DECEMBER 2007 / VOLUME 59 / NUMBER 12

➤ BY ALAN RICHTER, EDITOR

Long in the Thread

New technology is overcoming the challenges of thread milling long, internal threads.

Vardex

Designers frequently overengineer parts to provide a safety cushion, and that includes threaded-hole specifications. One example is a bolt hole. The thread engagement length to achieve the required clamping force for a bolt is about a 1:1 to 1.5:1 length-to-diameter ratio, but the thread length is typically longer. "You can argue up and down that you don't need it, but this is the United States, so we overengineer everything," said Ross Wegryn-Jones, international sales manager for Advent Tool and Manufacturing Inc., Lake Bluff, Ill.

Nonetheless, part specifications call for the vast majority of internal threads to have a 2:1 or longer length-to-diameter ratio. Mark Hatch, senior product manager for Emuge Corp., West Boylston, Mass., estimated that only about 10 percent of internal threading applications require a shorter length. "That is why there has been, for a long period of time, a limited use of thread mills because of their limitations in the length of cut," he said.

That situation is changing. Newer generations of thread mills combined with machine tools able to perform helical interpolation—a basic requirement for thread milling—enable thread milling longer internal threads. To perform helical interpolation, a machine must be capable of simultaneous movement in three axes. Two of the axes perform circular interpolation, while the third axis moves perpendicular to the circular plane.

Deflection Avoidance

"Anything with a length of thread that's two or more times the minor hole diameter is where it starts to get fun," Wegryn-Jones said. That's when side-load pressure becomes a factor. "With thread milling, horsepower is no longer a factor. Torque is no longer a factor,” he noted. “It’s all about the forces against the tool and the workpiece working against each other to create vibration, chatter and general chaos at the tool/workpiece interface. Deflection is the symptom of the problem, but high side-load pressure is the root cause.”

Side-load pressure is particularly problematic when generating long, coarse threads with a thread mill, such as a 7- or 8-pitch thread. That’s because there’s a greater distance between the minor and major diameters and therefore more workpiece material needs to be removed. The upside is coarse threads are typically found on larger diameter holes, so a larger, more rigid tool can be applied.

To minimize side-load pressure with the aforementioned conditions, Wegryn-Jones said Advent provides two options. The first is a smaller, shorter insert. “Rather than taking a full frontal assault and trying to mill all of a thread form in one shot in these applications, we’ll take a piece of it,” he said. That might involve applying a 1"-long insert in two passes to make a 2"-long thread instead of attempting to produce the entire thread in one pass.

Advent Tool and Manufacturing

One option to minimize side-load pressure when thread milling long threads is to apply a skip-tooth form, such as one where every other tooth is missing.

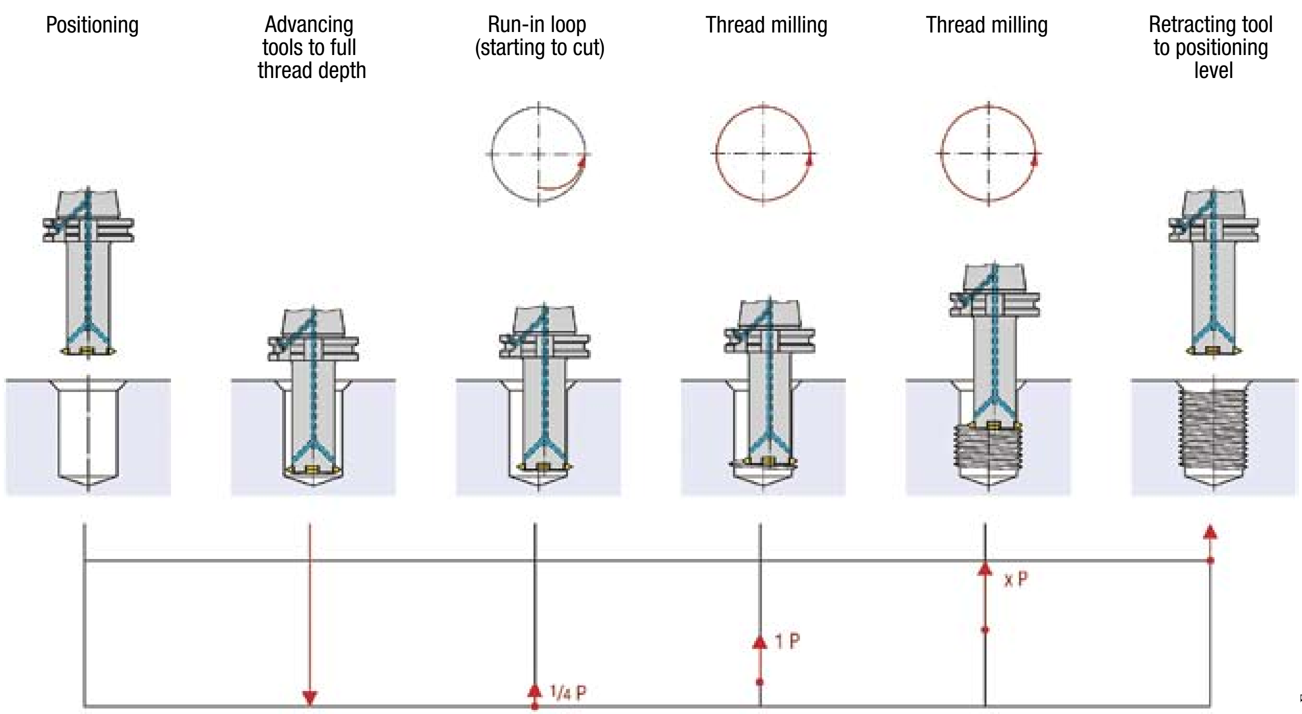
The second option is to apply an insert with a skip-tooth form where every second tooth or every second and third tooth are missing, mathematically reducing the side-load pressure by half or 66 percent, respectively. “The tool behaves more predictably because it’s not trying to do everything at once,” Wegryn-Jones said, adding that two or three passes are required to blend the threads and generate a complete form. “That’s not to mention a rough and a finish or spring pass,” he noted.

Of course, the additional passes make for a longer cycle time. “Your cycle time is going to go out the window,” Wegryn-Jones said. “Consumers need to know that up front, and they understand it.” He joked that the end user should slap the part designer for requiring such a thread, but because the designer is more often than not the customer, that’s probably not wise.

Vardex USA, Janesville, Wis., recently introduced another insert-type thread mill. Called the TMSD line, the insert is mounted at an angle in a hardened steel body. The tool is available in three styles. One features a cylindrical shank with the same dimension for its entire length, the second has a Weldon adaptor and the third is a shell mill design.

“This tool has extended reach capabilities,” said Linda Wilken, the toolmaker’s technical application engineer. “You can cut it to length to make your setup more rigid if need be or use the entire length.”

She said the cylindrical-shank style is able to generate threads with up to a 7:1 length-to-diameter ratio, while the Weldon-flat style is able to produce threads with up to a 5:1 length-to-diameter ratio. “It’s multieffective, so you may have three, four, five or six effective teeth, but having a single insert dramatically decreases tool pressure,” Wilken said. “Because you have the multiple stations, or effective teeth, you’re able to feed the tool faster than you would a solid-carbide thread mill where you have a tremendous amount of tool pressure when you engage 10, 12 or 14 teeth versus one tooth in a multiple station.”



Emuge

The sequence for conventional thread milling of a blind-hole.

Tool pressure is reduced by having less mass intersecting the workpiece at one time, she noted, explaining that 10 teeth intersecting the workpiece causes 10 times more deflection compared to one tooth entering the same workpiece. “The cutting edge of our single insert also has a positive rake edge and a chipbreaker,” Wilken said. “This enhances chip curling and breakage, creating a higher shear action when entering a workpiece.”

Wilken added that the tool can produce the desired thread in a single or double pass, depending on the material. In a softer material, it’s desirable to generate a thread in a single pass at full DOC, whereas a difficult-to-machine material generally requires two passes. “You break it into maybe a 70/30 split,” she said, “taking the majority of the thread initially and then taking the balance in the second pass.”

Solid Tooling

Andrew Carr, product manager for Allied Machine & Engineering Corp., Dover, Ohio, said AMEC doesn’t currently offer insert-type thread mills, but plans to offer an insertable-style thread mill in the first quarter of 2008. The majority of AMEC’s solid-carbide thread mills have two to five flutes,

Helically fluted, solid-carbide thread mills, such as these from Allied Machine & Engineering Corp., are one tool option for generating internal threads.

The following companies contributed to this report:

which help stabilize the cutting action, according to Carr. “More flutes equal more cutting edges that you have engaged with the material,” he said as an example. “The more cutting edges you have engaged with the material, the more stable the tool becomes. As stability increases, you reduce chatter and side-load pressure because you have a more balanced force against the workpiece.” In addition to stabilizing the cutting tool, multiple helical flutes enable more aggressive machining parameters, Carr noted.

He added that AMEC offers a standard helical-fluted thread mill with a thicker core for producing threads in a single pass with an approximate 2.5:1 length-to-diameter ratio and will make specials for longer lengths.

Emuge also makes solid-carbide thread mills with a thicker core for enhanced rigidity, which results in an enlarged cutter diameter. Hatch added that this new generation of thread milling technology makes obsolete the old rule-of-thumb that the cutter diameter should not exceed two-thirds of the major thread diameter being produced to avoid having a deviated flank angle error. This means true-to-gauge threads can now be produced with enlarged cutter diameters because of the development of advanced tool-grinding techniques.

According to Hatch, in addition to the thicker core, Emuge is able produce tools with long milling sections that have multiple teeth "through the science of precise relief geometries, rake angle and flute shape." He said this enables an end user to make a full thread length in one rotation. "Customers said 'Yeah, I like the benefits of thread milling; however I can't live with the long cycle time,'" Hatch said. "Now we have a long milling section without any interruption of the cutting teeth."

Concerning flute shape, Hatch added that the tool needs an equal, symmetrical division of the flutes to prevent harmonic excitation and avoid subjecting the cutting edges to uneven cutting pressures as they move in and out of the cut similar to an endmill.

Emuge

A thread milling system from Emuge features partial-profile technology to generate thread lengths with up to a 3:1 length-to-diameter ratio.

Easing into the Cut

Entering the cut requires care to avoid overloading the cutting edge and shortening tool life. Therefore, a tangential arc approach is the proper course. "Our basic recommendations are to helically ramp into the thread, do a 360° cut and ramp off," said Joe Mazzenga, president of toolmaker JBO-USA, Troy, Mich., representatives for JBO (Johs. Boss GmbH & Co. KG) of Germany. "That ramp is formed by the radius from the tool's OD out to the major diameter of the thread."

He added that instead of climb milling, where the tool starts at the bottom of the hole and mills up the thread, JBO recommends conventional thread milling down the thread. "Conventional milling provides a straighter, or less-tapered, thread," he said.

Vardex's Wilken concurred that the tangential arc approach is preferred over cutting straight into the side wall, adding that the feed rate should be reduced by 20 to 30 percent during tool entry. "That does dramatically increase tool life by about 20 to 30 percent," she said, "and as the tool exits the cut, it does the same thing. It creates a perfect blend."

For a coarse thread, Advent Tool's Wegryn-Jones said the usual default value for ramping into the cut is from 45° to 90°, but he suggests 180° or more. "That's 180° from the time the tool touches the material until it gets into the programmed DOC," he noted. "A major reason for this is carbide tools like to be pushed hard, but they do not like surprises or shocks."

To generate a program's code, thread mill manufacturers offer software, typically available free of charge on their Web sites, to generate a thread milling subroutine that includes speeds and feeds based on the workpiece material, the thread mill, the thread specifications and the hole's dimensions. "The software steps you through, helps you create CNC program data and generates the code," Wilken said. That code can then be cut and pasted or dragged and dropped into the overall CNC program.

Emuge's Hatch added that canned software programs include modules to

Keywords

thread:
Portion of a screw thread encompassed by one pitch. On a single-start thread, it is equal to one turn.

thread form:
Defines the basic thread shape (V, square, buttress, etc.) and specification of radii or flats at the crest or root of the thread.

threading:
Process of cutting, turning and rolling of internal or external threads into a workpiece material. Standardized specifications are available to determine the desired results of the threading process. Numerous thread-series designations are written for specific applications. Threading often is performed on a lathe. Specifications such as thread height determine the threads' strength. The material used is taken into consideration in determining the expected results of any particular application for that threaded piece. To perform internal threading, the exact diameter to bore the hole is critical before threading. The threads are distinguished from one another by the amount of tolerance or allowance that is specified.

—CTE Metalworking Glossary

enable end users to write CNC code for thread milling and that newer controllers on CNC machines have canned cycles for thread milling, similar to those for tapping and drilling, for example. The canned cycles are for threading only-type tools, and modifications are needed when performing a multifunction operation that might also include drilling and chamfering.

Raking It

The primary tool geometry on a thread mill besides the thread form is the rake angle, which varies from a neutral, or zero, angle to a positive angle. AMEC's Carr explained that a neutral rake is appropriate for materials that form chips easily, such as cast iron, and a positive rake is for gummier workpieces, such as stainless steel. "The neutral is like an all-purpose tool," he said, adding that a positive rake helps shear more elastic materials while providing stability.

Advent offers thread mills with positive rake geometries, but Wegryn-Jones said that design can be less than optimal unless the entire process and workpiece material are known. "The positive rake needs to cut under the material and in some materials the positive geometry has a tendency to grab and dive into the material," he said. The positive geometry is also not as tough as a neutral one and therefore more prone to chipping in some workpiece materials.

In addition, the rake angle can be variable or constant, according to Hatch. He noted that toolmakers produce a variable rake, which is more like a hook, using a flat wheel while a constant rake is created with a form wheel, which takes more time. "The advantage of a constant rake is it provides a stronger cutting edge and better chip formation so the tool becomes less susceptible to chipping and breakage," Hatch said. He added that there are no practical benefits to a variable rake on a thread mill because that type of geometry would only be suitable for soft, nonferrous materials, such as aluminum, and those materials are not commonly thread milled.

A standard indexable tool from Advent Tool with a skip-tooth insert for thread milling 1"-8 UNC threads 2.125" deep.

Advent Tool and Manufacturing

Evacuating Chips

Once chips are created, they must be efficiently evacuated from the hole. Fortunately, the comma-shaped chips a thread mill creates are generally small and easily managed. Because the thread mill's cutting diameter is generally two-thirds or less of the minor thread diameter, there's plenty of room in the workpiece for chip evacuation.

"Flood coolant does a wonderful job of getting the chips out," said Wegryn-Jones. "If it's through-the-tool coolant, so much the better, but between the two, I don't see a huge amount of difference as far as what you can achieve in tool life in most applications."

Hatch noted that Emuge's thread mills come standard with through-coolant capability, which is helpful for blind-hole applications. "With an internal-coolant supply in a blind-hole," he said, "we are assured of complete and efficient evacuation of the flute area, so we don't get any recutting of chips."

Gravity may aid chip evacuation in vertical through-holes, but redirecting the coolant in a thread mill designed to have coolant travel through the tool's center can also aid chip evacuation. Mazzenga explained that for those applications, JBO-USA can plug the tool's end face and place exit holes in the flutes so coolant travels radially from the tool or grind grooves along the tool's shank to allow coolant to flow down to the cutting edge. "There is no one way to address that problem," he said. "We use coolant extensively to negate the concern about chip buildup in the flutes."

Besides properly directed coolant, a rigid machine tool helps when thread milling long threads. In addition, a stable toolholder helps ensure success when threads have a 2:1 or greater length-to-diameter ratio. Mazzenga rates shrink-fit holders as the No. 1 option, followed by milling chucks, ER collet chucks and endmill holders, but recommends against using hydraulic chucks. "They're great for drilling and other axial work," he said, "but when it comes to side loads, or radial forces, there's a little sponginess because a hydraulic chuck has an inner membrane that clamps the tool shank and an outer sleeve with oil in between." △

Thread milling is popular for medical applications, such as this stainless steel orthopedic implant.

JBO-USA

There's more than one way to skin a cat, er, make a thread

Thread milling is often considered a higher-quality alternative to tapping for internal threadmaking applications. That's because a thread mill—unlike a tap—can generate a full thread form in all machinable materials and produce a thread form within one thread pitch of the bottom of a blind-hole. Full-bottom threading is problematic when tapping because a tap is designed with a chamfer of two or three threads in length. "If you make the chamfer too short, it makes for a very poor cutting tool," said Mark Hatch, senior product manager for Emuge Corp.

In addition, one thread mill can produce all the different pitch diameters that may be required for a part. By making a CNC programming change, it can cut from the bottom up or the top down, and if a thread mill breaks during cutting it can be easily removed from the hole whereas a broken tap is stuck. A stuck tap requires machining it out or scrapping the part—an expensive prospect. "Generally, with threading being the last operation, you could lose your shirt if you're scrapping parts," Hatch said.

That doesn't mean taps don't have their place in the metalcutting world. "Something we run into time and time again is someone taking a process that had traditionally been tapped and deciding to thread mill it when it wasn't set up to be thread milled," said Ross Wegryn-Jones, international sales manager for Advent Tool and Manufacturing Inc. "Sideload pressure wasn't brought into the equation or the thread mill's holder is a small collet that is just perfect for a tap but absolutely bloody useless for a thread mill."

Wegryn-Jones added that a toolmaker must spend significant time to understand an end user's process, equipment and goals to optimize a thread milling process.

Also, depending on the thread specification and the raw material cost, tapping can be more economical. "If you're looking for a rough cut thread blown into a $5 part, tapping is faster," said Andrew Carr, product manager for Allied Machine & Engineering Corp. "If you're looking at a $10,000 Inconel investment casting and you snap the tap off, the entire part can become scrap. This is a primary example of when thread milling is more advantageous than tapping. Both style tools have their place in the market, it just depends on what is needed for a customer's application."

Hatch concurred that "thread milling is not the answer for all," noting that Emuge sells both types of tools. Sometimes a part will have a set of holes where tapping is appropriate and others that should be thread milled. "It will be a combination of both technologies that ultimately brings you the lowest manufacturing cost," he said.

Nevertheless, thread milling can provide significant savings in the right application. Hatch said one customer was tapping 200 holes in an Inconel subsea instrumentation component and consumed 40 taps per part. Switching to thread milling cut the cycle time in half, enabled one tool to thread all the holes and saved about $6,800.

—A. Richter