WHITE PAPER

HOW TO FINE-TUNE PARTS FOR EFFICIENT CNC MACHINING

Explore how to fine-tune part designs for manufacturability (DFM), as well as key drivers of machining time and cost

While a part's design may meet all the right functional requirements, that doesn't mean the design is suited for cost-effective CNC machining. In fact, some part features may be difficult or even impossible to machine as designed, which drives cost, quality and yield problems.

In this white paper, we'll present some real-world examples of how we fine-tune part design for efficient machining — a practice known as design for manufacturability (DFM). In particular, we will identify some key drivers of machining time and cost:

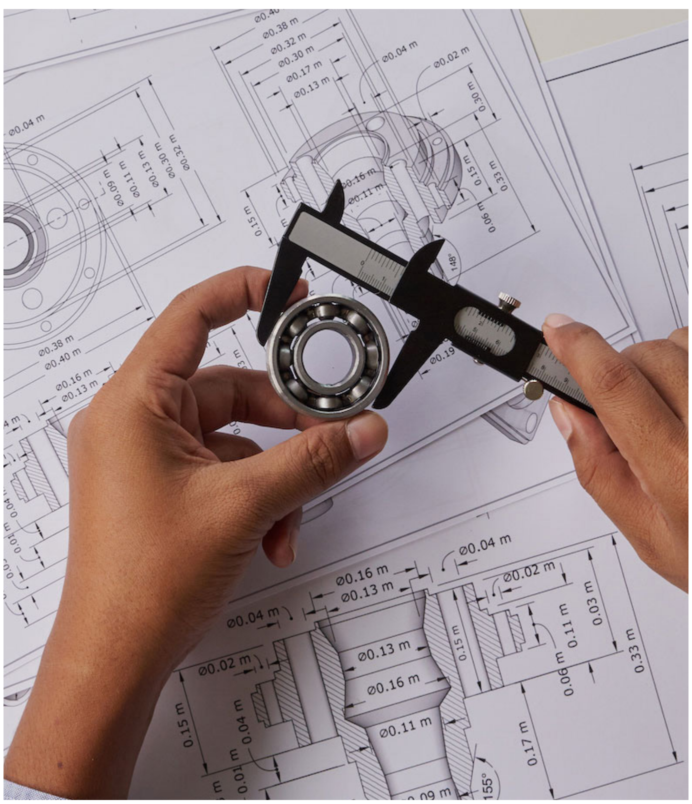
- **Tolerancing**. It's easy to specify overly tight dimensional tolerances, increasing production time, reducing yields and driving up costs.
- **Feature sizes**. You'll want to make sure your part features can be manufactured using standard, rather than custom, tools.
- **Surface finish**. Designing parts with a finer surface finish than what you need will lead to costly, inefficient CNC production.

- **Radius, chamfers, crossholes and deburring**. How your part's edges are broken around holes and other features affects cost — making part radii, chamfers and crossholes important variables to consider during the design process.
- **Contours**. Just because a part can have many contours doesn't mean it should. Simplifying a part so that it can be made using a three-axis machine, rather than a five-axis machine, will put money back into your pocket during production.
- **Inspection**. It pays to tweak part designs to facilitate inspection processes — making this variable an important, yet subtle part of the design process.

Dimensional Tolerances: Pay Attention to the Details

When overly tight, tolerances can easily drive up production time, lead you into more costly machining territory and reduce your yields. That's why it's always a good idea to ask yourself if a part really needs the tightest of tolerances from a functional standpoint. If the part does, then we're equipped to handle even the most demanding of tolerances. But if it doesn't, then loosening the tolerances where appropriate can drastically improve the part's machinability in terms of time and cost.

Tolerances refer to the acceptable amount of dimensional variation that will still allow an object to function correctly.



They can apply to the nature of a part's form, whether flat, straight or circular, or to location, whether symmetry or concentricity. Other types of tolerances include feature orientation, profile and runout.

The drawbacks of overly tight tolerances boil down to time and cost. For example, at a certain dimensional threshold, hole sizes will require custom or specialized tools, adding cost. Or the machine shop may have to switch from machining, to electrical discharge machining (EDM), jig boring or water jet cutting to hit the tightest hole size specifications, adding time and skilled labor costs.

Even if the tolerances are dialed in functionally, the manufacturing process itself can introduce challenges, leading to low yields and additional costs. For example, it's not uncommon for a machined part to meet tolerance on the warm shop floor, but cooler temperatures during inspection can throw the part out of tolerance.

A Few Real-World Examples

One example of a part with overly tight tolerances is a spider pack, which is part of a rod cluster control assembly for nuclear power plants. This component consists of 24 legs, each of which features a hole with a dimensional tolerance of 0.280 inch — plus or minus half a thousandth. To put these numbers into perspective, the spider pack must include tolerances that are 8 times tighter than a strand of human hair.

Even if one tolerance is off by a tenth of an inch, the part must be scrapped. Making matters even more challenging, the spider pack consists of 24 holes, leading to very high scrap rates and inconsistent delivery times — both of which significantly drive up manufacturing costs.

In a second example, we recently machined an aerospace component that will be sent to the moon. To reduce weight, we were tasked with cutting out pockets with a tolerance of plus or minus 5 millimeters. Bear in mind, these pockets served no other functional purpose than to reduce overall part weight, begging the question: why pay more for such tight tolerances?

It's always a good idea to ask yourself if a part's tolerances need to be so tight. Many down-the-line manufacturing issues related to tolerancing can be easily avoided if addressed during the initial design phase. In many cases, taking the part back to engineering once it already hits the shop floor will prove too costly an endeavor.

Avoid Custom Cutting Tools To Save Cost

Dialing in tolerances is just one of the many ways we can fine-tune part designs for efficient machining. A related manufacturability issue has to do with designing part features whose dimensions require the use of custom-sized end mills and other cutting tools. Holes, grooves, radii and chamfers are all examples of part features that may require a costly custom cutting tool if the feature's callout dimensions don't match a standard-sized tool.

For example, we recently received a part with a 0.18-inch radius, begging the question: Given the availability of standardized three-eighths-inch end mills at 0.1875 radius, are these 0.0075 inches important enough, from a functional standpoint, to trigger the added expense and lead time for a custom end mill?

Or in another recent example, we received an order for a part with a 0.188-inch corner radius that was required to hit a depth of 1.900 with a 0.375-diameter tool. In this case, we would have to take many slow step-down cuts to get to this depth. We also learned this radius would not interfere with any holes or other functional properties. After going back and forth with the part's design engineer, we were able to bump the radius up to 0.260 inches, which meant we could use a 0.500 endmill to full depth with next to no push off and in one third the time — saving the customer the cost and lead times of procuring the custom tools.

Why Engineers Go Custom

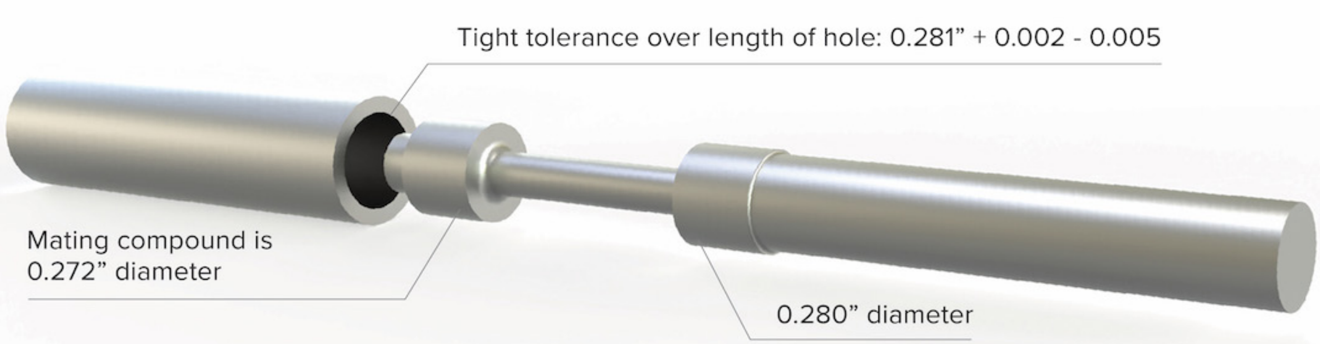
Both of these examples, and countless others we've seen over the years, had no functional requirement for the non-standard feature size. So why the unusual dimensions? In some cases, the culprit is an "exact copy" mentality where dimensions from earlier cast parts or prototypes are carried through into the production drawings. Other times, it's a metric conversion of the called out feature dimension. Keep in mind, however, that standard cutting tools are available in metric, not just in U.S. Imperial sizes. In these cases, it pays to give your machine shop the drawing with the original metric measurements, rather than wasting time and energy to convert the callouts to something that may not correspond to a standard U.S. tool size.

The consequences of calling out feature dimensions that won't work with standard cutting tools can be significant. Some of that cost is the price of the cutting tool itself, which can be exacerbated by the fact that custom-sized tools tend to break and wear prematurely. Then there's the opportunity cost associated with longer lead times to get your parts.

And finally, there's a hidden cost to special feature sizes that may be less obvious: Features machined with standard cutting tools can often be inspected with simple gage pins and similar inspection tooling. Features produced with non-standard cutting tools may need a trip to the coordinate measuring machine (CMM), further adding to the production costs and lead times.

OVER-SPECIFIED FEATURE EXAMPLE

In this example, the tight tolerance on the hole diamenter extends for the length of the part, but only a portion of the part actually needs that tolerance given the size of the mating component.



If your part has features that require a custom tool to meet functional requirements, we can make just about anything you want to put on the drawing. But it's worth investigating, early in the design process, whether there are features whose dimensions can be eased to match standard tools without interfering with the functionality of the part.

Surface Finish: A Balancing Act

Whilte it's clear that over-specifying certain part features — such as dimensional tolerances, grooves, holes and radii — will drive up your production time and costs, surface finish can be a different story. While it's true designing a part with a finer surface finish than what you need can lead to inefficient, costly CNC machining, more often than not we see the opposite occur.

In our experience, the surface finish specified on a drawing does not always reflect what a customer wants. For example, a 125-microinch Ra surface finish will appear nice and smooth to the naked eye, while a 250-microinch Ra finish will appear rougher. And whether for aesthetic or functional reasons, we have had customers come back to us after the fact because they want the surface to be smoother.

In other words, although the drawing may indicate 250, what the customer really wants — and expects — is 125 or smoother. This is especially true for part features like water holes. And in many cases, this back-and-forth will drive up production time.

On the other hand, it's important to remember that the finer the surface finish, the more labor will be required to achieve it. For this reason, overspecifying finishes can also drive up time and cost. For example, while we can easily hit 125–250 surface finishes with water jets and CNC milling, achieving smoother surfaces will require more specialized tooling or bench work, translating to significantly higher costs.

Work With Your Machinist

While it's important to know what you want in terms of surface finish before working with your machinist, the best machining companies will ensure they deliver what you're looking for up front. For example, at L&S, we work with our customers during every step of the machining process to avoid under- or over-delivering surface finishes. And as a result, we keep your cost and production time to the minimum.

Learn more at www.lsmachineco.com.

Ra (microinch)	**RMS** (microinch)	**Surface Finish Appearance**
250 – 320	275 – 352	Visible machining marks with cutter lines that you can feel.
125 – 200	137.5 – 220	Visible, but not obvious, machining marks with cutter lines that are much harder to detect.
63 – 100	69.3 – 110	Machining marks blur together, but the direction is obvious.
32 – 50	35.2 – 55	Directional marks are visible but not obvious.
16 – 25	17.6 – 27.5	Directional marks are blurred, and the cutter lines cannot be picked up.
8 – 12.5	8.8 – 13.75	Directional marks are not visible – closer to a mirror finish.
4	4.4	Mirror-like finish.

Figure 2.
Determining Surface Finish
What certain surface finishes may look like. Roughness Average (Ra) and Root Mean Square (RMS) are both common representations of surface roughness, but each one is calculated differently.