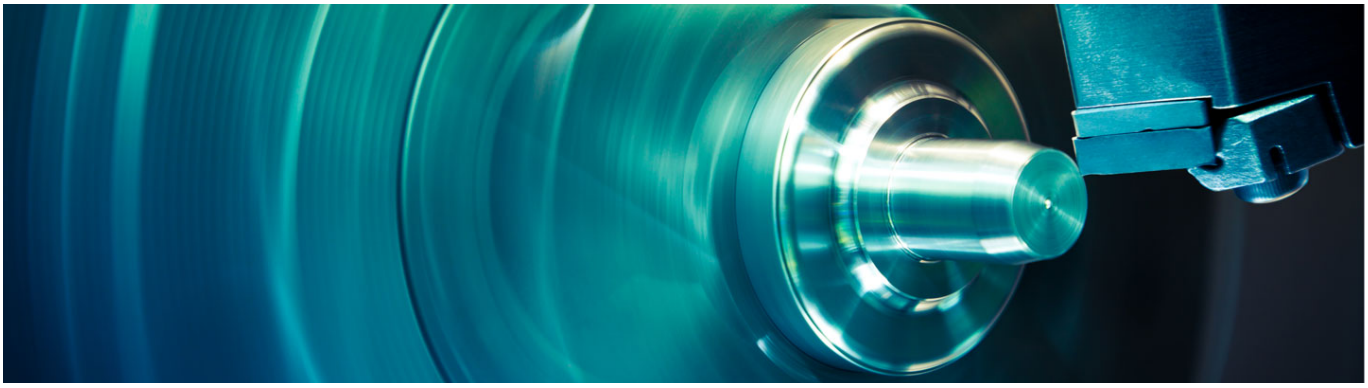

Machining

5 Ways to Prolong Tool Life and Amplify Productivity

Kip Hanson | Mar 26, 2019

What's the secret to optimizing throughput and lengthening tool life? We boil it down to the top methods that will deliver the biggest part-making bang for the buck.

Choosing the right feeds and speeds when machining demands a delicate balance between productivity, part quality and tool selection. If you push machines and cutting tools too hard, the time needed to change tools might eat up any productivity gains—and machinists might burn through their tools too quickly.

On the other hand, babying machines might reduce tooling costs at the expense of throughput. And we all know that in the part-making business, time is money. That's why finding the sweet spot of optimal tool life and productivity can seem like an elusive quest, one that even experienced machinists and programmers struggle to satisfy.

Here are five techniques that will help machine operators increase tool life and find the right balance of speeds and feeds for any application.

Tool Life: Focus on Speeds and Feeds

Of the three cutting parameters—feed, speed and depth of cut—cutting speed is easily the most important. It's measured in surface feet per minute (or meters/minute for you metric shops), and you can think of it as the rate at which the tool and workpiece move past one another.

Always start any machining operation by determining the optimal cutting speed based on the material being cut and the carbide grade and coating used to cut it—higher cutting speeds produce more heat and therefore increased flank wear, but running faster might just help to reduce built-up edge (BUE), the chief cause of failure in nickel-based alloys.

Better MRO is here to help: Find out your key metrics with our machine and productivity calculators.

Consider Flank Wear in Your Feed Rate Formula

Here in the U.S., lathe operators measure feed rates in inches per revolution (IPR), while milling people use inches per minute (IPM), determined by chip load per tooth times the number of teeth.

In either case, feeding faster produces more heat and increased flank wear, but it also produces more parts in less time.

Flank wear is the best wear to have, so it's a good idea to feed at the upper end of the cutting tool manufacturer's recommendations. That's assuming, of course, that your machine tool is sturdy enough to handle it and that you have a firm grip on the workpiece.

> ## The Importance of High-Pressure Coolant
>
> Re-cutting chips is a killer to productivity—and the time as money equation.
>
> The best way to help optimize chip cutting is by applying a stream of well-filtered, high-pressure, ***through-the-tool cutting fluid***. Doing so flushes chips away from the work zone while reducing heat and improving available lubrication, which in turn eliminates a host of tool wear problems, including built-up edge (BUE), chipping, cratering and more.
>
> Smaller tools require more pressure and less flow, while larger ones are just the opposite.
>
> For general-purpose work, 1,000 psi and 10 gpm are good places to start. If your shop does both large and small parts—or small part features, consider a variable flow system. These systems might seem expensive, but they'll pay for themselves in no time at all.

Optimize Your Depth of Cut

Determining the optimal depth of cut (DOC) with turning applications is largely a matter of how much material you can remove per pass without yanking the part out of the chuck, although DOC should never exceed one-half of the insert's inscribed circle (IC).

On machining centers, DOC values are radial and axial. Radial measures the width of the cut. Axial measures the depth into the workpiece.

Trochoidal and other modern milling strategies generally rely on a light radial DOC and as much axial engagement as the cutting tool, toolholder and machine can handle.

DOC has the least effect on tool life, even though a light DOC means more passes will be needed to finish the roughing operation, which could mean greater insert consumption.

A word of caution: An overly aggressive DOC can lead to chatter, broken tools, or parts flying across the shop.

Take Advantage of Precise Tool Grades

It’s easy to make a case against “too many inserts” in the tool crib. Excess inventory can add up, so more insert styles, grades, coatings and brands than are absolutely needed to meet the shop’s machining needs may cause unintentional waste.

There’s another side to the coin, however, and that’s the fact that cutting tool manufacturers go to great lengths to optimize their wares for specific material classes and even specific materials to maximize performance.

General-purpose cutting tools are fine for general machining work, but with high-volume part-making with difficult materials such as ***composites*** like CFRPs, ***titanium*** and others, it pays to fine-tune your ***insert selection***.

Use Proper Programming

The perfect feed, speed and DOC combination is meaningless if the toolpaths are off. That’s why machinists should evaluate all of the available techniques in a programmer’s bag of tricks.

There are the trochoidal toolpaths, for example, that leverage the ***chip-thinning effect*** to improve virtually every aspect of machining.

Slicing can be used to “clean out the corners” in milled workpieces. Arcing into and out of the cut is effective on CNC mills and lathes alike. The same can be said for pre-chamfering of parts to reduce cutter shock on entry.

Plunge milling is an alternate but effective way to rough out deep pockets. And high-feed milling strategies drive cutting forces up into the spindle taper, which is helpful when faced with long tool overhangs.

There’s more: Positive ***lead-angle*** cutting tools do wonders to increase productivity, but be sure to increase feed rates to compensate for the thinner chip, lest rubbing will occur. Stay strong and minimize runout with high-quality milling chucks, shrink-fit, and hydraulic ***tool holders***.

Use hydraulic workholding wherever possible. And as far as feeds and speeds are concerned, always start with the cutting tool provider’s published guidelines. Chances are good they’re higher than expected (and no, that’s not so they can sell more inserts).

Any DOC horror stories to tell? Talk to your peers in the Better MRO forum [registration required].

