MALOYA
METAL FABRICATION & MANUFACTURING

From Concept To Manufacturing

Everything You Need To Know About CNC Machining

From the Basics to Cost Reduction Techniques, this guide walks you through everything you need to know about CNC Machining.

Table of Contents

Fundamentals of CNC Machining

To get from concept to manufacturing, it’s helpful to have a solid foundation of knowledge about what CNC machining is and its many applications. That’s why we’ve created this guide, which covers the fundamentals and major concepts you need to know from start to finish.

DEFINITION & PROCESS

CNC Machining is a digital technology that uses CAD files to manufacture parts with a high level of accuracy and with superior physical properties.

Unlike additive or formative technologies like 3D and injection molding, CNC machining uses material removal methods to form the final product.

An engineer designs a part with a CAD model which is then converted into a CNC program by the machinist, who then sets up the machine. The automated CNC system performs all the machining, removing material to execute the final design.

CNC machining is highly versatile and appropriate for just about any material, including metals, plastics, foam, composites, and wood.

Because so much of the CNC machining process is automated, it offers highly competitive pricing for projects that involve one-off, custom parts or for mid-range-volume productions.

HOW DO I KNOW IF CNC MACHINING IS THE BEST CHOICE FOR MY PROJECT?

Sometimes it's easier to understand this machining approach in the context of similar methods to better answer the basic question of whether CNC machining is the right process for your project.

Undoubtedly, 3D printing technology revolutionized the machining industry. But it's not right for every project (or budget).

Other technologies are great for extremely high-volume projects but would be cost-prohibitive for projects requiring 1,000 parts or less.

So how do you know if CNC machining is the right approach for your project?

WHEN IS 3D PRINTING A GOOD CHOICE?

If you're looking to manufacture an inexpensive plastic prototype or a part designed using specialty materials, 3D printing may be your best bet.

3D printing is a good choice if your part has intricate geometrical requirements, or if you're looking for a speedy turnaround. SLS metal 3D printing allows for highly complex exotic designs with little to no "set up costs."

Unlike CNC manufacturing, however, 3D printing doesn't scale well for larger volumes since the unit price is fairly stable.

WHAT ABOUT FORMING TECHNOLOGIES?

When you're seeking economy of scale, investment or die casting for extremely high-volume metal part projects is a good choice.

Similarly, if your design requires plastic and you need a lot of parts, injection molding could be your best option.

Either of these methods is generally only cost-effective for 1,000 or more parts and both involve very high start-up costs.

WHEN IS CNC MACHINING THE BEST OPTION?

There are some basic guidelines to follow when making this decision, but the bottom line is that if your part design is conducive to metal, involves relatively simple geometry, and can be manufactured using a subtractive process (more on that later) CNC machining is an excellent choice.

CNC manufacturing is less expensive for small-to-medium volume and one-off projects while offering a greater level of dimensional accuracy with superior mechanical properties.

Because CNC machined parts are characterized by exceptional physical properties, they're ideal for designs requiring very high levels of performance and tight tolerances. Also, because such a wide variety of hard materials can be CNC machined, designers have a great deal of flexibility when drafting their concept.

And because this process doesn't require special tooling, on-demand or custom one-off parts and prototypes aren't cost-prohibitive to produce.

The CNC machining process involves a significant start-up investment, mainly as a result of the process planning aspect. (Unlike 3D printing, for example, CNC machining requires the input of a specialist at the outset which can be expensive.)

But because CNC start-up costs are fixed and can be spread over multiple parts, designers can take advantage of economies of scale since the cost per unit goes down with higher volumes.

However, since CNC uses subtractive technology, costs go up with more complex geometries, so simpler projects will be less expensive to produce. (Bear in mind, CNC machining generally comes with more design restrictions. Parts with internal geometry or steep undercuts would be difficult to full machine.)

State of the art CNC technology, and CAM software packages have all made it possible to achieve accelerated production times, sometimes with turnaround times comparable to 3D printing, at a very low cost.

3-Axis vs. 5-Axis Machining Basics

CNC milling and turning machines are a very basic method of machining which allows for movement of the cutting tool itself within three axes relative to the part being formed, those being up-down, back-forth, and right-left axes.

3-AXIS CNC MILLING

These workhorse machines are very common since they accommodate most regular geometries, and because they're relatively easy to program and operate, using them for a project tends to be highly cost-effective.

However, limited tool access can mean design restrictions apply since with this type of machine not all areas of the part may be easily accessible (or accessible at all). And if a workpiece needs to be repositioned multiple times to gain access, labor and machining costs are higher, commensurate with this added layer of complexity.

3-AXIS CNC TURNING (LATHES)

Unlike CNC milling, CNC lathing can produce parts much faster and at a lower price point per unit. Obviously, this has benefits for high-volume projects.

A major difference between turning and milling is that 3-axis CNC turning is restricted to designs with cylindrical profiles. So if a part requires multiple profiles, it means an added step to mill the non-cylindrical element separately.

Summary of 3- & 5-Axis CNC Machining Characteristics

	Workpiece Orientation	Process	Tool Operation	Benefits	Limitations
3-Axis CNC Milling	Part is held stationary on the machine bed or in a vice.	Cutting tools or drills rotate at a very high rate to remove material from the workpiece.	Tools are attached to a spindle with capability to move along a 3-linear axis.	Most parts with simple geometries can be produced with this type of machine, and can be manufactured with high-level accuracy and extremely tight tolerances at a lower cost.	Designs are limited by tool access and workholding restrictions, and any manual repositioning required for more complex designs can compromise accuracy.
CNC 3-Axis Turning (Lathes)	Spindle holds workpiece and rotates at a very high speed.	A center drill or cutting tool forms the geometry by tracing the inner or outer perimeter.	The tool itself does not rotate. Movements are restricted to radial and lengthwise directions.	Can accommodate very high volume production requirements at the lowest cost per unit than any other CNC machining method.	Can only accommodate designs with rotational symmetry and simple geometric profiles.

5-AXIS CNC MACHINING

There are three system variations for multi-axis CNC machining, but they're all fundamentally based on milling or lathing technology:

1. 5-Axis indexed milling
2. Continuous 5-axis milling
3. Mill-turning centers with live tooling

These are all basically milling or lathing machines that offer a wider degree of freedom than 3-axis CNC machines. For example, a part with multiple geometric profiles can be produced in one single step as opposed to with 3-axis CNC turning.

To illustrate with an example, 5-axis CNC milling centers let the machinist rotate the machine bed or the tool head (or even both) in addition to the three linear axis points (up-down, right-left and back-forth).

This technology comes with a slightly higher price tag, not just because of the advanced capabilities it offers and the necessary expertise to execute those capabilities, but because it requires specialized machinery.

5-axis CNC machining isn't ideal for projects involving a complex topography or intricate design. Such projects likely would be better suited to 3D printing.

BENEFITS

INDEXED 5-AXIS CNC MILLING

These milling systems provide two additional levels of freedom which is why they are sometimes referred to as 3+2 CNC milling machines.

During the machining process, the tool doing the cutting can move along the same three linear axes as discussed above, but in between operations, the tool head, as well as the bed, can be repositioned without the need to manually rotate the workpiece. This ability to rotate tools and beds gives a wider range of access points for machining more complex geometries.

Costs are higher than for 3-axis machines, but an indexed 5-Axis milling machine can produce parts with more involved geometry, at a faster rate, and with higher accuracy levels than any of the 3-axis CNC machines.

However, if a design requires highly accurate contoured surfaces, it's likely a continuous 5-axis CNC milling machine is the best option.

CONTINUOUS 5-AXIS CNC MILLING

These Although these machines have many of the same architectural features as the indexed 5-axis milling machine, they provide a lot more freedom since all five axes can move at the same time during machining operations.

Although some access restrictions still apply, the cutting tool can move along the aforementioned three linear axes, as well as along two rotational axes relative to the workpiece which provides the ability to manufacture parts with more complex geometries with a smooth surface and fewer machining marks.

Plus, they offer these capabilities at a much higher level of accuracy than any other CNC machining method. (Bear in mind this premium performance level comes at a higher cost per part than all the other processes.)

5-AXIS MILL-TURNING

5-axis mill turning offers the lowest cost of all the 5-axis CNC machining methods. These machines leverage the productivity of turning with the flexibility of milling since both lathe and milling tools are used to remove material to achieve the final workpiece geometry.

To do this, a spindle rotates at either a high rate of speed (similar to a lathe) or much like the 5-axis CNC mill, the workpiece can be positioned at a precise angle.

5-axis mill turning machines offer low-cost high-production capability with a lot more design freedom than other methods and are best suited for manufacturing cylindrical parts with irregular rotational symmetry.

Summary of 3- & 5-Axis CNC Machining Characteristics

3-Axis CNC Milling	3-Axis CNC Turning	Indexed 5-Axis CNC Milling	Continuous 5-Axis CNC Milling	5-Axis Mill-Turning
Low cost, high accuracy, best for manufacturing parts with simple geometries.	Lowest cost per unit but are restricted to designs that involve rotational symmetry.	Best for parts with features that don't align with one of the main axis, provides very high accuracy and quickly.	Highest-cost per part of all CNC machining methods, capable of executing highly complex designs involving intricate geometries and smooth contours.	Lowest cost of all 5-axis systems, combines lathing and milling capabilities to manufacture complex parts.

Applications

There are many reasons people choose CNC machining over other methods for their manufacturing projects. Let's take a look at just a few of the wide range of applications:

SPACE

Because CNC machining is an excellent choice when high accuracy and material properties are paramount, this process is one of only a very few that are suitable for space applications. Additionally, this process provides a wide variety of choices for surface treatments that may be applied to parts following CNC machining.

AEROSPACE

Used for both the development stages and for aircraft parts, CNC is ideal for manufacturing lightweight parts with superior physical properties and extremely tight tolerances. In fact, aerospace was one of the very first industries to use CNC machining for just these reasons.

AUTOMOTIVE

From prototyping to manufacturing, CNC machining is a popular application in the automotive sector, since high-performance custom parts are required.

PRODUCT DESIGN & DEVELOPMENT

Because it's possible to produce custom parts with required levels of detail for many projects, CNC machining is a common choice for manufacturing metal parts with superior dimensional accuracy for functional prototypes. This is of course, hugely important during later stages of design and development.

ELECTRICAL & ELECTRONIC MANUFACTURING

From enclosures to high-power electronic sensing systems to PCB prototyping, CNC machining has a wide range of applications in the electrical and electronic manufacturing industry.

TOOLING & INDUSTRIAL MANUFACTURING

Though a project might be ultimately designed for other applications like injection molding, CNC machining is often used in the fabrication of necessary parts such as the molds themselves.

SPORTS & MOTORSPORTS

CNC manufacturing is ideal for producing sports and motorsports equipment because it allows companies in this industry to increase performance while reducing weight.

Designing For CNC Machining

As with any manufacturing processes, CNC machining is limited by some design restrictions which are largely the result of the mechanics of the cutting process.

The major areas of design restriction are:

TOOL ACCESS

If a surface can't be reached by a cutting tool, it can't be CNC machined. So parts designed with internal or "hidden" geometries or with deep undercuts aren't ideal candidates for CNC machining process.

TOOL GEOMETRY & STIFFNESS

The vast majority of CNC cutting tools have cylindrical features with a flat (or sometimes spherical) head. So internal vertical corners of a part will always introduce a radius, even if smaller tools are used.

WORKHOLDING

This design restriction comes into play when considering the benefits and drawbacks of 3-axis versus 5-axis machining processes.

The geometry of a part dictates how the CNC machine will hold it, and will also determine the number of required set-ups. If there are multiple set-ups, manual repositioning can produce small but still significant positional errors.

Maloya solves some of these issues by machining custom "soft jaws" to constrain the work piece. Custom holders can also improve efficiency by holding multiple work pieces at once. Having one or more of these holder greatly improves efficience by minimizing load/unload times. Maloya also utilizes vacuum chucks to hold thinner flat parts which may not be suitable to clamp in a vise.

WORKPIECE STIFFNESS

CNC machined parts are limited by wall thickness and aspect ratios of taller features because, during the cutting process, vibrational movements and process-generated temperature changes can cause deformities. As well as tool deflection as longer tools are needed for geometry with deeper/taller features.

BASIC CNC MACHINING DESIGN RULES

When designing for CNC machining, there are recommendations and feasibility aspects to consider for different scenarios:

TALL FEATURES

Tall features are prone to vibrations, and that makes them difficult to machine accurately. The recommended maximum aspect ratio is less than 4.

CAVATIES & POCKETS

Because the recommended cavity/pocket depth is 4 x cavity width (and feasible depth is 10 x tool diameter or 25 cm), deeper cavities require tools with larger diameters as they affect the fillets of the internal edges.

INTERNAL EDGES

The recommendation for internal verticle edges is larger than 1/3 x cavity depth--- and the larger the fillet the better. Cavity floor edges should have a .1 mm or 1 mm radius or should be sharp.

MINIMUM WALL THICKNESS

Recommendations and feasibility limitations for wall thickness vary between metals and plastics. For metals, the recommendation is 0.8 mm (with feasibility at .05 mm), while for plastics 1.5 mm is recommended (with feasibility at 1.0 mm). (Plastics are prone to warping and thermal softening which is why a larger minimum wall thickness is required.)

These limitations result from the relationship between wall thickness and stiffness of the workpiece; decreased wall thickness increases vibrations which limits achievable tolerances.

HOLES

Holes are limited by diameter and depth. Standard diameter designs are preferred since parts can be produced with standard drill bits.

Different requirements apply depending on designs with features like blind holes and non-standard diameters.

Recommendations for diameter are based upon the size of standard drill bits. The recommended depth is 4 x the normal diameter (with a maximum depth of 10 x nominal diameter).

THREADS

The recommended length for threads is 3 x the nominal diameter, while the recommended size is M6 or larger (with a feasibility size of M2). Threads that are longer than 3 x the nominal diameter are unnecessary, and threads should be designed as cosmetic in your CAD package, and you'll need to include a technical drawing in the order.

In general, it behooves you to choose the largest thread possible since they're easier to machine.

SMALL FEATURES

Micro-machining elements should be avoided unless necessary. 2.5mm (0.1") cavities and holes can be CNC machined with standard cutting tools. The feasibility recommendation is 0.11 mm (.010").

MAXIMUM PART SIZE

While 5-axis CNC machines typically have smaller build volumes, very large CNC machines can produce parts with dimensions of up to 2000 x 800 x 1000 mm.

The typical CNC milling recommendation is 400 x 250 x 150 mm, and the typical CNC turning recommendation is ⌀500 mm x 1000 mm.

TOLERANCES

Unilateral, bilateral, interference or geometric tolerances should absolutely be defined on all critical features, with a strong warning against over-tolerance. Generally, the standard recommendation of +/- .0125 mm is held if no tolerance is specified in a technical drawing, and feasibility is +/- 0.025 mm.

DESIGNING FOR UNDERCUTS

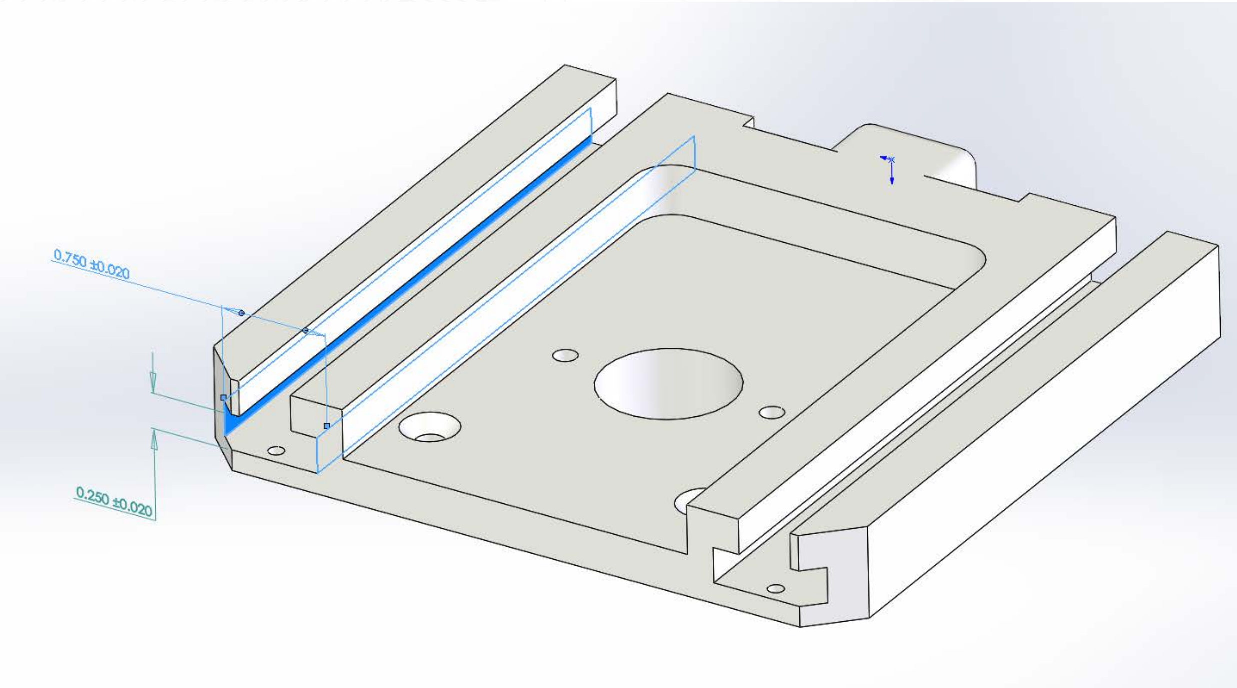
If designed correctly, undercuts can be machined using special t-shaped, lollipop-shaped or v-shaped cutting tools, although certain restrictions apply.

Dimension plays a big role in designing for undercuts, with the recommendation being that you design them with widths in whole millimeter increments (or standard inch fractions).

For non-standard dimension undercuts a custom cutting tool is required since standard tools have a cutting depth of approximately twice the length of their width, which limits feasible depths.

The recommended dimensions for undercuts are 3 mm to 40 mm width and a maximum depth of 2 times the width.

Clearance is another design consideration when it comes to undercuts. The recommended minimum clearance is 4 times the depth, and for internal faces with undercut features, you should add enough clearance between opposing walls to allow tool access.



The Basics Of CNC Machining Materials

Providing they are hard enough, almost any material can be CNC machined. But selecting the material is crucial in the design process. Obviously, the specific use for the part as well as design requirements must be considered (surface finishes can alter CNC machined parts, for example) but CNC machining offers a wide variety of material options from metals to plastics.

METALS

Metals and alloys are the most common materials used in CNC machining, with Aluminum 6061 being the most common. Each material type has certain benefits. Aluminum alloys, for example, have superior strength-to-weight ratios and offer natural protection against corrosion, while brass is often selected for its aesthetics.

Stainless steel alloys offer excellent protection against corrosion as well and can be welded, machined and polished. General use alloy steel offers numerous benefits like toughness, fatigue and wear resistance (more so than mild steels), and hardness, but is not good for chemical resistance.

Mild steel is attractive for its low cost, weldability, and mechanical properties, while tool steel affords high hardness and stiffness, and its resistance to abrasion and temperature makes it ideal for dies, molds, and stamps.

PLASTICS

Because plastics are lightweight and offer flexibility in terms of physical properties--- they're valuable for their chemical resistance and electrical insulation qualities for example -- they're often used for prototyping purposes prior to injection molding.

ABS

This common, lightweight thermoplastic material has excellent mechanical properties and impact strength which makes them a very popular CNC machining material choice.

POLYCARBONATE (PC)

This highly adaptable, temperature resistant material can be colored or transparent, lends itself to outdoor environments and is known for its superior impact strength and toughness.

NYLON

Nylon offers remarkable chemical resistance, and its versatile mechanical properties make it a standard CNC machining material.

POM (DELRIN)

With its thermal stability, extremely stiff characteristics, and superior frictional aspects, this thermoplastic material is the easiest to machine.

PEEK

High-performance PEEK (polyetheretherketone) is a thermoplastic that is best used for the most difficult engineering challenges.

SURFACE FINISHES

Once the machining aspect has been completed, various surface finishes may be applied. They change surface features like color, roughness, hardness or chemical resistance.

Powder coating adds a thin layer of polymer paint and is great for functional applications since it offers higher impact resistance than anodizing and is corrosion-resistant. Powder coating cannot be applied to internal feature surfaces, however, and isn't a good solution for smaller parts.

As-machined parts lend themselves to the tightest tolerances but come with visible tool marks.

Bead blasting is another option that's typically only applied for aesthetic reasons since it adds a smooth matte or satin surface finish to a machined part. It's a low-cost solution, although tool marks can still be visible.

Anodizing a part mans you'll get durability and the ability to use a variety of colors that are aesthetically pleasing and which can be applied even to internal cavities. The application of this method is limited to aluminum and titanium parts and it's more brittle than powder coating.

Another surface finish is hardcoat anodizing, which is known for corrosion and wear resistance and is suitable for a wide variety of functional applications. Additionally, critical areas can be protected during this process to preserve tight tolerances. However, it's more brittle than powder coating and like standard anodizing is limited to parts consisting of aluminum and titanium.

Finally, silk screening is a low-cost method for printing things like logos or text on a flat surface of a machined part and while it can be used along with other finishes, it is used for aesthetic purposes only.

Maloya commonly uses their vibratory finishing systems as a secondary process to finish machined parts. This can soften sharp edges and minimize tooling marks. Various size ceramic media can be utilized for this depending on the part geometry.

How To Keep Your CNC Machining Project On Budget

There are a number of factors that affect CNC machining costs, and here we'll go over some tips to keep your project on budget.

Perhaps the most simplified recommendation for keeping costs low is to choose a design with simple geometries and standardized features. The more complex a part's geometry, the longer the process will take to machine it and that means higher costs.

In general, CNC machining costs correlate to:

- Machining time and model complexity
- Start-up costs
- Material and finishing costs.

For example, start-up costs are directly related to CAD file preparation and process planning, while the type of material and its cost also affect costs. (Bear in mind, high start-up costs can be amortized over time so you can take advantage of economies of scale.)

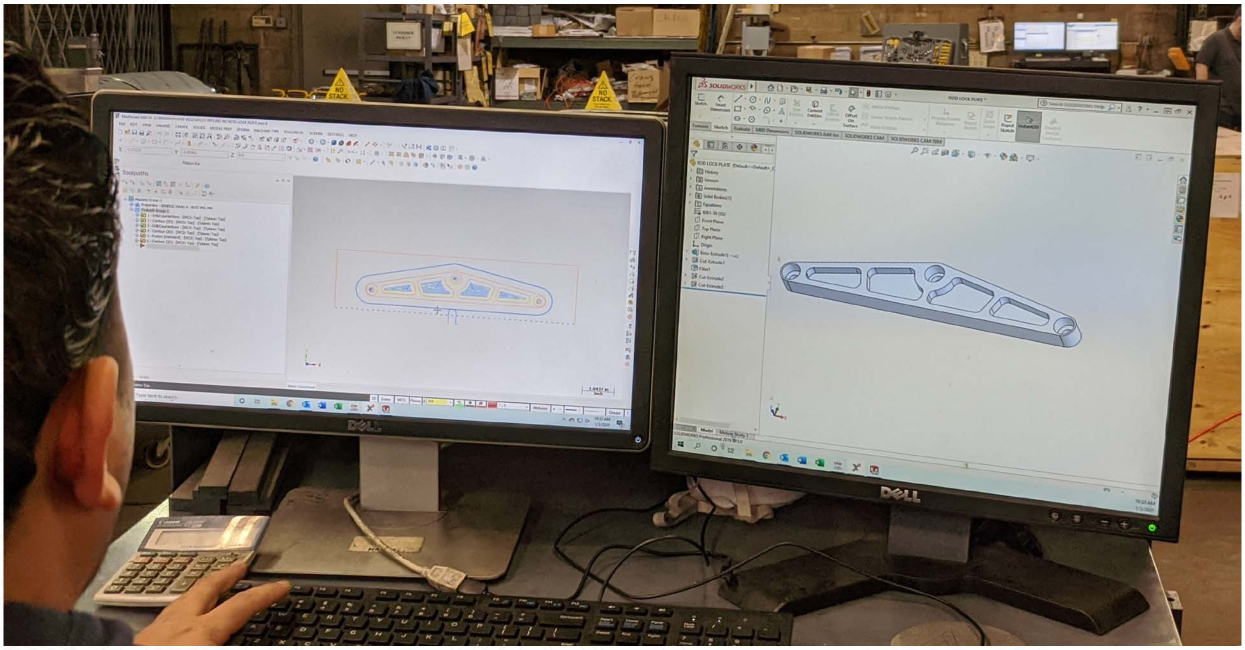
Doing things like adding fillets that are as large as possible to all internal and external vertical edges reduces the time it takes to machine a part. Among several other benefits, a larger tool can be used, and more material can be removed with each cut.

And using the same radius for all the edges of a part makes it easier and faster to change tools, which also saves on costs.

You can also reduce costs by reducing the number of machine set-ups. Designing parts with only one or two orientations when using a 3-axis CNC mill saves a lot of money. If that's not possible, consider a design with multiple geometries that can be split up and machined in one setup, then assembled later on.

Material costs also affect overall costs. Aluminum and POM (Delrin) plastics represent the lowest cost of all the materials, while stainless steel 304 and PEEK represent the highest. Other metal and plastic material costs fall somewhere in between. The goal is to select the material with the lowest cost that meets all design requirements.

From Design To Manufacturing In 4 Basic Steps



STEP 1: CNC-Compatible CAD File Formate

The most predominant formats for exporting CAD files are STEP and IGES. They're open source and standardized and better yet, lend themselves to use across platforms. When using the STEP format, plan to export designs directly from your native CAD software.

STEP 2: Technical Drawing

It's always recommended--- though not required -- that you include a technical drawing in your order since it will contain information not included in a STEP file format.

A technical drawing is always required when:

- A design requires that certain surfaces are finished differently,
- A design contains threads
- Certain tolerances are specified

STEP 3: Get A Quote

You'll need to understand how the complexity of machining services will affect pricing, as well as what the production capacity is so you can get quotes that also give you lead-time estimates. There are 3D imaging software packages available that can provide you with instant quotes.

STEP 4: Start Machining

Now that you've understood the basics of CNC machining, gotten design tips and recommendations for materials and gotten a quote that's in line with your design complexity and materials requirements, congratulations--- you're ready to start machining!