Tapping Questions
Handbook

TAPMATIC

For Immediate Assistance call (800) 854-6019
or E-mail us at info@tapmatic.com

Table of Contents

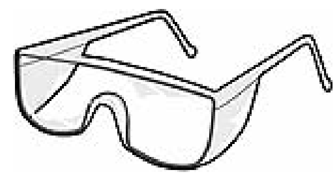
Tapping Safety

⚠ WARNING

To avoid serious injury and ensure best result for your tapping operation, Please Read Carefully All operator and safety instructions provided for your tapping attachment as well as all other safety instructions that are applicable, especially those for your machine tool.

1. Proper Clothing: The rotating spindle of a machine tool can snag loose fitting clothing, jewelry or long hair. **Never** wear jewelry, long sleeves, neckties, gloves or anything else that could become caught when operating a machine tool. Long hair must be restrained or netted to prevent it from becoming entangled in rotating spindle. Steel toed boots should also be worn in any machine environment.

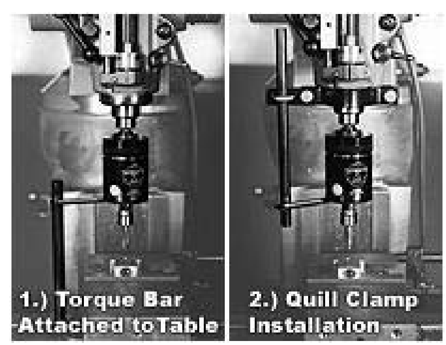
2. Proper Eye Protection: Always wear safety glasses with side shields to protect your eyes from flying particles.



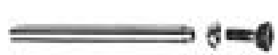
3. Proper Work Piece Fixturing: Never hold the work piece or the vise it is held in, by hand. The work piece must be clamped firmly to the table of the machine so that it cannot move, rotate or lift.

4. Proper Stop Arm / Torque Bar Installation
For Self-Reversing Attachments On Conventional Machines:

Quill Clamp Capacity	**Order #**	**Max Tap Size**
1 1/2 – 2 3/8	29099	1/2
2 3/8 – 4 1/2	290991	3/4



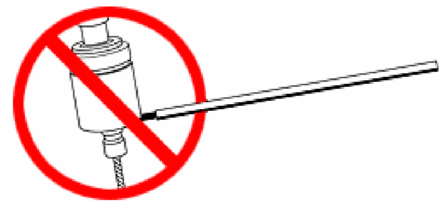
Torque Bar Assembly	**Order #**	**Max Tap Size**
Table Mount	29097	3/4
Heavy Duty Table Mount	29096	1 3/4



(Continued)

Tapping Safety (Continued)

Never extend the length of the standard stop arm supplied with your tapping attachment. A lengthened stop arm could break free, hitting the operator and causing serious injury.

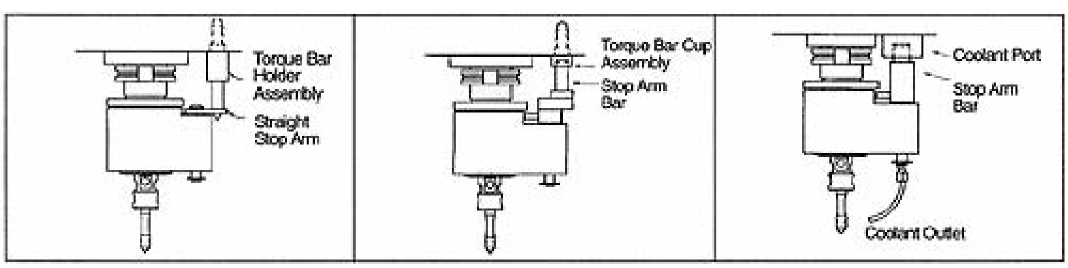


Never hold the stop arm by hand. On reversal, full power of the machine is transmitted through the stop arm and the operator could be seriously injured.

Always mount a torque bar to hold the tapping attachment's stop arm from rotating. The torque bar **must** be mounted securely to the table or quill of your machine. The torque bar installation **must** be stronger than the largest tap in the capacity range of your tapping attachment.

7. Continuous High Production Manual Tapping: Models for use on conventional drill press or milling machines. Speed is a critical factor in tapping. Please always refer to recommended tapping speed chart. Tapmatic Torque Control Reversing Tapping Attachments employ a planetary gear reversing mechanism that increases speed by a 1.75 x 1 ratio. This means that a machine speed of 2,000 RPM results in a reversing speed of 3,500 RPM. It is strongly recommended that you consider the **AVERAGE TAPPING SPEED** rather than machine speed when calculating your cycle time. For example, if machine speed is 1,500 RPM, reverse speed is 2,625 RPM, making your **AVERAGE TAPPING SPEED 2,062 RPM.**
You must not exceed the maximum allowable speed marked on your tapping attachment.

8. On Machining Centers: The same rule for installation applies whether using the torque bar holder assembly with stop arm, torque bar cup assembly or stop block assembly. "**Always** be sure that the installation is stronger than the largest tap being used."



9. Always Be Aware Of The Potential Hazards Of A Machining Operation: Sometimes working with your machine can seem routine. You may find that you are no longer concentrating on the operation. A feeling of false security can lead to serious injury. **Always** be alert to the dangers of the machines with which you work. **Always** keep hands. Body parts, clothing, jewelry and hair out of the areas of operation, when the machine spindle is rotating. Areas of operation include the immediate point of machining and all transmission components including the tapping attachment. Never bring your hand, other body parts or anything attached to your body into any of these areas until the machine spindle is completely stopped.

(Continued)

Tapping Safety (Continued)

10. The tapping attachment housing, drive spindle and tap itself can become hot to the touch after operation. Use caution when removing the attachment from the machine of handling.

11. Be aware of any other applicable safety instructions requirements.

Check List For Good Tapping:

Never use the tapping unit before reading all safety instructions for it as well as those for the machine it is to be used on.
Is tap sharp and of correct design for current job?
Is tap in proper alignment with drilled hole?
Is machine speed correct?
Is machine feed correct?
Is machine stop set properly so tap releases in neutral rather than bottoming in work piece or fixture?
Is work piece held rigidly against rotation and upward movement?
Is clearance between the drilled hole and tap sufficient at start position to allow the tap to clear the hole upon retraction?
Is the stop arm of the tapping attachment held rigidly against rotation by the torque bar extending from the machine?

References for this Safety Information include but are not limited to:

American National Standards Institute
ANSI B11.8-1983 (Adopted May 31, 1983 by Department of Defense)
Coastal Video Communications Corporation
Machine Guarding Copyright 1994

Society Of Manufacturing Engineers
Tool and Manufacturing Engineers Handbook Volume 1 Machining
(Library of Congress Catalog No. 82-060312)

Tapping Checklist

Work Piece
All Applications

- What are the size and depth of the hole?
- Will it be a through or blind hole?
- Is the hole drilled to the correct size?
- Is the work piece rigidly held against rotation and upward movement?
- If tapping a bottom-hole, does drilled depth allow for chamfer teeth of tap and sufficient clearance to keep tap from bottoming out in hole?
- What is material and hardness of the work piece?

Tap
All Applications

- Do you have the correct tap design for the application?
- What are the tap sizes and styles?
- What material is the tap made from?
- Is the tap sharp?
- Is the tap properly aligned with the drilled hole?
- Is there sufficient clearance between the tap and the hole to allow for retraction?
- Who is tap manufacturer? What speed do they recommend for optimum performance of their tap in this material?

Machine Tool
Manual Applications

- Is machine stop set so the tap releases in neutral to prevent bottoming?
- Is the machine retraction correct for tapping attachment being used?
- Is the torque control set to prevent tap breakage?
- Is depth control set to correspond with machine stop to provide the total thread depth required and prevent bottoming?

CNC Applications

- What type of machine is in use?
- What is the horsepower?
- What is the spindle taper?
- What is the method of fixturing?
- Are machine feed and speed set correctly?

All Applications

- Is the proper cutting fluid or lubricant being used for lubricating the tap?

Tapping Attachment
All Applications

- Is the correct Tapmatic tapping attachment being used for the specific job requirements?

CNC Applications

- Is the machine retraction correct for the tapping attachment being used?

(Continued)

Tapping Checklist (Continued)

Tapping Head Installation and Machine Set Up All Applications

- Never perform any installation or programming, before reading the operator instructions ac companying the tapping attachment and the machine as well as the tap manufacturers' recommendations.
- With a self-reversing tap chuck for manual or CNC operations, it is important to make sure that stop arm is strong enough to prevent torque bar from bending or deflecting. Machine torque bar must be stronger than largest tap.
- Is clearance between the drilled hole and tap sufficient at start position to allow the tap to clear the hole upon retraction?
- If a bottom hole is being tapped is there sufficient chip clearance?

Manual Applications

- If torque control attachment is being used, is torque set correctly so that tap will not break if accidentally bottomed?
- If depth control feature is employed, is it set correctly to cooperate with machine stop, provide total thread depth required and prevent engagement with bottom?

CNC Applications

- The machine retraction must be correct for the tapping attachment being used.
- What is the feed rate?
- What is the actual tapping speed?
- What is the clearance plane height?
- Is the potentiometer canceled?
- Be sure to follow programming instructions for the tool.
- When using a self-reversing head, has the ramp, dwell or exact stop been disabled?

Determining Correct Speed Within Specified Range

Compilation of Guidelines From Tap Manufacturers And Other Sources For Cutting or Cold-Forming of Threads In Relation To Work Piece Material

Cutting Speed For Tapping: Several factors, singly or in combination can cause very great differences in the permissible tap-pingspeed. Theprincipalfactorsaffectingthetappingspeedarethepitchofthethread,thechamferlengthonthetap, the percentage of full thread to be cut, the length of the hole to be tapped, the cutting fluid used, whether the threads are straight or tapered, the machine tool used to perform the operation, and the material to be tapped. From Machinery's Handbook 23rd edition.
If your coolant does not contain EP additives or its lubrication quality is low, start from the lower speeds in the range. Roll form taps in particular require good lubrication because of the high friction forces involved. As the lubrication quality of a coolant is often unknown, we recommend you start from the lower speeds in the range.

These Factors Apply to *Everyone's* Tapping Speed Charts

–%	Ten Factors Requiring Lower Speeds		Ten Factors Permitting Higher Speeds	+ %
-20	Poor Lubrication	1	Good Lubrication	+20
-15	High Tensile Strength Of Material	2	Low Tensile Strength Of Material	+15
-15	Large Thread Diameter	4	Small Thread Diameter	+15
-10	High Alloy Materials	3	Low Alloy Materials	+10
-10	Thread Depth More Than 1.5 x Dia.	5	Thread Depth 1.5 x Dia. Or Less	+10
-10	Thread Pitch Coarse	6	Thread Pitch Fine	+10
-5	Drill Size More than 65% of Thread	7	Drill Size 65% or Less Thread	+5
-5	Tap Lead Less Than 3.5 Thread	8	Tap Lead More Than 3.5 Threads	+5
-5	Blind Holes	9	Through Holes	+5
-5	Free Running Spindle Inaccurate Pitch Control Hydraulic/Air Feed	10	Synchronous Spindle Lead Screw CNC Control	+5

Example:
Tap Size: 1/4"-28 Coated, **Material:** Aluminum Die Cast, From Chart 688-1375 RPM **RPM Spread** = 687

Minus Factors:		Plus Factors:	
High Tensile Strength	-15	Coolant With Good EP	+20
Thread Depth 3 x Dia.	-10	Small Thread Diameter	+15
Drill Size = 75% Thd.	-5	Pitch Fine	+10
Blind Hole	-5	Lead 3.5 Threads	+5
TOTAL	-35	CNC Machine	+5
		TOTAL	+55

Apply The Factors Against The RPM Spread of 687
+.55 X 687 = 378 Added to minimum RPM 688 = **1066 New Mininimum RPM**
-.35 X 687 = 240 Substracted from maximum RPM 1375 = **1135 New Maximum RPM**
Common Sense Rule: Begin with min RPM and work up to optimum efficiency and tap life.

TAPMATIC SFT, SynchroFlex, Rigid Tap Drivers

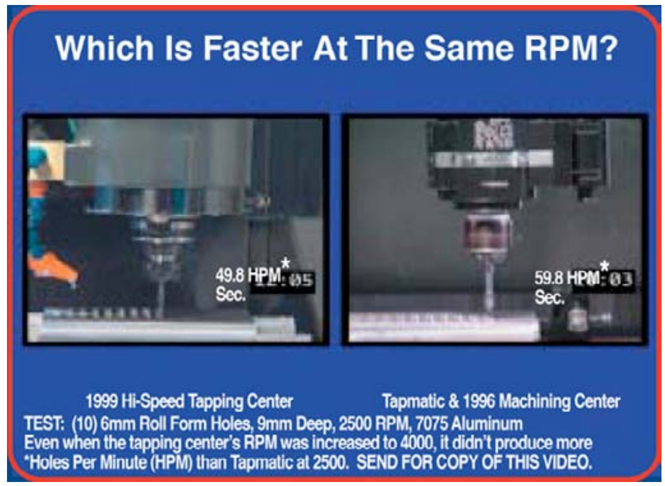
Model	RPM	Capacity
SFT10	Limited	#2 - #10-32
SFT50	Only By	#10 - 1/2"
SFT75	Rigid	1/2"-13 - 3/4"-10
SFT100	Tapping	3/4"-10 - 1"
	Program	

TAPMATIC CST, Constant Speed, Self-Reversing Tapping Heads

Model	Spindle	RPM	Spindle	RPM	Capacity
RDT15 HD and HS	Heavy Duty	5000 RPM	High Speed	6000 RPM	(#0-#6, M1-M3)
RDT25 HD and HS	Heavy Duty	3500 RPM	High Speed	4000 RPM	(#4-1/4", M2.5-M6)
RDT50 HD and HS	Heavy Duty	2000 RPM	High Speed	2500 RPM	(#10-1/2", M4.5-M12)
RDT75		1500 RPM			(#3/8"-3/4", M10-M20)
RDT85		1200 RPM			(#1/2"-1", M12-M25)

Tools come with Popular Inch & Metric Size Straight Shanks adaptable to CAT, BT and HSK. All sizes available. See Price List.

Comparison Test Between Fastest Rigid Tapping Method And Constant Speed Tapping



Seven Essential Steps For Trouble Free Performance With Self-Reversing Tapping Heads

1. **Choose the proper tap:** Follow your tap manufacturers recommendations for your specific application.
2. **Calculate the correct tapping speed** from the adjacent charts and the rules on this page.
3. **Common sense rule:** Begin conservatively and increase speed until optimum results are obtained.
4. **Select the best tool** for your application or applications. High production with one tap size *(Don't compromise)*, low production with a variety of taps, *(Choose the tool that best covers range.)*
5. **Follow our programming instructions exactly,** *and absolutely* ***make sure ramp or exact stop has been eliminated from tapping cycle.*** Leaving it in will increase tapping time 30% increase thread depth variations substantially, and wear out the tapping head prematurely.
6. **Follow our installation instructions exactly** and lock orientation collar in place once stop arm is in proper position. Then fix it positively with the locking screw provided.
7. **Schedule preventative maintenance.** Disassembly, cleaning, re-lubricating, and reassembly takes no more than half an hour. Just consider what the head does for the machine by eliminating its reversal related wear and tear. Simple maintenance will keep the head working efficiently, and pay big dividends in trouble free production.

(Continued)

Speed Chart/Standard Taps

	Low Carbon Steel, Medium Carbon Steel	High Carbon Steel, High Strength Steel, Tool Steel	High Strength Steel, Tool Steel Hardened	Stainless 303, 304, 316	Stainless 410, 430, 17-4 Hardened	Stainless 17-4 Annealed	Titanium Alloys	Nickel Base Alloys	Aluminium Alloys	Aluminium Die Cast	Magnesium	Brass, Bronze	Copper	Cast Iron
A) m/min B) m/min	10–20 20–40	8–12 15–25	4–6 12–18	6–12 8–15	3–5 4–10	6–12 8–20	4–8 8–15	3–5 5–10	15–25 40–60	10–15 30–40	15–25 40–60	15–25 40–60	8–12 15–25	10–20 20–30
Tap size	A) rpm for HSS-E uncoated taps B) rpm for HSS-E coated taps													
M2 (#2)	1600–3200 3200–6350	1250–1900 2400–4000	640–800 1900–2850	800–1900 1250–2400	480–800 640–1600	800–1900 1250–3200	640–1250 1250–2400	480–800 800–1600	2400–4000 6350–9550	1600–2400 4750–6350	2400–4000 6350–9550	2400–4000 6350–9550	1250–1900 2400–4000	1600–3200 3200–4750
M3 (#5)	1050–2100 2100–4250	850–1250 1600–2650	420–530 1250–1900	530–1250 850–1600	320–530 420–1050	530–1250 850–2100	420–850 850–1600	320–530 530–1050	1600–2650 4250–6350	1050–1600 3200–4250	1600–2650 4250–6350	1600–2650 4250–6350	850–1250 1600–2650	1050–2100 2100–3200
M4 (#8)	800–1600 1600–3200	640–950 1200–2000	320–400 950–1450	400–950 640–1200	240–400 320–800	400–950 640–1600	320–640 640–1200	240–400 400–800	1200–2000 3200–4750	800–1200 2400–3200	1200–2000 3200–4750	1200–2000 3200–4750	640–950 1200–2000	800–1600 1600–2400
M5 (#10)	640–1250 1250–2550	510–760 950–1600	250–320 760–1150	320–760 510–950	190–320 250–640	320–760 510–1250	250–510 510–950	190–320 320–640	950–1600 2550–3800	640–950 1900–2550	950–1600 2550–3800	950–1600 2550–3800	510–760 950–1600	640–1250 1250–1900
M6 (1/4)	530–1050 1050–2100	420–640 800–1350	210–270 640–950	270–640 420–800	160–270 210–530	270–640 420–1050	210–420 420–800	160–270 270–530	800–1350 2100–3200	530–800 1600–2100	800–1350 2100–3200	800–1350 2100–3200	420–640 800–1350	530–1050 1050–1600
M7	450–910 910–1800	360–550 680–1150	180–230 550–820	230–550 360–680	140–230 180–450	230–550 360–910	180–360 360–680	140–230 230–450	680–1150 1800–2750	450–1150 1350–1800	680–1150 1800–2750	680–680 1800–2750	360–550 680–1150	450–910 910–1350
M8 (5/16)	400–800 800–1600	320–480 600–990	160–200 480–720	200–480 320–600	120–200 160–400	200–480 320–800	160–320 320–600	120–200 200–400	600–990 1600–2400	400–600 1200–1600	600–990 1600–2400	600–990 1600–2400	320–480 600–990	400–800 800–1200
M9	350–710 710–1400	280–420 530–880	140–180 420–640	180–420 280–530	110–180 140–350	180–420 280–710	140–280 280–530	110–180 180–350	530–880 1400–2100	350–530 1050–1400	530–880 1400–2100	530–880 1400–2100	280–420 530–880	350–710 710–1050
M10 (3/8)	320–640 640–1250	250–380 480–800	130–160 380–570	160–380 250–480	100–160 130–320	160–380 250–640	130–250 250–480	100–160 160–320	480–800 1250–1900	320–480 950–1250	480–800 1250–1900	480–800 1250–1900	250–380 480–800	320–640 640–950
M12 (1/2)	270–530 530–1050	210–320 400–660	110–130 320–480	130–320 210–400	80–130 110–270	130–320 210–530	110–210 210–400	80–130 130–270	400–660 1050–1600	270–400 800–1050	400–660 1050–1600	400–660 1050–1600	210–320 400–660	270–530 530–800
M14 (9/16)	230–450 450–910	180–270 340–570	90–110 270–410	110–270 180–340	70–110 90–230	110–270 180–450	90–180 180–340	70–110 110–230	340–570 910–1350	230–340 680–910	340–570 910–1350	340–570 910–1350	180–270 340–570	230–450 450–680
M16 (5/8)	200–400 400–800	160–240 300–500	80–100 240–360	100–240 160–300	60–100 80–200	100–240 160–400	80–160 160–300	60–100 100–200	300–500 800–1200	200–300 600–800	300–500 800–1200	300–500 800–1200	160–240 300–500	200–400 400–600
M18 (11/16)	180–350 350–710	140–210 270–440	70–90 210–320	90–210 140–270	50–90 70–180	90–210 140–350	70–140 140–270	50–90 90–180	270–440 710–1050	180–270 530–710	270–440 710–1050	270–440 710–1050	140–210 270–440	180–350 350–530
M20 (3/4)	160–320 320–640	130–190 240–400	60–80 190–290	80–190 130–240	50–80 60–160	80–190 130–320	60–130 130–240	50–80 80–160	240–400 640–950	160–240 480–640	240–400 640–950	240–400 640–950	130–190 240–400	160–320 320–480
M22 (7/8)	140–290 290–580	120–170 220–360	60–70 170–260	70–170 120–220	40–70 60–140	70–170 120–290	60–120 120–220	40–70 70–140	220–360 580–870	140–220 430–580	220–360 580–870	220–360 580–870	120–170 220–360	140–290 290–430
M24 (15/16)	130–270 270–530	110–160 200–330	50–70 160–240	70–160 110–200	40–70 50–130	70–160 110–270	50–110 110–200	40–70 70–130	200–330 530–800	130–200 400–530	200–330 530–800	200–330 530–800	110–160 200–330	130–270 270–400
M25 (1)	130–250 250–510	100–150 190–320	50–60 150–230	60–150 100–190	40–60 50–130	60–150 100–250	50–100 100–190	40–60 60–130	190–320 510–760	130–190 380–510	190–320 510–760	190–320 510–760	100–150 190–320	130–250 250–380
M26	120–240 240–490	100–150 180–310	50–60 150–220	60–150 100–180	40–60 50–120	60–150 100–240	50–100 100–180	40–60 60–120	180–310 490–730	120–180 370–490	180–310 490–730	180–310 490–730	100–150 180–310	120–240 240–370
M27 (1 1/16)	120–240 240–470	90–140 180–290	50–60 140–210	60–140 90–180	40–60 50–120	60–140 90–240	50–90 90–180	40–60 60–120	180–290 470–710	120–180 350–470	180–290 470–710	180–290 470–710	90–140 180–290	120–240 240–350
M28 (1 1/8)	110–230 230–450	90–140 170–280	50–60 140–200	60–140 90–170	30–60 50–110	60–140 90–230	50–90 90–170	30–60 60–110	170–280 450–680	110–170 340–450	170–280 450–680	170–280 450–680	90–140 170–280	110–230 230–340
M30 (1 3/16)	110–210 210–420	80–130 160–270	40–50 130–190	50–130 80–160	30–50 40–110	50–130 80–210	40–80 80–160	30–50 50–110	160–270 420–640	110–160 320–420	160–270 420–640	160–270 420–640	80–130 160–270	110–210 210–320
M32 (1 1/4)	100–200 200–400	80–120 150–250	40–50 120–180	50–120 80–150	30–50 40–100	50–120 80–200	40–80 80–150	30–50 50–100	150–250 400–600	100–150 300–400	150–250 400–600	150–250 400–600	80–120 150–250	100–200 200–300
M33 (1 5/16)	100–190 190–390	80–120 140–240	40–50 120–170	50–120 80–140	30–50 40–100	50–120 80–190	40–80 80–140	30–50 50–100	140–240 390–580	100–140 290–390	140–240 390–580	140–240 390–580	80–120 140–240	100–190 190–290
M36 (1 7/16)	90–180 180–350	70–110 130–220	40–40 110–160	40–110 70–130	30–40 40–90	40–110 70–180	40–70 70–130	30–40 40–90	130–220 350–530	90–130 270–350	130–220 350–530	130–220 350–530	70–110 130–220	90–180 180–270
M39 (1 5/8)	80–160 160–330	70–100 120–200	30–40 100–150	40–100 70–120	20–40 30–80	40–100 70–160	30–70 70–120	20–40 40–80	120–200 330–490	80–120 240–330	120–200 330–490	120–200 330–490	70–100 120–200	80–160 160–240
M40	80–160 160–320	60–100 120–200	30–40 100–140	40–100 60–120	20–40 30–80	40–100 60–160	30–60 60–120	20–40 40–80	120–200 320–480	80–120 240–320	120–200 320–480	120–200 320–480	60–100 120–200	80–160 160–240
M42 (1 3/4)	80–150 150–300	60–90 110–190	30–40 90–140	40–90 60–110	20–40 30–80	40–90 60–150	30–60 60–110	20–40 40–80	110–190 300–450	80–110 230–300	110–190 300–450	110–190 300–450	60–90 110–190	80–150 150–230

Tapping speed v_c = m/min
RPM n = U/min
Diameter d = mm

Calculation: $n = \frac{v_c \times 1000}{d \times}$

Example: M8 with 25 m/min $n = \frac{25 \times 1000}{8 \times 3.14} = 995$ m/min

- The speed given are guide values acc. to general information and can vary depending on tap manufacturer.
- For optimal production and correct speed, we suggest following the specifications of the tap manufacturer.
- Furthermore, the maximum speed of the tapping attachment must not be exceeded.

RDT15	RDT25 RDTIC25	RDT50 RDTIC50 RDTICXT50 RDTXT50 RCT50 sintered gears	RDT50 RDTIC50 RDTICXT50 RDTXT50 RCT50 cut gears	RDT85HS RDTIC85HS	RDT85HD RDTIC85HD	RDT100 RDTIC100	RDT150

(Continued)

Speed Chart/High Speed/Top Speed Taps

	Low Carbon Steel, Medium Carbon Steel	High Carbon Steel, High Strength Steel, Tool Steel	High Strength Steel, Tool Steel Hardened	Stainless 303, 304, 316	Stainless 410, 430, 17-4 Hardened	Stainless 17-4 Annealed	Titanium Alloys	Nickel Base Alloys	Aluminium Alloys	Aluminium Die Cast	Magnesium	Brass, Bronze	Copper	Cast Iron
A) m/min B) m/min	50–70 —	20–30 40–60	15–20 30–50	10–15 —	6–10 —	10–15 —	12–15 —	6–12 —	50–70 60–80	40–50 50–70	50–70 40–80	30–70 60–80	20–30 30–50	25–40 30–50
Tap size	A) rpm for HSS-E uncoated taps B) rpm for HSS-E coated taps													
M2 (#2)	7950–11150 640–800	3200–4750 6350–9550	2400–3200 4750–7950	1600–2400 640–800	800–1600 640–800	1600–2400 640–800	1900–2400 –	800–1900 –	7950–11150 9550–12750	6350–7950 7950–11150	7950–11150 6350–12750	4750–11150 9550–12750	3200–4750 4750–7950	4000–6350 4750–7950
M3 (#5)	5300–7450 420–530	2100–3200 4250–6350	1600–2100 3200–5300	1050–1600 420–530	530–1050 420–530	1050–1600 420–530	1250–1600 –	530–1250 –	5300–7450 6350–8500	4250–5300 5300–7450	5300–7450 4250–8500	3200–7450 6350–8500	2100–3200 3200–5300	2650–4250 3200–5300
M4 (#8)	4000–5550 320–600	1600–2400 3200–4750	1200–1600 2400–4000	800–1200 320–400	400–800 320–400	800–1200 320–400	950–1200 –	400–950 –	4000–5550 4750–6350	3200–4000 4000–5550	4000–5550 3200–6350	2400–5550 4750–6350	1600–2400 2400–4000	2000–3200 2400–4000
M5 (#10)	3200–4450 250–320	1250–1900 2550–3800	950–1250 1900–3200	640–950 250–320	320–640 250–320	640–950 250–320	760–950 –	320–760 –	3200–4450 3800–5100	2550–3200 3200–4450	3200–4450 2550–5100	1900–4450 3800–5100	1250–1900 1900–3200	1600–2550 1900–3200
M6 (1/4)	2650–3700 210–270	1050–1600 2100–3200	800–1050 1600–2650	530–800 210–270	270–530 210–270	530–800 210–270	640–800 –	270–640 –	2650–3700 3200–4250	2100–2650 2650–3700	2650–3700 2100–4250	1600–3700 3200–4250	1050–1600 1600–2650	1350–2100 1600–2650
M7	2250–3200 180–230	910–1350 1800–2750	680–910 1350–2250	450–680 180–230	230–450 180–230	450–680 180–230	550–680 –	230–550 –	2250–3200 2750–3650	1800–2250 2250–3200	2250–3200 1800–3650	1350–3200 2750–3650	910–1350 1350–2250	1150–1800 1350–2250
M8 (5/16)	2000–2800 160–200	800–1200 1600–2400	600–800 1200–2000	400–600 160–200	200–400 160–200	400–600 160–200	480–600 –	200–480 –	2000–2800 2400–3200	1600–2000 2000–2800	2000–2800 1600–3200	1200–2800 2400–3200	800–1200 1200–2000	990–1600 1200–2000
M9	1750–2500 140–180	710–1050 1400–2100	530–710 1050–1750	350–530 140–180	180–350 140–180	350–530 140–180	420–530 –	180–420 –	1750–2500 2100–2850	1400–1750 1750–2500	1750–2500 1400–2850	1050–2500 2100–2850	710–1050 1050–1750	880–1400 1050–1750
M10 (3/8)	1600–2250 130–160	640–950 1250–1900	480–640 950–1600	320–480 130–160	160–320 130–160	320–480 130–160	380–480 –	160–380 –	1600–2250 1900–2550	1250–1600 1600–2250	1600–2250 1250–2550	950–2250 1900–2550	640–950 950–1600	800–1250 950–1600
M12 (1/2)	1350–1850 110–130	530–800 1050–1600	400–530 800–1350	270–400 110–130	130–270 110–130	270–400 110–130	320–400 –	130–320 –	1350–1850 1600–2100	1050–1350 1350–1850	1350–1850 1050–2100	800–1850 1600–2100	530–800 800–1350	660–1050 800–1350
M14 (9/16)	1150–1600 90–110	450–680 910–1350	340–450 680–1150	230–340 90–110	110–230 90–110	230–340 90–110	270–340 –	110–270 –	1150–1600 1350–1800	910–1150 1150–1600	1150–1600 910–1800	680–1600 1350–1800	450–680 680–1150	570–910 680–1150
M16 (5/8)	990–1400 80–100	400–600 800–1200	300–400 600–990	200–300 80–100	100–200 80–100	200–300 80–100	240–300 –	100–240 –	990–1400 1200–1600	800–990 990–1400	990–1400 800–1600	600–1400 1200–1600	400–600 600–990	500–800 600–990
M18 (11/16)	880–1250 70–90	350–530 710–1050	270–350 530–880	180–270 70–90	90–180 70–90	180–270 70–90	210–270 –	90–210 –	880–1250 1050–1400	710–880 880–1250	880–1250 710–1400	530–1250 1050–1400	350–530 530–880	440–710 530–880
M20 (3/4)	800–1100 60–80	320–480 640–950	240–320 480–800	160–240 60–80	80–160 60–80	160–240 60–80	190–240 –	80–190 –	800–1100 950–1250	640–800 800–1100	800–1100 640–1250	480–1100 950–1250	320–480 480–800	400–640 480–800
M22 (7/8)	720–1000 60–70	290–430 580–870	220–290 430–720	140–220 60–70	70–140 60–70	140–220 60–70	170–220 –	70–170 –	720–1000 870–1150	580–720 720–1000	720–1000 580–1150	430–1000 870–1150	290–430 430–720	360–580 430–720
M24 (15/16)	660–930 50–70	270–400 530–800	200–270 400–660	130–200 50–70	70–130 50–70	130–200 50–70	160–200 –	70–160 –	660–930 800–1050	530–660 660–930	660–930 530–1050	400–930 800–1050	270–400 400–660	330–530 400–660
M25 (1)	640–890 50–60	250–380 510–760	190–250 380–640	130–190 50–60	60–130 50–60	130–190 50–60	150–190 –	60–150 –	640–890 760–1000	510–640 640–890	640–890 510–1000	380–890 760–1000	250–380 380–640	320–510 380–640
M26	610–860 50–60	240–370 490–730	180–240 370–610	120–180 50–60	60–120 50–60	120–180 50–60	150–180 –	60–150 –	610–860 730–980	490–610 610–860	610–860 490–980	370–860 730–980	240–370 370–610	310–490 370–610
M27 (1 1/16)	590–830 50–60	240–350 470–710	180–240 350–590	120–180 50–60	60–120 50–60	120–180 50–60	140–180 –	60–140 –	590–830 710–940	470–590 590–830	590–830 470–940	350–830 710–940	240–350 350–590	290–470 350–590
M28 (1 1/8)	570–800 50–60	230–340 450–680	170–230 340–570	110–170 50–60	60–110 50–60	110–170 50–60	140–170 –	60–140 –	570–800 680–910	450–570 570–800	570–800 450–910	340–800 680–910	230–340 340–570	280–450 340–570
M30 (1 3/16)	530–740 40–50	210–320 420–640	160–210 320–530	110–160 40–50	50–110 40–50	110–160 40–50	130–160 –	50–130 –	530–740 640–850	420–530 530–740	530–740 420–850	320–740 640–850	210–320 320–530	270–420 320–530
M32 (1 1/4)	500–700 40–50	200–300 400–600	150–200 300–500	100–150 40–50	50–100 40–50	100–150 40–50	120–150 –	50–120 –	500–700 600–800	400–500 500–700	500–700 400–800	300–700 600–800	200–300 300–500	250–400 300–500
M33 (1 5/16)	480–680 40–50	190–290 390–580	140–190 290–480	100–140 40–50	50–100 40–50	100–140 40–50	120–140 –	50–120 –	480–680 580–770	390–480 480–680	480–680 390–770	290–680 580–770	190–290 290–480	240–390 290–480
M36 (1 7/16)	440–620 40–40	180–270 350–530	130–180 270–440	90–130 40–40	40–90 40–40	90–130 40–40	110–130 –	40–110 –	440–620 530–710	350–440 440–620	440–620 350–710	270–620 530–710	180–270 270–440	220–350 270–440
M39 (1 5/8)	410–570 30–40	160–240 330–490	120–160 240–410	80–120 30–40	40–80 30–40	80–120 30–40	100–120 –	40–100 –	410–570 490–650	330–410 410–570	410–570 330–650	240–570 490–650	160–240 240–410	200–330 240–410
M40	400–560 30–40	160–240 320–480	120–160 240–400	80–120 30–40	40–80 30–40	80–120 30–40	100–120 –	40–100 –	400–560 480–640	320–400 400–560	400–560 320–640	240–560 480–640	160–240 240–400	200–320 240–400
M42 (1 3/4)	380–530 30–40	150–230 300–450	110–150 230–380	80–110 30–40	40–80 30–40	80–110 30–40	90–110 –	40–90 –	380–530 450–610	300–380 380–530	380–530 300–610	230–530 450–610	150–230 230–380	190–300 230–380

Tapping speed v_c = m/min
RPM n = U/min
Diameter d = mm

Calculation:

$$n = \frac{v_c \times 1000}{d \times}$$

Example: M8 with 40 m/min

$$n = \frac{40 \times 1000}{8 \times 3.14} = 1592\text{m/min}$$

- The speed given are guide values acc. to general information and can vary depending on tap manufacturer.
- For optimal production and correct speed, we suggest following the specifications of the tap manufacturer.
- Furthermore, the maximum speed of the tapping attachment must not be exceeded.

RDT15	RDT25 RDTIC25	RDT50 RDTIC50 RDTICXT50 RDTXT50 RCT50 sintered gears	RDT50 RDTIC50 RDTICXT50 RDTXT50 RCT50 cut gears	RDT85HS RDTIC85HS	RDT85HD RDTIC85HD	RDT100 RDTIC100	RDT150

(Continued)

Speed Chart/Roll From Taps

	Low Carbon Steel, Medium Carbon Steel	High Carbon Steel, High Strength Steel, Tool Steel	High Strength Steel Tool Steel Hardened	Stainless 303, 304, 316	Stainless 17-4 Annealed	Titanium Alloys	Nickel Base Alloys	Aluminium Alloys	Aluminium Die Cast	Copper
A) m/min B) m/min	30–40 40–60	20–30 30–50	15–25 25–40	10–15 12–20	10–15 —	5–15 —	3–5 8–12	30–60 50–70	20–40 30–50	15–25 25–50
Tap size	A) rpm for HSS-E uncoated taps B) rpm for HSS-E coated taps									
M2 (#2)	4750–6350 6350–9550	3200–4750 4750–7950	2400–4000 4000–6350	1600–2400 1900–3200	1600–2400 –	800–2400 320–1600	480–800 1250–1900	4750–9550 7950–11150	3200–6350 4750–7950	2400–4000 4000–7950
M3 (#5)	3200–4250 4250–6350	2100–3200 3200–5300	1600–2650 2650–4250	1050–1600 1250–2100	1050–1600 –	530–1600 210–1050	320–530 850–1250	3200–6350 5300–7450	2100–4250 3200–5300	1600–2650 2650–5300
M4 (#8)	2400–3200 3200–4750	1600–2400 2400–4000	1200–2000 2000–3200	800–1200 950–1600	800–1200 –	400–1200 160–800	240–400 640–950	2400–4750 4000–5550	1600–3200 2400–4000	1200–2000 2000–4000
M5 (#10)	1900–2550 2550–3800	1250–1900 1900–3200	950–1600 1600–2550	640–950 760–1250	640–950 –	320–950 130–640	190–320 510–760	1900–3800 3200–4450	1250–2550 1900–3200	950–1600 1600–3200
M6 (1/4)	1600–2100 2100–3200	1050–1600 1600–2650	800–1350 1350–2100	530–800 640–1050	530–800 –	270–800 110–530	160–270 420–640	1600–3200 2650–3700	1050–2100 1600–2650	800–1350 1350–2650
M7	1350–1800 1800–2750	910–1350 1350–2250	680–1150 1150–1800	450–680 550–910	450–680 –	230–680 90–450	140–230 360–550	1350–2750 2250–3200	910–1800 1350–2250	650–1150 1150–2250
M8 (5/16)	1200–1600 1600–2400	800–1200 1200–2000	600–990 990–1600	400–600 480–800	400–600 –	200–600 80–400	120–200 320–480	1200–2400 2000–2800	800–1600 1200–2000	600–990 990–2000
M9	1050–1400 1400–2100	710–1050 1050–1750	530–880 880–1400	350–530 420–710	350–530 –	180–530 70–350	110–180 280–420	1050–2100 1750–2500	710–1400 1050–1750	530–880 880–1750
M10 (3/8)	950–1250 1250–1900	640–950 950–1600	480–800 800–1250	320–480 380–640	320–480 –	160–480 60–320	100–160 250–380	950–1900 1600–2250	640–1250 950–1600	480–800 800–1600
M12 (1/2)	800–1050 1050–1600	530–800 800–1350	400–660 660–1050	270–400 320–530	270–400 –	130–400 50–270	80–130 210–320	800–1600 1350–1850	530–1050 800–1350	400–660 660–1350
M14 (9/16)	680–910 910–1350	450–680 680–1150	340–570 570–910	230–340 270–450	230–340 –	110–340 50–230	70–110 180–270	680–1350 1150–1600	450–910 680–1150	340–570 570–1150
M16 (5/8)	600–800 800–1200	400–600 600–990	300–500 500–800	200–300 240–400	200–300 –	100–300 40–200	60–100 160–240	600–1200 990–1400	400–800 600–990	300–500 500–990
M18 (11/16)	530–710 710–1050	350–530 530–880	270–440 440–710	180–270 210–350	180–270 –	90–270 40–180	50–90 140–210	530–1050 880–1250	350–710 530–880	270–440 440–880
M20 (3/4)	480–640 640–950	320–480 480–800	240–400 400–640	160–240 190–320	160–240 –	80–240 30–160	50–80 130–190	480–950 800–1100	320–640 480–800	240–400 400–800
M22 (7/8)	430–580 580–870	290–430 430–720	220–360 360–580	140–220 170–290	140–220 –	70–220 30–140	40–70 120–170	430–870 720–1000	290–580 430–720	220–360 360–720
M24 (15/16)	400–530 530–800	270–400 400–660	200–330 330–530	130–200 160–270	130–200 –	70–200 30–130	40–70 110–160	400–800 660–930	270–530 400–660	200–330 330–660
M25 (1)	380–510 510–760	250–380 380–640	190–320 320–510	130–190 150–250	130–190 –	60–190 30–130	40–60 100–150	380–760 640–890	250–510 380–640	190–320 320–640
M26	370–490 490–730	240–370 370–610	180–310 310–490	120–180 150–240	120–180 –	60–180 20–120	40–60 100–150	370–730 610–860	240–490 370–610	180–310 310–610
M27 (1 1/16)	350–470 470–710	240–350 350–590	180–290 290–470	120–180 140–240	120–180 –	60–180 20–120	40–60 90–140	350–710 590–830	240–470 350–590	180–290 290–590
M28 (1 1/8)	340–450 450–680	230–340 340–570	170–280 280–450	110–170 140–230	110–170 –	60–170 20–110	30–60 90–140	340–680 570–800	230–450 340–570	170–280 280–570
M30 (1 3/16)	320–420 420–640	210–320 320–530	160–270 270–420	110–160 130–210	110–160 –	50–160 20–110	30–50 80–130	320–640 530–740	210–420 320–530	160–270 270–530
M32 (1 1/4)	300–400 400–600	200–300 300–500	150–250 250–400	100–150 120–200	100–150 –	50–150 20–100	30–50 80–120	300–600 500–700	200–400 300–500	150–250 250–500
M33 (1 5/16)	290–390 390–580	190–290 290–480	140–240 240–390	100–140 120–190	100–140 –	50–140 20–100	30–50 80–120	290–580 480–680	190–390 290–480	140–240 240–480

Tapping speed v_c = m/min
RPM n = U/min
Diameter d = mm

Calculation:

$$n = \frac{v_c \times 1000}{d \times}$$

Example: M8 with 30 m/min

$$n = \frac{30 \times 1000}{8 \times 3.14} = 1194 \text{m/min}$$

- The speed given are guide values acc. to general information and can vary depending on tap manufacturer.
- For optimal production and correct speed, we suggest following the specifications of the tap manufacturer.
- Furthermore, the maximum speed of the tapping attachment must not be exceeded.

RDT15	RDT25 RDTIC25	RDT50 RDTIC50 RDTICXT50 RDTXT50 RCT50 sintered gears	RDT50 RDTIC50 RDTICXT50 RDTXT50 RCT50 cut gears	RDT85HS RDTIC85HS	RDT85HD RDTIC85HD	RDT100 RDTIC100	RDT150

TAPMATIC

Drill Selection: Inches / Metric

For Inch/Metric Taps & Decimal Equivalents

Note: Most drill size charts are based on using standard job drills which can drill over size by approximately .003. These charts are based on .003 over size condition to achieve the proper percentages of thread. With today's high precision drills, they are now capable of drilling to near net size. When using a high precision drill or a "G" drill you should refer to the drill size formula's in the "Tapping Formulas" section.

Fraction Or Drill Size	Decimal Equivalent	Tap Size	Fraction Or Drill Size	Decimal Equivalent	Tap Size	Fraction Or Drill Size	Decimal Equivalent	Tap Size
NUMBER SIZE DRILLS 80	.0135		39	.0995		15/64	.2344	
79	.0145		38	.1015	5-40	LETTER SIZE DRILLS B	.2380	
1/64	.0156		37	.1040	5-44	C	.2420	
78	.0160		36	.1065	6-32	D	.2460	
77	.0180		7/64	.1094		1/4 E	.2500	
76	.0200		35	.1100		F	.2570	5/16 - 18
75	.0210		34	.1110		G	.2610	
74	.0225		33	.1130	6-40	17/64	.2656	
73	.0240		32	.1160		H	.2660	
72	.0250		31	.1200		I	.2720	5/16 - 24
71	.0260		1/8	.1250		J	.2770	
70	.0280		30	.1285		K	.2810	
69	.0292		29	.1360	8-32, 36	9/32	.2812	
68	.0310		28	.1405		L	.2900	
1/32	.0312		9/64	.1406		M	.2950	
67	.0320		27	.1440		19/64	.2969	
66	.0330		26	.1470		N	.3020	
65	.0350		25	.1495	10-24	5/16	.3125	3/8 - 16
64	.0360		24	.1520		O	.3160	
63	.0370		23	.1540		P	.3230	
62	.0380		5/32	.1562		21/64	.3281	
61	.0390		22	.1570		Q	.3320	3/8 - 24
60	.0400		21	.1590	10-32	R	.3390	
59	.0410		20	.1610		11/32	.3438	
58	.0420		19	.1660		S	.3480	
57	.0430		18	.1695		T	.3580	
56	.0465		11/64	.1719		23/64	.3594	
3/64	.0469	0 - 80	17	.1730		U	.3680	7/16 - 14
55	.0520		16	.1770	12-24	3/8	.3750	
54	.0550		15	.1800		V	.3770	
53	.0595	1 - 64,72	14	.1820	12-28	W	.3860	
1/16	.0625		13	.1850		25/64	.3906	7/16 - 20
52	.0635		3/16	.1875		X	.3970	
51	.0670		12	.1890		Y	.4040	
50	.0700	2-56, 64	11	.1910		13/32	.4062	
49	.0730		10	.1935		Z	.4130	
48	.0760		9	.1960		27/64	.4219	1/2 - 13
5/64	.0781		8	.1990		7/16	.4375	
47	.0785	3 - 48	7	.2010	1/4-20	29/64	.4531	1/2 - 20
46	.0810		13/64	.2031		15/32	.4688	
45	.0820	3 - 56	6	.2040		31/64	.4844	9/16 - 12
44	.0860		5	.2055		1/2	.5000	
43	.0890	4 - 40	4	.2090		33/64	.5156	9/16 - 18
42	.0935	4 - 48	3	.2130	1/4-28	17/32	.5312	5/8 - 11
3/32	.0938		7/32	.2188		35/64	.5469	
41	.0960		2	.2210		9/16	.5625	
40	.0980		LETTER SIZE DRILLS 1	.2280		37/64	.5781	5/8 - 18
			A	.2340				

Fraction Or Drill Size	Decimal Equivalent	Tap Size
19/32	.5938	
39/64	.6094	
5/8	.6250	
41/64	.6406	
21/32	.6562	3/4 - 10
43/64	.6719	
11/16	.6875	3/4 - 16
45/64	.7031	
23/32	.7188	
47/64	.7344	
3/4	.7500	
49/64	.7656	7/8 - 9
25/32	.7812	
51/64	.7969	
13/16	.8125	7/8 - 14
53/64	.8281	
27/32	.8438	
55/64	.8594	
7/8	.8750	1 - 8
57/64	.8906	
29/32	.9062	
59/64	.9219	1 - 12
15/16	.9375	
61/64	.9531	
31/32	.9688	
63/64	.9844	1 1/8 - 7
1	1.0000	
1 3/64	1.0469	1 1/8 - 12
1 7/64	1.1094	1 1/4 - 7
1 1/8	1.1250	
1 11/64	1.1719	1 1/4 - 12
1 7/32	1.2188	1 3/8 - 6
1 1/4	1.2500	
1 19/64	1.2969	1 3/8 - 12
1 11/32	1.3438	1 1/2 - 6
1 3/8	1.3750	
1 27/64	1.4219	1 1/2 - 12
1 1/2	1.5000	

PIPE THREAD SIZES (NPSC)

THREAD	DRILL	THREAD	DRILL
1/8 - 27	11/32	1 1/2 - 11 1/2	1 3/4
1/4 - 18	7/16	2 - 11 1/2	2 7/32
3/8 - 18	37/64	2 1/2 - 8	2 21/32
1/2 - 14	23/32	3 - 8	3 1/4
3/4 - 14	59/64	3 1/2 - 8	3 3/4
1 - 11 1/2	1 5/32	4 - 8	4 1/4
1 1/4 - 11 1/2	1 1/2		

Metric Tap	Tap Drill mm	Decimal Equivalent
M1.6 x 0.35	1.25	.0492
M1.8 x 0.35	1.45	.0571
M2 x 0.4	1.60	.0630
M2.2 x 0.45	1.75	.0689
M2.5 x 0.45	2.05	.0807
M3 x 0.5	2.50	.0984
M3.5 x 0.6	2.90	.1142
M4 x 0.7	3.30	.1299
M4.5 x 0.75	3.70	.1457
M5 x 0.8	4.20	.1654
M6 x 1	5.00	.1968
M7 x 1	6.00	.2362
M8 x 1.25	6.70	.2638
M8 x 1	7.00	.2756
M10 x 1.5	8.50	.3346
M10 x 1.25	8.70	.3425
M12 x 1.75	10.20	.4016
M12 x 1.25	10.80	.4252
M14 x 2	12.00	.4724
M14 x 1.5	12.50	.4921
M16 x 2	14.00	.5512
M16 x 1.5	14.50	.5709
M18 x 2.5	15.50	.6102
M18 x 1.5	16.50	.6496
M20 x 2.5	17.50	.6890
M20 x 1.5	18.50	.7283
M22 x 2.5	19.50	.7677
M22 x 1.5	20.50	.8071
M24 x 3	21.00	.8268
M24 x 2	22.00	.8661
M27 x 3	24.00	.9449
M27 x 2	25.00	.9843
M30 x 3.5	26.50	1.0433
M30 x 2	28.00	1.1024
M33 x 3.5	29.50	1.1614
M33 x 2	31.00	1.2205
M36 x 4	32.00	1.2598
M36 x 3	33.00	1.2992
M39 x 4	35.00	1.3780
M39 x 3	36.00	1.4173

Drill Size Factors

- Tapped holes deeper than 1.5 diameters often call for a larger tap drill.
- Blind holes often require larger tap drills to reduce loads on the tap caused by chip buildup in the hole.
- Materials that tend to gall when tapped or when fasteners are installed should have larger drilled holes. Under tapping pressure, soft materials tend to extrude and enter the root area, necessiating a larger drilled hole.
- Materials that don't readily dissipate heat, should have larger holes to reduce the tooth contact area and minimize heat build up.
- When making threads with high helix angles using a larger tap drill will help reduce tap breakage.

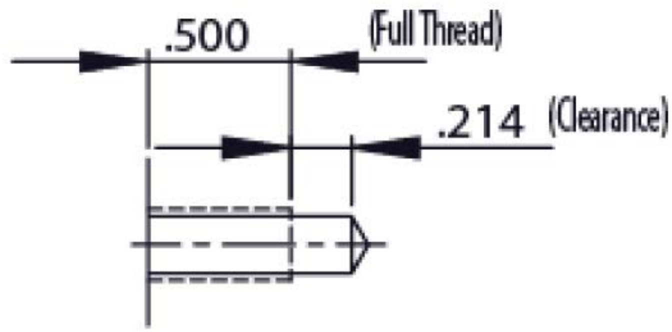
Drill Depth Clearance in Blind Holes

Chamfer Teeth + One Pitch + 1mm = Clearance

- Bottom Tap has 1 to 2 teeth in chamfer or lead.
- Semi Bottom Tap has 2 to 3 teeth in lead.
- Modified Bottom has 2 to 4 teeth in lead.
- Plug Tap has 3 to 5 teeth in lead.
- Modified Plug has 5 to 7.
- Roll Form Tap has typically 2 1/2.

Example:

1/4-20 Roll Form tap 1/2”deep.



Chamfer Teeth = 2.5 X pitch (.050) = .125

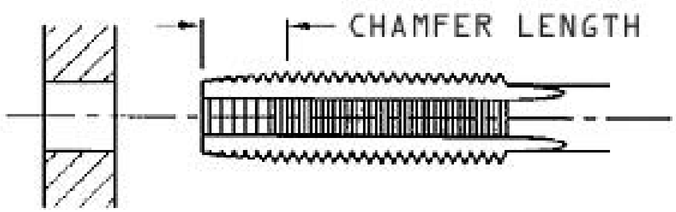
Chamfer Teeth + one pitch + 1mm = Clearance

.125 + .050 + .039 = .214

Tap Recommendations For Specific Materials

Tap Manufacturers offer their own unique geometries for specific materials and applications. This chart is meant to provide general information. For a specific tap recommendation for your application, please consult your tap supplier.

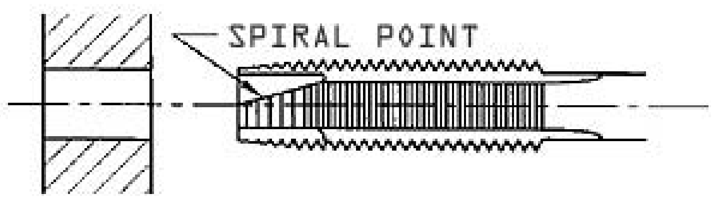
Standard Straight Fluted Tap
With 6 to 8 Threads Chamfer Length or Lead.



These taps do not transport the chips out of the hole. For this reason, they should not be used for deep hole tapping. They work best in shallow depth through holes and in materials that produce short chips.

Workpiece Materials	Recommended Tap Surface Treatments
Cast Iron	Nitrided or TiN
Brass, Short Chipping	Nitrided
Cast Aluminum	Nitrided
Short Chip Hard	Nitrided or TiN

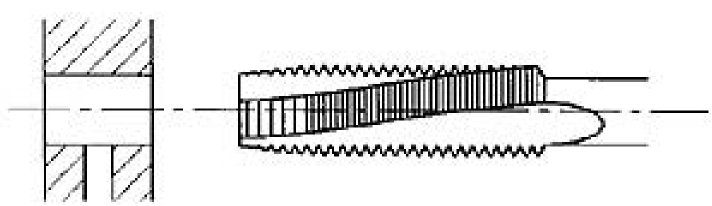
Straight Fluted Taps With Spiral Point
With 3.5 to 5 Threads Chamfer Length or Lead.



These taps push the chips forward. The chips are curled up to prevent clogging in the flutes. They are used for through holes.

Workpiece Materials	Recommended Tap Surface Treatments
Aluminum Long Chip	Bright, or Cr or TiN
Exotic Alloys	Nitrided or TiN
Stainless Steel	Nitrided or TiN
Steel	Bright or TiN or TiCN

Left Hand Spiral Fluted Tap
With Approximately 12 Degrees Spiral Flutes
With 3.5 to 5 Threads Chamfer Length.



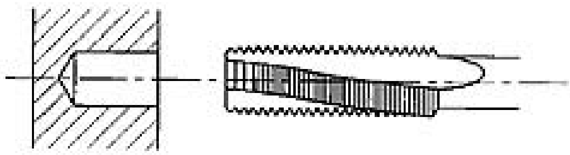
These taps are mostly used in thin walled parts or for holes interrupted by cross holes or longitudinal slots.

Workpiece Materials	Recommended Tap Surface Treatments
Aluminum Long Chip	Bright, or Cr or TiN
Exotic Alloys	Nitrided or TiN
Stainless Steel	Nitrided or TiN
Steel	Bright or TiN or TiCN

(Continued)

Tap Recommendations for Specific Materials (Continued)

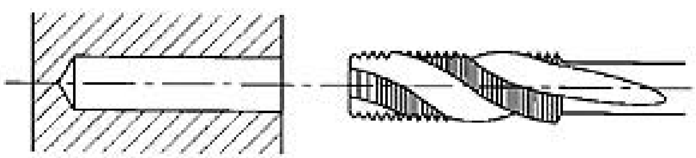
Right Hand Spiral Fluted Tap
With Approximately 15 Degrees Spiral Flutes
With 3.5 to 5 Threads Chamfer Length.



The spiral flutes transport chips back out of the hole. These taps are used in blind holes less than 1.5 times the tap diameter deep with materials that produce short chips.

Workpiece Materials	Recommended Tap Surface Treatments
Cast Aluminum	Nitrided
Titanium	Nitrided or TiN
Stainless Steel	Bright or TiN
Steel	Bright or TiN or TiCN

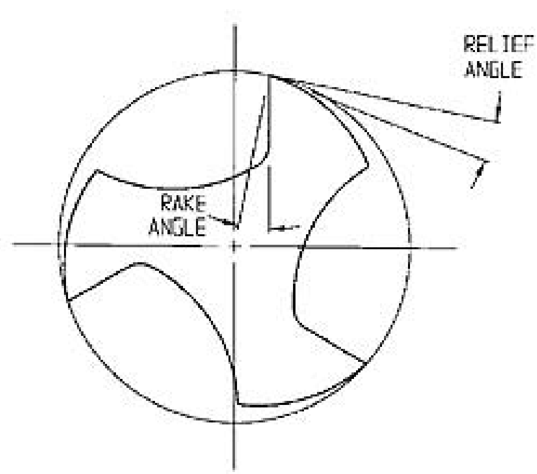
Right Hand Spiral Fluted Tap
With 40 Degrees to 50 Degrees Spiral Flutes.



The greater helix angle provides good transport of chips back out of the hole. These taps are used only in blind holes in materials that produce long chips. They can also be used in deeper holes up to 3 times the tap diameter.

Workpiece Materials	Recommended Tap Surface Treatments
Aluminum Long Chip	Bright, or Cr or TiN
Stainless Steel	Bright or TiN
Steel Alloy Cr-Ni	Bright or TiN or TiCN
Soft Material	Bright

Rake Angle
The best rake angle for a tap depends on the material. Materials that produce long chips normally require a tap with greater rake angle. Materials that produce short chips require a smaller rake angle. Difficult materials like Titanium or Inconnell require a compromise between greater rake angle for longer chips and smaller rake angle for more strength.



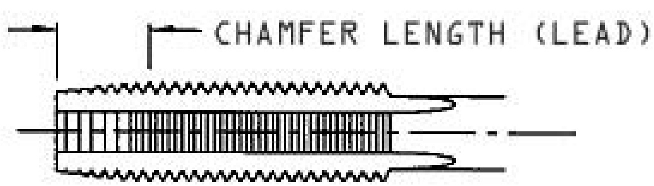
(Continued)

Tap Recommendations for Specific Materials (Continued)

Relief Angle In The Lead Of A Tap
A small relief angle can be used in soft materials. Harder materials like stainless steel can be cut easier with a tap having a greater relief angle which reduces the friction. Tough materials like Inconnel and nickel can be cut more easily with an even greater angle.

The relief angle is smaller on taps for blind holes that on taps for through holes so that the chip root can be sheared off when the tap reverses without breaking the taps cutting edge.

Chamfer Length (Lead)

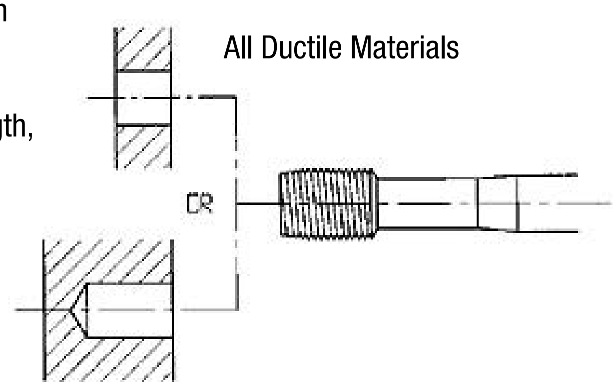


The actual cutting of the thread is done by the lead of the tap. When there are more threads in the chamfer length or lead the torque is reduced, producing the thread is much easier, and the life of the tap will be increased. In blind holes where there is not enough room to drill deep enough for a tap with a longer lead, taps with short leads are used. In some cases the lead of the tap is reduced to as little as 1.5 threads. This greatly increases torque and reduces tap life. Even when using taps with shortened lead it is still important to drill deep enough for adequate clearance. It is recommended to allow one thread length plus one mm beyond the lead of the tap as drill clearance.

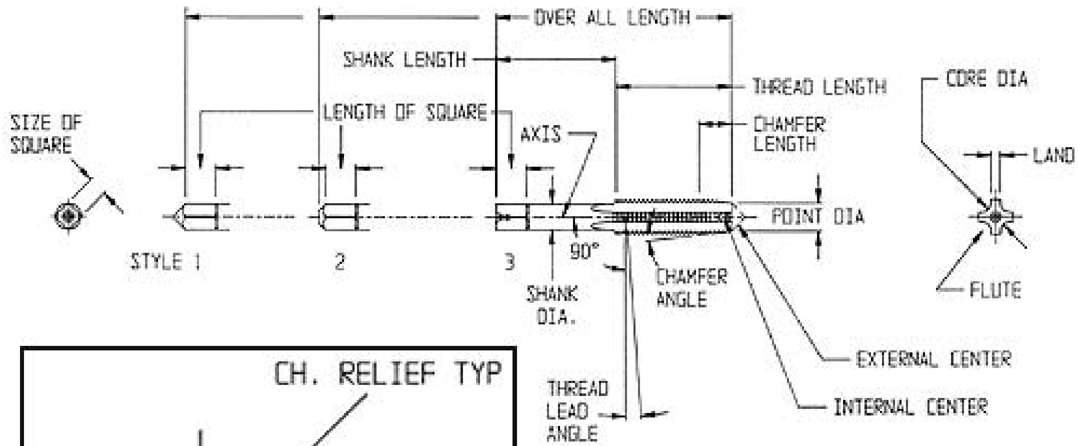
Relief Angle In The Thread Profile (Pitch Diameter Relief)
The relief angle effects true to gage thread cutting, and also the free cutting ability and life of the tap. It has an effect on how the tap is guided when it enters the hole. If the relief angle is too great pitch guidance and self centering of the tap can not be guaranteed especially in soft materials. In materials like stainless steel or bronze
the relief angle should be larger to allow free cutting and to allow more lubrication to reach the cutting and friction surfaces. A bigger relief angle can allow higher tapping speed provided the tap is guided concentrically into the hole by the machine and tap holder.

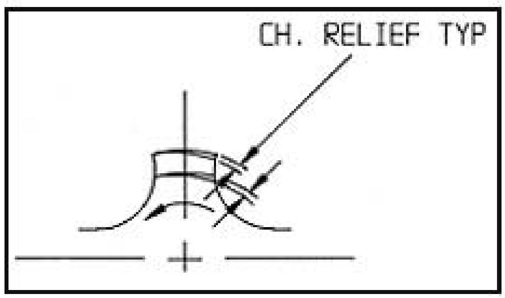
Roll Form Taps
These taps form the thread rather than cut. Since no chips are produced they can be used in blind or through holes. Cold forming is possible in all ductile materials. Advantages include no waste in the form of chips, no mis-cutting of threads, no pitch deviation, higher strength, longer tool life, and higher speed. Please note that the core hole diameter must be larger than with a cutting tap. Good lubrication is important , more torque is required, and the minor diameter of the thread will appear rough due to forming process.

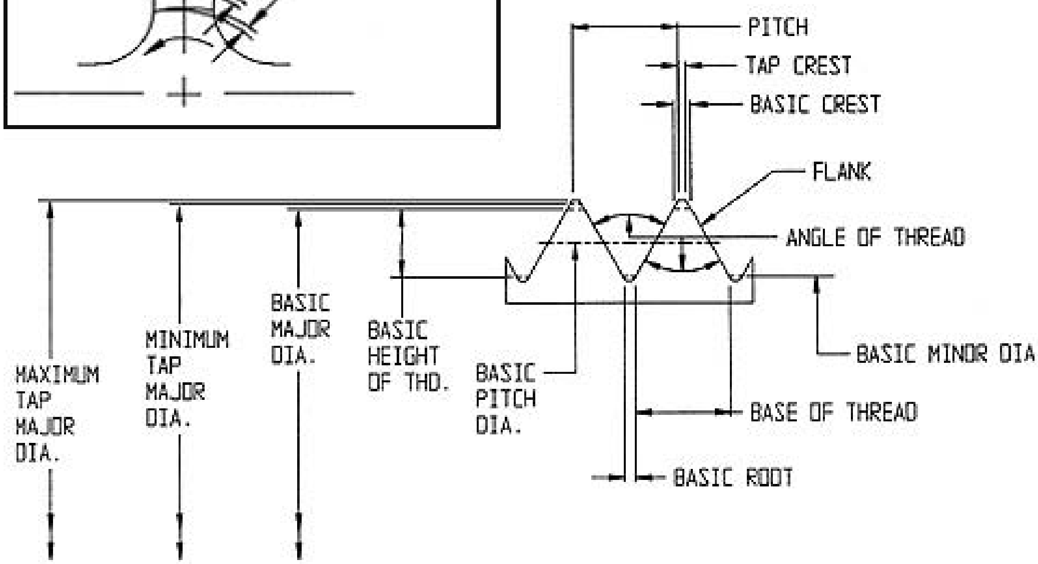
Workpiece Materials	**Recommended Tap Surface Treatments**
All Ductile Materials	

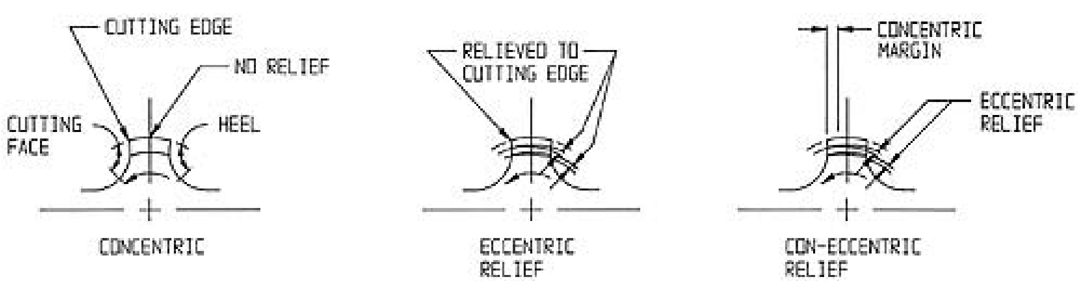


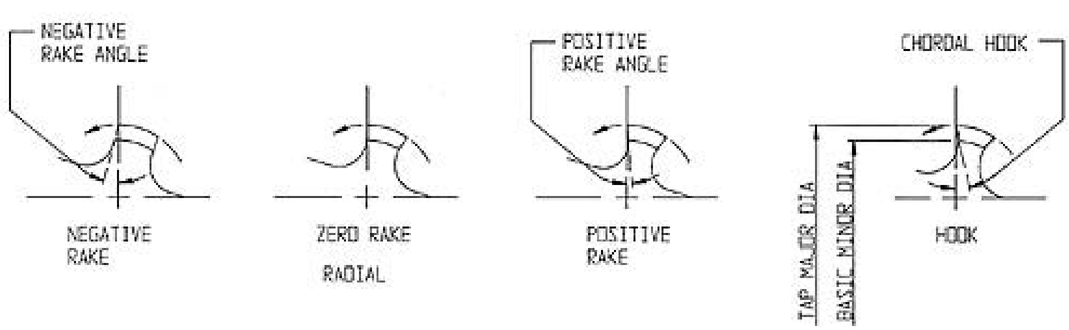
Terms for Tap Features











Common Thread Terms

Allowance: The minimum clearance or maximum interference which is intended between mating parts.

Angle of thread: The angle included between the flanks of a thread measured in an axial plane.

Back of taper: A slight taper on threaded portion of the tap making the pitch diameter near the shank smaller than that at the chamfer.

Basic: The theoretical or nominal standard size from which all variations are made.

Chamfer: The tapered and relieved cutting teeth at the front end of the threaded section. Common types of chamfer are: Taper, 8 to 10 threads long; Plug, 3 to 5 threads and Bottoming, 1.5 threads.

Crest: The top surface joining the two sides or flanks of a thread.

Cutting face: The leading side of the land.

Flute: The longitudinal channels formed on a tap to create cutting edges on the thread profile.

Heel: The following side of the land.

Height of thread: In profile, distance between crest and bottom section of thread measured normal to the axis.

Hook face: A concave cutting face of the land. This may be varied for different materials and conditions.

Interrupted thread: Alternate teeth are removed in the thread helix on a tap having an odd number of flutes.

Land: One of the threaded sections between the flutes of a tap. Lead of thread: The distance a screw thread advances axially in one turn.

Major diameter: The largest diameter of the screw or nut on a straight screw head.

Minor diameter: The smallest diameter of the screw or nut on a straight screw head.

Neck: The reduced diameter; on some taps, between the threaded portion and the shank.

Pitch: The distance from a point on one thread to a corresponding point on the next thread, measured parallel to the axis.

Pitch diameter: On a straight screw thread, the diameter of an imaginary cylinder where the width of the thread and the width of the space between threads is equal.

Point diameter: The diameter at the leading end of the chamfered portion.

Radial: The straight face of a land, the plane of which passes through the axis of the tap.

(Continued)

Common Thread Terms (Continued)

Rake: The angle of the cutting face of the land in relation to an axial plane intersecting the cutting face at the major diameter.

Relief: The removal of metal behind the cutting edge to provide clearance between the part being threaded and a portion of the threaded land. Also, see back taper.

Chamfer Relief: The gradual decreasing land height from cutting edge to heel on the chamfered portion of the tap land to provide radial clearance for the cutting edge.

Con-Eccentric Relief: Radial relief in the thread form starting back of a concentric margin.

Eccentric Thread Relief: Radial relief in the thread form starting at the cutting edge and continuing to the heel.

Root: The bottom surface joining the flanks of two adjacent threads.

Side of flank of thread: The surface of the thread which connects the crest to the root.

Shank: The portion of the tap by which it is held and driven.

Spiral point: An oblique cutting edge ground into the lands to provide a shear cutting action on the first few threads.

Square: The squared end of the tap shank.

Thread: The helical formed tooth of the tap which produces the thread in a tapped hole.

Thread lead angle: The angle made by the helix of the thread at the pitch diameter; with a plane perpendicular to the axis.

Threads per inch: The number of threads in one inch of length.

Thread: SINGLE: A thread which is equal to pitch. DOUBLE: A thread in which lead is equal to twice the pitch. TRIPLE: A thread in which lead is equal to triple the pitch.

Coarse vs. Fine Threads

Coarse Threads

- For most applications, course threads offer these advantages:
- Easier and faster assembly, providing a better start with less chance of cross threading.
- Nicks and burrs from handling are less liable to affect assembly.
- They are less likely to seize in temperature applications and in joints where corrosion will form.
- Less prone to strip when threaded into lower strength metals.
- More easily tapped in brittle materials and or materials that crumble easily.

Fine Threads

Fine threads may make for a superior fastener for applications with specific strength or other requirements.

- They are about 10% stronger that coarse threads due to their greater cross-section area.
- In very hard materials, fine threads are easier to tap.
- They can be adjusted more precisely because of their smaller helix angle.
- Where length of engagement is limited, they provide greater strength.
- Thinner wall thickness can be used because of their smaller thread cross section.

Machining Recommendations For Cold Forming Tags

Cold Forming Internal Threads With Taps: Internal threads can be produced by a cold forming or swaging process. The desired thread is formed in the metal under pressure and the grain fibers, as in good forging, follow the contour of the thread. These grain fibers are not cut away as in conventional tapping. The cold forming tap has neither flutes nor cutting edges and therefore, it produces no chips and cannot create a chip problem. The resulting thread has a burnished surface.

Material Recommended: Care must be taken to minimize surface damage to the hole when tapping materials which are prone to work harden. This may be accomplished by using sharp drills, correct speed and feeds. Surface damage may cause torque to increase to a point of stopping the machine or breaking the tap.

Cold forming taps have been recommended for threading ductile materials. Examples of material classes which have been tapped are:

- Low carbon steels
- Leaded steels
- Austenitic stainless steels
- Aluminum die casting alloys (low silicon)
- Wrought aluminum alloys (ductile)
- Zinc die casting alloys
- Copper and copper alloys (ductile brasses)

Cold Forming Tap Application Information

Tapping Application The Same: Except for changes in hole size, the application of cold forming taps differs in no way from conventional cutting taps.

Blind Hole Tapping Possible: Whenever possible, in blind holes, drill or core deep enough to permit the use of the plug style taps. These tools, with four threads of taper, will require less torque, will produce less burr upon entering the hole, and will give greater life.

Torque: Where the operation calls for 75% of thread or less, the torque required varies with the material from no additional torque to 50% additional torque. On most applications, therefore, conventional equipment is suitable for driving cold forming taps.

No Lead Screw Necessary: These taps work equally well when used in a standard tapping head, automatic screw machine, or lead screw tapper. It is unnecessary to have lead screw tapping equipment in order to run the cold forming tap because the tool will pick up its own lead upon entering the hole.

Standard Lubrication: In general it is best to use a good cutting oil or lubricant rather than a coolant for cold forming taps. We recommend MQL Systems Dry-Cut Cutting Fluid.

(Continued)

Machine Recommendations for Cold Forming Tags (Continued)

Spindle Speeds: For most materials, spindle speeds may be increased over those recommended for conventional cutting type taps. Generally, the tap extrudes with greater efficiency at higher RPMs but it is also possible to run the tap at lower speeds with satisfactory results.

Counter Sinking or Chamfering Helpful: Because these taps displace metal, some metal will be displaced above the mouth of the hole during tapping. For this reason it is best to countersink or chamfer the hole prior to tapping, so that the extrusion will raise within the countersink and not interfere with the mating part.

Tapping Cored Holes Possible: Cored holes may be tapped with these taps provided the core pins are first changed to form the proper hole size. Because core pins have a draft or are slightly tapered the theoretical hole size should be at a point on the pin that is one-half the required length of engagement of the thread to be formed. In designing core pins for use with these taps, a chamfer should be included on the pin to accept the vertical extrusion.

Drill Selector Chart: The chart shown previously is based upon a formula derived from research statistical data and is designed to reflect the flow characteristics of all ductile materials. Laboratory experiment proved that there are only slight differences in the flow characteristics of the different metals as related to internal threading. It will be necessary to deviate slightly from the recommended hole size when tapping extremely ductile or extra hard metals.

The formula for these theoretical hole size determinations is as follows:

Theoretical Hole Size

(core, punch or drill size) + Basic Tap O.D. minus .0068 x % of Thread
Threads per Inch

Example: To determine the proper drill size to form 65% of thread with a 1/4-20 cold form tap.

$$\frac{\text{Basic Tap O.D.} = 1/4\text{" or } .250}{\text{Threads per Inch} = 20}$$

$$\text{drill size} = .250 \text{ minus } \frac{.0068 \times 65}{20}$$

drill size = .228

Tapping Torque and Horse Power

Note: Numbers are in inch-pounds. All values given are for 1010 mild steel. For other materials multiply the values by the factors given in the torque and horsepower calculation table.

Tapping Torque & Horsepower Requirements
Torque Setting Data For HSS Straight Flute Plug Taps

Tap Size And Pitch	*Minimum Tapping Torque*	*Maximum Tapping Torque*	*Tap Breaking Torque (low Strength)*	*Tap Breaking Torque (High Strength)*	*Required Horsepower (Minimum)*	*Tap Holder Setting (Normal)*	*Tap Holder Setting (Minimum)*	*Tap Holder Setting (Maximum)*
#0-#2								
NC & NF	10	18	25	50	1/4	20	15	25
#3 & #4								
NC & NF	10	20	30	50	1/4	20	15	30
#5 & #6								
NC & NF	10	20	30	50	1/4	20	15	30
8-30	20	30	40	60	1/4	25	20	40
8-32	20	30	40	60	1/4	25	20	40
10-32	20	30	40	60	1/3	25	20	40
10-24	25	50	40	60	1/3	30	25	50
12-28	25	50	40	70	1/3	30	25	50
12-24	25	50	40	70	1/3	30	25	50
1/4-28	30	60	50	100	1/2	40	30	60
1/4-20	40	80	50	100	1/2	50	40	80
5/16-24	40	80	75	150	1/2	60	50	100
5/16-18	60	120	75	150	1/2	90	80	150
3/8-24	60	120	180	260	3/4	90	80	150
3/8-16	100	200	180	260	3/4	130	110	220
7/16-20	80	160	180	300	3/4	130	110	220
7/16-14	100	200	180	300	1”	200	180	300
1/2-20	100	250	300	600	3/4	300	300	450
1/2-13	150	300	300	600	1”	300	300	450
9/16-18	150	350	500	800	3/4	350	350	500
9/16-12	200	500	500	800	1” 1/4	350	350	500
5/8-18	200	600	800	1200	3/4	450	450	650
5/8-11	300	800	800	1200	1” 1/2	450	450	650
3/4-16	300	800	1000	1500	1”	650	650	950
3/4-10	500	1000	1000	1500	1” 3/4	650	650	950
7/8-14	500	1000	1500	2000	1”	850	850	1500
7/8-9	800	1500	1500	2000	2	850	850	1500
1”-12 & 14	700	1800	2000	2500	1” 1/2	1100	1100	1800
1”-8	1000	1800	2000	2500	2” 1/2	1100	1100	1800

(Continued)

Tapping Torque and Horse Power (Continued)

Torque Setting Data For Pipe Taps

Tap Size And Pitch	*Minimum Tapping Torque*	*Maximum Tapping Torque*	*Tap Breaking Torque (low Strength)*	*Tap Breaking Torque (High Strength)*	*Required Horsepower (Minimum)*	*Tap Holder Setting (Normal)*	*Tap Holder Setting (Minimum)*	*Tap Holder Setting (Maximum)*
1/16-27	60	120	75	150	1/2	90	80	150
1/8-28	80	160	180	300	1/2	130	110	220
1/4-18	200	500	500	800	1	350	350	500
3/8-18	300	600	700	1000	1	400	400	550
1/2-14	800	1300	1500	2000	1 1/2	850	800	1400
3/4-14	1000	1500	1800	2300	1 3/4	1200	1000	1600

Torque And Horsepower Calculation Factors

Material	*Factor*
Aluminum	.2
Brass	.4
Bronze	.4
Cast Iron	.6
Copper	.5
Magnesium	.5
Malleable Iron	.7
Zinc	.4
Titanium	1.4

Carbon (Mild) steel 1008-1095		*Free Cutting Steel 1111-1213*		*Alloy Steel 1330-8642*		*Alloy Steel 1330-8642*	
BHN	*Factor*	*BHN*	*Factor*	*BHN*	*Factor*	*BHN*	*Factor*
90	1.0	140	.7	175	.9	240	1.5
130	1.1	170	.8	190	1.0	250	1.6
170	1.2	230	.9	200	1.1	330	2.1
190	1.3			205	1.2	390	2.5
250	1.4			210	1.3	470	2.9

Tapping Torque vs. Thread Strength

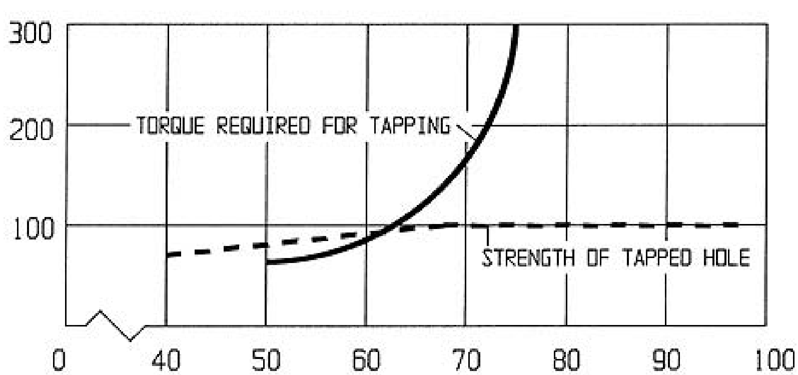
Suggested Percentage Of Full Threads In Tapped Holes

It stands to reason that it takes more power to tap to a full depth of thread than it does to tap to a partial depth of thread. The higher the metal removal rate, the more torque required to produce the cut.

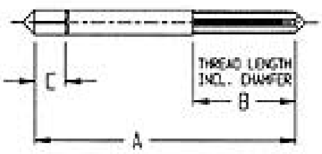
It would also stand to reason that the greater the depth of thread, the stronger the tapped hole. This is true, but only to a point. Beyond that point (usually about 75% of full thread) the strength of the hole does not increase, yet the torque required to tap the hole rises exponentially. Also, it becomes more difficult to hold size, and the likelihood of tap breakage increases. With this in mind, it does not make good tapping sense to generate threads deeper than the required strength of the thread dictates.

As a general rule, the tougher the material, the less the percentage of thread is needed to create a hole strong enough to do the job for which it was intended. In some harder materials such as stainless steel, Monel, and some heat-treated alloys, it is possible to tap to as little as 50% of full thread without sacrificing the usefulness of the tapped hole.

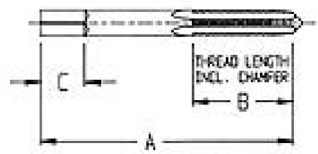
workpiece material	deep hole tapping	average commercial work	thin sheet stock or stampings
hard or tough cast steel drop forgings Monel metal nickel steel stainless steel	55% – 65%	60% – 70%	-
free-cutting aluminum brass bronze cast iron copper mild steel tools steel	60% – 70%	65% – 75%	75% – 85%



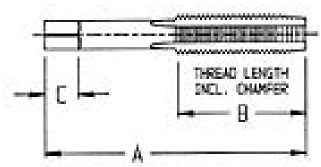
Standard Tap Dimensions (ANSI Shanks)



Style 1



Style 2

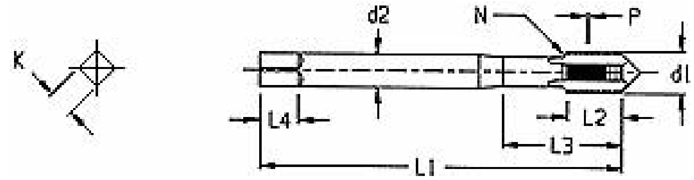


Style 3

Nominal Diameter Range - Inches		Mach. Screw Size No.	Nominal Fractional Diameter Inches	Nominal Metric Diameter Millimeters	STYLE	Tap Dimensions - Inches				
Over	To (Incl.)					Overall Length A	Thread Length B	Square Length C	Shank Diameter D	Size of Square
.052	.065	0	1/16	M1.6	1	1 5/8	5/16	3/16	.141	.110
.065	.078	1		M1.8	1	1 11/16	3/8	3/16	.141	.110
.078	.091	2		M2, M2.2	1	1 3/4	7/16	3/16	.141	.110
.091	.104	3	3/32	M2.5	1	1 13/16	1/2	3/16	.141	.110
.104	.117	4				1 7/8	9/16	3/16	.141	.110
.117	.130	5	1/8	M3, M3.15	1	1 15/16	5/8	3/16	.141	.110
.130	.145	6		M3.5	1	2	11/16	3/16	.141	.110
.145	.171	8	5/32	M4	1	2 1/8	3/4	1/4	.168	.131
.171	.197	10	3/16	M4.5, M5	1	2 3/8	7/8	1/4	.194	.152
.197	.223	12	7/32		1	2 3/8	15/16	9/32	.220	.165
.223	.260	14	1/4	M6, M6.3	2	2 1/2	1	5/16	.255	.191
.260	.323		5/16	M7, M8	2	2 23/32	1 1/8	3/8	.318	.238
.323	.385		3/8	M10	2	2 15/16	1 1/4	7/16	.381	.286
.385	.448		7/16		3	3 5/32	1 7/16	13/32	.323	.242
.448	.510		1/2	M12, M12.5	3	3 3/8	1 21/32	7/16	.367	.275
.510	.573		9/16	M14	3	3 19/32	1 21/32	1/2	.429	.322
.573	.635		5/8	M16	3	3 13/16	1 13/16	9/16	.480	.360
.635	.709		11/16	M18	3	4 1/32	1 13/16	5/8	.542	.406
.709	.760		3/4		3	4 1/4	2	11/16	.590	.442
.760	.823		13/16	M20	3	4 15/32	2	11/16	.652	.489
.823	.885		7/8	M22	3	4 11/16	2 7/32	3/4	.697	.523
.885	.948		15/16	M24	3	4 29/32	2 7/32	3/4	.760	.570
.948	1.010		1	M25	3	5 1/8	2 1/2	13/16	.800	.600
1.010	1.073		1 1/16	M27	3	5 1/8	2 1/2	7/8	.896	.672
1.073	1.135		1 1/8		3	5 7/16	2 9/16	7/8	.896	.672
1.135	1.198		1 3/16	M30	3	5 7/16	2 9/16	1	1.021	.766
1.198	1.260		1 1/4		3	5 3/4	2 9/16	1	1.021	.766
1.260	1.323		1 5/16	M33	3	5 3/4	2 9/16	1 1/16	1.108	.831
1.323	1.385		1 3/8		3	6 1/16	3	1 1/16	1.108	.831
1.385	1.448		1 7/16	M36	3	6 1/16	3	1 1/8	1.233	.925
1.448	1.510		1 1/2		3	6 3/8	3	1 1/8	1.233	.925
1.510	1.635		1 5/8	M39	3	6 11/16	3 3/16	1 1/8	1.305	.979
1.635	1.760		1 3/4	M42	3	7	3 3/16	1 1/4	1.430	1.072
1.760	1.885		1 7/8		3	7 5/16	3 9/16	1 1/4	1.519	1.139
1.885	2.010		2	M48	3	7 5/8	3 9/16	1 3/8	1.644	1.233
2.010	2.135		2 1/8		3	8	3 9/16	1 3/8	1.769	1.327
2.135	2.260		2 1/4	M56	3	8 1/4	3 9/16	1 7/16	1.894	1.420
2.260	2.385		2 3/8		3	8 1/2	4	1 7/16	2.019	1.514
2.385	2.510		2 1/2		3	8 3/4	4	1 1/2	2.100	1.575
2.510	2.635		2 5/8	M64	3	8 3/4	4	1 1/2	2.225	1.669
2.635	2.760		2 3/4		3	9 1/4	4	1 9/16	2.350	1.762
2.760	2.885		2 7/8	M72	3	9 1/4	4	1 9/16	2.475	1.856
2.885	3.010		3		3	9 3/4	4 9/16	1 5/8	2.543	1.907
3.010	3.135		3 1/8		3	9 3/4	4 9/16	1 5/8	2.668	2.001
3.135	3.260		3 1/4	M80	3	10	4 9/16	1 3/4	2.793	2.096
3.260	3.385		3 3/8		3	10	4 9/16	1 3/4	2.883	2.162
3.385	3.510		3 1/2		3	10 1/4	4 15/16	2	3.008	2.256
3.510	3.635		3 5/8	M90	3	10 1/4	4 15/16	2	3.133	2.350
3.635	3.760		3 3/4		3	10 1/2	5 5/16	2 1/8	3.217	2.413
3.760	3.885		3 7/8		3	10 1/2	5 5/16	2 1/8	3.342	2.506
3.885	4.010		4	M100	3	10 3/4	5 5/16	2 1/4	3.467	2.600

Standard Tap Dimensions (DIN Standard 371)

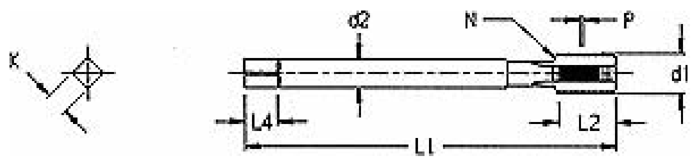
Metric Taps
To Din STD (371)
Metric Iso Threads



d_1 mm	P mm	L_1 mm	L_2 mm	L_3 mm	d_2 mm	k mm	L_4 mm	N
M1.6	.35	40	5	—	2.5	2.1	6	2
M2	.25	45	4	10	2.8	2.1	5	2
M2	.4	45	4	10	2.8	2.1	5	3
M2.2	.25	45	5	10	2.8	2.1	5	2
M2.2	.45	45	5	10	2.8	2.1	5	3
M2.3	.25	45	5	10	2.8	2.1	5	2
M2.3	.4	45	5	10	2.8	2.1	5	3
M2.5	.35	50	5	12	2.8	2.1	5	2
M2.5	.45	50	5	12	2.8	2.1	5	3
M2.6	.45	50	5	12	2.8	2.1	5	3
M3	.35	56	6	13	3.5	2.7	6	2
M3	.5	56	6	13	3.5	2.7	6	3
M3.5	.35	56	7	17	4	3	6	2
M3.5	.6	56	7	17	4	3	6	3
M4	.5	63	7	17	4.5	3.4	6	2
M4	.7	63	7	17	4.5	3.4	6	3
M4.5	.5	70	7	17	6	4.9	8	2
M4.5	.75	70	8	18	6	4.9	8	3
M5	.5	70	8	18	6	4.9	8	3
M5	.75	70	8	18	6	4.9	8	3
M5	.8	70	8	18	6	4.9	8	3
M6	.5	80	10	22	6	4.9	8	3
M6	.75	80	10	22	6	4.9	8	3
M6	1	80	10	22	6	4.9	8	3
M7	.75	80	10	22	7	5.5	8	3
M7	1	80	10	22	7	5.5	8	3
M8	1	90	13	28	8	6.2	9	3
M8	1.25	90	13	28	8	6.2	9	3
M10	1	90	13	28	8	6.2	9	3
M10	1.50	100	15	32	10	8	11	3

Standard Tap Dimensions (DIN Standard 374)

Metric Taps
To Din STD (374)
Metric Iso Threads

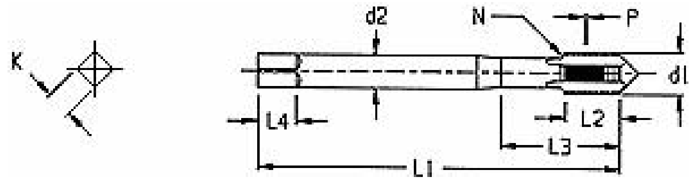


d_1 mm	P mm	L_1 mm	L_2 mm	d_2 mm	k mm	L_4 mm	K mm
M8	.75	80	10	6	4.9	8	3
M8	1	90	13	6	4.9	8	3
M9	1	90	13	7	5.5	8	3
M10	.75	90	12	7	5.5	8	3
M10	1	90	14	7	5.5	8	3
M10	1.25	100	15	7	5.5	8	3
M12	1	100	16	9	7	10	4
M12	1.25	100	16	9	7	10	4
M12	1.5	100	20	9	7	10	4
M14	1	100	16	11	9	12	4
M14	1.25	100	18	11	9	12	4
M14	1.5	100	20	11	9	12	4
M16	1	100	18	12	9	12	4
M16	1.5	100	22	12	9	12	4
M18	1	110	18	14	11	14	4
M18	1.5	110	22	14	11	14	4
M18	2	125	25	14	11	14	4
M20	1	125	20	16	12	15	4
M20	1.5	125	24	16	12	15	4
M20	2	140	25	16	12	15	4
M22	1	125	20	18	14.5	17	4
M22	1.5	125	24	18	14.5	17	5
M22	2	140	25	18	14.5	17	4
M24	1	140	22	18	14.5	17	5
M24	1.5	140	26	18	14.5	17	5
M24	2	140	28	18	14.5	17	5
M25	1.5	140	26	18	14.5	17	4
M26	1.5	140	26	18	14.5	17	4
M27	1	140	24	20	16	19	4
M27	1.5	140	26	20	16	19	4
M27	2	140	28	20	16	19	4
M28	1.5	140	28	20	16	19	4
M28	2	140	28	20	16	19	4
M30	1	150	26	22	18	21	4
M30	1.5	150	28	22	18	21	4
M30	2	150	28	22	18	21	4
M32	1.5	150	28	22	18	21	4
M32	2	150	28	22	18	21	4
M33	1.5	160	28	25	20	23	4
M33	2	160	30	25	20	23	4
M34	1.5	170	30	28	22	25	4
M35	1.5	170	30	28	22	25	4
M36	1.5	170	30	28	22	25	4
M36	2	170	30	28	22	25	4
M36	3	200	36	28	22	25	4
M38	1.5	170	30	28	22	25	5
M39	1.5	170	30	32	24	27	5
M39	2	170	30	32	24	27	4
M39	3	200	40	32	24	27	4
M40	1.5	170	30	32	24	27	5
M40	2	170	30	32	24	27	4
M42	1.5	170	30	32	24	27	5
M42	3	200	40	32	24	27	4
M45	1.5	180	32	36	29	32	5
M45	2	180	32	36	29	32	5
M45	3	200	40	36	29	32	4
M48	1.5	190	32	36	29	32	5
M48	2	190	32	36	29	32	5
M48	3	225	45	36	29	32	4
M50	1.5	190	32	36	29	32	5
M50	2	190	32	36	29	32	5
M52	1.5	190	32	40	32	35	5
M52	2	190	32	40	32	35	5
M52	3	225	45	40	32	35	5
M56	3	225	50	40	32	35	5

TAPMATIC

Standard Tap Dimensions (DIN Standard 376)

Metric Taps
To Din STD (376)
Metric Iso Threads



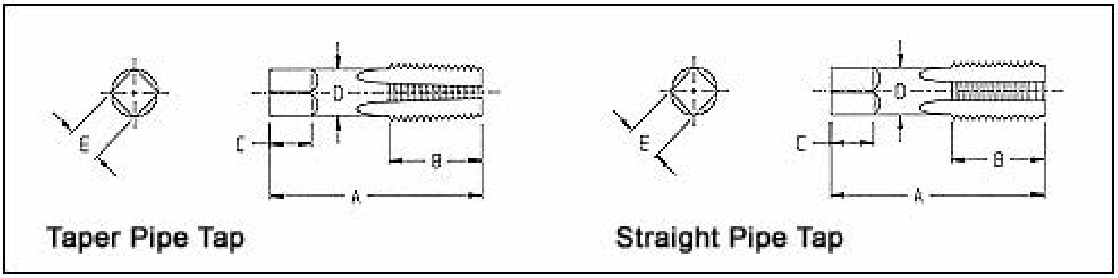
d_1 mm	P mm	L_1 mm	L_2 mm	L_3 mm	d_2 mm	k mm	L_4 mm	N
M1.6	.35	40	5	—	2.5	2.1	5	2
M2	.25	45	4	10	2.8	2.1	5	2
M2	.4	45	4	10	2.8	2.1	5	3
M2.2	.25	45	5	10	2.8	2.1	5	2
M2.2	.45	45	5	10	2.8	2.1	5	3
M2.3	.25	45	5	10	2.8	2.1	5	2
M2.3	.4	45	5	10	2.8	2.1	5	3
M2.5	.35	50	5	12	2.8	2.1	5	2
M2.5	.45	50	5	12	2.8	2.1	5	3
M2.6	.45	50	5	12	2.8	2.1	5	3
M3	.35	56	6	13	3.5	2.7	6	2
M3	.5	56	6	13	3.5	2.7	6	3
M3.5	.35	56	7	17	4	3	6	2
M3.5	.6	56	7	17	4	3	6	3
M4	.5	63	7	17	4.5	3.4	6	2
M4	.7	63	7	17	4.5	3.4	6	3
M4.5	.5	70	7	17	6	4.9	8	2
M4.5	.75	70	8	18	6	4.9	8	3
M5	.5	70	8	18	6	4.9	8	3
M5	.75	70	8	18	6	4.9	8	3
M5	.8	70	8	18	6	4.9	8	3
M6	.5	80	10	22	6	4.9	8	3
M6	.75	80	10	22	6	4.9	8	3
M6	1	80	10	22	6	4.9	8	3
M7	.75	80	10	22	7	5.5	8	3
M7	1	80	10	22	7	5.5	8	3
M8	1	90	13	28	8	6.2	9	3
M8	1.25	90	13	28	8	6.2	9	3
M10	1	90	13	28	8	6.2	9	3
M10	1.50	100	15	32	10	8	11	3

JIS

d_1 mm	P mm	L_1 mm	L_2 mm	d_2 mm	k mm	L_4 mm	N
M1	.25	30	8	3	2.6	5	2
M1.1	.25	32	9	3	2.6	5	2
M1.2	.25	32	9	3	2.5	5	2
M1.4	.3	34	11	3	2.5	5	2
M1.6	.35	36	13	3	2.5	5	2
M1.7	.35	36	13	3	2.5	5	2
M1.8	.35	36	13	3	2.5	5	2
M2	.4	40	15	3	2.5	5	2
M2.2	.45	42	15	3	2.5	5	3
M2.3	.4	42	15	3	2.5	5	3
M2.5	.45	44	16	3	2.5	5	3
M2.6	.45	44	16	3	2.5	5	3
M3	.5	46	18	4	3.2	6	3
M3.6	.6	48	18	4	3.2	6	3
M4	.7	52	20	5	4	7	3
M4.5	.75	55	20	5	4	7	3
M5	.6	60	22	5.5	4.5	7	3
M6	1	62	24	6	4.5	7	3
M7	1	65	26	6.2	5	8	3
M8	1.25	70	30	6.2	5	8	3
M9	1.25	72	30	7	5.5	8	3
M10	1.5	75	32	7	5.5	8	3
M11	1.5	80	36	8	6	9	4
M12	1.75	82	38	8.5	6.5	9	4
M14	2	88	42	10.5	8	11	4
M16	2	95	45	12.5	10	13	4
M18	2.5	100	48	14	11	14	4
M20	2.5	105	50	15	12	15	4
M22	2.5	115	56	17	13	16	4
M24	3	120	58	19	15	18	4
M27	3	130	62	20	16	18	4
M30	3.5	135	65	23	17	20	4
M33	3.5	145	70	25	19	22	4
M36	4	155	75	28	21	24	4
M39	4	165	80	30	23	26	4
M42	4.5	175	85	32	26	30	4
M45	4.5	180	85	35	28	30	4
M48	5	185	90	38	29	32	4

TAPMATIC

Standard Pipe Tap Dimensions (ANSI/DIN)



Taper Pipe Taps

Nominal Size	Threads/In		# of Flutes		Overall Length A	Length of Thread B	Square Length C	Shank Diameter D	Size of Square E
	Carbon Steel	High Steel	Regular	Interrupted					
1/16	—	27	4	—	2 1/8	11/16	3/8	.3125	.234
1/8	27	27	4	5	2 1/8	3/4	3/8	.3125	.234
1/8	27	27	4	5	2 1/8	3/4	3/8	.4375	.328
1/4	18	18	4	5	2 7/16	1 1/16	7/16	.5625	.421
3/8	18	18	4	5	2 9/16	1 1/16	1/2	.7000	.531
1/2	14	14	4	5	3 1/8	1 3/8	5/8	.6875	.515
3/4	14	14	5	5	3 1/4	1 3/8	11/16	.9063	.679
1	11 1/2	11 1/2	5	5	3 3/4	1 3/4	13/16	1.1250	.843
1 1/4	11 1/2	11 1/2	5	5	4	1 3/4	15/16	1.3125	.984
1 1/2	11 1/2	11 1/2	7	7	4 1/4	1 3/4	1	1.5000	1.125
2	11 1/2	11 1/2	7	7	4 1/2	1 3/4	1 1/8	1.8750	1.406
2 1/2	8	—	8	—	5 1/2	2 9/16	1 1/4	2.250	1.687
3	8	—	8	—	6	2 5/8	1 3/8	2.625	1.968

Straight Pipe Taps

Nominal Size	Threads/In		# of Flutes		Overall Length A	Length of Thread B	Square Length C	Shank Diameter D	Size of Square E
	Carbon Steel	High Steel	Regular	Interrupted					
1/8	—	27	4	—	2 1/8	3/4	3/8	.3125	.234
1/8	—	27	4	—	2 1/8	3/4	3/8	.4375	.328
1/4	—	18	4	—	2 7/16	1 1/16	7/16	.5625	.421
3/8	—	18	4	—	2 9/16	1 1/16	1/2	.7000	.531
1/2	—	14	4	—	3 1/8	1 3/8	5/8	.6875	.575
3/4	—	14	5	—	3 1/4	1 3/8	11/16	.9063	.679
1	—	11 1/2	5	—	3 3/4	1 3/4	13/16	1.1250	.843

Din Shank Pipe Taps NPT

Nominal Size	Threads/In		# of Flutes		Overall Length A	Length of Thread B	Square Length C	Shank Diameter D	Size of Square E
	Carbon Steel	High Steel	Regular	Interrupted					
1/16	—	27	4	—	2 1/8	11/16	3/8	.3125	.234
1/8	27	27	4	5	2 1/8	3/4	3/8	.3125	.234
1/8	27	27	4	5	2 1/8	3/4	3/8	.4375	.328
1/4	18	18	4	5	2 7/16	1 1/16	7/16	.5625	.421
3/8	18	18	4	5	2 9/16	1 1/16	1/2	.7000	.531
1/2	14	14	4	5	3 1/8	1 3/8	5/8	.6875	.515
3/4	14	14	5	5	3 1/4	1 3/8	11/16	.9063	.679
1	11 1/2	11 1/2	5	5	3 3/4	1 3/4	13/16	1.1250	.843
1 1/4	11 1/2	11 1/2	5	5	4	1 3/4	15/16	1.3125	.984
1 1/2	11 1/2	11 1/2	7	7	4 1/4	1 3/4	1	1.5000	1.125
2	11 1/2	11 1/2	7	7	4 1/2	1 3/4	1 1/8	1.8750	1.406
2 1/2	8	—	8	—	5 1/2	2 9/16	1 1/4	2.250	1.687
3	8	—	8	—	6	2 5/8	1 3/8	2.625	1.968

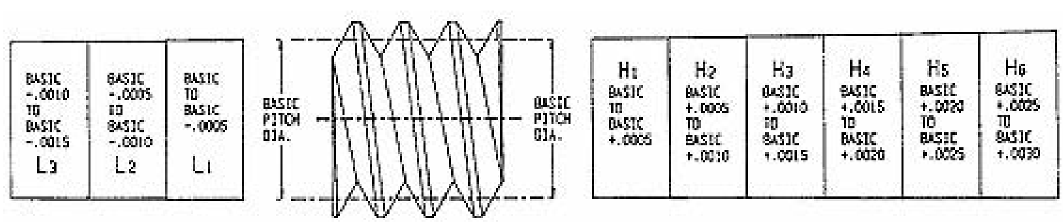
Class of Threads, H Limits

Classes Of Threads
There are (3) established Classes of Thread, designated in the unified series by adding: “A” for Screws and “B” for Nuts (or other intenal threads) to show definite limits and tolerances. Class 1B Thread is where a 1A screw can run in readily for quick and easy assembly. The hole is classified as 1B. The fit is a 1B thread, (very seldom used in modern metal working)

Class 2B Thread
Consists of a 2A screw in a 2B hole. 2B thread has wide applications. It is used to accomodate plating, finishing and coating to a limited extent and therefore, has fair tolerance allowances.

Class 3B Thread
3A screw in a 3B nut or internal threaded hole, used where tolerance limits are close.

GH Numbers
GH Numbers are listed below. “G” designates Ground Thread. “H” designates the pitch diameter is on high side of basic. These two letters (GH) are followed by a numeral indicating the Tolerance of Pitch diameter oversize.



H1	=	Basic to Plus .0005	
H2	=	Basic Plus .0005 to Plus .0010	
H3	=	Basic Plus .0010 to Plus .0015	
H4	=	Basic Plus .0015 to Plus .0020	
H5	=	Basic Plus .0020 to Plus .0025	
H6	=	Basic Plus .0025 to Plus .0030	H=Above Basic
H7	=	Basic Plus .0030 to Plus .0035	L=Below Basic

(Continued)

Class of Threads, H Limits (Continued)

Relation Of Tap Pitch Diameter to Basic Pitch Diameter

American Tap Manufacturers use a series of tap pitch diameter limits. These limits feature a .0005 tolerance in tap sizes #0 Thru 1" and a .001" or greater tolerance in tap sizes above 1" thru 1 1/2" diameter. **Example:** 1/4-20. Relationship between Tap Pitch diameter limits and basic nominal pitch diameter.

GH5 —-
.2200
GH4
—-.2190
Basic GH3
—-.2185
Pitch Diameter: GH2
.2175 —-.2180
GH1
—-.2175
GL1
—-.2170

Notes:

1. A tap cannot produce a class of thread it can produce a tapped hole within specific product limits.
2. Since the tap is used only in tapping a hole or producing an internal thread, a tap has no control over the fitting properties of the mating external thread.
3. To produce what is commonly referred to as a class of thread both external and internal threads must be within their respective product limits. Only when both members of a thread assembly fall within their desired class limits can the proper fit be assured.
4. The acceptability of any class of threaded hole is determined only by an accurate "GO" or "HI" Thread plug gage of corresponding class. The acceptability of the male part with an external Thread is also determined by a corresponding "GO" or "LO" Thread Ring gage.
5. Tap limits refer to the various sizes of tap manufactured. A tap whould be selected which will produce an internal Thread within the desired product limit. Tap limits are designated as L1, H1, H2, H3 etc.
6. Although ground taps are produced to precision tolerances under closely controlled manufacturing processes and are guaranteed for accuracy of individual elements, there is always the possibility of the presence of unknown factors which can be a detriment to good tap perfomance.

Surface Treatments for Taps

Nitride
A hard superficial case, approximately 68 HRC, on the surface of a finished tap produced by means of a cyanide salt bath.

Purpose to resist abrasion and increase wear resistance due to the higher surface hardness.

Application effective in both abrasive and tough materials, cast iron, plastic, stainless steel and high tensile strength steels.

Note! Care must be used when selecting nitride surface treatment because the increase hardness has a tendency to make the tap easy to chip and damage: Nitride not recommended for fast spiral flute taps and taps smaller than machine screw #2.

Double Nitride
Very similar to Nitride surface treatment with the exception that the hard case produced is deeper and harder than standard Nitride.

Application extremely abrasive materials, plastic and gray cast iron.

Oxided
Produced on surface of a finished tap by means of a steam furnace or cyanide salt bath. Well know heat treatment by which an oxide layer (Fe3O4) is formed on the surface of the tap. This will improve the adherence of threading agent which leads to improved output of taps.

Categories of Oxide

Steam Oxide: To counteract galling or loading lubricate tap surfaces. Best for low carbon, leaded steel, stainless and gummy material.

Nitride and Oxide: For stress relief and light coating. Copper alloys of medium machinability.

Nitride Plus Steam Oxide: To add wear life and reduce loading. High speed production tapping, poor lubrication. Steam Oxide Plus Nitride: To add wear life and provide self lubrication. Use in cast iron.

Heavy Nitride Plus Steam Oxide: To add wear life in hard and dense metals. For tapping hard steel alloys, titanium, exotic metals and hard copper alloys.

Black Oxide: Helps retain cutting fluid in the working portion of the tap. Improves Performance in stainless steel, steel forgings, tool and die steel, and hot and cold rolled steels.

(Continued)

Surface Treatment for Taps (Continued)

Hard Chrome
A surface treatment in the form of a thin hard chromium layer deposit (.0001 approximately). Increases the taps surface hardness and help reduce torque required to drive the tap.

Purpose: Proven very advantageous in non-ferrous materials, such as copper, brass and bronze.

PVD Process (TiN, and TiCN)
Used to resist abrasion and chip welding. Biggest potential is for ferrous materials below 40RC.

TiN Titanium Nitriding
In the PVD treatment a 2-4 micron layer is formed. The coating is a gold color with a hardness of about 2300 HV with good friction characteristics and coating adhesion for improved tool life. TiN coating remains resistant up to 600 degrees centigrade.

TiCN Titanium Carbonitriding
A similar PVD process as TiN coating. Friction characteristics are still better than TiN. The TiCN coating remains resistant up to 400 degrees centigrade. The coating is a grey-purple color.

Insulation
A method of surface treatment which has a marked influence on diminishing the possibility of “cold welding” especially good for machining softer steels.

Jetting
A surface treatment through which the sliding property of the tap is increased, especially for machining different nonferrous metals.

Tapping Formulas

RPM = (SFM x 3.82) divided by D   D = Diameter of Tap in inches

SFM = (3.14 x D x RPM) divided by 12   D = Diameter of Tap in inches

Inch Taps
Drill Size = Major Diameter of Tap minus $\dfrac{.01299 \text{ x \% of Full Thread}}{\text{\# of Threads / inch}}$

Inch Taps
% of Full Thread = Threads/in x $\dfrac{\text{Major Dia. of Tap minus Drill Diameter}}{.01299}$

Metric Taps
Drill Size = Major Diameter of Tap (mm) minus $\dfrac{\text{\% of Thread x Metric Pitch}}{76.98}$

Metric Taps
% of Full Thread = $\dfrac{76.980}{\text{Metric Pitch}}$ x (Basic Major Diameter (mm) minus Drilled Hole (mm))

Form Tap
Drill Size = Basic Tap OD minus $\dfrac{.0068 \text{ x \% of Threads}}{\text{Threads / Inch}}$

Inch Taps
IPM (For Threads) = RPM divided by TPI (Threads per Inch)

(Continued)

Tapping Formulas (Continued)

Inch Taps
IPR (For Threads) = 1 divided by TPI

Inch Taps
IPM (Inches Per Minute) = IPR x RPM

Inch Taps
IPR (Inches Per Rev.) = IPM divided by RPM

Metric Taps
MM/Min = RPM x Metric Pitch

Inch Taps
In/Min = RPM divided by # of Threads / in.

Distance = Rate x Time

Time = $\frac{\text{Distance}}{\text{Rate}}$

(Continued)

Troubleshooting

General Problem: Dimensional Accuracy

Specific	Cause	Solution
	Incorrect Tap	1. Use Proper GH limits of taps 2. Use longer chamfered taps.
	Chip Packing	1. Use spiral point or spiral fluted taps. 2. Reduce number of flutes to provide extra chip room. 3, Use larger hole size. 4. If tapping a blind hole, allow deeper holes where applicable or shorten the thread length of the parts. 5. Use proper lubricant.
Oversize Pitch Diameter	Galling	1. Apply proper surface treatment such as steam oxide or chrome. 2. Use proper cutting lubricant. 3. Reduce tapping speed. 4. Use proper cutting angle in accordance with material being tapped. 5. Use larger hole size.
	Operating Conditions	1. Apply proper tapping speed. 2. Correct alignment of tap and drilled hole. 3. Use proper tapping speed to avoid torn or rough threads. 4. Use tapping attachment with axial compensation. 5. Use proper tapping machine with suitable power. 6. Avoid misalignment of the tap and drilled hole from loose spindle or worn holder.
	Tool Conditions	1. Obtain proper indexing angle for the flutes at the cutting edge. 2. Grind proper cutting angle and chamfer angle. 3. Avoid too narrow a land width. 4. Remove burrs from regrinding.
Oversize Internal	Hole Size	1. Use minimum hole size. 2. Avoid tapered hole 3. Use proper chamfered taps.
	Galling	Galling solutions 1 through 4 can be applied to this specific problem.

(Continued)

Undersize Pitch	Incorrect Tap	1. Use Oversize taps. a. Use for cutting materials such as copper alloy, aluminum alloy and cast iron. b. Use for cutting tubing which will have “spring back” action after tapping. 2. Apply proper chamfer angle. 3. Increase cutting angle.
	Damaged Thread	1. Use proper reversing speed to avoid damaging tapped thread on the way out of the hole.
	Leftover Chips	1. Increase cutting performance to avoid any left over chips in the hole.
Undersize Internal Diameter	Hole Size	1. Use maximum drill size

General Problem: Surface Finish

Specific	**Cause**	**Solution**
Torn or Rough Thread	Chamfer Too Short	1. Increase chamfer length.
	Wrong Cutting Angle	1. Apply proper cutting angle.
	Galling	1. Use thread relieved taps. 2. Reduce land width. 3. Apply surface treatment such as steam oxide or chrome. 4. Use proper cutting lubricant. 5. Reduce tapping speed. 6. Use larger hole size. 7. Obtain proper alignment between tap and work.
	Chip Packing	1. Use spiral pointed or spiral fluted taps. 2. Use larger drill size.
	Tool Condition	1. Avoid too narrow a land width. 2. Do not grind the bottom of the flute.
Chattering on Tapped Thread	Tool Free Cutting	1. Reduce cutting angle. 2. Reduce amount of thread relief.

(Continued)

General Problem: Tool Life

Specific	Cause	Solution
Breakage	Incorrect Tap Selection	1. Avoid chip packing in the flutes or the bottom of the hole. Use spiral pointed or spiral fluted taps or fluteless taps. 2. Apply correct surface treatment such as steam oxide or bright.
	Excessive Tapping Torque	1. Use larger drill size. 2. Try to shorten thread length. 3. Increase cutting angle. 4. Apply a tap with more thread relief and reduced land width. 5. Use spiral pointed or spiral fluted taps.
	Operating Condition	1. Reduce tapping speed. 2. Avoid misalignment between tap and the hole and tapered hole. 3. Use floating type of tapping holder. 4. Use tapping holder with torque adjustment. 5. Avoid hitting bottom of the hole with tap.
	Tool Condition	1. Do not grind bottom of the flute. 2. Avoid too narrow a land width. 3. Remove all worn sections when regrinding the flutes. 4. Regrind tool more frequently.
Chipping	Incorrect Tap Selection	1. Reduce cutting angle. 2. Use a different kind of high speed steel tap. 3. Reduce hardness of the tap. 4. Increase chamfer length. 5. Avoid chip packing in the flutes or in the bottom of the hole by using spiral fluted or spiral pointed taps.
	Operating Conditions	1. Reduce tapping speed. 2. Avoid misalignment between tap and hole. 3. Avoid sudden return of reverse in blind hole tapping. 4. Avoid galling. 5. Use larger hole size.
Wear	Incorrect Tap Selection	1. Apply specially designed taps for tapping heat treated material. 2. Change to a type of high speed steel tap that contains vanadium. 3. Apply special surface treatment such as nitriding. 4. Increase chamfer length.
	Operating Conditions	1. Reduce tapping speed. 2. Apply proper cutting lubricants. 3. Avoid work hardened hole. 4. Use larger hole size.
	Tool Condition	1. Grind proper cutting angle. 2. Avoid hardness reduction from grinding process.

(Continued)

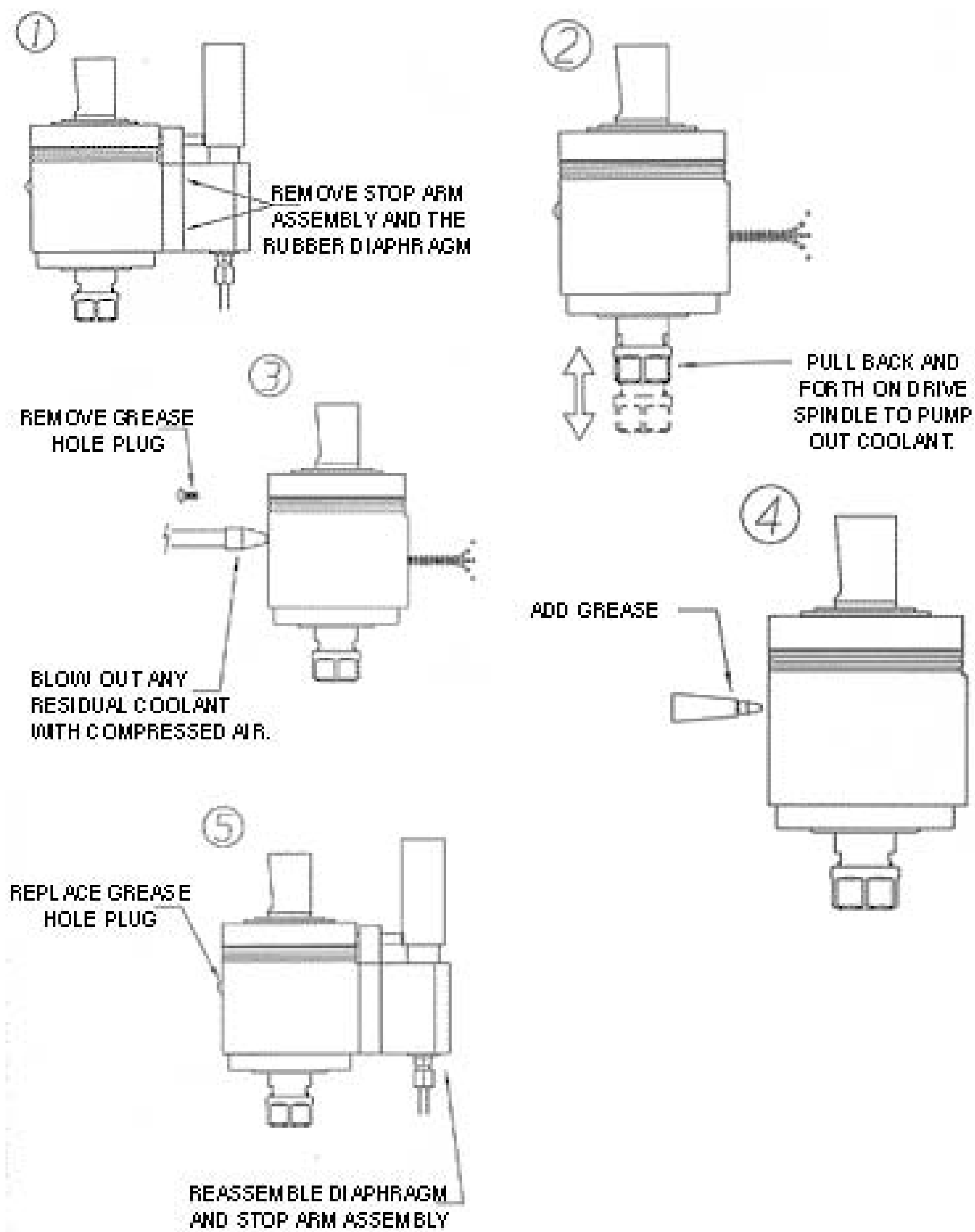
General Problems Related To Tap Holder

Specific	Cause	Solution
Tap stops before reaching required depth	Clutch of X, R or TC/DC is slipping with new tap.	1. Increase torque adjustment until a sharp tap can be driven to proper depth. Then add one half turn and lock adjustment
	Clutch of X, R or TC/DC is slipping with dull tap.	1. Change to new sharp tap.
	Clutch of X, R or TC/DC slips when fully tightened.	1. The torque required is too great for the tapping attachment. Use a larger model. 2. Clutch may be worn and needs replacing.
	The housing of the X, R or TC/DC stops rotating.	1. The torque required is too great for the machine and the machine spindle is stopping. A machine with more torque is needed.
	Drive disengages and goes into neutral.	1. Set machine stop to allow tap to feed deeper into hole.
	The self-feed adjustment of TC/DC is set to minimum self-feed and drive goes into neutral.	1. Back off self-feed adjustment for more forward drive engagement and adjust machine stop. 2. Note: There can be confusion between the clutch slipping and the drive going into neutral. You can determine what is occuring from the sound. With the R model, there is a loud ratcheting sound when the clutch slips. When the drive goes into neutral it is quiet. With the X or TC/DC model when the clutch slips there is no sound and when the drive goes into neutral there is a clicking sound.
Tap pulls out of collet chuck.	Tap chuck nut is not tightened securely.	1. Be sure to follow operator instructions for installation of tap.
	Tap square is not being driven properly.	1. Be sure to follow operator instructions for installation of tap.
	Back jaws for driving tap square are damaged.	1. Replace back jaws and please see operator instructions for installation of tap.
Tap stops and starts chattering on the way out of the hole.	With X, R or TC/DC speed increases by 1.75 times for reverse, operator is not feeding fast enough to keep up with tap.	1. Retract the machine spindle at a faster rate with a smooth motion.
	With RDT, ID or NCRT on a machining center, the feed rate is not keeping up with the tap.	1. Adjust machines feed rate correctly according to operator instructions. 2. Be sure that potentiometer over ride control is canceled. Please see programming in operator instructions. 3. Be sure that “Ramp” or “Exact Stop” is not in effect. Please see programming in operator instructions.

(Continued)

Self-reversing tapping attachment does not reverse.	Stop arm is not installed or prevented from rotating.	1. Please see operator instructions for stop arm and torque bar installation.
Coolant has entered the housing of the tapping attachment.	Tapping attachment is being flooded with external coolant.	1. Try to avoid flooding tapping attachment itself.
	Internal Coolant system is leaking due to pressure build up.	1. Please be sure not to exceed maximum recommended pressure. Please be sure outlet for coolant flow is adequate. Please see operator instructions.

Note:
Coolant can be removed by using the following procedure.



TAPMATIC

For Immediate Assistance call (800) 854-6019
or E-mail us at info@tapmatic.com