▶ BY PETER FRETTY

JANUARY 2001 / VOLUME 53 / NUMBER 1

Going Deep

Regal Cutting Tools

Deep-hole tapping takes on a new dimension when the workpiece material is aerospace-grade titanium.

Deep-hole tapping is rarely easy, and exotic workpiece materials further complicate the task.

Deep-hole tapping a complex component made from aerospace-grade titanium, for example, can be especially challenging. And scrapping a nearly complete part because of a broken tap can mean a substantial loss of time and money. Minimizing the scrap rate requires utilizing the proper tool and tapping techniques.

Before addressing those subjects, though, it's important to answer two questions: "What is *deep*, and why does it necessitate special considerations?" In drilling, deep is considered any hole depth that exceeds three times the tool's diameter. Deep-hole tapping refers to a threaded depth of 1½ times, or more, the nominal diameter of the tap. So, tapping a ¼-20 thread to a depth of ⅜" is considered "deep."

Tapping a deep thread means extending the tool-to-workpiece contact time, which increases the cutting forces on the tool and generates more heat. This situation can be especially difficult in machining components for the aerospace industry. There, the combination of deep-hole tapping small-diameter holes in exotic materials like titanium increases the likelihood of tool breakage and produces a higher level of thread inconsistency.

However, two approaches that address these difficulties are enlarging the hole and using a tap that was designed for deep-hole tapping.

Larger and Deeper

While drilling the proper size hole is crucial to a successful operation, an oversized hole for a threaded fastener reduces heat generation and tool stress during the tapping operation. It also reduces the percentage of thread-to-thread contact.

The International Titanium Association's R&D center (www.titanium.org) states that it's acceptable to tap holes with as low as 50 percent threads. This means that the depth threads are cut into the walls of the hole can be as little as 50 percent of the height of the deep-hole thread, as measured radially between the hole's major and minor diameters.

The National Institute of Standards and Technology also finds that tapping 50 percent threads is acceptable in deeper holes. This becomes especially important when tapping small holes in exotic or workhardenable materials. While the percentage of thread-to-thread contact is reduced in terms of thread height, the necessary amount of metal-to-metal contact can be achieved if the depth of thread is sufficiently increased in the drilled hole. When deep-hole tapping, thread-contact percentages are not as vital as they are with shallower depths. The reason is because as the depth of the hole increases, the likelihood of stripping is reduced. (The fastener will break before it strips.)

The permissible enlargement in hole diameter depends on part specifications regarding the required percent of full thread and the number of threads per inch. Once known, the values for percent of full thread and number of threads per inch can be plugged into the tap-drill size formulas found in the "Tapping and Thread Cutting" section of the *Machinery's Handbook* to generate the proper hole size for the intended fastener.

However, before enlarging a hole, prudence dictates that the customer be apprised of the change. This is especially so in aerospace applications, where tap size and hole size may be specified and critical to part performance.

While drilling the proper size hole is crucial to a successful operation, an oversized hole for a threaded fastener reduces heat generation and tool stress during the tapping operation.

Cutting Considerations

Often, aerospace parts are made of titanium because of its unparalleled resiliency, its high strength-to-weight ratio, its ability to withstand high temperatures and its corrosion resistance. These properties also make it a difficult material to machine.

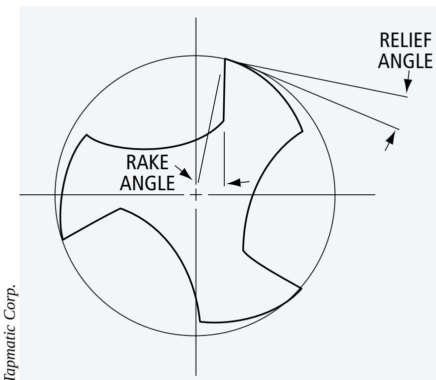
Following are some tips on cutting parameters and tool geometry that make the job easier.

Cutting rate. Commercially pure titanium has many characteristics applicable to aerospace grades. However, it does not possess any of the alloys that boost its resiliency and add to its deflection rate. Therefore, somewhat higher cutting rates (approximately 40 sfm) are common with nonaerospace-grade titanium.

The recommended cutting rate when tapping aerospace grades of titanium, especially deeper holes, ranges between 10 and 14 sfm. It is not advisable to cut at a slower rate since workhardening may occur. This leads to premature tool wear and broken tools, because the cutting edge cannot penetrate the workpiece.

Furthermore, the level of heat generated while machining materials like titanium contributes to galling.

Flute arrangement. When tapping deep holes, taps with fewer, but larger, flutes carry away more chips as they exit the hole, reducing the chance of tool failure due to chip blockage. However, the overall strength of the tool is lowered. Deeper flutes mean that the core diameter of the tap is smaller and, therefore, weaker and less able to handle the applied torque during the tapping operation. This, too, affects the cutting speed.



Tapmatic Corp.

Difficult-to-tap materials like titanium call for a compromise between a small rake angle, for tool strength, and a large rake angle that facilitates the cutting of longer chips. Also, it's recommended that a tap with a greater relief angle be used, to reduce friction.

Also, spiral-flute taps transport chips out of the hole better than straight-flute styles, minimizing the recutting of chips.

Rake and grind. Taps are available with grinds and rakes that are designed for specific materials. Titanium requires a compromise between a small rake angle, for tool strength, and a large rake angle that facilitates the cutting of longer chips. Some shops demand a tap with a larger radial relief than standard—as high as 40° when threading with a taper tap.

And, favorable results occur when the tool has every other tooth ground back.

Coolant. When working with exotic materials, it is vital to ensure that ample coolant reaches the cutting edge. This is necessary to remove the high levels of friction-generated heat that arise during a thread-cutting operation. In order to improve the flow of coolant, taps with interrupted threads are recommended.

And, if the diameter is large enough, you may want to consider taps that either have coolant channels or coolant-through capabilities.

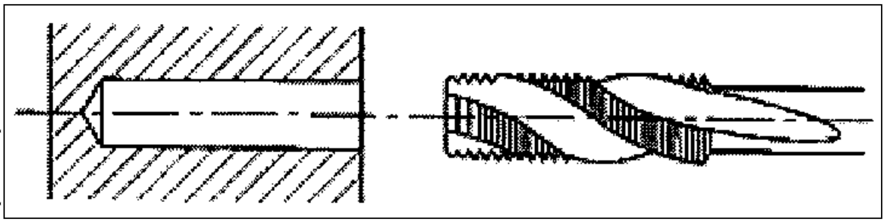
Take a Listen

An aircraft-component manufacturer that produces titanium parts for turbine engines takes a simple but effective approach to deep-hole tapping. One part the company taps is a heat shield, which begins as an investment casting that is thoroughly deburred prior to machining.

The material is Grade 7 titanium with medium hardness. The total depth of the tapped section is just less than 3", while the tap size is a nominal ½". The manufacturer found that machinists could tap the part successfully with a solid-carbide, interrupted-cut, spiral-flute tap with steel extensions on a CNC machining center at 13 sfm using flood coolant.

To ensure part conformance, the machinists remove taps from operation before they dull. Once a tap starts to dull, the probability of tool breakage in-

Scrapping a nearly complete part because of a broken tap can mean a substantial loss of time and money. Minimizing the scrap rate requires utilizing the proper tool and tapping techniques.

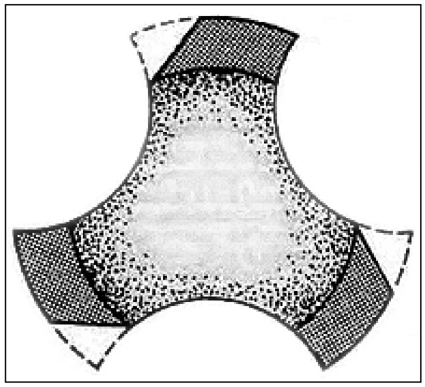


Spiral-flute taps transport chips from the hole more efficiently than straight-flute tools do.

creases, often leading to scrapped parts. Taps are removed early enough from the tapping operation to sharpen them for further use.

At the outset of a production run, machinists determine the number of holes tapped before tool wear sets in by listening to changes in the sound a tap emits as it wears. The operators swap out taps before the known number of holes are tapped.

The aerospace manufacturer has two tapping stations on each piece of equipment it uses for tapping. Both stations are fitted with the same size tap, allowing machinists to switch tools when one wears. Also, at the beginning of a shift, maintenance replaces both taps, if doing so will minimize the number of times worn taps will need to be replaced during a production run.



With titanium, it's important to use an interrupted-thread tap that has alternate teeth removed. Additionally, the greatest chip clearance is obtained by grinding a large chamfer in the tap's trailing edge.

Although changing out taps that still have life in them may not appear to be cost-effective, the practice has merit. The company spends nearly $100 to replace an $84 tap. However, that same tap can be resharpened once for about $10 and used a second time, saving $74 in tap purchases. If the tap were to break, it would cost $84 for a new tool. But the company would more than recoup its money by saving a $4,500 heat shield from being tossed in the scrap bin.

About the Author

Peter Fretty is a contributing editor to CUTTING TOOL ENGINEERING who has over 15 years of metalworking experience in application support, sales and marketing, and operations management.