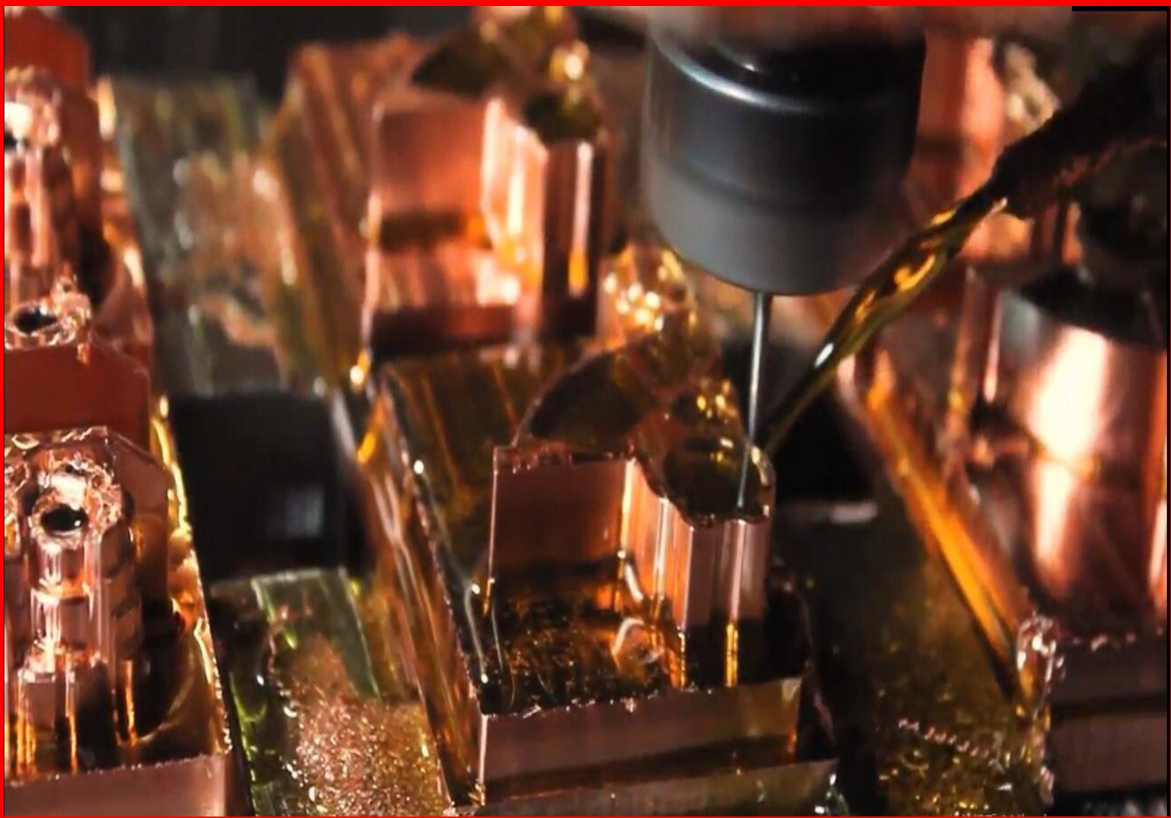
Handbook

CNC Machining Guide



For beginners and professionals

IN3DTEC V6.2.1

Contents

1 >> The basics

1.1 What is CNC machining

The term CNC stands for 'computer numerical control', and the CNC machining definition is that it is a subtractive manufacturing process which typically employs computerized controls and machine tools to remove layers of material from a stock piece—known as the blank or workpiece—and produces a custom-designed part. This process is suitable for a wide range of materials, including metals, plastics, wood, glass, foam, and composites, and finds application in a variety of industries, such as large CNC machining, machining of parts and prototypes for telecommunications, and CNC machining aerospace parts, which require tighter tolerances than other industries. Note there is a difference between the CNC machining definition and the CNC machine definition- one is a process and the other is a machine. A CNC machine is a programmable machine that is capable of autonomously performing the operations of CNC machining.

Subtractive manufacturing processes, such as CNC machining, are often presented in contrast to additive manufacturing processes, such as 3D printing, or formative manufacturing processes, such as liquid injection molding. While subtractive processes remove layers of material from the workpiece to produce custom shapes and designs, additive processes assemble layers of material to produce the desired form and formative processes deform and displace stock material into the desired shape. The automated nature of CNC machining enables the production of high precision and high accuracy, simple parts and cost-effectiveness when fulfilling one-off and medium-volume production runs. However, while CNC machining demonstrates certain advantages over other manufacturing processes, the degree of complexity and intricacy attainable for part design and the cost-effectiveness of producing complex parts is limited.

1.2 Overview of CNC machining process

Evolving from the numerical control (NC) machining process which utilized punched tape cards, CNC machining is a manufacturing process which utilizes computerized controls to operate and manipulate machine and cutting tools to shape stock material—e.g., metal, plastic, wood, foam, composite, etc.—into custom parts and designs. While the CNC machining process offers various capabilities and operations, the fundamental principles of the process remain largely the same throughout all of them. The basic CNC machining process includes the following stages:

- Designing the CAD model
- Converting the CAD file to a CNC program
- Preparing the CNC machine
- Executing the machining operation

CAD Model Design

The CNC machining process begins with the creation of a 2D vector or 3D solid part CAD design either in-house or by a CAD/CAM design service company. Computer-aided design (CAD) software allows designers and manufacturers to produce a model or rendering of their parts and products along with the necessary technical specifications, such as dimensions and geometries, for producing the part or product.

Designs for CNC machined parts are restricted by the capabilities (or inabilities) of the CNC machine and tooling. For example, most CNC machine tooling is cylindrical therefore the part geometries possible via the CNC machining process are limited as the tooling creates curved corner sections. Additionally, the properties of the material being machined, tooling design, and work holding capabilities of the machine further restrict the design possibilities, such as the minimum part thicknesses, maximum part size, and inclusion and complexity of internal cavities and features.

Once the CAD design is completed, the designer exports it to a CNC-compatible file format, such as STEP or IGES.

CAD File Conversion

The formatted CAD design file runs through a program, typically computer-aided manufacturing (CAM) software, to extract the part geometry and generates the digital programming code which will control the CNC machine and manipulate the tooling to produce the custom-designed part.

CNC machines used several programming languages, including G-code and M-code. The most well-known of the CNC programming languages, general or geometric code, referred to as G-code, controls when, where, and how the machine tools move—e.g., when to turn on or off, how fast to travel to a particular location, what paths to take, etc.—across the workpiece. Miscellaneous function code, referred to as M-code, controls the auxiliary functions of the machine, such as automating the removal and replacement of the machine cover at the start and end of production, respectively.

Once the CNC program is generated, the operator loads it to the CNC machine.

Machine Setup

Before the operator runs the CNC program, they must prepare the CNC machine for operation. These preparations include affixing the workpiece directly into the machine, onto machinery spindles, or into machine vises or similar work holding devices, and attaching the required tooling, such as drill bits and end mills, to the proper machine components.

Once the machine is fully set up, the operator can run the CNC program.

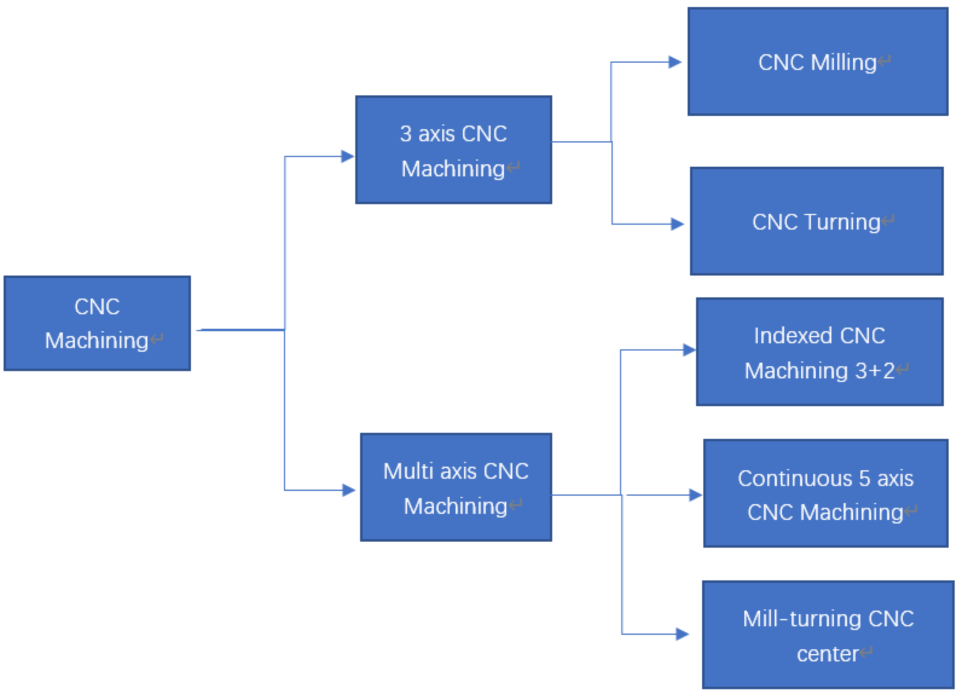
Machining Operation Execution

The CNC program acts as instructions for the CNC machine; it submits machine commands dictating the tooling's actions and movements to the machine's integrated computer, which operates and manipulates the machine tooling. Initiating the program prompts the CNC machine to begin the CNC machining process, and the program guides the machine throughout the process as it executes the necessary machine operations to produce a custom-designed part or product.

CNC machining processes can be performed in-house—if the company invests in obtaining and maintaining their own CNC equipment—or out-sourced to dedicated CNC machining service providers.

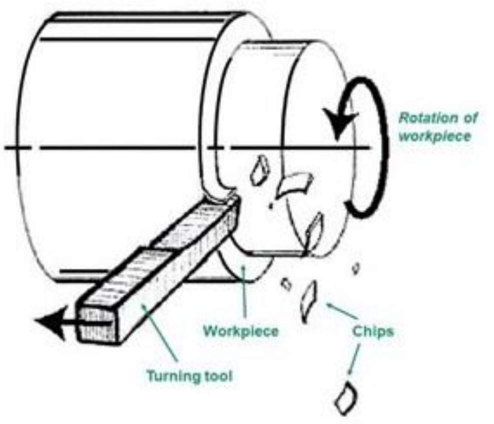
1.3 Types of CNC machines

In this guide, we will focus on CNC machines that remove material using cutting tools. These are the most common and have the widest range of applications. Other CNC machines include laser cutters, plasma cutters and EDM machines.

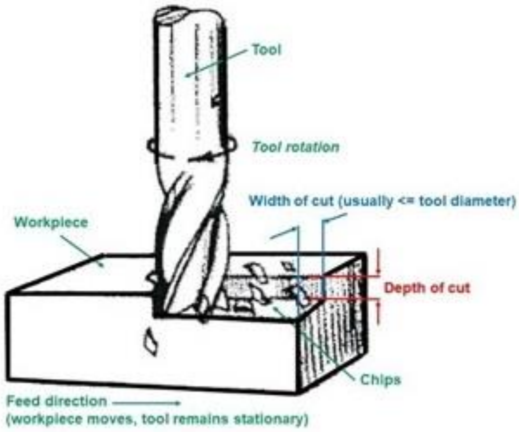


1.3.1 3-axis CNC machines

CNC milling and CNC turning machines are examples of 3-axis CNC systems. These "basic" machines allow the movement of the cutting tool in three linear axes relative to the workpiece (left-right, back-forth and up-down).



Turning



Milling

How does CNC milling work?

CNC milling is the most popular CNC machine architecture. In fact, the term CNC milling is often synonymous with the term CNC machining.

In CNC milling, the part is mounted onto the bed and material is removed using rotational cutting tools. Here is an overview of the basic CNC milling process:

1, First, the CAD model is converted into a series of commands that can be interpreted by the CNC machine (G-code). This is usually done on the machine by its operator, using the provided technical drawings.

2, A block of material (called the blank or the workpiece) is then cut to size and it is placed on the built platform, using either a vice or by directly mounting it onto the bed. Precise positioning and alignment is key for manufacturing accurate parts and special metrology tools (touch probes) are often used for this purpose.

3, Next, material is removed from the block using specialized cutting tools that rotate at very high speeds (thousands of RPM). Several passes are often required to create the designed part. First, an approximate geometry is given to the block, by removing material quickly at a lower accuracy. Then one or more finishing passes are used to produce the final part.

4, If the model has features that cannot be reached by the cutting tool in a single setup (for example, if it has a slot on it back side), then the part needs to be flipped and the above steps are repeated.

After machining, the part needs to be deburred. Deburring is the manual process of removing the small defects left on sharp edges due to material deformation during machining (for example, the defects created as a drill exists the far side of a through hole). Next, if tolerances were specified in the technical drawing, the critical dimensions are inspected. The part is then ready to use or post-process.

Most CNC milling systems have 3 linear degrees of freedom: the X, Y and Z axis. More advanced systems with 5 degrees of freedom also allow the rotation of the bed and/or the tool head (A and B axis). 5-axis CNC systems are capable of producing parts with high geometric complexity and may eliminate the need for multiple machine setups.

How does CNC turning work?

In CNC turning, the part is mounted on a rotating chuck and material is removed using stationary cutting tools. This way parts with symmetry along their center axis can be manufactured. Turned parts are typically produced faster (and at a lower cost) than milled parts.

Here is a summary of the steps followed to manufacture a part with CNC turning:

1, The G-code is first generated from the CAD model and a cylinder of stock material (blank) with suitable diameter is loaded in the CNC machine.

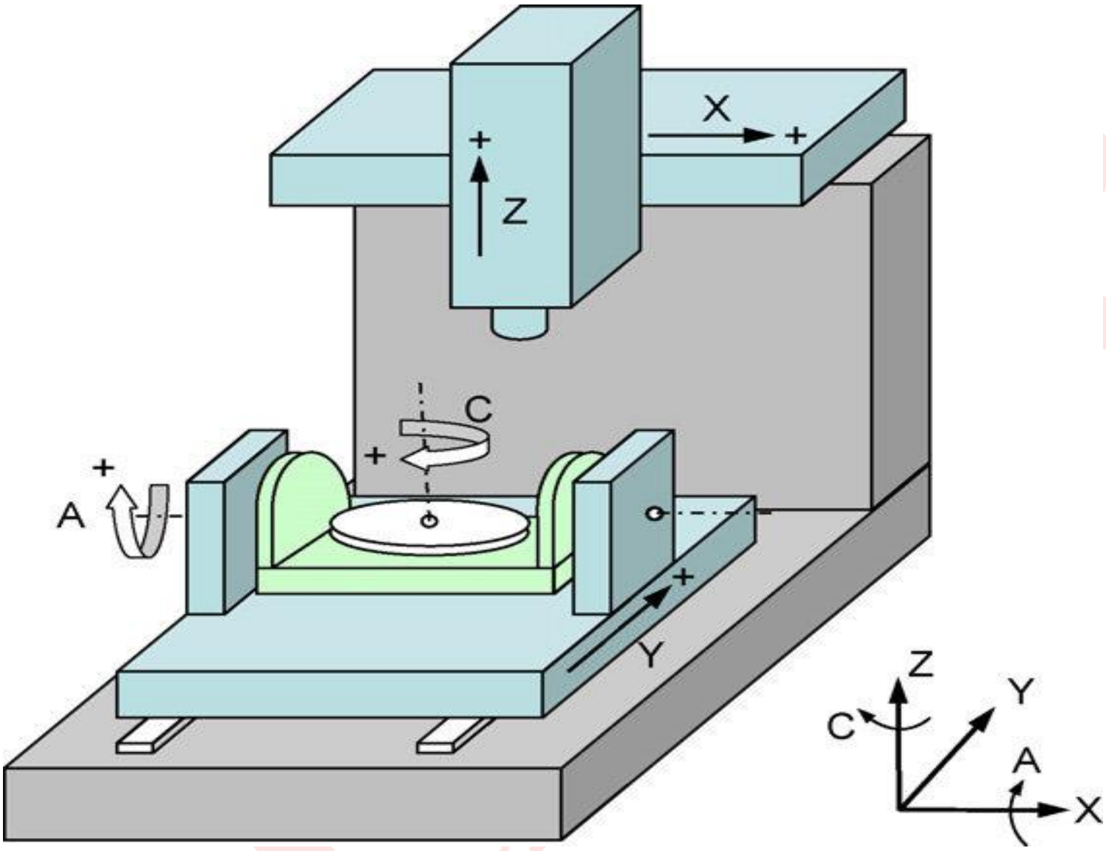
2, The part starts rotating at high speed and a stationary cutting tool traces its profile, progressively removing material until the designed geometry is created. Holes along the center axis can be also manufactured, using center drills and internal cutting tools.

3, If the part needs to be flipped or moved, then the process is repeated. Otherwise, the part is cut from the stock and it is ready for use or further post-processing.

Typically, CNC turning systems (also known as lathes) are used to create parts with cylindrical profiles. Non-cylindrical parts can be manufactured using modern multi-axis CNC turning centers, which are also equipped with CNC milling tools. These systems combine the high productivity of CNC turning with the capabilities of CNC milling and can manufacture a very large range of geometries with (looser) rotational symmetry, such as camshafts and radial compressor impellers.

Since the lines between milling and turning systems are blurry, the rest of the article focuses mainly on CNC milling, as it is a more common manufacturing process.

1.3.2 Multi axis CNC machines



Multi-axis CNC machining centers come in three variations: 5-axis indexed CNC milling, continuous 5-axis CNC milling and mill-turning centers with live tooling.

These systems are essentially milling machines or lathes enhanced with additional degrees of freedom. For example, 5-axis CNC milling centers allow the rotation of the machine bed or the toolhead (or both) in addition to the three linear axes of movement.

The advanced capabilities of these machines come at an increased cost. They require both specialized machinery and also operators with expert knowledge. For highly complex or topology optimized metal parts, 3D printing is usually a more suitable option though.

Indexed 5-axis CNC milling systems are also known as 3+2 CNC milling machines, since they are using the two additional degrees of freedom only between machining operations to rotate the workpiece.

The key benefit of these systems is that they eliminate the need of manually repositioning the workpiece. This way parts with more complex geometries can be manufactured faster and at higher accuracy than in a 3-axis CNC mill. They lack though the true freeform capabilities of continuous 5-axis CNC machines.

Continuous 5-axis CNC milling systems have a similar machine architecture to indexed 5-axis CNC milling machines. They allow, however, for the movement of all five axes at the same time during all machining operations.

This way, it is possible to produce parts with complex, 'organic' geometries that cannot be manufactured at the achieved level of accuracy with any other technology. These advanced capabilities come of course at a high cost, as both expensive machinery and highly-trained machinists are needed.

Mill-turning CNC centers are essentially CNC lathe machines equipped with CNC milling tools. A variation of the mill-turning centers are swiss-style lathes, which have typically higher precession.

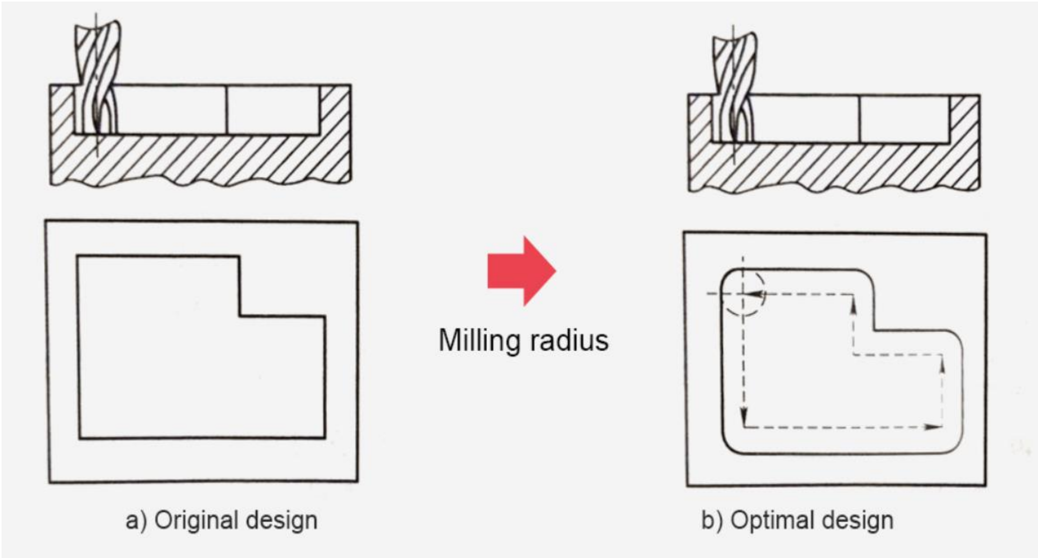
Mill-turning systems take advantage of both the high productivity of CNC turning and the geometric flexibility of CNC milling. They are ideal for manufacturing parts with 'loose' rotational symmetry (think camshafts and centrifugal impellers) at a much lower cost than other 5-axis CNC machining systems.

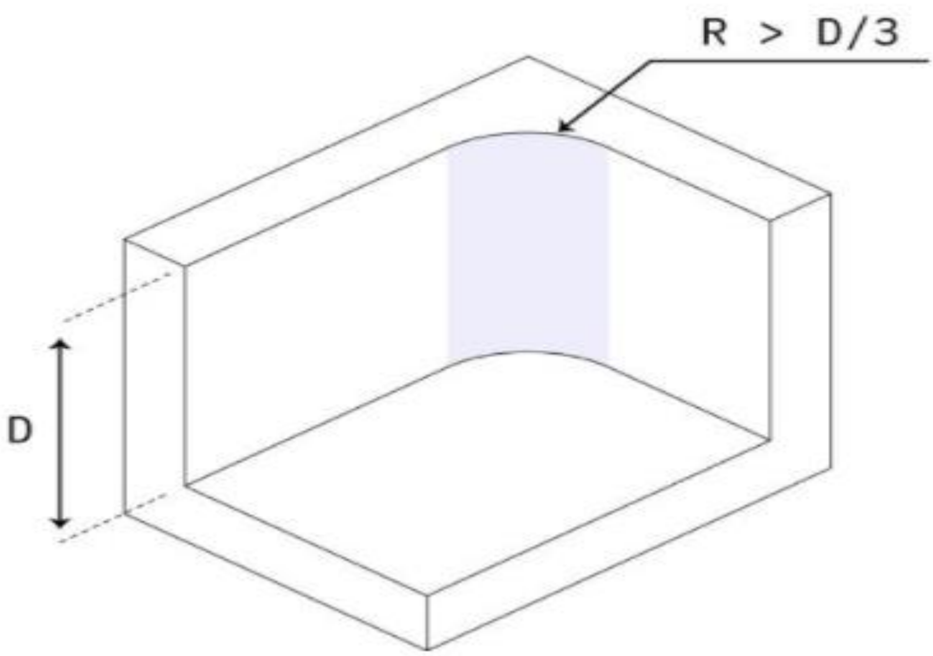
2 >> 7 Design tips for CNC machining

2.1 Rounded internal corners

The corner radius should be remained in the corner of large sunken area of the part, and the corner radius should be consistent with the diameter of standard milling cutter. As shown in the figure of original design, the corner is right angles, the part is not available for milling, and should be manufactured via more expensive process (such as EDM). While in the optimized design, milling can be realized by adding the corner radium.

When a 90° internal edge is needed, reducing the radius will not do the job. In these cases, use an undercut instead.





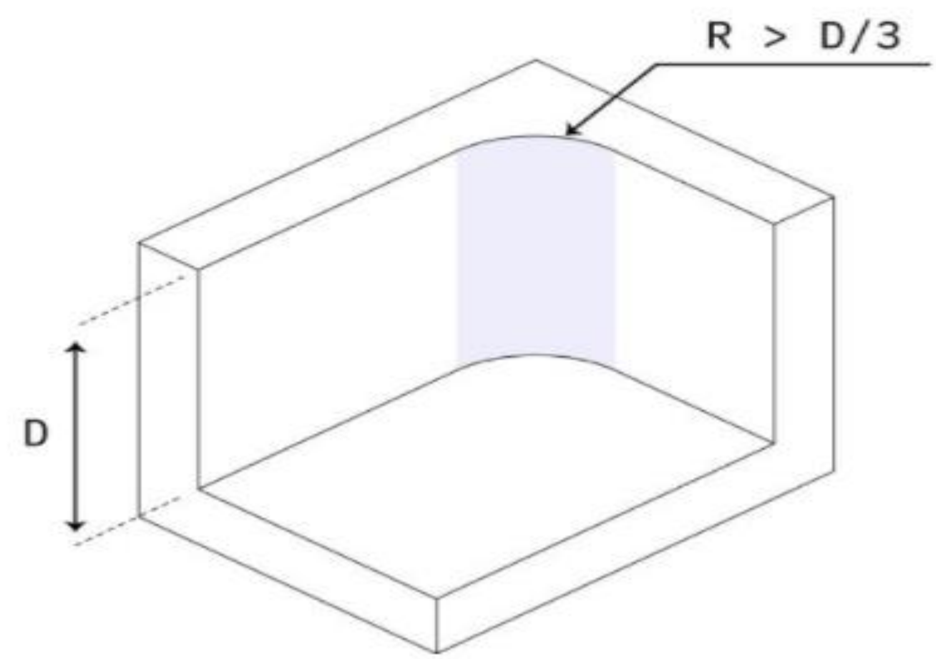
2.2 Ratio (depth to radius) ⩽ 3:1

A CNC machining tool – such as an end mill or milling cutter – will automatically leave rounded inside corners. The wider this radius, the less passes that the tool will need to make to remove material.

By contrast, a narrow inside corner radius requires both a small tool to machine away material and more passes — often at slower speeds to reduce the risk of deflection or tool breakage —

resulting in more machining time.

For optimum design, a radius that is slightly larger than 1/3 of the depth of the cavity. In addition, try to keep internal corner radii the same, if possible. This helps eliminate tool changes, which adds complexity and can increase run time significantly.



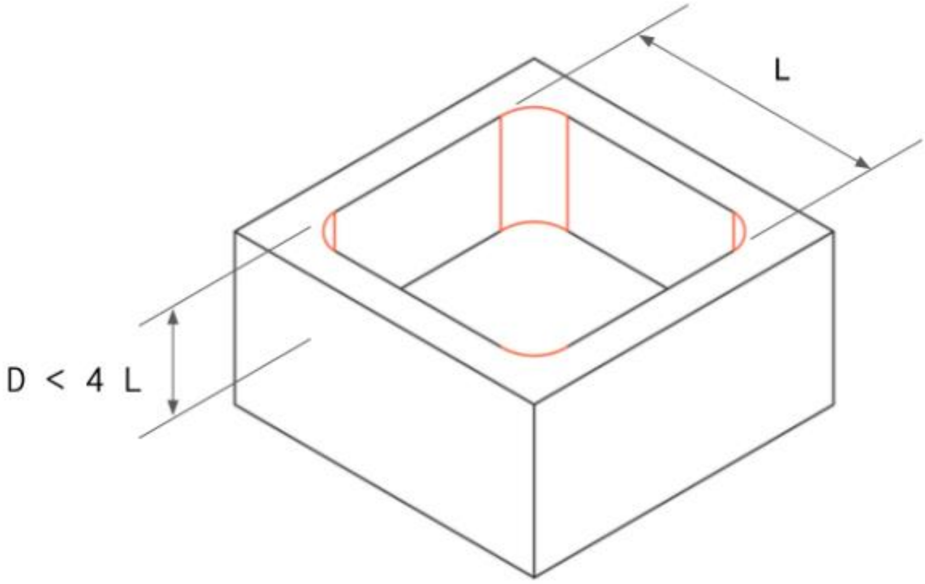
2.3Limit the depth of cavities

Machining deep cavities affects the cost of CNC parts dramatically, as a lot of material needs to be removed, which is very time-consuming.

It is important to keep in mind that CNC tools have a limited cutting length: typically they will work best when cutting cavities with a depth of up to 2-3 times their diameter. For example, a ø12 milling tool can cut cavities safely up to 25 mm deep.

Cutting deeper cavities is possible (up to 4x the diameter of the tool or greater), but this will increase the cost, as special tooling or multi-axis CNC systems are required.

The milling area should not be too deep, and the depth-to-width ratio should not exceed 4:1. Otherwise, the milling cutter will be easy to break off, then increasing the cost.



2.4Pay attention to the thin walls

Thin walled CNC parts tend to chatter, which will slow down the processing speed. They also cause deformation, which makes it difficult to maintain tolerances. The result could be additional costs.

In order to keep the processing cost low, the minimum width of thin wall should be 0.8mm for metal parts. For plastic parts, keep the minimum wall thickness above 1.5 mm.

If very thin walls are required, it is usually more economical to use other manufacturing methods, such as sheet metal manufacturing.

2.5Optimize threaded hole

When it comes to threaded holes, two factors that may increase costs are hole depth and thread size.

Generally, increasing the length of the thread in the hole does not make the bolt tighter. In fact, all the work is the first two or three turns. Therefore, it is not necessary to screw the diameter of the hole more than 3 times (if possible, shorten it as much as possible). Drilling deep will only increase the risk of tapping fracture and increase the time of tapping operation.

In addition, using standard tap sizes can help reduce costs. For example, 4-40 taps are more common and common than 3-48 taps. Also, avoid very small threaded holes if possible. Anything less than 2-56 inches requires manual tapping, which increases a lot of time and risk.

2.6Avoiding tight tolerance

Tolerances usually increase processing costs and time and should be specified only when necessary. The precision of CNC machine tool is very high, when there is no custom tolerance in the design, standard tolerance will be produced. Tight tolerances should be specified only when necessary as they are difficult to apply. They also require more processing time and manual inspection. We also recommend that you use GD & T in your drawings, which define more relaxed tolerances.

2.7Complex parts, the less the better

In some cases, if complex parts are designed and processed as separate parts, then welded or bolted together, the cost will be lower.

This is especially true for parts with deep grooves, which will require hours of processing time to remove material, plus additional material costs.

The same is true for parts that need to operate on multiple faces, which increases the fixation and setup time.

3 >> Materials and finishes for CNC machining

Material selection an incredibly important part of any manufacturing process, but engineers and product teams must be especially diligent when it comes to selecting materials for CNC machining. Since this process is compatible with a wide variety of materials — from metal to plastic — it's easy to mistakenly choose a material that's sufficient for the project but not the best-suited. In this article, we'll touch on key considerations for CNC machining material selection and give an overview of some common options.

3.1Overview of common CNC metal alloys

Material	Grade	Strenght 1-3		Hardness1-4	Machinablility1-5	Cost1-5	Chemical resistance 1-5
Aluminum	6061-T6	2	291-320 Mpa	2	5	1	4
	6082	2	140-340 Mpa	2	5	1	5
	7075-T6	3	434-580 Mpa	2	4	3	2
	5083	2	270-350 Mpa	1	5	2	5
Stainless steel	304	3	480-620 Mpa	2	2	3	3
	316	3	480-620 Mpa	2	2	4	5
	2205Duplex	3	640-950 Mpa	3	1	5	5
	303	3	528-639 Mpa	3	3	4	2
	"17-4	3	896-990 Mpa	4	2	5	5
Mild steel	1018	2	190-440 Mpa	2	3	2	1
	1045	3	620-680 Mpa	3	2	3	1
	A36	3	400-550 Mpa	2	3	2	1
Alloy steel	4140	3	915-1130 Mpa	3	2	3	2--1
	4340	3	670-820 Mpa	3	2	3	2--1
Tool steel	D2	3	2100-2500 Mpa	4	1	4	
	A2	3	2040-2360 Mpa	4	1	4	
	O1	3	2040-2360 Mpa	4	1	4	
Brass	C360	2	140 Mpa	2	5	2	2

Aluminum alloys have an excellent strength-to-weight ratio, a high thermal and electrical conductivity and natural protection against corrosion. They can be easily machined and have a low bulk cost, so they are often the most economical option of creating custom metal parts and prototypes.

Aluminium alloys typically have a lower strength and hardness than steels, but they can be anodized, creating a hard, protective layer on their surface.

Stainless steel alloys have high strength, high ductility, excellent wear and corrosion resistance and can be easily welded, machined and polished. Depending on their composition, they can be either (essentially) non-magnetic or magnetic.

Mild steels are also known as low-carbon steels and have good mechanical properties, great machinability, and good weldability. Due to their low cost, they find general-purpose applications, including the manufacturing of machine parts, jigs, and fixtures. Mild steels are susceptible to corrosion and attacks from chemicals.

Alloy steels contain other alloying elements in addition to carbon, resulting in improved hardness, toughness, fatigue and wear resistance. Similarly to mild steels, alloy steels are susceptible to corrosion and attacks from chemicals.

Tool steels are metal alloys with exceptionally high hardness, stiffness, abrasion and thermal resistance. They are used to create manufacturing tools (hence the name), such as dies, stamps, and molds. To achieve their good mechanical properties, they must undergo a heat treatments.

Brass is a metal alloy with good machinability and excellent electrical conductivity, ideal for applications that require low friction. It is also commonly used in architecture to create parts with a golden appearance for aesthetic purposes.

3.2Overview of common CNC plastics

Material	Strenght 1-3	Operating temperature	Machinablility 1-5	Cost1-5
ABS	2	up to 60℃	3	2
Nylon	3	up to 100℃	1	2
PC	3	up to 120℃	2	2
POM(Delrin)	2	up to 82℃	4	2
PTFE(Teflon)	1	up to 260℃	4	2
HDPE	1	up to 80℃	1	2
PEEK	3	up to 260℃	2	4

ABS is one of the most common thermoplastic materials offering good mechanical properties, excellent impact strength, high heat resistance and good machinability.

Nylon is also known as polyamide (PA) and is a thermoplastic that is often used for engineering applications, due to its excellent mechanical properties, good impact strength and high chemical and abrasion resistance. It is susceptible to water and moisture absorption though.

Polycarbonate is a thermoplastic with high toughness, good machinability and excellent impact strength (better than ABS). It can be colored, but typically it is optically transparent, making it ideal for a wide range of applications, including fluidic devices or automotive glazing.

POM is commonly known with the commercial name Delrin, and it is an engineering thermoplastic with the highest machinability among plastics.
POM (Delrin) is often the best choice when CNC machining plastic parts that require high precision, high stiffness, low friction, excellent dimensional stability at elevated temperatures and very low water absorption.

PTFE is commonly known as Teflon, and is an engineering thermoplastic with excellent chemical and thermal resistance and the lowest coefficient of friction of any known solid.

PTFE (Teflon) is one of the few plastics that can withstand operational temperatures above 200oC and is an outstanding electrical insulator. Nevertheless, it has pure mechanical properties and it is often used as a lining or insert in an assembly.

High-density polyethylene (HDPE) is a thermoplastic with a high strength-to-weight ratio, high impact strength, and good weather resistance.

HDPE is a lightweight thermoplastic, suitable for outdoor use and piping. Like ABS, it is often used to create prototypes before Injection Molding.

PEEK is a high-performance engineering thermoplastic with excellent mechanical properties, thermal stability over a very wide range of temperatures and excellent resistance to most chemicals.

PEEK is often used to replace metal parts due to its high strength-to-weight ratio. Medical grades are also available, making PEEK suitable also for biomedical application.

3.3 Surface finishes for CNC machining

Surface finishes are applied after machining and can change the appearance, surface roughness, hardness and chemical resistance of the produced parts. Below is a quick summary of the most common finishes for CNC.

As machined

CNC machining produces a part with an “as-machined” or “as-milled” finish as soon as the manufacturing process is completed. The part will have small but visible tool marks and blemishes. The average surface roughness is around 3.2 μm. As-machined parts have the tightest dimensional tolerances and are extremely affordable to produce because post-processing isn’t necessary.

This finish is a good choice for those who are more concerned with dimensional integrity than aesthetics. However, parts with as-machined finishes don’t rank very highly when it comes to protection. Their roughness and lack of protective coating renders them susceptible to nicking, scuffing, and scratching.

Anodizing

Type II decorative anodizing

Type III hardcoat anodizing

Anodizing is an electrochemical process that thickens a CNC machine part's natural oxide layer to make it thicker, denser, electrically non-conductive, and more durable.

This process can only be done using aluminum or titanium alloys because they conduct electricity well. During anodizing, the alloy is submerged in an acid electrolyte bath and acts as an anode. Once a cathode is placed in the anodizing tank and an electrical current passes through the acid, oxygen ions from the electrolyte and atoms from the alloy combine at the surface of the part.

Anodizing comes in two different varieties — Type II and Type III. The overall finishing process is the same, but Types II and III require that the part be submerged in a diluted sulfuric acid solution.

Anodizing Type II, also known as "decorative anodizing" (as the finished coating can be clear or colored), produces coatings up to 25 μm thick. The coating thickness range for clear parts is 4-8 μm and 8-12 μm for parts that have been dyed black. This process produces a part that is smooth, elegant, and resistant to corrosion and wear.

Anodizing Type III, also known as "hardcoat anodizing," can produce anodic coatings up to 125 μm thick. Parts with this coating have high density and are even more wear-resistant than anodizing Type II.

All in all, anodized finishes are durable and promise good dimensional control. Anodized finishes are best used in high-performance engineering applications, particularly for internal cavities and small parts. They are among the most aesthetically pleasing finishes for CNC machined parts, but often come at a higher price tag.

Powder coating

Powder coating is a lot like spray painting. First, the part is primed with a phosphating or chromating coat to make it more resistant to corrosion. Then, the part is "painted" with a dry powder coating from an electrostatic spray gun and cured in an oven heated to at least 200°C.

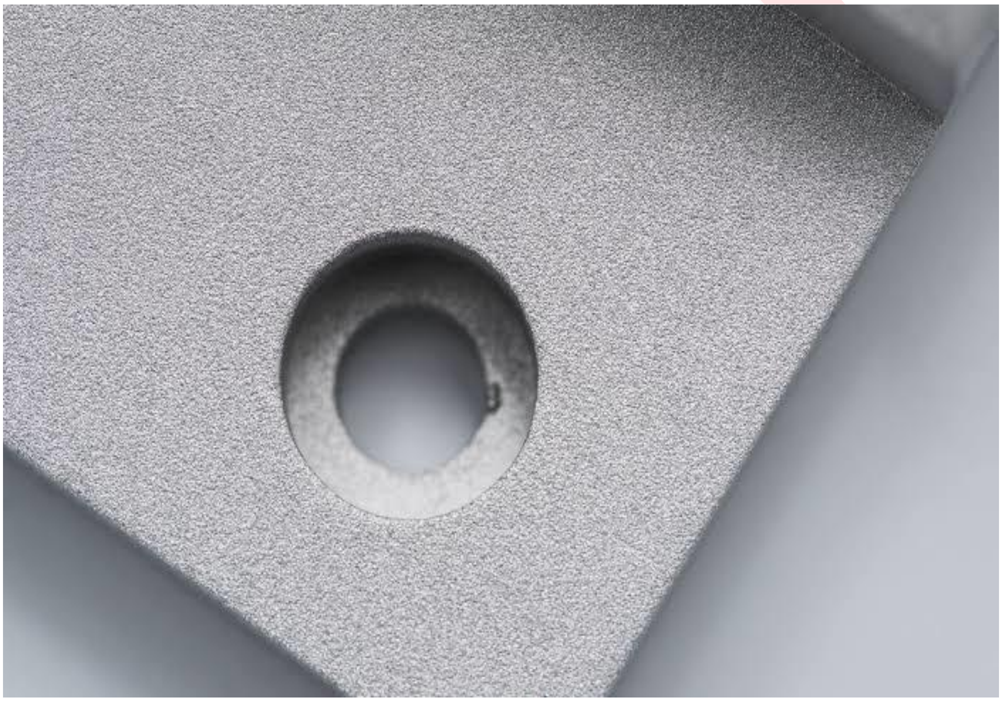
Multiple layers can be applied to increase thickness, which can reach 72 µm.

On its own, this finish creates a thin protective layer on the CNC machined part that is strong, wear-resistant, and aesthetically pleasing. This process can be combined with bead blasting to increase the part's corrosion resistance and create greater uniformity in texture and appearance.

Unlike anodizing, a powder-coated finish is compatible with all metals, less brittle, and offers greater impact resistance. This finish is suitable for many functional applications but may be particularly well-suited for military applications.

However, powder coating generally yields less dimensional control than an anodic finish, and powder coating is not recommended for use in small components or internal surfaces. What's more, powder coating's higher price point might make it prohibitively expensive for larger production runs.

Bead blasting



Bead blasting is used to add a matte or satin surface finish to a CNC machined part. During this process, a pressurized air gun shoots millions of glass beads at the part, effectively removing tool marks and imperfections, creating a consistent grainy finish. In contrast to other finishes, including anodizing and powder coating, bead blasting adds no chemical or mechanical properties to the part — it's purely visual. Unlike powder coating, which adds material to a part, bead blasting is a reductive finish, meaning it removes material from the part. This is an important consideration if your part has strict tolerances.

Bead blasting is one of the most affordable surface finishes, but it must be executed manually. As such, those considering bead blasting as a surface finish should be prepared to incur the cost of engaging an operator who has been formally trained in this process, and recognize that the final result will largely depend on how skilled the operator is. Bead size and grade will also affect the final finish.

In short, a post-processing finish such as anodizing or powder coating will likely prove an effective option for parts that don’t need to be picture-perfect but must maintain their original dimensions. Protecting or reinforcing an aluminum or titanium part may benefit from anodizing. If the part cannot be anodized but requires strength and impact resistance, powder coating provides an effective alternative. Finally, if cost-effectiveness is a higher priority than tolerance — and the part does not require a glossy finish — bead blasting may be the preferable route.

3 >> Cost reduction tips

If you’re interested in CNC machining cost reduction, first you have to understand the factors that impact the cost of CNC parts — machining time, start-up costs, material costs and more. Here is some background information on each of these factors that can be used to ultimately reduce CNC machining cost.

Machining Time: This is usually the primary cost driver when it comes to CNC machining. As a rule of thumb, the longer a part takes to machine, the more it will cost.
Start-up Costs: These costs are fixed. They are in place to cover the cost of CAD file preparation and process planning. For small volumes, start-up costs can be significant.
Material Costs: The cost of the bulk material you choose and how easy or hard the material is to machine is another big contributor to the cost of CNC machining.
Other Manufacturing Costs: Such as multiple finishes and so on.

4.1 7 design tips for CNC machining

Since machining time mostly depends on the design of product, 7 tips on design on chapter 2 will help you reduce the cost accordingly.

4.2Order large quantities

Since the start-up cost is fixed, the number of parts you order has a significant impact on the unit price. What's known as the economies of scale can be a powerful tool as you figure out how to reduce CNC machining costs. Increasing the quantity from one to fifty could decrease the unit price by more than 50 percent .

Consider ordering a higher quantity the next time you place an order.

4.3 Machinability of the materials

Machinability means that the material is easy to cut. The higher the machinability, the faster the CNC machining speed and the cost is lower.

The machinability of each material depends on its physical properties. In general, the softer the metal alloy, the easier it is to process.

Brass C360 is the metal with the highest machinability, allowing high-speed machining. Aluminum alloys such as aluminum 6061 and aluminum 7075 can also be easily machined.

The machinability of steel is 10 times lower than that of aluminum, and the processing time of steel is at least twice as long as that of aluminum. Please note that different steel grades have different machinability. For example, stainless steel 304 (the most common stainless steel alloy) has a machinability index of 45%, while stainless steel 303 (an alloy with very similar chemical composition) has a machinability index of 78%, making it easier to process.

The processability of plastics mainly depends on its stiffness and thermal properties. In the process of CNC machining, plastic is easy to melt and warp.

POM is the easiest plastic to process, followed by ABS. Peek and nylon are other common engineering plastics, which are difficult to process.

If you need help in selecting materials, please stay tuned for our next newsletter, in which we compare the most common materials used in CNC machining.

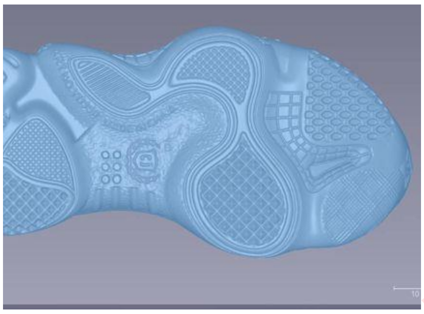
4.4 Reduce multiple finishes

The surface finish improves the appearance of CNC machining parts and the ability to resist harsh environment, but also increases the cost.

Requiring multiple different surface treatments on the same part further increases the price because additional steps are required.

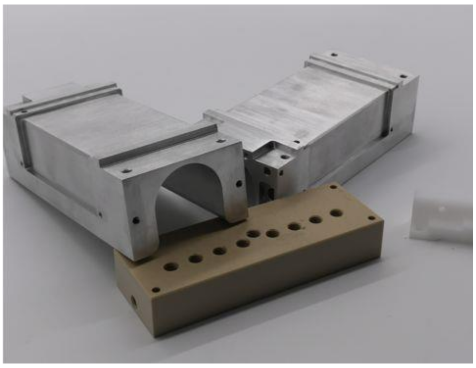
To minimize costs: 1, Select machined surface finish. 2, Multiple surface treatments are required only when absolutely necessary.

4 >> **About IN3DTEC** Our services



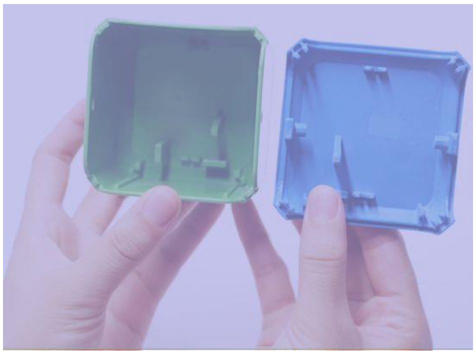
[3D Scanning & Reverse engineering>>](#)

[3D Printing Service>>](#)

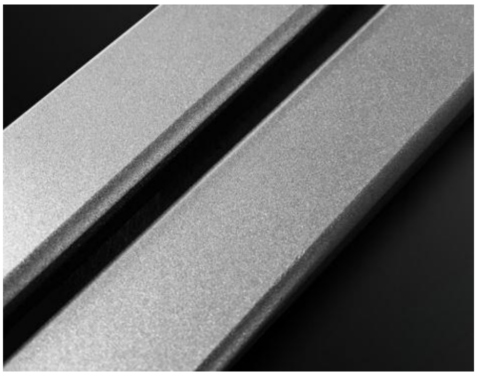


[CNC Machining Service>>](#)

[Vacuum Casting Service>>](#)



[Injection Molding Service>>](#)



[Surface finishes Service>>](#)

Thank you very much for taking the time to read this manual, and we welcome your comments or suggestions.

IN3DTEC

Bring your concept into reality

3DTEC was established in 2014 by a professional team in Shanghai with branches in Hong Kong, Suzhou, Shenzhen. We provide a full spectrum of technologies including 3D scanning, 3D printing, CNC machining, Vacuum Casting, Injection Molding and more manufacturing services, which enable us to become a one-stop station from prototyping to on-demand production.

In order to meet different market needs, we expend our offerings to service Aerospace, Automotive, Education, Electricals, Jigs ＆ Fixtures, Medical, Oil ＆ Gas, Transportation.

www.in3dtec.com; info@in3dtec.com; +86-21-5059 9658