Introduction to CNC machining

Written by Alkaios Bournias Varotsis

What is CNC machining?

CNC machining is the most widely used **subtractive manufacturing technology**. In CNC, material is removed from a solid block using a variety of cutting tools to produce a part based on a CAD model. Both metals and plastics can be machined with CNC.

CNC produces parts with **tight tolerances** and excellent **material properties**. CNC is suitable for both **one-off jobs** and **low-to-medium volume production** (up to 1000 parts), due to its high repeatability. When compared to 3D printing though, CNC has more design restriction, due to the subtractive nature of the technology.

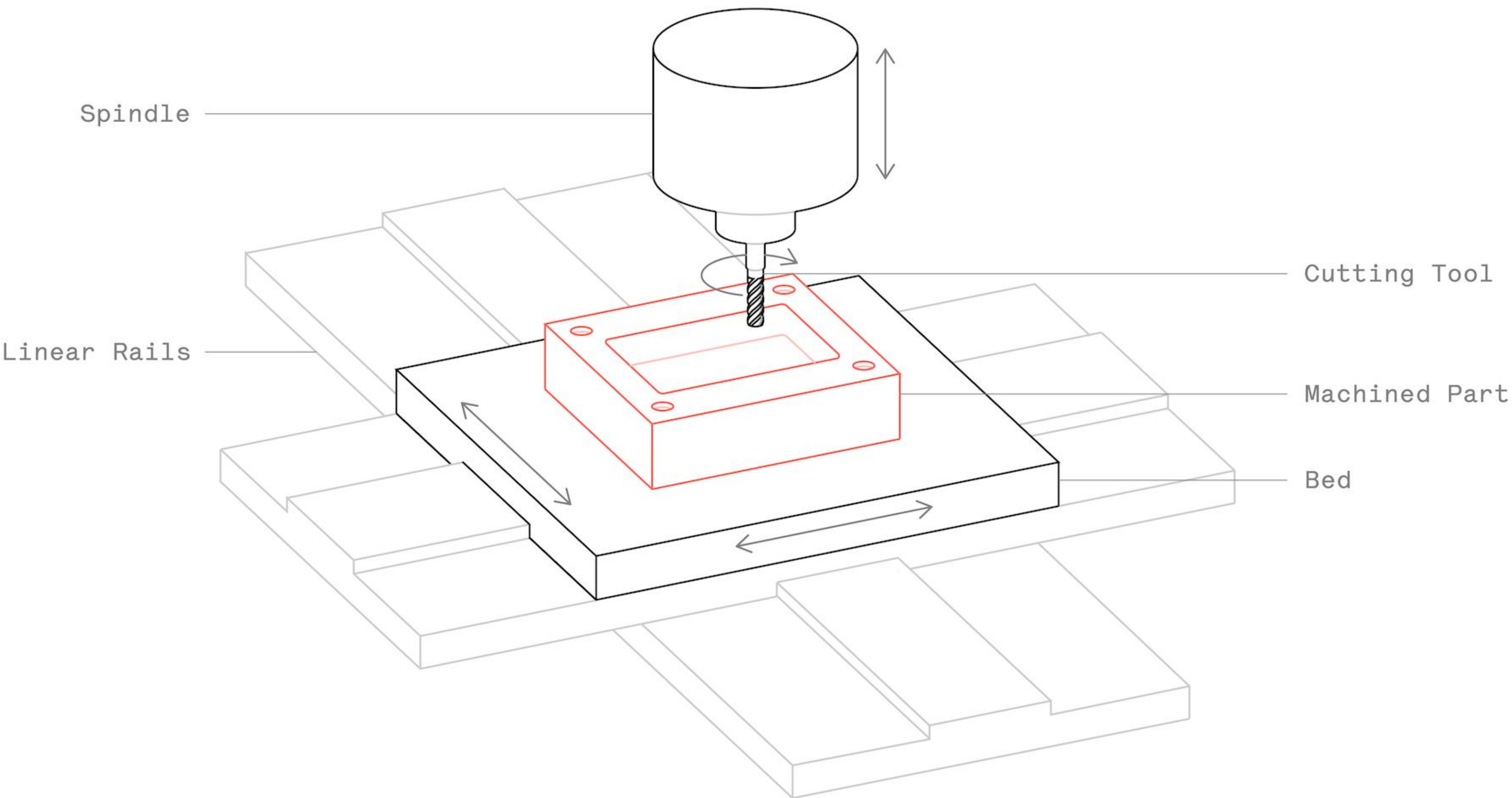
In this article, the differences between the two main CNC machine setups (milling and turning) are first explained, and then, the **characteristics** of the process are discussed. After reading this article, you will have an overview of the basic principles of the technology and how these relate to its key **benefits and limitations**.

How does CNC machining work?

There are two main types of CNC machining systems: **milling** and **turning**. Each is better suited for manufacturing different geometries, due to its unique characteristics.

Let's break down how parts are manufactured using these two machine setups...

How does CNC milling work?



Schematic of a typical CNC milling machine

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:

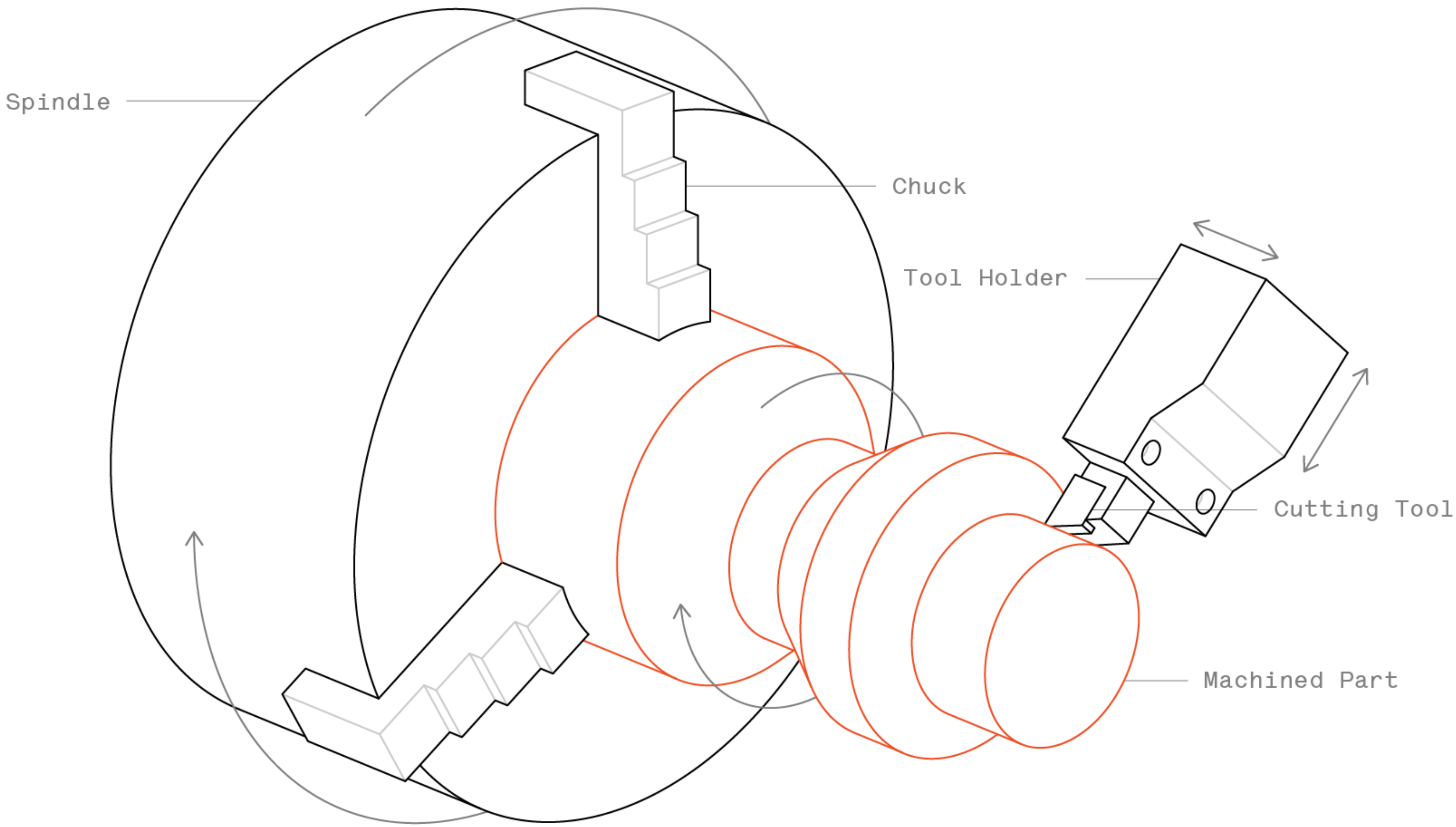
1. First, the CAD model is converted into a series of commands that can be interpreted by the CNC machine (G-code). This is usually done on the machine by its operator, using the provided technical drawings.
2. A block of material (called the blank or the workpiece) is then cut to size and it is placed 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.


A typical CNC milled part, manufactured by removing material from a rectangular blank

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?



Schematic of a typical CNC turning machine

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.

A typical CNC turned part, manufactured by removing material from a cylindrical blank

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.

Machine Parameters

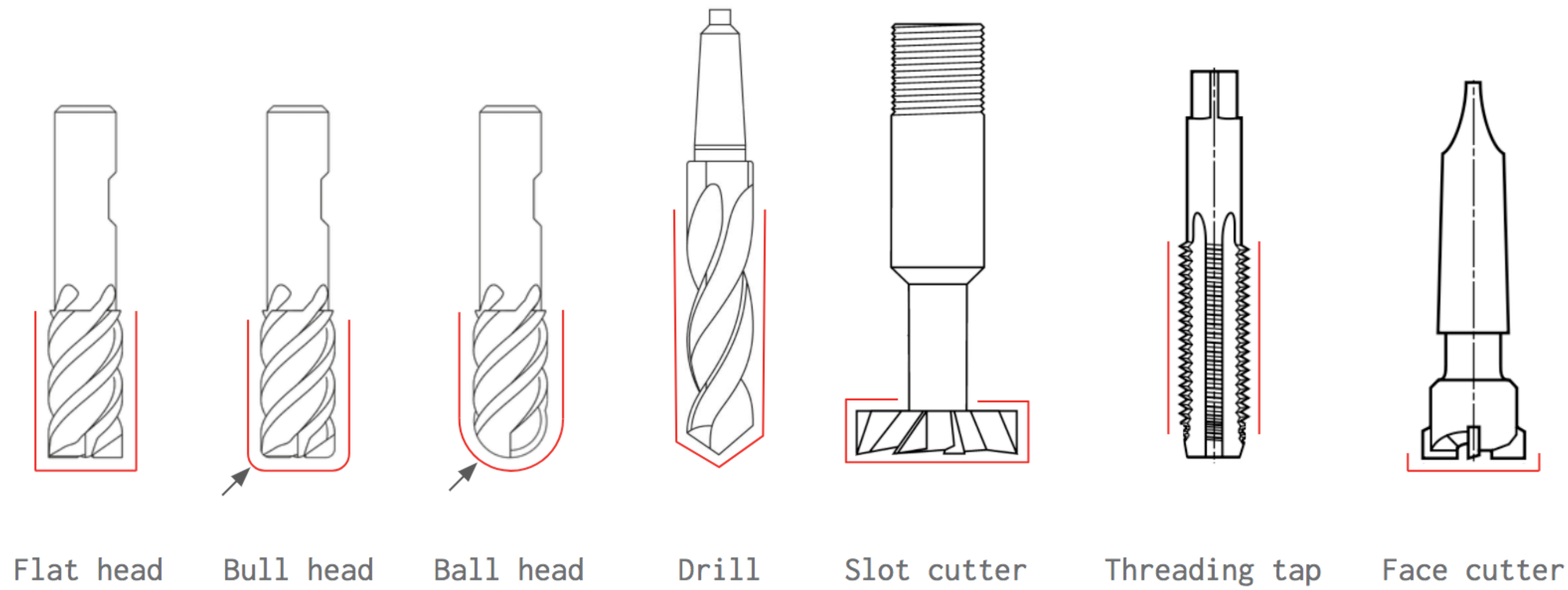
Most machining parameters are determined by the machine operator during the generation of the G-code and are usually of little interest to the designer. The machine parameters of interest are the **build size** and the **accuracy** of the CNC machine:

CNC machines have a **large build area**. CNC milling systems can machine parts with dimensions of up to 2000 x 800 x 100 mm (78'' x 32'' x 40'') and CNC turning systems can manufacture parts with diameter of up to Ø 500 mm (Ø 20'').

With CNC machining, parts with **high accuracy** and **tight tolerances** can be manufactured. If no tolerance is specified, then parts will be machined with a typical accuracy of ± 0.125 mm (.005''). Tight tolerances down to **less than half the diameter of an average human hair** (± 0.025 mm or .001'') can be achieved with CNC.

CNC Cutting Tools

To create different geometries, CNC machines use different cutting tools. Here are some of the most commonly used **milling tools** in CNC:



A selection of the most common CNC cutting tools (not in scale)

The **flat head**, **bull head** and **ball head end mill tools** are used to machine slots, grooves, cavities and other vertical walls. Their different geometry allows the machining of features with different details. Ball head tools are also commonly used in 5-axis CNC machining to manufacture surfaces with curvature and freeform geometries.

Drills are a common and fast way to create holes. You can find tables with the standard drill sizes here. For hole with non-standard diameter, a plunging flat head end mill tool (following a helical path) can used.

The diameter of the shaft of **slot cutters** is smaller than the diameter of their cutting edge, allowing these milling tools to cut T-slots and other undercuts by removing material from the sides of vertical wall.

Taps are used to manufacture threaded holes. To create a thread, precise control of the rotational and linear speed of the tap is needed. Manual tapping is also still commonly used in some machine shops.

Face milling cutters are used to remove materials from large flat surfaces. They have a larger diameter than end mill tools, so they require fewer passes to machine large areas, reducing the total machining time and producing flat surfaces. A face milling step is often employed early in the machining cycle to prepare the dimensions of the block.

An equally large range of cutting tools are also used in **CNC turning**, covering all machining needs, such as face cutting, threading and groove cutting.

Here is a video of a face milling cutter in action:

Geometric Complexity & Design Restrictions

CNC offers great design freedom, but **not every geometry can be CNC machined**. Unlike 3D printing, part complexity increases the cost, as more manufacturing steps are required.

5-axis CNC systems allow the cutting tool to access areas that are impossible to reach with 3-axis systems

The main restrictions in CNC have to do with the **geometry of the cutting tool**. For example, the internal edges of a slot will always be rounded, as they are machined using a tool with a cylindrical profile.

Tool access is another major restriction in CNC: material cannot be removed unless the tool can reach that area. Most CNC machines are 3-axis systems, so any feature must be designed so that it can be accessed directly from above. 5-axis CNC systems offer greater flexibility, allowing the creation of more intricate parts, as the angle between the part and the tool can be adjusted to gain access to difficult to reach areas.

Parts with **thin walls** or other **fine features** are difficult to CNC machine. Thin walls are prone to vibrations and are in danger of breaking due to the cutting forces. The minimum recommended wall thickness is 0.8 mm for metals and 1.5 mm for plastics.

An article with more design guidelines specifically for CNC machining can be found here.

Characteristics of CNC machining

A key strength of CNC machining is its ability to produce parts with **excellent material properties** from a very wide selection of materials: practically **all engineering materials** can be CNC machined.

In contrast with 3D printing, parts manufactured with CNC machining have fully-isotropic physical properties that are identical to the properties of the bulk material they were machined out of.

CNC machining is predominately used with metals, both for prototyping and larger production runs. Plastics are generally more difficult to machine, as they have lower stiffness and melting temperature. A common use case of plastic CNC machined parts, is the creation of functional prototypes before large-scale production with Injection Molding.

CNC machining materials

The **cost** of CNC materials varies greatly. For metals, Aluminum 6061 is the most economical option, at an approximate bulk cost of $25 for a blank with dimensions of 150 x 150 x 25 mm, while for plastics ABS has the lowest cost, at approximately $17 for a blank of the same size. The physical properties of a material can also greatly influence the overall cost of CNC. For example, Stainless Steel is much harder than Aluminium and that makes it more difficult to machine, increasing the overall cost.

Material	Characteristics
Aluminum 6061	Good strength-to-weight ratio Excellent machinability Low hardness
Stainless Steel 304	Excellent mechanical properties Excellent corrosion & acid resistance Relatively difficult to machine
Brass C360	High ductility Excellent machinability Good corrosion resistance
ABS	Excellent impact resistance Good mechanical properties Susceptible to solvents
Nylon (PA6 & PA66)	Excellent mechanical properties High toughness Poor moisture resistance
POM (Delrin)	High stiffness Excellent thermal & electrical properties Relatively brittle

Post processing & finishes

As machined CNC parts will have visible tool marks. Various post-processing methods can be employed to smoothen their surface and improve their wear, corrosion or chemical resistance and their visual appearance, such as **anodizing**, **bead blasting** and **powder coating**.

An article explaining the most common finishes in CNC can be found here.

A CNC machined part that has being anodized and dyed blue

Benefits & Limitations of CNC machining

The key advantages and disadvantages of the technology are summarised below:

CNC machining offers excellent accuracy and repeatability, and can produce parts with very tight tolerances, making it ideal for high-end applications
CNC materials have excellent and fully-isotropic physical properties and are suitable for most engineering applications.
CNC is the most cost-effective manufacturing process for the producing low-to-medium numbers of metal parts (from one-off prototypes, up to 1000 units)
Due to the subtractive nature of CNC machining, certain geometries are either very costly or impossible to manufacture.
The start-up cost of CNC machining is high compared to 3D printing, so CNC is less suited for low-cost prototyping (especially for plastics).
The lead times of CNC machining (10 days) are longer than the lead times of 3D printing (2-5 days), as CNC machines are not as widely available, since they require expert knowledge to opperate.

The main machine parameters of CNC machining are summarized in the table below:

	CNC machining
Materials	Metals & plastics
Dimensional accuracy	Typical: ± 0.125 mm (.005’’) Maximum: ± 0.025 mm (.001’’)
Minimum wall thickness	Metals: 0.75 mm (.030’’) Plastics: 1.5 mm (.060’’)
Maximum build size	Milling: 2000 x 800 x 100 mm (78’’ x 32’’ x 40’’) Turning: Ø 500 mm (Ø 20’’)

Rules of Thumb

- CNC machining is excellent for both one-off jobs and low-to-medium volume production (100's of part and up to 1000 parts).
- Use CNC machining for your metal prototypes, as it is the most price-competitive option.

- Use CNC machining when parts with the tightest tolerances are required.