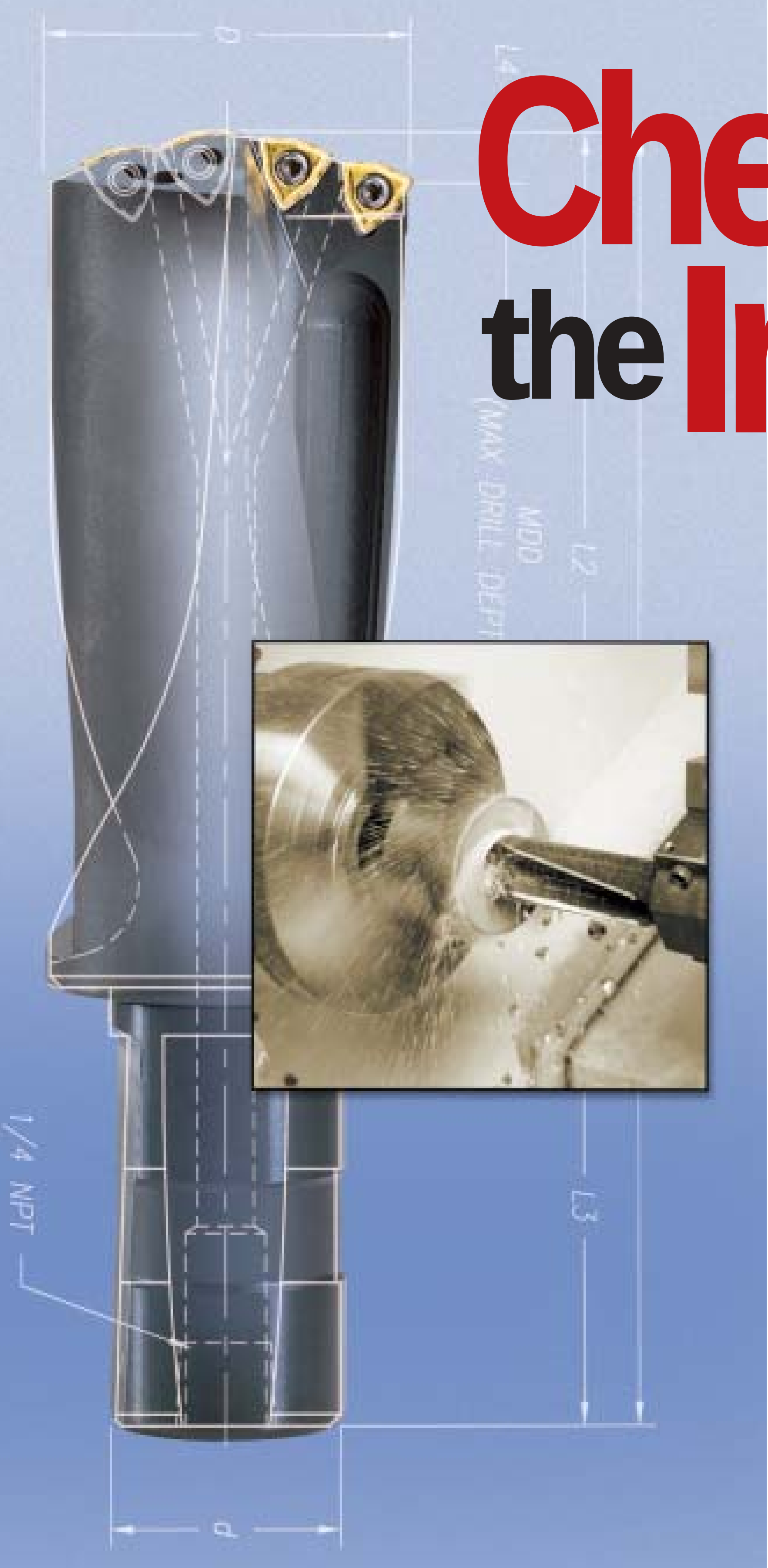
Indexable drills allow higher speed and feed rates.

Check the Index

▶ BY MICHAEL DEREN



Many a machinist has learned how to compensate for middling drilling processes. In fact, it's very much a way of life to some—a slick trick here, a secondary operation or two there.

However, in many cases, this kind of job shop sleight of hand is unnecessary. High-quality, streamlined holemaking may be just a phone call away. A supplier of indexable-insert drills can easily suggest alternatives to traditional drilling techniques that will result in better surface finishes and more productive operations. With a little schooling on the proper application of an indexable drill, any job shop can improve its capabilities and, possibly, open itself up to more business.

Where do indexable drills perform best? According to Greg Foreman, assistant manager for C&T Machining Solutions, Mercersburg, Pa., indexable drills perform equally well on machin-

Some indexable drills come with drill geometries that allow for aggressive performance in both lathe (inset) and mill applications with minimal horsepower consumption. Among OTM's line of indexable drills is its Holeshot drills (shown), Stinger smal-diameter drills and Coremaster core drills.

OTM

ing centers and turning centers. "Coolant capability is important. Rigidity is very important."

Matt Neuman, tool engineer, Manchester OTM, Wapakoneta, Ohio, added, "Indexable drills perform best when you require a better surface finish and want to run at a lot higher speeds and feeds."

Offset, On Target

There are several factors that influence performance. Inserts typically have molded chipbreakers, which were developed for high feeds. Advanced flute designs, which include ones with deeper gullets, faster or slower spirals and straight configurations, as well as wide flutes, ensure higher penetration rates by preventing chips from binding and degrading the hole finish. I prefer faster, spiral flutes due to their tendency to quickly remove chips.

The square- or rectangular-insert drill provides the ability to drill flat-bottom holes with effective chip control. Standard endmilling inserts, which have built-in chipbreakers and are material-specific in design, can be used in drilling applications. Square inserts provide four cutting edges, while rectangular ones provide two. (While there are double-sided inserts designed for milling operations, for the most part, there are no such inserts for drilling, as these are typically back-tapered and can only be used facing forward.)

The trigon-insert drill is probably the most popular style. Trigon inserts are stronger than triangular inserts and nearly as strong as square inserts. Like triangular inserts, most trigon inserts offer three cutting edges. The insert is generally a triangular shape angled out 10° to 24° at the midpoint of each length. This creates a very strong geometry. Trigon-insert drills can be operated at high metal-removal rates when drilling a wide variety of materials. Feed forces, designed to meet at the center of the tool, lead to low cutting pressures, low axial forces, longer tool life and a reduction of tool deflection and chatter, with the net result being an increase in hole accuracy.

An advantage that trigon, square and rectangular inserts have is that they can be offset to create different hole diameters. Usually, you can change the hole diameter by up to 0.040" on a milling machine and as much as 0.150" on a lathe. This avoids having to purchase another drill size when you need, for example, a 1⁄32" larger hole on a mill or 1⁄16" larger on a lathe. (Note: when you offset a drill, you should have a maximum feed rate of 0.006 ipr.)

Kyocera Ceratip

Drill adapters, such as this Magic Drill Sleeve from Kyocera Ceratip, hold indexable drills offset from center on machining centers. The markings on the adapters allow for precise radial adjustment.

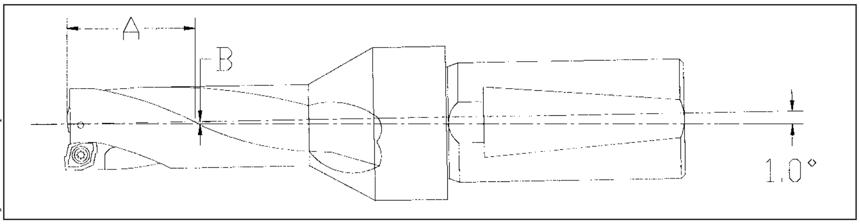
C&T Machining Solutions

A 1.5"-dia. indexable drill with high-pressure, through-spindle coolant.

To offset an indexable drill on a lathe, you offset the X-axis. As with any drill, select a holder with the least possible overhang and choose the shortest drill possible. Make sure the centerlines of the drill and spindle are aligned and parallel within 0.003" (the closer the better). With the top front of the outer insert facing the operator, make sure it is parallel to the X-axis. This allows for accurate offset adjustment for cutting. Drill a test hole about 1⁄4" deep. Measure the hole diameter. Then simply adjust the drill diameter with the tool-offset feature on the controller.

The use of an indexable drill on a lathe is also beneficial if trying to drill on an angled or uneven surface. If you have ever tried this with a conventinal drill, you'd know that you have to prepare the workpiece by counterboring a flat, followed by spot drilling. Only then are you finally able to drill. Indexable drills allow you to do this type of surface without the preliminary steps. Simply reduce your feed rate by half until you are totally engaged in the hole. Then, continue at full feed.

An important note for those of you considering using indexable drills in lathes: A slug is created when drilling through nonrotating parts. This slug can be thrown free from the chuck with



Kyocera Ceratip

With a 1° misalignment, the distance between the drill centerline and the spindle centerline increases linearly the farther one measures from the point at which the centerlines cross (in this case, at the tip of the tool).

tremendous force. Make sure all of the machine guards are in place.

When offsetting a drill on a machining center, a slightly different procedure is used. First of all, I recommend you purchase a small adapter for under $100, which holds the drill offset from center. You slide the drill into the adapter and slip both pieces into a toolholder. The ID of the adapter is slightly eccentric, and there are markings on the outside of the adapter to allow for precise radial adjustment.

Using the side-entry coolant port as a reference point, set the sleeve at zero. As on a lathe, drill a test hole about a ¼" deep. Measure the hole diameter. For a larger diameter, rotate the flange in the positive, or "+," direction. For a smaller diameter, rotate the sleeve in the negative, or "-," direction. Redrill a shallow test hole and measure. After each adjustment, tighten the setscrews on the toolholder. These adapters allow up to 1mm of offset. Best of all, one adapter handles several sizes of drills.

Higher Feeds and Speeds

Probably the biggest advantage of indexable drills is their ability to run at higher rates.

For example, in the case of applying a ¾"-dia. drill to aluminum, a conventional HSS drill run at 350 sfm with a 0.016-ipr feed would deliver 1,783 rpm and 28.5 ipm. A solid-carbide drill run at 550 sfm with a 0.012-ipr feed would produce application parameters of 2,801 rpm and 33.6 ipm. An indexable drill run at 1,200 sfm with a 0.005-ipr feed would have a spindle speed of 6,112 rpm and a 30.56-ipm table feed.

A process utilizing a ¾"-dia. drill in mild steel would find a conventional HSS drill run at 95 sfm with a 0.012-ipr feed would have a spindle speed of 484 rpm at 5.81 ipm. A solid-carbide drill run at 225 sfm with a 0.004-ipr feed delivers 1,146 rpm at 4.58 ipm. However, an indexable drill run at 950 sfm with a feed down to 0.003 ipr results in 4,839 rpm at 14.52 ipm.

Finally, a ¾"-dia. conventional HSS drill cutting stainless steel run at 55 sfm with a 0.008-ipr feed would produce 280 rpm at 2.24 ipm. A solid-carbide drill of the same diameter run at 200 sfm and a 0.004-ipr feed would translate into 1,019 rpm and 4.07 ipm. A ¾"-dia. indexable drill run at 560 sfm with a 0.003-ipr feed results in 2,852 rpm and 8.56 ipm.

These examples typify the definite productivity advantages indexable drills offer. (Note: the feeds listed for indexable drills in the mild steel and stainless steel examples are at the low end of a range applicable for the tools. The ipr can go as high as 0.007. Machinists should feel comfortable experimenting with the process and raising the feed incrementally.)

To calculate speed and feed, use these formulas:

rpm (spindle speed) = 3.82 × (sfm ÷ drill dia.)

ipm (table feed) = rpm × ipr

sfm (cutting speed) = 0.262 × drill dia. × rpm

ipr (feed) = fpt (feed per tooth) × effective teeth

If you were to try and drill a hole in ferrous materials with an HSS or solid-carbide drill at the feeds and speeds suitable for an indexable drill, you would either burn up or snap the drill. Another noteworthy factor: With indexable drills, you do not need to spot-drill the holes, because the drills are self-centering. Conventional HSS and carbide drills, due to their standard tip design, require a center or spot drill (1⁄16" or ⅛" deep) to prevent them from walking.

You will find that coolant requirements are more important for indexable drills. Most HSS and carbide drills use external coolant exclusively. In most cases with indexable drills, if you are drilling 1 diameter or less, external flood coolant is adequate. However, any depth over 1 diameter requires through-the-spindle coolant. In most cases, 70 psi suffices.

The deeper and larger the hole, though, the more coolant flow and pressure are required to help evacuate the chips. According to C&T Machining Solutions' Foreman, air can be sent through the tool as well. "You are strictly trying to evacuate the chips."

Whenever, Wherever

There will always be a need for conventional drills. However, if you re-examine your shop's applications, you might find a place for indexables. These drills are more expensive than conventional drills—maybe two to three times—but for certain jobs, they are well worth the investment.

About the Author

Michael Deren is a CNC applications engineer and a regular CTE contributor. You can email him at mderen@prexar.com.