DECEMBER 2008 / VOLUME 60 / ISSUE 12

Turn Right

cover story

The practice of turning threads on a lathe is well known but often not well-understood. Toolmakers offer their recommendations on how to do it the right way.

By Bill Kennedy, Contributing Editor

There are nearly as many ways to produce threads as there are types of parts that require them. Threads can be cut with a die, rolled, ground, tapped or milled, using lathes, machining centers or specialized machines.

The first method and still among the most common ways to produce threads is to turn them on a lathe. While new tooling technologies can help boost thread turning productivity, toolmakers say that close attention to the application requirements of thread-turning operations is just as important for success.

From a simply mechanical point of view, turning a thread involves matching the cutting tool's feed rate with the desired thread pitch. For example, turning a thread with a pitch of 8 threads per inch (tpi) requires the tool to feed 0.125" along the workpiece each time it revolves.

Optimizing thread turning operations can increase throughput and improve part quality. Although cutting tool makers continually introduce new tool materials and geometries that enable users to boost productivity via more aggressive cutting speeds, feeds and DOCs, the specialized nature of thread turning limits possible changes to cutting parameters. The 0.125-ipr feed rate required for an 8-tpi thread is significantly higher than the 0.010" to 0.015" feed of typical turning operations. In addition, the nose angle of a threading tool is narrow (generally 60° or less), and the nose radius may be 0.002" to 0.004" compared to a typical turning insert's minimum nose radius of 0.015". And excessively high cutting speeds generate sufficient heat to deform the relatively delicate threading tool tip. As a result, thread turning usually involves making multiple passes at moderate speeds and DOCs.

Simply maximizing the number of passes, however, can be counterproductive. Frank Battaglia, staff engineer at Kennametal Inc., Latrobe, Pa., noted that threading usually is one of the last

Learn more about thread turning

Get more information on thread turning by viewing an Interactive Report on www.ctemag.com (on the home page and also accessible through the "New and Events" drop-down menu). Also visit Bill Kennedy's blog in the CTE Community section, also at www.ctemag.com.

For optimal performance, laydown-style threading inserts must be mounted in the holder on a specific shim or anvil (gray component in assembled tool) that tilts the insert to match the thread's helix angle. The angle equalizes flank clearance, minimizing rubbing of the insert flanks and reducing the negative lead of the tool into the cut.

operations performed on a part that may already represent a significant investment in machining time. Accordingly, many shops adopt a conservative approach, believing a large number of passes is the safest solution. In reality, dividing the thread depth into a large number of passes results in DOCs that are too light, which decreases the insert's ability to cut effectively. Also, making more than the recommended number of passes increases tool wear because the cutting edge spends excessive time in contact with the workpiece. Chip control, always a concern when cutting threads, may suffer because a thinner chip can be more difficult to break than a thicker one. Finally, every unnecessary pass represents an increase in cycle time.

While excessive tool wear is a sign that too many passes are being taken, chipping of the insert may indicate tool overload and require that the number of passes should be increased.

Apportioning DOC

Even when an appropriate number of passes is employed in a thread-turning operation, the way the total thread depth is divided among those passes remains critical. Two approaches to apportioning DOC are constant-depth infeed, in which the tool takes the same DOC on every pass, and constant-volume infeed, where the DOC decreases on every succeeding pass in an effort to maintain a consistent load on the tool.

Constant infeed is easy to program; DOC for each pass can be determined by dividing the thread height (difference between major and minor diameters) by the number of passes. The main disadvantage of the constant-infeed approach is that the load on the insert increases with each pass deeper into the thread form because the tool removes successively greater volumes of material. Tool life and part quality suffer.

In the constant-volume approach, each pass is calculated to remove the same volume of material. As the thread deepens, the tool takes a smaller DOC on each subsequent pass. The method is not without special considerations. Removing the same volume of material on the first cut as in later passes may overload the tool tip on that first pass. To minimize chances of overload, manu-

Consistent results with constant-volume thread turning

IN THE CONSTANT-VOLUME thread turning approach, each pass is calculated to remove the same volume of material. The object is to maintain a constant load on the cutting tool. The formula to calculate constant volume states that the accumulated depth after each pass should equal the initial DOC times the square root of the number of the pass.

For example, an 8-pitch external thread has a depth of 0.0789". The first pass should be 25 percent of 0.0789" for a depth of 0.0197". Via the formula, the accumulated depth after the second pass should be 0.0197" times the square root of 2 (second pass), or 0.0279". Subtracting the depth of the first pass gives a 0.0082" as the depth required to reach a depth of 0.0279". The process can then be continued for subsequent passes.

Although this approach provides a constant chip volume, in practice manufacturers advise reducing the first pass by 20 percent (to 0.0158" in this example) and adding the reduced amount to the second pass (0.0039" + 0.0082" = 0.0121"). This effectively reduces the chip load in the first pass, avoiding overloading of the cutting edge.

—B. Kennedy

---

facturers recommend reducing the DOC on the first pass and adding the amount of the reduction to the second pass (see sidebar on this page).

Workpiece material considerations also may prompt some tweaking of the constant-volume approach. According to Kennametal's recommendations, the DOC on the final pass should be at least 0.002" to avoid workhardening and excessive abrasion of the cutting tool, especially with workhardening-prone materials such as stainless steel.

B. Kennedy

Either top-clamp (left) or laydown-style inserts can cut threads effectively. In general, laydown inserts work well in producing fine-pitch and high-helix threads, while top clamp styles offer strength and rigidity that can facilitate production of coarse-pitch threads. Top-clamp styles also are easy to modify to produce custom thread forms.

Infeed Angle

The angle at which the insert enters the workpiece also plays a role in tool life and productivity. Duane Drape, national sales manager for Horn USA, Franklin, Tenn., said radial infeed, in which the cutting edge enters the workpiece on a perpendicular path, is "the most common method, the easiest method to program and is going to result in the absolute worst tool life and the worst-looking threads." The insert cuts on its nose and both flanks "all at the same time on each pass. You are wearing the insert substantially more because you have more surface area in the cut," he said. Chip control can also be a problem, because radial infeed creates a V-shaped, undirected chip.

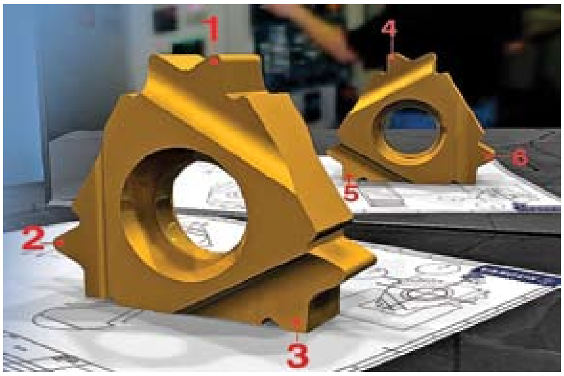
Chip flow is more directional and tool life is longer with the flank-infeed approach, which involves feeding the insert into the workpiece at an angle matching the thread's flank angle, so only one side of the tool does the cutting. Drape said flank infeed is a step in the right direction, but the tool flank rubs on the cut surface during each pass because it is directly aligned with the thread form. "You are wearing out the insert because the flank is not cutting, it is just rubbing," he said.

Insert flank rubbing can be minimized through a "modified flank" approach, in which the insert leads into the workpiece with at least 1° of clearance on the side of the tool. "You still get the same thread profile because the insert is actually coming down along that same 60°

or whatever the thread flank angle is, but you are not rubbing along that entire cutting edge." The modified-flank approach, Drape said, imparts a finer surface finish and extends tool life, and is not difficult to program.

The best way to balance tool wear is the alternating-infeed method, in which the insert makes cuts on each side of the thread form in turn. Drape said alternating infeed is, however, "the hardest to program." For most cases, the modified-flank approach offers the best combination of programming ease and tool life.

Jeff Major, national sales and marketing manager at Vardex USA, Janesville, Wis., pointed out that turning a thread is not like turning a diameter. "It's precise. You have to know the proper depth and the proper spindle speed to get the process to work." To help a shop optimize threading, he said, Vardex offers its TT Gen thread-turning tool selection software. A user can enter the desired thread pitch, length and other variables, and the software provides solutions. "It may give you two solutions; it may give you 25," Major said. "Then it gives you the number of passes, depth per pass, speeds and feeds. In 30 seconds, you can know all your parameters and have a recommended tool."



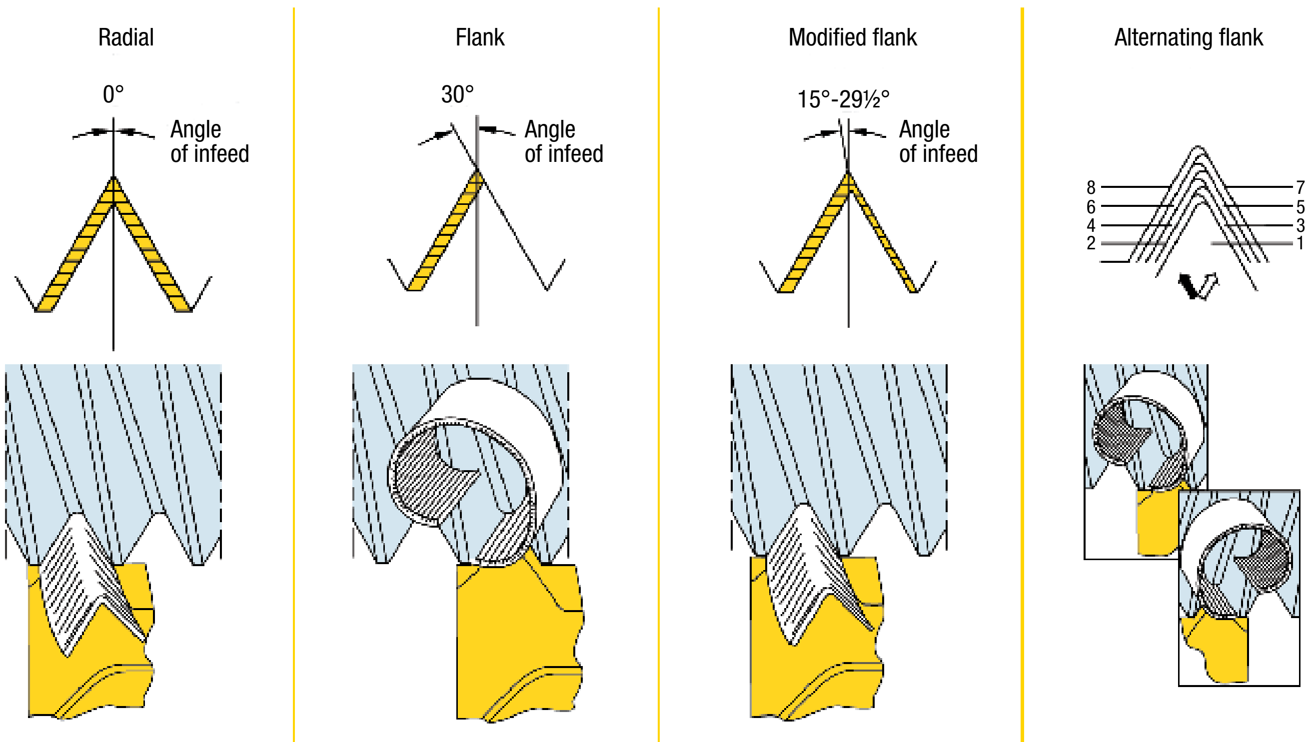
Vardex

Laydown threading inserts typically have three cutting edges; turning some inserts over presents three more edges, but they produce threads of the opposite hand compared to the first side. Recently, Vardex introduced a laydown threading insert that provides edges on both sides of the insert, for a total of six. A specific anvil supplied with the inserts enable them to be used in standard right-hand toolholders to cut right-hand threads with all six edges.

Laydown or On-Edge

Threading inserts can generally be categorized as laydown or on-edge styles. Laydown tools look like typical triangular turning inserts but feature three cutting points engineered to turn threads. Laydown inserts are held flat (wide dimension horizontal) in the holder via a vertical screw or a top clamp. On-edge inserts are available in a variety of configurations, including triangular, four-sided, double-ended, and bar-shaped "dogbone" styles. On-edge inserts are held in the holder via a horizontal screw or, for double-ended and dogbone styles, a top clamp.

Will Wright, Kennametal's global product manager for threading, grooving and cutoff, said laydown and on-edge styles offer their own performance advantages, but his field sales experience has shown him that many shops "tend to prefer just one style or the other. This can be because they've always used a particular style in their shop, or it may be that it's just what they were trained to use." Either style can cut threads effectively, he said, noting that Kennametal offers both LT laydown and Top Notch double-ended threading tools.



Kennametal

A threading insert's infeed angle plays a large role in tool life and productivity. Radial infeed is simple to program but hard on the tool; flank infeed is better but still subjects the tool flank to rubbing; modified flank provides perhaps the best compromise of programming ease and tool wear; and finally, alternating-flank infeed offers optimal tool wear but can present a programming challenge.

In general, laydown inserts work well in producing fine-pitch and high-helix threads, while top-clamp styles offer strength and rigidity that can facilitate threading coarse pitches. Top-clamp styles also are easy to modify to produce custom thread forms. Wright said producing such "custom solutions" is becoming an important part of toolmakers' efforts in the area of thread turning.

For optimal performance, laydown inserts must be mounted in the holder on a shim or anvil that tilts the insert to match the thread's helix angle. The angle equalizes flank clearance, minimizing rubbing of the insert flanks and reducing the tool's negative lead into the cut. Standard shims provide an inclination of about 1.5°. Shim angles vary in relation to the thread pitch and the workpiece diameter; manufacturers provide charts and formulas to determine the appropriate shim for a specific thread.

Laydown inserts typically have three cutting edges. Flipping some inserts presents three more edges that can produce threads of the opposite hand, compared to the first side. Recently, Vardex introduced a laydown threading insert that provides edges on both sides of the insert, for a total of six. A specific Vardex anvil supplied with the inserts enables them to be used in standard right-hand toolholders to cut right-hand threads with all six edges. The cutting edges on either side of the insert are offset to prevent them being damaged by chip flow.

Horn USA's Drape pointed out that on-edge inserts have the helix and reliefs built into the insert itself, making shims unnecessary. He added that Horn USA's on-edge threading inserts fit the company's grooving holders. "A thread is a groove; it just happens to be a spiral groove," he said. The grooving configuration provides an advantage in that "if you have to turn a thread between shoulders, you may not be able to complete it with a laydown system," Drape said. The configuration of on-edge threading inserts in grooving holders provides clearance that permits turning the threads between shoulders.

Partial or Full

In an effort to suit machine shops' differing production-volume requirements, toolmakers provide threading inserts in partial and full-profile configurations.

A full-profile insert is designed to produce a completed version of a specific-pitch thread. When the edge cutting the thread profile reaches the desired

contributors

Horn USA Inc.
(888) 818-HORN
www.hornusa.com

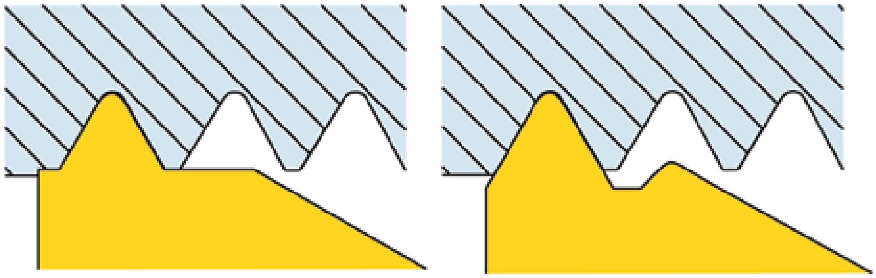
Kennametal Inc.
(800) 446-7738
www.kennametal.com

Vardex USA
(800) 828-8765
www.vardexusa.com

Walter USA Inc.
(800) 945-5554
www.walter-tools.com

depth (the minor diameter), a secondary edge cuts the outer, or major, diameter of the thread form. Partial-profile inserts, on the other hand, lack the secondary edge and leave the major diameter unfinished.

Patrick Nehls, product manager for indexable programs, Walter USA Inc., Waukesha, Wis., said a full-profile tool controls thread depth, “very accurately. It gives you a precise, clean thread,” he said, that requires no secondary finishing operations. He added that full-profile inserts are most beneficial in high-production applications where the same thread pitch is machined repeatedly.



Kennametal

A full-profile threading insert (left) is designed to produce a finished version of a specific pitch thread. When the edge cutting the thread profile reaches the desired depth (the minor diameter), a secondary edge cuts the outer, or major, diameter of the thread form. Partial profile inserts, on the other hand, lack the secondary edge and leave the major diameter unfinished. Choice usually depends largely on the volume in which a shop will be producing a particular thread.

On the other hand, partial-profile inserts provide the flexibility a shop may require to machine a variety of thread pitches with the same insert. Partial-profile inserts can be easily modified to produce nonstandard thread forms. The tradeoff for increased flexibility may be a need to perform a finishing or deburring operation on the major diameter of the thread. In addition, tool life may be shorter in some cases because the nose radius of the partial-profile insert is engineered to produce a range of thread pitches and may not be optimal for a particular thread profile.

Multitooth Productivity

Multiple-tooth inserts represent another tool configuration aimed at boosting throughput in production thread turning. Multitooth tools feature a series of cutting edges or teeth that increase in size. The initial teeth progressively rough and semifinish the thread form, while the last tooth finishes the thread to final dimensions.

Vardex’s Major said multitooth inserts enable a shop to reduce the number of passes required to complete a thread. “With a single point it may take you 12 to 16 passes to complete the thread; with a multitooth you can do it in a maximum of four,” he said.

Michael Trimble, Vardex cutting tool engineer, cited a case where application of the company’s threading inserts overcame setup shortcomings and boosted productivity. A shop was using a single-point on-edge threading insert to create two internal threads on a cast iron part. The setup was unsatisfactory in regard to tool life and produced chatter and poor surface finish. First, Vardex converted the operation to a single-point laydown insert, which resolved the chatter and finish issues. Later, a move to MultiPlus multitooth threading inserts permitted reducing the number of passes required for the operation from 10 to four, while maintaining good surface finish and increasing tool life by 75 percent.

Despite providing clear productivity advantages in many cases, it should be noted that multitooth tools produce higher cutting forces, which can pose a problem when dealing with less-rigid parts and setups and lower power machines. The tool's design precludes machining to a shoulder, and its extra length may be a disadvantage where runout room at the end of the part is limited. When applying multitooth threading inserts it is important to follow DOC recommendations so each cutting edge works as designed. Excessive DOC will wear or break the first tooth, while too light a cut will minimize the effect of the first tooth, requiring more passes and negating the benefits of the multitooth tool.

Cutting Edge

Kennametal's Battaglia said optimal performance in thread turning results from the application of inserts that feature PVD coatings and small, consistent edge preparation (less than 0.001"), which minimizes cutting forces. Threading insert geometries are for the most part positive, although neutral/chip control styles may be appropriate for some applications in steel and cast iron. Tool substrates need to be tough to handle the lower cutting speeds (perhaps 25 percent lower than routine turning) and high feed rates characteristic of threading. However, cutting speeds must be high enough to prevent built-up edge. As cutting speeds climb, a tool's ability to withstand heat and avoid deformation become more important.

No matter the tool materials or geometries, turning threads successfully requires a continual balancing of cutting parameters within the restrictions imposed by the nature of the threading-turning operations. CTE

www.ctemag.com

For more information, visit "Articles Archive Index" and select the "Threading" category.

About the Author: *Bill Kennedy, based in Latrobe, Pa., is contributing editor for* Cutting Tool Engineering. *He has an extensive background as a technical writer. Contact him at (724) 537-6182 or at billk@jwr.com.*