Types of Tap

HAND TAPS ISO 529

These are straight flute general purpose tools which can be used for both machine or hand tapping. They are generally the most economical tool for use on production runs, but are best on materials that produce chips, or where the swarf breaks readily. Where deep holes are to be tapped, in materials which produce stringy swarf, serial taps may be needed, especially for coarse threads.

ISO 529 hand taps can be supplied in sets of three; bottom, second and taper leads, or individually.

BOTTOM TAPS have a chamfer (lead) of 1–2 threads, the angle of the lead being around 18 degrees per side. They are used to produce threads close to the bottom of blind holes.

SECOND TAPS have a lead of 3-5 threads at 8 degrees per side. They are the most popular and can be used for through holes, or blind holes where the thread does not need to go right to the bottom.

TAPER TAPS have a lead of 7-10 threads at 5 degrees per side. The taper lead distributes the cutting force over a large area, and the taper shape helps the thread to start. They can therefore be used to start a thread prior to use of second or bottom leads, or for through holes.

IMPORTANT NOTE ON TERMINOLOGY! In the U.K. bottom taps are often referred to as 'plugs'. In North America second taps are often referred to as 'plugs'! This can easily lead to confusion. To avoid problems when ordering it is best to use the terms bottom, second and taper.
Another variation is that second taps are sometimes referred to as 'intermediate', and taper taps can be called 'firsts'.

SERIAL TAPS DIN 352

As a rule of thumb ISO 529 taps should not be used to produce threads longer than 1.5 times the tap diameter as breakages may occur. For deeper holes and in tougher or difficult to work materials serial taps can be used.

Serial taps come in sets of three and have leads similar to ISO 529 taps. They are however graduated in diameter so that the first tap used, recognised by one groove on the shank, is smaller than the second which has two grooves. The required thread diameter is achieved by the finishing tap, no grooves. The material is therefore removed in stages so that there is less stress generated than with ISO 529 taps. The disadvantage of these taps is the longer time taken to produce a thread.

MACHINE TAPS

Machine taps are so called because they are designed to be run at higher speeds, and need less cutting power than hand taps.

SPIRAL POINT **Also known as 'gun nose' or 'bull nose' or chip driver. These taps are dimensionally the same as a hand tap, second lead, but have the cutting face ground back relative to the axis of the tap, for the lead portion. This gives the flute a better cutting action, requiring less power, and pushes the cut material forward, allowing free flow of coolant along the flutes to the cutting edge. The flutes are not ground as deeply as for hand taps, giving the tap greater strength. It can therefore be run at higher speeds.**
Spiral point taps are ideal for machine tapping of through holes, or blind holes where there is enough clearance beyond the threaded portion to accommodate the swarf.

SPIRAL FLUTE **These taps are used to produce a thread close to the bottom of a blind hole and therefore have a very short lead. The right hand spiral cut of the flutes acts to force the swarf away from the cutting teeth to the rear of the flutes and out of the hole. They are better on materials which form long continuous stringy swarf, rather than chips. They are also better to tap a thread in a hole where there is a break in the material, e.g. another hole, as the spiral fluting helps the tap to pick up on the other side.**
Spiral flute taps can have slow or fast helix angles. Slow spirals are used for non ferrous materials, and fast spirals for most other materials. We supply fast spiral as standard.

FLUTELESS **Otherwise known as 'Roll', 'Forming' or Polygon taps. Used for the chipless production of threads in ductile materials such as copper, aluminium or soft brass. As the name implies they do not have flutes but lobes, which contact the work piece to form the thread by extrusion. They are operated at high speeds and are better at maintaining size. As long as all of the correct operating criteria are met, i.e. speed, hole size and lubrication, they have a longer life and less breakages than with other types of machine taps. Because they produce no chips they are very suitable for blind hole tapping.**
Fluteless taps require different tapping drill sizes and higher operating speeds than conventional taps, see later. They also produce stronger threads.

LONG SHANK TAPS

MACHINE NUT (BS949) **These taps were designed for tapping nuts on machines, and have a long thread, shank and square length. The shank diameter is reduced below the thread minor diameter to enable the tap to run through the work piece. This was originally to allow nuts to collect on the shank. For nut production, taper leads were used to distribute the cutting action, but these taps are now generally used as long reach, and are supplied also in bottom and second leads.**

DIN 376 and 374 **These are European standard taps that are dimensionally different to the standard ISO 529 UK types. They are generally longer and have reduced shanks as do Nut Taps.**
Basically DIN 376 are for standard coarse threads, and DIN 374 for fine threads. A third type DIN 371 is available. These have bigger shanks, so are not really classed as long reach. All of these taps are available in the same types as ISO 529, see previous pages.

DIN 376 STRAIGHT FLUTE

DIN 376 SPIRAL POINT

DIN 376 SPIRAL FLUTE

PIPE TAPS

We supply BSP, BSPT to BS 949.9 and ANP taps to BS 949.3. See tap dimensions sheets.

OTHER TYPES

There are many other types of tap available, such as interrupted thread, series, tandem etc, but these are normally for special applications. We would advise you on enquiry should you require anything other than the standard forms listed here. We do stock some taps other than normal standards, as you can see below:-

TRAPEZOIDAL TAP

SURFACE COATINGS

While selecting the correct type of tap for a job, the material to be tapped should also be considered. This may determine the surface coating that should be applied to the tap in order to extend its life.

Most taps are supplied with no surface treatment. They are referred to as 'Bright Finish'. These taps are mainly for use on non-ferrous materials, or steels that do not cold weld. Bright finish taps are therefore suitable for all hand operations, where speeds are too low for cold welding to occur, and for most machine operations.

STEAM TEMPERING & NITRIDING.
Nitriding hardens the tap's surface skin and gives a grey/black appearance. It extends tap life when used on abrasive materials such as Copper, Aluminium alloys and Plastics, but it can make the tap brittle and more prone to breakage. Steam Tempering gives a blue/black finish. It causes minute imperfections on the surface of the tap. These cause the tapping lubricant to be retained more readily, reducing friction and the possibility of cold welding. This helps the tap to cope with both mild and stainless steels.

Both above processes are normally only required for Spiral point and Spiral flute taps when the work piece material requires them. These taps are known as 'Steam tempered and nitrided' and have a blue/black finish.

Titanium nitrided taps.

TITANIUM NITRIDE (Tin) **This is a gold coloured coating only a few microns thick which is not metallic so reduces the chance of cold welding. It is applied by vapour deposition using special purpose machines. It imparts to the tool a surface hardness of around 80-85 Rockwell. It also imparts a good abrasive resistance and subsequent decrease in friction between the tool and the work piece. This can allow tapping at higher speeds. Suitable for most stainless steels.**
The process can increase the price of a tap significantly. When the tap is reground the coating will be removed. This coating is more appropriate to Fluteless taps than steam tempering and nitriding.

TITANIUM CARBIDE NITRIDING (TiCn) **This coating is applied to the tool in the same way as Tin, and gives a silver/grey finish. It has the same qualities as Tin coating, but more so. It is equally more expensive. Suitable for very abrasive and high tensile materials.**

These are the surface treatments that we can offer. We do not keep all taps with all treatments already applied on the shelf. Our price list is for bright finish, but we keep most common standard coarse thread spiral point and spiral flute taps in bright or steam tempered and nitrided, at the same price as bright. Please enquire.

We would be happy to advise you the best tap and coating for your job.

SPEEDS, FEEDS AND LUBRICANTS

Tapping speeds are determined by many factors. The main ones are:-

a) **Thread pitch**
b) **Material being tapped**
c) **Hole depth**
d) **Hole type, through or blind**
e) **Depth of thread**
f) **Lubricant quality and flow rate**

Tapping speeds can be calculated using:- $\text{RPM} = \dfrac{\text{Feed speed (SFM)}}{0.26 \times \text{Tap OD}}$

Recommended SFM are given later.

Tapping speeds should be decreased if :-

a) **Lubricant is poor, or flow is restricted**
b) **Bottom lead or Spiral flute taps are used**
c) **Thread depth (%) increases.**
d) **Thread pitch is coarse**
e) **Cutting taper threads (50% normal speed)**
f) **Cutting Acme or Trapezoidal threads (40% normal speed)**

Tapping speeds can be increased if:-

a) **Thread depth decreases**
b) **Thread pitch is fine**
c) **Coolant flow and quality is good**
d) **Spiral point taps are used**

RECOMMENDED FEED SPEEDS AND LUBRICANTS

Speeds given are for machine tapping using HSS taps, and are given in feet per minute

MATERIAL	FEED SPEED (SFM)	LUBRICANT
Aluminium	70-90	Soluble oil
Aluminium alloy	50-70	Soluble, light base or lard oil
Brass	60-100	Light base oil
Bronze	30-40	Light base oil
Copper	60-80	Light base oil
Gun metal	50-60	Soluble, light base or lard oil
Grey cast iron	30-60	Dry or soluble oil
Alloy cast iron	15-30	Sulphur based oil
Malleable iron	20-40	Soluble or sulphur based oil
Magnesium alloy	50-70	Soluble oil or paraffin with lard oil
Nimonic alloy	10-12	Very high pressure cutting oil
Plastics	50-70	Dry, freeze spray, liquid soap
Mild steel	30-50	Sulphur based oil
Carbon steel to 4%	20-40	Sulphur based oil
Carbon steel to 7%	20-30	Sulphur based oil
Carbon steel 7%+	15-25	Sulphur based oil
Steel alloys to 60T	15-25	Sulphur based oil
Steel alloys 60T+	10-15	Sulphur based oil
Stainless steels	10-20	Sulphur based oil
Tool steels	15-25	Sulphur based oil

With so many variables affecting tapping speeds there may need to be some experimentation to find the ideal. A good rule is to start at the slowest speed and work up.

TROUBLESHOOTING

Many factors can affect the quality of a tapped thread. Some more common problems are listed along with probable causes.

POOR THREAD FINISH

Misalignment of tap and work piece
Incorrect feed rate
Chips/swarf not being cleared properly
Tapping device or machine faulty
Insufficient or incorrect lubricant
Incorrectly ground or blunt tap
Wrong tap selection

OVERSIZE/BELL MOUTHED

Misalignment
Incorrect feed rate
Incorrect tapping drill
Tapping device or machine faulty
Insufficient or incorrect lubricant
Incorrectly ground or eccentric tap
Wrong tap selection

EXCESSIVE TAP WEAR

Wrong tap selection
Blunt or incorrectly sharpened tap
Insufficient or incorrect lubricant
Tapping speed too high
Hole work hardened

COLD WELDING

Wrong material composition
Blunt or incorrectly sharpened tap
Insufficient or incorrect lubricant
Tapping speed too high
Material too soft

TAPS BREAKING

Incorrectly sharpened/blunt tap
Tap hits bottom of hole
Machine or tapping device faulty
Wrong tap selection
Incorrect or insufficient lubricant
Tapping speed too high
Hole work hardened
Inefficient chip or swarf removal
Incorrect tapping drill size

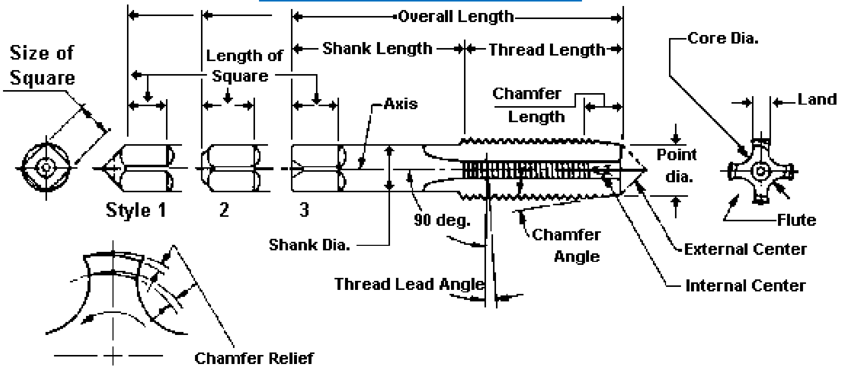
TAP TEETH CHIPPING

Incorrectly sharpened/blunt tap
Tap hits bottom of hole
Machine or tapping device faulty

In order to minimise problems the following rules should be followed:-

1) **Use a pitch controlled tapping attachment**
2) **Choose the correct lubricant**
3) **Use the correct type of tap for the job**
4) **Use the correct tapping drill size**
5) **Choose the correct speeds and feeds**
6) **Keep taps sharp. Regrind with a proper machine**
7) **Ensure accurate alignment**
8) **Check hardness of material, especially when changing batches**
9) **Ensure thread gauging is recently certified – they do wear!**

TAP TERMINOLOGY



CLASSES OF FIT AND TOLERANCES

The various standards for taps give certain allowances or tolerances for manufacture. Taps are generally manufactured to take advantage of these tolerances, so that as they wear their life is extended. The closest tolerances are applied to Class 1 taps, and the most generous to Class 3. Standard, off the shelf taps are manufactured to Class 2, which applies to all taps listed in this catalogue. Classes 1 and 3 can be manufactured to order.

Classes of fit apply to all thread forms:-

CLASS 1 **Produces tapped holes of 4H and 5H fit for Metric threads, 3B fit for American Unified threads and Close fit for Whitworth and BA threads.**
CLASS 2 **Produces tapped holes of 6H, 4G and 5G for Metric threads, 2B for Unified threads and Medium fit for Whitworth and BA threads.**
CLASS 3 **Produces tapped holes of 7H, 8H and 6G for Metric threads, 1B for Unified threads and Free fit for Whitworth and BA threads.**

Information on tolerances and fits for individual threads is available on request.

TAPPING DRILL SIZES

METRIC COARSE AND FINE THREADS

THREAD SIZE	DRILL (mm)	THREAD SIZE	DRILL (mm)	THREAD SIZE	DRILL (mm)
M1x0.25	0.75	**M9x0.75**	8.3	**M18x2**	16
M1.2x0.25	0.95	**M9x1**	8	**M18x2.5**	15.5
M1.4x0.3	1.1	**M9x1.25**	7.8	**M20x1**	19
M1.6x0.35	1.25	**M10x0.5**	9.5	**M20x1.5**	18.5
M1.8x0.35	1.45	**M10x0.75**	9.3	**M20x2**	18
M2x0.4	1.6	**M10x1**	9	**M20x2.5**	17.5
M2.2x0.45	1.75	**M10x1.25**	8.8	**M22x1**	21
M2.5x0.45	2.05	**M10x1.5**	8.5	**M22x1.5**	20.5
M2.6x0.45	2.15	**M11x1**	10	**M22x2**	20
M3x0.35	2.65	**M11x1.25**	9.8	**M22x2.5**	19.5
M3x0.5	2.5	**M11x1.5**	9.5	**M24x1**	23
M3.5x0.35	3.15	**M12x0.5**	11.5	**M24x1.5**	22.5
M3.5x0.6	2.9	**M12x0.75**	11.3	**M24x2**	22
M4x0.5	3.5	**M12x1**	11	**M24x3**	21
M4x0.7	3.3	**M12x1.25**	10.3	**M25x1.5**	23.5
M4.5x0.5	4	**M12x1.5**	10.5	**M25x2**	23
M4.5x0.75	3.75	**M12x1.75**	10.3	**M27x3**	24
M5x0.5	4.5	**M13x1**	12	**M30x3.5**	26.5
M5x0.75	4.25	**M14x1**	13	**M33x3.5**	29.5
M5x0.8	4.2	**M14x1.25**	12.8	**M36x4**	32
M6x0.5	5.5	**M14x1.5**	12.5	**M39x4**	35
M6x0.75	5.25	**M14x2**	12	**M42x4.5**	37.5
M6x1	5	**M15x1**	14	**M45x4.5**	40.5
M7x0.5	6.5	**M15x1.5**	13.5	**M48x5**	43
M7x0.75	6.25	**M16x1**	15	**M52x5**	47
M7x1	6	**M16x1.25**	14.8	**M56x5.5**	50.5
M8x0.5	7.5	**M16x1.5**	14.5	**M60x5.5**	54.5
M8x0.75	7.25	**M16x2**	14	**M64x6**	58
M8x1	7	**M18x1**	17	**M68x6**	62
M8x1.25	6.8	**M18x1.5**	16.5	**M72x6**	66

To calculate tapping drill size for metric threads subtract the pitch from the major diameter,
Select the next larger standard drill size.

UNC & UNF THREADS

UNC	DRILL (mm)	UNF	DRILL (mm)
1.64	1.55	**0.80**	1.25
2.56	1.8	**1.72**	1.55
3.48	2.1	**2.64**	1.85
4.40	2.3	**3.56**	2.1
5.40	2.6	**4.48**	2.4
6.32	2.8	**5.44**	2.7
8.32	3.4	**6.40**	2.9
10.24	3.9	**8.36**	3.5
12.24	4.5	**10.32**	4.1
1/4.20	5.1	**12.28**	4.6
5/16.18	6.6	**1/4.28**	5.5
3/8.16	8	**5/16.24**	6.9
7/16.14	9.4	**3/8.24**	8.5
1/2.13	10.8	**7/16.20**	9.9
9/16.12	12.2	**1/2.20**	11.5
5/8.11	13.6	**9/16.18**	12.9
3/4.10	16.5	**5/8.18**	14.5
7/8.9	19.5	**3/4.16**	17.5
1"8	22.2	**7/8.14**	20.5
1.1/8.7	25	**1"12**	23.2
1.1/4.7	28.2	**1.1/8.12**	26.5
1.3/8.6	30.8	**1.1/4.12**	29.5
1.1/2.6	34	**1.3/8.12**	32.8
1.3/4.5	39.5	**1.1/2.12**	36
2"4.1/2	45.3		

BSW & BSF THREADS

BSW	DRILL(mm)	BSF	DRILL(mm)
1/8.40	2.54	**3/16.32**	3.97
3/16.24	3.7	**1/4.26**	5.37
1/4.20	5.08	**5/16.22**	6.8
5/16.18	6.5	**3/8.20**	8.3
3/8.16	7.9	**7/16.18**	9.7
7/16.14	9.3	**1/2.16**	11.1
1/2.12	10.6	**9/16.16**	12.7
9/16.12	12.2	**5/8.14**	14.1
5/8.11	13.6	**11/16.14**	15.6
3/4.10	16.5	**3/4.12**	16.9
13/16.10	18.1	**13/16.12**	18.5
7/8.9	19.4	**7/8.11**	19.9
15/16.9	21	**1!10**	22.9
1"8	22.2	**1.1/8.9**	25.8
1.1/8.7	24.9	**1.1/4.9**	28.9
1.1/4.7	28.1	**1.3/8.8**	31.8
1.3/8.6	30.7	**1.1/2.8**	34.9
1.1/2.6	33.9	**1.5/8.8**	38.1
1.5/8.5	36.2	**1.3/4.7**	40.8
1.3/4.5	39.4	**1.7/8.7**	44
1.7/8.4.1/2	42	**2"7**	47.2
2"4.1/2	45.2		

PIPE THREADS

BSP	DRILL (mm)	BSPT	DRILL (mm)	NPT	DRILL (mm)	NPTF	DRILL (mm)	PG THREAD	DRILL (mm)
1/16.28	6.8	**1/16.28**	6.2	**1/16.27**	6.3	**1/16.27**	6.3	**7**	11.3
1/8.28	8.8	**1/8.28**	8.2	**1/8.27**	8.5	**1/8.27**	8.4	**9**	13.9
1/4.19	11.8	**1/4.19**	10.9	**1/4.18**	11	**1/4.18**	10.9	**11**	17.3
3/8.19	15.3	**3/8.19**	14.4	**3/8.18**	14.5	**3/8.18**	14.25	**13.5**	19
1/2.14	19.1	**1/2.14**	18	**1/2.14**	18	**1/2.14**	17.75	**16**	21.2
5/8.14	21.1	**3/4.14**	23	**3/4.14**	23	**3/4.14**	23	**21**	27
3/4.14	24.6	**1”11**	29	**1”11.5**	29	**1”11.5**	29	**29**	35.5
7/8.14	28.4	**1.1/4”11**	38	**1.1/4”11.5**	38	**1.1/4”11.5**	37.8		
1”11	30.9	**1.1/2”11**	44	**1.1/2”11.5**	44	**1.1/2”11.5**	43.8		
1.1/8.11	35.6	**2”11**	55	**2”11.5**	56	**2”11.5**	55.8		
1.1/4.11	39.6	**2.1/2”11**	71						
1.3/8.11	42	**3”11**	83						
1.1/2.11	45.5								
1.3/4.11	51.4								
2”11	57.3								
2.1/4”11	63.4								
2.1/2”11	72.9								
2.3/4”11	79.2								
3”11	85.6								

TAPPING DRILLS FOR FLUTELESS TAPS

METRIC	DRILL (mm)	UNC	DRILL (mm)	UNF	DRILL (mm)
1.6x0.35	1.45	**1.64**	1.67	**0.80**	1.38
2x0.4	1.8	**2.56**	1.98	**1.72**	1.69
2.5x0.45	2.3	**3.48**	2.27	**2.64**	2
3x0.5	2.8	**4.40**	2.56	**3.56**	2.31
3.5x0.6	3.2	**5.40**	2.89	**4.48**	2.6
4x0.7	3.7	**6.32**	3.14	**5.44**	2.91
4.5x0.75	4.2	**8.32**	3.8	**6.40**	3.22
5x0.8	4.6	**10.24**	4.34	**8.36**	3.84
6x1	5.6	**12.24**	5.01	**10.32**	4.47
8x1.25	7.4	**1/4.20**	5.77	**1/4.28**	5.94
10x1.5	9.3	**5/16.18**	7.3	**5/16.24**	7.5
12x1.75	11.2	**3/8.16**	8.8	**3/8.24**	9
14x2	13.1	**7/16.14**	10.3	**7/16.20**	10.5
16x2	15.1	**1/2.13**	11.8	**1/2.20**	12.1

BA THREADS

THREAD	DRILL (mm)
0	5
1	4.4
2	3.9
3	3.4
4	2.95
5	2.6
6	2.25
7	2
8	1.75
9	1.5
10	1.35

Calculating tapping drill sizes.

As a rule of thumb, for Metric threads, the tapping drill can be calculated by subtracting the pitch from the diameter of the thread.
Eg for an M5x0.75 thread the tapping drill is 5-0.75mm = 4.25mm.

For Imperial (inch) threads the tapping drill size is calculated in the same way, diameter minus pitch.
Eg for 3/4.10 UNC pitch = 0.1, diameter = 0.75, tapping drill = 0.65 = 16.5mm.