FEBRUARY 2003 / VOLUME 55 / NUMBER 2

▶ BY ALAN RICHTER, MANAGING EDITOR

Down to Size

There is an alternative to threading small-diameter holes with taps—using a thread mill.

One of the primary misconceptions about threadmaking is that small-diameter holes—those smaller than ¼"—can't be thread-milled because there won't be enough clearance to feed into the thread being cut and remove the chips.

Most thread mill manufacturers carry standard products down to 8-32, said Michael Massey, operations manager for toolmaker Schmarje Tool Co., Milwaukee. (That translates as a theoretical hole size of 0.1503" when producing 65 percent of the full thread).

Other toolmakers don't stop there. Evans, Ga.-based Kennametal Greenfield IPG produces solid-carbide, multithread thread mills down to 2-64, which have cutter diameters of 0.060".

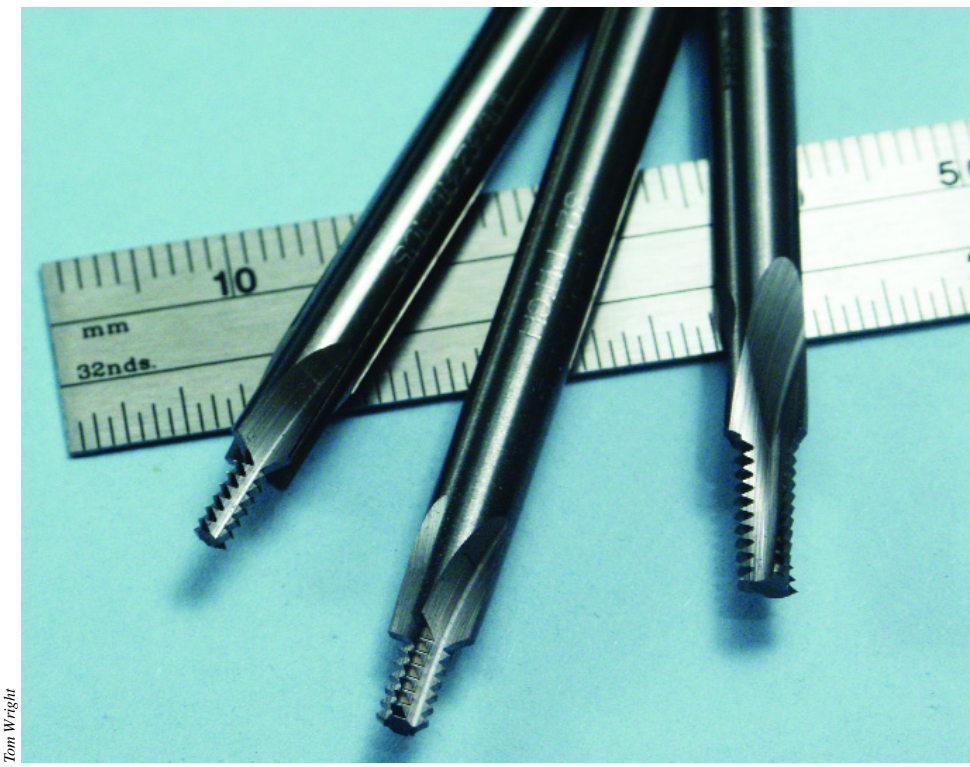
Cutters with a single thread and one flute are another style of thread mill. Jon Baker, president of A.B. Tools Inc., Lincoln, Calif., said his company stocks these solid-carbide thread mills down to a No. 3. "If you go any smaller, then there's too much flexing [for a 1-flute thread mill]."

A.B. Tool's No. 3 thread mills have a cutter diameter of 0.072", a neck diameter of 0.040", a 0.001" crest and a neck length of 0.160". The company can grind longer necks into its tools, "but we let the customers know that they're on their own as to performance and flex avoidance," Baker said.

He recommends a 1.5:1 to 2:1 depth-to-diameter ratio.

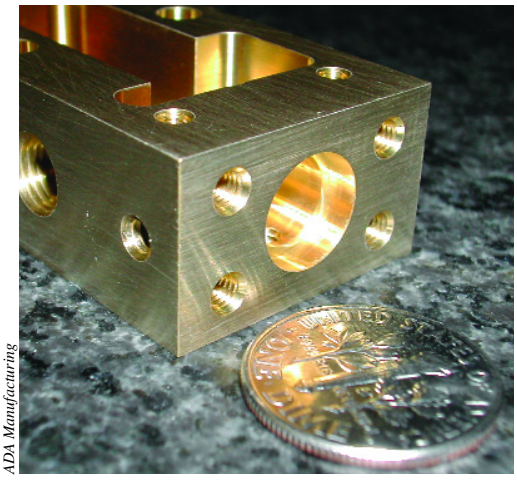
Baker added that the thread mills have a 60° included angle, a neutral rake and a primary relief of 12° to 15° for the smaller tools. This isn't to say adequate relief isn't critical for generating a thread in a hole where the thread's major diameter is as small as 0.090". "You need plenty of relief," Baker said. "It determines whether the thread mills work."

You also need a CNC machine tool that can move a thread mill simultaneously in the X, Y and Z axes, allowing it to helically interpolate. During helical interpolation, the thread mill moves in a circular motion on the X and Y axes



Tom Wright

Thread mill manufacturers have been producing smaller tool sizes, such as this selection from Advent Tool, for nearly a decade or more but haven't offered them as standards until more recently. The reason is because more of today's end users have machine tools with the capabilities to effectively thread-mill small-diameter holes.



ADA Manufacturing

Thread milling often offers the best solution for generating threads in cross-holes.

with a simultaneous Z-axis movement that produces the thread.

Thread Mills vs. Taps

Why would someone threading holes less than ¼" in diameter apply a thread mill rather than a tap?

Alan Baker, founder of A.B. Tools, said thread milling has six advantages over tapping. He explained that thread milling enables more accurate positioning of the threaded hole than does tapping. It offers the ability to thread blind holes within one thread of the hole's bottom, improves chip removal—since the tiny chips are cleared more efficiently—and eliminates bell-mouthed holes, since all threads are exactly the same from bottom to top. In addition, single-flute thread mills are able to mill special threads or metric threads. (Producing such threads with a tap would require a special tool.) Finally, thread mills generate stronger threads when hole depths are limited because, compared to tapping, more complete threads are possible.

Nicholas Korfias, national sales manager for Advent Tool & Mfg. Inc., Lake Bluff, Ill., said that being able to easily wash out of the hole the small "6s and 9s" that thread mills create is one of the operation's significant benefits. (Advent offers standard Unified-thread, solid-carbide, 4-flute, multithread cutters as small as 4-40.)

"In general, whether you're thread milling horizontally or vertically, those chips flood right out," Korfias said. "It's not like tapping, where there's this long, nasty, wiry, bird's nest-type chip."

He added that when thread milling with a multiflute cutter, only a portion of the tool's cutting surface is contacting a portion of the workpiece's ID rather than the hole's entire diameter at once. "With a tap, you're 'scoring' the walls all the way around as all the cutting surfaces are engaged at one time," Korfias said. "So that tool is undergoing a great deal of torsional force, whereas a thread mill cuts on only one side of the wall at a time."

And with the tap undergoing this "violent" torsional force, Korfias said, there's a greater chance of tool breakage, which clogs the hole. "When you break off a tap, you either have to EDM it out or you've got guys in the shop with vise grips, hammers and chisels trying to knock it out," he explained. "Or you scrap the part."

If a thread mill breaks, it comes out of the hole easily.

The Only Option

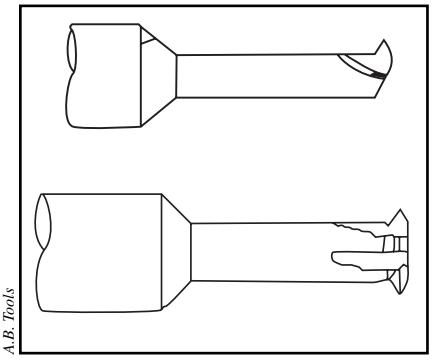
Sometimes, thread milling isn't an alternative to tapping but the only option. Tom Daniel, president of ADA Manufacturing Inc., Santa Clara, Calif., said ADA can only apply single-thread milling tools to thread some blind holes with depths up to ⅜" and tools as small as a No. 4. "We can't tap with a thin floor at the bottom of the hole because the floor will bulge up on the other side," he explained.

Thread milling, which generates a "cleaner" thread than tapping, is also the best solution for cutting threads in intersecting holes, even though the cycle time is longer, Daniel said. "The thread mill is cutting rather than pushing the thread, so there's no deburring or thread retapping required of previously tapped cross holes."

He added that thread milling also imparts a high-quality surface finish. "The finish is 100 percent better than when tapping."

However, Daniel emphasized that thread mills are not cost-effective unless there are cross threads in blind holes. "You don't want to thread-mill through-holes," he said. "It takes too long."

Cycle time also depends on the workpiece material. Daniel said ADA, which specializes in semiconductor and microwave parts, typically thread-mills aluminum, brass and 416 stainless



A.B. Tools

Single-thread milling cutters can generate any thread pitch.

The following companies contributed to this article:

A.B. Tools Inc.
(916) 408-2442
www.abtoolsinc.com

ADA Manufacturing Inc.
(408) 727-2585
www.adamanufacturing.com

Advent Tool & Mfg. Inc.
(800) 847-3234
www.advent-threadmill.com

Kennametal Greenfield IPG
(888) 434-4311
www.gfii.com/it

Schmarje Tool Co.
(800) 790-8665
www.schmarje.com

Threading 101 Inc.
(414) 425-4849

Material	Cutting Speed (sfm)	Feed (ipt)
Ferrous	150 to 300	0.0003 to 0.001
Nonferrous	500 to 1,000	0.0005 to 0.0015

A.B. Tools

Suggested started-point thread-milling parameters

steel. To create threads with 75 percent of the full thread in aluminum and brass, he said a minimum of two passes is required. Three to four passes are needed for thread milling holes in stainless steel.

In addition, Daniel noted that tool life is excellent for an uncoated thread mill, which is discarded when its tip is worn or chipped or the tool can't hold the required pitch.

Other Considerations

Even after deciding to thread-mill holes, there is still the choice of going with a single- or multiple-thread tool.

Alan Baker pointed out that a single-thread tool can make any thread pitch, because it is cutting only one thread at a time as the hole's entire thread is progressively formed. "It cuts the threads all the way up the hole," he said, "so a ¼"-deep hole requires ¼" of thread milling."

But, a multithread cutter is faster, since one helically interpolated revolution of the tool can produce the thread. The downside is that the multithread cutter is limited to cutting a single thread pitch. Nonetheless, Baker estimated that multithreaders have an 80 to 90 percent share of the thread-mill market.

Although it's not an option for the tiniest thread mills, multithread specials down to 8-32 can be ordered with a staggered- or skipped-tooth pattern (every other tooth is missing) or a design where each tooth form is followed by two missing teeth, said Schmarje's Massey. Such a pattern breaks the chip up more finely but also makes it easier to build up an accumulated pitch error, he said. Staggered-tooth specials cost 10 to 15 percent more, Massey added.

Regardless of the thread mill being applied, Massey said the machine tool's spindle needs to be in decent condition and able to run at 10,000 rpm or faster to create an adequate chip load per tooth when milling with small-diameter tools.

Al McBride, president of Threading 101 Inc., Muskego, Wis., noted, "All cutting is done according to the surface footage per minute, not revolutions per minute. So as the sfm goes up, the rpm requirement goes up as well."

But, too much feed might break the mill, Massey added, while not enough causes premature wear.

The thread mill needs to be held in a high-quality mill chuck, a shrink-fit toolholder or, if those aren't available, a double-contact collet. "Never use an endmill holder because the tolerance in the ID, shank and set screw builds up tool runout," Massey said. "The tool needs to run dead concentric, with almost zero TIR, to run properly."

Although standard thread mills are indeed available for threading hole diameters under ¼", Massey indicated that many shops are still running older machines that aren't capable of simultaneous 3-axis movement or don't have the spindle speed for applying small-diameter thread mills efficiently. He also acknowledged that tapping remains popular because it requires less programming and technical skill to perform. "The tap guys have a good foothold."

There are advantages to thread milling and lots of end users are sold on them, but application is everything and there are taps for nearly every threading application. "It's dog eat dog as tap makers improve their tools to recapture some of the market share lost to thread milling," said Alan Baker. "I don't see thread mills taking up more market share."

Tapping market share

Depending on the application, thread milling has its advantages, but one of them isn't providing the lowest cost per threaded hole, said Nobuaki Kamiya, vice president of marketing for OSG Tap & Die Inc., Glendale Heights, Ill.

He noted that a standard ⅜", spiral-flute, HSS tap costs less than $20, while a comparable thread mill, which is commonly solid carbide, costs $150 or more. In addition, thread milling requires time- and money-consuming CNC programming, whereas the tapping program is automatically generated by the CNC.

With the cost disadvantage, it's little wonder thread milling isn't as popular as tapping. Thread mills' slice of the pie for threading tools, in general, is less than 5 percent, Kamiya said, and they only have a 1 to 2 percent market share for threading holes under ⅜" in diameter. "Thread milling is a niche market," he said, "but it's growing a little bit as programming is simplified."

Kamiya emphasized that the workpiece material represents the biggest consideration when deciding which type of tool to apply, and thread mills can be more effective for threading holes in aerospace-grade materials, for example. "It's a case-by-case decision," he said.

For more information about OSG's taps, drills, endmills and dies, call (630) 790-1400, or visit www.osgtool.com.

—A. Richter