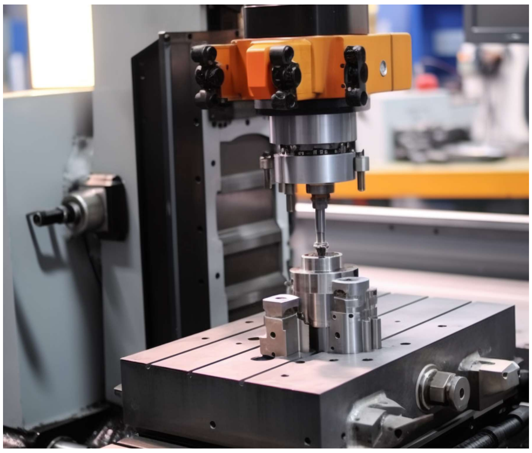
LVMA

DESIGN GUIDE

CNC Machining

Contents

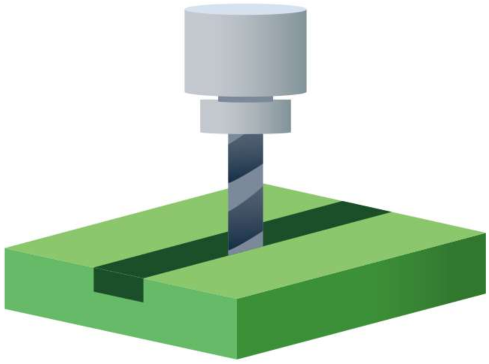


Part One
What is CNC Machining?

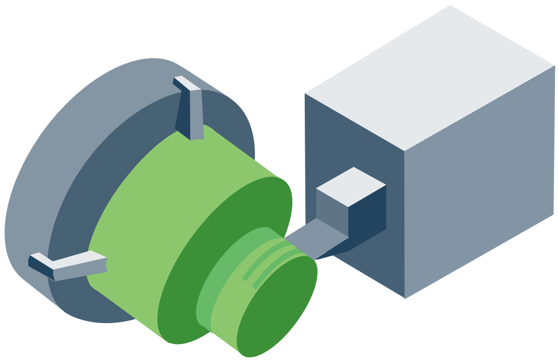
CNC Computer Numerical Controlled Machining is a means to remove material using high speed, precision machines that use a wide variety of cutting tools to create the final design. CAM computer aided manufacturing software, in conjunction with the CAD computer aided design model provided by the customer, is used to program the instructions the machines will use to produce parts.

CNC Milling Machines

With CNC mills, parts are manufactured by holding down the stock material or workpiece to the machine bed while a fast turning spindle holding the cutting tool removes material. Horizontal and vertical movements of the spindle and bed are used to manipulate the workpiece s position, allowing various shapes and depths to be cut. In machines with an additional axis of control, such as the rotary axis in 5 axis machines, the tooling can access multiple faces and hard-to-reach areas to create complex features with reduced setups.



CNC Lathes



Complex cylindrical shapes can be manufactured more cost effectively using a CNC lathe versus a 3 or 5-axis CNC milling machine. With a CNC lathe, the part stock turns while the cutting tools remain stationary. To create the geometry of a part, the CNC computer controls the rotational speed of the stock, as well as the movement and feed rates of the stationary tools. If square features are required on an otherwise round part, typically, the round geometry is created first using a lathe, then the part is moved to a milling machine to create the square features. Lathes with live or driven tools take exception to this and can perform certain milling operations such as drilling, slotting, and tapping within the lathe itself.

Part Two
General Manufacturing Standards

Unless otherwise specified, LVMA, manufactures CNC machined components to the following standards:

- As-machined surface finish is Ra 3.2 or better. Machine tool marks may leave a swirl-like pattern.
- Sharp edges will be broken and deburred by default. Critical edges that must be left sharp should be noted and specified on a print.
- Clear or transparent plastics will be matte or have translucent swirl marks on any machined face. Bead blasting will leave a frosted finish on clear plastics.
- Tolerances on foam or similar compressible materials cannot be guaranteed.

General Tolerances

In CNC machining, standard tolerances are used to ensure that the finished part meets the desired specifications and functional requirements. Unnecessary tight tolerances can increase the cost and time of machining. By specifying standard CNC machining services tolerances, manufacturers can reduce the need for secondary operations and improve the overall efficiency of the machining process.

Tolerance Standard	General Linear Tolerances	General Tolerances for External Radius and Chamfer Heights	Angular Dimensions
ISO 2768–Medium (Standard)	±0.1-2 mm depending on the nominal length from 0.5 to over 2000 mm	± 0.2-1 mm depending on the nominal length from 0.5 to over 6 mm	±1°-0°5′ depending on the nominal length from up to 10 to over 400 mm
ISO 2768–Fine	±0.05-0.5 mm depending on the nominal length from 0.5 to over 2000 mm	±0.2-1 mm depending on the nominal length from 0.5 to over 6 mm	±1°-0°5′ depending on the nominal length from up to 10 to over 400 mm
ISO 286–Grade 8	Standardized tolerance value ranges from 0.014 to 0.23 mm depending on the nominal size from 0.5 to 2000 mm		
ISO 286–Grade 7	Standardized tolerance value ranges from 0.010 to 0.150 mm depending on the nominal size from 0.5 to 2000 mm		
ISO 286–Grade 6	Standardized tolerance value ranges from 0.006 to 0.092 mm depending on the nominal size from 0.5 to 2000 mm		

Note: These tolerances apply to machined metal components. The tolerance grades for plastic and composite materials are ISO 2768 coarse or medium for general tolerances, and ISO 286 grade 8 or bigger for specific tolerances.

If tighter tolerances (less than the standard, e.g. ±0.1-2 mm) are required, information regarding which dimensions require tighter tolerances must be communicated. A technical drawing or specification sheet is the best way to share this information.

Tight Tolerances

General tolerances for CNC machining are typically starting at 0.1 mm. Tight tolerances typically describe tolerances smaller than the general standard. With CNC machining, we can achieve tolerances as tight as 0.01 mm.

With specialized setups and additional operations such as reaming, grinding, etc., even tighter tolerances are possible for some features depending on the material and part geometry. Overall geometric tolerances (GD&T) can also be applied to the drawing for the part; however, these may lead to longer inspection times due to the tools and time required to check them.

While tighter tolerances can improve the form, fit, and function of a part, there are some disadvantages.

Tighter tolerances can lead to:

- Higher scrap rates
- Additional fixture requirements
- Special measuring tools
- Longer cycle times
- Increased price and lead times

Depending on the tolerance requirements and its geometry, a part can cost more than twice as much as a standard tolerance.

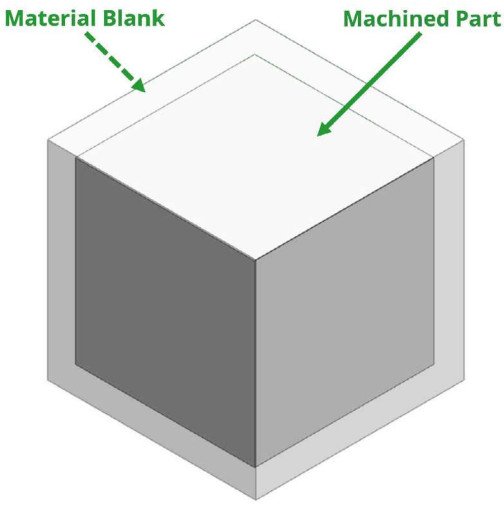
Size Limitations

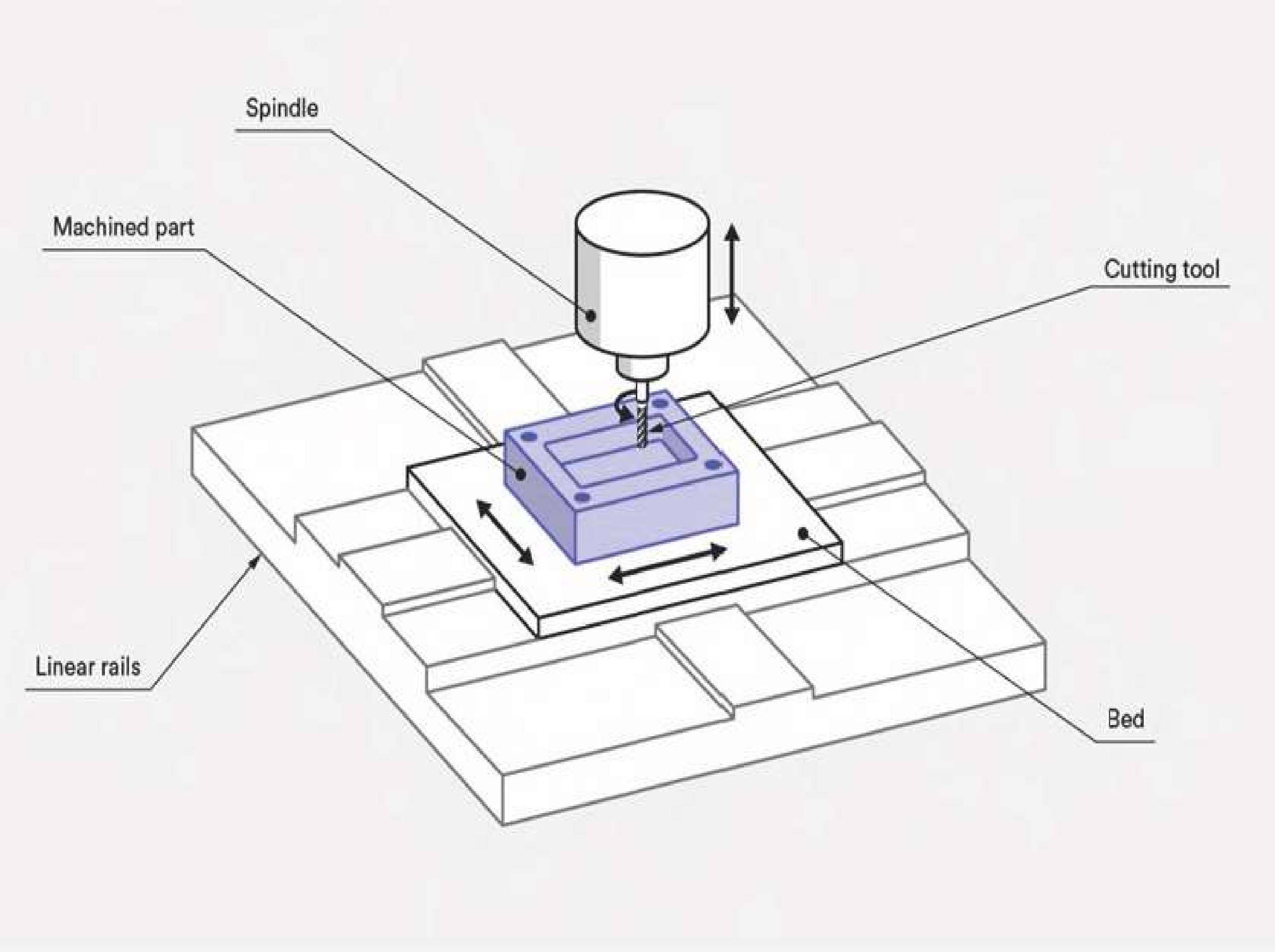
Milled Parts
Part size is limited to the machine s capabilities and depth of cut required by a parts features. LVMA can typically mill parts up to X-2000 mm, Y-750 mm, Z-600 mm. The features and size of each unique part will determine that part s machinable height. If your part goes beyond 2 in machinable height, it will require an additional manual review for manufacturability.

Lathe Parts
LVMA capabilities allow for turned parts up to D-500 mm, L-1500 mm. In addition to standard 2 axis lathes, LVMA's manufacturing facilities utilize specialized equipment such as live tooling systems, multi spindle machines, and swiss lathes, which are great for producing lathe parts with milled features or small, delicate features.

Material Blank Size
Material blank or simply blank refers to the size of the raw material used to create the finished part. Blanks typically need to be slightly larger than the finished part s measurements to allow for variations in the raw material and to cut away the rough faces of the raw material. For example, if the final dimensions are to be 20 × 20 × 20 mm, then a suitable blank for the part would be roughly 23 × 23 × 23 mm.

Designers should keep blank sizes in mind when designing their parts. Optimising your design to allow for smaller and standardized blank sizes is a good way of reducing part cost and waste. Remember that some blank sizes are more common in particular materials than others.





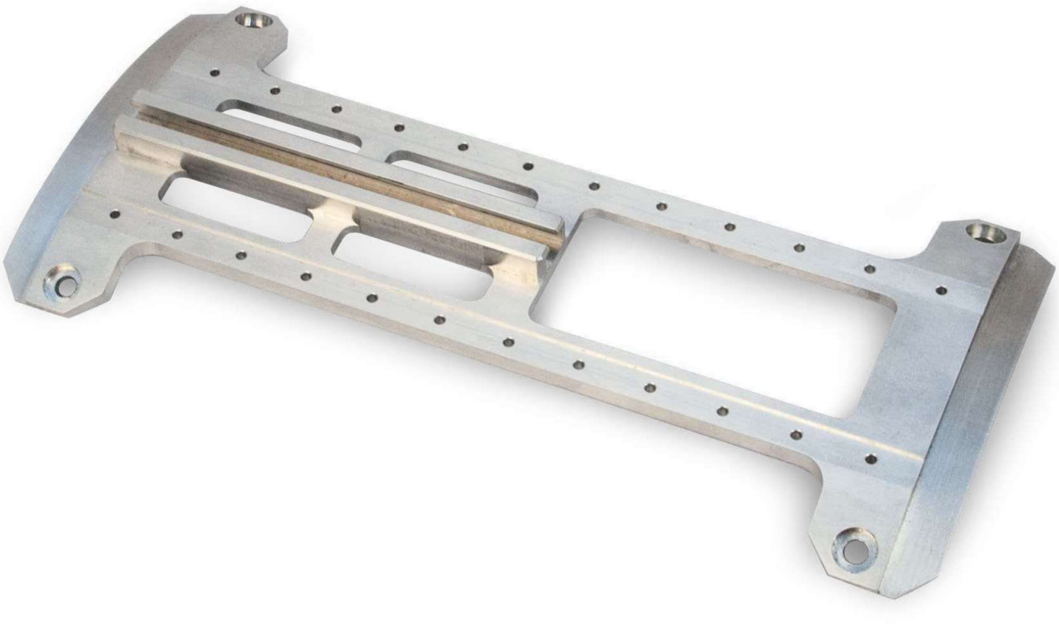
Part Three
Design Guidelines

Adhering to certain design guidelines can reduce the costs associated with CNC machining while maintaining high standards of quality and precision. In this section, we will explore various CNC design guidelines that can be incorporated into your design process to reduce the CNC machining cost.

Part Complexity

CNC machining can effectively produce highly complex designs; however, that does not mean you should not strive to simplify your designs. A part with contoured geometry or multiple faces that need to be cut will typically take longer to machine and thus have a higher cost when compared to a piece that only requires one setup and three axes (X, Y, and the tool movement of the Z) . Minimal cuts are made with small tools to create a complex curved surface with a suitable surface finish. These tiny cuts take significantly longer to machine than the more significant cuts that can be made on broader or planar geometries, increasing the cost.

To help minimize cost and machining time, try to design parts using on axis planes as much as possible. Avoid unnecessary draft angles and contoured or organically shaped geometry. Minimizing feature variations, such as internal corner radii and tapped holes, will also help reduce tool changes, thus further saving time and cost through a simplified design.



Fillets

When using a CNC vertical or horizontal milling machine, interior vertical walls cannot be left sharp and will be machined with a radius. Radii must be present because the material is removed using a round tool spinning at high RPMs. Part designers must consider where radii will occur due to this limitation.

Inside Corner Fillets

When it comes to inside corner fillets, the radii size is key. Sizing corner radii appropriately can improve not only cutting efficiency and cost but quality as well. Consider the following for internal corner radii in your designs:

- Use a radius that does not correlate with standard tool sizes
- Radii should be above 0.5 mm
- Use as large a radius as possible
- Avoid small, deep radii

Most cutting tools come in standard sizes, such as Ø12, Ø6, Ø3, etc. Avoid these standard sizes; if the tool radii match the designed corner radii, it will not have the proper clearance to turn into the cut. Instead, the tool must come to a complete stop, pivot 90 degrees, then resume cutting. These abrupt cutting paths reduce efficiency and lead to quality issues such as chatter.

Floor Fillets

Generally speaking, floor fillets can be time consuming and dif cult to machine and thus should be avoided unless vital to your part s form and function. When creating a floor radius that meets a corner, it is much easier to machine if the floor radius is smaller than the wall radius. By having the floor radius smaller than the wall radius, the same tool can be used to remove the material, which creates a smooth flow through the corner.

Pro Tip:
Use a radius 1.3 times the radius of the closest standard tool size and aim for a radii-to-depth ratio of 1:4 for pocket radii. A smaller radius of 1:8 is also possible to a certain extent, but it would increase the machining time and thus the cost.

Larger radii also enable bigger tools for machining the parts that remove more material with each cut, reducing machine time and overall cost. When the cut s depth exceeds two times the diameter of the cutting tool, the tool s feed rate must slow down, increasing the cycle time and part cost. Though small radius tools down to a 0.25 mm radius are available, sometimes the depth of cut required makes it impossible because the tool is not manufactured in the required length. Even if the tool exists, the part cost will increase significantly due to the extra manufacturing time required to machine a part using only minuscule cuts.

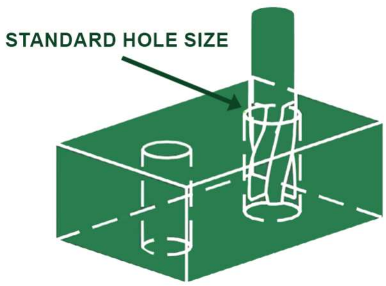
Pro Tip:
For better manufacturability of floor fillets, use a standard bull nose mill radius.

Holes

Holes are typically created using drill bits that plunge into the workpiece to remove material. Drilling is a fast and efficient method of creating holes and is what most machinists will defer to when they can. More significant or oddly sized holes can be made via helical milling with an end mill, but this is slower and less efficient than drilling methods. In either case, designers should make a few considerations when designing holes in their parts.

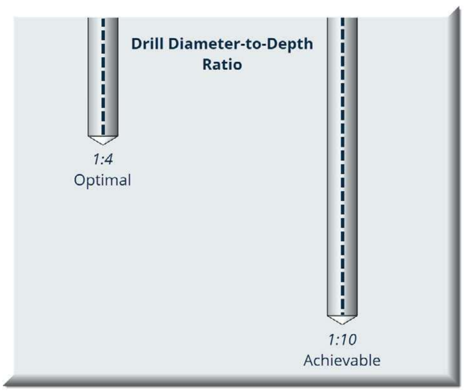
Standard Drill Sizes

Designers should become familiar with standard drill bit sizes and design holes to match, allowing fast drilling and accurate hole sizes. Non-standard sizes may require expensive custom tooling or additional passes with end mills and reamers to achieve the dimension, which increases cycle time. The standard metric drill size is an increment of 0.1 mm from 1 mm to 13 mm. Anything bigger than 13 mm is usually an increment of 0.5 mm.



Hole Depth to Diameter Ratios

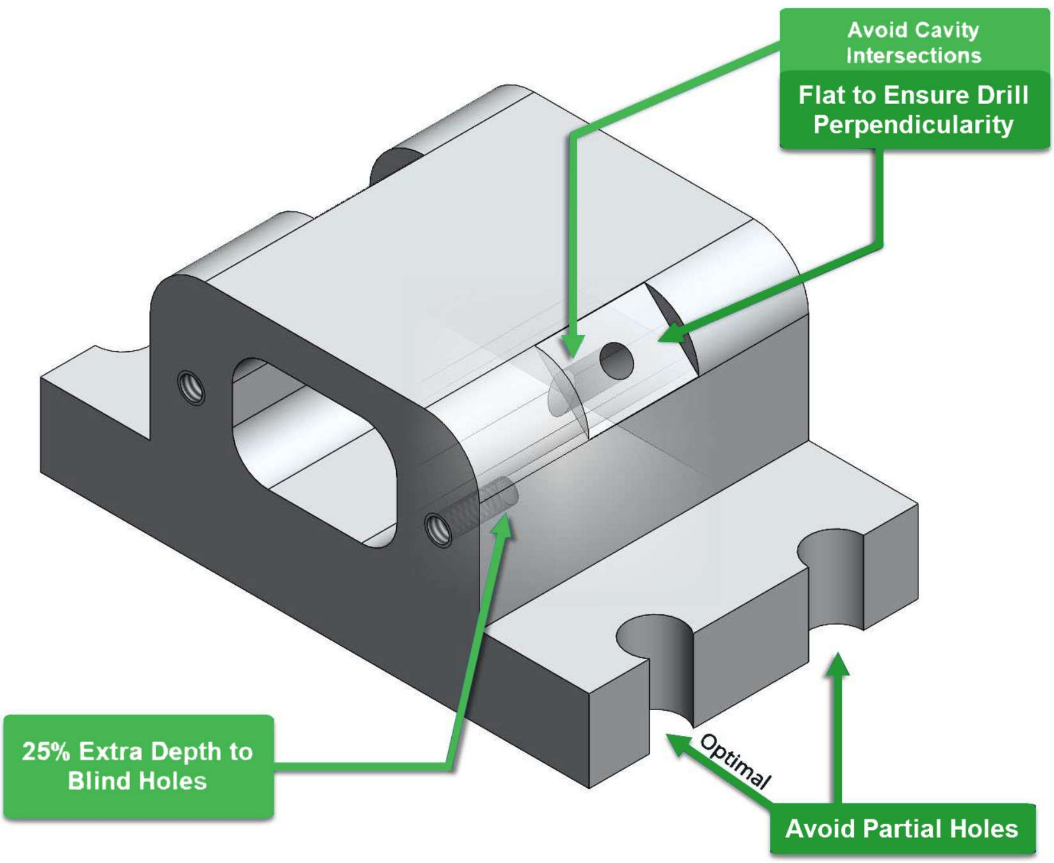
As the depth of a hole increases, so does the manufacturing difficulty. Excessively deep and narrow holes can lead to manufacturing issues such as tool breakage, drill walking, and chip evacuation issues, among others. Hole depth to diameter should be kept as low as possible. Holes with significant depth to diameter ratios may require specialized tooling, such as gun drilling, to achieve the geometry.



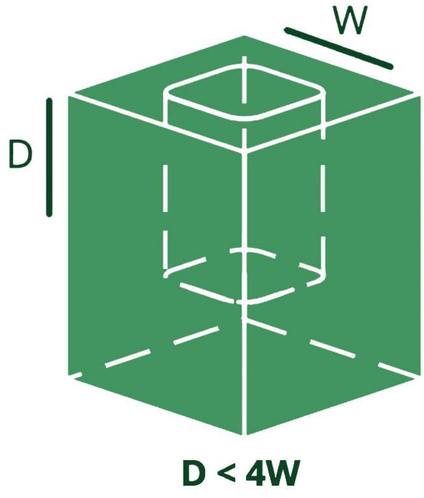
Other Hole Design Tips

Here are some other quick tips and considerations you can follow to improve hole manufacturability of your parts:

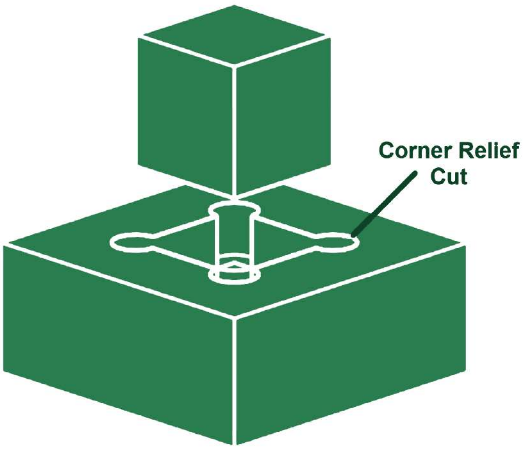
- Avoid partial e.g., on edge holes; these are dif cult to manufacture. If necessary, keep at least 75 of the hole inside the part edge.
- Keep holes and pockets at least 0.8 mm from walls to avoid defects in metal parts. This value doubles for milled plastic or composite materials.
- Keep the hole axis perpendicular to the surface; avoid drilling on sloped or curved surfaces. Adding a flat to curved surfaces where the hole is will ensure the drill enters perpendicularly.
- Use through holes over blind holes when possible; they are more accessible to machine, ream, and tap.
- If you need to use blind holes, add 25 additional depth than you require to account for drill points and chip evacuation.
- Avoid designing holes that intersect with cavities, which can lead to manufacturing issues. If an intersection is unavoidable, keep the center of the drill axis away from the cavity.



Pockets and Cavities



Pockets and excessively deep cavities can pose manufacturing issues such as tool deflection, chip evacuation problems, and tool breakage. Cavities greater than six times deep than they are wide are considered too deep; the ideal width to depth ratio is $D < 4 \times W$.



If your design requires deeper cavities, it is advisable to consider implementing a variable cavity width one that is wider at the top and gradually narrows toward the bottom. This tapered design facilitates improved tool access and maneuverability at the lower sections of the cavity, reducing the risk of tool deflection, enhancing machining accuracy, and simplifying chip evacuation during manufacturing.

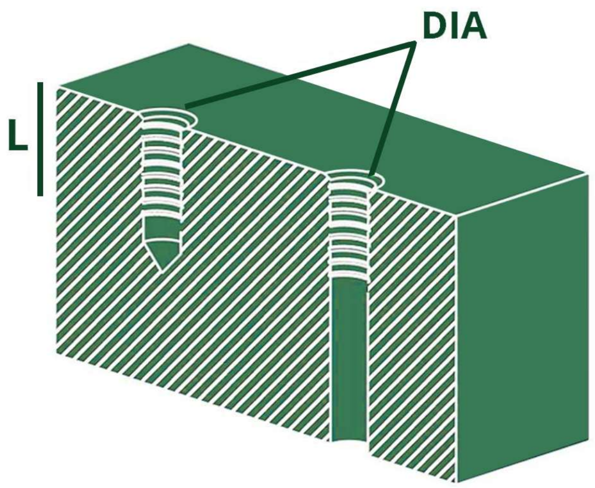
Pro Tip:
When a straight rectangular part will be assembled into a cavity,and a sharp corner is desired,adding corner reliefs or dog bone cuts is better than usinga small radius.

	Recommended	Feasible
Cavity Depth	4 times cavity width	10 times tool diameter or 25 cm

Threads and Tapped Holes

There are several ways to create threads in a part: cut taps, form taps, or thread mills. All of these methods are effective, but designers should keep the following in mind:

- Only thread to the length necessary; going beyond twice the hole diameter is not usually needed for metals. Deep, threaded holes can increase the part cost as specialized tooling may be required to meet the depth requirements.
- Consider using threaded inserts for softer materials such as aluminum or plastics.
- Always choose the largest thread size allowed by design making the manufacturing process more manageable.
- The smaller the tap, the greater the chance it will break during production. Threads below M2 in size become risky to form due to a high potential for tool breakage.
- Avoid using uncommon or custom thread specifications; these may require costly taps or custom tools.
- For blind holes, add an unthreaded length of at least half the diameter of the hole after the thread to allow for tap lead and chip evacuation. It is not necessary to design a drill relief into the 3D model but should be called out as allowable on a technical drawing.
- Add threads to your quote and attach a specified drawing to communicate your requirements. Drawing specifications should fully define the tapped feature, including thread type, hole size and depth, and any blending treatment, such as countersinks.



L < 2 x Diameter

Wall Thickness and Machined Text

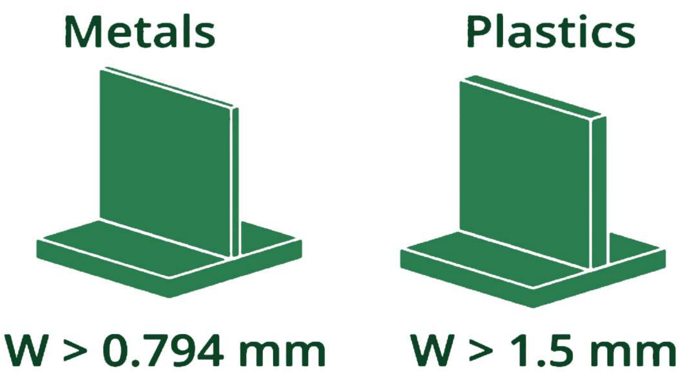
Wall Thickness

Walls should be kept thick enough to ensure strength and rigidity. When thicknesses become excessively thin, they are prone to warping, breakthrough, and general failure when under stress. Additionally, as rigidity is lost, vibrations from the machining process can result in chatter, forcing the machinist to slow things down to mitigate this issue. It is also more difficult to maintain accuracy when cutting walls that are not rigid enough due to being too thin.

Minimum wall thickness should correspond with the following:
Metal Materials: 0.79 mm
Plastic & Composite Materials: 1.5 mm

Machined Text

Machined text can be designed in one of two ways: embossed text that rises above the surface or engraved text that sits below the surface. Of these methods, we recommend creating text as engraved instead of embossed. Engraving requires minimal material removal, unlike embossing, which involves a large amount of material removal adjacent to the text to create the embossed effect. If you do not require machined markings, consider laser marking as an alternative method for adding text to your part.

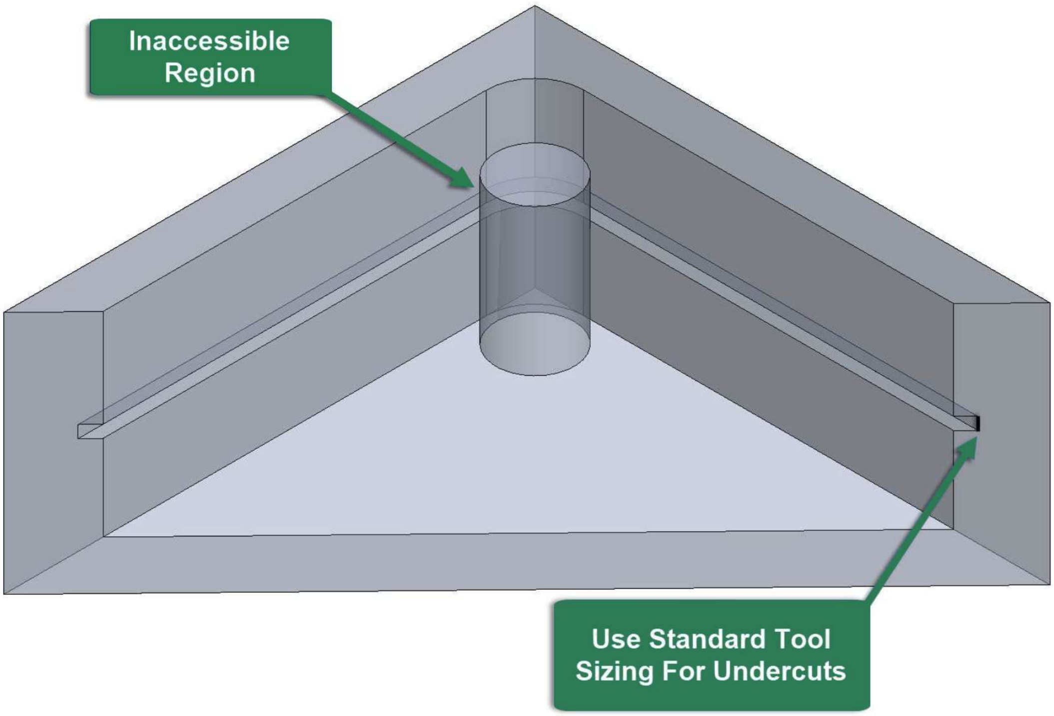


Undercuts

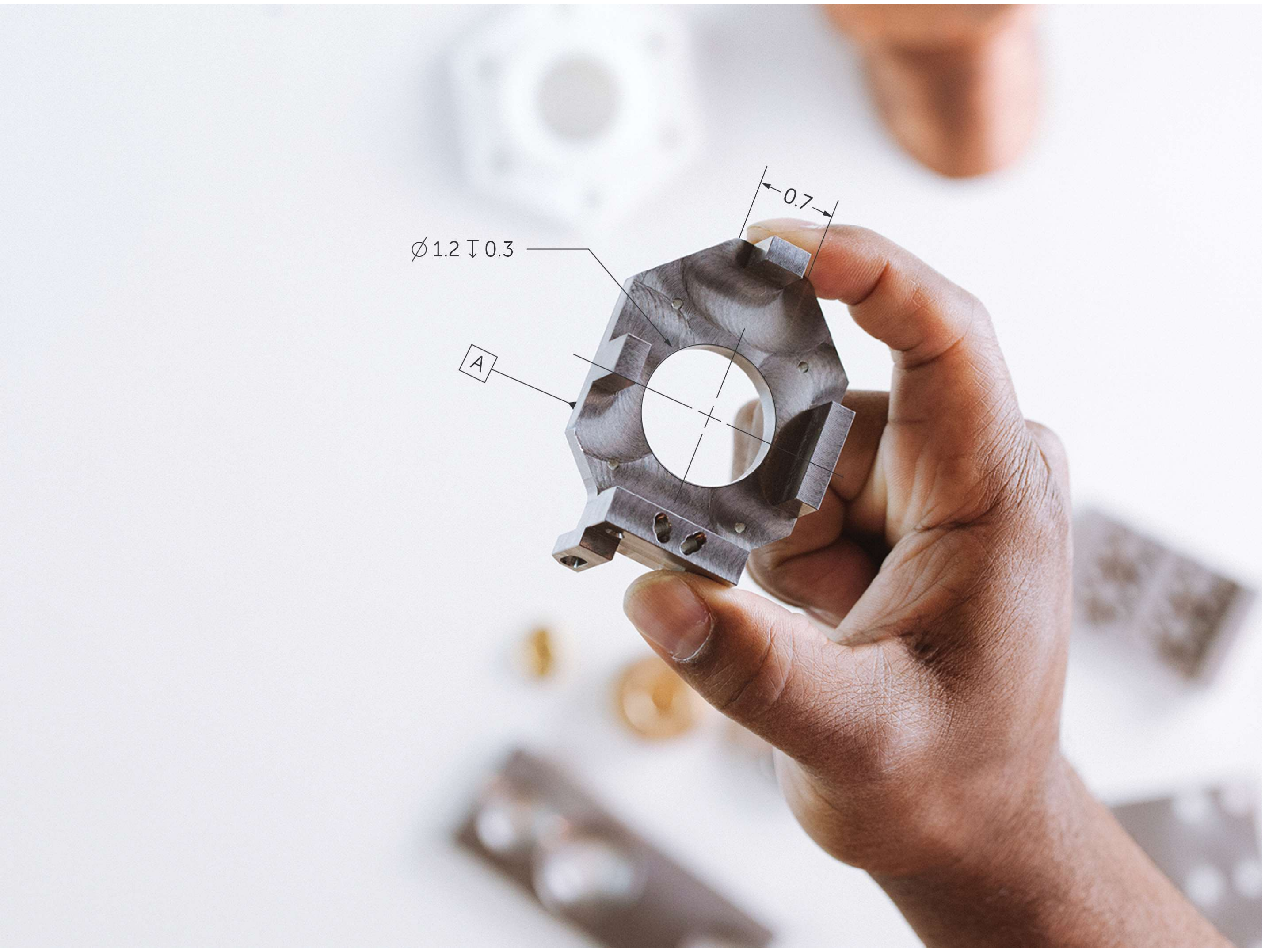
Some features cannot be reached by a standard machining tool, thus creating an undercut region on the part. Care must be taken when designing an undercut for two reasons:

First, suppose the feature dimension does not correlate to a standard cutter size. In that case, the undercut may require creating a costly custom tool, causing part cost and lead time to increase significantly especially if only a few parts are to be manufactured. If a standard radius were to be used, the price is greatly reduced since standardised tooling can be used.

Second, there are limits to the cut depth due to the tool s construction typically a key seat cutter, a horizontal cutting blade attached to a vertical shaft . There is no standard depth for undercuts, but the shallower the better. Designing undercuts in accessible places is also critical. The figure to the right depicts an undercut feature that cannot be manufactured via a machining process.



Undercuts should be carefully evaluated during the design phase to avoid unnecessary complexity and cost. Non-standard feature sizes may require expensive custom tooling, significantly increasing production time and expense—especially for low-volume manufacturing. Additionally, limitations in tool geometry restrict the achievable depth and location of undercuts, making shallow and easily accessible designs more practical.



Part Four
Material Selection Guide

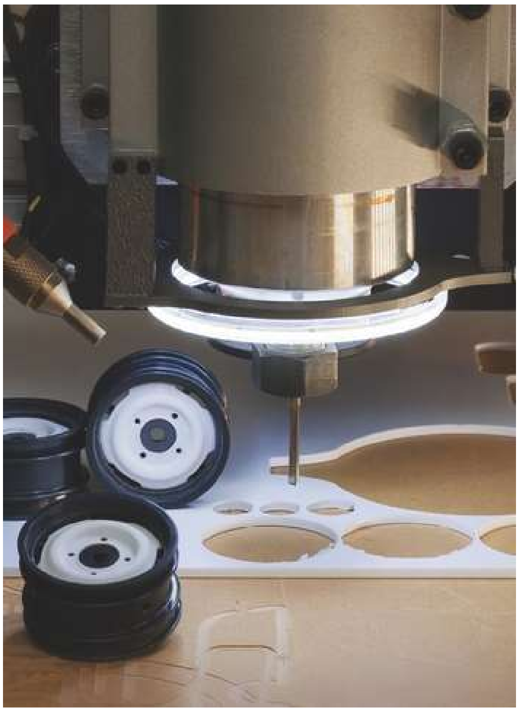
Material selection is an essential aspect of the CNC machining design guide. The material's properties will affect the machinability, cost, and overall quality of the finished part. When selecting CNC machining materials, you must consider machinability, mechanical properties, cost, availability, and environmental impacts.

Metals

Metals are known for their high strength and durability, making them ideal for use in CNC machined parts that will be subject to high stress and heavy loads. They also have good machinability, heat corrosion resistance, and they are highly versatile in producing components for different applications.

Some of the common metals used in CNC machining include:

- Aluminum
- Copper
- Brass
- Stainless steel
- Titanium
- Steel



Plastics

Plastics are widely used in CNC machining due to their low cost,lightweight,and ability to be molded into complex shapes.Some plastics also have good chemical resistance, making them ideal for use in parts that will be exposed to harsh chemicals or corrosive environments.

Some common plastics used in CNC machining are:

- Acetal(POM)
- Polycarbonate (PC)
- Nylon
- Polyphenylene Oxide (PPO)
- Acrylic(PMMA)
- Polyethylene (PE)

Part Five

Post-Processing

Post-processing refers to the finishing steps after CNC machining, enhancing part performance and appearance. Common techniques include deburring, polishing, anodizing, sandblasting, and coating. These processes improve surface quality, corrosion resistance, and dimensional accuracy, making parts ready for end use or assembly.

Inserts

Inserts are a common method for creating strong, reliable threads in parts. They are especially useful in softer materials such as aluminum or plastics, where tapped threads are more prone to wear and tear. If you require inserts, be sure to list the number of inserts required per part on your quote. LVMA and its manufacturing partners regularly install inserts such as:

- Helica/Helicoil Inserts
- Heat-Set Inserts
- Key-Locking Inserts
- Tapping Inserts
- Press-Fit/Self-Clinching Inserts

Part Markings

Part marking is a great way to add high-contrast markings, part numbers, logos, and more. The table below compares the different types of marking methods we offer.

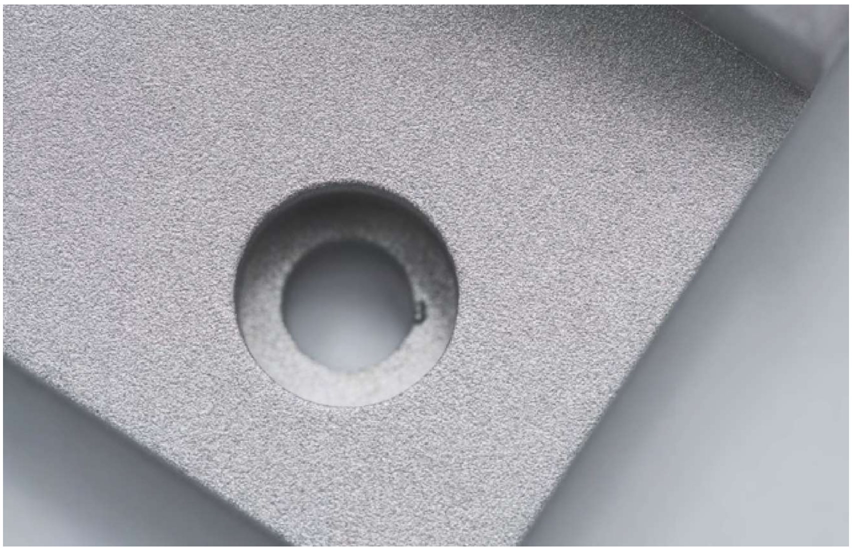
Marking Method	Common Uses	Pros	Cons
Laser Marking and Engraving	•Graphics •Part numbers •Text	•Extremely durable markings •Crisp detail	Cannot produce coloured markings
Bag and Tag	•Serialisation •Part numbers •Bulk packaging	•Very low cost •Can speed up inventory and receiving processes	Non-permanent solution

As Machined

This is the raw surface finish that results from the CNC machining process. The surface of an as-machined part typically has a finish similar to 125 uin Ra, although tighter tolerances can be achieved by requesting a finer finish of 63, 32, or even 16 uin Ra. The surface may have visible tool marks, and the finish may not be uniform.

Bead Blasting

For a sleek, matte texture, bead blasting is a great option. This process involves propelling fine glass beads at the machined part's surface in a controlled manner. The resulting finish is smooth and uniform. Different materials, such as sand, garnet, walnut shells, and metal beads, can be utilized depending on the desired outcome and the purpose of the bead blasting, whether it's for cleaning or as a pre-treatment for further surface finishing.



Anodizing (Type II Or Type III)

Anodization is a versatile and popular surface treatment for CNC machined components, offering superior resistance to corrosion, increased hardness, wear resistance, and improved heat dissipation. It's widely used for painting and priming due to its high- quality finish. At LVMA, we offer two forms of anodization: Type II, known for its corrosion protection, and Type III, which provides an additional layer of wear resistance. Both processes can be tailored to produce a range of color finishes to suit your specific needs.

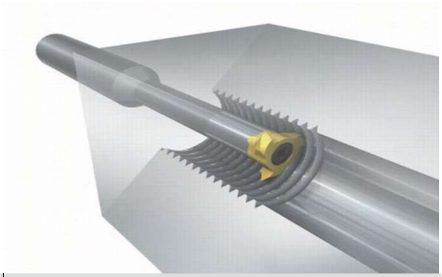
Powder Coating

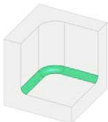
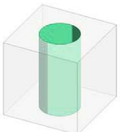
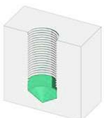
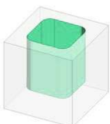
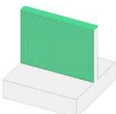
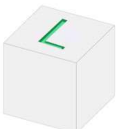
The powder coating process is a highly effective way to protect machined parts from wear, corrosion, and elements. In this method, a special type of powdered paint is applied to the part's surface, and then it is subjected to high heat in an oven. This process creates a long-lasting,protective coating with a multitude of color options to choose from. Whether you need a classic or bold look, powder coating provides a versatile and durable solution for your machined parts.

Custom

These surface treatments are tailored to meet specific design requirements and aesthetic preferences. These finishes can range from simple color changes to complex textured patterns. Custom finishes are essential for improving machined parts' appearance, durability, and performance and can be important in creating a unique brand identity.

Quick Design Reference Chart



	Design Feature	Guideline	Comments
	Interior corner fillet	R depth ÷ R edge ≤ 4 R edge = 1.3 × R tool	The larger the radii, the lower the cost
	Floor Fillet	Less than wall radii	Increase manufacturability sing standard ball nose mill sizes
	Hole Diameter-to-Depth Ratio	Less than 1:10	A ratio of 1:4 is optimal
	Thread Depth	Length < 2 × Diameter	Consider using inserts for plastics
	Additional Hole Depth	25% additional depth after threads and inserts	Allows room for tool leads and chip evacuation
	Cavity Width-to-Depth Ratio	Depth < 4 × Width	Use corner reliefs where tight assembly fits are needed
	Wall Thickness	Metals ≥ 0.794 mm Plastics ≥ 1.5 mm	Avoid designing walls at minimum thickness, thicker is better
	Machined Text	20pt + Sans-Serif	Use machine engraved text instead of embossed text for lower costs
	Edge Chamfer	45°	Use chamfers instead of fillets for edge breaks to lower cost

+86 18157733126/ +86 18958820137

info@lvma-cnc.com

www.lvma-cnc.com

No. 6688 Xuyang Road, Yueqing City,
Wenzhou City, Zhejiang Province, China