

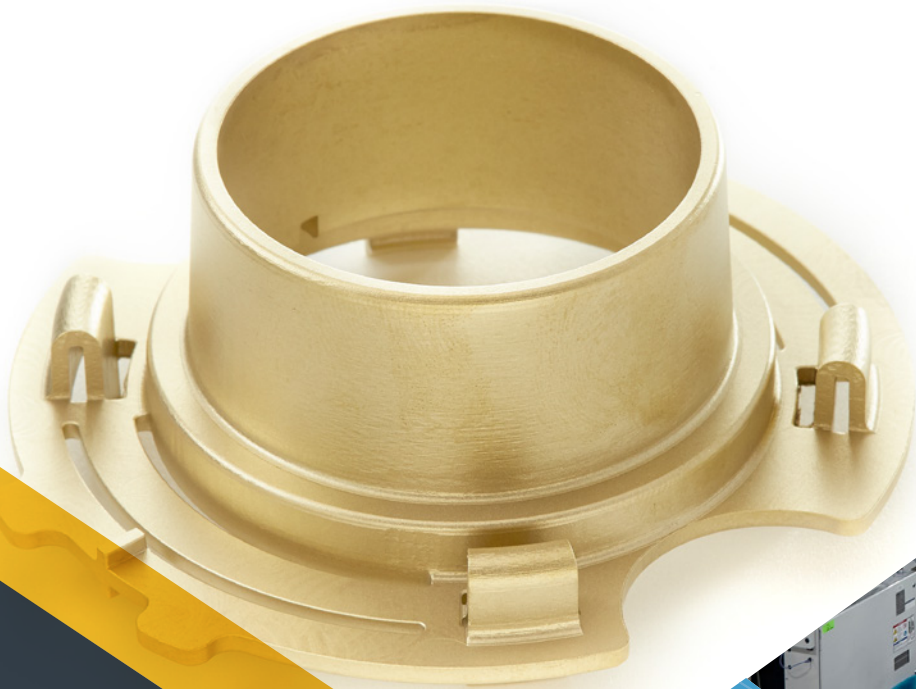


DESIGN ESSENTIALS

for CNC Machining



PROTOLABS
Manufacturing. Accelerated.



Contents

- 3 6 Ways to Cut Machining Costs
- 6 Turning to Lathe for Cylindrical Parts
- 8 The Ins and Outs of Proper Threading
- 12 Moving Between 3D Printing and Machining for Plastic Parts
- 15 Jigs, Fixtures, and Capacity Relief

Machining isn't new on the manufacturing scene, but it is evolving from year-to-year. Understanding its latest capabilities can reap big rewards for product designers and engineers. After helping thousands of customers with custom prototypes and end-use, production parts, we've compiled our most valuable design tips to help you take full advantage of what CNC machining has to offer. Whether you need to choose between milling and turning, select the right material, or just want to reduce costs, we've got you covered.



Got a 3D CAD model ready to be quoted? Get free design for manufacturability (DFM) analysis and real-time pricing.

[GET FREE DFM](#)

6 Ways to Cut Machining Costs

Machining gets a bit more complex every year, and as a result, it can be challenging to keep pace with the do's and don'ts of part design. But lowering the cost of machined parts while improving functionality can still be achieved by a few relatively simple adjustments to your part design or material selection.

At Protolabs, we use automation software to quote parts and highlight features that need design attention. The software will catch features that are unmanufacturable up front, but it will also highlight areas that do not necessarily require change, but can improve the design's overall machinability—thin walls, engraved text, deep holes, and complex geometries, to name a few.

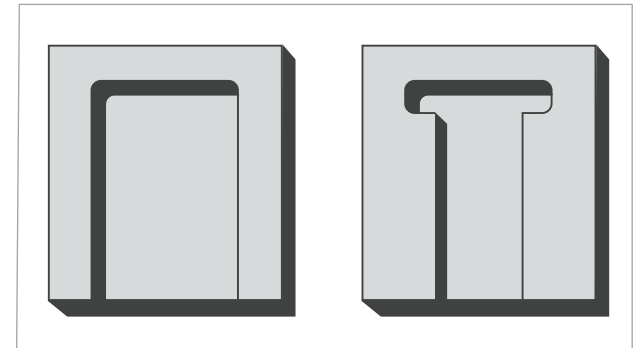
Here are a few tips to help you design more cost-effective machined parts.

1. Provide Relief to Corner Pockets

Consider the corners of a machined pocket—the inside of an electronics housing, perhaps, or a bracket used to capture the body of a rectangular component. One common design oversight is leaving the intersection of the vertical walls on those part features perfectly sharp. To illustrate, think about machining a stainless steel box to hold a collection of baseball trading cards. The only way to get the perfectly square corners needed to fit those Babe Ruths and Hank Aarons is with EDM, a slow and expensive process.

Instead, we'll equip one of our machining centers with the smallest end mill available to clean out the corners. In 304 stainless steel, that means a 0.031 in. (0.8mm) end mill, which leaves a corner radius of 0.016 in. (0.4mm). That's pretty sharp, but the depth is limited—the length of most steel-cutting end mills in this size range maxes out at five times the cutter diameter, barely deep enough to fit your favorite center fielders. Machining with small end mills such as this is also slow and delicate work, driving up the cost of your project.

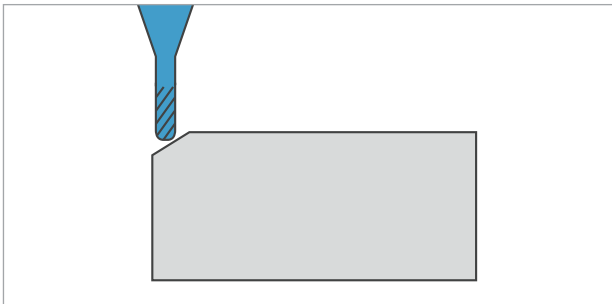
A more budget-friendly approach is machining a relief in each corner of the pocket. This removes that pesky radius, leaving a U- or C-shaped clearance instead (see illustration). It also allows for far deeper pockets—by cutting a 0.25 in. (6.35mm) wide relief in each corner, functionally sharp corners to around 1-1/4 in. (32mm) in depth are possible. And by switching to aluminum or even plastic, pocket depths twice that of steel are possible. Best of all, designing pockets in this manner reduces part cost, since larger end mills can be used and material removal rates increased accordingly.



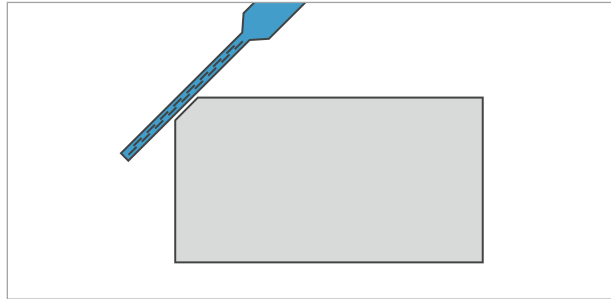
The left image illustrates resulting corner radii from milling. Consider adding reliefs to sharp corners (right image) to improve fit.

2. Deburr Edges Yourself

Avoiding corner breaks is another radius-related, cost-cutting measure. In an attempt to remove burrs and break sharp corners, designers will often smooth external part intersections with chamfers or corner radii. It's understandable, and sometimes quite necessary, but it can also be expensive. With stainless steel parts, we provide an automated deburring option, but all other materials are delivered as-machined. If the part design calls for an edge break, we have to call up an additional tool (a ball end mill) and machine these corners using a 3D profiling motion. We generally run these tools at very high rpm, and very small amounts of material are removed, but it's still a lengthy process to go back and forth until each corner is smooth. Many customers opt to save a few bucks by deburring these parts on their own with a file, some abrasive paper or a buffing wheel.



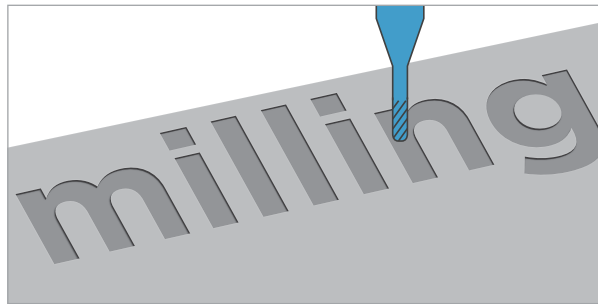
Adding chamfer or radii to corners can create longer machining times due to ball end mills.



When an edge needs softening, use a file or deburring tool to break the edge instead of milling it.

3. Avoid Text Until Molding

Similarly, text engraving is an aesthetically pleasing but time-intensive operation, one that might be best to avoid if possible. Here again, a ball end mill is used to trace whatever letters, numbers and symbols are called for on the CAD model. It looks cool, and might be a valid requirement on your machined part, but is probably more appropriate on injection-molded parts, where additional machining time is amortized over higher part volumes.



Small tool diameters add machining time so consider removing text or logos from machined prototypes.

4. Be Cautious of Thin Walls and Features

Our standard part tolerance is ± 0.005 in. (0.127mm). If you have a feature that is 0.020 in. (0.5mm) or smaller, our automated quoting system will highlight it as a thin wall geometry, but keep in mind that we'll still allow it to be machined—so the machined part may differ slightly from your original design.

Any thin walls that are 0.020 in. or less are not only subject to breakage during the machining operation, but may flex or warp afterwards. Beef them up as much as your part design allows.

5. Keep It Simple

Very deep pockets are a no-no, even if the corners are relieved. Not only does it take a lot of machining time to remove all that material, but any residual stress in the raw material tends to show its ugly face as pockets get deeper and walls taller. Gussets or support structures might be used to support these workpieces and prevent movement due to stress, but these tend to increase machining costs. Keep it simple is the best advice for any manufacturer or product designer of machined parts.

The same principle applies to overall part geometry as well. Don't attempt to make parts do more than they should. Maximizing material usage may create workholding or machining problems, in turn increasing costs. If the design gets too complex, consider breaking it into multiple components and using fasteners to assemble them. No one likes assembly costs, or the complexity that goes with multiple pieces, but it might be the best approach for difficult-to-machine parts. Sculptured surfaces, cavernous slots (think heatsinks), super deep holes and threads—these are some of the common machining cost drivers that can chip away at your project budget.

6. Explore Alternative Materials

One of the simplest ways to stay within budget—assuming it meets your requirements—is by switching to a more machine-friendly or less expensive material. Our material selection includes a range of metals and plastics, each with its own engineering attributes, aesthetics, machining considerations, and material cost.

- 17-4 PH stainless steel is a bear to cut. If high strength and corrosion resistance isn't critical, try 316L or 304 instead.
- Copper is a great electrical conductor, but it's far more expensive than aluminum, which is nearly as conductive and easier to machine besides.
- If hardness is a concern, 4140 might be your first choice, but 1018 is very low cost and takes an admirable case hardening.
- And one sure way to make a machinist smile is to give him or her an order for some brass or magnesium parts—both are free machining, and might just have the mechanical and chemical properties needed for your application.

- On the plastics side, we have nearly three dozen to choose from. All of them are relatively easy to cut, a factor that often—but not always—equates to lower cost parts. Some plastics offer superior wear, corrosion, or chemical resistance, others perform well under heat or flame, while still others offer excellent strength, impact, or electrical properties.

There are too many metal and plastic to list here, but feel free to check out the complete material list or call one of our application engineers for advice.

Turning to Lathe For Parts with Cylindrical Features

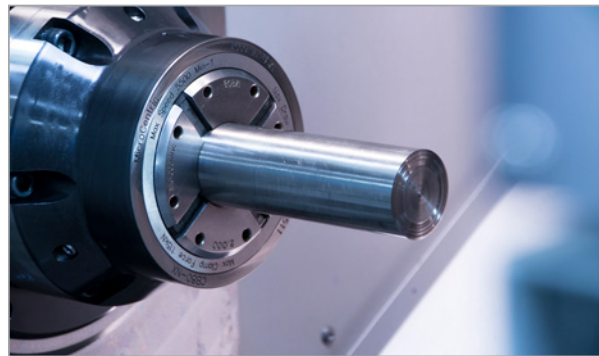
It was once a no-brainer. Round parts were turned on lathes; non-round parts were machined on mills. With the advent of CNC machining centers, which interpolate round part features with ease, the line between the two machining processes became blurred.

The situation grew even more confusing when CNC lathes attained live-tool capability. Operations that were once the exclusive domain of the milling department were now coming off the lathe complete. As a result, deciding which machine is the best fit for producing any given part has become far more complicated than it once was.

Some parts are obvious lathe candidates. Consider the piston for a spool valve, or a hydraulic fitting. The cylindrical symmetry of these components, coupled with complex external geometry and challenging internal features, makes them permanent residents of the turning department.

Conversely, the rectangular valve body that mates with those turned parts, with its large milled surfaces, detailed pockets and intersecting bores will never be spun on a lathe, no matter how live-tool capable that machine may be.

These are some of the reasons why our CNC machining service has turned to, well, turning. By adding live-tool (end mill) equipped CNC turning centers to our already extensive milling service, we're able to produce a better surface finish on cylindrical features and at a typically lower price for customers. Lathe also makes more efficient the manufacture of those parts that may skate on the edge of our milling capabilities. And if the goal is eventual low-volume production, turned parts are good candidates.



Metal rod stock is firmly fixtured in a high-speed CNC lathe before turning and milling begin.

Have a design concept for the next best camera lens housing? How about a revolutionary drive shaft for a lawnmower, one with milled flats and threaded cross-holes? Chances are, we can deliver a turned prototype for that geometry.

Turning parts such as these offers several advantages over milling. As implied previously, long length to diameter ratios on pistons and

shafts give mill operators an upset stomach. Turning a set of candlestick holders for your Aunt Martha's 60th birthday, however, is a piece of cake on a lathe.

If you're still unsure about what parts are lathe-worthy, consider a few household objects. A pint glass, for example, with its smooth, regular shape and length several times greater than the outside diameter is a straightforward exercise on a lathe. Not so on a machining center. A coffee cup, on the other hand, with its jutting handle and finger-ready hole, is impossible to turn.

Those 3 lb. dumbbells collecting dust in the closet could be turned fairly easily on a lathe. The wide, relatively deep recessed area where your hand grips the bar can be turned with a simple grooving routine, a feature that would be murderous to cut on a milling machine.

A small teacup saucer could go either way. Interpolating the concentric ridges and curved surfaces is equally possible on a mill or a lathe, requiring nothing more than accurate G-code and a suitable cutter. That said, it would almost certainly be faster to turn the saucer, and far more efficient in terms of material use.

How about a hose barb for the sprinkler? The v-shaped grooves would require a special cutter on a machining center, whereas a lathe can use a standard turning tool. The same holds true with a

replacement wheel for the barbecue grill, although milling those little mag wheel-like cutouts on the face would be challenging or outright impossible on most turning machines.

Soup cans and salt shakers, water bottles and flower pots—these shapes are what lathes are all about. Milk cartons and picture frames? Not so much. That's not to say we'll turn anything that comes our way. For the time being, we're offering parts in aluminum, steel and some stainless steel materials, brass, copper, and titanium.

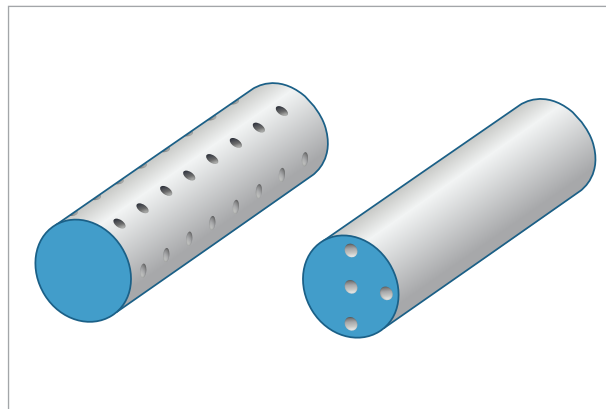
Size wise, we'll tackle parts up to 2.95 in. (75mm) in diameter by 9 in. (228mm) long, and as small as 0.03 in. (4mm) diameter by 0.060 in. (1.5mm) long. Sharp conical points are okay, as long as the angle is greater than 30 degrees. No promises on ballpoint pen prototypes, or models of soda straws.

As mentioned earlier, our new lathes have milling capability. Drilling a side hole or milling a flat is well within their means, as long as that feature is perpendicular to the long axis of the part.

Any milled grooves should be wider than 0.047 in. (1.2mm), but because all of our lathes are equipped with a Y-axis, we can machine slots or holes off-center (within reason).

Need your company's name engraved on the parts? Raised text is best, but we can machine most any lettering as long as the line width and character spacing measures at least 0.020 in. (0.5mm) across. Small holes don't scare us off, nor do threads. In fact, we'll drill radial holes down to 0.08 in. (2mm) and axial holes half that size. Likewise, we support a range of internal and external UNC, UNF and metric threads.

If you've previously used our machining service, you may find that turned surfaces are smoother and more round than their milled counterparts.



Protolabs has the ability to machine on-axis and axial holes as well as radial holes.

Milled features may have visible tool marks, but we can bead blast these surfaces to give them a matte finish and knock down any small burrs that remain after the machining process.

Like all machined parts, the process begins with entering your CAD model into our automated quoting system. Our software will attempt to determine the best machining method (milling or turning) for each material. This decision, however, can sometimes be overridden when you configure the quote. Each process has different capabilities for creating desired features and threading; pricing also varies between the two processes.

When you configure the quote, select the machining process that best meets your needs after weighing these factors. If a part does not qualify to be turned in a given material, this option will not be available in the quote for that material.

We've found over the years that making round parts out of square or rectangular stock can leave a few things to be desired. Milling away the square corners to get to the round part underneath takes longer than turning it from bar stock. It also creates more material waste. Turning those parts in a machine designed for such work is certain to be a win-win for everyone.

The Ins and Outs of Proper Threading

You can easily add threaded holes to your machined parts at Protolabs, and the addition of turning capability has created even more threading options. There are two basic types of machined threads: internal threads and external threads.

Internal Threads

Internal threads are machined using a single-lip threading tool—not a traditional threading tap. On parts with internal hole features in need of threading, the actual threads need to be removed from your CAD model, leaving only the pilot diameter. Our software recognizes a hole for threading if:

- it falls within the diameter range for the desired thread and,
- is in one of the three cardinal axes for milling or,
- is perpendicular to the axis of revolution for turning

Protolabs supports right-hand threaded holes on machined parts for UNC and UNF threads ranging from a #2 and up to a ½ in.; metric threads are also available ranging from M2 to M12. Location and method of manufacturing may limit some threads from becoming eligible.

Another key factor to keep in mind is our maximum depths for threads based on our threading tools. Our #2 threads have a maximum thread depth from the top of surrounding features of 0.200 in. and our ½ in. threads can go much deeper—up to 1 in. of threading depth is available. For metric threads, capabilities range from 5.08mm and 25.4mm in depth. Threads depths will gradually increase as the diameter of the threads increase. More on that later in the design tip.

Internal threads are also available on turned parts. Many of the same rules apply for machining threads using live tooling on a lathe, but the available threads are more limited than milling. UNC and UNF threads range from #4-40 to ½-20 and metric threads from M3 to M10. With the addition of live tooling, we are able to produce threaded features radially, axially, and on-axis.

This impacts the threading as some threads are available on-axis only. Please pay close attention to your quote or contact an application engineer for further clarification. A complete chart of turned threaded holes are available at [protolabs.com](https://www.protolabs.com), and be sure to select turned parts under standard holes to view the availability of these options.

In machining internal threaded holes, a hole may often times be longer than what our threading tools are able to reach. In this case, you have a few options depending on your needs.

- With a long through hole that exceeds the maximum reach, select the hole from the side that you anticipate the screw to be started from as shown in image 1 (page 9). If your screw is required to pass the entire way through the part, you would also have to pass a tap through the hole (in a secondary process) to complete it.
- You can also select both sides of the feature to be threaded as shown in image 2 (page 9), but notice the maximum thread depths as they overlap with each other in the hole. This raises concern with threading the features from both sides, you risk the threads of being cross threaded and a screw may not pass all the way through the part cleanly. But as long the threads don't intersect as seen in image 3 (page 9), selecting threads from both sides is typically fine.

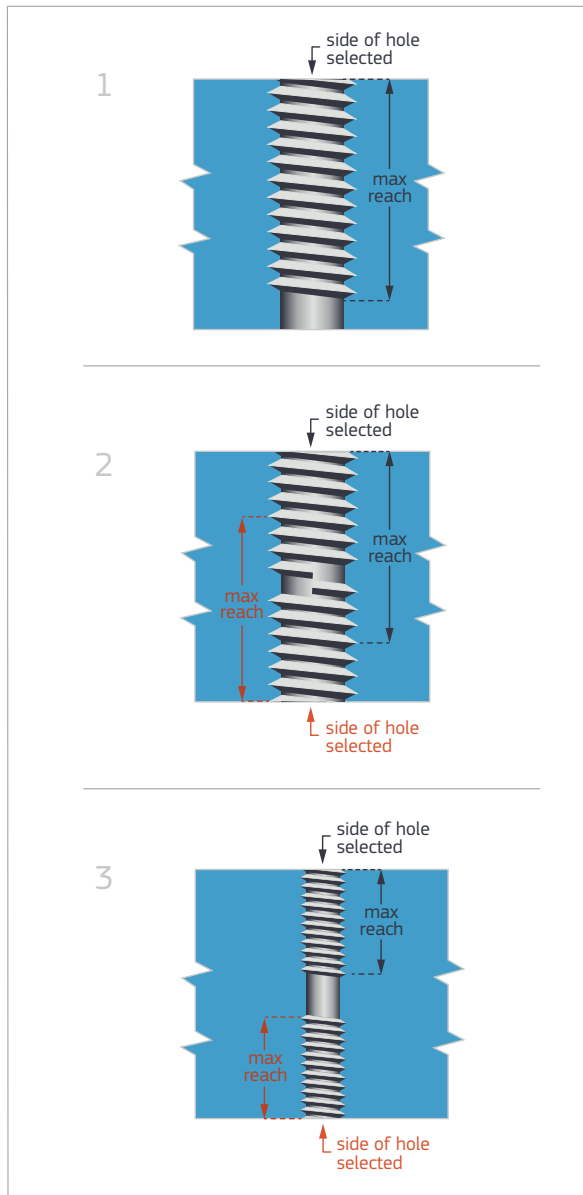


Image 1, 2, and 3 depict three methods of approaching maximum thread depths.

External Threads

The introduction of lathe has improved our external threading capabilities, and you can now get external threads for select sizes as long as your parts qualifies for turning. We still use a custom threading tool with a selected number of thread sizes, depths and placements within the part geometry. However, our advanced turning process offers external threads on the centerline of the part as well as live tooling that allows for internal holes to be threaded as long as they follow similar guidelines as milling. And it's not limited to on-axis holes—axial and radial holes are also available.

Just like internal threads, external thread design for turned parts need to have the thread removed from the CAD model in order for our software to recognize it. Additionally, please model your external threads for milling; don't model them for turning. After you receive your quote for turning, you will then have the ability to select the appropriate thread size.

External milled threads are designed into parts much less frequently than internal threads, but nonetheless, can still be machined effectively. External milled threads are produced on the half diameter, and then the part is rotated 180 degrees where the opposing threads are machined

in the same way. If you have larger, coarse external threads this does work well—we're able to produce quality ½ in. threads, but you may still want to chase the threads to remove any remaining material or mismatch as the threads are produced in two different machining set-ups.



A finished piece with external threads that has been turned on a lathe.

Smaller external threads such as a #6-32 are much more difficult to produce with a ball or flat end mill as a larger radii would be left in the root of the thread since the pitch is too tight. You would be required to chase the threads with a thread cutting die in order to remove the remaining material. On many parts, a 0.008 in. to 0.016 in. radius would be left.

Coil Inserts

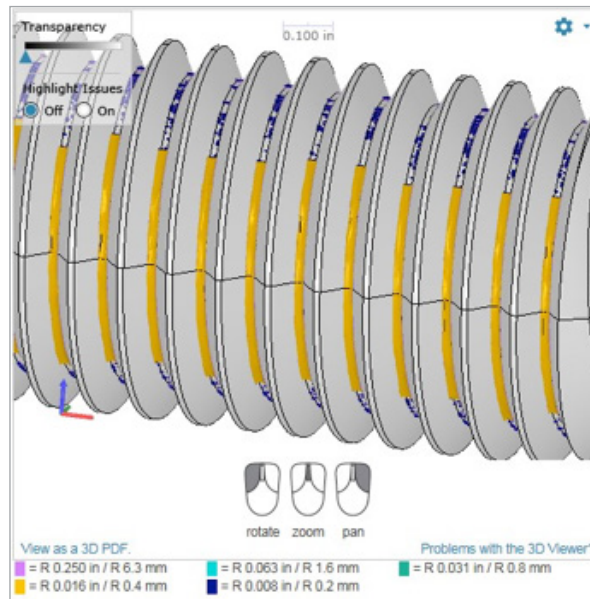
Coil inserts are also available if you require stronger threads. Coil insert threads are applied to the holes, but the coil inserts are not installed during manufacturing. The inserts would need to be installed by the user after the part is manufactured, but before the part can be assembled. Coil Inserts are available in UNC and UNF threads ranging from a #2 to 1/2 in., and metric from M2 to M12 with the same depths as standard threaded holes. Protolabs is optimized for HeliCoil brand inserts and we recognize standard sizes and lengths.

Threading in CAD Programs

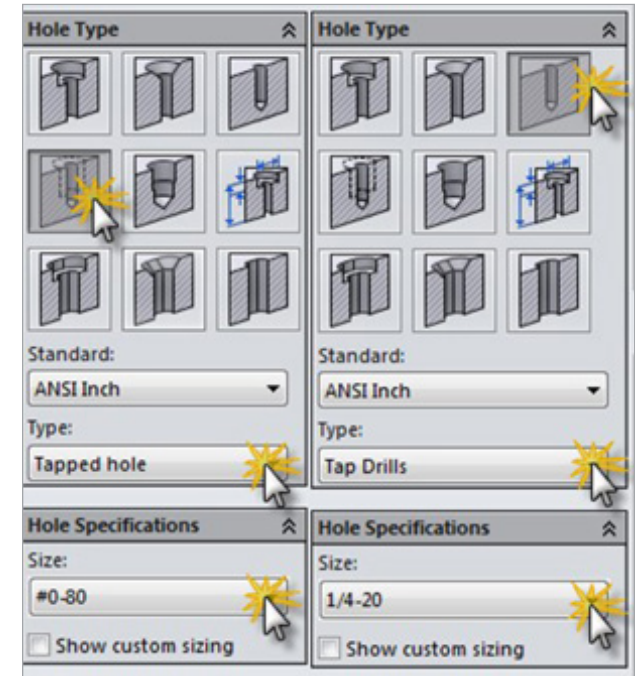
Many CAD packages allow you to display threaded features in a couple different ways including tap drill, cosmetic thread or by the major diameter of the thread. We suggest selecting the pilot diameter as long as it is designed at approximately 75 percent of the thread diameter.

The CAD file should then be submitted in a file format other than STL; we discourage uploading STL file formats for machined parts as our software is unable to recognize features like pilot holes in that format. A neutral file format such as IGES or STEP should be used if possible.

On milled parts that require external threads, you must design the threads on your part as this would then be in our standard milling procedure that uses ball and flat end mills. As stated earlier, this would not be the preferred method for producing threads as you may be required to perform a secondary process of chasing a cutting die over the threads to ensure the parts can be assembled correctly.



The CAD model illustrates an as-milled view of an external thread with resulting radii highlighted in yellow. This would be for milled parts only.



A pilot hole is designed—before threading—into a CAD model in a program like Solidworks.

On turned parts, external threading improves greatly because the part is spinning on center and a sharp single-lip threading tool can produce a quality thread. The design of an external turned thread is similar to that of the internal hole. You must remove the threads so software can view the outside diameter to determine the type of thread needed.

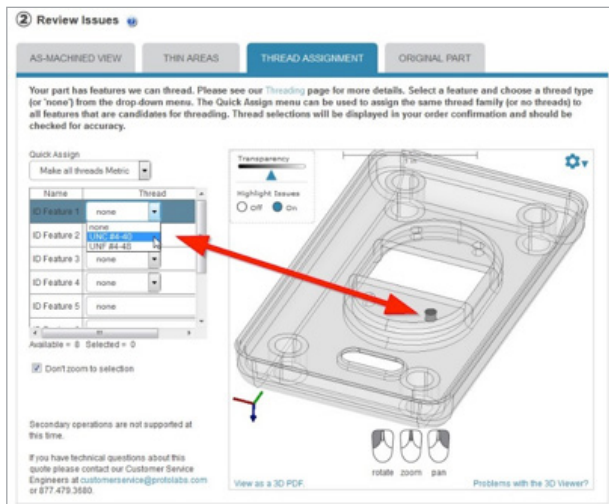
Threading in Quotes

Within your Protolabs quote, you have the ability to select both internal and external threads on part features. On the Thread Assignment tab of your quote you will notice a fully interactive model that allows you to select the threaded features that are available. Each eligible feature will be highlighted and you can manually select threads. Threads can also be mass-assigned in UNC, UNF and metric, which can be faster than assigning threads to individual holes, but be aware that you might inadvertently thread holes you weren't intending to.

If a thread you want isn't available, you may need to double check the diameter of your features to make sure they are within the guidelines for threading. Note that while reviewing our threading charts, you can toggle between our milled and turned options within each tab to review the availability of threads for each method of manufacturing. All threads need to be selected and saved before proceeding with an order. If you change your manufacturing process or material at any point, please check the Thread Assignment tab again as selections may change.

Protolabs offers threaded features for milled parts in plastic and metal, and threaded features on turned parts are only available in metal materials at this time. We can accommodate UNC, UNF and metric threads along with coil inserts (but do not supply or install the coil inserts).

[Click to view a complete list of milled and turned threading guidelines.](#)



Within Protolabs' interactive quotes, you can assign threaded features on your CAD model.

Moving Between 3D Printing and CNC Machining for Plastic Parts

Not so long ago, having prototypes or a few production parts machined was a more costly and time consuming process. You mailed a paper drawing to your favorite machine shop, had a short chat with the owner or shop foreman to discuss the due date and price, and waited. If you were in a hurry, maybe you paid for the overtime to get the parts done sooner, but prototyping was still an exercise in patience. You could get real parts, but at the cost of valuable production time and money.

Then came the service bureaus with their 3D printers. Plastic prototypes could now be ordered quickly and far more affordably than hiring a local shop to set up a machine. The only problem was the material—those early generation stereolithography (SL) systems were limited to one or two types of liquid resin photopolymers, good only for “show and tell,” or as patterns for casting and molding.

Additive manufacturing was in its infancy. Soon other methods with their own confusing acronyms became available. One of these is selective laser sintering (SLS), which uses a powder bed of nylon-based material rather than liquid resin. This eliminated the need for support structures during the build process as with SL, thus reducing post processing and saving time and money.

Stereolithography wasn't sitting on its laurels during this time. SLA parts today are made from a range of resins that mimic some of the properties of engineering plastics. These are available in different colors and strengths, making them suitable for more than dog-and-pony shows.

Machining has evolved over the years as well. For example, Protolabs' CNC machining service can deliver machined parts, in both plastic and metal, in less time than it takes to get a quote from other, more traditional machine shops. Upload a CAD file, choose the material and part quantity, and tell us when you need the parts. Chances are you can have parts by tomorrow and at a cost that is comparable to 3D printing, if not cheaper for certain geometries.

Like shopping for smartphones, the problem today is one of choice. How does a product designer or engineer know which process is best suited for his or her part design? Since machining has been around at least a century longer than additive manufacturing, let's start there.

Material Selection

For the most part, plastic is very easy to cut. Granted, glass-filled materials are somewhat hard on the end mills, and acrylics can chip. Teflon is slipperier than a politician. But for the most part machining plastic is as easy as it comes.

Protolabs can machine more than three dozen types of engineering-grade thermoplastics. Some of these are also available on the 3D printing side of Protolabs, but when it comes to material selection and quality, machining wins hands down. Acrylic, acetal, ABS, nylon, PC, PP, PEI, and PEEK—these and a host of other plastics cover the needs of nearly any part design imaginable. Final machined plastic parts can be used for form and fit testing, and even as functional parts in many cases.

On the 3D printing side of plastics, material options are more limited. SLA parts are built from photopolymer resins that mimic some properties of plastic like ABS, PC and PP. They work well for prototyping form and fit, but for functional parts, machined plastic is the route to travel if design will allow. Now the story is a bit different with SLS, which uses actual thermoplastic nylons to build parts. This process tends to bring added durability and stiffness to 3D-printed parts, but color options are literally black and white and surface finishes aren't nearly as nice as for machine parts.



Many milled plastic parts are used in functional, end-use applications.

Geometry

Where the machining process sometimes comes up short is part geometry. Protolabs uses 3- and 5-axis indexed CNC machining centers. An engine bracket, a camera body, the baseplate for a thermostat or even the thermostat housing itself—these are good candidates for machining.

Sharp internal corners on vertical walls are challenging, as are undercuts and pockets deeper than a couple of inches, but almost everything else is fair game. We're planning to launch a CNC lathe service for plastics to match our current CNC lathe for metals process. Cylindrical plastic parts—a drive shaft for a snowmobile, or the next greatest trailer ball hitch—can still be milled (or 3D printed) until then.

And then there are some part designs that machining can't touch. That's when additive steps up to the plate. It would be impossible, for

example, to machine a whiffle ball, whereas SLA and SLS can build that with ease. Machining the internal cooling channels in a heat exchanger would likewise be a challenge, but is possible in additive manufacturing.

Other parts are a tossup. A class ring can be machined or printed, but milling away the material inside the ring would be both wasteful and time consuming. The same holds true with a picture frame or a set of napkin holders. Because it can print just the material needed, additive wins.

Another large consideration to factor in with geometry is whether or not the part will eventually move into a manufacturing process with the ability to produce larger quantities, e.g., injection molding. A machined part typically has an easier path to molding than an additive part for two reasons: The material used in machining can more easily be duplicated in the molding process and printed parts with highly complicated geometries will most likely have to be modified in order to be molded effectively.

Tolerances and Surface Finish

Aside from material choices, there are other important differences between plastic 3D printing and machining processes. In terms of accuracy and surface finish, machining tends to be more accurate and has better long-term dimensional stability than SLA parts, with surface finishes being approximately equal between the two



SLA can produce intricate 3D-printed parts, such as this microfluidic component.

processes. SLS parts have rougher finishes than SLA or machined parts, and are more stable than SLA parts and approximately as dimensionally stable as machined nylon parts.

Prototyping with Injection Molding

The wild card in the additive versus machining plastic conversation is quick-turn injection molding. Even though injection molding at Protolabs is often used for low-volume production, most customers also use the service for manufacturing small quantities of prototypes and production parts with a similar turnaround as 3D printing and machining. If this is the case, different design considerations in areas like the aforementioned material selection, geometry and tolerances come into play.

Jigs, Fixtures, and Capacity Relief

Permission to speak freely? Protolabs is not here to replace machine shops.

Rather, we are often used as a pressure release valve for shops looking to manufacture and assemble parts faster with custom jigs and fixtures, and an external standby for when internal capacity is reached. Our mills let your skilled machinists remain focused on high-priority orders without interruption from one-off parts, and prevent delays in (or the loss of) new orders when all of your machines are running.

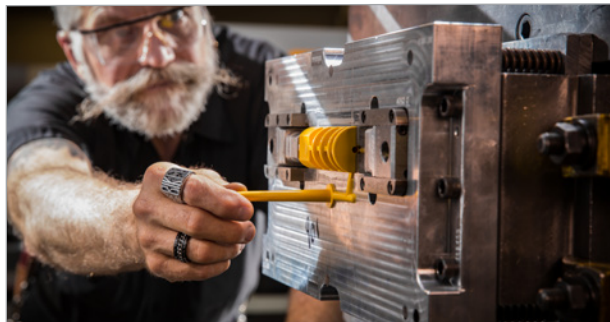
Jigs and fixtures are typically viewed as synonymous with each other; however, they technically vary slightly. Jigs support the relationship between a piece of work and the tool, while fixtures support the piece of work itself.

They both represent the hard-working, blue-collar sector of metal and plastic parts that are critical for production inside the manufacturing industry. Nearly all end-use products that are cut, formed, sheared, assembled or positioned involve some type of jig or fixture.

Avoiding Interruption

Progressive loading is a technique regularly employed by shops that house a limited number of machines. The bay on mill one is loaded, a part is machined, and the part shifts to the second machine where the next set of features are milled while another part is simultaneously loaded into bay one. Parts move through the required number of machines until all of their features are milled into a final part. Each machine needs a fixture to hold the parts as the milling tasks are performed.

Furthermore, it benefits shop owners to use the same material and setup for each machine during the entire progressive milling process. If a set of five machines are milling an order of aluminum parts, halting production for a one-off steel part not only delays a shop's delivery time, it contaminates the scrap metal bin of virgin aluminum material, which can be recycled.



Quick-turn injection molding is regularly used for low-volume prototyping and production of plastic parts.

It's an instance where shop owners and floor managers can turn to the mills at Protolabs for quick one-off machining that would otherwise slow down their own shop's production.

Drilling

Facilities that perform secondary manufacturing operations, for example, may have an order of plastic components that need threaded holes drilled into them. To do so, a fixture to cradle the components in the proper position is used to drill multiple holes in a single, fixed component while also achieving consistent hole placements from part to part. Cylindrical steel jig bushings can be inserted into the fixture to give further guidance to the drill. With thousands of parts that may need drilling, a fixture works to bring consistency and speed to the production line.

Assembly

Product assembly is another secondary process where jigs and fixtures lend a hand, and again, speed and consistency are paramount. They work as a template for assembly of end-use products in a factory or plant with a focus on a single, repetitive task that needs to be exact every time.

Welding

Shops use jigs and fixtures for welding support as well, but it's important to note the materials of both the weldments and the piece securing the weldments. A set of dissimilar materials do not run the risk of welding the fixture and the parts together whereas identical fixture and weldment materials can become joined together.

Materials Selection

For most jigs and fixtures, the material chosen is typically based on a combination of application, durability, and—like welding—the material of the pieces that it will interact with. The metal or plastic that is used in fixturing is often of a similar hardness, if not softer than the piece it will be holding, to prevent any marring on the piece.

You typically wouldn't use an aluminum jig for an electrical application because of the potential for static electricity and interference. In welding, dissipating heat is important, so materials like copper or brass, which dissipate heat well, may be advantageous as a welding fixture.

Does it need to be disposable? Sometimes fixtures have to break in order to work properly, and now you have the ability to make a disposable fixture as needed. Don't buy an expensive fixture set before you've tested it, because you can essentially prototype a fixture before you've truly committed to a shape.

Increasing Capacity

Jigs and fixtures are one-off parts that help machine shops run more fluidly, but what happens when a shop's capacity is reached? Orders may be delayed or turned away entirely, creating an unfavorable situation for shop owner and customer. Protolabs is used by shops as an extension of their own services. We don't require a fixturing fee, and due to hundreds of in-house CNC machines, we can mill and ship parts to you in as fast as one day. That allows shops to continue providing parts to customers even when their own mills are running at full capacity.



CNC machining at Protolabs is leveraged by machine shops for jigs and fixtures.

Further capacity relief is seen when shops use Protolabs to quickly mill the majority of a part or run of parts (often faster than a shop can order material), which is then sent back to the shop for subsequent milling. This allows shops to offer their customers additional value by:

- Drilling deeper and off-axis holes.
- Cleaning up critical features that have tighter tolerances than Protolabs' standard ± 0.005 .
- Adding any secondary operations that Protolabs does not currently support.

Not Just Shops

Jigs and fixtures are, of course, not only limited to machine shops. They are used by various companies in all forms of production lines and during research and development testing. Some of the most agile companies will even prototype their jigs and fixtures at the same time they prototype their parts, which drastically speeds up the product development cycle. But perhaps, that's a conversation for another time.



A custom fixture holds a part in place to provide consistent results each time.