JUNE 2001 / VOLUME 53 / NUMBER 6

▶ BY KIMBERLY PONTIUS

Tom Wright

Low-Silicon Lowdown

Simplifying tapping operations on low-silicon aluminum.

Tapping a nonferrous alloy such as aluminum always presents a quandary. One can easily be lulled into believing that because the material is softer and more yielding, it should be easier to work. And for the most part, this is true. But tapping low-silicon-aluminum alloys can be problematic if approached in a cavalier fashion.

Cool It

Aluminum varies according to the chemical composition of its metal matrix. When machined, low-silicon, or eutectic, aluminums create less wear on HSS and carbide cutting tools than their more abrasive, higher-silicon counterparts.

Aluminum is prone to galling, which is the buildup of material on the cutting tool and/or workpiece due to severe friction and abrasion. Galling can cause catastrophic tool failure and usually results in scrapped workpieces. One way to counter galling is by directing coolant—and plenty of it—at the cutting zone.

The preferred method is high-pressure coolant delivered through the shank of the tapping attachment and the tap. In addition to minimizing galling, this method invariably improves thread quality and extends tool life. Commonly, 700 psi is the recommended coolant pressure, although some tapping attachments can withstand pressures up to 1,000 psi without suffering seal failure.

The friction and resulting heat from tapping forms a vapor barrier around the tool/workpiece interface, and low-pressure coolant is not powerful enough to flush the chips through this barrier. The result is that chips fall back into the cutting zone.

Conversely, the force of high-pressure coolant allows chips to break through the barrier, flushing them away while properly lubricating the tool.

If you're ordering new tapping equipment, by all means purchase the optional high-pressure-coolant system. The productivity gains from repairing fewer damaged threads and removing fewer broken taps will justify the extra cost.

When tapping low-silicon-aluminum alloys, it is imperative to choose the proper fluid. Cutting oils, especially those high in sulfur, can create chemical reactions that may alter the material's cosmetic appearance or contribute to its deterioration.

For years, solvent-based tapping fluids for aluminum were standard in the shops where I worked. Though they performed well, the fluids emitted an unpleasant odor and weren't packaged for high-volume production.

Soluble oil is the choice for most high-volume tapping today, and there are many brands from which to choose.

Recently, I've been experimenting with some of the new, environmentally friendly tapping fluids. They are a non-staining, odorless alternative to some of the solvent-based products and, based on initial results, they work quite well.

Tool Choice

Tap selection is critical when working in aluminum. Many tool manufacturers recommend carbide taps for increased rigidity, accuracy and repeatability.

Standard straight-flute plug taps, taper taps and bottoming-style taps are perfectly acceptable for low-silicon aluminum. (When hand-tapping, these types of taps will produce fine results if properly guided and supported.) However, most manufacturers recommend that users apply spiral-point or spiral-flute taps that have as few flutes as possible. Fewer flutes enhance chip evacuation. These styles of taps are particularly useful for high-speed, high-production operations.

Spiral-flute taps, like drills, are designed to direct chips out of the hole. This proves particularly helpful when tapping a blind hole on a vertical machine.

Spiral-flute taps won't help much, though, if through-coolant isn't used. In my experience, the rotation of the spiral body restricts coolant travel down to the cutting zone.

Another option is the roll-form tap. These tools "form" the thread rather than cut it. Since no chips are produced, they're often a good choice for tapping blind holes.

Roll-form tapping can be performed in all ductile materials. The advantages of this style of tapping include no improperly cut threads, no pitch deviation, greater thread strength, longer tool life and higher tapping speeds. In situations where part cleanliness requires the operation to be free from chips and/or a wash process is not in place, roll-form tapping offers distinct advantages over cut tapping.

More torque is required when roll-form tapping, and the minor diameter of the thread will appear rough due to the forming process. Also, when roll-form tapping, the core hole diameter must be larger than it would be with a cutting tap (see equations, this page).

T.M. Smith Tool International

High-pressure through-coolant taps minimize galling.

When deciding on the percentage of thread required, remember that a 100 percent thread is only 5 percent stronger than a 75 percent thread, but it takes three times the power (torque) to turn the 100 percent threading tap. This extra torque can cause the spindle to slow down or stall, leading to excessive tool breakage.

The tap's surface finish can help prevent galling and facilitate chip removal. Many tap manufacturers recommend nitrided taps for cast aluminum, while a bright or chrome finish tool is recommended for long-chip aluminum. While these finishes extend the tool's wear characteristics, their contributions to lubricity and anti-galling are more important.

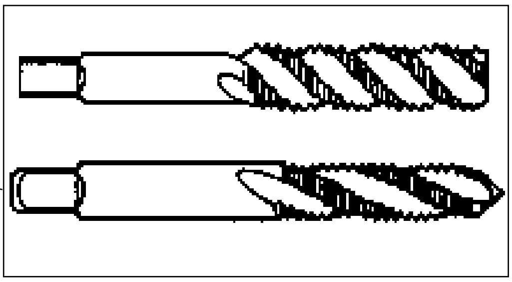
Tapping equations

The following equations, which are for American National threads and pertain to all materials, can be used to determine the percent of thread contact and the drill size required when tapping:

$$\text{\% full thread} = (\text{threads/inch}) \times \left(\frac{\text{major diameter of tap} - \text{drill diameter}}{0.01299}\right)$$

$$\text{Tap drill size (cutting)} = \text{major diameter of tap} - \left(\frac{0.01299 \times \text{\% of full thread}}{\text{threads/inch}}\right)$$

$$\text{Hole size (roll-form tap)} = \text{tap OD} - \left(\frac{0.0068 \times \text{\% full thread}}{\text{threads/inch}}\right)$$



Union Butterfield

Spiral-flute taps, like drills, are designed to direct chips out of the hole. This proves particularly helpful when tapping a blind hole on a vertical machine.

Angle Considerations

The best rake angle for a tap depends on the material machined. Materials that produce long chips normally require a tap with a greater rake angle, whereas materials that produce short chips require a tap with a smaller rake angle. Short-chip aluminum requires a rake angle of 12° to 14°, while long-chip aluminum can be tapped better with a tool possessing a 20° to 25° rake angle.

A small relief angle can be used in low-silicon aluminum, unlike harder materials, such as stainless steel, which require a tap having a greater relief angle to reduce friction. In most cases, the relief angle should be smaller when tapping blind holes than through-holes, so that the chip's root can be sheared off when the tap reverses without breaking the cutting edge.

Union Butterfield

Roll-form taps don't cut chips but require more spindle power to produce the thread.

The actual thread cutting is done by the lead of the tap. When there are more threads in the chamfer length or lead, the torque is reduced, the thread is produced much easier and the life of the tap increases.

In blind holes lacking the room to drill deep enough for a tap with a longer lead, taps with short leads are used. In some cases, the lead of the tap is as little as 1.5 threads. This greatly increases torque and reduces tap life.

Even when applying taps with shorter leads, it is still important to drill deep enough for adequate clearance. Tap manufacturers recommend allowing one thread length plus 1mm beyond the lead of the tap for drill clearance. When tapping aluminum, it's desirable to have as much clearance as the part design allows.

The relief angle impacts true-to-gage thread cutting, as well as the tap's free-cutting ability and tool life. Too little relief can result in unnecessary pressure and distortion of the thread, and cause premature tool failure. If the relief angle is too great, pitch guidance and the self-centering capability of the tap cannot be guaranteed—especially in soft materials.

In materials like stainless steel or bronze, the relief angle should be larger, to allow free cutting and more fluid to reach the cutting surfaces. A bigger relief angle can allow a higher tapping speed, provided the tap is guided concentrically into the hole by the machine and tapholder.

Union Butterfield

Two- and 3-flute taps are particularly useful for high-speed, high-production operations where maximum chip evacuation is important.

The use of a compensating tapholder is highly recommended, due to fluctuations in the lead screw that can result in following error. If this error is not compensated for, thread distortion and tap breakage can occur.

Guidelines for Success

Although low-silicon aluminum is regarded as a material that's easy to machine, adhering to the following guidelines will help ensure success when machining it:

- Make sure there is plenty of fluid applied; high-pressure, through-coolant taps are preferred.
- Select the proper hole size for the type of tap you wish to apply and the percent of thread desired.
- Choose the proper tapping speed.
- Keep the tap sharp. The chamfer angle is critical; resharpening should be done on a machine.
- Check for true alignment of the tap to the hole.

About the Author

Kimberly Pontius is operations manager of TD&M Production Machining, New Haven, Ind., and a regular contributor to CUTTING TOOL ENGINEERING.