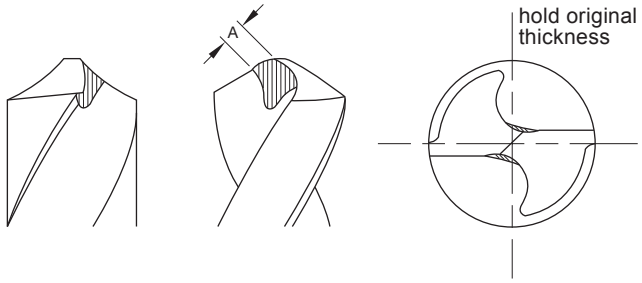
Hints on Use & Maintenance

Web Thinning

On most drills the web increases in thickness towards the shank with the result that, as the drill is shortened by repeated sharpening, the chisel edge will become wider. As the chisel edge does not cut but forces the metal out of the way, too wide a chisel edge will result in more pressure required for penetration, leading to greater heat generation and a resultant loss of life.



Cutting Fluids

The use of cutting fluids is an advantage in most drilling operations and an essential in some. The two main functions of the cutting fluid are lubrication
and cooling. The purpose of lubrication is to reduce friction by lubricating the surfaces tool and work, to facilitate easier sliding of the chips up the flute
and to prevent the chips welding to the cutting edges. In production work, particularly when drilling deep holes, the cooling action of the fluid is often more important than the lubrication. Overheating will shorten the life of the drill. Intermittent feed on deep holes, where possible, not only clears the chips but permits more effective cooling.

Speeds

The speed of a drill is the rate at which the periphery of the drill moves in relation to the work being drilled. As a rule, with a drill working within its speed range for a specific material, more holes between sharpening will be achieved if the speed is reduced and less holes if the speed is increased. Thus, for each production run, a speed must be established which will result in the highest rate of production without excessive breakdown time or drill usage. The factors governing speed are: component material, hardness of material, depth of hole, quality required, condition of drilling machine, efficiency of cutting fluid.

Feeds

The feed of the drill is governed by the drill size and the component material. As with speeds, an increase in feed will lessen the number of holes produced sharpening but it is essential that a constant feed be maintained. If a drill is allowed to dwell, breakdown of the cutting edges will result.

Small Drill Feeds and Speeds

Breakdown of small drills can most often be attributed to two faults: speed too high and feed too low. A feed which will produce CHIPS not POWDER, coupled with a speed compatible with the strength of the drill is essential for small hole drilling. Feeds must be based on thickness of chip, not mm/min, and speeds adjusted accordingly. EXAMPLE: A 1mm drill is to operate at a feed of 0.013mm /rev, drilling steel. While the material may permit a speed of 30m/min or 9,500 RPM it is obvious that the drill could not withstand a load of 0.013mm feed at this speed; a penetration rate of 124mm/min. The correct procedure is to retain the feed but reduce the speed to obtain a penetration within the capacity of the strength of the drill.

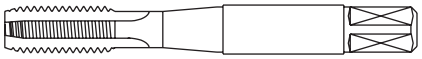
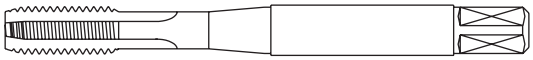
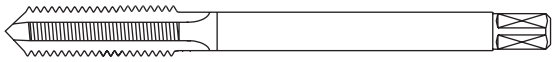
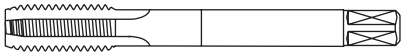
Deep Hole Drilling

When drilling deep holes, speeds and feeds should be reduced as follows:

Depth of hole	Reduction percent %	
	Speed	Feed
3 times drill diameter	10	10
4 times drill diameter	30	10
5 times drill diameter	30	20
6 to 8 times drill diameter	35 to 40	20

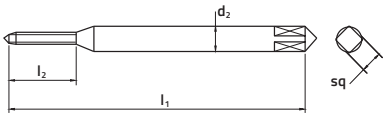
suttontools

Construction dimensions / designs

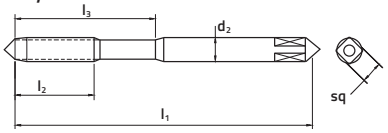
Short Machine & Hand Taps	ISO 529 JIS (J TYPE)	
Reinforced Shank Taps	DIN371	
Reduced Shank Taps	DIN374 / DIN376 / DIN5156	
Machine Nut Taps	ANSI B949 Standard	
Pipe Taps	Rc(BSPT), G (BSPF), Rp (BSPPL) - ISO2284 Standard NPT, NPTF, NPSF - ANSI B949 Standard	

Tap Styles

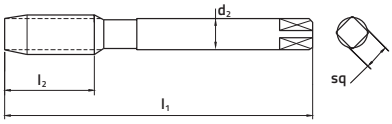
Style 1



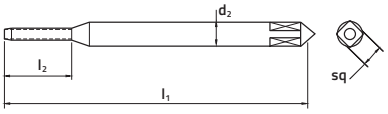
Style 2



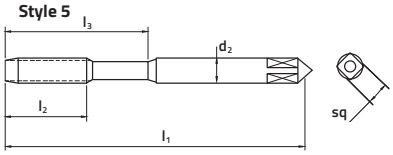
Style 3



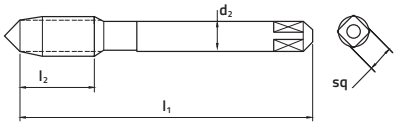
Style 4



Style 5

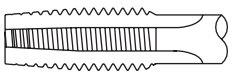
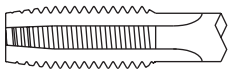
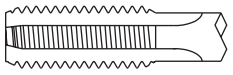
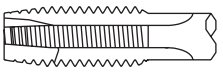
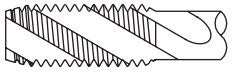


Style 6



Chamfer Type / Length

Table below is in accordance with ISO8830 / DIN2197

Terminology	Form	Number of threads on lead	Chamfer angle (≈)	Type of flute	Main area of application	Illustration
TAPER	A	6 to 8	5°	Hand or straight flutes	Short through holes	
INTERMEDIATE	D	3.5 to 5	8°	Hand or straight	Generally for through holes	
BOTTOMING	E*	1.5 to 2	23°	Hand or straight flutes	Blind holes with very short thread run out	
INTERMEDIATE	B	3.5 to 5	10°	Straight, with spiral point	Through holes in medium & long chipping materials	
BOTTOMING	C	2 to 3	15°	Spiral fluted	Generally for blind holes	

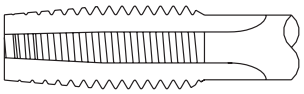
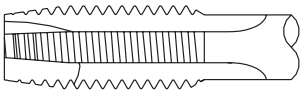
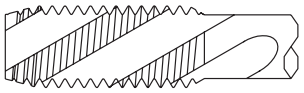
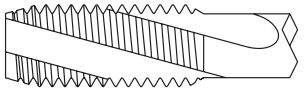
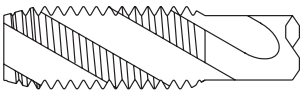
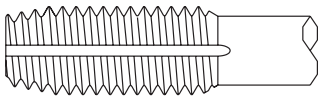
* Use of this type is not recommended

Technical Information Taps

suttontools

Tap Types - Helix direction/ Helical pitch / Fluteless

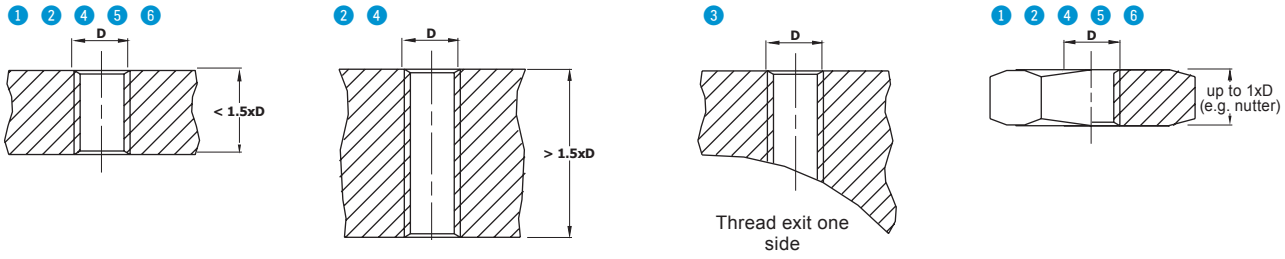
The helix angle depends primarily upon the hole form, eg. Through hole , blind hole, deep blind hole, etc., but the material, eg short chips, long chips, also has a strong influence on the direction of the helix. The following basic forms have derived during the development of taps:

Description	Illustration
❶ Straight Flutes (Hand) - Suitable for through or blind holes. The flutes only have room for a small amount of chips. The chips are not transported axially. Therefore, it is not advisable to cut deep through or blind holes (except in short chipping materials), with this type.	
❷ Straight Flutes with (Gun) – Suitable for through holes, the gun point curls the chip forward ahead of the tap & out of the hole. Therefore, chip clogging is avoided and coolant can flow without problems.	
❸ Spiral Flutes (LH Spiral, right hand cutting) – Suitable for interrupted through holes, where cross-holes exist. The direction of the flutes, curls & transports the chips forward of the tap, similar to Gun taps (also, opposite to RH spiral flutes). However, in applications where another hole intersects with the tapped hole, the helical flutes maintain the pitching of the thread.	
❹ 15° Spiral Flutes (RH Spiral) – Suitable for blind holes, best suited to tough short chipping materials, up to 1.5 x D in depth. This particular tap design has no advantages for soft, and long chipping materials, especially over 1.5 x d_1 in depth. Due to the slow helix angle not transporting the chips well, clogging is possible.	
❺ 40° to 50° Spiral Flutes (RH Spiral) – Suitable for blind holes, best suited to long chipping materials, the high helix angle & the direction of the flutes, curls & transports the chips back out of the hole. This particular tap style is required to cut on reversal; therefore flute rake is required on the both front & back flute faces.	
❻ Thredflo/Roll taps (fluteless) - Suitable for blind & through holes. This type of tap internally rolls a thread, therefore displacing the metal rather than cutting, like the above mentioned styles. Due to torque generated when producing roll threads, much higher machine power is required. Roll threads also produce much stronger threads than cut threads, as the grain structure of the thread remains uniform through the thread form profile. Note! Tapping drill size is not the same as a cut thread tap.	

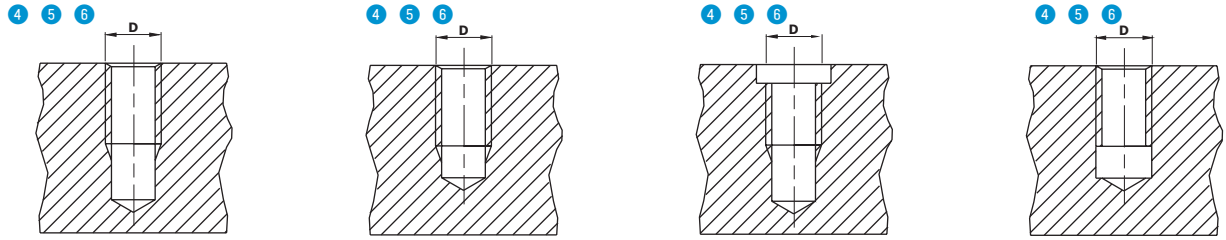
The above basic tool types are available in different variations, which have been designed & developed in respect to the specific materials and working conditions.

Tap Hole Types

Through Holes



Blind Holes



For blind holes, there are generally two thread run out forms used at the bottom of the tap hole. One form has a recessed diameter at the bottom of the hole, and the other form has a standard run out. Other types of holes are respective to construction designs, eg.

a) The bore is smaller than the tap hole diameter (typical for pipes)
b) As step hole, where the following diameter (second step), is smaller than the tap hole diameter.