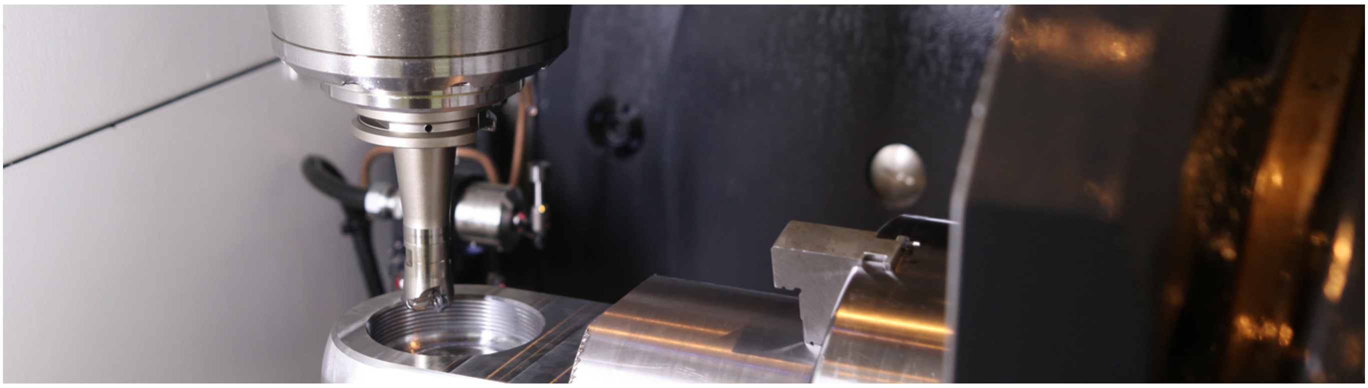

Technology

How to Maximize Machine Productivity: Chip Thinning

Kip Hanson | Sep 12, 2017

What You Need to Know:

Chip thinning is often radial in nature, but can be axial for face milling.
Trochoidal toolpaths allow for lighter radial cuts at higher feedrates and longer axial engagement, thus reducing cutting forces and improving tool life.
Newer, high-performance end mills, harder carbide, chip evacuation and firmer grips—and other tips and tricks, are all part of the new programming paradigm for maximizing machine productivity.
Every tool needs a retirement plan: Use a predictable, repeatable process to manage tool life.

From extending tool life to embracing patented metal removal rate tools and techniques, chip thinning is central to maximizing machine productivity in metalworking today.

For as long as machinists have stood before manual milling machines, they've known that as the width of cut decreases, so do cutting forces. The spindle becomes quieter; the handle easier to turn. Take too light of a cut without simultaneously increasing feedrate, however, and tool life suffers as the end mill rubs on itself—and becomes dull like an old butter knife. Learn how to use chip thinning to your advantage with today's technologies. Whether radial or axial chip thinning—or trochoidal toolpaths—the proof is in the tool and the technique.

Feedrates for Radial Chip Thinning and Face Milling: Do the Math

Radial chip thinning, which occurs whenever radial-cutter engagement falls below 50 percent of the cutter diameter, is just as relevant on CNC machining centers as it is on manually-operated knee mills. For example, a 1/2-inch, 4-flute end mill feeding at 0.01-inch per tooth (IPT) with a 1/4-inch or greater stepover (radial depth of cut) produces a chip thickness equal to the programmed IPT feedrate, or 0.010-inch. But decrease the stepover to 10-percent (0.050-inch) and the IPT value must be bumped up to 0.0167-inch to achieve comparable chip thickness.

Not all chip thinning, however, is radial. Face milling operations are typically performed with indexable 45-degree lead angle or round insert button cutters, both of which thin chips axially. They also direct cutting forces up and into the spindle, which is exactly where you want them to go. In either case, the feedrate will likely need to be increased to compensate for the chip thinning effect, oftentimes far higher than expected.

The chip thinning scenario becomes even more complex when cutting tools enter corners, race around islands, and dive deep into pockets. Feedrates must be continuously adjusted (as do the toolpaths themselves) to compensate for changing angles of cutter engagement and its impact on chip thickness. Fortunately, most CAM systems today are quite capable of making these calculations, and many leverage the chip thinning effect to greatly increase metal removal rates.

Getting Dynamic with Trochoidal Toolpaths

By taking lighter radial cuts at higher feedrates and longer axial engagement (oftentimes burying the end mill along its entire length), cutting forces are reduced and tool life improved. These "trochoidal" toolpaths present substantial opportunities to the machining community. The days of simple racetrack-style stepovers and incremental Z-axis cuts are being abandoned in favor of "Dynamic Motion," "Volumetric Milling," "Adaptive Clearing," and similar patented roughing strategies, most promising metal removal rates 1-1/2 to 2 times greater than traditional methods.

If your shop hasn't investigated this new programming paradigm, it's high time to do so. But even for those that are already using such toolpaths, a number of additional machining strategies should be implemented if maximum productivity is to be achieved.

Employ Newer End Mills

Yesterday's end mills might be unsuitable for today's toolpaths. Higher feedrates and spindle speeds are more likely to cause chatter, a condition that is often exacerbated on the lighter, faster, linear guideway machine tools common in shops these days. A wide variety of ***high-performance end mills*** are available, many equipped with high shear, variable pitch, and sometimes variable helix geometries that stop chatter before it can begin. Some have 5, 6, and even 7 flutes, making them especially well-suited to light cuts and high feedrates.

Take Advantage of Chip Evacuation

Opinions vary on this one, but few would argue that most of the heat generated during efficient milling operations leaves with the chip. Because of this, some suggest that an air blast to get the chip out of the work zone and avoid re-cutting is all that's needed when machining many steels and superalloys. Others swear by high-pressure coolant systems, capable of sending clean, filtered ***cutting fluid*** through the tool at 1000-psi or more. Which approach you take depends on the material, cutting tool, and tool coating, but both are worth consideration (as is minimum quantity lubrication, or MQL, an increasingly popular alternative to traditional cutting fluid).

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

Use Harder Carbide

Conventional "hogging" methods call for a tough carbide, able to withstand heavy cutting forces and vibration that comes with burying a tool. But light radial depths of cut and high feedrates mean that a harder grade of carbide can be used, in most cases promising longer tool life. And since greater amounts of heat are generated during high feed milling (HFM), a TiAlN or comparable multiphase tool coating should be used to provide lubricity, improve wear characteristics, and further extend tool longevity and predictability.

Find Terrific Toolholders

Ripping through a chunk of stainless steel or titanium at several hundred inches per minute is a beautiful thing, but only if there's no chance the cutter's going to come loose, sending shrapnel from one end of the shop to the other. HFM strategies such as these require ***high-quality hydraulic or shrink fit toolholders***, some of which have special grooves or flats to prevent pullout. And if you're going to spin the tool at more than 8,000 RPM (which you almost certainly will), you'll want to balance it as well. Runout and vibration are productivity killers.

Take a Firm Grip

While you're at it, how are you hanging on to the part? It's true that cutting forces are generally lower with HFM compared to the old rip and snort approach, but that doesn't mean the beat up machinist's vise you've been using since the first Bush Sr.'s presidency is the first choice for workholding. A variety of high-precision, powerful, and in many cases ***hydraulic or pneumatic clamping*** solutions exist on the market today. While you're at it, check out some of the quick change options, because once you adopt a HFM milling strategy, you're sure to be setting up jobs more frequently.

Of course, there's more to HFM than chip thinning and trochoidal toolpaths. Cutting tools should be eased into and out of the cut with gradual arcing motions. Internal corners may need to be "picked" to avoid excessive cutter engagement, possibly breaking the tool. Avoid sharp turns and 90-degree corners by using radiused toolpaths.

Create a Predictable, Repeatable Process for Tool Retirement

Above all, strive for predictable processes. All cutting tools will eventually fail, but knowing when and how wear occurs and then replacing tools before catastrophic failure is a no-brainer, even if it means retiring a tool before its time. Document your setup, and keep a continuous log of feed or speed changes, what tools were changed and why, and whatever machining observations seemed relevant at the time. Doing so will prevent surprises and allow you to maximize metal removal.

Is your shop embracing chip thinning techniques? Share your best, most effective or challenging applications.