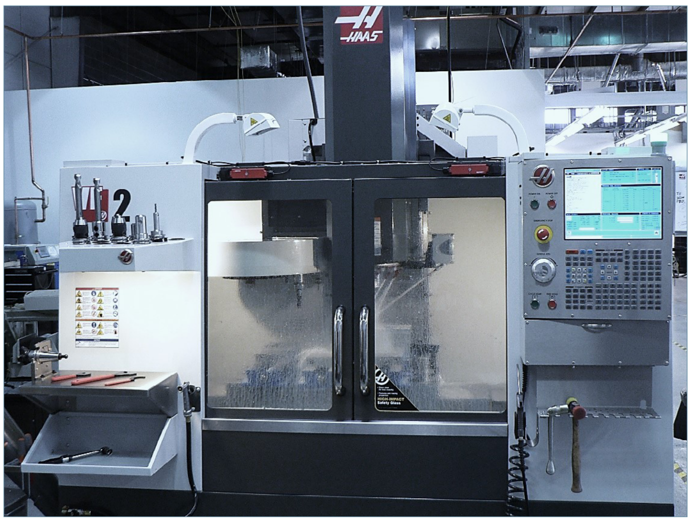
Design Guide: CNC Machining

VERSION 3.4

Table of Contents

Overview

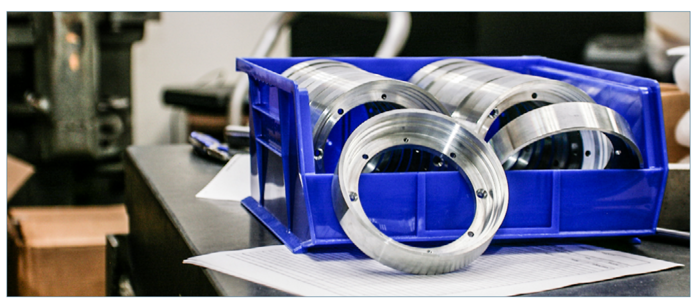
CNC (Computer Numerical Controlled) Machining is a means to remove material using high speed, precision machines that use a wide variety of cutting tools to create the final design. Common CNC machines include vertical milling machines, horizontal milling machines, and lathes.



CNC Vertical Milling Machine

To successfully make a part on a CNC Machine, programs instruct the machine how it should move. The programmed instructions given to the CNC machine are encoded using CAM (computer aided manufacturing) software in conjunction with the CAD (computer aided design) model provided by the customer. The CAD model is loaded into the CAM software and tool paths are created based on the required geometry of the manufactured part. Once the tool paths are determined, the CAM software creates machine code that tells the machine how fast to move, how fast to turn the stock and/or tool, and where to move in a 5-axis X, Y, Z, A and B coordinate system.

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. Conversely, on a CNC mill, the cutting tools move while the stock remains fixed. 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 needed on an otherwise round part, the round geometry is first created on the CNC lathe followed by the square features on a CNC mill.



Because the computer controls the machine movement, the X, Y, and Z axes can all move simultaneously to create everything from simple straight lines to complex geometric shapes. However, despite advancements in tooling and CNC controls, some limitations do still exist in CNC Machining and not all shapes and features can be created. These limitations will be discussed in this guide.

Tolerances

General Tolerances

If a drawing or specification sheet has not been provided by the customer, Xometry will manufacture the product from the model to the specifications listed below:

- For features of size (Length, width, height, diameter) and location (position, concentricity, symmetry) +/- 0.005″
- For features of orientation (parallelism and perpendicularity) and form (cylindrical, flatness, circularity, and straightness) apply tolerances as follows:

0-12″	Tolerance of 0.005″	Angularity	+/- ½ degree
Over 12″-24″	Tolerance of 0.010″	Angularity	+/- ½ degree
Over 24″-36″	Tolerance of 0.015625″ (1/64)	Angularity	+/- 1 degree
Over 36″-60″	Tolerance of 0.03125″ (1/32)	Angularity	+/- 1 degree
Over 60″	Tolerance of 0.0625″ (1/16)	Angularity	+/- 1 degree

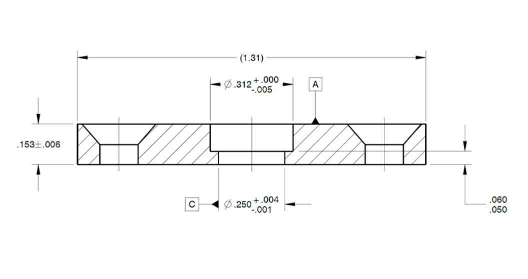
Part Tolerances

Tolerance is the acceptable range for a dimension which is determined by the designer based on the form, fit and function of a part. Unless specifically called out by the designer, the standard tolerance used by Xometry is +/-.005” for metal parts and +/-.010” for plastic parts. If tighter tolerances (less than the standard, e.g. +/-.002”) are required, information regarding which dimensions require the tighter tolerances must be communicated to Xometry. As a point of reference, a piece of paper is about 0.003” thick.

It is important to keep in mind that a tighter tolerance can result in additional cost as a result of increased scrap, additional fixturing, special measurement tools and/or longer cycle times (the machine may need to slow down in order to hold the tighter tolerance). Depending on the tolerance call out and geometry associated with it, the part cost can be more than double what it would be with a standard tolerance.

Overall geometric tolerances can also be applied to the drawing for the part. Based on the geometric tolerance and type of tolerance applied, additional costs may be incurred due to the additional inspection time required.

To help minimize cost, to only apply tight and/or geometric tolerances to critical areas.

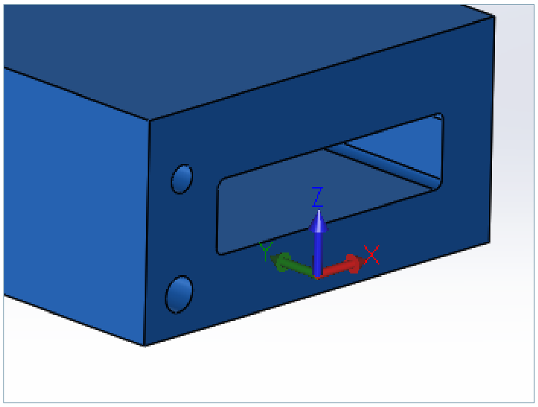


A part with non-standard tolerances

Size Limitations

Milling

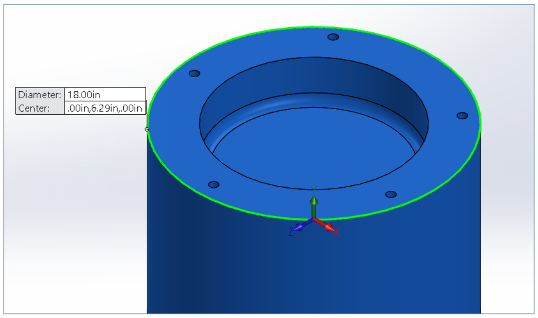
Part size is limited to the machine's capabilities and depth of cut required by a feature in the part. Xometry's equipment has a 64″ (X) by 32″ (Y) by 38″ (Z) travel limitation. The Z travel of 38″ does not, however translate to a part height or depth of 38″: depending on part size and features in the Z dimension, the Z height of the part will need to be less than the 38″ to allow for tool clearance. The features and size of each unique part will determine that part's machinable height.



Milling size limitations

Lathe

Xometry's lathe capabilities allows for successful machining of parts up to 18″ (457.2mm) in diameter, but special cases may be made for larger parts. Xometry is capable of utilizing a live tooling lathe, which dramatically decreases lead times and increases the amount of features that can be machined.



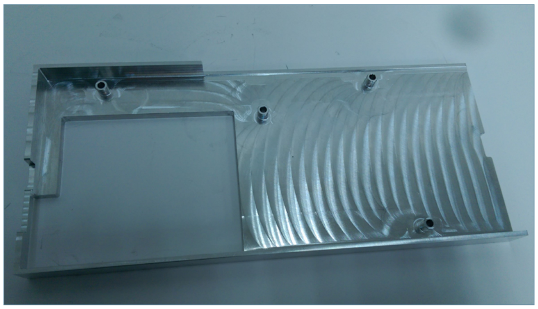
Lathe size limitations

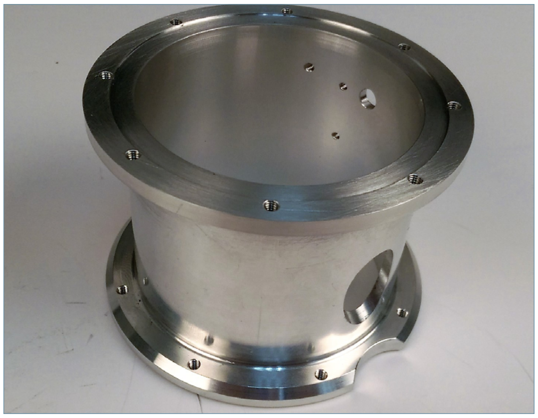
Material Selection

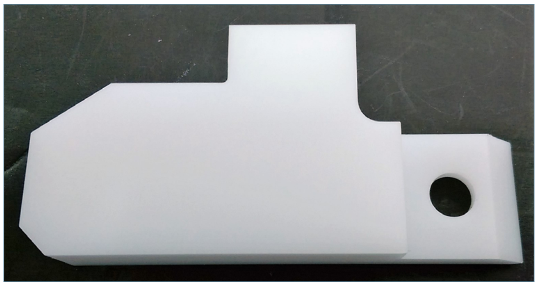
Material selection is critical in determining the overall functionality and cost of a part. The designer must define the material characteristics key to the part's design – hardness, rigidity, chemical resistance, heat treatability, thermal stability, and so on. Xometry machines a wide variety of metal and plastic materials, listed below, as well as other custom materials upon request.

MATERIAL BLANKS

"Material blank" or simply "blank" refers to the size of the raw material that will be used to create the finished part. A good rule to follow is to account for a blank that is a minimum of 0.125" larger on each dimension than the finished part's measurements to allow for variations in the raw material. For example, if the final dimensions are to be 1" x 1" x 1", then the blank for the part would be 1.125" x 1.125" x 1.125". If the part's form, fit, and function would not be negatively affected, the designer might consider reducing the final part dimensions to 0.875" x 0.875" x 0.875". This way a standard 1" x 1" x 1" block could be ordered to save on material cost.







CNC parts made with various materials

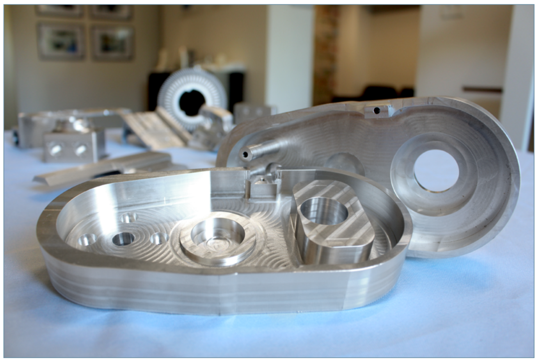
Metals

Xometry offers the following metals for CNC machining:

- Aluminum
- Brass
- Bronze
- Copper
- Stainless Steel
- Carbon Steel
- Titanium
- Other custom metals

Plastics and softer metals (e.g. aluminum and brass) in general machine easily and subsequently require less machine time, reducing the cost of machining. Harder materials like stainless steel and carbon steel must be machined with slower spindle RPMs and machine feed rates which makes for longer cycle times over the softer materials. As a baseline estimate, aluminum will machine about 4 times faster than carbon steel, and stainless steel will machine half as fast as carbon steel.

Material type is also a critical driver in determining the overall cost of a part. For example, 6061 aluminum bar stock is approximately $^1/_2$ the price of aluminum plate per pound, and 7075 aluminum bar stock can be 2 to 3 times the cost of 6061 bar stock. 304 stainless steel costs about 2 to 3 times as much as 6061 aluminum and about 2 times as much as 1018 carbon steel. Depending on the size and geometry of a part, the material cost can be a significant portion of its overall price. If the design does not warrant the properties of a more expensive material like carbon or stainless steel, consider choosing a less expensive material like 6061 aluminum to minimize material cost.



Metal CNC parts

Plastics

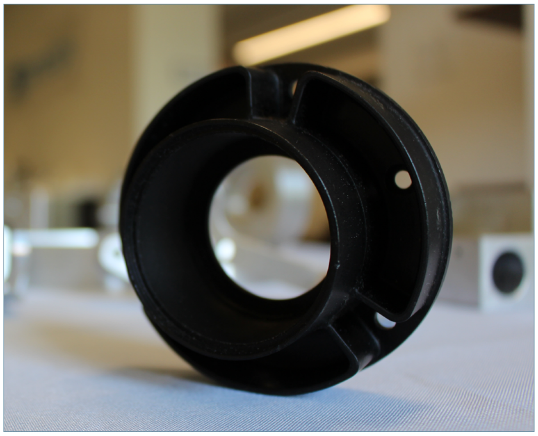
Xometry offers the following plastics for CNC machining:

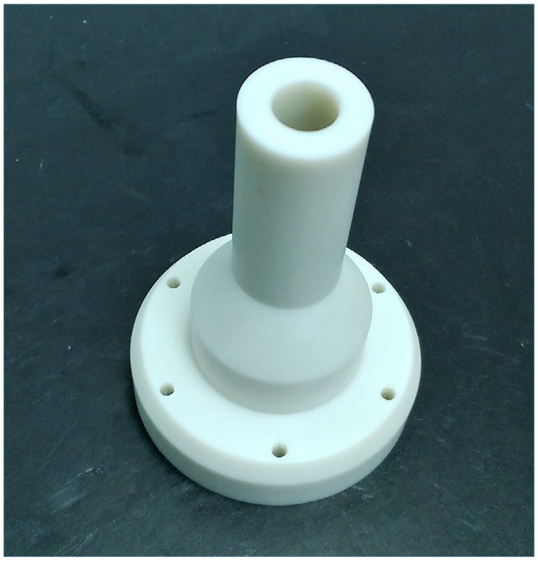
- ABS
- Garolite
- Acetal (Delrin®)
- Nylon
- PEEK
- Polypropylene
- PTFE
- Polycarbonate
- Ultem
- Other custom plastics

Plastic can be a less expensive alternative to metal if a part's design does not require the rigidity of metal. Polyethylene, for example, is easy to machine and is about $^1/_3$ the cost of 6061 aluminum. In general terms, ABS is about $1^1/_2$ times the cost of acetal, while nylon and polycarbonate are approximately 3 times the cost of acetal.

NOTE:

Depending on a part's geometry, tight tolerances can be harder to hold with plastics. Parts may also warp after machining as a result of the stress created when material is removed.





Plastic CNC parts

Complexity & Limitations

CNC Machining can effectively produce simple and more complex designs

The more complex the part—i.e. a part with contoured geometry or multiple faces that need to be cut—the more costly it becomes due to the additional setup and machining time required. When a part only requires one setup and 3 axes (for example X and Y, and the tool movement making Z), the setup and machining can be accomplished faster, thus minimizing the cost.

To create a complex surface with a suitable surface finish, very small cuts are made. These small cuts take significantly longer to machine than larger cuts on broader or planar geometries, which in turn increases the cost. To help minimize cost and machining time, try to design parts using on-axis planes possible. Keeping features such as internal corner radii and tapped holes consistent will also help save time and money on parts by reducing the need for tool changes.

CNC parts ranging in complexity

Interior Fillets

When using a CNC vertical or horizontal milling machine, all interior vertical walls will have a radius. This is because material is removed with a round tool spinning at high RPMs. Part designs must take into account areas where radii will occur as a result of this limitation.

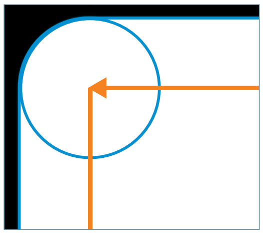
INSIDE CORNER FILLETS

For internal corner radii, it may be better to use a non-standard radius. This is because endmills need clearance to turn and continue milling when tracing the internal corner (see Fig. 1).

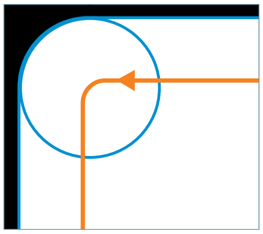
If a part features a 0.25" interior radius, the standard endmill would need to hammer the corner, come to a complete stop, pivot 90 degrees, and then resume cutting. Doing this slows down machining speed (creating additional cost), and also causes vibration (creating chatter marks). By adding 0.02" (0.508mm) - 0.05" (1.27mm) to internal radii, the cutter will be able to turn slightly without coming to a complete stop. This will not only reduce the part's cost, it will also improve the part overall (see Fig. 2).

The larger the radius, the lower the cost—larger tools can be used to machine larger parts, resulting in more material being removed with each cut, which in turn reduces machining time. For example, in the illustration to the left (see Fig. 3), using a tool with a 0.125" diameter (0.063" radius) would take approximately $1\frac{1}{2}$ times longer than using a 0.187" diameter tool and approximately 2 times longer than a 0.250" diameter tool.

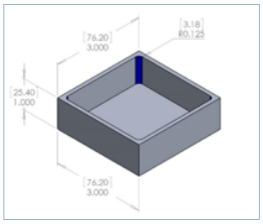
Though small radius tools (down to a .015" radius) are available, sometimes the depth of cut required makes the cut impossible because the tool is not manufactured. If the tool is manufactured, the part cost will increase significantly as a result of the increased manufacturing time required to machine a part using only small cuts.



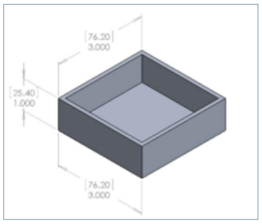
(Fig. 1a) Cutter path comes to a sharp corner



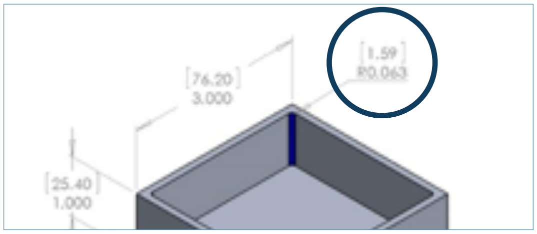
(Fig. 1b) Center line of cutter path has a radius



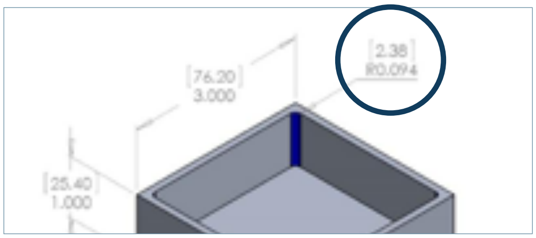
(Fig. 2a) Interior radii ensure a round endmill can cut the internal cavity



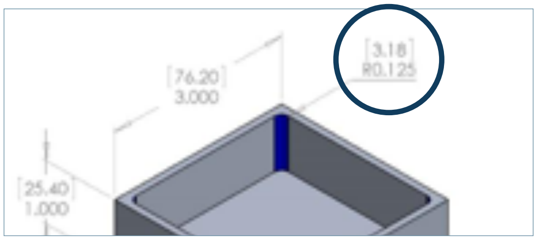
(Fig. 2b) Square internal corners cannot be manufactured



(Fig. 3a)



(Fig. 3b)



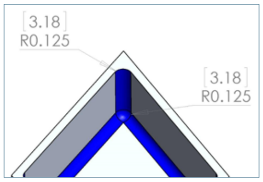
(Fig. 3c)

INSIDE CORNER FILLETS (CONTINUED)

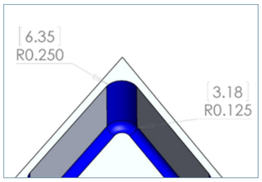
When the depth of cut becomes greater than 2 times the diameter of the cutting tool, the tool's feed rate must slow down, which increases the cycle time and part cost. For every doubling of the length of cut, feed rate is more than halved, which more than doubles the time to cut the feature. he maximum cut depth to tool diameter ratio is 4 times for pockets and 10 times for drilled or reamed holes. Ratios greater than this may require special tooling. For example, using a 0.125″ diameter tool, the max cut depth would be 0.50″ and drill depth is 1.25″ before a custom tool would be required.

FLOOR FILLETS

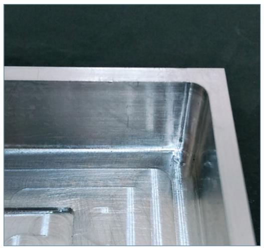
When creating a floor radius that meets to a corner, it is much easier to machine if the floor radius is smaller than the wall radius. With modern CAD systems, it is easy for a designer to have the computer generate the same size floor and wall radii with a few clicks, but this makes it very difficult to remove the material in the corner. 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.



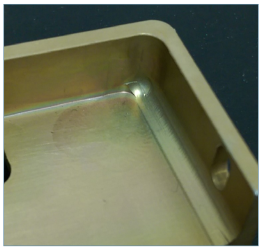
Identical floor and wall radii are difficult to machine



The part is easier to machine if the floor radius is smaller



Machined wall fillet



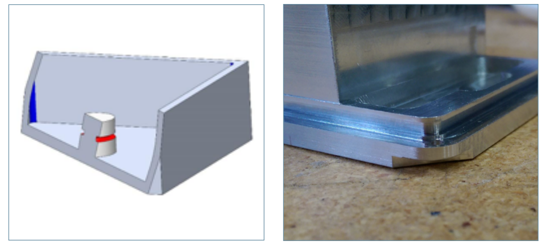
Machined floor fillet

UNDERCUTS

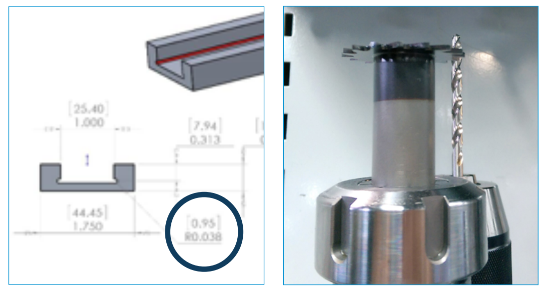
Some features cannot be reached by a standard machining tool, thus creating an undercut region on the part. Care must be taken when designing an undercut for two reasons:

First, if the feature is not a standard dimension, the undercut may require the creation of a custom tool. In the example at left (Fig. 1), the radius in the slot is 0.053”. A costly custom tool would be necessary to create the geometry, causing part cost to increase significantly—especially if only a few parts are to be manufactured. If a standard .062” radius were to be used, then the tool’s cost would be less than half that of a custom tool.

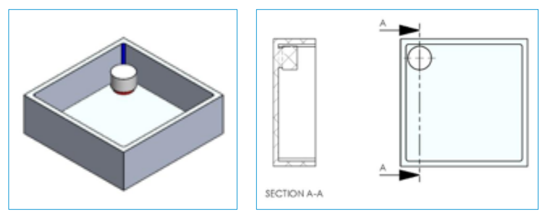
Second, there are limits to the depth of cut due to the construction of the tool (a horizontal cutting blade attached to a vertical shaft). There is no “standard depth” for undercuts, but the shallower the better. Designing undercuts in accessible places is also critical. The illustration at left (Fig. 2), for example, depicts an undercut feature that cannot be reached in the machining process.



Examples of undercuts



Custom tools increase build cost (Fig. 1)



Undercut in close proximity to wall (incorrect) (Fig. 2)

Finishes

Threads

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:

- Always choose the largest thread size possible allowed by the design—it makes the manufacturing process easier.
- The smaller the tap, the greater the chance it will break during production.
- Only thread to the length necessary. Deep, threaded holes can increase part cost as specialized tooling may be needed to meet the depth requirements. Try to use off-the-shelf thread sizes wherever possible to keep costs down.
- Be sure to add threads to your quote *and* attach a specified drawing, or else parts will be machined to the specified diameter.

NOTE:
Please see a list of Xometry's supported threads on our Support page here.

A threaded hole on a machined part

Surface Finish

Xometry offers a wide variety of finishes:

STANDARD, AS MILLED FINISH

This finish is equivalent to a 125 RMS finish where minor tool marks are visible on the part. Increasing surface finish requirements to 63, 32, or 16 RMS can increase costs as feed rates may need to be reduced and/or additional post processing may be required.

BEAD BLAST FINISH

A light texture with a matte finish is created by blowing small glass beads against the part in designated areas. Additional costs may be incurred if the design requires significant masking of surfaces or holes that do not require bead blasting.

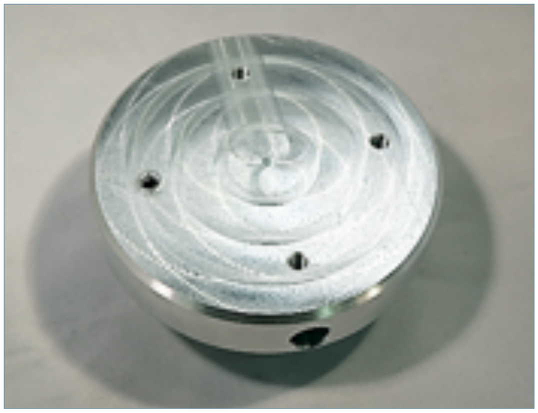
ANODIZING, TYPE II

This type creates a corrosion-resistant finish. Parts can be anodized in different colors—clear, black, red, and gold are most common—and is usually associated with aluminum.

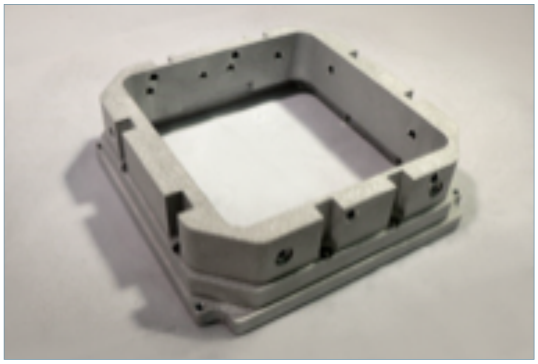
ANODIZING, TYPE III (HARD)

This type is thicker and creates a wear-resistant layer in addition to the corrosion resistance seen with Type II.

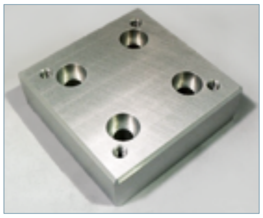
NOTE:
Reference Mil-A-8625A for additional information on anodizing.



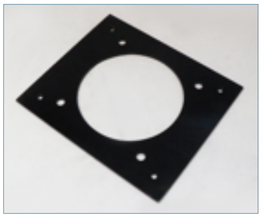
As milled - 63 finish



Bead blast



Type II anodized



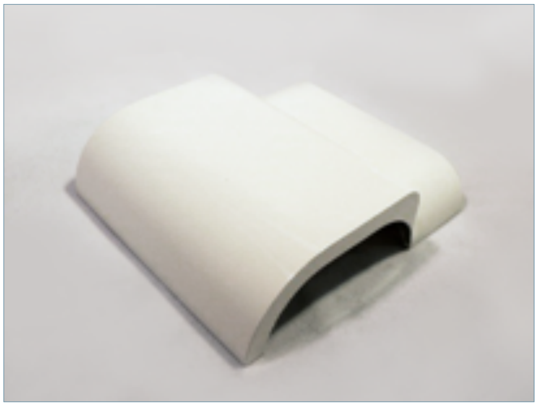
Type III anodized

POWDER COAT FINISH

This is a process where powdered paint is sprayed onto a part which is then baked in an oven. This creates a strong, wear- and corrosion-resistant layer that is more durable than standard painting methods. A wide variety of colors are available to create the desired aesthetic.

OTHER

Other types of finishes, including iridite, are available upon request.



Powder coat

Resources at Xometry

Online Instant Quoting

Web: Upload your CAD file at get.xometry.com/quote

CAD: Download the free Xometry Add-In for SOLIDWORKS: xometry.com/solidworks

Accepted File Types: .stl, .step, .stp, .x_t, .x_b, .sldpart, .ipt, .prt, .sat, .catpart (max file size: 300MB)

Capabilities: CNC Machining, Sheet Metal Fabrication, 3D Printing, Urethane Casting, Injection Molding

Live Engineering Support

Hours: M-F 8:00 AM - 9:00 PM EST

Email: support@xometry.com

Phone: (240) 252-1138

Online: xometry.com/support offers live chat, FAQs, and other helpful articles.

ITAR Registered

All uploads are secure and confidential. Contact us for assistance with large orders and existing Xometry quotes.

Need to use a PO? We offer same day credit application processing. Contact us for details.