfers between machines.

5.1.4 Cutting Tools, Fixtures, and Quality Control

Deep pockets, narrow slots, small internal radii, deep holes, and difficult-to-machine materials may require extended-reach tools, special tools, or more frequent tool changes. Thin-walled, irregular, and multi-sided parts may also require soft jaws or dedicated fixtures.

Tight tolerances and low surface-roughness requirements also increase:

- Finishing and compensation adjustments
- First-article and in-process inspection
- CMM inspection or inspection with dedicated gauges
- Material certificates and inspection reports
- Risk of rework and scrap

Tight tolerances and low roughness values should be limited to areas related to mating, positioning, sealing, and motion functions, rather than applying the highest standard uniformly to the entire part.



Figure 31 : CNC Cutting Tooling

5.1.5 Post-Processing and Project Conditions

Heat treatment, anodizing, plating, sandblasting, polishing, and laser marking add post-machining handling, inspection, and quality-control requirements. Some treatments may also change part dimensions, hardness, or surface condition, requiring machining allowance and a clearly defined final inspection stage.

Order quantity and lead time also affect pricing:

- Larger batches allow programming and fixture costs to be distributed across more parts.
- Stable production volumes may justify multi-part workholding or automated production.



Figure 32 : Anodizing

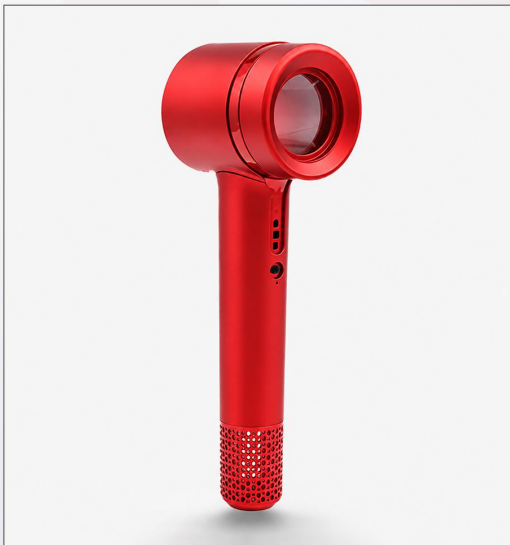


Figure 33 : Spraying

- Urgent delivery may require schedule changes, overtime, and expedited material procurement.
- Strict packaging, traceability, and quality-documentation requirements increase project-management costs.

5.2 CNC Machining by Production Volume

Order quantity changes the main cost drivers in CNC machining. Prototype projects generally emphasize lead time and design flexibility, while production runs place greater emphasis on cycle time, fixture efficiency, and long-term quality consistency.

5.2.1 Prototypes

One-off or low-quantity prototypes generally use standard stock, general-purpose tools, and flexible workholding to minimize investment in dedicated tooling.

Key project priorities include:

- Rapid design and functional validation
- Control of programming and preparation time
- Flexibility for design changes
- Avoidance of excessive tolerances in nonfunctional areas
- Verification of manufacturability for subsequent production

The unit cost of prototype machining is generally high, but no mold investment is required. It is suitable for design validation, engineering testing, and new-product development.

5.2.2 Low-Volume Production

Low-volume production requires a balance between machining flexibility and unit cost. Soft jaws, simple fixtures, toolpath optimization, and multi-part workholding can reduce auxiliary time and improve repeatability.

Key project priorities include:

- Reasonable allocation of programming and fixture costs
- Standardized workholding methods and operating steps
- Control of tool life and tool-change frequency
- First-article approval and in-process sampling inspection
- A degree of adaptability to design changes

5.2.3 Production Machining

As order volume increases, the cost-effectiveness of continued CNC machining should be evaluated. For high-volume parts with stable designs, casting, forging, extrusion, or injection molding may be more economical, while CNC machining can be retained for critical holes, mating surfaces, and other high-precision features.

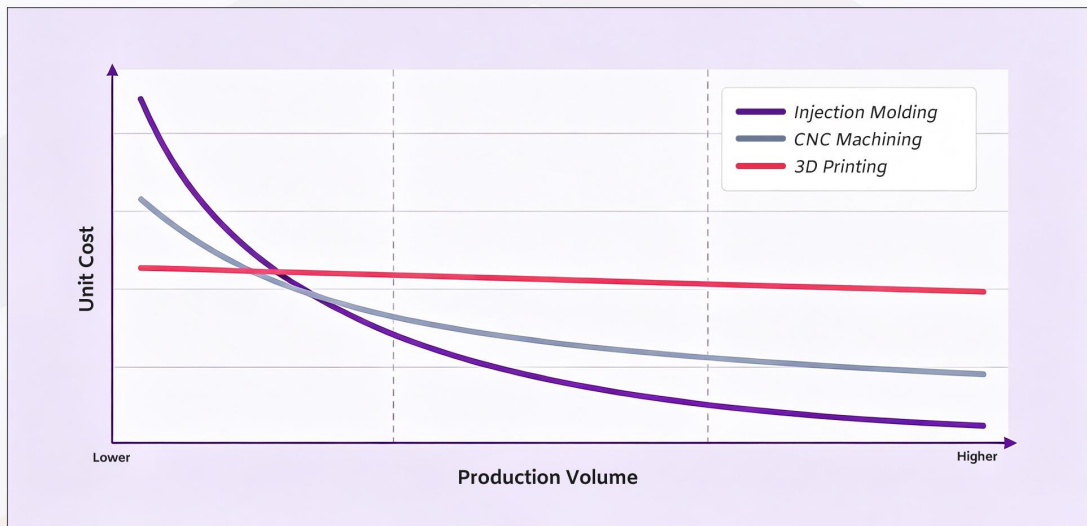


Figure 34 : Indicative Unit Cost Comparison by Production Volume

5.3 CNC Supplier Evaluation

CNC supplier selection should not be based on unit price alone. Equipment capability, process experience, inspection resources, quality systems, and sustained delivery capability directly affect project risk and final cost. The following areas should be evaluated:

- **Equipment and Machining Range**

The supplier should have equipment compatible with the part size, geometry, and accuracy requirements and should be able to explain the application range of three-axis, four-axis, five-axis, turning, and secondary processes.

- **Material and Similar-Part Experience**

Metals and engineering plastics differ significantly in cutting, workholding, cooling, and dimensional control. The supplier should have experience with the relevant materials and similar part geometries.

- **Process Review Capability**

A qualified supplier should identify risks such as deep pockets, thin walls, small radii, tight tolerances, and difficult workholding before machining and provide practical DFM recommendations.

- **Fixture and Tooling Strategy**

For complex or production parts, evaluate whether the supplier can design fixtures, select tools, optimize toolpaths, and consolidate operations.

- **Inspection and Quality System**

In addition to machine accuracy, verify that the supplier has CMM, vision measurement, contour, and surface-roughness inspection capabilities and can provide inspection records and quality documents as required by the project.

- **Capacity and Delivery Stability**

The supplier should have equipment, personnel, and scheduling capacity appropriate to the order volume, together with management processes for materials, machining, outsourced operations, and nonconformance handling.

- **Material and Process Traceability**

For automotive, medical, and other regulated-industry projects, confirm that material certificates, production batches, inspection records, and critical processes are traceable.

6. Conclusion

Successful CNC machining depends on more than machine accuracy or axis count. Part geometry, material, tool access, workholding, tolerances, surface finish, production volume, and inspection requirements must be considered together.

Early manufacturability review helps reduce unnecessary setups, machining time, cost, and quality risks. The most suitable process is the one that meets the project's functional and quality requirements efficiently.

First Mold Manufacturing operates more than 80 CNC machines and supports 3-axis, 4-axis, and 5-axis machining for metal and engineering plastic parts, from prototypes to volume production. Our services cover process planning, fixture design, machining, inspection, and post-processing.

If you are developing a CNC-machined part, First Mold's engineering team can review the design and provide manufacturing recommendations based on the part geometry, material, production volume, and quality requirements.



First Your Need ***Mold Your Part***

Contact Details



<https://firstmold.com>



sales@firstmold.com



+86 13925326660



Mold Making



Injection Molding



CNC Machining



3D Printing



Die Casting



Sheet Metal

