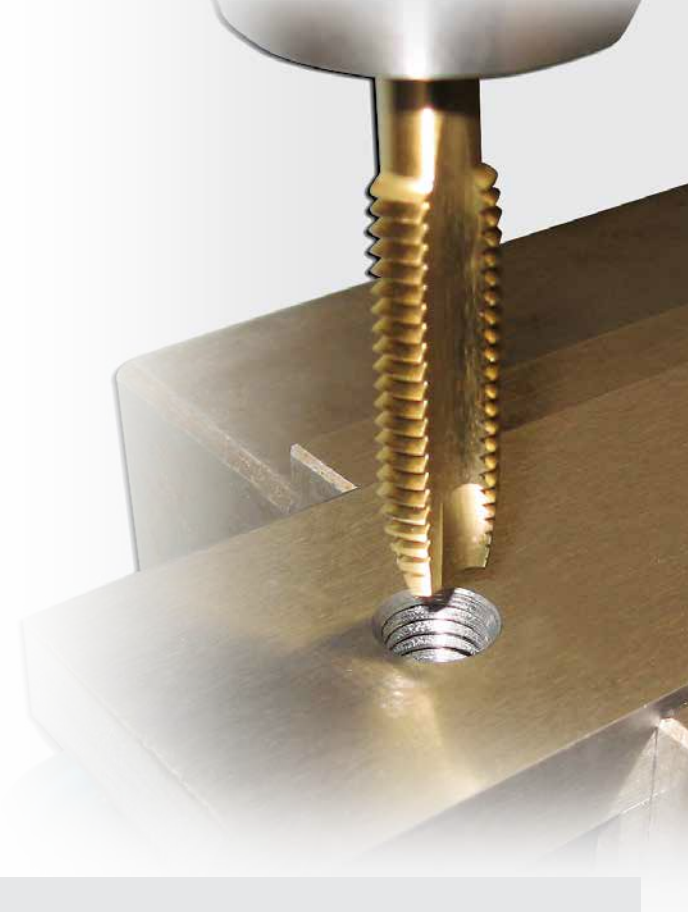
TAPPING RIGHT

AN OVERVIEW OF THE DIFFERENT TAP STYLES AND HOW TO APPLY THEM

Holemaking ranks among the most frequently performed of all metalworking operations, and of the countless holes produced each year, many are threaded to accept a screw, stud or bolt. **The question then becomes, what's the best way to go about producing all these screw threads, and what cutting tool should we use?**

A TAP IS USED TO CREATE THREADS IN AN EXISTING HOLE

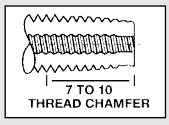


HAND ME DOWN

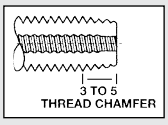
Thanks to a man named Albert Kehl, who filed for a patent on his thread-cutting invention in 1914, that tool could very well be the hand tap. Today, hand taps are inexpensive, widely available, and compared to thread mills (their more modern-day alternative), hand taps don't require any additional special or very expensive equipment. Still, it's not always clear which of the several styles of hand tap are most effective for a given application.

For starters, **three chamfer styles exist: taper, plug, and bottoming**. All have straight flutes, and all share the same basic shape. The difference lies within the chamfer length—the section of the tap's business end that does the actual cutting. The chamfered area on a taper tap is approximately nine threads long. Plug taps come in at roughly four threads, while bottoming taps have a chamfer length of only one or two cutting edges.

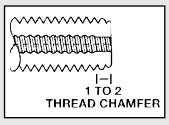
TAPER:



PLUG:



BOTTOMING:



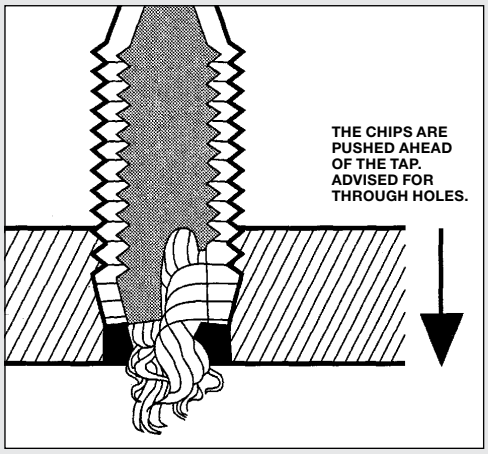
WHO CARES?

If you're wondering why this is important, listen up: the longer the chamfer length, the straighter and more freely the tap cuts. As a rule, this translates to better part quality and longer tool life. That doesn't necessarily mean that taper taps are always the most appealing or most suitable, though; while they're ideal for through holes, most blind holes require threads all the way to the bottom, which in turn calls for—you guessed it—bottoming taps. That said, bottoming taps don't work as well in tougher materials like stainless steel and especially superalloys. In these situations, plug taps offer a nice balance of maximum thread depth and efficient cutting, and is why they are generally more popular than the other two styles.

SPIRALING TO SUCCESS

There's more to the story than thread depth, however. Chip control can be problematic in any tapping operation, particularly in blind holes or in longer through-holes. For the former, a **spiral flute tap** is the best bet, at least in ductile, long-chipping materials. That's because the spiral flutes help to direct chips up and out of the hole, exactly where you want them. Be careful to select the right helix angle, however—as with drills, a "faster" helix is better for soft metals like aluminum and copper, while a slow spiral should be used on the aforementioned superalloys.

GET TO THE POINT

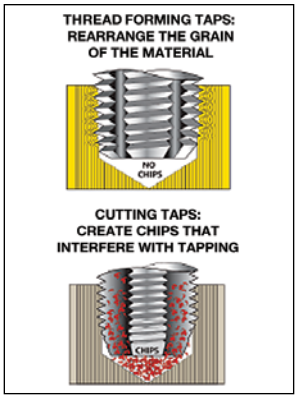


Similarly, **spiral point taps** are your best bet for through-holes—particularly on thicker materials—as they push the chip forward, ahead of the tap. The spiral point also tends to make the tap freer cutting, a good thing in any material. Unlike hand taps, spiral-fluted taps are most commonly available in bottoming, or modified bottoming (which is a compromise between the bottoming and plug chamfers), and although spiral point taps are also available in bottoming, it's the plug style chamfer that is much more popular. Regardless of their flute geometry, the plug-style taps are the most versatile.

THE TAP RECAP

This is only the tip of the tapping iceberg. There are also different thread forms, including UNC, UNF, Whitworth, Acme, and STI (screw thread insert) to name a few. Different "H" and "D" limits are available (H limits are on standard "ANSI" threads, while D limits are used for metric "ISO" threads), giving machinists a way to make the thread diameter larger if necessary. There are also **thread forming taps**, (see following page) which as their name implies, cold form the thread rather than cutting it. The result? No chip problems, stronger threads, and better tool life—assuming the customer allows you to use them, that is. Some don't, so always ask.

THREAD FORMING TAPS DO NOT PRODUCE CHIPS; RATHER, THEY FORM THE THREAD



THREAD FORMING TAPS VS. CUTTING TAPS

Finally, there are several toolholding considerations, and lubricants, and programming (for you CNCers), stuff we don't have time to cover today. The message for now is this: if you're stuck using the century-old hand tap and are looking for a better way to make threads, it's probably time to review what's available. You might be pleasantly surprised.

Have more tapping questions or require assistance with your current tapping application? Call Travers Tool's expert technical support team at **800.234.9985** or email **tech@travers.com**

CUT TAPS VS. FORMING TAPS

Taps are the most common method to produce threads in a part. Taps are easy to use and they produce good results, quickly and with little training. Taps can be applied in a multitude of different ways that include hand tapping, pneumatic tapping arms, and canned tapping cycles in CNC machines. There are two different types of taps to choose from when machining aluminum – 'cut taps' and 'thread forming taps', which are also known as 'form rolling taps'.

BENEFITS OF CUT TAPS

- Most common type of taps and thread making.
- Simple method including hand and power tools.
- Different types for different applications: taper, plug, or bottoming.
- More versatile and can be used in certain materials that do not produce threads
- Market is mature, so a wide variety and selection to choose from at competitive prices.

BENEFITS OF FORMING TAPS

- No chips to deal with!
- Works well on smaller threads g. 4-40 threads or M2.5 X 0.45
- Excellent thread quality, threads are tighter and stronger.
- Advanced substrates and coatings are more common in form taps such as carbide substrates or DLC coatings that work tremendously well on aluminum.
- Form taps with advanced substrates and coatings offer increased tool life, allow increased speeds and decrease the propensity for scrapped parts & tap breakage.

CUT TAPS

Cut taps are general purpose and very common in the market, with a variety of styles available including straight flute, spiral fluted, spiral pointed, pipe taps, acme taps, S.T.I taps and more. Cut taps are used to cut both ferrous metals and non-ferrous materials alike. Cut taps are designed to remove material as it is fed through a predrilled hole to produce the desired thread shape. The removed material or chip is then guided, depending on the design of the tap. Bottoming taps will bring the chips back towards the entry hole, while plug taps will push the chips down the opposite way (machinist note: plug taps are stronger but must have the clearance to remove the chip). These taps are great to have around the shop for non-production jobs, maintenance jobs, or simply to keep in stock for those unknown projects because of their price point. Most cut taps are made from High Speed Steel (vanadium) and may have a coating for added protection. While they certainly have their benefits, all taps endure a tremendous amount of stress when rotating, and this stress may break or snap the taps. In certain situations, if a tap breaks inside the part, the part must be scrapped, which can be costly. Cut taps do tend to break more often than 'roll forming taps'.

FORMING TAPS

Thread forming (roll forming) taps deform material to produce the desired thread shape. Since the workpiece is deformed, as opposed to being cut, the produced thread is stronger. The main benefit of the deformation process is that there are no chips produced (and therefore, no chips to manage), so there is no need to select a bottoming, plug or tapered style tap. Another benefit from a forming tap is their better performance in smaller threads, such as 4-40 or M2.5-0.45 threads. However, roll forming taps are limited to use in softer materials that can be deformed, and are not ideal for materials such as cast iron or high hardness materials. Thread forming taps are offered with advanced substrates, coatings and some are even offered with coolant ports. A carbide substrate significantly increases tool life and productivity and can be further enhanced with a high-performance coating that has a positive reaction with non-ferrous materials such as aluminum, like a DLC (diamond like carbon) coating for instance. When tapping smaller threads, roll forming taps are very productive. These taps can be more costly, application specific, and require a different size drill vs. cut taps, as the same 10-32 thread requires a different size drill from a roll forming tap and a cut tap. Roll forming taps also perform the best when the tap is centered and parallel with the hole.

Both types of taps have their benefits and choosing which type can help productivity significantly. When using a machine tool to tap aluminum, thread forming taps do provide a great argument but like most things in the shop – it comes down to quality, productivity and economics. A prototype or small run may not warrant use of a thread forming tap. However, if you are producing many tapped holes, thread forming taps should be considered.

Both types of taps have their benefits and choosing which type can help productivity significantly. When using a machine tool to tap aluminum, thread forming taps do provide a great argument but like most things in the shop – it comes down to quality, productivity and economics. A prototype or small run may not warrant use of a thread forming tap. However, if you are producing many tapped holes, thread forming taps should be considered.