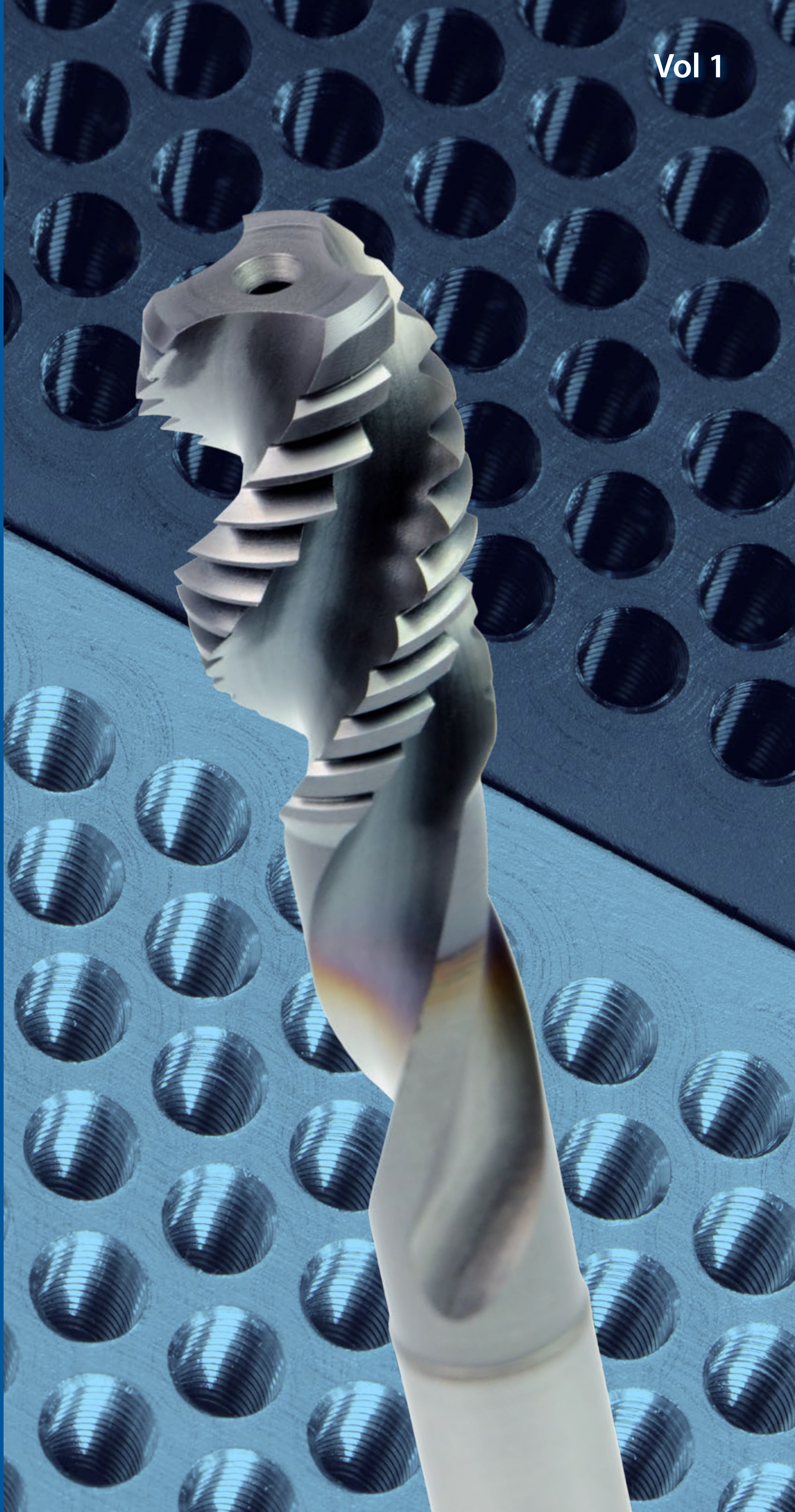
TECHNICAL GUIDE
TAPPING



For more information
scan the QR code to visit:
osgtool.com/threading

Part 1: Tap Basics 3

1.1 What is a tap 3
1.2 Features of tapping. 3
1.3. Tap Anatomy. 3

Part 2: Tap Function. 4

2.1 Thread 4
2.2. Chamfer lead. 4
2.3 Flute 7

Part 3: Tap Types And Features. 9

Part 4: Tap Class 9

4.1 Effective diameter 9
4.2 Outer diameter. 9
4.3 Valley diameter 11
4.4 Pitch 11
4.5 Thread angle 11

Part 5: Tap Substrate Material 12

5.1 High Speed Tool Steel (HSS) 12
5.2 Powdered HSS (PM) 14
5.3 Cemented Carbide 15

Part 6: Tap Surface Treatment 16

6.1 Steam oxide treatment 18
6.2 Nitride treatment. 19
6.3 Coating treatment. 21

Part 7: Cutting Torque 22

7.1 Empirical formula for calculating maximum cutting torque. 23
7.2 Calculation of required power 25
7.3 Factors affecting cutting torque. 26
7.4 Cutting torque of taper tap for pipe 27

Part 8: Tapping Condition 27

8.1 Work material 28
8.2 Screw pilot hole diameter 30
8.3 Cutting fluid 31
8.4 Cutting speed 33
8.5 Equipment and machinery 38

Part 9: Roll Form Taps. 40

9.1 Thread structure and features. 40
9.2 Manufacturing range. 40
9.3 Tap types and main applicable materials . . 40
9.4 Usage. 42
9.5 Tapping torque. 45
9.6 Female thread chevron 46
9.7 Threaded part accuracy 47

Part 10: Measurement of Female Thread with Thread Gauge 50

Part 11: Tap Regrinding 51

11.1 Regrind timing 51
11.2 Regrind part 51
11.3 Regrind of groove 51
11.4 Regrinding of chamfer. 53
11.5 Spiral tap regrind. 54
11.6 Point tap regrind 54

Part 12: Troubleshooting Tapping Problems. 55

Part 13: Threaded Hole Diameter . 57

Part 1: Tap Basics

1.1 What is a tap?

A tap is a cylindrical or conical tool with threads around its outer diameter that cuts or forms internal (female) threads in a previously prepared (typically drilled) hole. Tapping is the machining process for cutting internal threads by combining rotational speed, axial feed, and thread lead or spacing.

1.2 Features of Tapping

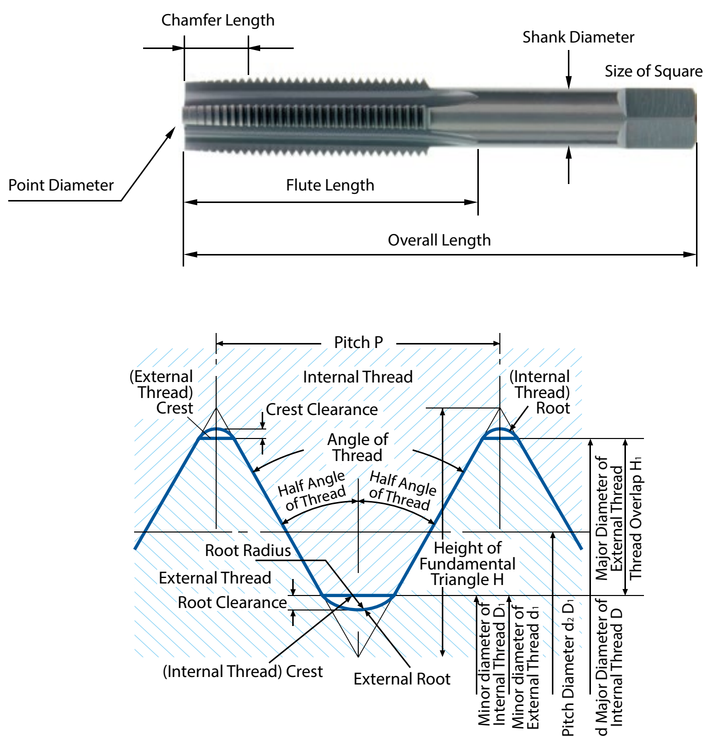
Table 1 – Benefits and Drawbacks

Comparing with other female threading process	• Produces highly accurate threads efficiently • Inexpensive due to mass production of tools • Easy to make a thread and does not require specialized knowledge • Easy to thread into large or complex workpieces • Can be processed by both mechanically and manually
Compared with other cutting processes	• Chips are easily packed or tangled • Selection range of cutting conditions is small • Tap accuracy cannot be adjusted • Taps need self-guiding portion • Machine reversal mechanism required • Process is easily affected by the pilot hole

Although the tapping method is simple, it is important to properly select the type of tap and set the cutting conditions to avoid damage that is typical and unique to taps.

1.3 Tap terms

Table 1 – Tap Dimensions

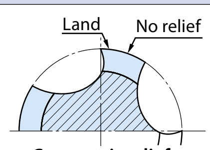
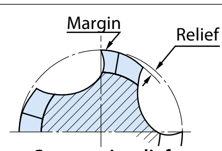
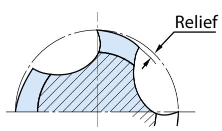


Part 2: Tap Function

2.1 Thread

The chamfer portion of the tap – generally the first 2-5 pitch lengths – performs the entirety of the cutting operation in threading. All threads after the chamfer portion serve as guides to keep threading accurate. In some cases, such as tapping materials with poor machinability or tapping large diameter threads, the resistance between the tap and the internal thread can be extreme. When this occurs, welding between the tap and the work material can occur causing damage to the internal threads. To prevent such occurrences, taps are designed with a thread relief, which is a partial removal of concentric tool material following the cutting edge to provide clearance between the tap and work material. In addition, there is usually a back taper (a slight decrease in the major diameter of each thread) after the chamfer that continues towards the shank end which provides additional clearance.

Figure 3 – Types of Thread Relief

Type of Tap Relief	Features
 Concentric relief	• Small diameter tap less than M10 • Excellent self-guidance
 Concentric relief	• M10 or larger taps • Less contact with internal threads • Excellent self-guidance
 Eccentric relief	• Minimal contact with internal threads • Suitable for stainless steel and high hardness materials • Suitable for synchronized machines • Inferior self-guidance

2.2 Chamfer

2.2.1 Length and Angle of Chamfer

The cutting by the tap is entirely done by its chamfer. The sharpness, tool life, precision of the thread, and surface finish are all affected by the chamfer. Thus, careful selection of the appropriate chamfer length is extremely important. For through holes, a longer chamfer is preferred, and for blind holes a shorter chamfer is preferred, as blind holes do not leave much clearance. If the prepared holes are long enough, it is recommended to have a longer chamfer length, even for blind holes. Table 2-2 shows the chamfer length of standard OSG taps, and Fig 4 shows the standard length and angle of slopes for various types of chamfers.

Figure 4 – Chamfer Length

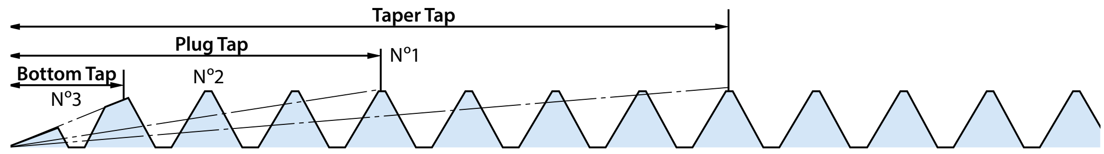


Figure 3

Tap type	Chamfer Length	Angle
Hand tap (taper)	9 threads	4°
Hand tap (plug)	5 threads	7.5°
Hand tap (bottom)	1.5 threads	24°
Nut tap	75% of thread length	1.5°
Pipe thread taper tap	2.5 threads	20°
Pipe thread parallel tap	4 threads	11°
Spiral flute tap	2.5 threads	15°
Spiral point tap	5 threads	7.5°

2.2.2 Mechanism of Tapping

In general, tapping of the internal thread is done by the cutting edges of the chamfer, which are truncated threads at the front of the tool. As stated in section 2.2.1, the threads after the chamfer act as a guide inside the previously cut internal threads. This mechanism is referred to as self-guiding.

As the tap revolves, the cutting edges of the chamfer sequentially cut out smaller pieces of the thread while the tool moves forward. The entire chamfer is used to cut out the complete thread form.

Fig 5 is an example of how a 5-thread-long chamfer on a four-fluted tap cuts a full thread form.

Fig 5 – Cutting Process of Tap (in case of 4 flute)

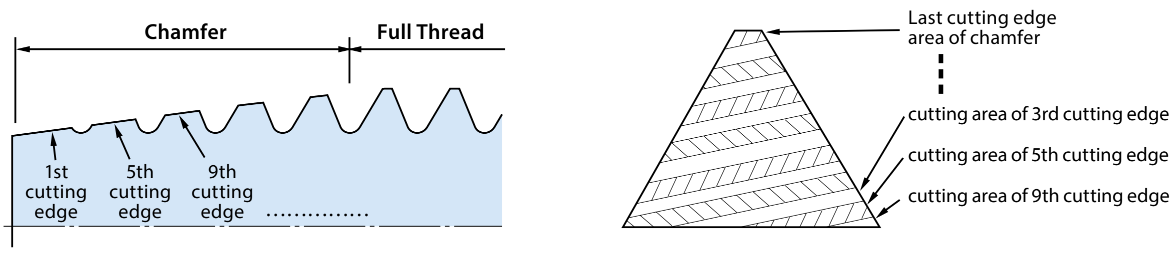
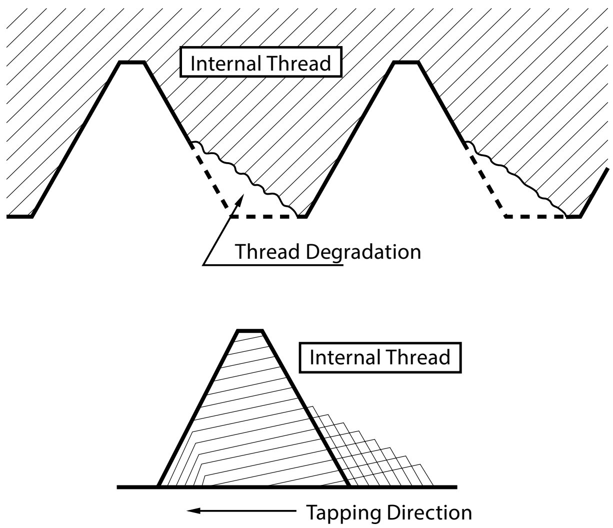
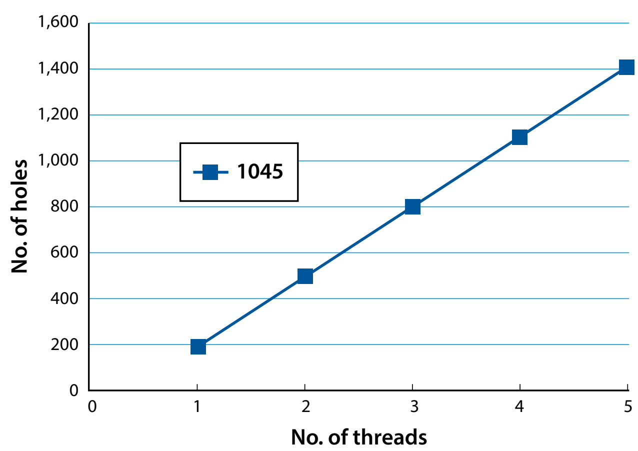


Fig 6 – Internal Thread – Overfeed of Tap



By making the chamfer length longer, or by increasing the number of flutes, the number of cutting edges will increase. Table 3 shows how the nominal diameter, number of flutes, and the length of the chamfer affect the cutting amount.

Fig 7 – Length and Tool Life of Chamfer



Tap:	Spiral Fluted Tap M10×1.5
Tap Drill:	φ8.5×20mm
Tap Length:	20mm
Speed:	315min-1
Coolant:	Non-soluble Cutting Fluid
Machine:	CNC Drilling Machine

Table 3 – Tap Cutting Amount Thread per Tooth

unit:mm

Tap Type \ Thread Size	M3×0.5	M6×1	M10×1.5	M24×3	M48×5
No. of Flutes	3	3	4	4	4
Hand tap (bottom)	0.012	0.023	0.026	0.052	0.087
Hand tap (plug)	0.022	0.044	0.049	0.098	0.163
Hand tap (taper)	0.068	0.136	0.153	0.305	0.508
Nut tap	0.004	0.009	0.01	0.02	0.033
Spiral flute tap	0.043	0.086	0.097	0.194	0.324
Spiral point tap	0.028	0.055	0.062	0.124	0.206

2.3 Flute

Like other cutting tools, taps need appropriate rake angles based on the work material. Normally if the rake angle is high the cutting edge gets sharper and the finish is cleaner, but on the other hand the edge is easier to break, and the precision of the internal thread is unstable.
Therefore is it customary to set the rake angle high for soft materials to focus on the sharpness, and to make the angle lower for hard materials to prevent breakage.

The cutting edge of taps has either a rake style or a hook style (Fig 8) and are chosen along with the rake angle. Rake style is set to increase the toughness of the tool, whereas the hook style is used to increase the sharpness.
Because of this, it is typical to use the rake style for hard materials and hook style for soft materials.
The flutes of a tap have three basic categories based on the tap style. Straight flutes have the flutes parallel to the length of the tool, spiral fluted taps are twisted along the length of the tool, and spiral pointed flutes are twisted only within the chamfer area. (See Table 8 on page 21)

Table 3 – Rake Angle by Material

Work material	Rake angle
Low carbon steel	10~13
High carbon steel	5~7
Tool Steel	5~7
Stainless steel	10~13
Chrome steel	10~13
Manganese steel	10~13
Cast steel	10~16
Cast iron	2~4
Aluminum	16~20
Aluminum alloy	12~14
Copper	16
Brass	3~5
Bronze	1~3
Synthetic resin	3~5

Fig 8 – Rake Style and Hook Style

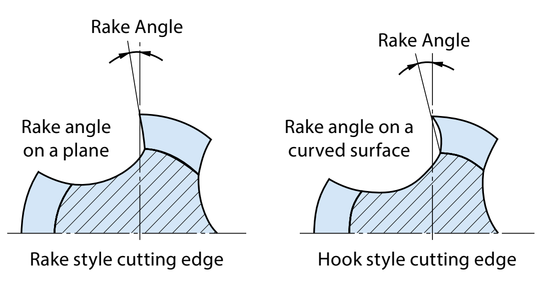
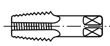
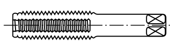
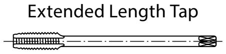
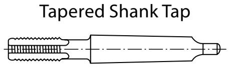
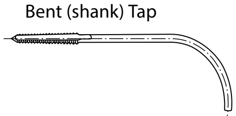
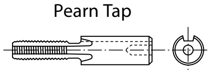
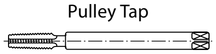
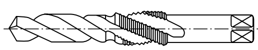
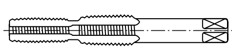
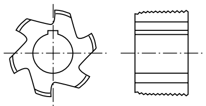
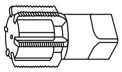


Table 8 – Classification and Features of Flute Design

Classification	Features	Application
Straight fluted tap 	• Straight flute • Strong cutting edge • Multiple options of chamfer length • Easy to regrind	• High hardness work material • Work material that easily causes tool wear • Short-chipping materials • Through holes, shallow blind holes
Spiral Point tap 	• Point flute • Evacuates chips forward, into hole • No chip packing around the shank • Highly resistant to breakage and failure • Sharp cutting edge	• Long-chipping and soft material • Through hole • High-speed tapping
Spiral fluted tap 	• Spiral flute • Threads close to bottom of blind holes • Chips are evacuated out of the hole • Easily able to engage into the pre-tap hole • Sharp cutting edge	• Long-chipping and soft material • Blind hole
Roll form tap (form tap) 	• Forms internal threads by deformation of the material • Does not produce chips • Internal thread accuracy is stable • Highly resistant to breakage and failure	• For material that readily deforms without breakage • Can be used for both through and blind holes

Table 9 – Tap Types

Classification	Types
Pipe Thread Tap	 for Tapered Pipe Threads  for Parallel Pipe Threads
Pipe Thread Tap	Extended Length Tap  Tapered Shank Tap  Bent (shank) Tap  Pearn Tap  Pulley Tap 
Combination Tap	 Drill Tap  Multi-step Tap  Guided Tap  Reamer Tap
Exchangeable Thread Tap	 Shell Tap  Indexable Tap

Part 3: Tap types and features

Depending on the application, there are many suitable types of taps to be chosen from. The various classifications are divided into flute style and shank geometry. As taps must be used in pre-existing holes, the primary factor in selection of basic geometry is the method by which chips will evacuate. By understanding the type and style of a tap one can select the appropriate tool for any threading application. Table 8 shows the classification by flute types, and Table 9 shows various tap types. Additional options, such as tapered pipe taps and thread milling cutters, also may be applicable.

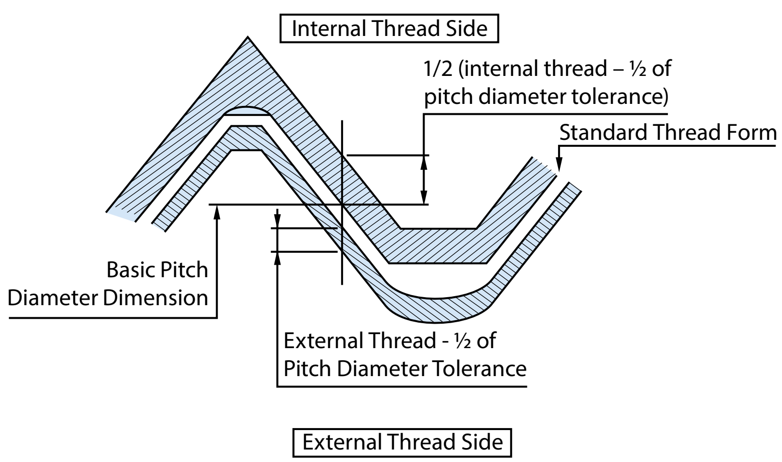
Part 4: Tap class

In determining the accuracy of the internal thread, the pitch diameter of the tap is one of the most important factors. It is defined as "The diameter of a virtual cylinder (or circular cone) that is formed when the height of the crest and the height of the root are equal."

Typically the accuracy that is often used for taps is ANSI 2B or ISO 6H, but this makes the range of selection very narrow. Thus a system of tolerance is in use to define the accuracy and size of every tap – for ANSI, these are H-limits, while ISO uses D-limits.

Fig 9 shows the positional relationship of the internal and external thread used to calculate the pitch diameter, and Fig 10 shows the relationship of the internal thread and taps used to calculate the pitch diameter.

Fig 9 – Pitch Diameter Position Relationship Between Internal and External Thread

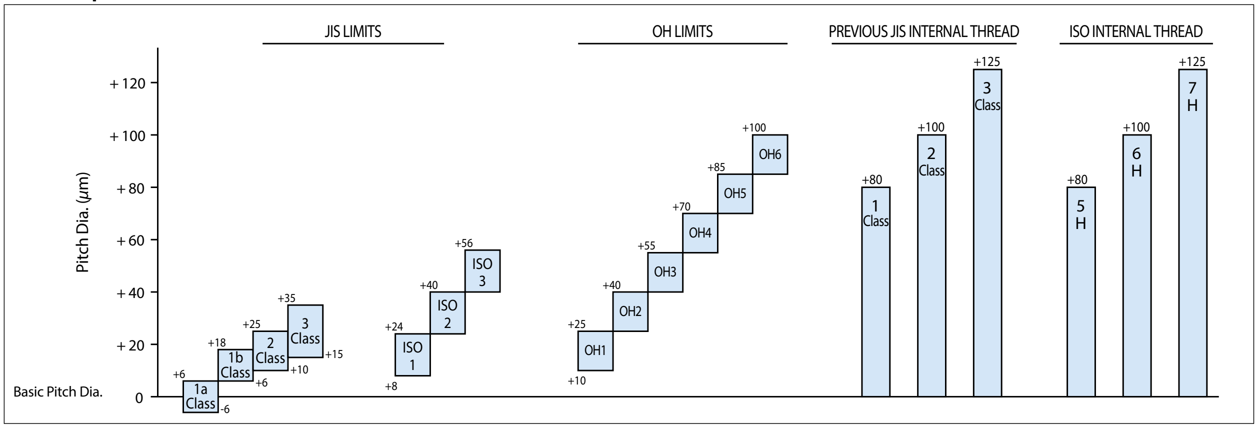


OH Limit

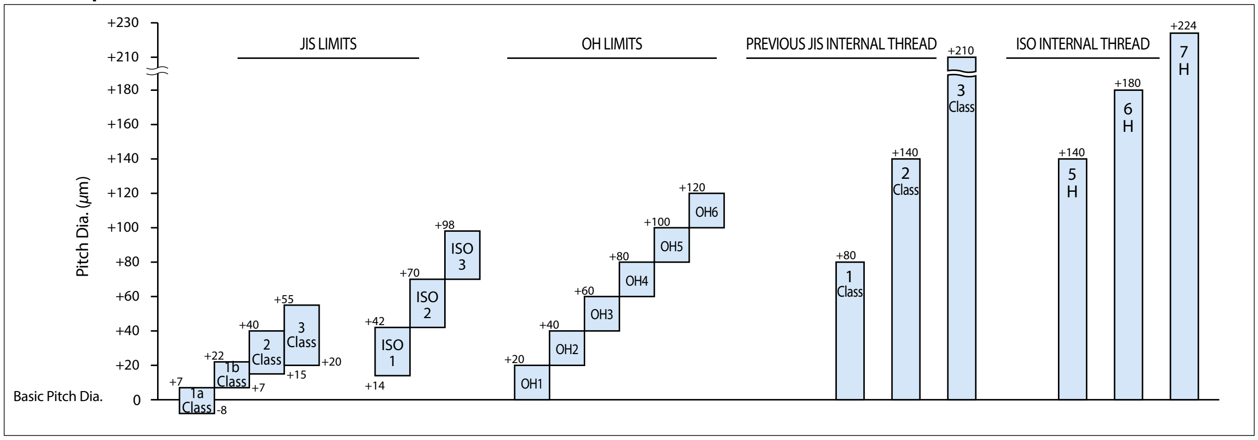
1. P≦ 0.6 (T.P.I. ≧ 40)	2. P≧0.7 (T.P.I. ≦ 36)
Upper Limit: 0.010＋0.015×n	Upper Limit: 0.020×n
Lower Limit: (upper limit) −0.015	Lower Limit: (upper limit) −0.020

Unit:mm (n=OH number)

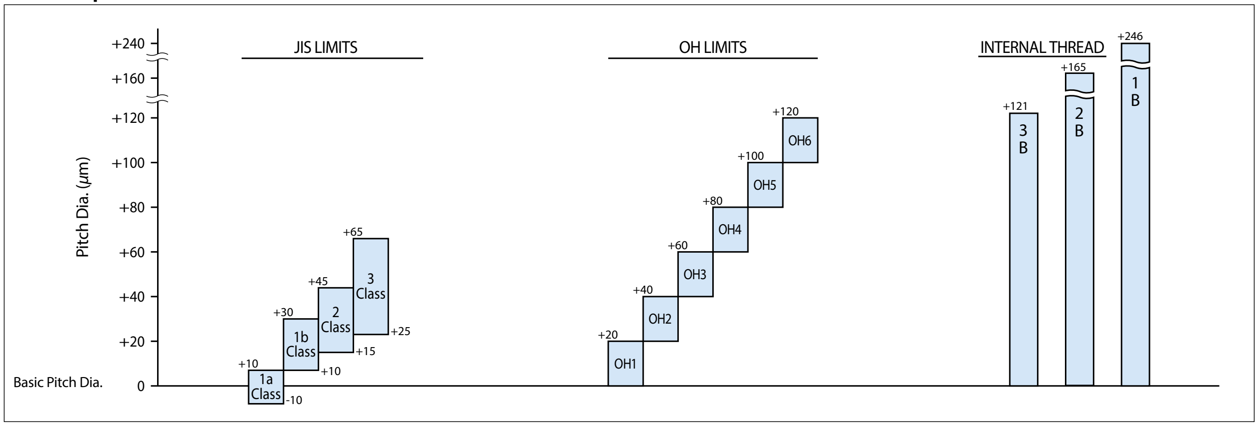
■ Example M2×0.5



■ Example M10×1.5



■ Example ½-13UNC



4.2 Major Diameter

The diameter of the cylinder (or circular cone) that runs along the largest diameter of thread is called the major diameter. The tolerance of the major diameter is set to create a slightly larger tool than the nominal size in order to avoid a potential interference fit with a fastener. Taps generally have their crests cut off, or truncated, in order to provide a longer tool life by removing a sharp point.

4.3 Minor Diameter

The diameter of the cylinder (or circular cone) that runs along the bottom of the thread is called the minor diameter. The minor diameter of the tap does not cut or otherwise interact with the drilled hole while threading.

4.4 Pitch

In the axial cross section of the tap, the distance between two equivalent points of adjacent threads is called the pitch.

4.5 Thread Angle

In the axial cross section of the tap, the angle that the two adjacent flanks form is called the angle of the thread. It may also be called the full angle of the thread.

Part 5: Tap Substrate

5.1 High Speed Tool Steel (HSS)

As tap substrates improve, the trend has been to move from tool steel (SKS) to high speed steel and, eventually, to carbide. SKS taps are still used predominantly in manual operations and small production lots and is typically not used in mass production atmospheres, such as automotive, that require high efficiency and reliability. The primary difference between SKS and HSS materials is their hot hardness abilities. During tapping, heat is elevated and concentrated at the cutting edge, which causes wear and dulls the edge. This heat is greater during longer tapping cycles and deeper threads. HSS taps are capable of holding their hot hardness up to 600ºC, so they are less vulnerable to wear during prolonged usage (see Fig 12).

Compared to high tungsten HSS tools, high molybdenum HSS tools have better toughness. Because of this, Mo taps are used when sharper edges are required, but can chip or fracture somewhat more easily. This downside has caused the popularity of the M7 substrate to rise, but currently vanadium-added tools are more popular because of their high wear resistance, allowing for higher performance.

The addition of cobalt in a HSS composition provides increased hardness at high temperatures and increased heat resistance capabilities. Adding vanadium also increases hardness at high temperatures, but the resulting substrate makeup has the effect of increasing the overall abrasion resistance. Therefore, when tapping heat-treated or hardened materials, a HSS tap with added vanadium (OSG's HSSE) is the ideal choice.

Fig 12 – Comparison of High Temperature Hardness

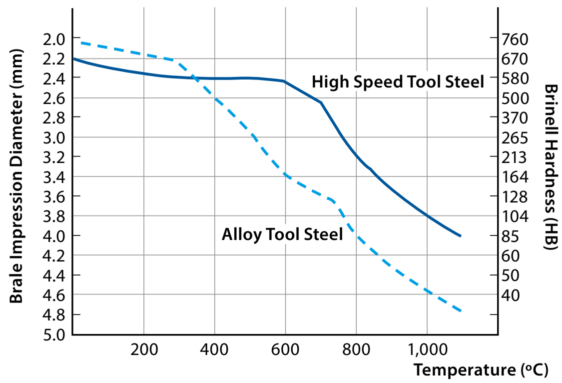


Table 12 – High Speed Tool Steel for Taps

Classification	Symbol		Chemical composition (%)					
	JIS	AISI	C	Cr	Mo	W	V	Co
W type	SKH 2	T1	0.8	4.0	–	18.0	1.0	–
	SKH 3	T4	0.8	4.0	–	18.0	1.0	5.0
Mo type	SKH51	M2	0.8	4.0	5.0	6.0	2.0	–
	SKH52	M3-1	1.05	4.0	5.0	6.0	2.4	–
	SKH53	M3-2	1.2	4.0	5.0	6.0	3.0	–
	SKH55	M35	0.8	4.0	5.0	6.0	2.0	5.0
	SKH56	M36	0.9	4.0	5.0	6.0	2.0	8.0
	SKH58	M7	1.0	4.0	8.8	1.8	2.0	–
	–	M41	1.1	4.3	3.8	6.8	2.0	5.0
	SKH59	M42	1.1	3.8	9.5	1.5	1.2	8.0
CPM	SKH10	T15	1.5	4.0	–	12.0	5.0	5.0

5.1.1 Characteristics

• As can be seen from Fig 13, Fig 14, HSSE is a high grade high speed tool steel that has the necessary toughness and abrasion resistance needed for taps.

• Marked as "HSSE" to differentiate from the regular HSS. (Example) M10X1.5 OH3 OSG HSSE.

Fig 13 – Mechanical Properties

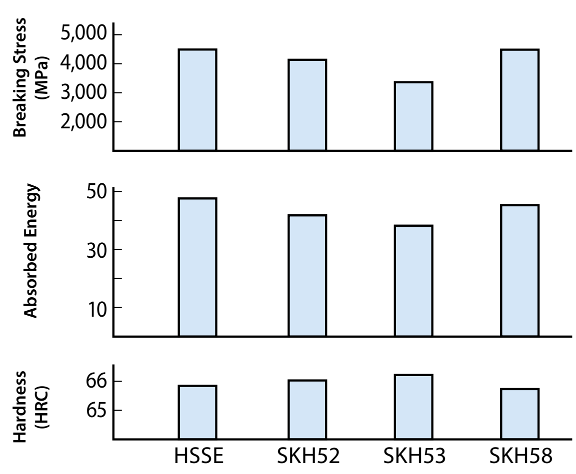
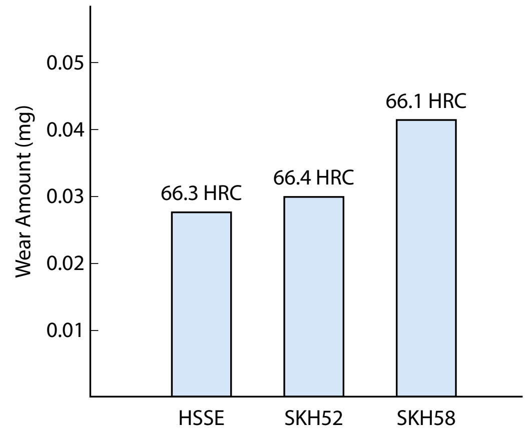


Fig 14 – Wear Resistance



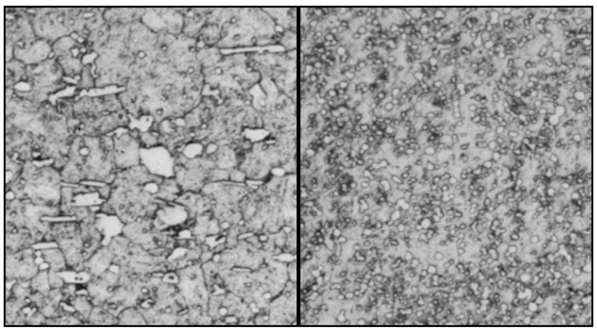
Test piece: φ5×70ℓ **Load:** 10kg
Sandpaper Type: SiC500# Grit
Speed: $980min^{-1}$ **Feed:** 60mm/min
Friction Length: 1000mm

5.2 Powdered Metal Steel

The idea of increasing the vanadium content in a substrate to increase its performance has been experimented with for some time. Instead of standard steel processing by remelting, the powder metallurgy method can allow for elevated vanadium content without decreasing the tool's grindability for manufacturing.

Page 25, Table 12 shows the composition chart for high speed tool steel (Powdered HSS) made by the powder metallurgy method. In the chart, PM T15, a typical material for tap steel, has 5% vanadium and 5% cobalt content. Figure 15 is a microscopic view of the composition. One can see the carbide grains are very fine and spread out evenly throughout the substrate for the powdered HSS. Table 16-1 shows the experimental data of the effect that the tap substrate has, based on the work material and coolant oil. The diagram reveals that when the work material is hard, the higher the vanadium content of the tool, the longer the tool life becomes. Also, non-water soluble coolant oil works better than water soluble coolant oil. A substrate with 5% vanadium (CPM T15), has approximately 3 times the tool life of a substrate with 2% vanadium (M7). High alloy powdered HSS substrates are under continual development to improve heat and abrasion resistance for taps.

Fig 15 – Microscopic Structure (×800)



Standard HSS | Powdered HSS (CPM)

Fig 16-1 – Tap Material and Life

Tap	Straight flute tap M10×1.5OH2 Plug tap			
Work Material	S45C Quenched and Tempered 34~35HRC		SCM440 Quenched and Tempered 39~40HRC	
Cutting Speed	13.7m/min (435min^{-1})		9.9m/min (315min^{-1})	
Cutting Fluid	Non-water Soluble Cutting Fluid	Water Soluble Cutting Fluid	Non-water Soluble Cutting Fluid	Water Soluble Cutting Fluid
Machining Method / No. of Holes / Material	Machine: NC Drill Machine RND1003; Endurance Limit; Breakage			
	200 400 600 800 X	200 400 X	100 200 300 400 X	100 200 300 X
SKH58 (V2%)	172, 227, 272; X: 224	60, 114, 145; X: 109	128, 84, 89; X: 100	35, 37, 65; X: 47
SKH53 (V3%)	488, 480, 424; X: 464	330, 222, 202; X: 251	236, 182, 145; X: 188	67, 92, 42; X: 67
CPMT15 (V5%)	590, 563, 641; X: 598	330, 353, 472; X: 385	260, 281, 329; X: 290	122, 108, 120; X: 117

Recently, OSG has developed a new HSS XPM grade for additional wear resistance in especially high-hardness materials. This material is a high-grade powdered HSS containing 5% vanadium and 10% cobalt and is used for taps for high hardness steel such as V-XPM-HT and V-XPM-TPT; the high hardness of the substrate enables tapping 42-52 HRC hardened steel.
Fig 16-2 shows the tool life/durability of V-XPM-HT

Fig 16-2

Tap	Surface Treatment	Machined Holes (10 30 20 40 50)	Cutting Condition
Straight Flute Tap for Hardened Materials V-XPM-HT M6×1 2.5 XPM OH3	V coating (TiCN)	51 Holes	Work Material: H13 50HRC Tap Drill: φ5×25mm (thru hole) Speed: 2.3m/min Cutting Fluid: Non-water Soluble Cutting Fluid

5.3 Carbide

In order to lengthen the life of taps, carbide alloys have been used more frequently. Fig 17 shows the data for cutting Cast Iron, which illustrates tool life up to 30,000 holes. The increase in cutting torque due to wear is significantly lower than HSS taps, and tool life is over 10 times longer. Carbide taps can be seen regularly in mass production atmospheres cutting cast iron and aluminum materials.
Carbide substrates are ideal for cast iron, as well as most non-ferrous (aluminum, magnesium) or non-metallic materials (graphite, thermosetting plastic). Additionally carbide substrates are good for steels over 50HRC, which are either too hard or too abrasive for HSS taps. The carbide alloys that are used in OSG taps are premium, ultrafine-grain carbides with high rigidity.
As seen in Fig 18, carbide taps are more susceptible to breakage problems than HSS taps. When using carbide taps, one must be careful the tap and the prepared hole are properly aligned to minimize torque. However, with the development of advanced carbide grades, carbon steel tapping is possible despite the higher torque involved.

Fig 17 – Carbide Tap Performance

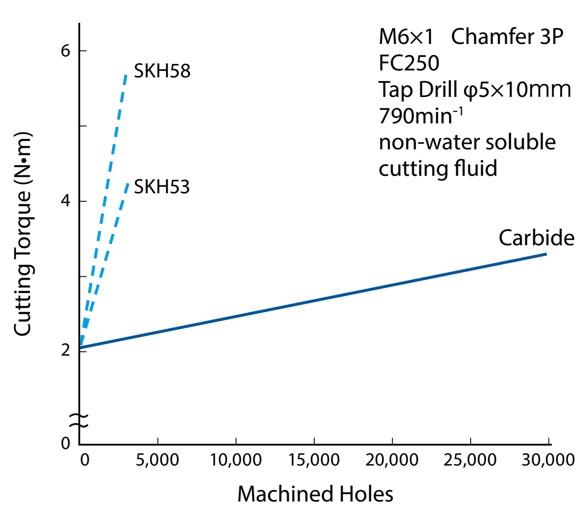
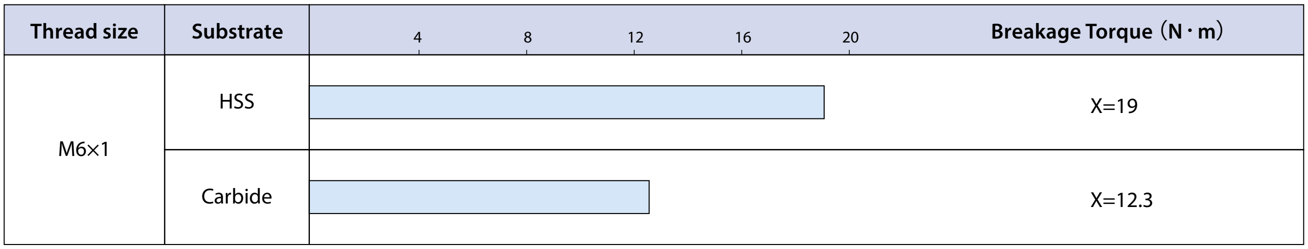


Fig. 18 – Breakage Torque of Carbide Tap



Part 6: Coating and Surface Treatments

The performance of a tap is affected by its form, substrate, and heat treatment, but the coating can have a drastic effect upon tool life as well. For work materials with a high affinity to welding or shrinkage, like stainless steel, coating is often a necessary addition.

Some advantages of coatings are:
•Strengthens the surface hardness.
•Increases the abrasion resistance.
•Prevents of adhesion and welding of the tool and work material.
•Reduces of the coefficient of friction between the tool and work material.

To fit these requirements, various coatings have been developed. But, since the metal tempering temperature (the process of heating the steel to a certain temperature in order to make the quenched steel tougher) is 550ºC to 580ºC, the coating process must be done below this temperature.

Table 13 – Types and Characteristics of Surface Treatment

OSG Name	Color	Coating Composition	Coating Hardness (Hv)	Coefficient of Friction	Oxidation Temperature (°C)	Characteristic • Uses	
Steam Oxide	Black	Fe_3O_4	———	———	———	• Surface treatment, no coating added – retains the cutting edge sharpness. • 1-3µm treated depth . • Porous surface which retains cutting fluid . • Reduces coefficient of friction.	• Prevents welding. • Not suitable for non-ferrous materials. • Suitable for easy-to-weld materials (Stainless steels, Titanium alloy, SS400 (400 series SUS) , S15C (1015 low carbon steel), and other soft steels.
Nitride Treatment	———	Substrate Penetration	1,000~ 1,300	———	———	• Surface treatment, no coating added – retains the cutting edge sharpness. • 30-50µm treated depth.	• Improved wear resistance. • Suitable for abrasive work materials (cast iron, silicon alloy, thermosetting resin).
TiN Coating	Gold	TiN Single Layer	2,000	0.4	500	• Coating thickness 2-5µm. • Improved wear resistance. • Reduction in coefficient of friction. • Prevents welding. • Improved heat resistance.	• Suitable for easy-to-weld and abrasive work materials (400 series SUS, low carbon steel (1015) or soft steels S45C 1045 or hard steels Alloy tool steel, Hardened steels.
V Coating	Blue Gray	Multi-layer of TiN and TiCN	2,700	0.3	400	• Coating thickness 2-5µm. • Improved wear resistance. • Reduction in coefficient of friction. • Prevents welding. • Improved heat resistance.	• Suitable for easy-to-weld and abrasive work materials (400 series SUS, low carbon steel (1015) or soft steels S45C 1045 or hard steels Alloy tool steel, Hardened steels.
CrN Coating	Silver Gray	CrN 2 Layer	1,800	0.25	700	• Coating thickness 2-5 µm. • Improved wear resistance. • Reduction in coefficient of friction. • Prevents welding. • Improved heat resistance.	• Suitable for easy-to-weld work materials (copper and copper alloy, aluminum and aluminum alloy, Inconel and other heat resistant materials).
FX Coating/ EXO Coating	Black	Multi-layer of 2 Types of TiAlN	2,800	0.3	850	• Coating thickness 2-5µm. • Improved wear resistance. • Reduction in coefficient of friction. • Prevents welding. • Improved heat resistance.	• Suitable for high speed machining or dry machining high hardness materials (65 HRC and under hardened steel, for hard steels such as 1045 (S45C), alloy tool steel, for soft steels such as 400 SUS (SS400), 1015 (S15C)).
DLC Coating	Interference Color	DLC	3,000	0.1	300	• Coating thickness 0.2µm. • Improved wear resistance. • Reduction in coefficient of friction.	• Prevents welding. • Suitable for dry aluminum machining/applications. • Not suitable for ferrous materials.
DIA Coating	Black	Microcrystal Diamond	3,500	0.3	1,300	• Coating thickness 3-10µm. • Improved wear resistance. • Reduction in coefficient of friction.	• Suitable in high silicon aluminum. • Not suitable for ferrous materials.

6.1 Steam Oxide

Steam oxide processing is a type of oxidation treatment. The tool is heated for 30 to 60 minutes in a 500ºC-550ºC steam bath, in which a layer of Fe_3O_4 is applied.

Iron has three kind of oxides, FeO, Fe_2O_3, and Fe_3O_4. FeO can only be applied at 570ºC and thus cannot be applied in a steam bath. Fe_2O_3 (Red Rust) is not applicable due to performance. Therefore, Fe_3O_4 is the only Steam Oxide treatment allowable. The usual thickness of the coating layer is 1 to 3 µm.

6.1.1 Treatment Characteristics

The following points show the improvement in performance by Steam Oxide processing:

- The porous nature of the treatment holds the coolant oil better, resulting in a decrease of heat from friction.
- Prevents the adhesion and welding of the work material onto the tap surface.
- Reduces the internal stresses from grinding that are present on the tap surface.

Although the treatment does improve performance, the treatment does not increase the hardness of the tool, so the surface layer itself does not improve abrasion resistance.

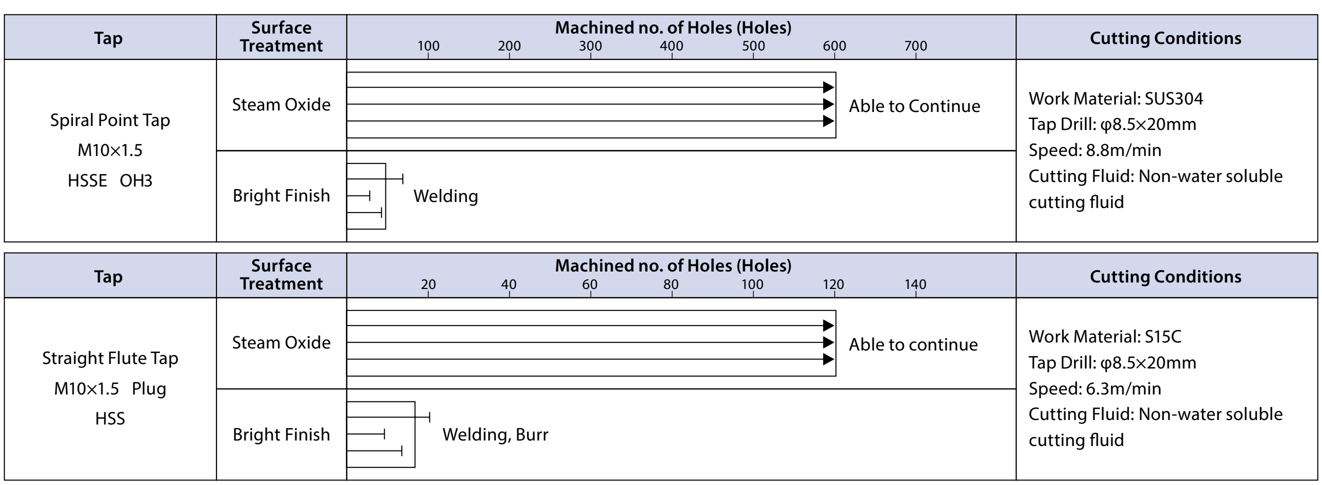
6.1.2 Range of Application

Steam oxide works well on materials that are prone to welding or steel based work materials, such as stainless steels, cast steels, low carbon steels, nickel steels, and chrome steel.

6.1.3 Cutting Performance

Fig 19 shows the cutting test for taps with a steam oxide treatment. In austenitic stainless steel the steam oxide layer provided a tool life ten times longer than conventional. In low carbon steel hand-tapping there was also a marked improvement.

Fig19 – Tool life of Steam oxide taps

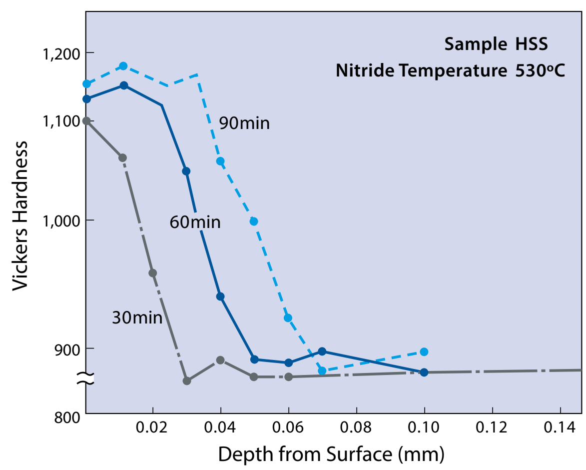


6.2 Nitride Treatment

The special nitride treatment (processed at 500ºC-560ºC for 30-90 minutes) has a smooth surface and is strongly resistant to defects. It provides an even surface hardness layer to the tool.

The hardness of the nitride layer is determined by the temperature and time of the process, but hardness of the base material must be taken into consideration. For taps the process is done between 500ºC at 30 minutes and 560ºC at 70 minutes. Fig20 shows an example of the distribution of the hardness of the Nitride layers.

Fig 20 – Hardness distribution of nitride layer



6.2.1 Treatment Characteristics

The nitride treatment can improve the tool's performance by:

•Improving the hardness thus improving the abrasion resistance
•Improving heat resistance thus allowing for higher performance at elevated temperatures.

From this treatment, the surface hardness of the tap can be increased up to 1300 HV (Vickers Hardness scale). This, when compared to the substrate's hardness of ~870 HV, is 1.5 times harder than the tap alone.

6.2.2 Range of application

Nitride-treated taps are effective for thermosetting polymers, grey cast iron, aluminum die cast, and aluminum cast metal, as these require high abrasion resistance. Steel applications must be taken with special care, as there is a possibility of chipping and breakage. The hardness is usually between 1,000 and 1,300 HV, while for steel it is usually under 1,100 HV. This treatment is effective for tools that the chips are cut short, such as with hand taps, and is commonly not applied to spiral pointed and spiral fluted taps.

6.2.3 Cutting performance

Fig 21-1 and 21-2 shows the difference in cutting torque (rotational resistance when tapping) of the taps with 1200 HV Nitride treatment and those without. The improvement in wear resistance for the nitride-treated tap is clear.

Figure 21-1 – Tool Life of Nitride-Treated Tap

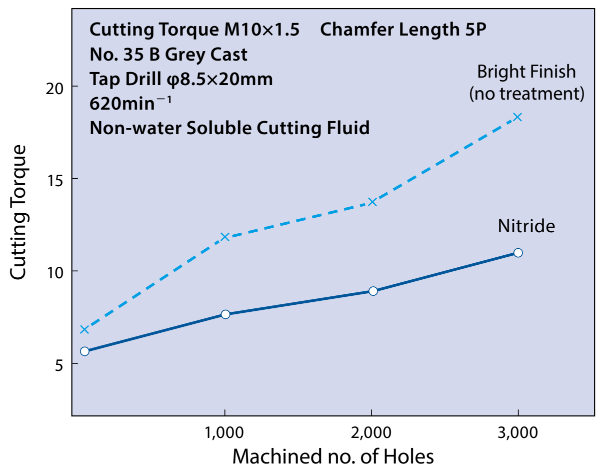


Fig 21-2 – Tool Life of Nitride Tap

Tap	Surface Treatment	Machined no. of Holes (Holes) 50 100 150 200 250 300 350	Cutting Conditions
Straight Flute Tap HT M6×1 Plug HSS	Nitride	Able to Continue	Work Material : Si Alloy Steel Tap Drill: φ5.0×20mm (Blind) Speed: 10.2m/min Cutting Fluid: Non-water Soluble Cutting Fluid
	Bright Finish	Wear	

Tap	Surface Treatment	Machined no. of Holes (Holes) 50 100 150 200 250 300 350	Cutting Conditions
Straight Flute Tap HT M4×0.7 Plug HSS	Nitride	Able to Continue	Work Material: Phenolic Resin with Glass Fiber Tap Drill: φ3.3×9mm Speed: 15m/min Cutting Fluid: None
	Bright Finish	Wear	

6.3 Coating Process

There are two major methods for the coating process, the PVD (Physical Vapor Deposition) method and the CVD (Chemical Vapor Deposition) method. Coating types such as TiN (Titanium Nitride), TiCN (Titanium carbon nitride), CrN (Chromium nitride), and TiAlN (Titanium aluminum nitride), are usually applied by the Ion Plating Method, a form of the PVD. PVD methods typically run at 550ºC or less, thus can be applied to HSS tools. Typical PVD methods include the HCD (Hollow Cathode Discharge), or the AIP (Arc Ion Plating), which produces high adhesion to the tools.

For extreme wear resistance, DLC (Diamond-Like Carbon) or DIA (Diamond) coatings may be applied.

6.3.1 Cutting Performance

The upper part of Fig22 shows the duration test results of spiral pointed taps with no coating, steam oxide, and TiN coating in precipitation hardening stainless steel (SUS630). The work material is prone to welding and has a hardness of 45 HRC, so the tools would require an anti-adhesion property and high abrasion resistance. The steam oxide tools show 3 times the tool life of tools with no coating, and the TiN coated tools show 3 times the tool life of the steam oxide tools. The lower part of Fig22 shows the difference of steam oxide and TiN coated NRT, with TiN coated taps lasting about 3 times longer.

Fig 22 – Tool Life of TIN Coated Tap

Tap	Surface Treatment	Machined no. of Holes (Holes) 100 200 300 400 500 600	Cutting Conditions
Spiral Point Tap EX-POT M10×1.5 HSSE OH3	Bright Finish	Wear, Welding	Work Material: SUS630 (45HRC) Tap Drill: φ8.5×20mm Speed: 8.5m/min Cutting Fluid: Non-water Soluble Cutting Fluid
	Steam Oxide	Wear, Welding	
	TiN Coating	Wear	

Tap	Surface Treatment	Machined no. of Holes (Holes) 1,000 2,000 3,000	Cutting Conditions
Spiral Point Tap EX-POT M10×1.5 HSSE OH3	Bright Finish	Wear, Welding	Work Material: 1045 Tap Drill: φ8.5×20mm Speed: 13.7m/min Cutting Fluid: Non-water Soluble Cutting Fluid
	TiN Coating	Wear	

Tap	Surface Treatment	No. of Tapped Holes 10,000 20,000 30,000 40,000	Cutting Conditions
Form Tap NRT M6×1 RH7 Chamfer Length 4P	Steam Oxide	Wear, Welding	Work Material: SS400 Tap Drill: φ5.5×10mm Speed: 10m/min Cutting Fluid: Non-water Soluble Cutting Fluid
	TiN Coating	Wear	

Fig 23 shows the difference in durability of the NRT with steam oxide treatment and V coating. The upper part is the result of tapping of high-carbon steel at 20m/min, which requires high abrasion resistance. The V coated tools show approximately 5 times the tool life of the steam oxide tools. The lower part is the result of tapping of mild steel with water soluble coolant oil diluted to 30 times, and the V coated tools show approximately 20 times the tool life of steam oxide tools.
Fig 24 shows that the CrN coated tools have approximately 1.5 to 6 times the tool life when working with copper. This is because the CrN coating is known to show especially high abrasion resistance and welding resistance when working with copper.

Fig 23 – Tool Life of V Coated Tap

Tap	Surface Treatment	No. of Tapped Holes (2,000 / 4,000 / 6,000)		Tapping Condition
Form Tap NRT M6×1 RH7-B	Steam Oxide	GPOUT	$\bar{X}$: 766	Work Material: 1045 91～94HRB Tap Drill: φ5.5×15mm Speed: 20m/min Thread Lenght: 10mm Cutting Fluid: non- water soluble cutting fluid
	V Coating	GPOUT	$\bar{X}$: 3,761	

Tap	Surface Treatment	No. of Tapped Holes (2,000 / 4,000 / 6,000)		Tapping Condition
Form Tap NRT M10×1.5 RH7-B	Steam Oxide	Spindle Stall	$\bar{X}$: 200	Work Material: SS400 70～75HRB Tap Drill: φ9.25×25mm Speed: 15m/min Thread Length: 20mm Cutting Fluid: Dilution rate x30
	V Coating	Breakage	$\bar{X}$: 3,915	

Fig 24 – Tool Life of CrN Coated Tap

Tap M5×0.8	Surface Treatment	No. of Tapped Holes (2,000 / 4,000)		Tapping Condition
CU-POT	CrN coating	Able to Continue	X:4,000	Work Material: C1100 (Copper) Tap Drill: φ4.2 Speed: 15.7m/min (1,000min^{-1}) Thread Length: 10mm (thru hole) Cutting Fluid: Non-water soluble cutting fluid Dilution rate x10
V-POT	V coating	Breakage	X:=2,299	
EX-SUS-POT	Steam Oxide	GPOUT	X:=1,666	

Tap M6×1	Surface Treatment	No. of Tapped Holes (1,500 / 3,000)		Tapping Condition
CU-SFT	CrN coating	Able to Continue	X:=3,000	Work Material: C1100 (Copper) Tap Drill: φ5.0 Speed: 10.6m/min (560min^{-1}) Thread Length: 10mm (blind hole) Cutting Fluid: Non-water soluble cutting fluid Dilution rate x10
V-SFT	V coating	GP OUT	X:=1,200	
EX-SUS-SFT	Steam Oxide	GP OUT	X:=600	

Part 7: Cutting Torque

The rotational resistance that occurs during tapping is called the cutting torque of the tap. This resistance occurs on the tangent of the chamfer circumference. There are numerous factors that affect the torque, and these combine to form the resistance during tapping. These include factors of the tap form, like the type of the tap, rake angle, substrate, or chamfer length. There are also factors of the cutting condition like the type of the work material, length of tapping, and the size of the drill hole diameter.

7.1 Calculation Formula for the Maximum Cutting Torque

The maximum cutting torque for tapping is calculated by the following formula:

$$Tc = \frac{\tan\theta}{24{,}000} \cdot kc \cdot K(D - Do)^2 \cdot (D + 2Do)$$

Tc: Cutting Torque (N•m)

Kc: Cutting Resistance

K: Tap Style, and constant value due to chips

D: Tap Major Diameter (mm)

Do: Tap Drill Diameter (mm)

θ: Half Angle of Thread (°)

Of these, the kc and K are shown in Table 14 and 15, and examples using the formula are shown in Table 16, 17 and Fig 25. The formula does not take into account the length of tapping.

Table 14 – Specific Cutting Resistance by Work Material

Work Material		Cutting Resistance
SK5	175HB	5300
SS400	133HB	3700
1045	141HB	3600
1035	162HB	3700
1045	188HB	3900
1055	188HB	4000
4140	193HB	3600
4140	30HRC	4900
4140	40HRC	5500
SUS304	209HB	4200
Brass	–	2300
Cast Aluminum	–	1300
No. 35 B Grey Cast	193HB	2900

Table 15 – Tap Shape Constant Value

	Constant Value					
	Steel		Cast Iron, Aluminum alloy		Brass	
	Coarse	Fine	Coarse	Fine	Coarse	Fine
Spiral Point Tap	0.95	1	0.8	1	0.75	1
Spiral Flute Tap	1.15	1.25	1.05	1.1	0.85	–
Straight Flute Tap (Taper)	0.95	1.20	–	–	1.20	–
Straight Flute Tap (Plug)	1.35	1.15	1.25	1.08	1.60	1.1
Straight Flute Tap (Bottom)	1.43	1.50	1.30	1.25	1.68	1.12

Table 16 – Tap Cutting Torque

(N • m)

Thread Percentage / Thread Size	100%	90%	75%	60%
M1	0.02	0.017	0.01	0.007
M2	0.11	0.09	0.07	0.04
M3	0.27	0.22	0.16	0.1
M4	0.69	0.58	0.41	0.25
M6	2.16	1.76	1.23	0.83
M8	4.56	3.77	2.65	1.72
M10	8.23	6.86	4.8	3.09
M12	13.7	11.1	7.64	4.8
M16	24.5	18.6	13.5	9.02
M20	46.8	39.2	26.4	17.1
M24	78.4	65.7	45.6	29.4
M30	139.7	112.7	78	53.9
M48	460	377.5	268.7	163.7

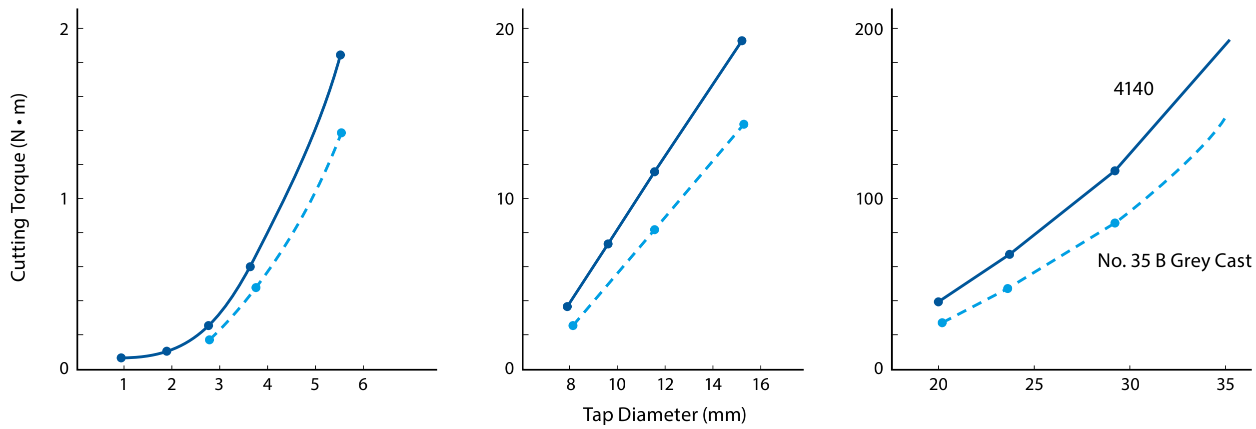
Work Material: 4140 (193HB)
(Specific Cutting Resistance: 3600N/mm)
Tap: Straight Flute Tap (Plug) Coarse
(Tap Constant Value: 1.35)

(N • m)

Thread Percentage / Thread Size	100%	90%	75%	60%
M1	0.016	0.013	0.009	0.006
M2	0.082	0.069	0.05	0.032
M3	0.21	0.17	0.12	0.08
M4	0.53	0.43	0.3	0.2
M6	1.6	1.31	0.93	0.61
M8	3.37	2.78	1.95	1.27
M10	6.11	5.02	3.57	2.29
M12	10.1	8.18	5.56	3.56
M16	18.3	13.8	10.6	6.74
M20	34.6	27.6	19	12.6
M24	58.3	48.4	33.6	21.4
M30	103.4	83.6	57.5	39.2
M48	337.4	279.9	198.4	122.4

Work Material: No. 35 B Grey Cast (193HB)
(Specific Cutting Resistance: 2900N/mm)
Tap: Straight Flute Tap (Plug) Coarse
(Tap Constant Value: 1.25)

Fig 25 – Cutting Torque of Straight Flute Tap <Plug Tap, Thread Percentage 90%>



7.2 Calculation of required power

Pc: Energy (kW)
Tc: maximum torque (N•m)
n: speed (min^{-1})

In reality, when the increase in torque through usage and other breakage is taken into account, the necessary energy required would be approximately four times the calculated value.

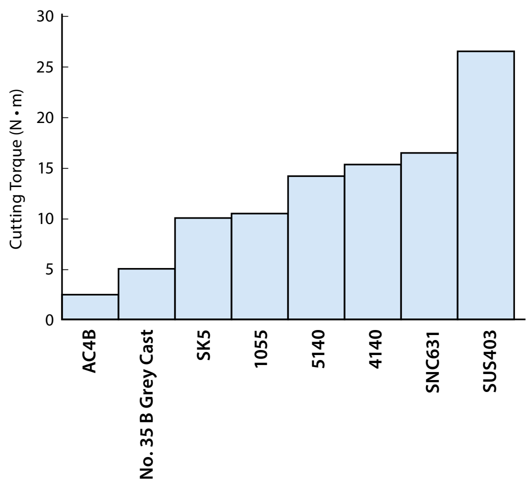
$$Pc=\frac{2\pi \cdot n \cdot Tc}{102 \cdot 9.8 \cdot 60} 0.000104n \cdot Tc$$

Pc: Energy (kW)
Tc: Maximum Torque (N•m)
n: Speed (min^{-1})

7.3 Factors affecting cutting torque

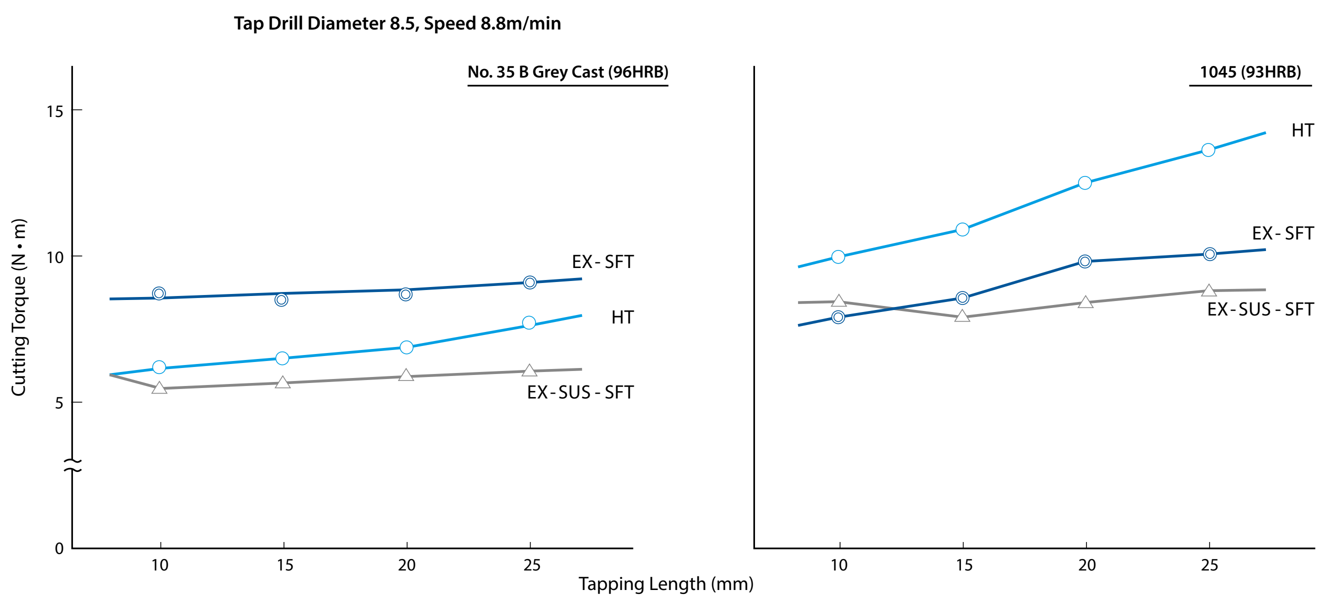
As shown in Fig26 and Fig 27, the cutting torque changes due to the change in various factors. Machining with a new tap will result in a lower torque than machining with a worn or used tap.

Figure 26 – Work Material and Cutting Torque



Tap: Straight flute tap M10X1.5 II
HSS chamfer length 5 threads
Tap Drill, Thread Length: 8.5X10mm (Thru Hole)
Speed: 4.6m/min
Cutting Fluid: Non-water Soluble Cutting Oil

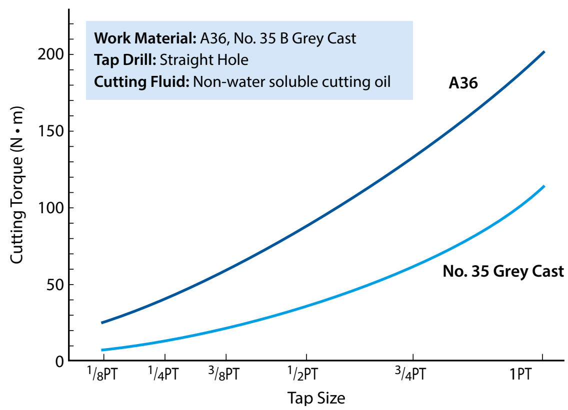
Fig 27 – Tapping Length, Work Material, Tap Type and Cutting Torque



7.4 Cutting torque for taper taps for pipes

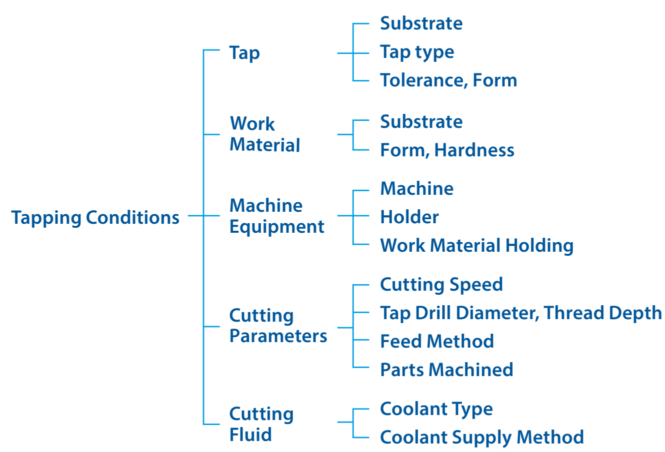
Unlike Straight taps, taper pipe taps cut using the entirety of the thread length, so friction will continuously increase during tapping. The cutting torque ends up being 2 to 3 times that of hand taps. Table 7-8 shows the experimental data. Fig 28 shows the testing values.

Fig 28 – Cutting Torque of Taper Pipe Thread Tap



Part 8: Tapping Conditions

Table 18 – Tapping Conditions



The cutting mechanics of tapping is the hardest process to explain between all the cutting operations. Numerous tapping conditions overlap and enmesh with each other, and it is hard to isolate and solve single problems. Therefore, one must check the relationship between the complicated tapping conditions, tapping method, and tool quality to pick out the best tap to use in each situation. Table 18 shows the key points of the tapping conditions.

8.1 Work material

During tapping operations, the characteristics of the work material have a drastic effect on the tool life, internal thread accuracy, and thread finish. For example, the optimum specification of the rake angle is 16 degrees for copper, but 3 degrees for copper alloys like brass and bronze. Other than the rake angle, the pitch diameter, hardness, and coating must be taken into account to find the best tap for the work material. Table 19 shows the characteristics of typical work materials and Fig. 29 shows the types of taps and their durability against special hard materials.

Fig 28 – Cutting Torque of Taper Pipe Thread Tap

Work Material	Material Properties	Tap Considerations
Low Carbon Steel	• Soft material, easy to weld and has poor surface finish	• Increase rake angle • Steam oxdie is effective
Cast Steel, Mid/High Carbon Steel, Manganese, Chrome Steel/Tool Steel	• Heavy tool wear	• Increase tap hardness
Stainless Steel	• High weldability • High chance of work hardening • Tough and long chips	• Increase rake angle • Helical flute tap • Steam oxide
Cast Iron	• Heavy/excessive wear • High chance of oversize • Powdery chips	• Weak rake angle for wear prevention • Nitride treatment • Oversize
Aluminum	• Soft and highly malleable • Poor surface finish likely to occur • Overlapping chips	• Sharper rake angle • Thinner tap land • Form tapping is best
Aluminum Alloy	• Similar to cast iron but has a tendency to get poor surface finish	• Sharper rake angle • Oversize • Form tap is best
Copper	• Soft and highly malleable • Connecting chips • Hole expansion chance is minimal	• Sharp rake angle • Spiral flute tap • Oversize • Form tap is best
Copper Alloy	• Has tendency to chatter/poor surface finish • Expansion is minimal	• Weak rake angle • Oversize • Roll form tap is best
Thermosetting Resins	• Heavy/excessive wear • Expansion is minimal, tendency to undersize • Powder chips	• Weak rake angle • Oversize • Nitride treatment

Fig 29 – Tap Type and Durability in High Hardness Material

Tap	Machined No. of Holes (50, 100, 150, 200, 250, 300)	Cutting Condition
CPM-SFT	Undersize	M10×1.5 Work Material: 4140 40HRC Tap Drill: φ8.5×20mm Speed: 6.3m/min Cutting Fluid: Non-water soluble cutting fluid
EX-SFT	Undersize	

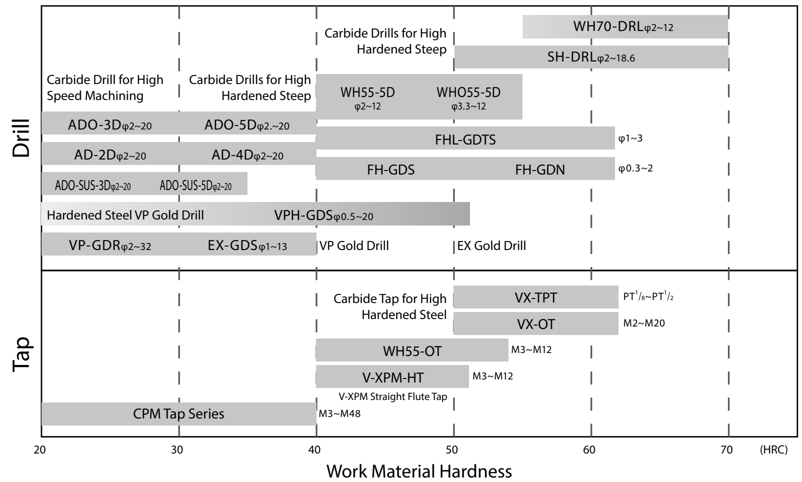
VX-OT Performance in High Hardness Materials

Tap	Material Hardness	Machined No. of Holes (20, 40, 60, 80, 100)	Cutting Condition
Straight Flute Carbide Tap for High Hardness Steel M6×1 3 UMA OH3	59HRC	30 holes	Work Material: D2 Tap Drill: φ5.1×20mm (Thru Hole) Thread Depth: 9mm Speed: 59HRC 2.3m/min 56HRC 2.1m/min Cutting Fluid: Non-water soluble cutting fluid
	56HRC	over 100 holes	

V-XPM-HT Performance in High Hardness Materials

Tap	Material Hardness	Machined No. of Holes (20, 40, 60, 80, 100)	Cutting Condition
Straight Flute Tap For High Hardness Steel M10×1.5 5 XPM OH4	50HRC	76 holes	Work Material: H13 Tap Drill: φ8.5×25mm (Thru Hole) Thread Depth: 15mm Speed: 2.2m/min Cutting Fluid: Non-water soluble cutting fluid

Hole making - Tool Selection Map



*When using a drill with an oil-based cutting fluid, lower the cutting speed by 20% before use.

8.2 Tap drill diameter (cutting tap)

The difficulty of tapping is drastically affected by the diameter of the drill hole (Percentage of Thread). As long as the diameter of the drill hole is within the tolerance of the internal thread diameter, there is no marked difference in the strength of the thread, so it is generally taken as large as possible within the tolerance. If the diameter of the drill hole is small, there is an increase in the size of chips and cutting torque increases, possibly resulting in breakage.

Because it directly affects the tool life, operating efficiency, and the accuracy of the internal thread, the drill hole diameter is set as large as the internal thread diameter limit allows. The suggested drill hole diameter for Metric coarse threads, Metric fine threads, Unified coarse threads, and Unified fine threads can be found in standard ASME B1.1-2003. The ratio of the in-contact portion of the internal and external thread to the theoretical full thread height is called the Percentage of Thread, and is calculated from the following formula.

$$\text{Thread Percentage} = \frac{(\text{External Thread Major Diameter}) - (\text{Internal Thread Minor Diameter})}{2 \times (\text{External Thread Height})} \times 100$$

Thread Percentage = (External Thread Major Diameter) – (Internal Thread Minor Diameter) / 2 X (Basic Thread Height) X 100

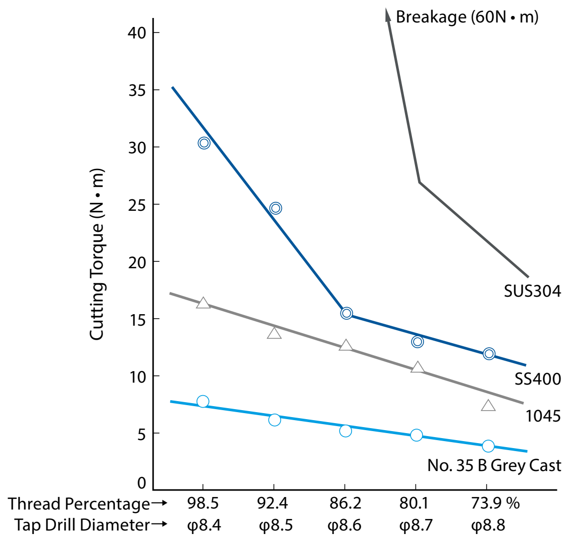
For cutting taps, the minor diameter of the internal thread is equal to the drill hole diameter. Given the pitch (P), major diameter of external thread (d), and the required percentage of thread, the proper selection of drill hole diameter is found by the below formula.

$$\text{Tap Drill Diameter} = d - 2 \times 0.541266P \times \frac{\text{Thread Percentage}}{100}$$

Tap Drill Diameter = d - 2 X 0.541266P X (thread percentage/100)

As Fig. 30 shows, as the percentage of thread increases (through hole diameter decreases) the cutting torque increases drastically, and tapping itself becomes difficult to the point of tool breakage.

Fig 30 – Percentage of Thread and Cutting Torque



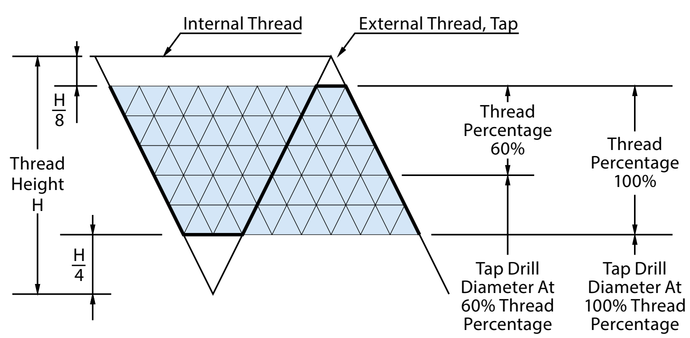
Straight Flute Tap: M10×1.5 HSS Plug Tap
Thread Depth: 20mm (Thru Hole)
Speed: 8.5m/min

By keeping the external thread the same with the baseline angle and changing the percentage of thread, to 100% and 60% for example, and comparing with the number of triangles of the cross section of the internal thread shown in Fig 31, we get

(13 + 11 + 9) / 45 x 100 = 73.3%

So even if the percentage of thread is 60%, the cross section of the screw thread is 73.3%. The strength of the thread is proportional to the cross section of the thread, so changing the height of the thread does not decrease the strength of the thread.
The attrition rate of the cross section of the area being tapped is 26.7% {(7 + 5) / 45 x 100}, whereas the attrition rate of the cross section of the area being tapped is 44.4% {(9 + 11) / 45 x 100}. From this, it can be seen that it is better to keep the internal diameter as large as possible.

Fig 31 – Diagram/Illustration of Thread Strength



8.3 Cutting fluid

Tapping involves an extremely complex cutting action, so the specification of the tap greatly affects the finishing surface roughness, tool life, and the cutting torque. But just as important as the specification are the types of the cutting fluid, and the method of application.
Cutting fluids are mainly used for lubrication, cooling, and prevention of welding or galling. In tapping, the cutting speed is usually slow and it is hard to clear out the chips, so it is common for the tap to chip or break from built-up edge or chips getting stuck. From this, it is common to use sulfuric chloride based fluids and pastes.
Water-soluble cutting fluids are not as good as water-insoluble fluids in terms of lubrication, but they are good for cooling and are more environmentally friendly. There are coolant types with extreme pressure agents for use in steel, so careful selection based on the uses must be taken.
Characteristics of the cutting fluids are shown on Table 8-9, and examples of the effects they have on the tap's durability are shown in Fig 32 and Fig 33.

Table 20 – Dutting Fluid/coolant Characteristic

	Non-Water Soluble Coolant/Cutting Fluid			Water Soluble Coolant	
	Oil	Inert Extreme Pressure Type	Active Extreme Pressure Type	Emulsion Type	Soluble Type
Lubricity	○	◎	◎	○	△
Anti-Welding	△	○	◎	—	—
Cooling Ability	○	○	○	◎	◎
Infiltration	◎	◎	◎	△	○
Anti-Rust	◎	○	○	△	△
Smoking/Flammability	△	△	△	◎	◎

Fig 32 – Cutting Fluid and Tap Tool Life

Coolant	Machined No. of Holes		Cutting Parameters
Water Soluble Cutting Fluid	100, 200, 100	133	Tap: Spiral Point Tap M6×1 Oh2 Work Material: SUS304 85~87HRB Tap Drill: φ5×10mm (Thru Hole) Speed: 11.3m/min Failure Mode: Welding On Tap And Thread Flank Area
Activated Non-water Soluble Cutting Fluid	400, 200, 300	300	
Super Activated Non-water Soluble Cutting Fluid	1,000, 1,000, 1,000	able to continue over 1,000	

Fig 33 – Tool Life and Dilution Ratio of Water Soluble Coolant

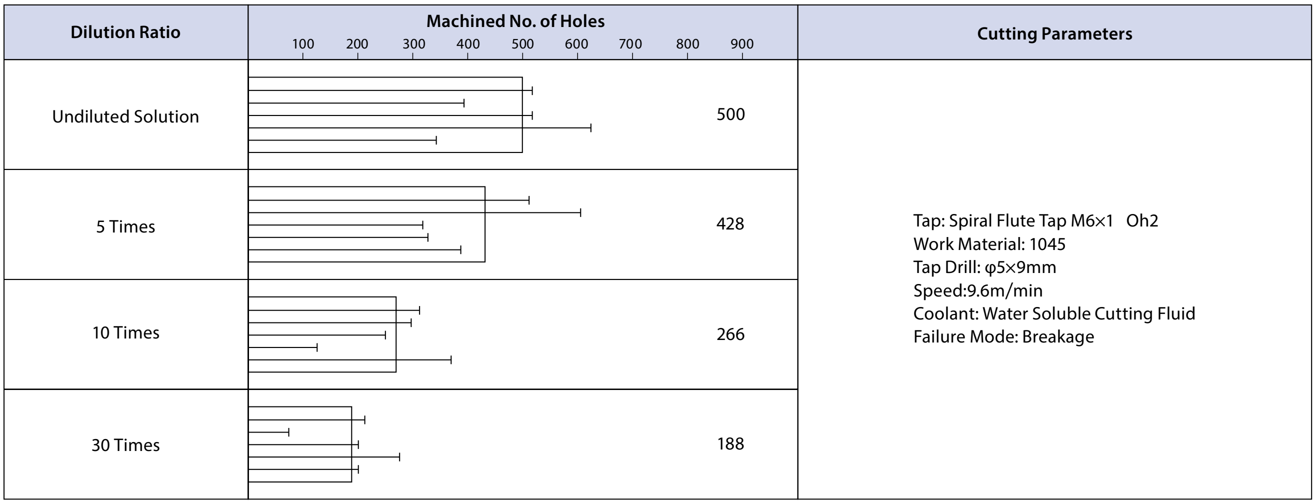
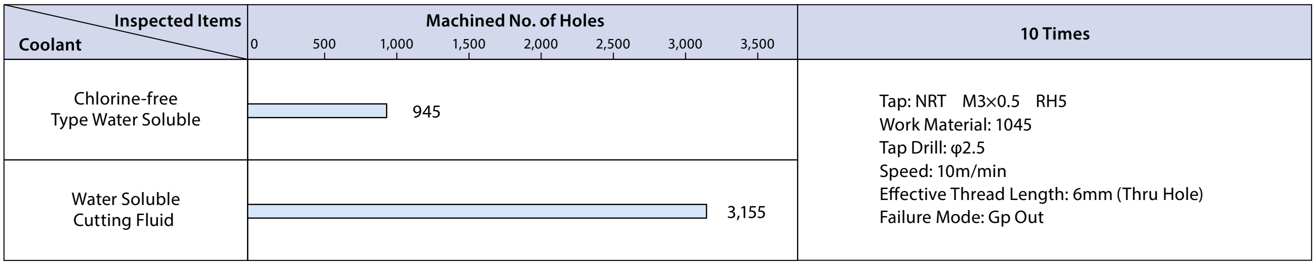


Fig 34 – Tool Life of Water Soluble Coolant and Chlorine-free Water Soluble Coolant



The latest trend is the use of dry tapping, tapping that does not use any cutting fluids, in order to lessen the impact on the environment by eliminating the requirement to clean finished parts. Dry tapping has two methods: complete dry tapping, which uses no cutting fluids, and semi-dry tapping, which sprays a limited amount of fluid in mist form. Semi-dry tapping is usually called MQL (Minimum Quantity Lubrication), in which 4 to 10cc per hour of the mist is applied, providing lubrication during tapping in effort to extend the tool life. In the semi-dry tapping, the work material does not require as much cleaning afterwards, so pollution of the environment is minimal.
Compared to tapping with cutting fluids, reduction of the tool life and cutting conditions cannot be avoided. Fig 35-1 shows the difference in performance between the complete dry, semi-dry, and tapping with water-soluble cutting fluids.

Tapping is nearly impossible completely dry, but the semi-dry application performed at about 83% of the tapping with water-soluble fluid. Fig 35-2 shows the results of the complete dry tapping with and without coating applied.

Fig 35-1 – Dry Tapping Tool Life

Tap	Coolant	Machined No. of Holes (1,000 / 2,000)	Cutting Condition
Slow Helix Spiral Flute Tap (Special Form) M10×1.25 Chamfer 2.5P CPM OH3 V Coating	Water Soluble Coolant Dilution Rate x10 (10% oil)	Stopped tapping due to 0.2mm wear amount on chamfer at 1,750 holes	Work Material: 1045 Tap Drill: φ8.7×15mm (Blind Hole) Thread Depth: 10m/min Speed: 10mm Machine: Horizontal Machining Center
	Semi-dry 3cc/h	Stopped tapping due to 0.2mm wear amount on chamfer at 1,450 holes	
	Complete Dry (No Air)	Internal thread undersized on 4th hole	

Fig 35-2 – V Coated Tap Tool Life in Dry Machining/tapping

Tap	Coolant	Machined No. of Holes (1,000 / 2,000 / 3,000)	Cutting Condition
V-POT	V coating	over 3,000	Work Material: 1045 Tap Drill: φ5×10mm (blind hole) Thread Depth: 10m/min Speed: 10mm Cutting Fluid: Dry Machine: Horizontal Machining Center
EX-POT	Bright	157	

The cutting speed is affected by the type of the tap, size and shape of the prepared hole, work material, and the cutting fluid used. If the speed is appropriate, tool life is increased, the internal thread is more accurate, and the surface roughness is improved. For example, in Fig. 36, the cutting speed is too fast, resulting in the built-up edge getting bigger and causing the internal thread to fall out of the tolerance range.

Furthermore, the built-up edge induces chipping as the friction from tapping heats up the cutting edge and makes it softer (this happens especially when tapping tool steels). These conditions result in the cutting edge getting worn down, and decreasing tool life, as shown in Table 8-16.

To prevent built-up edge and the additional cutting heat, the cutting speed must be carefully calculated. The cutting speed of the tap can be calculated from the velocity of rotation by the following formula.

$$V_c = \frac{(D \times \pi \times n)}{12} \text{ (m/min)}$$

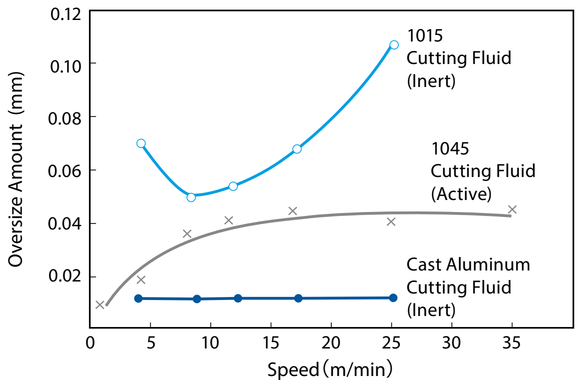
Vc: Cutting Speed (m/min)

D: Nominal Tap diameter

n: Rotation Speed

π : (3.14) Pi

Fig 36 – Speed and Oversize Amount



Straight Flute Tap: M10×1.5
HSS class 2, chamfer 5P
Tap Drill: φ8.5×10mm (Thru Hole)
Thread Length: 10mm
Machine: Radial Drill Press/machine
Cutting Fluid: Non-water Soluble Cutting Fluid

Fig 37 – Speed and Tool Life

Work Material	Speed (m/min)	Tool Life (No. of Holes) 0 500 1,000 1,500	Cutting Parameters
No. 35 B Grey Cast	9.6	X̄=1,474	Failure Mode: When the GO thread Gauge (class 2) fails/does not go thru Tap: Straight Flute Tap M6X1 HSS Class 2, Chamfer 5P Tap Drill: φ5×10mm (Thru Hole) Machine: Tapping Drill Press Cutting Fluid: Non-water Soluble Cutting Fluid
	18.2	X̄=1,062	
	29.7	X̄=1,016	
1045 (95HRB)	9.6	X̄= 576	
	18.2	X̄= 507	
	29.7	X̄= 348	

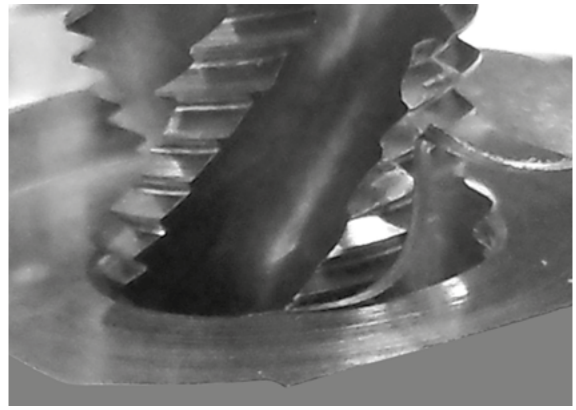
Table 21 – Standard Cutting Speed and Applicable Cutting Fluid

◎ Optimal ○ Applicable △ Usable — Cannot use/Unusable

Work Material		Cutting Speed (m/min)							Cutting Fluid			
		Hand Tap*1	Spiral Tap	Point Tap*1	Carbide Tap*1	Form Tap*1	High Speed Synchronized Tap	Pipe Tap	Cutting Oil	Water Soluble	Semi-dry	Dry
Low Carbon Steel	Under C0.25%	8~13	8~13	15~25	-	8~13	27~32	3~6	◎	○	—	—
Medium Carbon Steel	C0.25~0.45%	7~12	7~12	10~15	-	7~10	27~32	3~6	◎	○	—	—
High Carbon Steel	Over C0.45%	6~9	6~9	8~13	-	5~8	22~27	2~5	◎	○	—	—
Alloy Steel	SCM	7~12	7~12	10~15	-	5~8	22~27	2~5	◎	△	—	—
Heat-Treated Steel	25~45HRC	3~5 (4~8)	3~5 (4~8)	4~6 (6~10)	-	-	-	2~5	◎	△	—	—
Stainless Steel	SUS	4~7	5~8	8~13	-	5~10	-	3~6	◎	○	—	—
Stress-Hardened Stainless Steel	SUS630 SUS631	3~5	3~5	4~6	-	-	-	2~5	◎	—	—	—
Tool Steel	SKD	6~9	6~9	7~10	-	-	-	2~5	◎	—	—	—
Cast Steel	SC	6~11	6~11	10~15	-	-	17~22	2~5	◎	○	—	—
Cast Iron	FC	10~15	-	-	10~20	-	-	2~5	◎	○	○	○
Ductile Cast Iron	FCD	7~12	7~12	10~20	10~20	-	-	4~8	◎	○	○	—
Copper	Cu	6~9	6~11	7~12	10~20	7~12	27~32	2~5	○	○	—	—
Brass/Brass Casting	Bs • BsC	10~15	10~20	15~25	15~25	7~12	27~32	5~10	○	○	○	○
Bronze	PB • PBC	6~11	6~11	10~20	10~20	7~12	-	6~11	○	○	—	—
Rolled Aluminum	Al	10~20	10~20	15~25	-	10~20	100~300*2	5~10	◎	○	—	—
Aluminum Alloy Casting	AC • ADC	10~15	10~15	15~20	10~20	10~15	80~300*2	10~15	◎	○	—	—
Magnesium Alloy Casting	MC	7~12	7~12	10~15	10~20	-	-	10~15	◎	○	—	—
Zinc Alloy Casting	ZDC	7~12	7~12	10~15	10~20	7~12	27~100	10~15	◎	○	—	—
Thermosetting Plastic	Bakelite Phenol Epoxy	10~20	-	-	15~25	-	-	5~10	—	○	○	○
Thermosetting	Vinyl Chloride Nylon Duracon	10~20	10~15	10~20	10~20	-	27~32	5~10	—	○	—	—

1. This table assumes general machining conditions. Different usage conditions may require changes to speed selection.
2. For tap selection, refer to the application-based tap selection table.
3. The value in parentheses in the tempered steel column is the CPM series cutting speed.
*1 For coated taps, select 30% to 50% higher than the values in the table. Note
*2 Depends on the maximum synchronous feed speed of the machine.

Due to their design, spiral fluted taps can easily chip their cutting edges from chips getting caught while tapping. By setting the cutting speed at the appropriate level, the shape of the chips is kept constant, and the chances of them getting caught are kept at a minimum.
Fig. 38 shows results of how the cutting speed affects the chip shape. The chip shapes that result from speeds of between 7.5 ~ 12.5m/min are the easiest to evacuate from the hole.



Re-cutting chips

Fig 38 – Cutting Speed and Chip Formation with Spiral Flute Taps

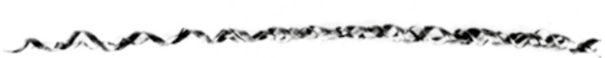
Test Product	Testing Parameter
Product Name: EX-SFT Size: M16×2 Substrate: HSSE Thread Limit: OH2	Work material: SS400
	Speed: 20, 40, 100, 150, 200, 250, 300, 400min^{-1}
	Tap Drill: φ15×25mm
	Threading Length: 25mm
	Feed: 90%
	Cutting Fluid: Non-Water Soluble Cutting Fluid
	Machine Used: Horizontal Machining Center

1m/min (20min^{-1})

2m/min (40min^{-1})

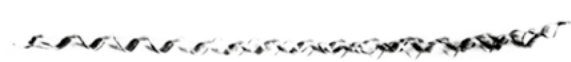
5m/min (100min^{-1})

7.5m/min (150min^{-1})

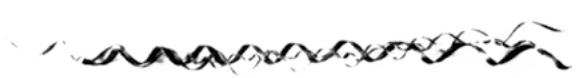


10m/min (200min^{-1})

12.5m/min (250min^{-1})



15m/min (300min^{-1})



20m/min (400min^{-1})

0 2 4(cm)

8.5 Equipment and Machinery

The equipment used in tapping can be categorized in the following categories: Machine, machine specifications, machine capability, rigidity

- Machine: Mechanism, capability, rigidity
- Tap holder: Structure, capability, floating, shaft runout
- Workpiece support: Support method, holding force, alignment

8.5.1 Machine/feeding mechanism

The machines require enough capability so that they can withstand the tapping torque. The accuracy of the internal thread diameter is heavily affected by the feeding mechanism. This makes a synchronized mechanism the preferred choice, and automated tapping machines with the latest synchronized feeding mechanisms are becoming popular. Other methods include hydraulic pressure feeding, cam feeding, and manual feeding in standard presses. Whichever method is used, setting the feed rate to the appropriate value leads to better accuracy of the internal thread diameter. For NC machines and machining centers that do not have the rotational direction of the main axis and the feed rate synchronized, tap holders with float mechanisms in the axial direction are used to minimize the difference. Generally vertical machines work better than horizontal machines in terms of chip evacuation and cutting fluid circulation. Table 23 shows the types of feeding mechanisms and their disadvantages, and Table 24 shows the features of the synchronized feeding.

Table 23 – Feeding Method and Problems

Feed Method		Type of Machine Used	Machining Issues/Problems	Prevention
Manual Feed		Tapping drill machine/press	Overfeed of tap due to improper thrust force.	Depends on the skill of the operator.
Forced Feed	Gear Feed	Tapper, custom machine, NC machine, transfer machine	Due to the backlash of the gear feed, “tearing” and “galling” may occur due to the delay in the thrust direction during the reverse rotation.	Extension/tension of the built-in spring holder will absorb extra thrust.
	Lead Screw Feed	NC machine Dedicated machine	The feed error shifts to the tap due to the rotation control of the lead screw and the wear of the lead screw.	Absorbed by the tension/extension of the built-in spring of the holder. Reduce feed by about 5%.
	Hydraulic/ Pneumatic Feed	NC machine Dedicated machine	Thrust force is constantly applied and the thread of the female screw is thinned.	Attach a device to control the pressure after tap chamfer engagement.
	Cam Feed	Automatic lathe tapping machine	Adjustment of thrust force required until engagement.	Adjust the spring.

Table 23 – Feeding Method and Problems

Machine	Machining Method	Tap Holder
Tapping Machine Machining Center	Being able to compensate for spindle rotation and feed errors, including pitch error and backlash compensation, with the complete synchro feed control and one pitch (lead) feed per spindle revolution, high speed, high precision and improved tool life tapping is possible.	Direct tapping with fixed type without tapper.

Remark: please use the high speed synchro series for taps.

8.5.2 Tap holder

The holder is an important part to ensure the fullest performance of the tap. Generally these are divided into two types: rigid and floating. The floating holder is further subdivided into three styles: axial (or tension/compression), lateral, or combo. T/C is common in non-synchronized machines. Lateral floating holders are intended to compensate for misalignment between the pre-drilled hole and the tapping location, to avoid breakage of taps due to improper alignment.

Table 25 – Tap Holder Type and Features

Full synchro feed

Features: Machining centers that have a completely synchronized feed and speed mechanism (synchronized, rigid, direct) do not require a tapper with a tension/compression function; use a drill/end mill collet holder. If not 100% synchronized, poor quality, torn threads/burrs may occur as well as unstable tool life.

No rotational and feed synchronization mechanism

Mechanism
◯ Axial (Axial Direction) Floating Mechanism Features: Mechanism that automatically absorbs and corrects the error between the machine spindle feed and tap feed. Prevents thread thinning due to feed error.
◯ Radial (Radial Direction) Floating Mechanism Features: Mechanism that automatically absorbs and corrects the misalignment between the tap and tap drill hole during tapping. Align the center of the tap with the tap drill hole to prevent breakage due to misalignment and internal thread inclination.
◯ Automatic Sizing Type Features: Automatically absorbs and corrects poor bite of taps and variations in screw depth due to inertial rotation after the machine spindle is stopped.
◯ Built-In Reverse Rotation Mechanism Features: Mechanism to remove the tap by rotating the machine spindle with the tapper without rotating the machine spindle in reverse. Since the machine spindle is not reversed, failure due to overheating and machine drive part wear are reduced, thread machining time is greatly shortened and machine power consumption is also reduced.
◯ Torque Limiter Type Features: When a torque value is over the set torques value (adjusted arbitrarily according to the work material) is applied to the tap, it is equipped with mechanism to prevent tap breakage and prevent damage to the machine drive part due to excess torque.

8.5.3 Workpiece support

There are various support methods depending on the shape of the work piece, but it is important that there is no inclination of the tap drill hole, misalignment between the tap and tap drill hole, and no deflection of the work piece due to the tapping torque while machining.

Part 9: Roll form taps

9.1 Thread form and features/characteristics

Generally the characteristics of form taps are:
1 - No chips are produced from tapping operations.
2 - Longer tool life.
3 - Produces a stronger internal thread than cutting taps.
4 – Stronger tap body to resist breakage

However, the OSG XPF forming taps have additional characteristics such as:
1 – Due to manufacturing there is no discontinuity in form between the chamfer and full thread portions, leading to more accurate threading and longer tool life overall.
2 – Without a sharp point and with a continuously curved relief, it provides a better surface finish and thread tolerance is more accurate.
3 – They have oil grooves to improve lubrication during tapping.
4 – They have high rigidity to prevent breakage, forming with minimal friction, thus increasing tap life.

9.2 Production range

OSG form tap standard production ranges are the following:
Metric threads M0.5~M50 (0.125~3 pitch)
Unified threads No. 0~2" (80~8 threads per inch)
Parallel pipe thread 1/8~1-1/2" (28~11 threads per inch)
Special sizes outside the above range may be considered on a case-by-case basis.

9.3 Tap types and main applicable materials

Where conventional cut taps produce threads by removing material with their cutting edges, form taps make use of the material's ductility to roll the threads from the drilled hole surface. Materials listed in Table 26 are optimal form tapping work materials. Furthermore, it is necessary to properly differentiate the use of steel type and non-ferrous type.
TiN coated Nu-roll taps (NRT) • VP Nu-roll taps (EXO-NRT) •X Performer form tap (XPF) can be used for both steel and non-ferrous alloys.

Fig 39 – Comparison of Tap Profile/thread Structure (Representative Examples Shown)

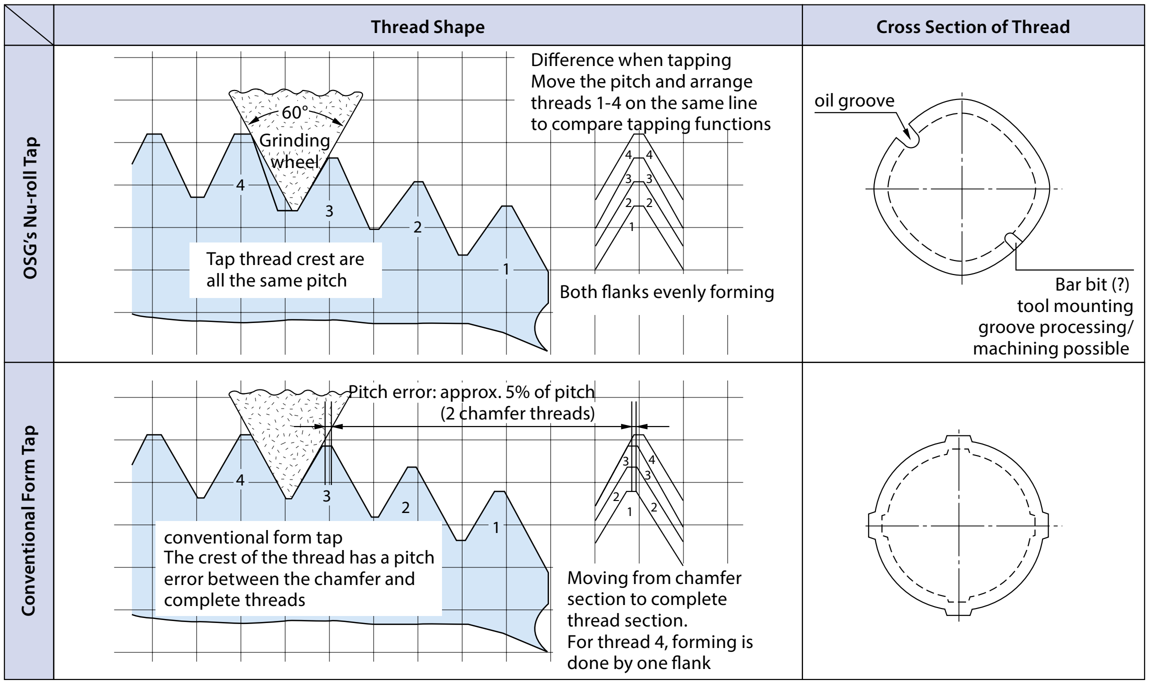


Table 26 Types of taps and applicable work material

Tap	Applicable Work Material	Types of Applicable Materials
VP Nu-roll (VP-NRT) X Performer form tap (S-XPF)	Steel	Steels under 35 HRC
For steel (NRT) TiN coating (TIN-NRT) VP Nu-roll (VP-NRT) X Performer form tap (S-XPF)	Steel	Steels under 20HRC, Mild Steel, Free-cutting Steel, Electromagnetic Mild Steel, Stainless Steel, etc.
For non-ferrous alloy (B-NRT) TiN coating (TIN-NRT) VP Nu-roll (VP-NRT) X Performer form tap (S-XPF)	Aluminum and Al alloys	Die casting, castings, drawn and rolled materials
	Zinc and Zn alloys	Castings, drawn and rolled materials
	Copper	Drawn and rolled materials
	Brass	Drawn and rolled materials

9.4 Usage

9.4.1 Tap drill hole diameter

Since NRT taps produce the internal thread by plastic deformation, the pre-drilled hole must be larger to accommodate the material deformation into the space between threads on the tap.

$$dN = D - 0.2P - 0.00403 \cdot P \cdot f_1 + 0.0127 \cdot n - ①$$

dN: Tap Drill Hole Diameter
D: Tap Basic Major Diameter
n: RH Accuracy Number
f_1: Thread Percentage (%)
P: Pitch

For example: M10x1.5 RH7P when tapping a class 2 internal thread and % of thread being 90%,
$dN = 10.000 - 0.2 \times 1.5 - 0.00403 \times 1.5 \times 90 + 0.0127 \times 7 = 9.24$

However, some work materials may have less ductility and higher hardness. Therefore, do not apply the calculated value directly to the tapping operation, use a diameter that is slightly larger than this calculated value. If possible observe the condition of the formed internal thread and adjust the size of the diameter gradually. Fig. 40 shows the relationship between the prepared diameter of different work materials and the minor diameter of the internal thread after the tapping operation.

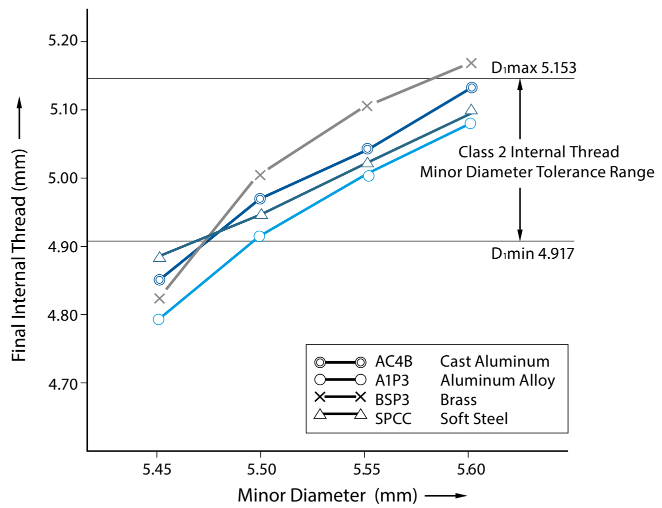
9.4.2 Management of tap drill hole

For cutting taps, the prepared diameter becomes the minor diameter of the internal thread. However, for form taps it is necessary to set a diameter of the prepared hole for proper precision of rolling operations, because due to the deformation process, the internal diameter changes during tapping. The maintenance of the prepared hole is a most important consideration for the proper operation of NRT taps. For example, a diameter of a prepared hole for a M6x1 6H metric internal thread having a percentage of thread range of 80%-100% is:

$$\text{Minimum hole diameter} = D - 0.603P + RH*n \text{ ——— ②}$$
$$\text{Maximum hole diameter} = D - 0.54255P + RH*n \text{ ——— ③}$$
$$② - ③ = 0.603P - 0.54255P = 0.06045P \text{ ——— ④}$$

The diameter must remain with the range of equation ④.
This is the basic procedure for setting the diameter for a prepared hole for NRT taps. However, in ordinary operations, calculate a basic size of prepared diameter from equation ① and determine a drill diameter.

Fig 40 – Relationship Between Tap Drill Hole Diameter and Finished Internal Diameter of Internal Thread



Tap: NRT M6×1 RH7B
Machine: Tapping Machine
Coolant: Nitrate-based Non-water Soluble Cutting Fluid
Thread Length: 6mm
Speed: 7.2m/min

9.4.3 Chamfer of the tap drill hole

Since NRT taps use plastic deformation, occasionally burrs will occur around the entrance of the hole or on the work surface if there is no chamfering operation (Table 9-4). To prevent burrs, apply a chamfering tool with a point angle of 60°~70° as shown in Table 9-5.

However, if the chamfer requires the same angle as cutting taps, 118°, because of the drill operation, then set the diameter of the chamfered surface to (Tap's Major Diameter +2P). This will avoid the burrs reaching the work surface and make it possible to have a chamfer angle of 118°.

Fig 41 – Entrance/hole Edge Without Chamfer

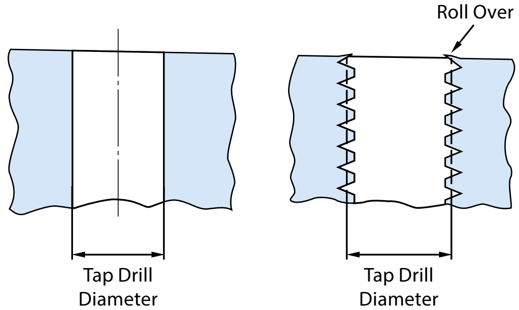
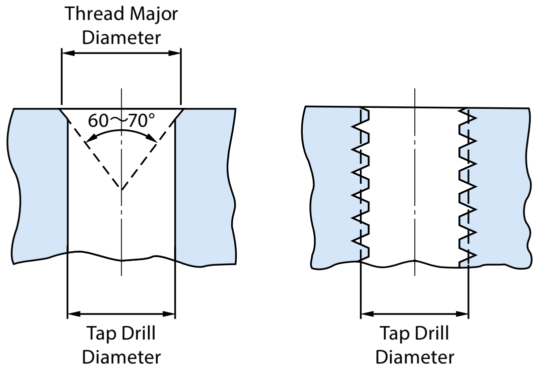


Fig 42 – Entrance/hole Edge with Chamfer



9.4.4 Tapping Speed

The tapping speed for NRT taps is practically the same as cutting taps. However, in most cases, doubling the standard speed will result in both production efficiency and improved thread condition. This requires having proper tapping speeds for different work pieces and dimensions of the internal thread. Table 9-6 shows the recommended tapping speeds for typical work materials.
Table 27 shows recommended tapping speeds general application materials.

Table 27 – Selection Criteria of Cutting Speed for Various Work Materials

Work Material	Speed (m/min)
Aluminum, Copper, Brass (Soft)	7 ~ 12 (10 ~ 30)*
Aluminum Alloy and Die Casting/Free, Cutting Steel, Copper Alloy, Brass	12 ~ 20 (10 ~ 30)*
Regular Steel (35HRC or less) Mild Steel, Stainless Steel	5 ~ 10 (5 ~ 15)*

*XPF recommendations

9.4.5 Tapping Oil

Since NRT taps produce internal threads via plastic deformation, it is recommended to apply a tapping solution with high lubricity and containing sulfur or chlorine constituents. This will produce a longer tap life and an excellent machined surface on the internal threads. Table 28 shows different tapping oil selections for typical work materials.

Table 28 – Selection Criteria for Tapping Oil for Various Work Materials

Work Material	Tapping Oil
Aluminum And Alloy Die Casting, Zinc Alloy Die Casting	• Chlorine Extreme Pressure Water Soluble Oil • Chlorinated Non-water Soluble Oil • Oil Based Non-water Soluble Oil • Water Soluble Oil
Copper, Brass	• Oil Based Non-water Soluble Oil • Water Soluble Oil
Extremely Mild Steel, Electromagnetic Mild Steel, Free Cutting Steel	• Sulfochlorinated Non-water Soluble Oil • Sulfur Chloride Paste • Water Soluble Oil
Regular Steel, Mild Steel, Stainless Steel (Steel With Hardness less Than 35 HRC)	• Chlorine Extreme Pressure water Soluble Oil • Sulfur Chloride Paste • Sulfochlorinated Non-water Soluble Oil • Water Soluble Oil

9.4.6 Selection for precision of internal thread and tap pitch diameter

The precision of the internal thread depends on the tapping conditions and methods. In most cutting tap applications, the size of the internal thread is oversize, typically within a range of 30-40μm or less. If there is no sufficient control on the tapping conditions, it is very difficult to precisely control the size of the internal thread. However, NRT taps oversize very slightly due to the rolling process, making it capable of producing internal threads very close to exactly on-size. Thus it is very easy to limit the oversize condition on internal threads to 20μm or less. They can also produce ANSI 4H and 3B threads easily. Additionally, as the concern of oversize thread is considerably lessened, the NRT style can use a pitch diameter near the top of the tolerance range, allowing the NRT taps to create the internal thread with proper precision for a long period of continuous tapping.

9.5 Tapping Torque

During the plastic deformation process of internal threads, a large amount of friction is generated on the chamfer portion of the tap. This friction causes torque that can be as twice that of cutting taps. If a machine is near its operating limits with a cut tap, a form tap of identical size may exceed those limits, stalling the machine.
Testing taps in various materials provides the below curves for torque:

Fig 43 – Tapping Torque of Cut Tap and Form Tap

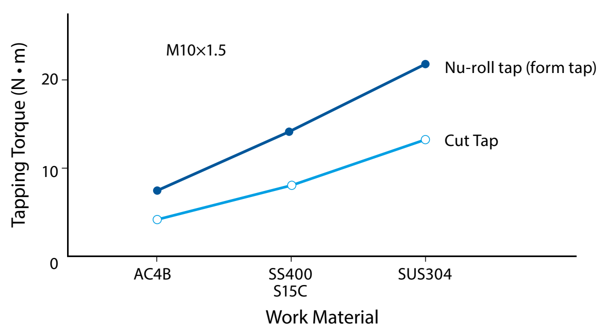
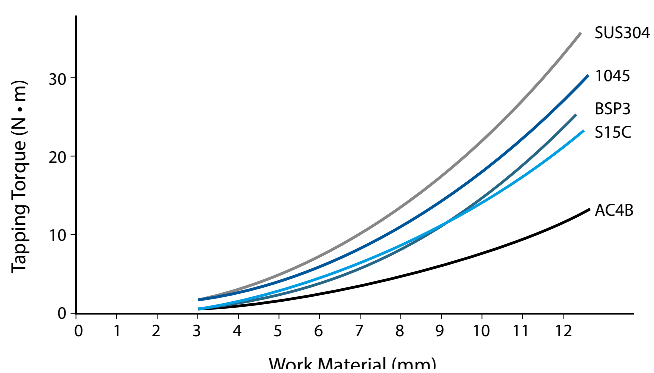


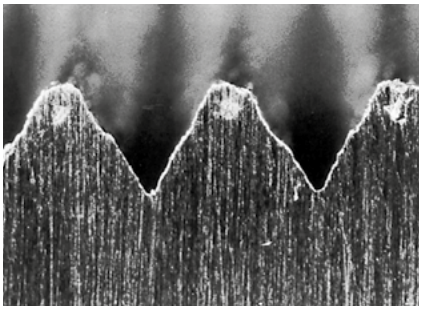
Fig 44 – Tapping Torque of Form Tap

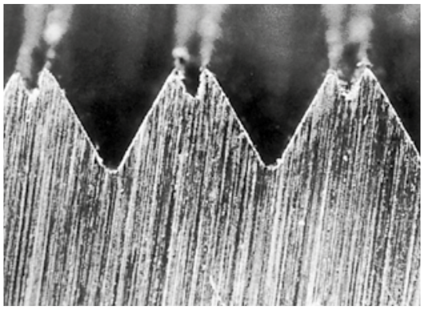


9.6 Internal thread form

After the deformation process, the internal threads have a hollow crest, distinct from the solid crest created by cut taps. If this hollow area causes any quality problems, it may be necessary to select a cutting tap.
In any material with low ductility, the thread flank will have a visually brilliant appearance, but the internal diameter may not have the same polished look. While visually inconsistent, this issue does not qualify as a quality problem, it is simply aesthetic.
Table 45 shows the percentage of thread and the internal thread profile.

Fig 45 – Thread Percentage and Thread Form M10X1.5


Over Formed


Normal

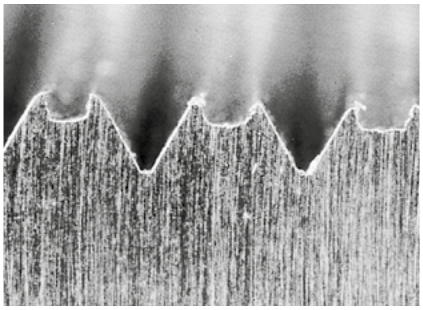

Under Formed

Fig. 43 Tapping torque of cutting tap and new roll tap (NRT)
Fig. 44 Tapping torque of new roll tap (NRT)

NRT $T = 0.09806 \times kc \times D_2 \times P^2$
XPF $T = 0.06864 \times kc \times D_2 \times P^2$
T: Plastic deformation torque (N•m)
kc: Work material coefficient
D_2: Nominal pitch diameter (mm)
P: Thread pitch (mm)

Kc value

Work Material	Coefficient
Aluminum	2
Aluminum Die Cast	3 ~ 4
Brass	6 ~ 8
Copper	10 ~ 11

9.7 Thread limit

9.7.1 Selection of thread limit

The tap thread limit is determined by the required internal thread grade (old JIS class 1, class 2, class 3, 3B, 2B etc).

Table 29 – Metric Thread

Thread Size	Pitch	Tap Thread Limit					
		Class 1 Internal Thread	Class 2 Internal Thread			Oversize	
						class 2+0.02~+0.03	class 2+0.04~+0.05
M1 M1.2	0.25	RH2	RH3	**RH4**	RH5	RH6	RH7
M1.4	0.3	RH2	RH3	**RH4**	RH5	RH6	RH7
M1.6 M1.7 M1.8 M2.5	0.35	RH2	RH3	**RH4**	RH5	RH6	RH7
M2 M2.3	0.4	RH2	RH3	**RH4**	RH5	RH6	RH7
M2.2 M2.5 M2.6	0.45	RH2	RH3	**RH4**	RH5	RH6	RH7
M3 M4 M4.5 M5 M5.5	0.5	RH3	RH4	**RH5**	RH6	RH8	RH9
M3 M3.5	0.35	RH3	RH4	**RH5**	RH6	RH7	RH8
M3.5	0.6	RH3	RH4	**RH5**	RH6	RH7	RH8
M4	0.7	RH4	RH5	**RH6**	RH7	RH9	RH10
M5	0.8	RH4	RH5	**RH6**	RH7	RH9	RH10
M6	1	RH4	RH6	**RH7**	RH8	RH9	RH10
M4 M4.5 M6	0.75	RH4	RH5	**RH6**	RH7	RH8	RH9
M7 M8 M9	1	RH4	RH6	**RH7**	RH8	RH9	RH10
M10 M11	0.75	RH5	RH6	**RH7**	RH8	RH9	RH10
M10 M11 M12	1	RH5	RH6	**RH7**	RH8	RH9	RH10
M8 M9 M10 M12	1.25	RH5	RH6	**RH7**	RH8	RH9	RH10
M10 M11 M12	1.5	RH5	RH6	**RH7**	RH8	RH9	RH10
M12	1.75	RH5	RH7	**RH8**	RH9	RH10	RH11
M14 M15 M16	1	RH5	RH8	**RH9**	RH10	RH11	RH12
M17 M18 M20 M22 M24	1	RH5	RH8	**RH9**	RH10	RH11	RH12
M14 M15 M16	1.5	RH5	RH8	**RH9**	RH10	RH11	RH12
M18 M20 M22 M24	1.5	RH6	RH9	**RH10**	RH11	RH12	RH13
M14 M16	2	RH6	RH9	**RH10**	RH11	RH12	RH13
M17	1.5	RH9	RH10	**RH11**	RH12	RH13	RH14
M18 M20 M22	2	RH6	RH10	**RH11**	RH12	RH13	RH14
M18 M20 M22	2.5	RH6	RH10	**RH11**	RH12	RH13	RH14
M24	2	RH6	RH10	**RH11**	RH12	RH13	RH14
M24	3	RH8	RH11	**RH12**	RH13	RH14	RH15

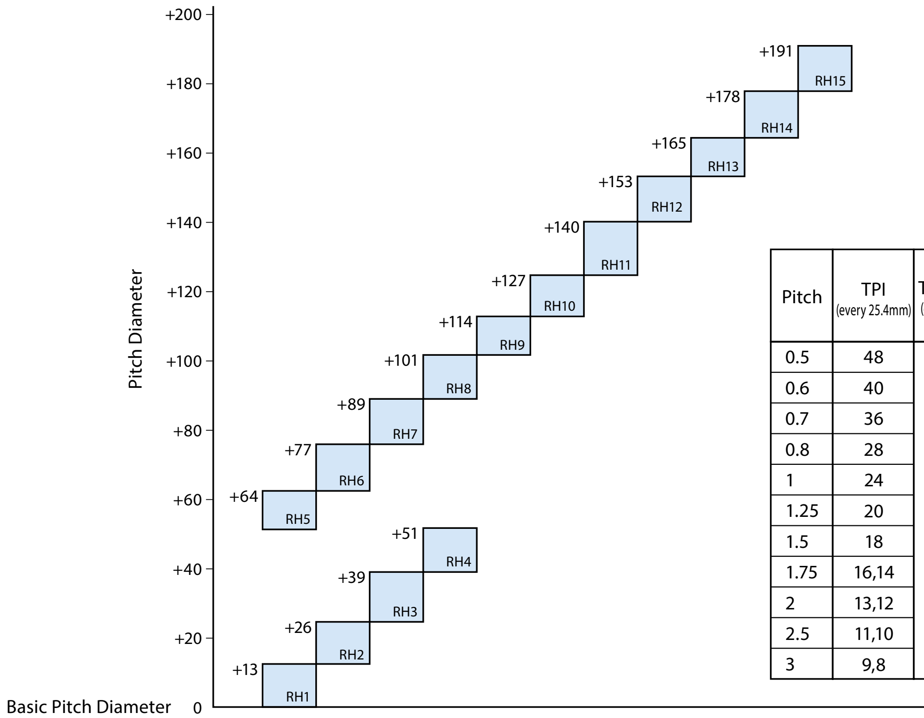
Table 30 – Unified Thread

Thread Size	TPI	Tap Thread Limit					
		3B Internal Thread	2B Internal Thread			Oversize	
						2B class+0.02～+0.03	2B class+0.04～+0.05
№2	64	RH2		**RH3**	RH4	RH5	RH6
№2 №3	56	RH3		**RH4**	RH5	RH6	RH7
№3 №4	48	RH3		**RH4**	RH5	RH6	RH7
№5	44	RH3	RH4	**RH5**	RH6	RH7	RH8
№4 №5 №6	40	RH3	RH4	**RH5**	RH6	RH7	RH8
№8	36	RH4		**RH5**	RH6	RH7	RH8
№6 №8 №10	32	RH4		**RH5**	RH6	RH7	RH8
№10 №12 №12	24 28	RH^{4}_{4}	RH5	**RH^{6}_{5}**	RH^{7}_{6}	RH^{8}_{8}	RH^{9}_{9}
U 1/4	20 28	RH^{4}_{4}	RH5	**RH^{6}_{5}**	RH^{7}_{6}	RH^{8}_{7}	RH^{9}_{8}
5/16	18 24	RH^{5}_{5}	RH6	**RH^{7}_{6}**	RH^{8}_{7}	RH^{9}_{8}	RH^{10}_{9}
3/8	16 24	RH^{5}_{5}	RH6	**RH^{7}_{6}**	RH^{8}_{7}	RH^{9}_{8}	RH^{10}_{9}
7/16	14 20	RH^{5}_{5}	RH^{6}_{6}	**RH^{7}_{7}**	RH^{8}_{8}	RH^{9}_{9}	RH^{10}_{10}
1/2	13 20	RH^{6}_{5}	RH^{7}_{6}	**RH^{8}_{7}**	RH^{9}_{8}	RH^{10}_{9}	RH^{11}_{10}
9/16	12 18	RH^{8}_{7}	RH^{9}_{8}	**RH^{10}_{9}**	RH^{11}_{10}	RH^{12}_{11}	RH^{13}_{12}
5/8	11 18	RH^{8}_{7}	RH^{10}_{8}	**RH^{11}_{9}**	RH^{12}_{10}	RH^{13}_{11}	RH^{14}_{12}
3/4	10 16	RH^{9}_{7}	RH^{11}_{9}	**RH^{12}_{10}**	RH^{13}_{11}	RH^{14}_{12}	RH^{15}_{13}
7/8	9 14	RH^{9}_{8}	RH^{11}_{10}	**RH^{12}_{11}**	RH^{13}_{12}	RH^{14}_{13}	RH^{15}_{14}
1	8 12	RH^{10}_{9}	RH^{12}_{11}	**RH^{13}_{12}**	RH^{14}_{13}	RH^{15}_{14}	RH^{16}_{15}

Thread Size	Pitch	Tap Thread Limit	
		Class 1 Internal Thread	Class 2 Internal Thread
M3	0.5	**RH3**	RH4
M4	0.7	**RH4**	RH5
M5	0.8	**RH4**	RH5
M6	1	**RH4**	RH5
M8 M10 M12	1.25	**RH5**	RH6
M10 M12	1.5	**RH5**	RH6
M12	1.75	**RH5**	RH7

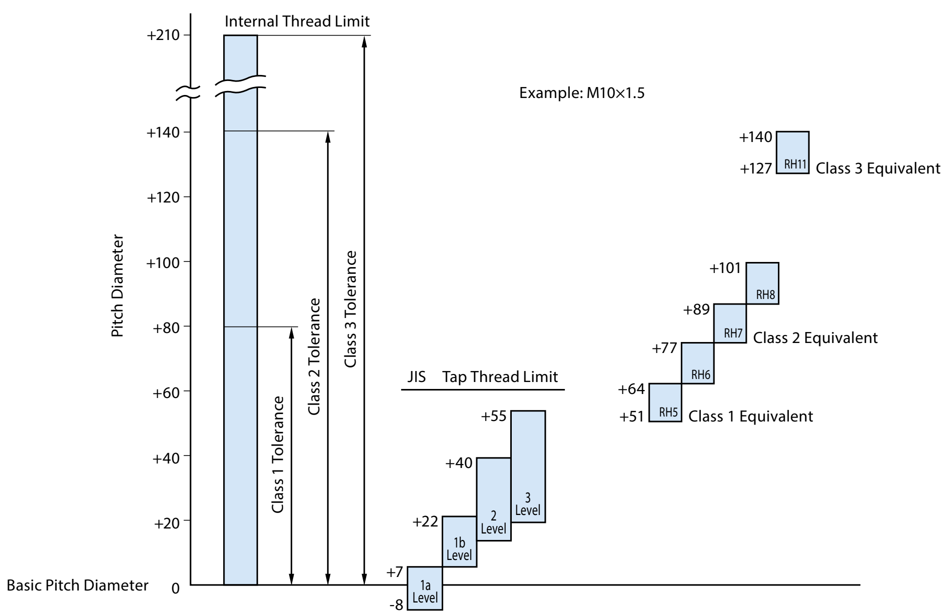
Thread Size	Tpi	Tap Thread Limit	
		A Class Internal Thread	B Class Internal Thread
PF1/8	28	**RH6**	RH12
PF1/4 PF3/8	19	**RH7**	RH14
PF1/2 PF3/4	14	**RH8**	RH16
PF1	11	**RH10**	RH20

9.7.2 Positional relationship of RH tolerance



Pitch	TPI (every 25.4mm)	Pitch Tolerance (every 25.4mm) ±	Half Thread Tolerance (Minutes)
0.5	48	0.010	30
0.6	40		
0.7	36		
0.8	28		
1	24		
1.25	20		
1.5	18		25
1.75	16,14		
2	13,12		
2.5	11,10		
3	9,8		

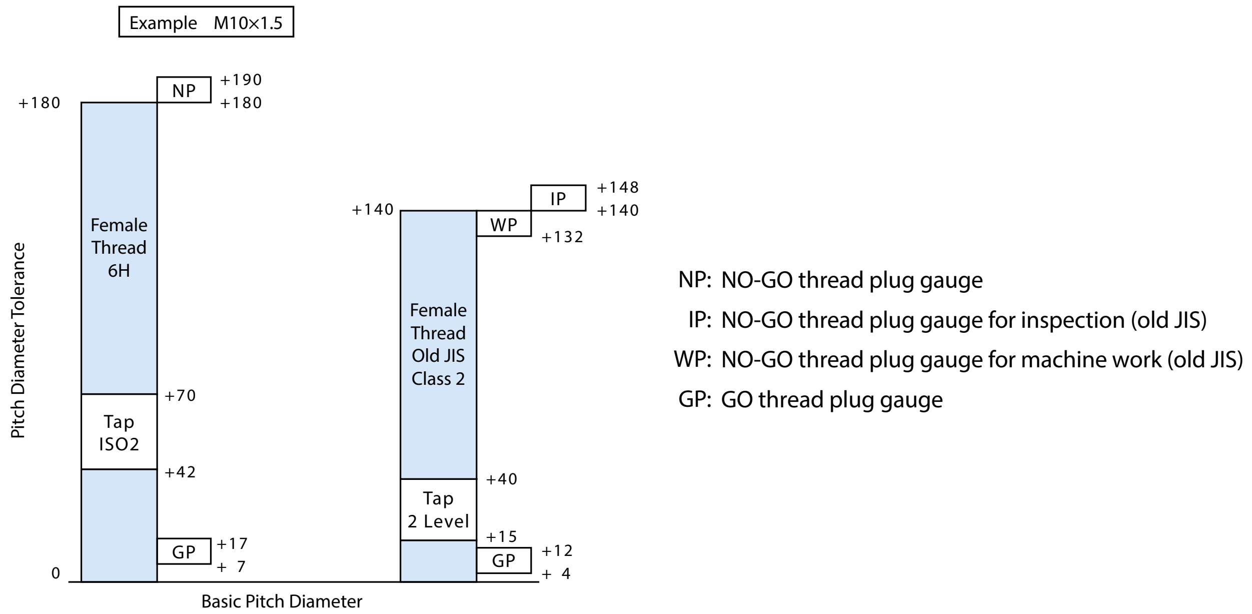
Note 1. RH tolerance/limit pitch diameter and tolerance (unit µm)
Upper tolerance/limit: 12.7n (n: RH number)
Lower tolerance: upper tolerance – tolerance
Tolerance: 12.7 µm



Part 10: Measurement of internal thread by thread gauge

The accuracy check of internal threads is done by thread limit plug gages (GO screw plug gages and NO-GO screw plug gages). The typical standard requires that the GO gage thread smoothly into the hole while the NO-GO gage does not turn more than two full rotations into the hole.
There is no set standard for the pitch tolerance and the angle tolerance of the thread, but it is included in the pitch diameter as its equivalent (shown in Fig 47). Fig 46 shows the relation of the internal thread, external thread, and the plug gage in reference to the pitch diameter tolerance.

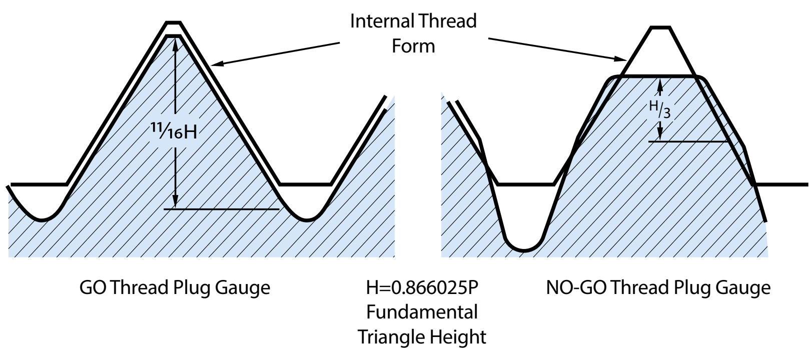
Fig 46 – Position Relationship of the Pitch Diameter



Thread profile of thread gauge

For the purpose of checking the virtual pitch diameter of the internal thread on the GO side and checking the simple pitch diameter on the NO-GO side, the ridges on the GO side have a profile similar to the basic profile, as shown in Fig. 47. The height is low and the contact height of the flank of the thread is minimal.

Fig 47 – Thread Form of Thread Plug Gauge



Part 11: Tap regrinding

By effectively determining the timing and method of regrinding, the tool life can be extended and will lead to cost savings. But for small diameter and SKS taps, careful consideration of the cost effectiveness must be taken.

11.1 Timing of regrinding

The following must be considered to determine the timing of when to send out the tools for regrinding.

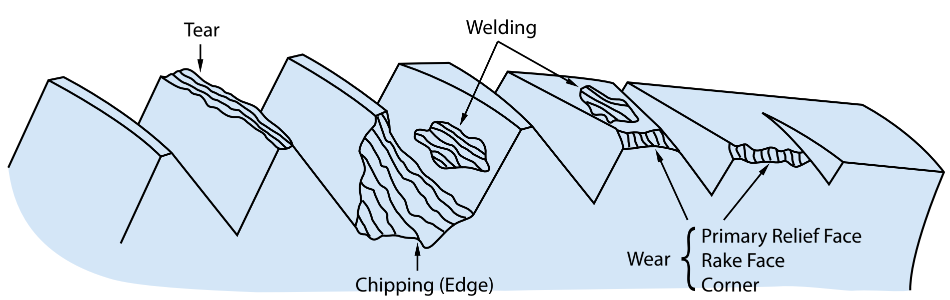
a. The tool has been damaged.
b. The tolerance of the internal thread does not pass standards.
c. The finish of the machined internal thread is poor.
d. Increased cutting resistance when tapping.
e. Unusual grinding sound when tapping.
f. Large changes to the chip shape.

In mass production plants, it is beneficial to investigate the average tool life and routinely send out the tools for regrinding as they reach the average tool life.

11.2 Regrinding part

Fig. 48 shows typical damage types on a tap. When regrinding, it is optimal to regrind both the flutes and the chamfer, but depending on the damage only one may suffice. It is best practice to keep the amount required for cutoff and regrind to a minimum.

Fig 48 – Tap Damage Types/Condition



11.3 Regrinding of the flute

11.3.1 Machine

A universal tool grinder is generally used for flute grinding. Using a crushing roller to form the grinding wheel is preferable but in general it can also be formed using bricks.

11.3.2 Grinding method and conditions

Flute regrinding requires only a depth of 1.5 to 2 times the thread height to provide a new surface. Additional grinding is permissible, but adversely impacts the tool strength and the number of allowed regrinds for that tool.
Also, the core diameter has a taper of 1/60 to 1/100 to provide additional strength; it is thus recommended to regrind according to this gradient.
Table 33 shows standard regrinding conditions.

Table 33 – Tap Flute Grinding Standard/General Conditions

Item \ Division	For General Purpose	For Mass Production
Grinding Wheel	WA60 ~ 0K	CBN120 ~ 170
Grinding Wheel Speed (m/min)	600	1,500 ~ 1,800
Depth of Cut (mm)	0.03	0.01 ~ 0.05
Feed (m/min)	1	1 ~ 3
Coolant	Wet	Wet

11.3.3 Rake angle

The rake angle is the most crucial aspect to regrinding the flute. If the rake angle becomes incompatible with the cutting material, many issues may develop, including inconsistency in internal thread tolerance, poor finish, and decreased tool life. In order to get the proper positive or negative cutting angles on the rake, according to the tap shank center, adjust the center of the grinding wheel either to the left or right. (Fig 49). To accurately inspect the cutting rake angle after it has been reground, follow the procedure shown in Fig 50, using a dial gage.

Fig 49 – How to Add/Apply/ Grind a Rake Angle

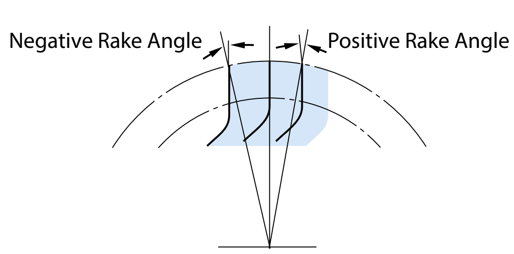
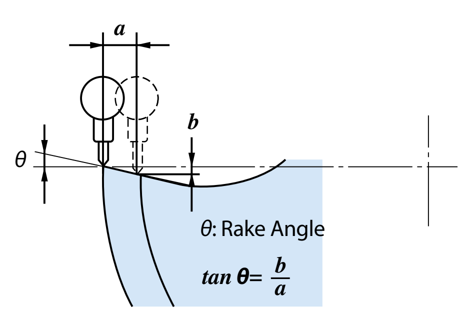


Fig 50 – Rake Angle Inspection Method

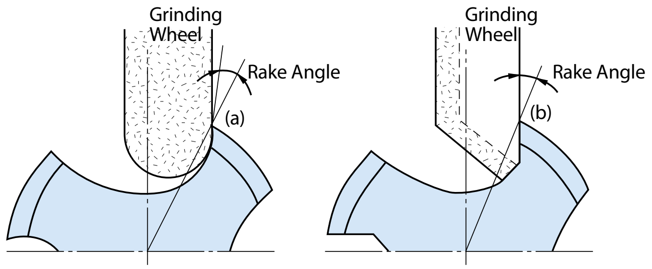


11.3.4 Other precautions

(a) Use an indexing system to properly regrind multiple cutting edges.
(b) Use proper cutting oil and avoid excessive cutting to prevent grind burn.
(c) Avoid edge wear.
(d) Keep the surface roughness at about 3.2Rz.
(e) Remove burrs with a buffer after regrind.

The limit of flute regrind, considering the influence on the thread tolerance and tool rigidity, is recommended to be 1/3~1/2 of a new tool edge thickness. Fig51 shows regrind conditions.

Fig 51 – Regrind conditions



(a) Hook angle grind by formed grinding wheel
(b) Rake angle grind by dish shape grinding wheel

11.4 Regrinding of the chamfer

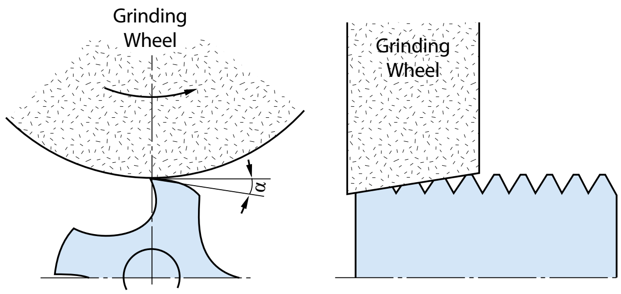
11.4.1 Machine, Procedures, Requirements

There are many ways, but the most common way, as shown in Fig 52, is to line up the centers of the tap and grind parallel to each other. The angle of the grinding wheel is set so that there is a relief (alpha), and either the tap or the wheel is moved by the cam.
In this method the tap is usually held by both center ends, but depending on the conditions only one end may be held.

The following is a basic regrind application guide:

Grinding Wheel	WA80K~L
Grinding Wheel Speed	1,500m/min
Cutting amount	0.03mm

Fig 52 – Regrind conditions



11.4.2 Chamfer relief

The chamfer relief angle may vary due to material, tap edge thickness, chamfer length, and thread lead. Angle is generally 2-3deg for hard materials and 3-4deg for soft materials.

11.4.3 Other precautions

(a) Keep the cutting edge runout to be within 0.03mm.
(b) Avoid grind burns.
(c) Avoid edge wear.
(d) Keep the surface roughness at about 3.2Rz.
(e) Keep the point diameter of the tap slightly smaller than the prepared hole.

In case of unified and metric threads: Max point diameter = Tap nominal size – 1.2 x pitch

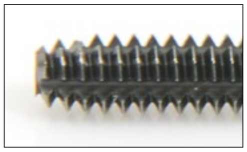
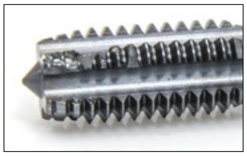
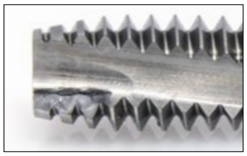
11.5 Regrinding spiral flute taps

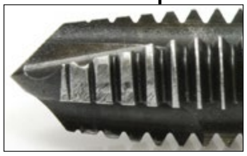
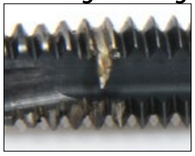
A special machine is needed when regrinding spiral fluted taps due to their unique helical flute design, which evacuates the chips back towards the shank. Generally, regrind is only done on the chamfer, but for any spiral fluted taps with a low helix angle flute regrinding is possible. There is a higher frequency of problems with flute tolerance or burrs, which may cause thread expansions to occur. In order to regrind the chamfer, it is necessary to have a machine that can regrind along the spiral groove and grind the relief. If the machine does not have this feature, the chamfer relief will not properly connect from the edge point to the back.

11.6 Regrinding spiral point taps

Spiral pointed taps have a 10° "gun" grind on the point of a straight fluted tap. Regrinding the chamfer requires the same procedure as a straight fluted tap. For flute regrinding one may either hold the tap between centers or by one end horizontally, creating the gun grind by tilting 10deg in the vertical direction. Make the rake angle 3° higher than a straight fluted tap.

Part 12: Troubleshooting

Problem	Cause	Solution
Chip Packing (Back Threaded Portion) 	Inappropriate spindle speed	Adjust RPM (lower or higher) for proper chip form
	Helix angle too large	Decrease helix angle or choose tap with low helix angle
	Chips not coiling / breaking properly	Use alternate coating
Chip Packing (Single Thread) 	*Occurs predominantly in horizontal applications*	
	Weak rake angle (positive)	Decrease rake angle
	Chips not evacuating properly	Use a POT style tap or a LHH / RHF
	Chips not coiling / breaking properly	Use alternate coating
Chipping During Reversal 	Chips left behind in flute during tap reversal	Improve wear resistance of tap
		Improve / add surface treatment / coating
	Material shrinkage	Increase coolant volume / concentration to control heat
Chipping Due to Wear 	Tap substrate not suitable for work material	Improve wear resistance of tap
		Improve / add surface treatment / coating
	Cutting action work hardened material	Shorten chamfer length
Chipping of Land Edge 	Occurs when tap either hits bottom or entrance of hole	Avoid hitting the bottom of the hole, check stroke length, alignment and hole size
Chipping of Land Axially 	Occurs when tap either hits bottom or entrance of hole	Avoid hitting the bottom of the hole, check stroke length, alignment and hole size
Chipping of Chamfer 	Tap substrate not suitable for work material	Improve wear resistance of tap
	Inappropriate pre-drill size	Select suitable pre-drill size

Problem	Cause	Solution
Premature Tap Wear 	Inappropriate spindle speed	Reduce spindle speed
	Possible work hardening of pre-drilled hole	Prevent work hardening of pre-drilled hole
	Inappropriate thread relief	Use proper thread relief
	Inappropriate chamfer length	Adjust chamfer length
	Inappropriate lubrication	Change coolant method
		Increase volume / concentration
		Apply surface coating / treatment
Welding / Galling 	Inappropriate spindle speed	Reduce spindle speed
	Inappropriate lubrication	Change coolant method
		Increase volume / concentration
		Apply surface coating / treatment
Deformed Lobes 	Possible work hardening of pre-drilled hole	Prevent work hardening of pre-drilled hole
	Inappropriate spindle speed	Reduce spindle speed
	Inappropriate pre-drill size	Increase pre-drill hole size as much as possible
	Inappropriate lubrication	Change coolant method
		Increase volume / concentration
		Apply surface coating / treatment
	Tap substrate not suitable for material	Improve wear resistance of tap
Tap Breakage 	Possible chip packing	Avoid chip packing
	Inappropriate pre-drill size	Increase pre-drill hole size as much as possible
	Inappropriate spindle speed	Reduce spindle speed
	Possible runout or tapered hole	Reduce runout and assure hole is straight
	Too high of torque generated	Use tap holder with torque adjustment / limiting feature
	Possible tap collision with bottom of hole	Avoid hitting the bottom of the hole, check stroke length, alignment and hole size
Overcutting / Oversized Threads 	Inconsistent feed of spiral fluted style tap	Use compensating tension / compression tap holder
		Adjust feed rate appropriately
		Check CNC program
	Inconsistent feed of spiral pointed style tap	Use compensating tension / compression tap holder
		Adjust feed rate appropriately
		Check CNC program
Tearing on Flanks 	Inappropriate thread relief / rake angle	Use sharper / freer cutting relief and angle
	Inappropriate lubrication	Change coolant method
		Increase volume / concentration
		Apply surface coating / treatment
Extremely Torn Threads 	Possible welding / galling	Select appropriate cutting conditions
	Possible chip packing	Select appropriate cutting conditions
	Inappropriate thread relief	Use sharper thread relief
	Inappropriate lubrication	Change coolant method
		Increase volume / concentration
		Apply surface coating / treatment
Chips Remain at Bottom 	Inappropriate geometry of tap	Reduce chamfer relief angle
		Use thinner land width
		Reduce chamfer length
		Reduce cutting angle

Part 13: Threaded Hole Diameter

Decimal Equivalents

Drill Size	Decimal	Drill Size	Decimal	Drill Size	Decimal	Drill Size	Decimal	Drill Size	Decimal
0.1mm	0.0039	49	0.0730	4.5mm	0.1772	P	0.3230	16mm	0.6299
0.2mm	0.0079	48	0.0760	15	0.1800	21/64	0.3281	41/64	0.6406
0.3mm	0.0118	5/64	0.0781	14	0.1820	Q	0.3320	16.5mm	0.6496
80	0.0135	47	0.0785	13	0.1850	8.5mm	0.3346	21/32	0.6562
79	0.0145	2mm	0.0787	3/16	0.1875	R	0.3390	17mm	0.6693
1/64	0.0156	46	0.0810	12	0.1890	11/32	0.3438	43/64	0.6719
0.4mm	0.0157	45	0.0820	11	0.1910	S	0.3480	11/16	0.6875
78	0.0160	44	0.0860	10	0.1935	9mm	0.3543	17.5mm	0.6890
77	0.0180	43	0.0890	9	0.1960	T	0.3580	45/64	0.7031
0.5mm	0.0197	42	0.0935	5mm	0.1969	23/64	0.3594	18mm	0.7087
76	0.0200	3/32	0.0938	8	0.1990	U	0.3680	23/32	0.7188
75	0.0210	41	0.0960	7	0.2010	9.5mm	0.3740	18.5mm	0.7283
74	0.0225	40	0.0980	13/64	0.2031	3/8	0.3750	47/64	0.7344
0.6mm	0.0236	2.5mm	0.0984	6	0.2040	V	0.3770	19mm	0.7480
73	0.0240	39	0.0995	5	0.2055	W	0.3860	3/4	0.7500
72	0.0250	38	0.1015	4	0.2090	25/64	0.3906	49/64	0.7656
71	0.0260	37	0.1040	3	0.2130	10mm	0.3937	19.5mm	0.7677
0.7mm	0.0276	36	0.1065	5.5mm	0.2165	X	0.3970	25/32	0.7813
70	0.0280	7/64	0.1094	7/32	0.2188	Y	0.4040	20mm	0.7874
69	0.0292	35	0.1100	2	0.2210	13/32	0.4063	51/64	0.7969
68	0.0310	34	0.1110	1	0.2280	Z	0.4130	20.5mm	0.8071
1/32	0.0313	33	0.1130	A	0.2340	10.5mm	0.4134	13/16	0.8125
0.8mm	0.0315	32	0.1160	15/64	0.2344	27/64	0.4219	21mm	0.8268
67	0.0320	3mm	0.1181	6mm	0.2362	11mm	0.4331	53/64	0.8281
66	0.0330	31	0.1200	B	0.2380	7/16	0.4375	27/32	0.8438
65	0.0350	1/8	0.1250	C	0.2420	11.5mm	0.4528	21.5mm	0.8465
0.9mm	0.0354	30	0.1285	D	0.2460	29/64	0.4531	55/64	0.8594
64	0.0360	29	0.1360	1/4&E	0.2500	15/32	0.4688	22mm	0.8661
63	0.0370	3.5mm	0.1378	6.5mm	0.2559	12mm	0.4724	7/8	0.8750
62	0.0380	28	0.1405	F	0.2570	31/64	0.4844	22.5mm	0.8858
61	0.0390	9/64	0.1406	G	0.2610	12.5mm	0.4921	57/64	0.8906
1mm	0.0394	27	0.1440	17/64	0.2656	1/2	0.5000	23mm	0.9055
60	0.0400	26	0.1470	H	0.2660	13mm	0.5118	29/32	0.9063
59	0.0410	25	0.1495	I	0.2720	33/64	0.5156	59/64	0.9219
58	0.0420	24	0.1520	7mm	0.2756	17/32	0.5313	23.5mm	0.9252
57	0.0430	23	0.1540	J	0.2770	13.5mm	0.5315	15/16	0.9375
56	0.0465	5/32	0.1563	K	0.2810	35/64	0.5469	24mm	0.9449
3/64	0.0469	22	0.1570	9/32	0.2813	14mm	0.5512	61/64	0.9531
55	0.0520	4mm	0.1575	L	0.2900	9/16	0.5625	24.5mm	0.9646
54	0.0550	21	0.1590	M	0.2950	14.5mm	0.5709	31/32	0.9688
1.5mm	0.0591	20	0.1610	7.5mm	0.2953	37/64	0.5781	25mm	0.9843
53	0.0595	19	0.1660	19/64	0.2969	15mm	0.5906	63/64	0.9844
1/16	0.0625	18	0.1695	N	0.3020	19/32	0.5938	1	1.000
52	0.0635	11/64	0.1719	5/16	0.3125	39/64	0.6094		
51	0.0670	17	0.1730	8mm	0.3150	15.5mm	0.6102		
50	0.0700	16	0.1770	O	0.3160	5/8	0.6250		

Tap Drill Sizes - For 70% Thread - Inch

Size	TPI	Tap Drill	Size	TPI	Tap Drill	Size	TPI	Tap Drill	Size	TPI	Tap Drill
NO. 0	80 NF	0.0486	NO. 8	36 NF	0.1387	7/16	20 NF	0.3920	1	12 NF	0.9242
NO. 1	64 NC	0.0588	NO. 10	24 NC	0.1521	1/2	13 NC	0.4301	1-1/8	7 NC	0.9951
NO. 1	72 NF	0.0604	NO. 10	32 NF	0.1616	1/2	20 NF	0.4545	1-1/8	12 NF	1.0492
NO. 2	56 NC	0.0698	NO. 12	24 NC	0.1781	9/16	12 NC	0.4867	1-1/4	7 NC	1.1201
NO. 2	64 NF	0.0718	NO. 12	28 NF	0.1835	9/16	18 NF	0.5120	1-1/4	12 NF	1.1742
NO. 3	48 NC	0.0801				5/8	11 NC	0.5423	1-3/8	6 NC	1.2235
NO. 3	56 NF	0.0828				5/8	18 NF	0.5745	1-3/8	12 NF	1.2992
NO. 4	40 NC	0.0893	1/4	20 NC	0.2045	11/16	11 NS	0.6048	1-1/2	6 NC	1.3485
NO. 4	48 NF	0.0931	1/4	28 NF	0.2175	11/16	16 NS	0.6307	1-1/2	12 NF	1.4242
NO. 5	40 NC	0.1023	5/16	18 NC	0.2620	3/4	10 NC	0.6591	1-3/4	5 NC	1.5681
NO. 5	44 NF	0.1043	5/16	24 NF	0.2746	3/4	16 NF	0.6932	2	4-1/2 NC	1.7979
NO. 6	32 NC	0.1096	3/8	16 NC	0.3182	7/8	9 NC	0.7740			
NO. 6	40 NF	0.1153	3/8	24 NF	0.3371	7/8	14 NF	0.8101			
NO. 8	32 NC	0.1356	7/16	14 NC	0.3726	1	8 NC	0.8863			

Tap Drill Sizes - For 70% Thread - Metric

Size	Pitch	Drill (mm)	Inch Equiv.	Size	Pitch	Drill (mm)	Inch Equiv.	Size	Pitch	Drill (mm)	Inch Equiv.	Size	Pitch	Drill (mm)	Inch Equiv.
M1.6	0.35	1.28	0.0505	M8	1.25	6.86	0.2702	M16	1.5	14.64	0.5762	M33	3.5	29.82	1.1739
M2.0	0.4	1.64	0.0644	M8	1.0	7.09	0.2792	M18	2.5	15.73	0.6192	M33	2.0	31.18	1.2276
M2.5	0.45	2.09	0.0823	M10	1.5	8.64	0.3400	M18	1.5	16.64	0.6550	M36	4.0	32.36	1.2741
M3.0	0.5	2.55	0.1002	M10	1.25	8.86	0.3489	M20	2.5	17.73	0.6979	M36	3.0	33.27	1.3099
M4.0	0.7	3.36	0.1324	M12	1.75	10.41	0.4098	M20	1.5	18.64	0.7337	M39	4.0	35.36	1.3922
M5.0	0.8	4.27	0.1682	M12	1.25	10.86	0.4277	M24	3.0	21.27	0.8375	M39	3.0	36.27	1.4280
M6.0	1.0	5.09	0.2004	M14	2.0	12.18	0.4796	M24	2.0	22.18	0.8733				
M6.3	1.0	5.39	0.2122	M14	1.5	12.64	0.4975	M30	3.5	26.82	1.0558				
M7.0	1.0	6.09	0.2398	M16	2.0	14.18	0.5583	M30	2.0	28.18	1.1095				

Pipe Tap Drill Sizes

National Pipe Threads								British & ISO Pipe Threads					
Size	TPI	NPT		NPTF		NPS		Size	Whit. Form TPI	BSPT (Taper)		BSPP (Parallel)	
		Tap Drill	Decimal	Tap Drill	Decimal	Tap Drill	Decimal			Tap Drill (mm)	Inch Equiv.	Tap Drill (mm)	Inch Equiv.
1/16	27	C	0.246	D	0.246	1/4	0.250	1/16	28	6.3	0.248	6.5	0.256
1/8	27	Q	0.332	R	0.339	11/32	0.344	1/8	28	8.4	0.331	8.8	0.346
1/4	18	7/16	0.438	7/16	0.438	7/16	0.438	1/4	19	11.2	0.441	11.8	0.465
3/8	18	9/16	0.562	37/64	0.578	37/64	0.578	3/8	19	14.75	0.581	15.25	0.600
1/2	14	45/64	0.703	45/64	0.703	23/32	0.719	1/2	14	18.25	0.719	19.0	0.748
3/4	14	29/32	0.906	59/64	0.922	59/64	0.922	3/4	14	23.75	0.935	24.5	0.965
1	11-1/2	1-9/64	1.141	1-5/32	1.156	1-5/32	1.156	1	11	30.0	1.181	30.75	1.211
1-1/4	11-1/2	1-31/64	1.484	1-1/2	1.500	1-1/2	1.500	1-1/4	11	38.5	1.516	39.5	1.555
1-1/2	11-1/2	1-23/32	1.734	1-47/64	1.7342	1-3/4	1.750	1-1/2	11	44.5	1.752	45.5	1.791
2	11-1/2	2-3/16	2.203	2-7/32	2.219	2-7/32	2.219	2	11	56.0	2.205	57.0	2.244

shaping your dreams

Safe use of cutting tools

- Use safety cover, safety glasses and safety shoes during operation.
- Do not touch cutting edges with bare hands.
- Do not touch cutting chips with bare hands. Chips will be hot after cutting.
- Stop cutting when the tool becomes dull.
- Stop cutting operation immediately if you hear any abnormal cutting sounds.
- Do not modify tools.
- Please use appropriate tools for the operation. Check dimensions to ensure proper selection.

FOR MORE INFORMATION CONTACT US

800-837-2223 • osgtool.com