TRAK® MACHINE TOOLS
SOUTHWESTERN INDUSTRIES, INC.

January 3, 2022

White Paper

What is the largest tap you can use on the TRAK LPM?

As most of you know, there are many factors that come into play when determining how big of a cut you can make on any machine tool. This is no different when it comes to tapping. The following is a long list of factors that will determine just how big of a tap you can get away with.

1. Material type – in general soft materials like aluminum and brass are much easier to tap and hence require less torque than steels or really hard materials like Inconel
2. Material Hardness – the harder the material, (for example as measured by Briness Hardness BHN) the more torque that is required to tap the hole
3. Tap Types – different types of taps will require more or less torque. Below is a summary of the typical taps you will find.
4. Sharpness of Tap – Believe or not, this is a big factor when it comes to the maximum sizes you can attain. Typically a dull tap will take 50% more torque than a new one.
5. Depth of Hole you are tapping – the deeper the hole the more torque it will take to tap the hole.
6. Tap Pitch – courser taps like ¾-10 will require significantly more torque than a fine thread like ¾-16. The torque is affected by the square of the pitch.

Torque = K × p2 × d

where K depends on:

- the material (2 to 3 times lower for light alloys than for steels)
- the type and therefore the geometry of the tap
- the diameter of the pilot hole

7. Lubricant used – the type of lubricant you use will also affect the outcome. Different materials perform better with different lubricants. The cutting lubricant also plays an important role of clearing away chips, which can cause the torque required to be higher.

8. Tap Drill Size – the small the hole you drill the more torque that is required. The drill size you use may vary depending on the class of thread you are trying to produce.
9. Thread Forming – in general, thread forming can require up to 50% more torque versus cutting the thread.

Recommended Tapping Speeds

Terminology

- SFM – surface feet per minute
- RPM – revolutions per minute
- D – tap diameter
- Ipr – inches per revolution
- Ipm – inches per minute

Formulas

- RPM = 3.82 x SFM/D
- Feed (ipr) = 1/pitch
- Feed (ipm) = ipr x RPM

The following table represents the recommended tapping speeds for general purpose taps.

Material	**Tapping Speed (SFM)**
Aluminum	90 - 110
Brass	80 - 100
Bronze	40 - 60
Copper	70 – 90
Copper – Beryllium	40 – 50
Inconel, Hastalloy, Waspalloy	5 – 15
Iron – Cast	65 – 75
Iron – Malleable	30 – 60
Magnesium	90 – 110
Plastic	60 – 90
Steel – Cast	30 – 40
Steel – Free Machining	50 – 80
Steel – Chromium	25 – 40
Steel – Alloy	20 – 35
Steel – Stainless	15 – 30
Titanium	10 – 25
Zinc – Die Cast	80 - 120

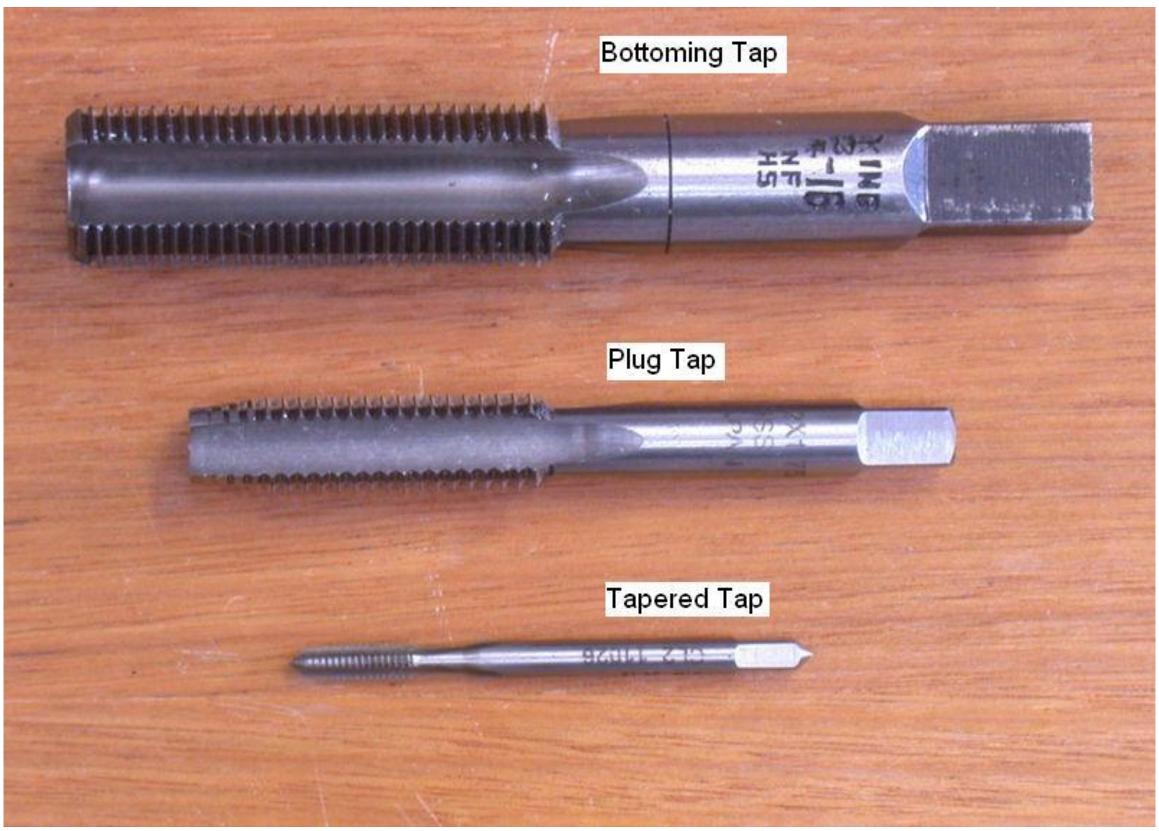
The Different Types of Taps

As mention above, there are quite a few different types of taps you can use to tap a hole. It is important to understand the differences because some taps require significantly more torque to cut than others. The following is a summary of the various taps you can buy.

Chamfer Length on Taps

1. Taper taps – have a chamfer angle that reduces the height of the first 8 to 10 teeth. This tap would require the least amount of torque.
2. Plug taps - have a chamfer angle that reduces the height of the first 3 to 5 teeth
3. Bottoming tap - have a chamfer angle that reduces the height of the first 1 ½ teeth. This tap would require the most torque.

The following taps are straight flute taps. The 1st 2 taps have 4 flutes. In general 4 flute taps are for hand tapping and should not be used on CNC machines. Most taps will come with 2, 3 or 4 flutes.



4. Spiral Point taps – these work well at removing long continuous chips from ductile materials. Most spiral taps are of the plug type, but some come as bottoming. The tap in the 2nd picture is good for tapping deep holes since it will push the chip out of a through hole, but this tap also has what is called a neck relief directly behind the cutting teeth to help evacuate chips.

5. Spiral fluted taps – these have a helical flute with helix angle being between 15 and 52 degrees and the land of the helix is the same as that of the threads on the tap. They are ideally suited for tapping blind holes and they are available as plug or bottoming types. A higher helix angle is suited for very ductile materials. For harder materials, chipping at the cutting edge may result and the helix angle should be reduced. In general, these types of taps will require less torque than a straight flute tap.

6. Roll or Form Taps – these taps do not cut material, they actually form the thread and generally require more torque to do so. The following picture shows various types of roll or form taps.

Maximum Recommended Tap Sizes

The following table illustrates the approximate tap sizes you can attain on the TRAK LPM. Once again, the data below assumes the best cutting conditions. The recommendations in the table below were made by calculating the estimated torque required for a given tap in a given material and then factoring in the maximum spindle torque of the LPM.

WARNING!
These approximations are based upon theoretical values and are intended as recommendations only. Actual results may vary.

Yellow – all style of taps should work
Orange – will require a sharp spiral fluted tap
Red – exceeds the capacity of the LPM

Tap Size	Brass	Aluminum & Leaded Brass	200 BHN Steel	300 BHN Steel	400 BHN Steel
#6					
#8					
#10					
¼”					
5/16”					

3/8”					
7/16”					
½”					
9/16”					
5/8”					
¾”					
7/8”					
1”					
1 ¼”					
1 ½”					
1 ¾”					
2”					

Important Notes

- The recommendations found above assume a 65% thread depth. This means you are using 65% of the thread found on the tap you are using. Threads that are deeper than this will require more torque.

The following table gives you an idea of how hard it is to tap other materials relative to 1010 mild steel, which has a Brinell Hardness (BHN) of approximately 115. For example, the table below says that Titanium takes 40% more torque to tap than 1010 steel.

Material	**Factor**
Aluminum	0.2
Brass	0.4
Bronze	0.4
Cast Iron	0.6
Copper	0.5
Magnesium	0.5
Malleable Iron	0.7
Zinc	0.4
Titanium	1.4