AUGUST 2001 / VOLUME 53 / NUMBER 8

cover story

Carboloy Inc.

BY MIKE RESTALL

The Ins and Outs of Thread Milling

Advantages of and operating guidelines for thread mills.

Thread milling is the process of producing internal or external threads on a stationary workpiece with a specialized cutter. Holes smaller than ¼" in diameter can be thread-milled with a solid cutter. Insert-type thread mills are practical for ¾"-dia. and larger holes.

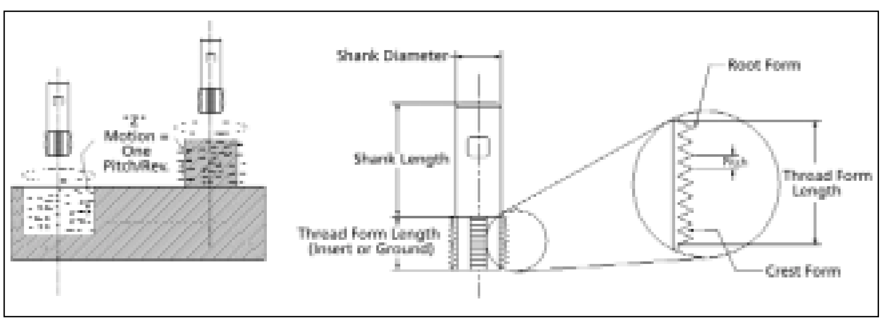
Thread milling requires a machine capable of helical interpolation—the ability to move simultaneously in three axes. The cutting tool moves in a circular pattern—X- and Y-axis movements—and at the end of one complete revolution around the circumference of the thread, the cutter has advanced—Z-axis movement—into the workpiece a distance equal to one thread pitch.

Thread milling offers a number of advantages over other threading methods. It is obviously better when a tap would be impractical or the workpiece is unsuitable for turning on a lathe. And because a single thread mill can thread holes with different diameters, it can replace a number of taps.

When thread milling, the operator can change the pitch diameter of a thread by making a CNC programming change. Thread milling is also useful for applications in which keyways, side holes or other voids interrupt the area being threaded. The entry and exit burrs are minimal when thread milling, compared to other threading methods. Often, the thread-milled hole requires no additional cleanup.

Plus, the flexibility of modern machine tools allows the thread mill to cut from either the bottom up or the top down. In a blind hole, cutting from the bottom up avoids recutting chips and marring the surface finish, or damaging the cutter.

Thread mills run at high speeds require less power than taps to produce



A machine capable of helical interpolation—motion in three axes—is required for thread milling. With both internal and external thread milling, the tool moves in a circular pattern (X and Y axes) while simultaneously moving into the hole (Z-axis).

good threads. Tapping requires a lot of horsepower at the spindle and on the Z-axis servomotors, which are much smaller than the spindle motors. Also, CNC machines usually have low torque at low rpm—a problem when tapping on machining centers. Thread milling makes it possible to cut threads on lower-horsepower machines, saving time and money.

Considerations that must be addressed when deciding whether or not to thread-mill include thread diameter, thread pitch, depth of thread, percent of thread engagement, workpiece material and hardness, and tool-to-machine rigidity. In general, small-diameter holes—up to ¾" in diameter—are better cut with taps than with thread mills.

Material	Chip Load Per Tooth	Surface Speed (sfm)
Aluminum	0.0015"-0.0060"	800-1,200
Brass	0.0020"-0.0065"	400-700
1045	0.0010"-0.0016"	320-420
4140	0.0008"-0.0009"	260-290
316 ss	0.0008"-0.0018"	200-320
420 ss	0.0012"-0.0015"	240-280

M. Restall

Recommended starting speeds and feeds for TiN-coated thread mills with indexable inserts.

Tool Materials, Styles

Thread mills can be made from a variety of materials, depending on the workpiece material. Thread mill inserts are usually carbide. Solid cutters are made of HSS, solid carbide or have steel bodies with carbide tips brazed onto them. Through-coolant styles are available and, as with any cutter, a suitable coating extends thread mill life and may allow higher cutting speeds.

The simplest cutter is the laydown insert in a threading bar. It produces the best thread form but is slow, since the cutter has only one effective tooth and must make a complete revolution around the circumference of the thread to produce a thread length of one pitch. Tools with several sets of threads ground into them and multiple edges produce threads faster, because the thread length per revolution is increased and multiple edges allow higher feed rates.

Advantages of thread milling

Threading operations are much more demanding than straight turning operations because the cutting forces are considerably greater in threading, and the cutting edge of a threading insert is much smaller and weaker. Various threading methods are available, including tapping, single-point turning, die threading, thread grinding and thread rolling.

Another method for cutting threads is thread milling, which offers the following advantages:

■ One thread mill can be applied to cut many ID thread sizes, whereas a specific tap is required for each ID thread size.

■ One thread mill can be applied to cut many OD thread sizes, whereas a specific die is needed for each OD thread size.

■ Thread milling is more accurate than single-point turning, tapping or die threading.

■ Thread milling is also faster than single-point threading, tapping or die threading, because the thread is cut with several teeth in one pass.

A right-hand thread mill only rotates in a right-hand direction, but by using up or down milling and changing the axial feed direction, right-hand and left-hand threads can be cut externally and internally.

—Information supplied by Carboloy

Company specializes in 'superbolts'

Superbolt Inc. manufactures a patented, mechanical, multiple-jackbolt tensioner in diameters from ½" to 30".

Superbolt® products replace conventional hex nuts and are easier and safer to install. Any diameter stud or bolt can be tensioned with the use of hand tools.

The tensioners consist of a short, thick-walled cylinder with internal threads. Additionally, the ends of the cylinder provide a flat surface that is drilled and threaded for a series of hardened jackbolts.

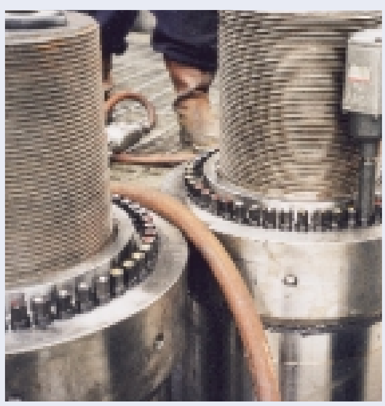
In service, the threaded cylinder is screwed into position on a bolt. As the jackbolts are tightened, they generate tension on the bolt or threaded stud, as do conventional hex nuts when tightened with a wrench. The internal threads in the cylinder blank, as well as the threads for the jackbolts, often are thread-milled.

The threading is performed on vertical machining centers. Machines with 40-taper spindles thread small and medium-size units, and 50-taper spindles mill the threads in bolts 12" in diameter and over. Generally, the cycle time for thread milling varies with material and the thickness of the multi-jackbolt tensioner.

The production process begins when Superbolt receives made-to-order, heat-treated, extremely thick-wall tubing in 16' to 20' lengths. After the material hardness has been verified, a cut-off section of tubing becomes the basic element for a Superbolt tensioner.

After initial machining to dimensional requirements, the thread milling begins and generates threads that conform with ANSI B 1.3M-1992 System 22 and System 23 standards. Superbolt produces threads with a differential analysis of less than 0.001", which is the difference between actual and functional pitch diameters, according to ANSI System 23.

—M. Restall



Superbolt

Superbolt thread-mills its mechanical, multiple-jackbolt tensioners, which are available in diameters from ½" to 30".

To cut threads longer than the tool length when using a multiple-tooth cutter requires that the tool move down into the workpiece the length of the cutter minus one thread pitch. The overlapping of the threads at the end of the cutter allows for the continual production of good threads.

Toolholders, Tools

The length of thread that can be cut successfully depends on the rigidity of the machine and toolholder. Rigidity is critical for producing high-quality threads. Producing the best threads requires applying the largest-diameter cutter that fits into the holder while allowing the cutter to engage the workpiece and enter and exit the cut properly. Machines with 50-taper spindles are capable of cutting the majority of threads used today.

While large-diameter cutters that accept more inserts allow faster cutting, there is a greater chance of the tool moving away from the work and, as a result, cutting unacceptable threads.

Superbolt

Thread mills can cut internal and external threads in one pass.

There should be no runout in the tool/toolholder setup or any possibility of the tool slipping. Thread milling holders—or any of the new-generation hydro-grip or shrink-fit holders—work well with straight-shank tools. Collets are not recommended.

Straight-shank tools are the most universally used style, but not necessarily the most rigid. Cutters with built-in CAT tapers offer more rigidity, but at an extra cost. Bolt-on cutters for arbors are available in larger diameters.

When using insert-style thread mills, it is important that the inserts are ground accurately and have a positive seating

Thread milling equations

In a linear milling operation, feed per tooth depends on the ratio of the radial DOC (A_r) to the cutter diameter (D) and the average chip thickness (H_{zm}). When thread milling, the cutting tool is rotating through the helix—not a flat plane. Therefore, the radial DOC must be calculated prior to calculating the actual feed per tooth and the rate of table travel.

To determine A_r, first calculate the major (D_1) and minor (D_2) diameters and plug the dimensions into the equation:

$$D_1 = D_2 + (2 \times \text{thread depth})$$

For example, if the depth of an internal thread is 0.048" and the minor diameter, or hole diameter, is 3.187", then the major diameter is:

$$3.187" + (2 \times 0.048") = 3.283"$$

For an external thread:

- Subtract twice the thread depth from the part diameter to find the minor diameter.
- The major diameter equals the part diameter.

Use the results of the previous calculation to determine the radial DOC.

$$A_r = \frac{(D_1^2 - D_2^2)}{4 \times (D_1 - D)}$$

Therefore, if the cutter diameter is 1.20", the radial DOC is:

$$A_r = \frac{(3.283^2 - 3.187^2)}{4 \times (3.283 - 1.20)} = 0.074"$$

Next, use the following formula for average chip thickness to calculate the actual feed per tooth (S_z).

$$S_z = H_{zm} \times \sqrt{D/A_r}$$

With an average chip thickness of 0.003", the feed per tooth is:

$$S_z = 0.003 \times \sqrt{1.20/0.074} = 0.012 \text{ ipt}$$

When milling an internal diameter, avoid plunging the cutter in the radial direction to the full DOC. Insert chipping can occur and tool life is likely to be shortened. The preferred technique is to increase the DOC gradually up to its full value, traveling around the inner diameter 1.25 times. If you must use the straight in-feed method, reduce the feed per insert by one-half until the desired DOC is achieved, then circular-mill at recommended feed rates.

Finally, calculate the rate of table travel (ipm) at the cutter diameter (IPM_D) and centerline (IPM_{CL}).

$$IPM_D = S_z \times rpm \times N_i$$

and

$$IPM_{CL} = \frac{(D_1 - D)}{D_1} \times IPM_D$$

Continuing with the example, if the rpm is 1,241 and the thread mill has one insert (N_i), then the ipm at the cutter diameter is:

$$IPM_D = 0.012 \times 1{,}241 \times 1 = 14.892 \text{ ipm}$$

And the ipm at the cutter centerline is:

$$IPM_{CL} = \frac{(3.283 - 1.20)}{3.283} \times 14.892 = 9.45 \text{ ipm}$$

For an external thread, the following formula is for determining the speed at the cutter centerline:

$$IPM_{CL} = \frac{(D_1 + D)}{D_1} \times IPM_D$$

Therefore:

$$IPM_{CL} = \frac{(3.283 + 1.20)}{3.283} \times 14.892 = 20.34 \text{ ipm}$$

Equations Key:

- A_r = radial depth of cut
- D = cutter diameter (inches)
- D_1 = major diameter (part OD for external threads)
- D_2 = minor diameter (hole ID for internal threads)
- H_{zm} = average chip thickness (inches)
- IPM_{CL} = ipm at cutter centerline
- IPM_D = ipm at cutter diameter (at the insert)
- N_i = number of inserts
- S_z = feed per tooth (ipt)

—Information supplied by Carboloy

method. Shimmed cutting tools offer some protection in the case of a machine crash.

At Superbolt Inc., where we threadmill both internal and external threads in AISI 4140, we apply tools with a neutral rake. Softer materials, like aluminum and stainless steel alloys, require high-positive-rake angles. The best tools for our applications are 3"-dia., multiflute thread mills. They accept the 20 to 30 styles of inserts needed for the variety of threads we produce (see sidebar, page 31).

With materials such as 4140, in the hardness range of 25 to 30 HRC, we find that 6 threads per inch is about the coarsest pitch that can be cut in one pass on a 40-taper machine with a multiple-insert cutter having a diameter less than 1.5". Threads that are coarser than 5 threads per inch, greater than 2" in diameter and have thread lengths exceeding 2-times the diameter should be produced on a machine with a 50-taper spindle.

When possible, it's beneficial to use through-the-spindle coolant. Coolant directed to the cutting edge improves tool life and flushes chips away. Also, the action of the milling cutter makes smaller chips and prevents creation of the "bird's nest" that often occurs when stringy materials are threaded.

Programming

Properly chosen programming parameters lead to the production of the best threads. As such, many thread-mill manufacturers offer software that selects the tool and allows the operator to write the program by selecting parameters on the CNC's screen.

The operator inputs information such as thread type, size and pitch, and whether the threads are internal or external, right- or left-hand. The software also processes information about cutting conditions, such as chip thickness (chip load) and surface footage.

The better software allows users to specify, in degrees, the amount of ramp-in and ramp-out. Ramping in prescribes the incremental cut into the material through a specified arc of the circle at the beginning of the machining cycle. Ramping in eliminates plunge cutting to full depth, which can lead to starting-point chatter and premature cutter failure. Failure to ramp in properly usually results in chipped cutter teeth.

In general, the objective is to cut the thread in one pass. However, in cases where the thread pitch is coarse and the material is tough to machine, it may not be possible to get an acceptable thread in one pass.

The surface finish produced by thread milling should be better than that produced by tapping and, in many cases, equal to single-point cutting on a lathe. As in any thread-cutting operation, poor stability in the setup leads to chatter. A scalloped appearance on the threads indicates the feed rate is too high.

About the Author

Mike Restall is the production manager for Superbolt Inc., Carnegie, Pa.