Machining

Improving Cycle Times: 3 Experts Expose Wasteful Machine Shop Practices

Kip Hanson | Apr 05, 2022

Whether it's a trade publication, a website like this one, or the cutting tool salesperson who stops by every month to show you the latest product offering, there's no shortage of ideas on how to improve machining processes. And that's a good thing, because learning about new technology in all its forms is a key to shorter cycle times and increased competitiveness. But did you ever stop to wonder why your cycle times are so long to begin with?

Maybe you experienced poor tool life on a long-running job, and instead of taking the time to troubleshoot the problem, you just turned the feed rate override down a few notches and left it there. Or perhaps you broke too many taps on that batch of Inconel parts last month, and instead of calling the local sales rep for help, you decided to thread them by hand.

Unfortunately, situations like these are all too common, and left unchecked, are a cancer that only metastasizes as a ***shop*** begins to accept such competitiveness-killing time-sucks as normal.

Breaking Bad Production Habits

Eric Gardner has seen plenty of them. The North American application specialist at Seco Tools LLC ticked off several pet peeves, starting with overly long tool stickouts. "Shops often try to reduce the number of tools needed for a job or family of parts and end up sticking some cutting tools out farther than they should," Gardner says. "So instead of using a stubby 1/2-inch diameter end mill for roughing a pocket and an extended-length tool for milling the periphery, they use the longer tool for everything and have to drop their feed rates or depths of cut to compensate for the lack of rigidity."

For one or two parts, he notes, that's no big deal. It's a sure cycle-time killer on larger quantities, however. So are extended-length toolholders, and for the same reasons—poor rigidity and all of the problems such as chatter, deflection, and reduced feed rates that come with them. As such, shops should always opt for the shortest gage-length holders available, even if it means bumping up the tooling budget to do so. Contrary to what many in the machining world have been told over the years, the best way to do this might be moving to a century-old and oft-maligned technology, the Weldon-style side lock toolholder.

Read more: Manufacturing Guide: How to Become a CNC Machinist

“The bore on a high-quality side lock is ground a few microns off-center, so when it’s tightened, the tool runs perfectly true,” Gardner says. “This eliminates any arguments about runout, and as I suggested earlier, Weldon toolholders offer some of the shortest gage lengths available, bringing the cutter up very close to the spindle face. I've worked on numerous high-production applications where I've intentionally used side lock holders for that very reason.”

How to Stop Babying Tools

Thomas Raun, chief technical officer for Iscar USA, agrees on the last point, although for a different reason. “When we suggest going to traditional Weldon side lock technology, people sometimes look at us like we're crazy because of all they've been told about the higher tool runout,” he says. “But let’s face it, when you're hogging with an end mill, that little bit of runout has very little effect on tool life and is easily offset by the anti-pullout assurance provided by Weldon-style clamping.”

Unless you’re willing to spend the money on a Safe-Lock or comparable anti-pull toolholder, he adds, a side lock is the best way to avoid the end mill creep that will quite possibly scrap an expensive workpiece. It also gives shops the confidence to increase feedrates and depths of cut to their proper levels, thereby eliminating any cycle time lost to babying the cutting tool.

> **"I go into far too many shops to introduce a new cutting tool or troubleshoot a machining process and the response is, ‘Sorry, we want a tool that's just like the one we've been using for the past 20 years, except at a lower price.'"**
> Evan Duncanson
> EMUGE-FRANKEN

Raun seconds what Gardner says about rigidity and tool lengths. One area where they might disagree, however, is programming methodology. Where Gardner points out that many shops remain reluctant to adopt trochoidal and similar high-feed, light depth-of-cut milling strategies, Raun points out that, given a sufficiently rigid setup and machine tool, more traditional “hogging cuts” might be a better idea.

“Instead of embracing one programming approach to the exclusion of others, shops should instead focus on maximizing their material removal,” he says. “That’s the surest path to increased productivity.”

Read more: How to Optimize Your Machine Setups to Minimize Chatter

Cutting tool selection follows in the same vein. Machinists have long been taught that the bigger the tool, the better. And yet, using a shell mill or face mill too large for the machine tool will force the machinist to throttle back on the depth of cut and/or feedrate, wasting time and shortening tool life. A better approach is to go with a smaller cutter that can be pushed to the correct cutting parameters without stalling the machine’s spindle or servomotors.

On the CNC-turning side of the shop, Raun says customers try to save money sometimes by buying smaller stick tools and then shimming them up. “Say the lathe turret is designed for 1-inch tooling, but they go with 3/4-inch tools because they're cheaper. Now, they've reduced rigidity, and are sacrificing tool life and performance on every job because they didn't want to spend a little more money on the right stick tool upfront. That’s where the proverbial stepping over a dollar to pick up the pennies came from, right?”

Poor Purchasing Decisions

For Evan Duncanson, a milling application specialist at EMUGE-FRANKEN, one of the biggest time wasters is the well-known “that's the way we've always done it” attitude. “I go into far too many shops to introduce a new cutting tool or troubleshoot a machining process and the response is, ‘Sorry, we want a tool that's just like the one we've been using for the past 20 years, except at a lower price,’” he says.

Duncanson’s job is to steer people away from that low-cost cutter mentality. He and others like him are there to improve machining processes, not replicate those used in the ’90s, but improvement requires the adoption of ***modern cutting tools and techniques***. Unfortunately, it isn’t always possible because A) the customer is unwilling to change, or B) the purchasing department puts up roadblocks on tooling spend. “I worked with a shop just last week that wouldn’t spend 20 percent more for an end mill, even after I proved to them it would increase tool life by a factor of 10 and double their output,” Duncanson says.

Other examples from Duncanson’s time-waster hit parade include continuing to use a pecking cycle after switching to a through-the-tool coolant drill, refusing to try newer technologies such as thread milling or circle segment milling (also known as conical barrel milling), applying the wrong tool geometry or coating for the workpiece material, and as Raun suggests, using cutters that are too large for the application.

"For example, I'll see shops milling little 1/4-inch-wide parts with a 1-inch rougher and then wonder why it’s making such a terrible racket,” Duncanson says. “And sometimes you’re standing there talking to the operator and notice that the feed rate and spindle overrides are set at 75 percent and no one really knows why, while others say they can't utilize trochoidal toolpaths because their outdated CAM system doesn't support it. There's no shortage of time wasters out there. The trick is to get shops to see their bad habits and then get them to try a better, more cost-effective and productive approach.”

What routines are slowing cycle times at your shop and how are you addressing them? Tell us in the comments below.