Metalworking

New Tools and Tactics to Tame Drilling’s Trickiest Tasks

Kip Hanson | Sep 05, 2023

Holemaking is one of the most common metalworking operations. Unfortunately, that doesn’t keep it from being one of the most difficult.

Even the stiffest drill tends to wander when cutting conditions are less than perfect, especially in very deep holes. Poor chip evacuation is also normal, a phenomenon that leads to premature tool wear and even breakage.

Similarly, it can be difficult to introduce sufficient coolant into drilled holes, further increasing tool wear and degrading both hole accuracy and surface finish.

Advanced tools and improved methods, however, make it easier to avoid the heartburn and produce holes that are straight, round and smooth, according to Paul Larson, product manager for drills and thread mills at Guhring Inc., and Corey Schwenke, Sandvik Coromant’s product manager for solid round tools in the Americas.

Larson and Schwenke each have years of holemaking experience, and they spend their workdays sharing the latest developments and best practices with machinists and other metalworking professionals (like you). They back up their recommendations with broad selections of high-quality, solid carbide drills, many of them tailor-made for specific metals and holemaking applications.

Best Practices in Holemaking

Larson and Schwenke agree on many points, beginning with this one: that solid carbide drills are generally the preferred choice for making holes under ¾ inch or so in diameter, especially smaller holes—say ¼ inch and below—where carbide’s higher cost relative to high-speed steel (HSS) drills is less of an issue.

“If you perform a cost-per-hole analysis, you’ll find that carbide pays for itself very quickly, especially on smaller drills,” Larson says.

Additional benefits include:

- **Hole straightness:** Carbide is less prone to drill walk. That’s because the ***Young’s modulus*** (a measure of its stiffness) for tungsten carbide is nearly 700 GPa (gigapascals), several times that of

high-speed steel.
- **Wear resistance:** On the Rockwell scale, carbide comes in at a whopping 90 HRC, compared to 58 to 68 HRC for high-speed steel. This means faster cutting speeds—at least four times that of high-speed steel—with greater abrasion resistance, increased tool life and commensurately higher productivity.
- **Simplified setups:** The attributes listed above also simplify holemaking operations. Where a high-speed steel drill will deflect and possibly break without a starter hole of some kind, carbide can often be applied without one. And because there's less wandering, machinists can reduce the number of tools needed for deep holes.
- **Forget the peck:** Where HSS drills need regular pecking to clear chips and introduce cutting fluid, their carbide counterparts are usually "coolant through." That means no pecking, and when used with high-pressure coolant, carbide produces more holes in less time and with better hole quality.

Alternatives to Carbide: Modular and Indexable Drills

Drilling is used in a broad range of jobs, however, and what's good for one holemaking application might be less effective for others. For instance, while solid carbide offers many benefits, particularly when making smaller holes, what about larger ones where its cost becomes prohibitive?

Here again, cutting tool manufacturers have come to the rescue with various cost-effective solutions. For example, modular drills boasting quick-change carbide heads ease into solid drilling territory at diameters of around ½ inch and continue up to 1½ inches and beyond, depending on the manufacturer.

Beyond this are the ***indexable drills*** that machinists have been using for the past few decades, albeit versions with better carbide, coatings and geometries, followed by annular cutters and trepanning tools, which are technically not drills at all.

Then there's gun drilling. Like the methods just mentioned, it's also a specialty application and is beyond the purview of "regular" holemaking, but machine shops are likely to benefit from understanding its capabilities as an alternative (but probably outsourced) holemaking technique.

Part of that understanding is the realization that gun drilling might be unnecessary—Sandvik Coromant, Guhring and other cutting tool manufacturers offer coolant-fed, solid carbide drills in lengths 30xD or greater. Holes of that depth were, at one time, gun-drilling-only candidates.

"Just be sure that you slow these long drills way down—say 100 rpm or so—until they're inside the pilot hole," Schwenke says. "And if you do need to clear the chips, set the G83 clearance plane to just below the part's upper surface, keeping the drill contained."

Back to Basics with Solid Carbide

Returning to solid carbide drills, which can (or should) make up the lion's share of all holemaking tools, some basic ground rules apply if you want to maximize tool life and get the most parts out the door in the least amount of time:

- **Minimize runout:** Whether using a drill, end mill or reamer, rotary tool runout must be kept to a minimum. Larson and Schwenke suggest that high-quality, well-maintained hydraulic toolholders are among the best ways to achieve this but agree that shrink-fit and mechanical holders have their place. Whatever the toolholding solutions, double-check the assembly with a dial indicator or, better yet, an optical or vision-equipped offline tool presetter.
- **Spotting success:** While "spot drilling" or drilling of a pilot hole is often unnecessary with a solid carbide or modular drill, that perk vanishes beyond a certain length-to-diameter ratio, which most experts say starts at 10:1, less in tough materials like Inconel and austenitic (300-series)

stainless steels. Here, drilling to 1 diameter deep with a stubby drill is advisable, followed by progressively longer drills as needed to reach final hole depth.

It's also important to adhere to the manufacturer's recommended feeds and speeds, and when possible, apply at least 60 bar (870 psi) of filtered, properly mixed cutting fluid.

Read More: ***From Carbide to Coolant-Through: Deep-Hole Drilling, Simplified***

Oil is another alternative, but it does not offer the same cooling capabilities as water-based alternatives (but boasts greater lubricity). "Coolant pressure is an important consideration, especially with smaller drills," Schwenke says. "Good filtration for your coolant system is also needed and should not be overlooked."

When Ideal Cutting Speeds Are Out of Reach

All this good advice becomes even more crucial with "micro" drills, or those under 1 millimeter in diameter. Here, even the smallest amounts of tool runout or failing to follow feed and speed recommendations are deal breakers (and quite possibly, drill breakers).

The catch is that achieving recommended rates can be tough. Do the math on a 0.02-inch microdrill like those offered by Sandvik Coromant, Guhring and many other cutting tool providers and you'll quickly see that your machine tool might be inadequate.

For instance, a cutting speed of only 200 surface feet per minute (SFM) requires a spindle speed of 38,200 rpm, far faster than most machining centers can achieve. Most modern cutting tools can run at surface speeds much higher than that, even in superalloys and hardened steels.

Larson's advice is to do the best you can with your equipment. "A speeder head might help in this situation, but in most cases, you're going to be under the recommended cutting speed. About the only thing you can do here is adjust the feed rate downward to maintain the proper chip load."

