Technology

How to Maximize Throughput and Part Quality When Threading

Kip Hanson | Oct 19, 2017

What You Need to Know:

Whether turning or milling, thread-making is tough work. You need the right tool with the right method for the right application.

Thread cutting methods go back a long, long time, but newer techniques and technology continue to evolve.

CNC lathes may have made single-point threading easier, but there is a fair amount to understand about the right tools for these jobs, such as the variety of inserts and how to program them.

Thread milling is an excellent method. If you're going to mill, solid carbide tools are the fastest, most rigid and most productive option available.

Learn about the most popular and effective methods and tools for maximizing your shop's threading operations.

Threading is tough work. Cutting forces are high, tolerances are tight and tools wear quickly, so selecting the best *threading* tool possible is always a best practice. An equally important best practice, however, is using the right thread-making method with the "right" tool in the "right" application. To understand what is *right* for threading, one needs a better overview of threading in action.

Here are the three most popular options—tapping, single pointing and thread milling, along with key technical advice on the methods or tools to use in these specific applications—and advice on what to avoid to make your threading work perform at the highest possible levels.

High-Performance Threading: Tap and Die Sets

For as long as people have been making threads, they've been using *taps* and *dies* to cut them. While dies have pretty much gone the way of carbon steel tools, *taps* are alive and well. However, tapping methods and thread-making technology have kept up with the times. Here's how you can use tapping to your advantage:

Always use a plug tap when possible. True, bottom taps can get closer to the bottom of the hole, but thread quality is usually not as good, nor is tool life. Now that you're using a plug tap, be sure to drill the minor diameter as deep as possible (and all the way through if you can). This provides more room for chips.

Don't settle for a general purpose tap. Chances are good that a tap has been developed specifically for the metal and thread style you are machining. For problem threads, squirt a small amount of machining wax or tapping compound into the hole before tapping. Some machine tools have an option to automate this step. Many taps are available with through-the-tool coolant. If your machine is so equipped, by all means use it.

Use roll or form tapping when it makes sense. Roll or form tapping is an excellent option in ductile materials such as aluminum and stainless steel and provides a strong, smooth thread. Be sure to drill the hole according to the tap manufacturer's recommendations, and towards the upper limit if possible.

Be aware of the tap's H-value. Depending on the class of thread, an H2 or H3 is a good place to start. Each change in H-value increases or decreases the tap's pitch diameter by 0.0005 of an inch.

Be aware of tap holder types. If your machine is equipped with rigid tapping (most new ones are), be sure to use a "synchro" tap holder specifically designed for this. Otherwise, use a floating (mill) or self-releasing holder (lathe). For high-volume threading on a machining center, a self-reversing tapping head might be just the ticket.

CNC Machining: Single-Point Threading

Anyone who has cut threads on an engine lathe knows that single-point threading can be challenging. What angle should you tip the compound slide? How many passes should you take? What is the right number of spring passes? Thanks to CNC lathes, it is all programmable. But while the threading process has become simpler, finding the right tool has become more complicated. Here is what you need to know about single-point threading tools:

Full-profile threading inserts cut the entire thread form. They eliminate the need for a separate turning operation to trace the thread's major diameter (or minor, for internal threads). However, each insert can only cut one specific thread pitch and profile, increasing tooling inventory and costs.

Partial-profile inserts can cut multiple thread pitches with a single insert. They offer greater flexibility on size control, but they do increase cycle time slightly over full-profile "topping" inserts.

Double-sided laydown-style inserts offer the lowest cost per edge. A special anvil is required to tip the insert at the required pitch angle. **Take note:** Use the wrong anvil and you'll have a pile of scrap.

Try using G76 canned threading cycle. Still using a G92 or G32 canned threading cycle? It may be time to pick up the programming manual and master G76. It's far more powerful. To increase tool life and part quality, use the compound feed option.

Be careful with spindle rpm, as it's easy to overfeed the machine. For example, the feed rate per revolution on a 1/2-13 thread is 0.0833 of an inch. At 3,000 rpm, this equals a traverse rate of roughly 250 inches per minute. Even if your machine is capable, that might be too fast for repeatable thread accuracy.

Tool coatings may be less effective in threading work, leading to built-up edge (BUE). Because cutting speeds during threading are often lower than turning or boring operations, you need to watch out. Uncoated or TiN inserts are the best bet, especially on high-temp alloys and very coarse threads.

Watch your Z-axis positioning. The Z-axis starting position should always be three to four times the thread pitch in front of the part. Also, turn off constant surface speed (G96) with a G97 command when threading.

> **Traditionally limited to machining centers, thread milling is becoming widespread on turn-mill lathes and multitasking machines. It's a bit slower than tapping but offers far greater control and flexibility.**

Thread Milling

Traditionally limited to machining centers, thread milling is becoming widespread on turn-mill lathes and multitasking machines. It's a bit slower than tapping but offers far greater control and flexibility. Where taps are limited to hole threading, *thread mills* can cut internal and external threads alike, often with the same tool. Thread milling is an excellent method, but you need the right tool and the right program to make it work.

Thread milling is the way to go for lower part volumes, difficult materials or where thread quality is critical (and where single pointing is not possible). Here's what you need to know:

Solid carbide thread mills are the fastest, most rigid and most productive tool available. They can cost more than other types but often offer the lowest cost per part.

Indexable thread mills require more time to cut the thread. Typically, this is due to having fewer flutes than their solid carbide cousins. For low production quantities or where the tooling budget is limited, however, they are often a good choice.

Single tooth, multi-flute thread mills should be used on delicate or thin-walled parts, or where long reach is required. Start the thread from the bottom and work your way out of the hole.

Most CAM systems today generate excellent threading routines. CAM systems are very useful. Check with your software partner if you need help programming one. Don't have a CAM system? Most thread mill supplier websites offer program generators for free. Plug in the values, then cut and paste the code snippet into your program.

Use cutter compensation to adjust the thread size (one of the big advantages of thread milling).

Whatever approach you take, always buy the best cutting tool possible. Use a high-quality neat oil or clean, properly mixed cutting fluid. The setup should be rigid, the tool on center (lathe), with no runout (mill) and held securely (either) in the appropriate toolholder. Plug in the correct program values, and get threading.

What tools and methods is your shop using for high-performance threading?

Copyright ©2025 MSC Industrial Supply Co.