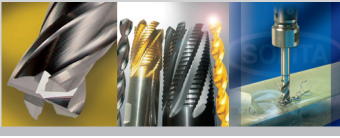
SOMTA

World Class Cutting Tools

S O M T A U S E R G U I D E

Technical Assistance



SOMTA TECHNICAL SERVICES

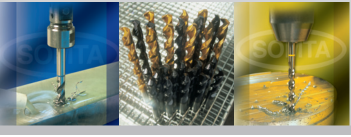
This handbook is intended to help you get maximum performance from SOMTA cutting tools. Whilst the information covers most common uses and problems it is not possible to deal with every situation. Our trained sales representatives are available to further assist and advise, fully backed up by factory technical services.

FULL SPECIFICATIONS IN SOMTA CATALOGUES

SOMTA TOOLS (PTY) LTD is a world class manufacturer producing precision cutting tools to international standards and specifications which include British Standard, DIN, ISO, ANSI and JIS. Full details of specifications are listed in our catalogues which are available from leading Industrial Distributors or directly from the Somta factory.

PRODUCT RANGE STANDARDS & SPECIALS

The SOMTA range consists of nearly 25 000 standard items and we have a cutting tool available for almost every application. Sometimes a special tool is needed and our product engineers at the SOMTA factory can design a special purpose tool to do the job. These can also be manufactured to customers' specifications or to a sample.



Introduction

Manufacturers & Suppliers of Drills, Reamers, End Mills, Bore Cutters, Taps & Dies, Toolbits, Custom Tools and Surface Coatings

Profile

SOMTA TOOLS specialises in the design and manufacture of drills, reamers, milling cutters, toolbits, threading tools and custom tools for the industrial and "DIY"markets, offering the innovative range of Balzers BALINIT® high performance coatings on all cutting tools.

SOMTA TOOLS vision as a world class provider of cutting tools, is supported by ISO 9001 accreditation.

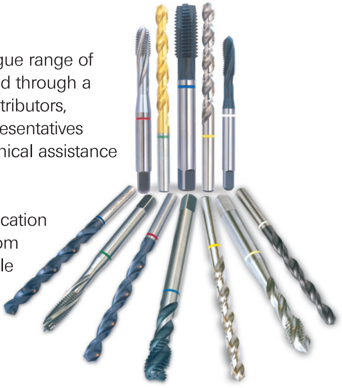
The Company

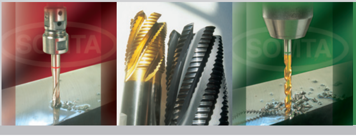
SOMTA TOOLS factory in Pietermaritzburg manufactures 25 000 standard items and a further 6 000 made-to-order items to serve the domestic and export markets in over 60 countries worldwide.

The Product

SOMTA TOOLS standard catalogue range of precision cutting tools are supplied through a extensive network of industrial distributors, backed up by technical sales representatives who are available to provide technical assistance where required.

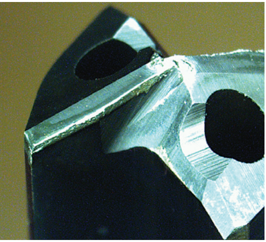
Should a special production application requiring specially designed custom tooling be requested, Somta is able to provide a full technical design service.



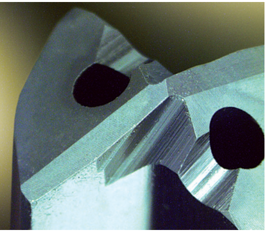


Manufacturers of precision industrial and
DIY coated cutting tools.

Standard and custom designed tooling



AN EXAMPLE OF BEFORE AND AFTER REGRINDING

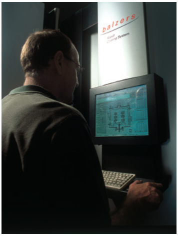


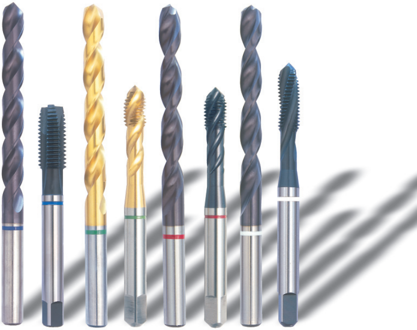
The major functions the end user, apprentice or student can expect from Somta's technical representatives and design departments are as follows:

- Provision of technical literature to assist with the correct application of Somta's tooling.
- Assistance with and solution of any tooling design and application relating to Somta's custom tooling.
- Advice, suggestions and recommendations on product improvement or innovation. Provision of training in the use, care and resharpening of high speed steel cutting tools on request.
- Liaison between the customer and Somta's factory estimator and production staff, to deliver the required tooling on time according to specification, to the customers complete satisfaction.

Contact our technical department on **Tel: +27 33 355 6600** or **e-mail: tech@somta.co.za** for technical assistance if experiencing a cutting tool problem.

State of the art . . .



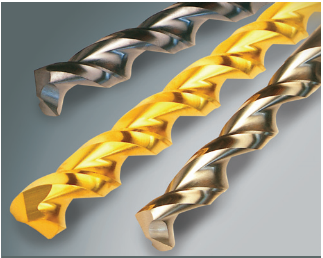


Somta has integrated a **state of the art Balzers PVD Rapid Coating System** into it's manufacturing programme, offering the innovative range of **Balzers BALINIT®** high performance coatings on all cutting tools.

Application
products

Colour Band Drills & Taps

A premium range of colour coded drills & taps, each designed with different geometry for specific material groups with enhanced efficiency over general purpose drills & taps. Major efficiency improvements are experienced in machining of high tensile, tough and ductile materials.

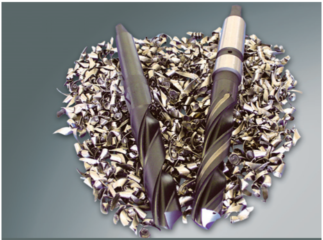


UD Parabolic Flute Drills

A comprehensive range of heavy duty drills designed with improved point and flute geometries for enhanced penetration and chip removal in long chip forming, short chip forming and abrasive material groups. This range of Parabolic Flute Ultra Drills are designed to meet the challenges of a broad spectrum of difficult drilling applications.

Roughing End Mills

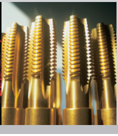
An efficient range of roughing end mills, designed for rapid stock removal and superior finish in a single operation. The flat crest provides a better finish than conventional knuckle form cutters, while the centre cutting end teeth up to 20mm diameter provide plunge and cavity cutting capabilities, ideal for CNC machining applications.



Chipbreaker Drills

This outstanding development increases drill cutting efficiency by means of chip control. As the chipbreaker rib extends along the full length of the flutes, the chipbreaker form is maintained throughout the drill life irrespective of the number of resharpenings. The cutting action of the Chipbreaker Drill is the same as a standard drill, the cut being effected by the full length of the cutting edge, with the chip down the face of the flutes. Contact with the chipbreaker rib then curls and breaks the chip into short lengths. Drill resharpening is the same as with standard drills.





Standard

products

Straight Shank Drills

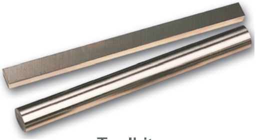
End Mills and Shank Cutters

Threading Tools

Morse Taper Shank Drills

Reamers, Countersinks & Counterbores

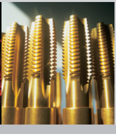
Bore Cutters



Toolbits

Contents





Contents

IF YOU CANNOT FIND AN ANSWER TO YOUR PROBLEM IN THIS BOOKLET PLEASE CONTACT THE SOMTA FACTORY.

CUTTING TOOL MATERIALS

Somta cutting tools are manufactured from the finest steel available. The heat treatment process is controlled by our Metallurgical laboratory using advanced computerised and electronic instrumentation. High Speed Steel contains various elements such as Molybdenum, Tungsten, Cobalt and Vanadium and must be specially heat treated to produce the ideal combination of strength, toughness and wear resistance. SOMTA products are manufactured from one of the following High Speed Steels depending on the product and application.

	C	Cr	W	Mo	V	Co	Hardness (HRC)
M2	0.9	4	6	5	2	-	63 - 65
M35	0.9	4	6	5	2	5	64 - 66
M42	1.1	4	1.5	9.5	1	8	66 - 68.5(70)
M9V	1.25	4.2	3.5	8.5	3	-	64 - 66

M2 is the standard High Speed Steel and is used where toughness is important, together with a good standard of wear resistance and red hardness.

M35 is a development of M2 and contains 5% cobalt which gives improved hardness, wear resistance and red hardness. It may be used when cutting higher strength materials.

M42 can be heat treated to very high hardness levels of up to 70 HRC (1 000 HV) although normally a slightly lower figure will be employed to retain toughness. This steel is ideal for machining higher strength materials and work hardening alloys such as stainless steels, nimonic alloys etc. Despite its high hardness, M42 has good grindability characteristics due to lower vanadium content.

M9V material is mainly used in the manufacture of machine taps because of its good wear resistance, good grinding capabilities, high hardness and excellent toughness.

Cutting tools may shatter
eye protection should be worn

SURFACE TREATMENTS

Bright Finish

A bright finish tool has no surface treatment and is suitable for general purpose use.

Blue Finish

A blue finish is achieved by steam tempering - a thermal process which imparts a non-metallic surface to the tool. This surface is porous and by absorbing lubricant, helps prevent rusting, reduces friction and cold welding, resulting in increased tool life.
Steam tempered products can successfully be used at slightly increased machining rates or on more difficult to machine materials.

Gold Oxide

This is a metallic brown coloured surface treatment achieved by a low temperature temper and is normally only used on cobalt products for identification purposes.

Nitriding

Nitriding imparts a hard surface to the tool and is used for prolonging tool life and machining difficult to machine materials. Because nitriding makes the edge more brittle, care must be exercised in the type of application.
Nitrided tools are normally also steam tempered.

Titanium Nitride Coating (TiN)

TiN coating is a very hard, gold coloured surface coating a few microns thick which is applied by means of a complex process, called Physical Vapour Deposition (PVD), by advanced modern equipment. The coating is non-metallic and therefore reduces cold welding.
In certain applications increased speed and feed rates can be achieved because of:

(a) The hardness of the coating.
(b) The reduction in cutting force required due to a decrease in friction between the tool and the workpiece.

Tool performance will deteriorate after re-sharpening.

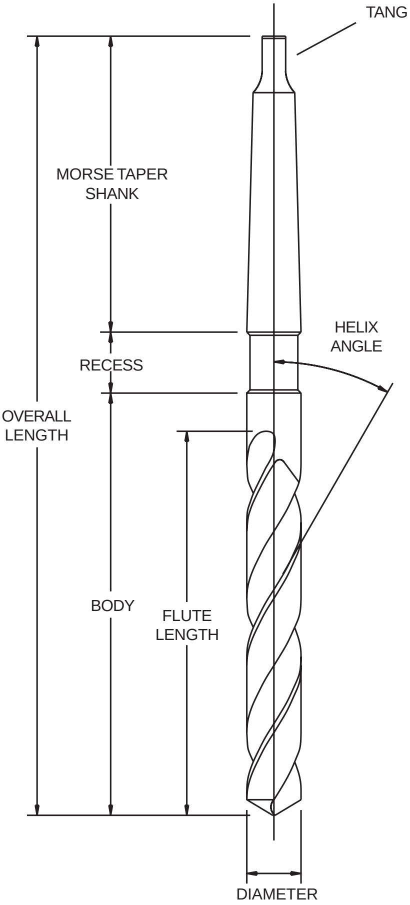
TiCN (Titanium Carbonitride)

The addition of carbon to TiN results in a significant increase in the hardness of TiCN over TiN. TiCN also has a much lower coefficient of friction which enhances the surface finish of components machined with TiCN coated tools, higher productivity can be achieved on a wide range of materials but, in particular stainless steel, titanium and nickel based alloys. It is now generally accepted that TiCN coating has been superseded by TiAlN for most machining applications.

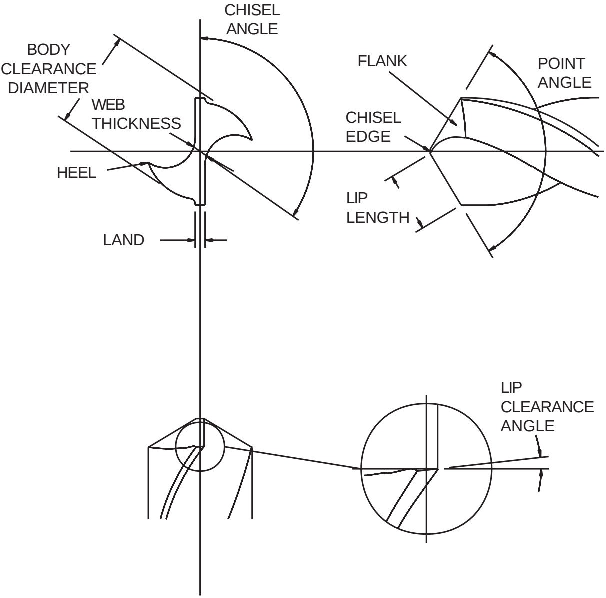
TiAlN (Titanium Aluminium Nitride)

In addition to a higher hardness than both TiN and TiCN the aluminium in the coating imparts a much greater oxidation stability. This is as a result of a very thin film of (Aluminium Oxide) being formed on the surface of the TiAlN. The film is self repairing, leading to additional increased service life. These improvements allow the coating to withstand much higher temperatures which in turn allows increased cutting conditions, especially useful when machining Cast Iron and tough steels.

DRILL NOMENCLATURE



DRILL NOMENCLATURE



DETAILED VIEW OF LIP CLEARANCE

SELECTING THE CORRECT DRILL

Drills for general use

These twist drills are designed to drill the common materials under normal operating conditions.
The following standard drills are available ex-stock from Somta.

Jobber Drills

General purpose drilling.

Long Series Drills

General purpose long reach drilling.

Stub Drills

A short robust drill suited to portable drill applications.

Reduced Shank Drills 1/2" / 12,7 mm Shank

General drilling for use in hand power tools.

Left hand Jobber Drills

General purpose drilling in the left hand direction.

MTS Drills

General purpose drilling.

MTS Drills, HSS-Co

General purpose drilling in difficult materials.

Drills for specific applications

More efficient drilling can be achieved by using a drill designed for a specific application.
The following drills are available ex-stock from Somta.

Sheet Metal Drills

Self centring drill designed to produce accurate holes in thin materials. 135° split point.

Double Ended Sheet Metal Drills

Self centring drill designed to produce accurate holes in thin materials. 135° split point.

Tanged Jobber Drills

Designed to fit tang drive sleeve.

NDX - Heavy Duty Jobber Drills, HSS-Co

Drilling high tensile steels and other difficult materials. 135° split point.

TiN Coated Jobber Drills

For drilling in a production environment where higher speeds and or feeds are required.

Extra Length Drills

Extra deep hole drilling. (Details on page 18)

UDL Deep Hole Drills Long Series, HSS-Co

Parabolic Flute Form and Heavy Duty, for general purpose long reach drilling.

UDL Deep Hole Drills Extra Length, HSS-Co

Parabolic Flute Form and Heavy Duty, for extra deep hole drilling.

UDL Jobber Drills, Long Chip, HSS-Co

Parabolic Flute Form and Heavy Duty, for use on NC and CNC machines where high productivity and accurate holes are required.

UDL - Stub Drills, Long Chip, HSS-Co

Parabolic Flute Form and Heavy Duty, for use on NC and CNC machines where high productivity and accurate holes are required.

UDS - Jobber Drills, Short Chip, HSS-Co

Parabolic Flute Form and Heavy Duty, for use on NC and CNC machines where high productivity and accurate holes are required.

UDC - Jobber Drills, Cast Iron, HSS-Co

Parabolic Flute Form and Heavy Duty, for use on NC and CNC machines where high productivity and accurate holes are required.

MTS Extra Length Drill

Extra deep hole drilling. (Details on page 18 & 19)

MTS Core Drills

For enlarging diameters of existing holes whether drilled, punched or cored. (Details on page 21)

MTS Armour Piercing Drills, HSS-Co

Heavy duty drilling in work hardening and heat treated steels.

Drills for Special Applications

Subland Drills

For drilling stepped holes in one operation.

Drill Reamers

For drilling and reaming holes in one operation (hole tolerance wider than H7).

Coolant Feed Drills

For drilling extra deep holes. (Details on page 19)

Cotter Pin Drills

For heavy duty drilling using a tang drive sleeve.

Somta Sorger

Patented Auger for wood drilling

Rail Drills

Developed for drilling work hardening railway lines. Alternative to the Armour piercing drill.

DRILL TECHNICAL DATA

WORKPIECE MATERIAL		TYPICAL PHYSICAL PROPERTIES			CODE TYPE	
TYPE	GRADE	HARDNESS BRINELL (MAX)	TONS PER SQ.INCH (MAX)	N / mm² (MAX)	STUB DRILLS	JOBBER DRILLS
CARBON STEEL & ALLOY STEEL	FREE CUTTING 0.3 to 0.4% Carbon	150 170	35 40	540 620	141 - 142 147 - 148 151 - 152 153● - 163●	101 - 102 105 154● - 164● 155 - 161 167 - 177
	0.3 to 0.4% Carbon 0.4 to 0.7% Carbon	248 206	59 47	910 720		
	0.4 to 0.7% Carbon	286	67	1030		
	Low Alloy Tool	248	59	910		
	Steels High Alloy Tool Steels Heat Treatable Steels	330 380	75 87	1150 1300		
STAINLESS STEEL	Martensitic (400 Series)	248	54	810	AS ABOVE	AS ABOVE
	Austenitic (Work Hardening) (300 Series)	300	65	1000		
HEAT RESISTING ALLOYS	Inconell, Hastelloy Nimonic Alloys	350	78	1200	AS ABOVE	AS ABOVE
TITANIUM	Commercially Pure	275	65	1000	AS ABOVE	AS ABOVE
	Commercially Alloyed	350	78	1200		155●
CAST IRONS	Grey Irons Nodular Irons Malleable Irons	110 - 300	-	-	AS ABOVE	AS ABOVE 156●
MANGANESE STEEL		AS SUPPLIED			AS ABOVE	AS ABOVE
ALUMINIUM MAGNESIUM ALLOYS	Wrought Alloys Cast Alloys Silicon Alloys	AS SUPPLIED			AS ABOVE	156● AS ABOVE
COPPER ALLOYS	Free Cutting Alloys	LEADED COPPER ALLOYS FREE CUTTING BRASS MEDIUM TO HIGH LEADED BRASS			AS ABOVE	AS ABOVE
	Moderately Machineable Alloys	LOW TO HIGH SILICON BRONZE MANGANESE BRONZE ALUMINIUM SILICON BRONZE			AS ABOVE	AS ABOVE 155●
	Difficult to Machine Alloys	COMMERCIAL BRONZE 90% PHOSPHOR BRONZE 5 - 10% ALUMINIUM BRONZE				
PLASTICS	Soft Hard Reinforced	AS SUPPLIED			AS ABOVE	AS ABOVE 156●

DRILL TECHNICAL DATA *(cont.)*

CODE TYPE ● DENOTES RECOMMENDED				COOLANT	SPEED METRES / MIN	FEED CURVE See Page 14
LONG SERIES	EXTRA LENGTH	MORSE TAPER STANDARD	MORSE TAPER E/LENGTH			
116 - 117 109● - 110●	118● - 119● 120● - 121 122 - 123 124 - 125 126	201 - 202 203 - 204 205 - 206 208●	241 - 242 244 - 245 251 - 252 254 - 255	SOLUBLE OIL OR SEMI-SYNTHETIC OIL	25 - 30 15 - 20 10 - 15	H
				SOLUBLE OIL	15 - 24	F
				SOLUBLE OIL EXTREME PRESSURE	10 - 15 4 - 8	H
AS ABOVE	AS ABOVE	AS ABOVE 261● 279●	AS ABOVE	SOLUBLE OIL EXTREME PRESSURE OR SULPHO-CHLORINATED	12 - 16 6	H C
AS ABOVE	AS ABOVE	AS ABOVE 261● 279●	AS ABOVE	SOLUBLE OIL EXTREME PRESSURE OR SULPHO-CHLORINATED	5 - 10	E
AS ABOVE	AS ABOVE	AS ABOVE 261● 279●	AS ABOVE	SOLUBLE OIL SULPHO-CHLORINATED EXTREME PRESSURE CHLORINATED OIL	15 - 25 7 - 11	F C
AS ABOVE	AS ABOVE	AS ABOVE	AS ABOVE	DRY OR DETERGENT WATER - SOLUBLE EMULSION	25 - 35 15 - 30 25 - 30	K
AS ABOVE	AS ABOVE	AS ABOVE 261● 279●	AS ABOVE	DRY OR NEAT E.P. OIL	4 - 6	C
AS ABOVE	AS ABOVE	AS ABOVE	AS ABOVE	SOLUBLE OIL (1 : 25)	Up to 45 30 - 35	L
				LOW VISCOSITY MINERAL OIL	40 - 100	L
AS ABOVE	AS ABOVE	AS ABOVE	AS ABOVE	SOLUBLE OIL (1 : 20)	40 - 50	M
				SOLUBLE OIL (1 : 20) LIGHT MINERAL OIL	30 - 36 15 - 20	L
				DRY OR SOLUBLE OIL	25 - 30 < 20	-

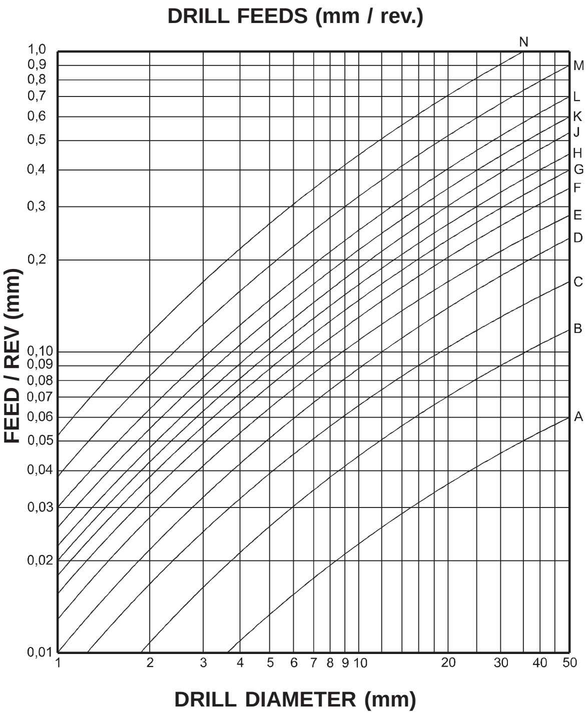
UD DRILL TECHNICAL DATA

MATERIAL TYPES		HARDNESS HB	TENSILE STRENGTH N/mm²
Steel	Free Cutting steels	≤120	≤ 400
	Structural steel. Case carburizing steel	≤200	≤ 700
	Plain carbon steel	≤250	≤ 850
	Alloy steel	>250	≤ 850
	Alloy steel. Hardened and tempered steel	>250 ≤350	> 850 ≤1200
	Alloy steel. Hardened and tempered steel	>350	>1200
Stainless Steel	Free machining Stainless steel	≤250	≤ 850
	Austenitic	≤250	≤ 850
	Ferritic + Austenitic, Ferritic, Martensitic	≤300	≤1000
Cast Iron	Lamellar graphite	≤150	≤ 500
	Lamellar graphite	>150 ≤300	> 500 ≤1000
	Nodular graphite, Malleable Cast Iron	≤200	≤ 700
	Nodular graphite Malleable Cast Iron	>200 ≤300	> 700 ≤1000
Titanium	Titanium, unalloyed	≤200	≤ 700
	Titanium, alloyed	≤270	≤ 900
	Titanium alloyed	> 270 ≤ 350	> 900 ≤1200
Nickel	Nickel, unalloyed	≤150	≤ 500
	Nickel, alloyed	≤270	≤ 900
	Nickel, alloyed	>270 ≤350	> 900 ≤1200
Copper	Copper	≤100	≤ 350
	Beta Brass, Bronze	≤200	≤ 700
	Alpha Brass	≤200	≤ 700
	High strength Bronze	≤470	≤1500
Aluminium Magnesium	Al, Mg, unalloyed	≤100	≤ 350
	Al alloyed Si < 0.5%	≤150	≤ 500
	Al alloyed, Si > 0.5% < 10%	≤120	≤ 400
	Al alloyed, Si > 10% Al-alloys, Mg-alloys	≤120	≤ 400
Synthetic Materials	Thermoplastics	-	-
	Thermosetting plastics	-	-
	Reinforced plastic materials	-	-

UD DRILL TECHNICAL DATA *(cont.)*

NORMAL CHIP FORM	DRILL TYPE & SURFACE TREATMENT	SURFACE SPEED METRES PER MINUTE	FEED CURVE see Page 14
extra long	UDL TiN	35 - 45 50 - 70	H J
middle/long	UDL TiN	25 - 35 40 - 50	H J
long	UDL TiN TiCN TiAlN	25 - 30 35 - 40	G I
long	UDL TiN TiCN TiAlN	25 - 30 35 - 40	G I
long	UDL TiN TiCN TiAlN	15 - 20 25 - 30	E G
long	UDL TiN TiCN TiAlN	15 - 20 20 - 25	E G
middle	UDL TiN TiCN TiAlN	18 - 21 27 - 32	E G
long	UDL TiN TiCN TiAlN	8 - 10 12 - 15	K M
long	UDL TiN TiCN TiAlN	10 - 15 16 - 22	E G
extra short	UDC TiAlN	30 - 35 45 - 55	G I
extra short	UDC TiAlN	25 - 30 35 - 45	G I
middle/short	UDC TiAlN	18 - 21 25 - 35	E G
middle/short	UDC TiAlN	12 - 17 22 - 26	E G
extra long	UDL TiCN	20 - 25 30 - 35	E G
middle/short	UDS TiCN	13 - 17 20 - 25	E G
middle/short	UDS TiCN	5 - 6 7 - 11	C E
extra long	UDL TiCN TiAlN	12 - 16 20 - 25	G I
long	UDL TiCN TiAlN	6 - 8 10 - 12	G I
long	UDL TiCN TiAlN	5 - 6 10 - 12	C E
extra long	UDL TiN	55 - 65 80 - 95	L N
middle/short	UDS TiN	60 - 70 90 - 105	L N
long	UDL TiN	30 - 40 45 - 50	L N
short	UDS TiN	27 - 33 40 - 50	K M
extra long	UDL TiN	75 - 85 110 - 125	N N
middle	UDL TiN	65 - 75 100 - 115	N N
middle/short	UDS TiN	55 - 65 80 - 100	L N
short	UDS TiN	27 - 33 40 - 50	K M
extra long	UDL TiN	75 - 85 110 - 125	L N
short	UDS TiN	55 - 65 80 - 100	J L
extra short	UDC TiN	15 - 20 20 - 30	J L

DRILL FEED CURVE CHART



HOW TO USE THE DRILL FEED CHART

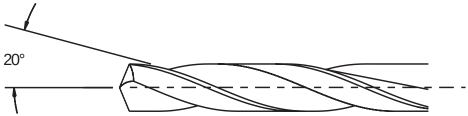
1. Locate Feed Curve (as given in the application data pages 11 & 13) on the right hand side of the drill feed chart.
2. Locate Drill Diameter along bottom axis of chart.
3. Determine point of intersection of Feed Curve and Drill Diameter.
4. Project horizontally from point of intersection to left hand side of chart and read off nearest FEED / REV (mm).
5. Select nearest feed on drilling machine within ± 20% of chart figure.

General Drilling Feeds (mm per revolution)

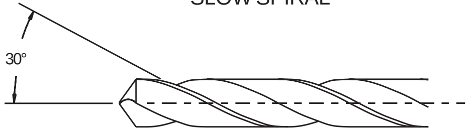
Drill Diameter Range (mm)	Feed Range	Drill Diameter Range (mm)	Feed Range
1 - 3	0.03 to 0.075	16 - 20	0.25 to 0.53
3 - 5	0.05 to 0.18	20 - 25	0.28 to 0.56
5 - 8	0.10 to 0.28	25 - 30	0.30 to 0.60
8 - 12	0.15 to 0.35	30 - 40	0.35 to 0.68
12 - 16	0.20 to 0.45	Over 40	0.40 to 0.75

When setting to drill material of unknown machinability the slowest speed and lightest feed should be used and these should be gradually increased until optimum output per regrind is obtained.

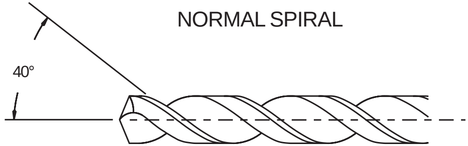
HELIX ANGLE OR SPIRAL



SLOW SPIRAL



NORMAL SPIRAL



QUICK SPIRAL

PERIPHERAL SPEED

METRES PER MIN	5	10	20	30	40
Drill Dia. mm	Revolutions				
1.0	1591	3182	6364	9546	12728
2.0	795	1590	3182	4770	6360
3.0	530	1060	2120	3180	4240
4.0	398	795	1590	2385	3180
5.0	318	636	1272	1908	2544
6.0	265	530	1060	1590	2120
7.0	227	455	910	1365	1820
8.0	199	398	796	1194	1592
9.0	177	353	706	1059	1412
10.0	159	318	636	954	1272
11.0	145	289	578	867	1156
12.0	133	265	530	795	1060
13.0	122	245	490	735	980
14.0	114	227	454	681	908
15.0	106	212	424	636	848
16.0	100	199	398	597	796
18.0	89	177	354	531	708
20.0	80	159	318	477	636
22.0	73	145	290	435	580
24.0	67	133	266	399	532
26.0	61	122	344	366	488
28.0	57	144	228	342	456
30.0	53	106	212	318	424
35.0	45	91	182	273	364
40.0	40	80	160	240	320
45.0	35	70	140	210	280
50.0	32	64	128	192	256
63.0	25	50	100	150	200
75.0	21	42	84	126	168
100.0	16	32	64	96	128

TO rpm CONVERSION CHART

50	60	70	80	90	100

per Minute

15910	19092	22274	25456	28638	31820
7950	9540	11130	12720	14310	15900
5300	6360	7420	8480	9540	10600
3975	4770	5565	6360	7155	7950
3180	3816	4452	5088	5724	6360
2650	3180	3710	4240	4770	5300
2275	2730	3185	3640	4095	4550
1990	2388	2786	3184	3582	3980
1765	2118	2471	2824	3177	3530
1590	1908	2226	2544	2862	3180
1445	1734	2023	2312	2601	2890
1325	1590	1855	2120	2385	2650
1225	1470	1715	1960	2205	2450
1135	1362	1589	1816	2043	2270
1060	1272	1484	1696	1908	2120
995	1194	1393	1592	1791	1990
885	1062	1239	1416	1593	1770
795	954	1113	1272	1431	1590
725	870	1015	1160	1305	1450
665	798	931	1064	1197	1330
610	732	854	976	1098	1220
570	684	798	912	1026	1140
530	636	742	848	954	1060
455	546	637	728	819	910
400	480	560	640	720	800
350	420	490	560	630	700
320	384	448	512	576	640
250	300	350	400	450	500
210	252	294	336	378	420
160	192	224	256	288	320

THE CORRECT USE OF DRILLS

A guide to successful drilling

- Make sure the workpiece is securely held and supported. Should it bend or move, it could cause the drill to break.
- Use a good socket and thoroughly clean both the socket and the taper shank of the drill. Do not use steel objects to seat the drill.
- Straight shank drill chucks must be able to hold the drill securely.
- Keep the drill sharp. Do not allow it to become blunt as it will require extra-grinding to get it sharp again.
- Direct an adequate supply of the recommended coolant to the point of the drill. (see page 11).
- Do not allow chips to clog the drill flutes.
- When re-sharpening take care to achieve the correct point geometry (see page 22/24) and do not overheat the drill when grinding.
- Use core drills for enlarging existing holes - 2 flute drills are not designed for this purpose.
- Use the correct drill to suit the application (see page 6-9).

Deep Hole Drilling

A general guide

A hole deeper than 3 times its diameter is considered a “deep hole”. Deep holes are successfully drilled by reducing speed and feed rates, as shown in the table on page 11. Care must be taken not to clog the flutes with chips. In very deep holes it may be necessary to withdraw the drill frequently to clear the flutes. Extra length drills should be used with a guide bush as close to the workpiece as possible to support the drill.

Recommended Speeds for Deep Holes

Depth of Hole	% Speed Reduction
3 X Drill Diameter	10%
4 X Drill Diameter	20%
5 x Drill Diameter	30%
More than 6 X Drill Diameter	40%

Recommended Feeds for Deep Holes

Depth of Hole	% Feed Reduction
3 to 4 X Drill Diameter	10%
5 to 8 X Drill Diameter	20%

Extra Length "Deep Hole" Drills (UDL Form)

The SOMTA "Deep Hole" drill has a specially shaped flute form, commonly known as Parabolic, which gives rigidity for deep hole drilling and improves chip flow, enabling the full depth of the hole to be drilled without withdrawal.

These drills are of special robust design for use on tougher materials such as steels and cast irons with hardness up to 1000 N/mm^2. Similar drills for softer materials such as aluminium, mild steel etc. with hardness up to 500 N/mm^2 are available on special request.

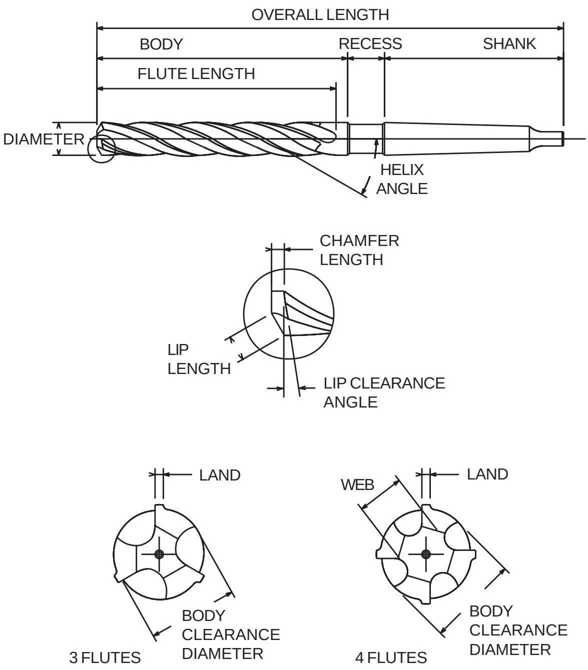
Coolant Feed Drills

Higher production rates can be achieved when deep hole drilling by using coolant feed drills.

Harmful heat generation at the drill point is prevented by the supply of coolant to the cutting face. This allows higher speeds and feeds and improved chip flow, thus eliminating the need to clear the flutes by withdrawal.

Core Drilling

Core Drill Nomenclature



Core Drills

Cutting Diameter Tolerance on Core Drills

Core Drill Diameter (mm)		Diameter Tolerance (mm)	
Above	Up to	Plus	Minus
-	6	+ 0	- 0,018
6	10	+ 0	- 0,022
10	18	+ 0	- 0,027
18	30	+ 0	- 0,033
30	50	+ 0	- 0,039

A Guide to Core Drilling

Core drills are only used for enlarging diameters of existing holes whether drilled, punched or cored. Having no point, the drill is only able to cut on the chamfer. The maximum amount of material that can be removed is restricted by the chamfer root diameter to 60% of the core drill diameter.
Because of its multi-flute construction the core drill gives better hole size and surface finish than a two flute drill. Two flute drills should not be used to enlarge existing holes as they will tend to chip and break.

Speed and Feed rates for Core Drills

Speed - As for 2 flute drills

Feed - **3 Flute**

1 to 1,5 X 2 flute drill feed rate

4 Flute

1,5 to 2 X 2 flute drill feed rate

Cutting diameter tolerance

SOMTA Twist Drills are manufactured to h8 tolerance.

2 Flute Drills

Cutting Diameter Tolerance on Twist Drills

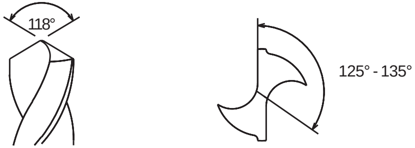
Drill Diameter (mm)		Diameter Tolerance (mm)	
Above	Up to	Plus	Minus
-	3	+ 0	-0,014
3	6	+ 0	-0,018
6	10	+ 0	-0,022
10	18	+ 0	-0,027
18	30	+ 0	-0,033
30	50	+ 0	-0,039
50	80	+ 0	-0,046

Back Taper on Fluted Portion

The drill diameter is normally reduced over the fluted portion to prevent jamming. The amount of back taper is a maximum of: 0,08 mm on diameter per 100 mm length.

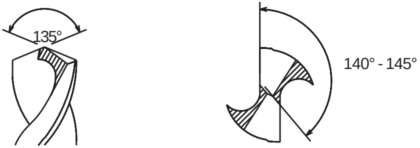
Back taper is usually only applied to sizes over 6 mm.

Standard Point



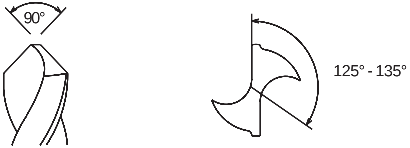
This point is suitable for general purpose drilling.

Split Point



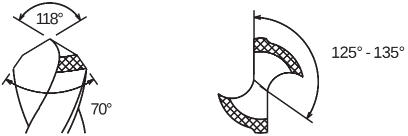
The split point minimises end thrust and is self centering.

Long Point



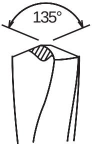
Used for wood, plastic, hard rubber, fibres etc.

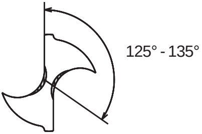
Cast Iron Point ("DX" Point)



The secondary angle reduces wear on the outer corners.

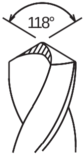
Heavy Duty Notched Point

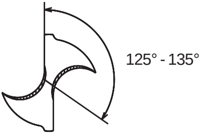




The notched point reduces end thrust and optimises centre cutting efficiency with chisel strength. It is recommended for hard and high strength materials.

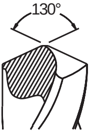
Web Thinned Point

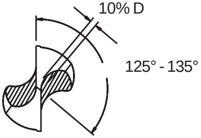




The web thinned point reduces end thrust and improves centre cutting efficiency.

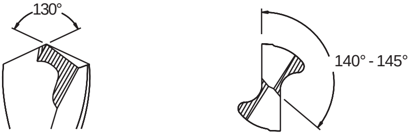
"UX" Point





The 130° special notched "UX" point style provides self centering, easier penetration, improved hole accuracy and improved load distribution. This special notch geometry gives a corrected rake angle of 15° which provides strong point for harder materials, as well as preventing snatching with materials such as Aluminium, Brass, Bronze and Plastics. Available on UDL and UDS drills.

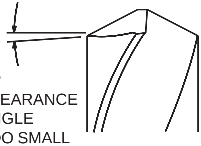
Part Split Point



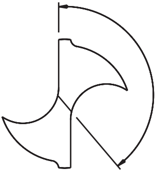
The 130° part split point is similar to the conventional split point. The part split point has a wider chisel edge. Provides easy penetration, self centering and optimises centre cutting efficiency with chisel strength.

Common Re-Sharpening Errors on Standard Drill Points

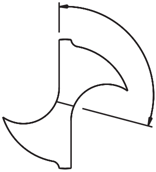
LIP CLEARANCE ANGLE TOO GREAT



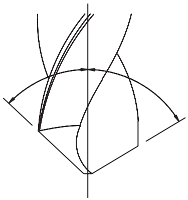
LIP CLEARANCE ANGLE TOO SMALL



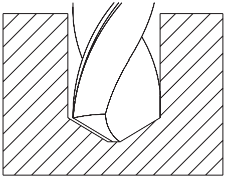
CHISEL ANGLE TOO GREAT



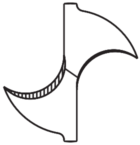
CHISEL ANGLE TOO SMALL



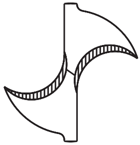
LIPS UNEQUAL LENGTH (DIFFERENCE IN RELATIVE LIP HEIGHT.)



DIFFERENCE IN RELATIVE LIP HEIGHT WILL DRILL AN OVERSIZE HOLE.



UNEVEN WEB THINNING



WEB THINNING TOO GREAT

Web thinning is recommended for:

1. restoring chisel edge to the original length after several regrinds.
2. larger drills where the machine thrust is limited.
3. difficult materials.

Lip Clearance Angle

Drill Size (mm)	Angle (°)
Up to 3	18 - 24
3.1 - 6	14 - 18
6.1 - 12	10 - 14
12.1 - 20	8 - 12
Above 20	6 - 10

Broken or Twisted Tangs

(a) Possible Cause
Bad fit between the drill sleeve and the shank of the drill.
Solution
(i) Use only sleeves which are in good condition (avoid worn or damaged sleeves).
(ii) Ensure the drill shank and sleeve are thoroughly clean.

Note:
The tang is not intended to transmit the drive - it is only used for ejection. The Morse Taper is self-holding and relies on a good fit in the sleeve to transmit the drive.

Drill Web Split

(a) Possible Cause
The feed is too great.
Solution
Use the correct feed for the drill size material - see page 14.

(b) Possible Cause
Insufficient lip clearance behind the cutting edge.
Solution
Check that the lip clearance is as per information on page 24/ 25.

(c) Possible Cause
Excessive web thinning.
Solution
The web thickness should not be less than 10% of the drill diameter.

(d) Possible Cause
Using a hard object to seat the drill in the sleeve.
Solution
Use soft material e.g. copper or wood, to seat the drill.

Worn outer Corners

(a) Possible Cause
The peripheral speed is too high for the material being drilled.
Solution
Use the recommended speed - see page 11.

Broken outer Corners

(a) Possible Cause
Drilling thin material particularly when not properly supported.
Solution
Use a sheet metal drill and clamp the workpiece securely.

(b) Possible Cause
Using a 2 flute drill to enlarge the diameter of an existing hole.
Solution
Only core drills should be used for this purpose.

Chipped or Broken Lips

(a) Possible Cause
This is usually caused by excessive lip clearance angles behind the cutting edge.
Solution
Check that the lip clearance is as per information on page 24/25.

Oversized and Out of Round Holes

(a) Possible Cause
Unequal point angles.
Solution
This usually results when hand grinding the point. Use a point grinding fixture or machine.

(b) Possible Cause
Unequal cutting edge length (lip height).
Solution
When re-grinding ensure that the same amount of material is removed from both flanks.

(c) Possible Cause
Loose spindle or worn drill sleeve.
Solution
Use equipment which is in good condition.

(d) Possible Cause
The workpiece moves.
Solution
The workpiece must be securely clamped.

Drill rubbing and not cutting

(a) Possible Cause
Too little lip clearance behind the cutting edge.
Solution
Check that the lip clearance is as per information on page 24/25.

Cracks in cutting edges

(a) Possible Cause
The point is overheated and cooled too quickly when re-sharpening.
Solution
Use coolant when grinding or grind in stages, quenching frequently in soluble oil.

Rough hole finish

(a) Possible Cause
The drill is blunt.
Solution
Re-sharpen as per information on page 24/25.

(b) Possible Cause
Inadequate supply of coolant to the point.
Solution
The coolant must reach the point of the drill.

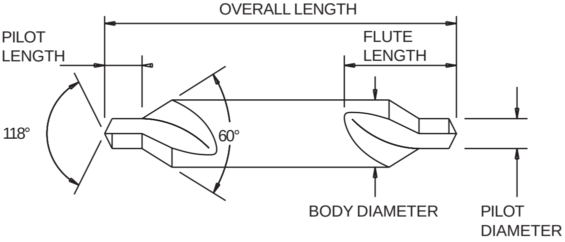
Drill breaks at flute runout

(a) Possible Cause
The workpiece moves during drilling.
Solution
The workpiece must be securely clamped.

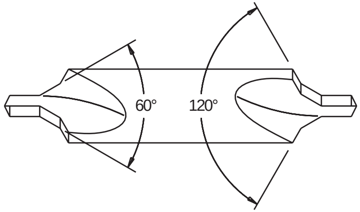
(b) Possible Cause
The flutes are clogged with swarf.
Solution
Clear the flutes by frequently withdrawing the drill, or use a drill more suited to the material e.g. a UDL drill for aluminium.

(c) Possible Cause
Using the wrong type of drill e.g. using a jobber drill for thin material.
Solution
See pages 6 to 9 for the correct drill to suit the application.

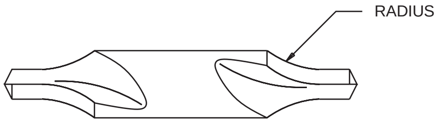
CENTRE DRILL NOMENCLATURE



TYPE 'A'



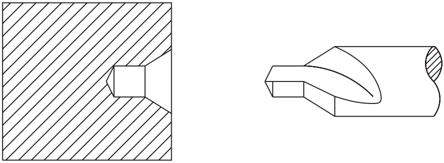
TYPE 'B'



TYPE 'R'

SELECTING THE CORRECT CENTRE DRILL

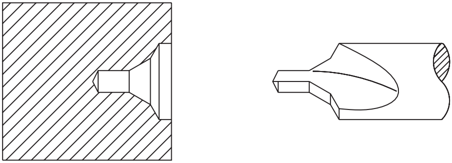
TYPE "A"



For general centering operations on workpieces requiring additional maching between centres.

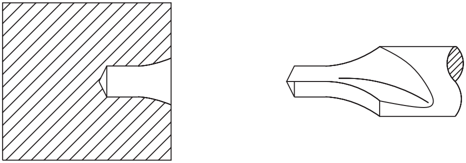
TYPE "B" (Protected Centre)

Sometimes called Bell Type



The 60° cone surface produced by this centre drill is recessed below the surface of the workpiece and is therefore protected from damage.

TYPE "R" (Radius)



The type "R" centre drill is also used for general centering operations, but produces a radius centre suitable for a variety of male centre angles eg. 60°, 82° or 120° can be used as an alternative to type "A" above.

THE CORRECT USE OF CENTRE DRILLS

A guide to successful drilling

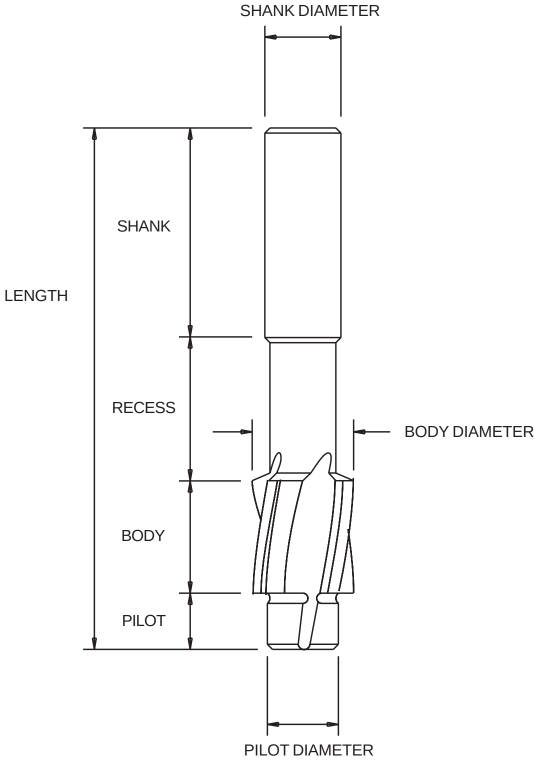
Recommended Speeds
The peripheral speeds for centre drills are the same as for 2 flute drills given on page 10-11. For calculation purposes the nominal diameter given below should be used.

Centre Drill Size	Nominal Diameter (mm)	Centre Drill Size (mm)	Nominal Diameter (mm)
BS 1	2	1	2
BS 2	3	1.25	2
BS 3	4	1.6	3
BS 4	6	2	4
BS 5	8	2.5	5
BS 6	11	3.15	6
BS 7	14	4	7
		5	9
		6.3	11
		8	14
		10	18

Recommended Feeds
Use the nominal diameter given above to establish the feed as given on page 19, and then reduce by 40% for centre drills.

Re-sharpening of Centre Drills
Centre Drill can be re-sharpened on the point only. refer to the re-sharpening guide for 2 flute drill on page 24/25.

COUNTERBORE NOMENCLATURE



A General Guide

Counterbores are used to create seatings for cap screw heads and are therefore identified by the cap screw they suit. They are available with straight or Morse Taper shanks.

Cap Screw Size	Pilot Drill Size (mm)	Counterbore Diameter (mm)
M 3	3.4	6
M 3.5	3.9	6.5
M 4	4.5	8
M 5	5.5	10
M 6	6.6	11
M 8	9	15
M 10	11	18
M 12	14	20

Speeds & Feeds

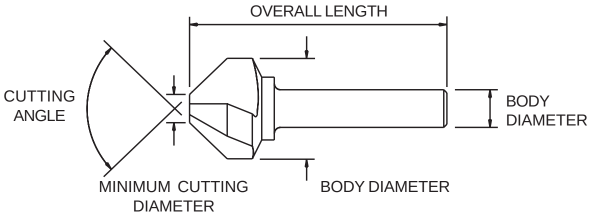
The speeds and feeds for counterbores are approximately 80% to 85% of those for drills as given on page 11.
The counterbore diameter given in the above table is used for this calculation.

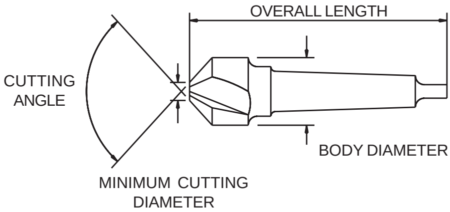
RE-SHARPENING

Counterbores are re-sharpened only by grinding the front cutting edges, maintaining the original relief angle of 6°- 8°.

COUNTERSINKS

COUNTERSINKS NOMENCLATURE





THE CORRECT USE OF COUNTERSINKS

A General Guide

Countersinks are normally used to produce a 60° or 90° chamfer recess which accommodates the corresponding 60° or 90° screw head. They are available in straight or Morse Taper Shank.

Speeds and Feeds

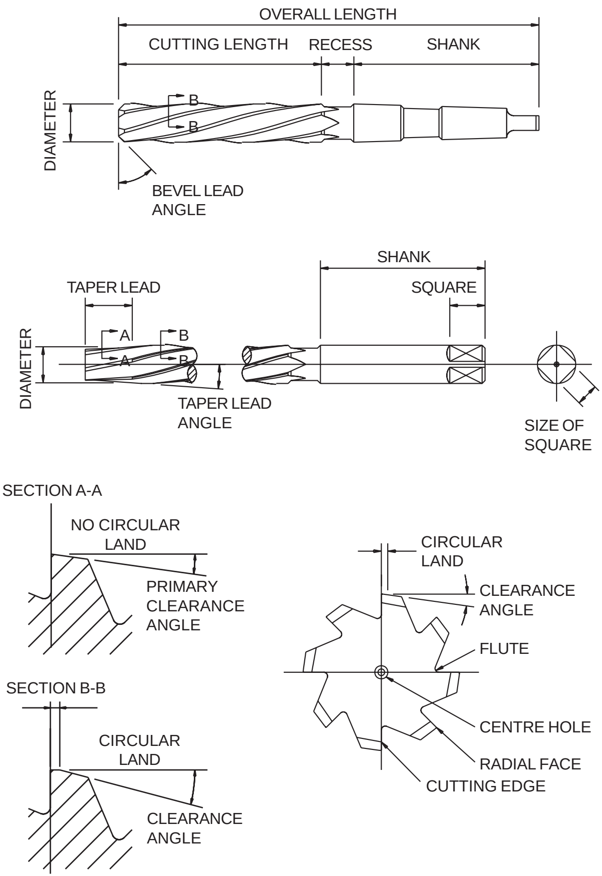
The speeds and feeds for countersinks are the same as those for drills (see page 11) and are based on the diameter midway between the largest and smallest diameter of the countersink.

RE-SHARPENING

The axial relief is critical to the performance of the countersink and should not be altered. When re-sharpening, grind only the flute face.

REAMERS

REAMER NOMENCLATURE



SELECTING THE CORRECT REAMER
Standard Reamers

Parallel Hand Reamers

General hand reaming.

MTS Parallel Machine Reamers

General machine reaming.

Machine Chucking Reamers, Parallel Shank

General machine reaming for deeper holes.

MTS Machine Chucking Reamers

General machine reaming for deeper holes.

MTS Taper Bridge Machine Reamers

For opening out existing holes for alignment on structural steel work.

Intermediate size reamers are available on request.

Hand Taper Pin Reamers - Metric

For reaming holes to suit standard metric taper pins with a taper of 1:50.

Hand Taper Pin Reamers - Fractional

For reaming holes to suit standard fractional taper pins with a taper of 1:48.

MTS Taper Socket Finishing Reamers

Finishing of Morse Taper holes.

MTS Taper Socket Roughing Reamers

Rough reaming Morse Taper holes.

Machine Chucking Reamers, Parallel Shank Tungsten Carbide Tip

Reaming of difficult to machine materials or mass production.

A guide to successful reaming

- Make sure the workpiece is securely held and supported. Should it bend or move, it could result in a poor finish or cause the reamer to break.
- Use a good morse taper sleeve and thoroughly clean both the sleeve and the taper shank of the reamer.
- As a reamer only cuts on the bevel lead and not on the peripheral land, it is essential to keep it sharp. A blunt reamer wears on the outer corners on the bevel lead, resulting in a poor fininsh, undersize holes and increased torque. (See page 39 for re-sharpening details.)
- Direct an adequate supply of the recommended lubricant to the cutting area. When reaming high tensile materials, an improved surface finish can be achieved by using chlorinated or sulphurised oils.

Stock Removal

Reamers are used to produce accurate holes with a good surface finish. It is a common fault to leave too little stock for removal by reaming. This results in a rubbing action and excessive wear of the reamer. The table below shows approximate amounts of stock to be removed by reaming.

Machine Reamers

Size of Reamed Hole (mm)		Pre-Drilled (mm)	Pre-Core Drilled (mm)
Above	Up to		
	1.5	0.3	0.2
1.5	3	0.3	0.2
3	6	0.3	0.2
6	13	0.4	0.25
13	25	0.5	0.3
25		0.5	0.3

Hand Reamers

The hand reaming allowance should be approximately two thirds of the machine reaming allowance.

* Feed Conversion Table

Reamer Diameter Range (mm)		Feed (mm/rev)		
Above	Up to	Light (L)	Medium (M)	Heavy (H)
	1.5	0.005 - 0.025	0.012 - 0.05	0.025 - 0.075
1.5	3	0.025 - 0.05	0.05 - 0.1	0.075 - 0.15
3	6	0.05 - 0.1	0.1 - 0.15	0.15 - 0.25
6	13	0.1 - 0.15	0.15 - 0.25	0.25 - 0.38
13	24	0.15 - 0.25	0.25 - 0.5	0.38 - 0.76
25		0.25 - 0.5	0.5 - 1	0.76 - 1.27

Tolerances

Somta reamers are manufactured to produce holes to H7 tolerance. The tolerance limits shown in the table below are added to the nominal reamer diameter.

eg. nominal diameter = 12mm

actual diameter = 12.008mm/12.015mm

Tolerance limits for reamers and hole sizes produced.

Reamer Diameter Range (mm)		Cutting Diameter Tolerance		Hole Diameter Tolerance H7	
Above	Up to	mm		mm	
1	3	+0.004	+0.008	0	+0.010
3	6	+0.005	+0.010	0	+0.012
6	10	+0.006	+0.012	0	+0.015
10	18	+0.008	+0.015	0	+0.018
18	30	+0.009	+0.017	0	+0.021
30	50	+0.012	+0.021	0	+0.025

Other useful tolerances can be found on page 121.

RE-SHARPENING

A reamer is only sharpened on the bevel lead which performs the cutting action. This operation must be done only by skilled operators on appropriate machine tools.

When re-sharpening it is essential to maintain both the original relief angle of 6°- 8° and the concentricity of the bevel lead.

REAMER TECHNICAL DATA

TYPE	GRADE	TYPICAL PHYSICAL PROPERTIES HARD NESS BRINELL	TONS PER SQ IN. (MAX)	N/mm² (MAX)	† Speed m/min	*Type of Feed
CARBON STEEL & ALLOY STEEL	FREE CUTTING	150	35	525	12-15	M-H
	0.3 to 0.4% Carbon	170	40	600		
	0.3 to 0.4% Carbon	248	59	900	7-10	M
	0.4 to 0.7% Carbon	206	47	700		
	0.4 to 0.7% Carbon	286	67	1000	5-8	L
		248	59	900	7-10	M
		330	75	1125	5-8	M
		380	87	1300	2-4	L
STAINLESS STEEL	Martensitic Free Cutting	380	54	810	5-8	M
	Martensitic Std. Grade				2-5	L-M
	Austenitic Free Cutting	As Supplied			5-8	L-M
	Austenitic Std. Grade				2-5	L-M
NIMONIC ALLOYS	Wrought	300	67	1000	2-5	L
	Cast	350	78	1200		
TITANIUM	Titanium Comm: Pure	170	40	600	7-10	M
	Titanium Comm: Pure	200	43	650		
	Titanium Comm: Pure	275	65	975		
	Titanium Alloyed	340	76	1140	2-4	L-M
	Titanium Alloyed	380	85	1275		
TOOL STEEL	HSS Standard Grades	225	48	720	7-10	M
	HSS Cobalt Grades					
	Hot Working Steel	225	54	800		
	Cold Working Steel					

† See Speed Conversion Chart on page 16/17.

* See table on page 39.

cont on page 41

REAMER TECHNICAL DATA

TYPE	GRADE	TYPICAL PHYSICAL PROPERTIES HARD NESS BRINELL	TONS PER SQ IN. (MAX)	N/mm² (MAX)	† Speed m/min	*Type of Feed
CAST IRONS	Grey	250	52	780	12-15	M-H
	Ductile				10-13	M-H
	Maleable	330	74	1100	12-15	M-H
	Hardened & Tempered				4-5	M
MANGANESE STEEL		As Supplied			2-3	L
ALUMINIUM ALLOYS		As Supplied			30-45	H
MANGANESE ALLOYS		As Supplied			35-60	H
ZINC ALLOYS		As Supplied			30-45	H
COPPER ALLOYS	Brass Free Cutting	As Supplied			20-35	H
	Brass Low Leaded				30-45	H
	Bronze Silicon				15-30	H
	Bronze Manganese				10-15	M
	Copper				15-45	M-H
	Bronze Aluminium Bronze Commercial Bronze Phospor				7-15	M
PLASTICS	Soft Hard Reinforced	As Supplied			12-15	M-H

† See Speed Conversion Chart on page 16/17.
* See table on page 39.

REAMING PROBLEMS: CAUSES AND SOLUTIONS

Poor Surface Finish

(a) Possible Cause
Incorrect speed and/or feed.
Solution
Use the recommended speed/feed - see page 40/41.

(b) Possible Cause
A Worn reamer
Solution
Do not allow the reamer to become too blunt. See page 39 for re-sharpening details.

(c) Possible Cause
Insufficient or wrong type of lubricant.
Apply and adequate supply of the correct lubricant to the cutting area.
See the drill table on page 11 for the recommended lubricants.

(d) Possible Cause
Damaged cutting edges.
Solution
Use a reamer which is in good condition.

Reamer Chattering

(a) Possible Cause
Lack of rigidity in set up.
Solution
Only use equipment which is in good condition and make sure the workpiece is securely held.

(b) Possible Cause
Feed too low.
Solution
Use the recommended speed/feed - see page 40/41.

Reamer showing rapid wear

(a) Possible Cause
Too little stock in the hole for reaming causing the reamer to rub and not cut.
Solution
See page 38 for recommended stock removal.

(b) Possible Cause
Speed too high or feed too low.
Solution
Use the recommended speed/feed - see page 40/41.

(c) Possible Cause
The workpiece material is too hard.
Solution
Use a HSS-Co reamer.

Tapered or Bell-Mouthed holes

(a) Possible Cause
Mis-alignment of the reamer and the hole.
Solution
Align the reamer and the hole.

(b) Possible Cause
The machine spindle and/or bearings are worn.
Solution
Only use equipment which is in good condition.

Reamer rubbing and not cutting

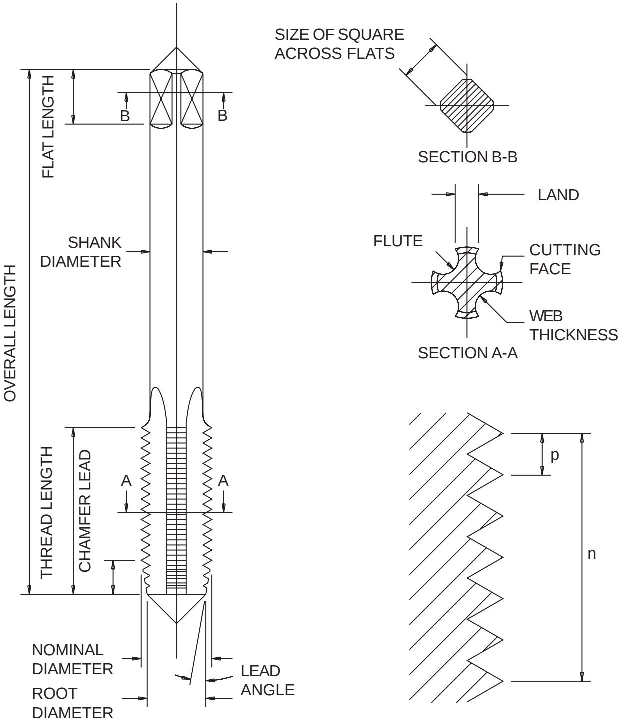
(a) Possible Cause
Too little reaming allowance in the hole.
Solution
See table of stock removal on page 38.

(b) Possible Cause
Reamer re-sharpened with too little or no relief on the bevel lead.
Solution
Re-grind the bevel lead to a 6°- 8° relief.

Oversized holes

(a) Possible Cause
Excessive run-out on the machine spindle or holding device eg. taper sleeve, collet or chuck.
Solution
Only use equipment which is in good conditon.

TAP NOMENCLATURE



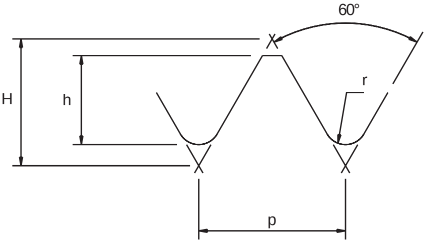
n = No. OF THREADS
PER INCH.

p = PITCH

Abbreviations for standard thread forms

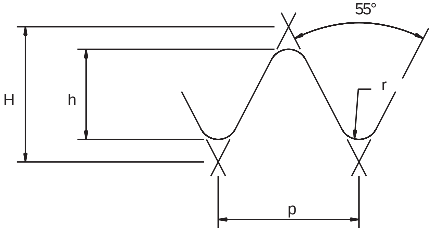
BA	- British Association
BSB	- British Standard Brass
BSP	- British Standard Pipe (Fine) "G" Series
BSPT	- British Standard Pipe Taper (Rc Series)
BSW	- British Standard Whitworth
BSF	- British Standard Fine
M	- Metric
MF	- Metric Fine
NPS	- National Pipe Straight
NPT	- National Pipe Taper
UNC	- Unified National Coarse
UNF	- Unified National Fine
UNEF	- Unified Extra Fine

ISO METRIC



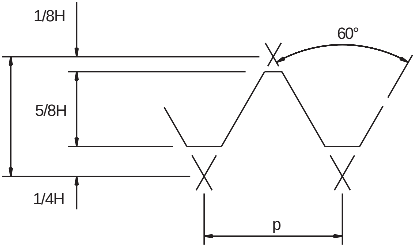
H = 0.866P
h = 0.708P
r = 0.1443P

WHITWORTH



H = 0.960491P
h = 0.640327P
r = 0.137329P

UNIFIED



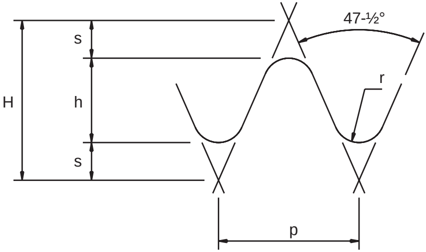
H = 0.86603P

5/8H = 0.54127P

1/4H = 0.21651P

1/8H = 0.10825P

BA



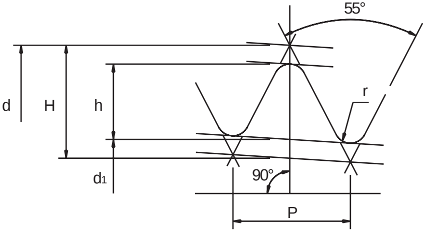
H = 1.1363365P

h = 0.6P

s = 0.26817P

r = 0.18083P

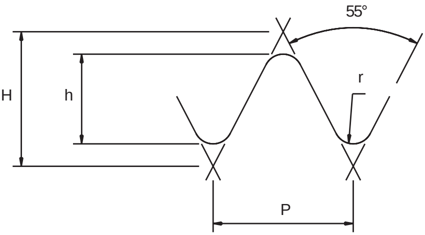
BSPT



H = 0.960237P
h = 0.640327P
r = 0.137278P

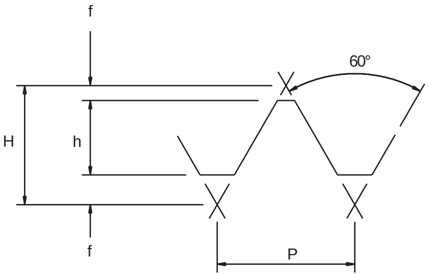
d = MAJOR DIAMETER AT GAUGE PLANE
d_1 = MINOR DIAMETER AT GAUGE PLANE
TAPER = 1 IN 16 ON DIAMETER

BSB



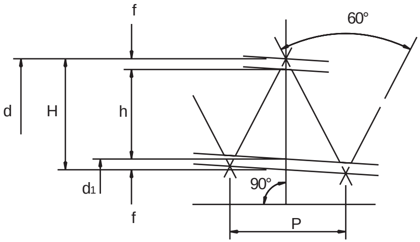
H = 0.86603P
h = 0.5237P
r = 0.1667P

NPS



H = 0.866P
h = 0.8P
f = 0.033P

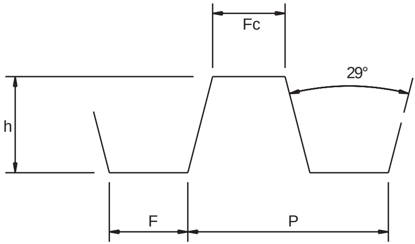
NPT



H = 0.866P
h = 0.8P
f = 0.033P

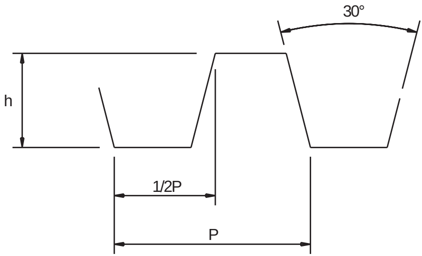
d = MAJOR DIAMETER AT GAUGE PLANE
d_1 = MINOR DIAMETER AT GAUGE PLANE
TAPER = 1 IN 16 ON DIAMETER

ACME



h = 0.5P + CLEARANCE
F = 0.3707P
Fc = 0.3707P - (0.256 X MAJOR DIAMETER ALLOWANCE)

TRAPEZOIDAL

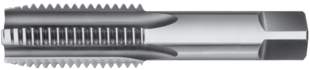


h = 0.5P + CLEARANCE

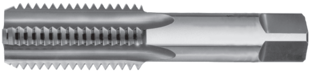
SELECTING THE CORRECT TAP

Short Machine and Hand Taps

Taper



Second



Bottoming

For general purpose hand or machine use for short production runs. Best suited for materials which do not present chip disposal problems.

Machine Taps

Spiral Point Tap

Sometimes called a gun nosed tap. For machine use on through holes. Suitable for a wide range of materials. The gun nose creates chip disposal ahead of the tap while the flute geometry allows an adequate supply of lubricant to the cutting area, making higher tapping speed possible.

Spiral Flute Tap

Mainly for work in blind holes and on ductile materials, such as aluminium and zinc alloys, which produce long stringy chips. The taps have a 35° right hand helix. The flute shape eliminates clogging and jamming, resulting in improved tap life.

Colour Band Application Taps (CBA)

The primary benefit of the CBA range is enhanced threading performance due to geometry designed for specific material application groups. The result is an improved quality of finish and an increased number of holes per tap, giving extended tap life and reduced cost per hole. Manufactured from HSS-EV steel (High Vanadium) for greater wear resistance.

Red Band


Designed for high tensile materials such as Tool Steels, Heat Treatable Steels, Spring Steel, Case Hardening Steel, Unalloyed Titanium, Nitriding Steel, Cold Drawn Constructional Steel and High Tensile Steel.Used to tap materials with hardness up to 470HB, tensile strength up to 1500N/mm². Spiral flute taps have 15° right hand helix which efficiently forces high tensile material swarf up out of the hole, while still maintaining correct cutting geometry. The red band tap is supplied as standard with TiAlN coating.

Blue Band


Designed for tough materials, such as Stainless Steel, Titanium Alloys, Cast Steel, Heat Resisting Steel and Work Hardening Steel. Used to tap materials with hardness up to 350HB, tensile strength up to 1250N/mm². Truncated thread after lead reduces frictional contact with the threaded hole and allows easier penetration of coolant. Spiral flute taps have 40° right hand helix allowing tough material swarf to be efficiently removed from the hole. Supplied as standard with TiAlN coating.

Colour Band Application Taps (CBA)

Yellow Band

Designed for more ductile materials such as Aluminium, Magnesium Alloys, Soft Brass (MS58), Plastics, Zinc Alloys and Copper. Used to tap materials with hardness up to 250HB, tensile strength up to 900N/mm². Wide flutes allow more efficient swarf removal which prevents clogging and excessive torque. High rake angle improves shear characteristic and reduces build-up on the cutting edge, allowing tap to cut more freely for longer periods. Spiral flute taps have 40° right hand helix, allowing ductile material swarf to be efficiently forced out of the hole. The yellow band tap is supplied as standard in bright condition.

White Band

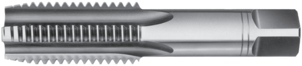
Designed for highly abrasive materials such as Cast Iron and reinforced plastics. Used to tap materials with hardness up to 300HB tensile strength up to 1000N/mm². Increased number of flutes reduces torque and increases tap life. Taps have 15° right hand helix. The white band tap is TiAlN coated as standard.

Fluteless Taps

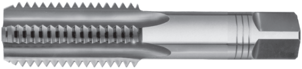
For machine use on through or blind holes. Best suited for ductile materials, such as aluminium and zinc alloys as the threads are cold formed, not cut like a conventional tap. For slightly tougher materials fluteless taps in the range of 5mm to 12mm can be supplied with a gash.

Serial Taps

Rougher



Intermediate

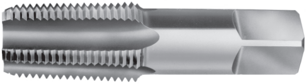


Finisher

For general purpose machine or hand use in tough materials, producing accurate threads with a high finish. Used in sequence to remove most of the material in stages before finally sizing with the Finishing tap.

Pipe Taps

For machine use on pipe work for parallel threads.



For machine use on pipe work for tapered threads.

Special taps are available on request.

RECOMMENDED TAPPING DRILL SIZES
(For 75% thread depth)

Metric Coarse

Size	Pitch	Tapping Drill Size (mm)
2	0.4	1.6 (1.8)*
2.5	0.45	2.05
3	0.5	2.5 (2.75)*
3.5	0.6	2.9 (3.2)*
4	0.7	3.3 (3.65)*
4.5	0.75	3.7 (4.1)*
5	0.8	4.2 (4.6)*
6	1	5 (5.5)*
7	1	6
8	1.25	6.8 (7.4)*
9	1.25	7.8
10	1.5	8.5 (9.25)*
11	1.5	9.5
12	1.75	10.2 (11.1)*
14	2	12
16	2	14
18	2.5	15.5
20	2.5	17.5
22	2.5	19.5
24	3	21
27	3	24
30	3.5	26.5
32	3.5	30.5
33	3.5	29.5
36	4	32
39	4	35
42	4.5	37.5
45	4.5	40.5
48	5	43
52	5	47
56	5.5	50.5

*Fluteless Tapping Drill Sizes

Metric Fine

Size	Pitch	Tapping Drill Size (mm)
2	0.25	1.75
2.5	0.35	2.15
3	0.35	2.65
3.5	0.35	3.15
4	0.5	3.5
4.5	0.5	4
5	0.5	4.5
6	0.75	5.25
7	0.75	6.25
8	0.75	7.2
8	1	7
9	1	8
10	1	9
10	1.25	8.75
12	1.25	10.75
12	1.5	10.5
14	1.25	12.75
14	1.5	12.5
16	1.0	15
16	1.5	14.5
18	1.5	16.5
18	2.0	16
20	1.5	18.5
20	2	18
22	1.5	20.5
22	2	20
24	1.5	22.5
24	2	22
25	1.5	23.5
25	2	23
27	2	25
30	1.5	28.5
30	2	28
32	1.5	30.5
33	1.5	31.5
36	1.5	34.5
36	2.0	34

Metric Fine (cont)

Size	Pitch	Tapping Drill Size (mm)
39	1.5	37.5
40	1.5	38.5
42	1.5	40.5
45	1.5	43.5
48	1.5	46.5
50	1.5	48.5
52	1.5	50.5

BSW

Nominal Diameter	TPI	Tapping Drill Size (mm)
3/32	48	1.9
1/8	40	2.55
5/32	32	3.2
3/16	24	3.7
7/32	24	4.5
1/4	20	5.1
5/16	18	6.5
3/8	16	8
7/16	14	9.3
1/2	12	10.5
9/16	12	12.2
5/8	11	13.5
3/4	10	16.5
7/8	9	19.5
1"	8	22
1-1/8	7	25
1-1/4	7	28
1-1/2	6	34
1-3/4	5	39
2"	4-1/2	45

BSF

3/16	32	4
7/32	28	4.7
1/4	26	5.4
5/16	22	6.8

BSF

Nominal Diameter	TPI	Tapping Drill Size (mm)
3/8	20	8.3
7/16	18	9.8
1/2	16	11
9/16	16	12.7
5/8	14	14
3/4	12	16.5
7/8	11	19.5
1"	10	22.5
1-1/8	9	25.5
1-1/4	9	29
1-1/2	8	34.5

UNC

No.3	48	2
No.4	40	2.25
No.5	40	2.6
No.6	32	2.75
No.8	32	3.4
No.10	24	3.8
No.12	24	4.4
1/4	20	5.1
5/16	18	6.6
3/8	16	8
7/16	14	9.4
1/2	13	10.8
9/16	12	12.2
5/8	11	13.5
3/4	10	16.5
7/8	9	19.5
1"	8	22
1-1/8	7	25
1-1/4	7	28
1-3/8	6	31
1-1/2	6	34
1-3/4	5	39
2"	4-1/2	45

UNF

Nominal Diameter	TPI	Tapping Drill Size (mm)
No.3	56	2.1
No.4	48	2.35
No.5	44	2.65
No.6	40	2.9
No.8	36	3.5
No.10	32	4.1
No.12	28	4.6
3/16	32	4
1/4	28	5.5
5/16	24	6.9
3/8	24	8.5
7/16	20	9.8
1/2	20	11.5
9/16	18	12.8
5/8	18	14.5
3/4	16	17.5
7/8	14	20.5
1"	12	23.5
1-1/8	12	26.5
1-1/4	12	29.5
1-3/8	12	32.5
1-1/2	12	36

BA

0	25.4	5.1
1	28.2	4.5
2	31.3	3.9
3	34.8	3.4
4	38.3	3
5	43.1	2.65
6	47.9	2.3
7	52.9	2.05
8	59.1	1.8
9	65.1	1.55
10	72.6	1.4

BSP

Nominal Diameter	TPI	Tapping Drill Size (mm)
1/8	28	8.8
1/4	19	11.8
3/8	19	15.5
1/2	14	19
5/8	14	21
3/4	14	24.5
7/8	14	28.5
1"	11	31
1-1/4	11	40
1-1/2	11	45.5
1-3/4	11	51.5
2"	11	57

BSPT

1/8	28	8.6
1/4	19	11.5
3/8	19	15.0
1/2	14	18.5
3/4	14	24.0
1"	11	30.25
1-1/4	11	39.0
1-1/2	11	45.0
2"	11	56.5

NPS

1/8	27	9.1
1/4	18	12.0
3/8	18	15.5
1/2	14	19.0
3/4	14	24.5
1"	11-1/2	30.5
1-1/4	11-1/2	39.4
1-1/2	11-1/2	45.5
2"	11-1/2	57.5

NPT

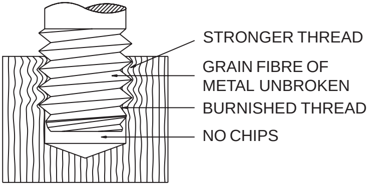
Nominal Diameter	TPI	Tapping Drill Size (mm)
1/8	27	8.4
1/4	18	11.0
3/8	18	14.25
1/2	14	17.5
3/4	14	23.0
1"	11-1/2	29.0
1-1/4	11-1/2	37.5
1-1/2	11-1/2	43.5
2"	11-1/2	55.5

Fluteless Taps

Fluteless taps are used for cold forming threads in ductile materials and have the following advantages.

(a) Increased strength and tap life resulting from:
  (i) Elimination of flutes which reduce the shear strength of the tap.
  (ii) The lack of cutting edges which, in a conventional tap, wear and break down.
  (iii) The lack of chips, which sometimes causes jamming.
(b) Better blind hole tapping due to the lack of chips and problems relating to chip removal.
(c) Higher productivity due to faster tapping speeds.
(d) Stronger threads.

FLUTELESS TAP



The grain fibres of the metal are not cut, but displaced, to form the threads, which are stronger than cut threads. It is accepted that a 60% cold formed thread is as strong as a 75% cut thread.

TAP TECHNICAL DATA

TYPE	GRADE	TYPICAL PHYSICAL PROPERTIES		
		HARDNESS BRINELL	TONS PER SQ IN.	N/mm²
CARBON STEEL	FREE CUTTING	150	33	500
	0.3 to 0.4% Carbon	170	38	570
	0.3 to 0.4% Carbon	248	54	800
	0.4 to 0.7% Carbon	206	44	650
	0.4 to 0.7% Carbon	286	63	950
ALLOY STEEL		248	54	810
	Tough	330	74	1100
	Hard	380	82	1250
STAINLESS STEEL	Martensitic Free Cutting	248	54	810
	Martensitic Std. Grade			
	Austenitic Free Cutting	As Supplied		
	Austenitic Std. Grade			
NIMONIC ALLOYS	Wrought	300	67	1000
	Cast	350	78	1170
TITANIUM	Titanium Comm: Pure	170	38	570
	Titanium Comm: Pure	200	43	650
	Titanium Comm: Pure	275	65	975
	Titanium Alloyed	340	76	1140
	Titanium Alloyed	380	85	1275
TOOL STEEL	HSS Standard Grades	225	48	720
	HSS Cobalt Grades			
	Hot Working Steel	225	54	810
	Cold Working Steel			
MANGANESE STEEL		As Supplied		

RECOMMENDED TAP TYPE		ALTERNATIVE TAP TYPE		*TAP PERIPHERAL SPEED m/min	LUBRICANTS
THROUGH HOLE	BLIND HOLE	THROUGH HOLE	BLIND HOLE		
Sp/Point	Sp/Flute	Str/Flute	Str/Flute	10-15 8-12 8-10	Sulphur based oil
Sp/Point	Sp/Flute	Str/Flute	Str/Flute	8-12	Sulphur based oil
Sp/Point	Sp/Flute	Str/Flute	Str/Flute	2-6	Heavy duty Sulphur based oil
See CBA Tap section pages 66 & 67.				2-4	Chlorinated oil
See CBA Tap section pages 66 & 67.				2-4	Chlorinated oil
Sp/Point	Sp/Flute	Str/Flute	Str/Flute	8-10	Sulphur based oil
Sp/Point	Str/Flute	Str/Flute	-	15-20	Sulphur based oil

* The tapping speeds for fluteless taps are 2-3 times higher than the recommended speeds given.

cont on page 64

TAP TECHNICAL DATA

TYPE	GRADE	TYPICAL PHYSICAL PROPERTIES		
		HARD NESS BRINELL	TONS PER SQ IN.	N/mm^2
CAST IRONS	Grey Ductile	240	52	780
	Maleable Hardened & Tempered	330	74	1110
ALUMINIUM ALLOYS	Long Chip Short Chip	As Supplied		
MANGANESE ALLOYS		As Supplied		
ZINC ALLOYS		As Supplied		
COPPER ALLOYS	Brass Free Cutting Brass Low Lead Bronze Silicon Bronze Manganese Copper Free Machining Copper Electrolytic Bronze Aluminium Bronze Commercial Bronze Phosphor	As Supplied		
PLASTICS	Soft Hard Reinforced	As Supplied		

TAP TECHNICAL DATA *(cont.)*

RECOMMENDED TAP TYPE		ALTERNATIVE TAP TYPE		*TAP PERIPHERAL SPEED m/min	LUBRICANTS
THROUGH HOLE	BLIND HOLE	THROUGH HOLE	BLIND HOLE		
Str/Flute	Str/Flute	Sp/Point	-	5-10 4 -8	Dry soluble oil or paraffin
Fluteless	Fluteless	Sp/Point	Sp/Flute	20-25 10-15	Sol. oil or light material oil
Sp/Point	Sp/Flute	Str/Flute	Str/Flute	15-20	Sul. B Oil
Fluteless	Fluteless	Str/Flute	Str/Flute	15-20	Soluble Oil
				15-20	Sol. oil or light mineral oil
Fluteless	Fluteless	Str/Flute	Str/Flute	25-30	
				10-12	
Sp/Point	Str/Flute	Str/Flute	Str/Flute	3-5	
Fluteless	Fluteless	Sp/Point	Sp/Point	15-20	Chlorinated oil or soluble oil
				8-12	
				10-12	
Sp/Point	Str/Flute	Str/Flute	Str/Flute	3-5	
				3-5	Sol. oil or light mineral oil
Str/Flute	Str/Flute	Sp/Point	-	4-7 12-15	Dry

* The tapping speeds for fluteless taps are 2-3 times higher than the recommended speeds given.

For optimum performance for machine tapping see Colour Band Application (CBA) section pages 66 & 67.

CBA TAP TECHNICAL DATA

Surface Treatment (Coating) TiN, TiCN, TiAlN coatings are available on request				
MATERIAL TYPES		HARDNESS HB	TENSILE STRENGTH N/mm²
Steel	Free Cutting steels	≤ 120	≤ 400
	Structural steel. Case carburizing steel	≤ 200	≤ 700
	Plain carbon steel	≤ 250	≤ 850
	Alloy steel	> 250	≤ 850
	Alloy steel. Hardened and tempered steel	> 250 ≤ 350	> 850 ≤ 1200
	Alloy steel. Hardened and tempered steel	> 350	> 1200
Stainless Steel	Free machining Stainless steel	≤ 250	≤ 850
	Austenitic	≤ 250	≤ 850
	Ferritic + Austenitic, Ferritic, Martensitic	≤ 300	≤ 1000
Cast Iron	Lamellar graphite	≤ 150	≤ 500
	Lamellar graphite	> 150 ≤ 300	> 500 ≤ 1000
	Nodular graphite, Malleable Cast Iron	≤ 200	≤ 700
	Nodular graphite Malleable Cast Iron	> 200 ≤ 300	> 700 ≤ 1000
Titanium	Titanium, unalloyed	≤ 200	≤ 700
	Titanium, alloyed	≤ 270	≤ 900
	Titanium alloyed	> 270 ≤ 350	> 900 ≤ 1200
Nickel	Nickel, unalloyed	≤ 150	≤ 500
	Nickel, alloyed	≤ 270	≤ 900
	Nickel, alloyed	> 270 ≤ 350	> 900 ≤ 1200
Copper	Copper	≤ 100	≤ 350
	Beta Brass, Bronze	≤ 200	≤ 700
	Alpha Brass	≤ 200	≤ 700
	High strength Bronze	≤ 470	≤ 1500
Aluminium Magnesium	Al, Mg, unalloyed	≤ 100	≤ 350
	Al alloyed Si < 0.5%	≤ 150	≤ 500
	Al alloyed, Si > 0.5% < 10%	≤ 120	≤ 400
	Al alloyed, Si > 10% Al-alloys, Mg-alloys	≤ 120	≤ 400
Synthetic Materials	Thermoplastics	-	-
	Thermosetting plastics	-	-
	Reinforced plastic materials	-	-

CBA TAP TECHNICAL DATA *(cont.)*

NORMAL CHIP FORM	SPEED M/Min		RECOMMENDED TAP TYPE (● Recommended X Suitable)			
	STANDARD	COATED	RED BAND	BLUE BAND	YELLOW BAND	WHITE BAND
extra long	12	18 - 27	X		●	
middle/long	12	18 - 27	X		●	
long	10	18 - 24	X		●	
long	10	18 - 24	●			
long	8	9 - 15	●			
long	5	9 - 15	●			
middle	9	18 - 24		●		
long	6	9 - 15		●		
long	5	8 - 15		●		
extra short	11	18 - 27				●
extra short	8	9 - 18				●
middle/short	11	18 - 27	X			●
middle/short	8	9 - 18				●
extra long	8	9 - 15	●			
middle/short	9	12 - 18	X	●		
middle/short	6	6 - 12	X	●		
extra long	9	12 - 18		●		
long	5	6 - 12	●		X	
long	4	5 - 11	●		X	
extra long	11	15 - 24			●	
middle/short	30	43 - 55			●	X
long	18	40 - 49			●	
short	5	6 - 12	●			
extra long	15	24 - 30			●	
middle	30	43 - 52			●	
middle/short	18	30 - 36	X		●	
short	15	24 - 30	X		●	
extra long	27	-			●	
short	11	15 - 21			●	
extra short	8	9 - 15	X			●

Metres / Min		4	6	8	9	10	12
Tap Size		Revolutions					
mm	Inch						
1.6	1/16	800	1194	1592	1791	1988	2386
1.8		708	1065	1415	1598	1768	2121
2		637	955	1274	1433	1591	1909
2.2	3/32	579	869	1158	1303	1446	1736
2.5		510	764	1019	1147	1274	1527
3	1/8	425	637	849	955	1061	1273
3.5		364	546	728	819	909	1091
4	5/32	318	478	637	718	796	955
4.5		283	425	566	637	707	849
5	3/16	255	382	510	573	637	764
6	1/4	212	319	425	477	530	636
7	9/32	182	273	364	409	455	546
8	5/16	159	239	319	358	398	477
9		142	212	283	318	354	425
10	3/8	127	191	255	286	318	382
11		116	174	232	260	289	347
12	1/2	106	159	212	238	265	318
13		98	147	196	220	245	294
14	9/16	91	136	182	205	277	273
16	5/8	80	119	159	179	199	239
18		71	106	141	159	177	212
20	3/4	64	96	127	143	159	191
22	7/8	58	87	116	130	145	174
24	1"	53	80	106	119	133	159
27		47	71	94	106	118	141
30	1-1/8	43	64	85	95	106	127
33	1-1/4	39	58	77	87	96	116
36		35	53	71	80	88	106
39	1-1/2	33	49	65	73	82	98
42		30	46	61	68	76	91
45	1-3/4	28	42	57	64	71	85
48		27	40	53	60	66	80
52		24	37	49	55	61	73
56	2"	23	34	46	51	57	68

TO rpm CONVERSION CHART

15	18	21	25	27	30	36
per Minute						
2983	3579	4176	4971	5369	5965	7158
2652	3182	3712	4419	4773	5303	6364
2386	2863	3341	3977	4295	4773	5727
2169	2603	3037	3616	3905	4339	5207
1909	2291	2673	3182	3436	3818	4582
1591	1909	2227	2651	2864	3182	3818
1364	1636	1909	2273	2455	2727	3273
1193	1432	1671	1989	2148	2387	2864
1061	1273	1485	1768	1909	2122	2546
955	1146	1337	1591	1719	1909	2292
795	954	1113	1326	1432	1592	1909
682	818	955	1136	1227	1364	1636
597	716	835	994	1074	1193	1432
531	637	742	885	955	1061	1293
477	573	668	795	859	955	1146
434	521	608	723	781	868	1041
398	477	557	663	716	796	955
367	441	514	612	661	734	881
341	409	477	568	614	682	818
298	358	418	497	537	597	716
265	318	371	442	477	530	636
239	286	334	398	430	477	573
217	260	304	362	391	434	521
199	239	275	331	353	398	477
177	212	245	295	318	354	424
159	191	223	265	286	318	382
145	174	203	241	360	289	347
133	159	186	221	239	265	318
122	147	171	204	220	245	294
114	186	159	189	205	227	273
106	127	149	177	191	212	255
99	119	139	166	179	199	239
92	110	129	153	165	184	220
85	102	119	142	153	170	205

A guide to successful machine tapping

- Use the correct tap to suit the application (see page 51/54).
- Select the correct tapping drill size (see page 55/61).
- Direct an adequate supply of the recommended lubricant to the cutting area of the tap (see page 63/65).
- Make sure the workpiece is securely held.
- Use a tapping attachment suited to the application and align the tap with the hole.
- When using a machine without lead screw feed, hand feed the tap until sufficient engagement produces self feed.
- When using a machine with lead screw feed, set the lead to correspond with that of the tap. This also applies on two and multi start taps.

Preparation of Holes

A good hole is a pre-requisite of a good thread. Some of the factors which contribute to inferior threads are:

(a) Out of round holes. The thread will be correspondingly out of round.
(b) Poor surface finish in the hole.
(c) Size of the hole. A hole which is too small will cause overloading of the tap with the possible breakage.
(d) Hard spots and abrasive surfaces in the cored holes. These holes should be pre-drilled.

Percentage Thread Depth

For general purpose work a thread depth of 75% is recommended. A drill size equal to the minor diameter of the tap produces a 100% thread depth. This practice is normally recommended fot the following reasons.

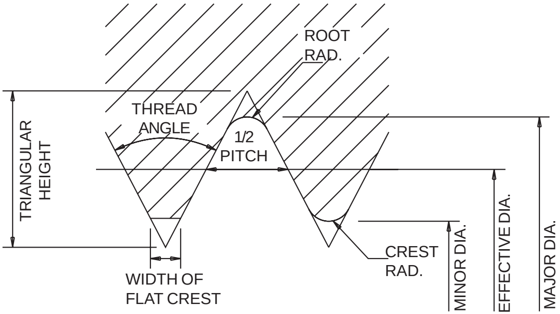
(a) 100% thread depth requires excessive power to turn the tap, with consequent possible breakage.
(b) 100% thread depth is only 5% stronger than the normal depth of 75%.
(c) Even a 50% thread depth still produces a thread stronger than its mating bolt.

Basic sizes and tolerance classes

To allow for clearance between mating internal and external threads, taps are manufactured with oversize allowances added to the basic diameters.
These basic diameters plus the oversize allowances establish:

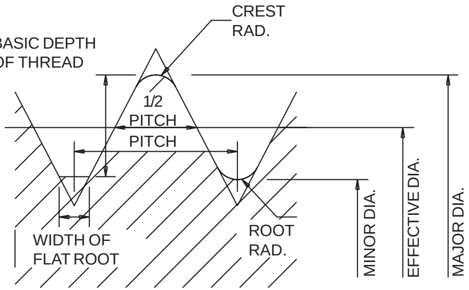
(a) the minimum effective diameter; and

NUT



(b) the minimum major diameter.

BOLT



Limits of Tolerance

Effective Diameter - The tolerance is the amount of variation allowed in the manufacture of the tap. This tolerance is added to the minimum effective diameter to establish the maximum effective diameter.

It follows that:
Basic Effective + Oversize
= Minimum Effective

Basic Effective + Oversize + tolerance
= Maximum Effective

The effective diameter can only be measured with special tap measuring equipment.

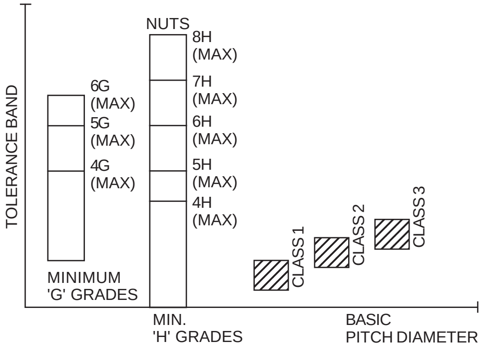
Major Diameter - The minimum major diameter is established by adding the oversize allowance to the basic major diameter (the nominal thread size). Therefore, on measurement, the major diameter of the tap is larger than the nominal thread size, and must not be used to judge the size of the tap.

The maximum major diameter of the tap is governed by the thread form and is therefore not subject to a tolerance.

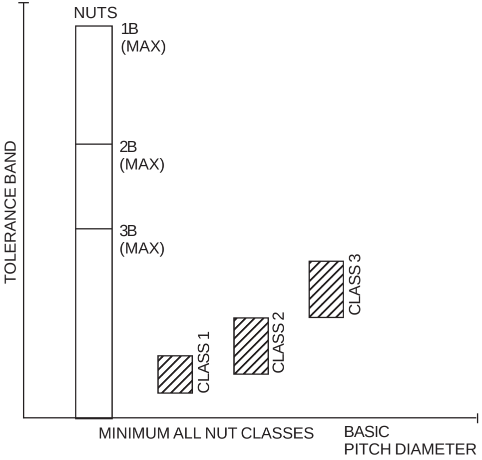
Tap Tolerance Classes

Relationships of Tap Classes to Nut Tolerances

Metric Threads



Unified Threads



Class 1 Tap
This is closest to basic, having little oversize allowance, and is normally specified for "close" fit threads, eg. Unified 3B, Metric 4H, 5H.

Class 2 Tap
This is normally specified for "medium" fit threads, eg. Unified 2B, Metric 6H, 4G, 5G.

Class 3 Tap
This is futhermost above basic size and used for "free" fit threads, eg. Unified 1B, Metric 7H, 8H, 6G.
Under favourable working conditions, the following thread tolerances should be produced by the new class taps.

	Class 1	Class 2	Class 3
Metric	4H, 5H	6H, 4G, 5G	7H, 8H, 6G
Unified	3B	2B	1B
Whitworth Form	Close Class	Medium Class	Free Class
BA	Close Class	Medium Class	Free Class

All Somta HSS taps are supplied to Class 2, 6H unless otherwise specified.

RE-SHARPENING
Maximum productivity and tap life can only be obtained from a tap that is kept in good condition and handled with care.

When re-sharpening becomes necessary, regrinding by hand is not recommended, though it is probably better than using chipped or worn taps. The recommended method is to use special tap grinding attachments or machines, and to follow the original form of the tap.

TAPPING PROBLEMS: CAUSES AND SOLUTIONS

Damaged tap threads in the hole

(a) Possible Cause
Mis-alignment of the tap with the hole before starting to tap.
Solution
Care must be taken to align the tap with the hole before starting to tap.

(b) Possible Cause
The tap is too dull
Solution
Use a tap which is in good condition.

(c) Possible Cause
Work hardened skin in the drilled hole.
Solution
Work hardening can be avoided when drilling by using the correct speeds,and coolants. See page 63/65. Use serial taps.

(d) Possible Cause
Incorrect rake angle
Solution
Use the recommended tap for the material. See page 51/54.

Poor finish of the thread

(a) Possible Cause
Using the incorrect tap.
Solution
Use the recommended tap.

(b) Possible Cause
The drilled hole is too small.
Solution
Use the recommended drill size. See page 55/61.

(c) Possible Cause
The tap is too dull.
Solution
Use a tap which is good condition.

(d) Possible Cause
Insufficient number of threads on the lead.
Solution
Use a tap with the correct lead.

(e) Possible Cause
Mis-alignment of the tap and the hole.
Solution
Care must be taken to align the tap with the hole before starting to tap.

(f) Possible Cause
Incorrect rake angle
Solution
Use the recommended tap for the material. See page 51/54.

Torn threads in the tapped hole

(a) Possible Cause
The flutes are clogged by chips.
Solution
Use a spiral point or a spiral flute tap.

(b) Possible Cause
Distortion of the walls in a thin walled workpiece.
Solution
Use a multi-fluted tap.

(c) Possible Cause
The threads on the tap are broken.
Solution
Use a tap which is in good condition.

(d) Possible Cause
Lack of/or the wrong type of lubricant.
Solution
Apply an adequate supply and the correct type of lubricant to the cutting area. See page 63/65.

(e) Possible Cause
Using the incorrect or unsuitable tap for the material.
Solution
Use the recommended tap for the material. See page 51/54.

(f) Possible Cause
Tap hitting the bottom of the hole.
Solution
Allow sufficient clearance at the bottom of the hole.

(g) Possible Cause
Incorrect rake angle.
Solution
Use the recommended tap for the material. See page 51/54.

Excessive Tap Wear

(a) Possible Cause
Mis-alignment of the tap and the hole.
Solution
Care must be taken to align the tap with the hole before starting to tap.

(b) Possible Cause
Lack of /or the wrong type of lubricant.
Solution
Apply an adequate supply and the correct type of lubricant to the cutting area.

(c) Possible Cause
The material is abrasive.
Solution
(i) Use the correct type of tap.
(ii) Use a surface treated tap.

(d) Using the incorrect tap.
Solution
(i) Use a tap with the correct lead.
(ii) Use a surface treated tap.

(e) Possible Cause
Incorrect rake angle
Solution
Use the recommended tap for the material. See page 51/54.

Over-Heating of tap

(a) Possible Cause
Lack of / or the wrong type of lubricant.
Solution
Apply an adequate supply and the correct type of lubricant to the cutting area.

(b) Possible Cause
The tap is too dull.
Solution
Use a tap which is in a good condition.

(c) Possible Cause
The wrong type of tap is used.
Solution
Use the recommended tap. See page 51/54.

(d) Possible Cause
Excessive tapping speed is applied.
Solution
Use the recommended tapping speed. See page 63/65.

Bell-Mouthed Tapped Hole

(a) Possible Cause
Mis-alignment of the tap and the hole.
Solution
Care must be taken to align the tap with the hole before starting to tap.

(b) Possible Cause
The workpiece is not rigidly held.
Solution
Secure the workpiece

(c) Possible Cause
Excessive pressure is applied when starting to tap.
Solution
Only sufficient pressure to initiate self-feeding should be applied.

(d) Possible Cause
Insufficient number of threads on the lead.
Solution
Use a tap with a longer lead.

(e) Possible Cause
The drilled hole is too small.
Solution
Use the recommended drill size. See page 55/61.

Over-size tapped hole

(a) Possible Cause
Using the incorrect tap.
Solution
Use the recommended tap. See page 51/54.

(b) Mis-alignment of the tap and the hole.
Solution
Care must be taken to align the tap with the hole before starting to tap.

(c) Possible Cause
Lack of / or wrong type of lubricant.
Solution

Apply an adequate supply and the correct type of lubricant to the cutting area.

(d) Possible Cause
Incorrect rake angle.
Solution
Use the recommended tap for the material. See page 51/54.

Tap binding in the hole

(a) Possible Cause
Using the incorrect tap.
Solution
Use the recommended tap.See page 51/54.

(b) Possible Cause
The drilled hole is too small.
Solution
Use the recommended drill size. See page 55/61.

(c) Possible Cause
Lack of / or the wrong type of lubricant.
Solution
Apply an adequate supply and the correct type of lubricant to the cutting area. See page 63/65.

(d) Possible Cause
The flutes are clogged with chips.
Solution
Use a spiral point or a spiral flute tap.

(e) Possible Cause
Incorrect rake angle.
Solution
Use the recommended tap for the material. See page 51/54.

Flutes clogged with chips

(a) Possible Cause
Using the incorrect tap.
Solution
Use a spiral point or spiral flute tap.

(b) Possible Cause
Lack of / or the wrong type of lubricant.
Solution
Apply an adequate supply and the correct type of lubricant to the cutting area.

Tap Breakage

(a) Possible Cause
Using the incorrect tap.
Solution
Use the recommended tap. See page 51/54.

(b) Possible Cause
The tap is too dull
Solution
Use a tap which is in good condition.

(c) Possible Cause
The drilled hole is too small.
Solution
Use the recommended tapping drill size. See page 55/61.

(d) Possible Cause
The drilled hole is too shallow.
Solution
Allow clearance at the bottom of the hole when drilling.

(e) Possible Cause
Mis-alignment of the tap and the hole.
Solution
Care must be taken to align the tap with the hole before starting to tap.

(f) Possible Cause
The flutes are clogged with chips.
Solution
Use a spiral point or spiral flute tap.

(g) Possible Cause
Excessive tapping speed is applied.
Solution
Use the recommended tapping speed. See page 63-65.

(h) Possible Cause
The tap holding device is not suitable.
Solution
Use the appropriate tapping attachment.

(i) Possible Cause
The work material is work hardened.
Solution
Use serial taps.

(j) Possible Cause
Lack of / or the wrong type of lubricant.

Lack of / or the wrong type of lubricant.

Solution

Apply an adequate supply and the correct type of lubricant to the chamfer lead of the tap.

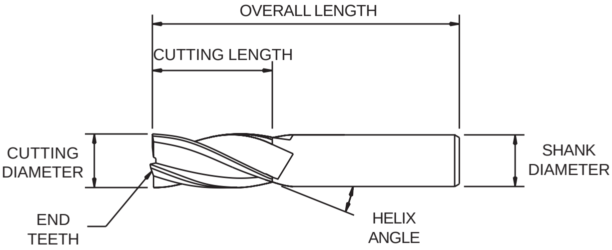
(k) Possible Cause

Incorrect rake angle.

Solution

Use the recommended tap for the material. See page 51/54.

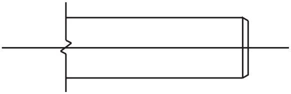
END MILL NOMENCLATURE



Shank Options

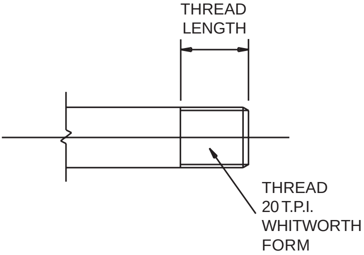
Plain Shank

Tolerance h7 on metric shank diameter (see page 121 for tolerance tables)



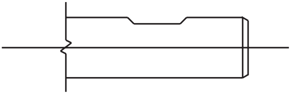
Threaded Shank

Tolerance h8 on metric / Fractional shank diameter



Flatted Shank

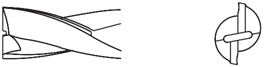
Tolerance h6 on metric shank diameter (see page 121 for tolerance tables



Typical End Mill Options

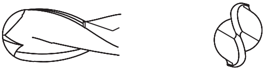
Two Flute End Mill

Tolerance e8 on cutting diameter (see page 121 for tolerance tables)



Ball Nose End Mill

Tolerance e8 on cutting diameter (see page 121 for tolerance tables)



Three Flute End Mill

Tolerance e8 on cutting diameter (see page 121 for tolerance tables)

Multi-Flute End Mill

Tolerance k10 on cutting diameter (see page 121 for tolerance tables)

Roughing End Mill

Tolerance k10 on cutting diameter (see page 121 for tolerance tables)

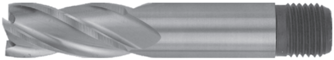
Two and Three Flute End Mills

Two and three flute end mills are shank type cutters with peripheral teeth and end teeth of the plunging type. Intended for general purpose use, they have right hand cutting, right hand helical teeth; they are used on keyway and closed slotting operations where the close minus tolerance of the cutting diameter allows slot widths to be produced in one pass. These cutters are also extensively used when profiling and end milling aluminium alloys, due to the greater chip space required by this material.

Ball Nose Two Flute End Mills

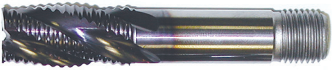
Ball nosed two flute end mills are manufactured to the same tolerances as the normal two flute end mill, and have a centre cutting ball end. They are used extensively in die making for cutting fillets, radiused slots, pocketing etc. These cutters have right hand cutting, right hand helical teeth.

Multi-Flute End Mills



Multi-flute end mills are shank type cutters with peripheral teeth and end teeth of the both plunging and non-plunging type. Designed for general purpose use they have right hand cutting, right hand helical teeth, and are used in stepping and profiling applications. They can also be used on slots where the plus tolerance of the cutting diameter is not critical.

Roughing End Mills



Shank type cutters with right hand cutting, right hand helical teeth on the periphery with roughing profile and with heavy duty end teeth. These cutters are robust and durable even under heavy cutting conditions on a wide range of materials. They are intended for rapid and heavy rates of stock removal where surface finish is of lesser importance. Available in coarse and fine pitch knuckle form and flat crest type.

HINTS FOR SUCCESSFUL END MILL USAGE

It is assumed that the workpiece clamping and machine size and power are adequate for the intended operation.

Always select the most suitable tool for the job on hand; a few minutes spent on selection can save hours of machining. Use roughing end mills when removing large amounts of stock; two or three flute end mills for deep slotting applications, for edge cutting and espically when machining light alloys. Use multi-flute end mills for edge cutting as well as for light finishing cuts.

Use threaded shank or flatted shank cutters where heavy stock removal and high tooth loads are involved. Plain shank cutters are particularly suitable for quick change CNC applications and for pre-setting off the machine.

Where possible check workpiece condition and hardness.

Check chucks and collects regularly ensuring that they are in good condition. Always clean cutter shanks and collets prior to assembly. Check that cutters are running true.

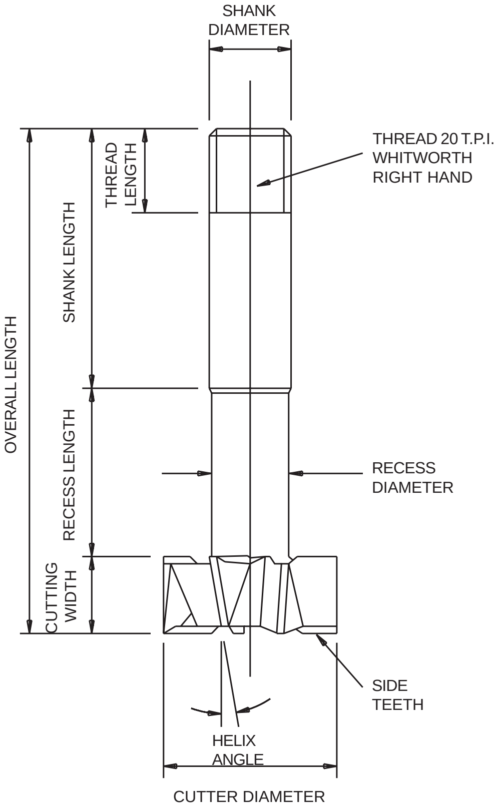
The most likely cause of cutter run-out is damaged chucks and collets.

Maintain cutters in a sharp condition to ensure maximum stock removal, surface finish and maximum power requirement.

Re-sharpen immediately when signs of wear are visible, since regrinding is then a relatively quick operation requiring little stock removal and with resulting increase in tool life. (See page 123 for resharpening details). Cutter storage is of paramount importance due to the brittle nature of the hardened cutting edges of all cutting tools. Poor storage often causes damage such as chipping of the cutting edges and breakage of corners, resulting in a tool which is useless. As in all machining operations cleaniless is essential.

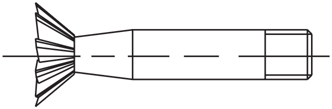
The best machining results are produced by cutters operating at the correct speed and feed to suit the material being worked. (See page 104/107 for technical data.)

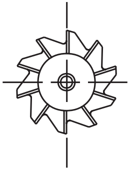
SHANK CUTTER NOMENCLATURE



Types of shank cutters

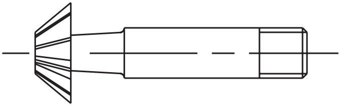
Dovetail Cutter

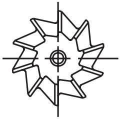




Tolerance js16 on cutting diameter
(see page 121 for tolerance tables)

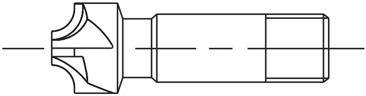
Inverted Dovetail Cutter

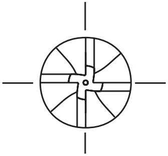




Tolerance js16 on cutting diameter
(see page 121 for tolerance tables)

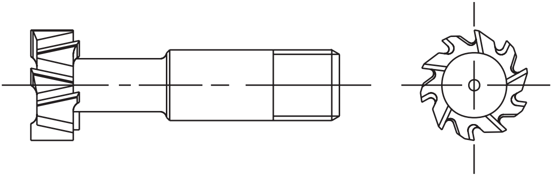
Corner Rounding Cutter





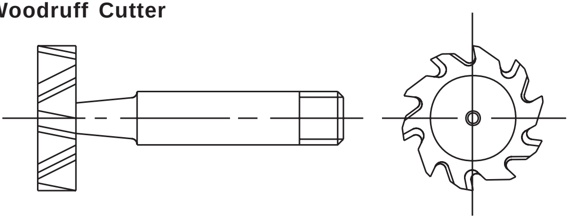
Tolerance H11 on radius and js14 on cutting tip
(see page 121 for tolerance tables)

T-Slot Cutter



Tolerance d11 on metric cutting diameter and width
Tolerance h12 on fractional cutting diameter and width
(see page 121 for tolerance tables)

Woodruff Cutter



Tolerance h11 on metric cutting diameter and e8 on width
(see page 121 for tolerance tables)

Tolerance on fractional diameter is

size	+0,381
	+0,127

and on width is

size	+0,000
	- 0,025

Dovetail Cutters

These angle cutters have right hand cutting straight teeth and non-plunging end teeth. They are used wherever dovetails or angles are required and are available in a range of angles and diameters.

Corner Rounding Cutters

Straight tooth cutters with right hand cutting teeth. Intended to produce a true convex up to 90° of arc.

T-Slot Cutters

Shank type cutters with right hand cutting alternate helical peripheral teeth as well as teeth on either face. Intended for opening out existing slots to form the T-slots used extensively on machine tables. They are produced in a range of diameters and widths to allow clearance on a standard range of bolt head sizes.

Woodruff Cutters

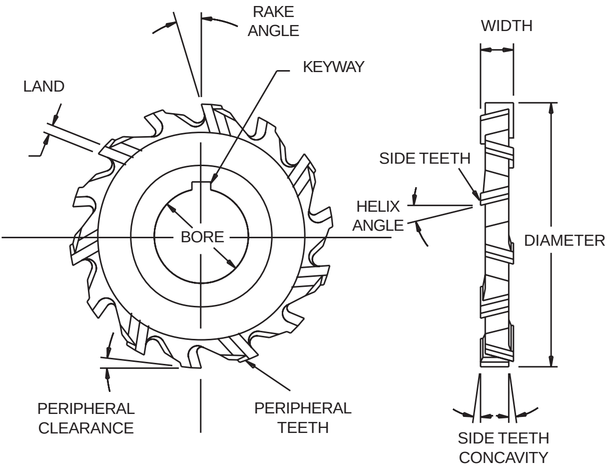
Shank type cutters with right hand cutting alternate helical peripheral teeth. Available in a range of diameters and widths. Designed to produce slots to suit standard woodruff keys.

HINTS FOR SHANK CUTTER USAGE

(See page 86 for hints on end mill usage)

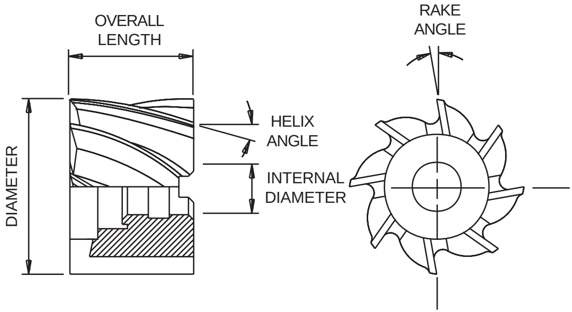
SIDE AND FACE CUTTER NOMENCLATURE

Side and Face Cutter- (Staggered Tooth shown)



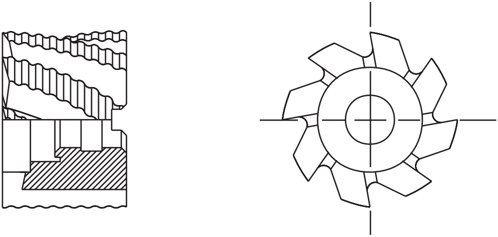
Tolerance js16 on metric cutting diameter and k11 on width (see page 121 for tolerance tables)

Plain Form



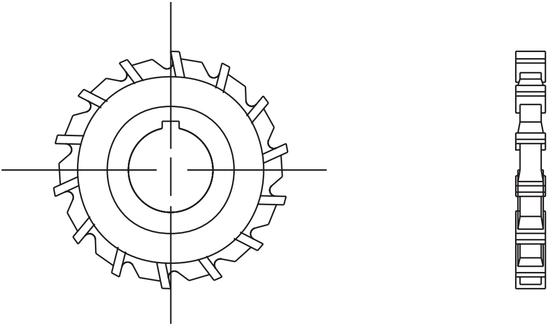
See page 98 for application.

Roughing Form



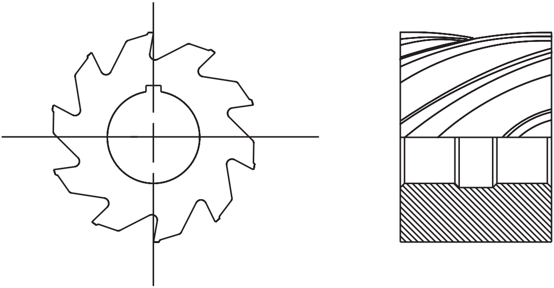
See page 98 for application.

Side and Face Cutter- Straight Tooth



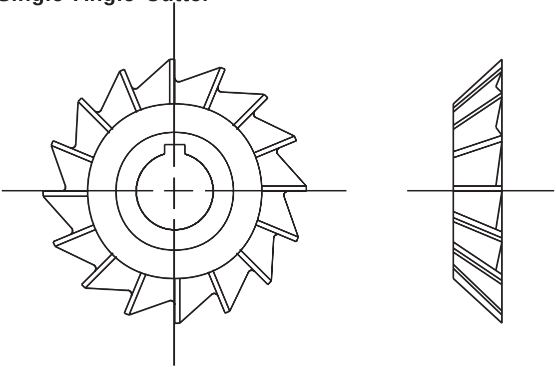
Tolerance js16 on metric/fractional cutting diameter and k11 on metric/fractional width
(see page 121 for tolerance tables)

Cylindrical Cutter



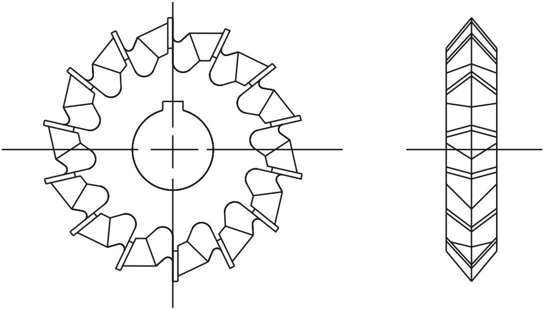
Tolerance js16 on cutting diameter and width
(see page 121 for tolerance tables)

Single Angle Cutter



Tolerance js16 on cutting diameter and js14 on width
(see page 121 for tolerance tables)

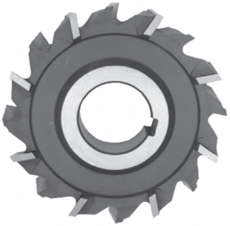
Double Angle Cutter



Tolerance js16 on cutting diameter and width
(see page 121 for tolerance tables)

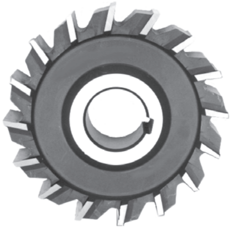
ARBOR MOUNTED CUTTER APPLICATIONS

Staggered Tooth Side and Face Cutters



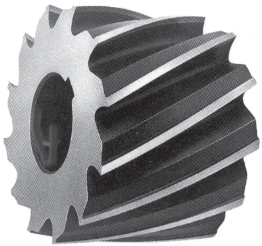
As the name suggests, side and face cutters have teeth on the periphery as well as on the sides, Designed with rugged alternate helical teeth, these cutters offer optimum performance when used for deep slotting with rapid stock removal; the cutting action of the alternate helical teeth combined with the coarse pitched side teeth giving excellent qualities of smooth cutting, efficient stock removal and good surface finish.

Straight Tooth Side and Face Cutters



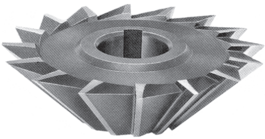
Intended for light cuts and shallow slotting operations, the straight tooth side and face cutter is often used in a straddle milling function where two parallel surfaces are machined simultaneously. It is considered to be a compromise tool due to the reduced cutting action of its straight teeth, which cause greater shock when meeting the workpiece than cutters with helical teeth.

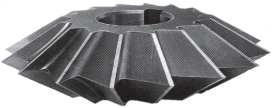
Cylindrical Cutters



Intended for medium/light surfacing cuts these helical cutters offer the benefits of shock reduction combined with a good cutting action.

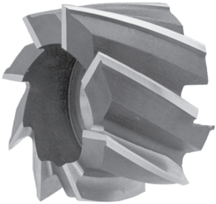
Angle Cutters





Produced with light duty straight teeth these cutters are used mainly for cutting dovetails, serrations and angled slots on less difficult materials.

Shell End Mills

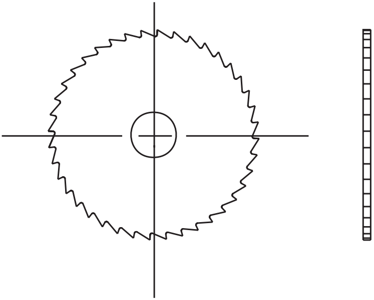


With helical peripheral teeth these cutters fillthe gap between normal shank cutters and the much larger facing cutters,this cuttter is better suited to light/medium cuts in a facing or stepping operation with its plain bore.

Shell End Mill (Roughing)

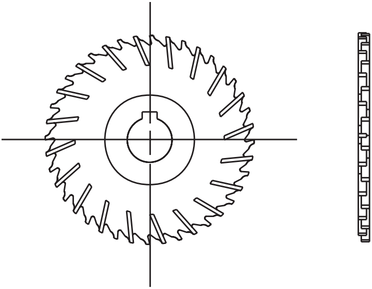
As the name implies, these cutters with their helical teeth and roughing profile are particularly efficient in areas where large volumes of stock must be removed at high speed and where tough materials are to be worked.

Slitting Saw - Plain



Tolerance js16 on cutting diameter and js10 on width
(see page 121 for tolerance tables)

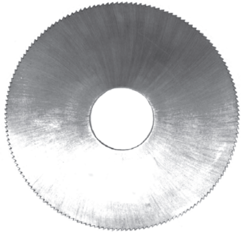
Slitting Saw - Side Chip Clearance



Tolernace js16 on cutting diameter and js10 on width
(see page 121 for tolerance tables)

SLITTING SAW APPLICATIONS

Slitting Saw - Plain

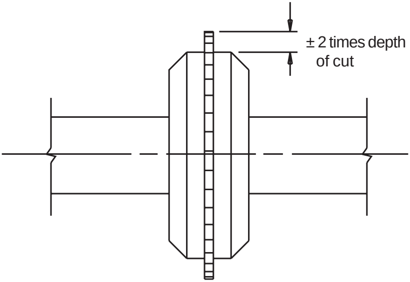


Intended for shallow cutting-off operations, these saws have straight teeth on the periphery and are tapered on width towards the bore to prevent binding. They are available in either coarse or fine pitch to suit the type and section of materials to be cut.

Slitting Saws - Side Chip Clearance

Intended for optimum production of deep narrow slots and for sawing operations, these saws have alternate helical teeth on the periphery combined with side teeth to ensure efficient stock removal, clean cutting action, and good surface finish.

HINTS FOR SUCCESSFUL SLITTING SAW USAGE



It is recommended that side plates be used with slitting saws.

HINTS FOR SUCCESSFUL ARBOR MOUNTED CUTTER USAGE

Some of the many factors governing efficient use of bore cutters are:-

1) Condition of machine
2) Machine power available
3) Machine capacity
4) Nature of the workpiece

Attention should be given to these factors prior to commencement. When using arbor mounted cutters the following points should be observed:-

Taper drive of arbor should be in good condition and fit correctly into machine drive.

Arbor and bushes should be kept in good and clean condition; dirty bushes cause run-out of cutters.

Arbors should be oiled and carefully stored when not in use; bent arbors are useless and expensive to replace.

Cutters should run true to prevent overloading of one or two teeth and extensive regrinding later.

Fit the cutter as closely as possible to the machine column with a support as near to the cutter as the workpiece will allow.

Running bushes and support bearings should be kept clean and in good running condition, particularly with regard to the bush faces. Lack of support will cause damage to the cutter and the workpiece. Always use correct lubricants.

Workpiece clamping should be rigid and able to withstand the forces acting upon it under the action of the cutter.

Select correct speeds and feeds for the cutter in use and the nature of the workpiece material and the size of the cut to be taken.

Use recommended coolants and direct flow to the point of cutting. Consult the coolant suppliers for specific recommendations. Adequate cooling is essential to prevent overheating of the cutter and failures associated with overheating.

Always use drive keys between the cutter and the arbor; friction between the cutter and the arbor bushes is seldom sufficient when cutters are under correct load.

Never force a cutter onto a arbor or over an ill-fitted key. Protect your hands by wrapping the cutter in a soft material when fitting or removing it from the arbor.

Due to the brittle nature of hardened tool steels it is not advisable to “remove” a cutter with a mallet once it has been tightened onto the arbor.

Maintain cutters in sharp condition. Regrind as soon as wear becomes apparent.

Store cutters carefully when not in use, using a light film of oil to prevent rusting.

Cleanliness of cutters and arbors is essential.

Use helically fluted cutters wherever possible to minimise shock as teeth contact the workpiece.

CUTTER TECHNICAL DATA

MATERIAL TYPE	GRADE	HARDNESS HB	TENSILE STRENGTH N / mm²
CARBON STEEL	FREE CUTTING	150	510
	0.3 to 0.4% Carbon	170	580
	0.3 to 0.4% Carbon	248	830
	0.4 to 0.7% Carbon	206	675
	0.4 to 0.7% Carbon	286	970
ALLOY STEEL		248	833
		330	1137
		381	1265
STAINLESS STEEL	Martensitic: Free Cutting	248	833
	Std. Grade	248	833
	Austenitic: Free Cutting	As Supplied	
	Std. Grade		
NIMONIC ALLOYS	Wrought	300	1030
	Cast	350	1200
TITANIUM	Titanium Comm: Pure	170	510
	Titanium Comm: Pure	200	660
	Titanium Comm: Pure	275	940
	Titanium Alloyed	340	1170
	Titanium Alloyed	350	1200
	Titanium Alloyed	380	1265
TOOL STEEL	HSS Standard Grades	225	735
	HSS Cobalt Grades	250	830
	Hot Working Steel	250	830
	Cold Working Steel	250	830
CAST IRONS	Grey, Malleable	240	800
	Hardened	330	1137

PERIPHERAL SPEED RANGE Refer to explanatory notes on page 32, 33				† CUTTING ANGLES		
TYPE *A	**TYPE *B**	**TYPE *C**	**TYPE *D**	**PRIMARY CLEARANCE**	**SECONDARY CLEARANCE**	**RADIAL RAKE**
30-40 24-32 18-25 24-32 16-25	28-40 24-32 18-25 24-32 16-20	24-32 20-26 14-20 20-26 12-20	30-40 24-32 18-25 24-32 16-25	8° - 20°	Add 10° to primary	9° - 14°
16-20 12-18 9-15	16-20 12-18 8-14	12-16 10-15 8-12	16-20 10-16 8-12	8° - 20°	Add 10° to primary	9° - 14°
10-20 5-10 10-20 5-10	12-16 5-10 12-16 5-10	8-15 4-8 8-15 4-8	10-20 5-10 10-20 5-10	8° - 20°	Add 10° to primary	9° - 14°
4-8	5-10	3-7	4-8	8° - 20°	Add 10° to primary	9° - 14°
7-12	5-12	5-10	7-12	8° - 20°	Add 10° to primary	9° - 14°
10-20 10-16 10-16 10-16	10-20 10-20 10-16 10-16	8-15 8-13 8-13 8-13	10-20 10-16 10-16 10-16	8° - 20°	Add 10° to primary	9° - 14°
16-20 12-16	16-20 10-14	12-16 10-12	20-28 16-22	8° - 20°	Add 10° to primary	9° - 14°

cont on page 106

CUTTER TECHNICAL DATA (cont)

MATERIAL TYPE	GRADE	HARDNESS HB	TENSILE STRENGTH N / mm²
ALUMINIUM ALLOYS	Wrought Wrought Cast	55 110 100	
COPPER ALLOYS	Brass : Free Cutting Low Leaded Bronze: Silicon Manganese Aluminium Phospor Copper	As Supplied	
PLASTICS		As Supplied	

Explanatory Notes

***Cutter types**

TYPE	CUTTER RANGE
A	End mills (2, 3 & Multi-Flute) T - Slot Cutters Dovetail & Inverted Dovetail Cutters Woodruff Cutters Corner Rounding Cutters
B	Side and Face Cutters Single and Double Angle Cutters Slitting Saws
C	Shell End Mills - Plain Tooth

PERIPHERAL SPEED RANGE Refer to explanatory notes on page 32, 33				† CUTTING ANGLES		
TYPE *A	TYPE *B	TYPE *C	TYPE *D	PRIMARY CLEARANCE	SECONDARY CLEARANCE	RADIAL RAKE
200-1500 100-250 40-100	120-180 100-180 50-70	50-180 50-100 30-80		10° - 20°	Add 10° to primary	20° - 28°
40-70 50-80 40-70 25-45	35-45 45-70 35-45 20-40	30-60 40-65 30-60 20-35		8° - 20°	Add 10° to primary	9° - 14°
15-25 15-25	15-25 15-25	12-20 12-20		8° - 20°	Add 10° to primary	9° - 14°
40-70	35-45	30-60		10° - 20°	Add 10°	20° - 28°
50-200	50-200			10° - 20°	Add 10° to primary	9° - 14°

***Cutter types** (cont)

TYPE	CUTTER RANGE
D	Shell End Mills - Roughing
Note:	For Roughing End Mills see page 85.

† Cutting Angles

Use higher angles for smaller diameters, reducing proportionately for larger diameters.

END MILLS: Feeds Per Tooth Sz (mm)

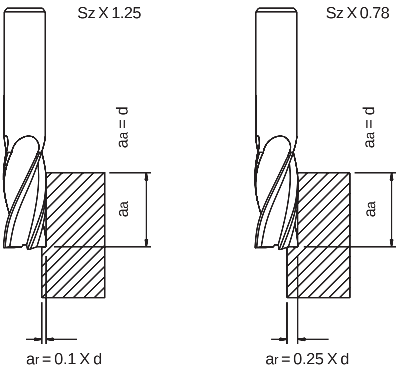


Table Shows Sz Values					
End Mill	Carbon Steels	Alloy Steels	Stainless Steels	Nimonic Alloys	Titanium
3	0.010	0.010	0.010	0.008	0.010
4	0.015	0.015	0.015	0.012	0.015
5	0.018	0.018	0.018	0.014	0.018
6	0.022	0.022	0.022	0.018	0.022
8	0.030	0.030	0.030	0.024	0.030
10	0.036	0.036	0.036	0.029	0.036
12	0.044	0.044	0.044	0.036	0.044
14	0.051	0.051	0.051	0.040	0.051
16	0.058	0.058	0.058	0.046	0.058
18	0.065	0.065	0.065	0.052	0.065
20	0.073	0.073	0.073	0.058	0.073
22	0.080	0.080	0.080	0.064	0.080
25	0.090	0.090	0.090	0.072	0.090
28	0.102	0.102	0.102	0.081	0.102
30	0.110	0.110	0.110	0.088	0.110
32	0.116	0.116	0.116	0.092	0.116
35	0.130	0.130	0.130	0.104	0.130
40	0.130	0.130	0.130	0.104	0.130
50	0.130	0.130	0.130	0.104	0.130

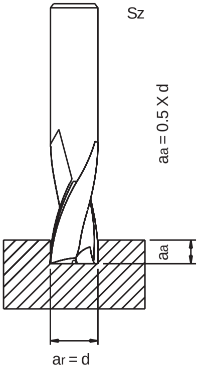


Table Shows Sz Values

Tool Steels	Cast Irons	Manganese Steels	Aluminium Alloys	Copper Alloys
0.009	0.010	0.008	0.013	0.013
0.013	0.016	0.012	0.019	0.019
0.016	0.022	0.014	0.023	0.023
0.020	0.028	0.018	0.028	0.028
0.027	0.036	0.024	0.039	0.039
0.032	0.040	0.029	0.046	0.046
0.040	0.045	0.036	0.057	0.057
0.046	0.056	0.040	0.066	0.066
0.052	0.064	0.046	0.075	0.075
0.058	0.070	0.052	0.085	0.085
0.065	0.080	0.058	0.092	0.092
0.072	0.088	0.064	0.104	0.104
0.080	0.095	0.072	1.117	0.117
0.091	0.110	0.081	0.132	0.132
0.100	0.120	0.088	0.143	0.143
0.104	0.127	0.092	0.150	0.150
0.117	0.142	0.104	0.170	0.170
0.117	0.142	0.104	0.170	0.170
0.117	0.142	0.104	0.170	0.170

ROUGHING END MILLS:
Peripheral Speed (m/min)
Feed Per Tooth Sz (mm)

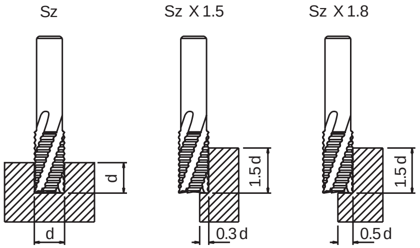


Table Shows Sz Values

End Mill Size	Material Group			
	1	2	3	4
6	0.008	0.008	0.009	0.010
8	0.013	0.013	0.015	0.015
10	0.017	0.020	0.020	0.021
12	0.023	0.025	0.025	0.033
14	0.026	0.030	0.030	0.037
16	0.030	0.038	0.038	0.044
22	0.032	0.040	0.040	0.048
25	0.035	0.042	0.042	0.050
28	0.035	0.045	0.042	0.050
30	0.040	0.045	0.045	0.056
32	0.042	0.050	0.050	0.064
35	0.013	0.013	0.015	0.015
38	0.045	0.057	0.057	0.070
40	0.045	0.057	0.057	0.070
45	0.047	0.059	0.060	0.075
50	0.060	0.074	0.075	0.090

Peripheral Speeds		
Material Group	Material Types	Cutter Speed (m/min)
1	Steels up to 500N/mm² Malleble Cast Iron up to 120 HB	28 - 40
2	Steels of 500 - 800 N/mm² Non - Alloyed Tool Steels Pure Titanium	24 - 32
3	Steels of 800 - 1200 N/mm² Hot Working Steels Cast Iron of 120 - 180 HB	18 - 25
4	Stainless Steels Titanium Alloys (Annealed) Cast Iron of more than 180 HB	12- 18
5	Titanium Alloys (Hardened)	7 - 12
6	Brass and Bronze (Cast)	35 - 45
7	Brass and Bronze (Rolled)	45 - 70
8	Plastics and similar	200 - 250

Table Shows Sz Values			
Material Group			
5	6	7	8
0.013	0.008	0.006	0.006
0.020	0.012	0.009	0.009
0.030	0.017	0.013	0.012
0.037	0.024	0.016	0.013
0.047	0.026	0.021	0.015
0.053	0.033	0.024	0.019
0.060	0.038	0.025	0.022
0.063	0.040	0.028	0.025
0.065	0.040	0.028	0.025
0.068	0.040	0.030	0.028
0.080	0.044	0.036	0.035
0.020	0.012	0.009	0.009
0.086	0.048	0.040	0.035
0.090	0.048	0.040	0.038
0.094	0.048	0.042	0.040
0.119	0.060	0.052	0.047

SIDE AND FACE CUTTERS - Staggered Tooth: Feed Per Tooth (mm)

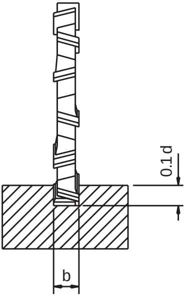


Table Shows Sz Values

Cutter Diameter	Cutter Width		Material Group		
	over	to	1	2	3
63	3	10	0.050	0.051	0.051
	10	18	0.052	0.054	0.054
80	4	12	0.063	0.063	0.070
	12	20	0.064	0.064	0.070
100	5	14	0.069	0.069	0.070
	14	25	0.070	0.069	0.070
125	7	16	0.077	0.078	0.080
	16	28	0.078	0.078	0.080
160	7	18	0.088	0.090	0.100
	18	32	0.090	0.090	0.190
200	8	18	0.093	0.093	0.194
	18	32	0.101	0.101	0.102
250	8	18	0.107	0.107	0.110
	18	32	0.105	0.105	0.106

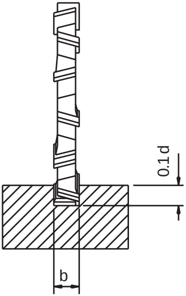


Table Shows Sz Values

Material Group				
4	5	6	7	8
0.050	0.051	0.050	0.046	0.020
0.052	0.053	0.052	0.048	0.020
0.063	0.063	0.063	0.056	0.020
0.063	0.063	0.063	0.056	0.020
0.070	0.070	0.070	0.062	0.020
0.070	0.070	0.070	0.070	0.020
0.078	0.080	0.080	0.080	0.020
0.078	0.080	0.080	0.080	0.020
0.090	0.090	0.090	0.090	0.020
0.090	0.090	0.090	0.090	0.020
0.093	0.094	0.093	0.093	0.020
0.102	0.102	0.101	0.101	0.020
0.108	0.110	0.108	0.108	0.020
0.104	0.105	0.104	0.104	0.020

SHELL END MILLS:
Feed Per Tooth (mm)

Plain Tooth

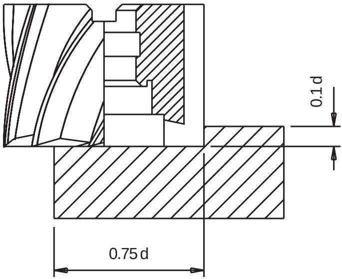


Table Shows Sz Values

Type	Cutter Diameter	Material Group		
		1	2	3
PLAIN	40	0.080	0.080	0.080
	50	0.080	0.080	0.080
	63	0.100	0.100	0.100
	80	0.100	0.100	0.100
	100	0.100	0.100	0.100
	125	0.100	0.100	0.100
	160	0.105	0.105	0.105
ROUGHING	40	0.060	0.060	0.060
	50	0.070	0.070	0.070
	63	0.075	0.080	0.070
	80	0.100	0.100	0.100
	100	0.110	0.110	0.110
	125	0.115	0.115	0.115
	160	0.120	0.120	0.125

Roughing Form

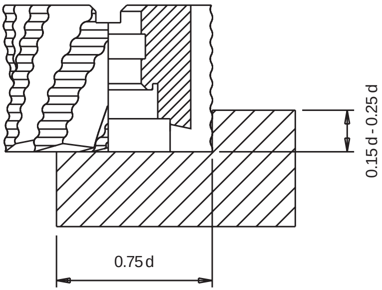


Table Shows Sz Values				
Material Group				
4	5	6	7	8
0.080	0.080	0.080	0.080	0.022
0.080	0.080	0.080	0.080	0.022
0.100	0.100	0.100	0.100	0.022
0.100	0.100	0.100	0.100	0.022
0.100	0.100	0.100	0.100	0.022
0.100	0.100	0.100	0.100	0.022
0.105	0.105	0.105	0.105	0.022
0.060	0.060	0.060	0.060	0.022
0.075	0.075	0.075	0.075	0.028
0.080	0.080	0.080	0.080	0.031
0.100	0.100	0.100	0.100	0.039
0.110	0.110	0.110	0.110	0.039
0.115	0.115	0.115	0.115	0.042
0.120	0.120	0.120	0.120	0.044

Speed and Feed Formulae

$$v = \frac{D.\pi.rpm}{1000}$$

$$Sz = \frac{S^1}{rpm.Z}$$

$$rpm = \frac{V.\ 1000}{\pi.D}$$

$$Sn = \frac{S^1}{rpm}$$

$$S^1 = Sz.\ Z.\ rpm$$

$$V = \frac{a.\ b.\ S^1}{1000}$$

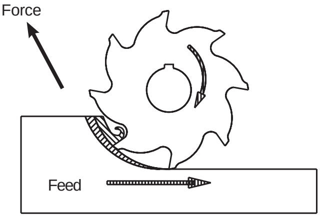
p = 3.1416

v	=	speed (m/min)
D	=	cutter diameter (mm)
rpm	=	revolutions/min
Sn	=	feed/revolution (mm)
S^1	=	feed/minute (mm)
Sz	=	feed/tooth (mm)
Z	=	number of teeth on cutter
V	=	chip volume (cm^3/min)
a	=	depth of cut (mm)
b	=	length of cut (mm)

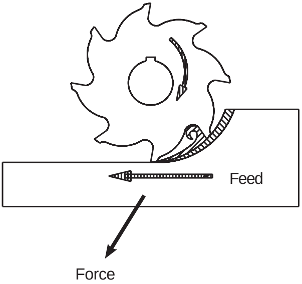
CLIMB OR CONVENTIONAL MILLING

From the very beginning of the milling process, it was found practical to always rotate the end mill in the opposite direction to the feed of the workpiece. This is termed conventional milling.

In conventional milling the end mill engages the workpiece at the bottom of the cut. The end mill teeth slide along until sufficient pressure builds up to break through the surface of the work. This sliding action under pressure tends to abrade the periphery of the end mill with resulting dulling. Also in horizontal conventional milling, the cutting action has a tendency to lift the workpiece, fixture and table from their bearings. In recent years, milling machines have been greatly improved through backlash elimination and greater rigidity so that climb milling is now possible. Climb milling improves surface finish and increases tool life.



Conventional Milling



Climb Milling

In climb milling the end mill rotates in the direction of the feed. The tooth meets the work at the top of the cut at the thickest portion of the chip. This provides instant engagement of the end mill with the workpiece producing a chip of definite thickness at the start of the cut without the rubbing action resulting from conventional milling. It further permits the gradual disengagement of the teeth and work so that feed marks are largely eliminated.

Climb milling will often provide better product finish, permit greater feed per tooth and prolong the cutter life per sharpening. It is particularly desirable to climb mill such materials as heat treated alloy steels and non-free machining grades of stainless steel for better tool life and to reduce work hardening. It is not recommended on material having a hard scale, such as cast or scaly forged surfaces, because abrasion would quickly ruin the cutting edges. Also some very soft steels do not lend themselves to climb milling because of their tendency to drag and tear.

Climb milling cannot be applied to every milling operation and should not be attempted if the material and the machine setup are not adapted to this type of milling.

PROBLEM SOLVING

Milling problems are often caused by one or more of the following factors, which should be carefully checked in a systematic and logical manner.

Speeds and Feeds
See page 104/107 for recommendations.

Coolants
Seek advice from your supplier.

Cutter Selection
Always select the correct type and quality of cutter to suit the application.

Arbors
Straightness/runout/size/wear/damage
Bushing-wear/damage.

Re-sharpening
Clearance angles. See page 123.
Runout
Burning/overheating
Surface finish

Milling Machines
Slides and gib strips
Lead screws and nuts
Backlash elimination
Attachments
Defective workheads
Worn tailstocks
Worn centres

Workholding
Workdolder condition
Workholder suitability
Workholder alignment
Workholder rigidity

Workpiece Condition
Machine suitability
Material specifications
Material hardness
Material surface conditions
Machining characteristics

Cutter Holders
Collets
Chucks
Draw bars
Runout
Damage

Tolerances in μm = 1 micron (1/1000mm)

	DIAMETER OR WIDTH							
Tol.	≤ 3mm	3 to 6mm	6 to 10mm	10 to 18mm	18 to 30mm	30 to 50mm	50 to 80mm	80 to 120mm
d11	-20 -80	-30 -105	-40 -130	-50 -160	-65 -195	-80 -240	-100 -290	-120 -340
e8	-14 -28	-20 -38	-25 -47	-32 -59	-40 -73	-50 -89	-60 -106	-72 -126
h6	0 -6	0 -8	0 -9	0 -11	0 -13	0 -16	0 -19	0 -22
h7	0 -10	0 -12	0 -15	0 -18	0 -21	0 -25	0 -30	0 -35
h8	0 -14	0 -18	0 -22	0 -27	0 -33	0 -39	0 -46	0 -54
h11	0 -60	0 -75	0 -90	0 -110	0 -130	0 -160	0 -190	0 -220
h12	0 -100	0 -120	0 -150	0 -180	0 -210	0 -250	0 -300	0 -350
js10	+20 -20	+24 -24	+29 -29	+35 -35	+42 -42	+50 -50	+60 -60	+70 -70
js14	+125 -125	+150 -150	+180 -180	+215 -215	+260 -260	+310 -310	+370 -370	+435 -435
js16	+300 -300	+375 -375	+450 -450	+550 -550	+650 -650	+800 -800	+950 -950	+1100 -1100
k10	+40 0	+48 0	+58 0	+70 0	+84 0	+100 0	+120 0	+140 0
k11	+60 -0	+75 -0	+90 -0	+110 -0	+130 -0	+160 -0	+190 -0	+220 -0
H7	+10 0	+12 0	+15 0	+18 0	+21 0	+25 0	+30 0	+35 0
H11	+60 0	+75 0	+90 0	+110 0	+130 0	+160 0	+190 0	+220 0

DIFFICULT TO MACHINE MATERIALS

There are number of materials which are generally regarded as being difficult to machine. In general terms the material being worked is considered to be difficult when it does not respond readily to normal machining techniques. Among these “difficult’ materials are aluminium alloys, stainless steel and work hardening steels.

Aluminium Alloys require relatively high speeds and feeds. They respond best to cutters with few teeth and correspondingly wide chip spaces, and can be worked very effectively by using two flute end mills, which have the advantage of fewer teeth engaged in the cut. In many cases coolant may not be needed to cool the cutter although it is of benefit in lubricating and particularly in removing chips. Climb milling gives definite advantages and shows significant benefits where a good quality surface finish is needed. These materials can be worked quite effectively with regular tooling, although benefits would be obtained from custom tools in the event of large volume production being the norm.

Stainless Steels require lower speeds and higher feed rates and often benefits are obtained from using corner radii and chamfers. These materials respond well to the conventional cutting method but rigidity of machine and setup are critical. Light finishing cuts are to avoided but where necessary should be taken at a feed rate as high as possible to meet with surface finishing requirements. It is crucial that these materials be “worked”, and “rubbing” of the cutter against the workpiece should be avoided. Selection of speed and feed rates is of great importance. Coolant must be used in large volume and be directed at the cutting area. Benefits are often obtained from a higer coolant concentration or from using cutting oils.

Work Hardening Steels such as some stainless and manganese steels can be successfully machined by using the same techniques as described for stainless steels above.

RESHARPENING AND CARE OF MILLING CUTTERS

The productivity of a milling machine depends to a large degree on the efficiency of the milling cutter. Best results in both production and cutter life are obtained by sharpening cutters correctly and carefully, and by taking proper care in handling and storage. A correctly sharpened cutter requires less driving power, produces better quality work and gives longer service than an incorrectly or hastily sharpened cutter.

The following factors should be considered:-

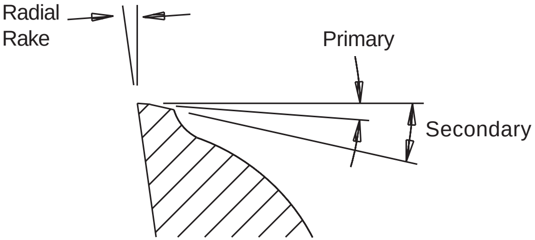
Correct handling and storage to prevent damage.

Restoration of the cutting edges to their original geometry using correct procedures.

Suitable wheel selection to ensure correct surface finish and stock removal. Consult wheel suppliers for specific recommendations.

Remember that milling cutters are precision tools and must be handled carefully. Damage due to incorrect handling or storage can be seen as a flaw upon the milled surface of a workpiece. Grinding should be needed only as a result of dulling due to use. Regrinding to remove damage caused by rough handling must be considered to be a wasted process which reduces the life of a cutter.

Correct clearance angles and radial rakes can be obtained from data given on page 104-107.



GENERAL INFORMATION

INCH-MILLIMETER CONVERSION TABLE

	0" mm	1" mm	2" mm	3" mm
0		25.400	50.800	76.200
1/64	0.397	25.797	51.197	76.597
1/32	0.794	26.194	51.594	76.994
3/64	1.191	26.591	51.991	77.391
1/16	1.588	26.988	52.388	77.788
5/64	1.984	27.384	52.784	78.184
3/32	2.381	27.781	53.181	78.581
7/64	2.778	28.178	53.578	78.978
1/8	3.175	28.575	53.975	79.375
9/64	3.572	28.972	54.372	79.772
5/32	3.969	29.369	54.769	80.169
11/64	4.366	29.766	56.166	80.566
3/16	4.762	30.162	55.562	80.962
13/64	5.159	30.599	55.959	81.359
7/32	5.556	30.956	56.356	81.756
15/64	5.953	31.353	56.753	82.153
1/4	6.350	31.750	57.150	82.550
17/64	6.747	32.147	57.547	82.947
9/32	7.144	32.544	57.944	83.344
19/64	7.541	32.941	58.341	83.741
5/16	7.938	33.338	58.738	84.138
21/64	8.334	33.734	59.134	84.534
11/32	8.731	34.131	59.531	84.931
23/64	9.128	34.528	59.928	85.328
3/8	9.525	34.925	60.325	85.725
25/64	9.922	35.322	60.722	86.122
13/32	10.319	35.719	61.119	86.519
27/64	10.716	36.116	61.516	86.916
7/16	11.112	36.512	61.912	87.312
29/64	11.509	36.909	62.309	87.709
15/32	11.906	37.306	62.706	88.106
31/64	12.303	37.703	63.103	88.503

INCH-MILLIMETER CONVERSION TABLE (cont)

	0" mm	1" mm	2" mm	3" mm
1/2	12.700	38.100	63.500	89.900
33/64	13.097	38.497	63.897	89.297
17/32	13.494	38.894	64.294	89.694
35/64	13.891	39.291	64.691	90.091
9/16	14.288	39.688	65.088	90.488
37/64	14.684	40.084	65.484	90.884
19/32	15.081	40.481	65.881	91.281
39/64	15.748	40.878	66.278	91.678
5/8	15.875	41.275	66.675	92.075
41/64	16.271	41.671	67.071	92.471
21/32	16.668	42.068	67.468	92.868
43/64	17.066	42.466	67.866	92.266
11/16	17.462	42.862	68.262	93.662
45/64	17.859	43.859	68.859	94.859
23/32	18.256	43.656	69.056	94.456
47/64	18.653	44.053	69.453	94.853
3/4	19.050	44.450	69.850	95.250
49/64	19.447	44.847	70.247	95.647
25/32	19.844	45.244	70.644	96.044
51/64	20.241	45.641	71.041	96.441
13/16	20.638	46.038	71.438	96.838
53/64	21.034	46.434	71.834	97.234
27/32	21.431	46.831	72.231	97.631
55/64	21.828	47.228	72.628	98.028
7/8	22.225	47.625	73.025	98.425
57/64	22.622	48.022	73.422	98.282
29/32	23.019	48.019	73.019	99.019
59/64	23.416	48.816	74.216	99.616
15/16	23.812	49.212	74.612	100.012
61/64	24.209	49.609	75.009	100.409
31/32	24.606	50.006	75.406	100.806
63/64	25.003	50.403	75.803	101.203

APPROXIMATE HARDNESS AND TENSILE STRENGTH CONVERSIONS

HRB	HRC	HV	HB	TENSILE STRENGTH	
				Tons/ inch²	MPa or N/mm²
50	—	95	90	21	320
55	—	100	100	23	350
60	—	110	105	25	390
65	—	120	110	27	420
70	—	130	120	29	150
75	—	140	130	31	180
80	—	150	140	34	520
85	—	165	160	37	570
90	—	185	175	40	620
95	—	205	195	45	690
100	20	230	220	50	770
—	22	240	230	53	820
—	24	255	240	56	860
—	26	265	250	59	910
—	28	280	265	62	960
—	30	295	280	65	1000
—	32	310	290	68	1050
—	34	325	310	72	1110
—	36	345	325	75	1150
—	38	360	345	78	1200
—	40	380	365	83	1280
—	42	405	385	88	1360
—	44	425	405	92	1420
—	46	450	430	96	1480
—	48	480	455	102	1540
—	50	505	480	108	1670
—	52	545	—	112	1720
—	54	580	—	117	1800
—	56	615	—	122	1890
—	58	655	—	130	2000
—	60	695	—	135	2100
—	64	790	—	150	2320
—	66	855	—	163	2510
—	68	940	—	179	2770
—	70	1075	—	197	3030
—	75	1480	—	—	—
—	80	1865	—	—	—

HRB = Hardness Rockwell B
HRC = Hardness Rockwell C
HV = Hardness Vickers. Also DPN, VPN, DPH, VPH
HB = Hardness Brinell. Also BHN

Note:

These values should be treated as approximate only and are suitable for calculating speeds and feeds or for general information purposes. Do not use for treated high speed steel.

HARDNESS CONVERSION CHART FOR HIGH SPEED STEEL

HV30	HRC
736	59-3/4
741	60
746	60-1/4
752	60-1/4
757	60-1/2
763	61
769	61
775	61-1/4
780	61-1/2
786	61-3/4
792	62
798	62-1/4
804	62-1/2
810	62-3/4
817	63
823	63-1/4
829	63-1/2
836	63-3/4
842	64
849	64-1/4

HV30	HRC
856	64-1/2
862	63-3/4
869	65
876	65-1/4
883	65-1/2
890	66
897	66
905	66-1/2
912	67
919	67
927	67-1/4
934	67-1/2
942	68
950	68
958	68-1/2
966	68-1/2
974	69
982	69-1/2
990	69-1/2
999	70

Typical hardness

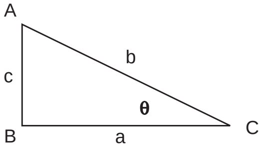
M2 823-876 HV30 - 63-65 HRC
M35 849-920 HV30 - 64-66 HRC
M42 897-966 HV30 - 66 - 68-1/2 HRC

Depending on the nature of the tool these hardnesses may be varied, particularly in the case of special tools where different hardnesses may be specified.

Note:
Undue reliance should not be placed on a general conversion chart unless it has been tested for a particular material. The above chart applies specifically to High Speed Steel.

Trigonometry

Formulae for the solution of RIGHT ANGLED TRIANGLES

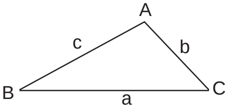


$$\text{Tan } \theta = \frac{\text{opposite}}{\text{adjacent}} = \frac{c}{a}$$

$$\text{Sin } \theta = \frac{\text{opposite}}{\text{hypotenuse}} = \frac{c}{b}$$

$$\text{Cos } \theta = \frac{\text{adjacent}}{\text{hypotenuse}} = \frac{a}{b}$$

Formulae for the solution of OBLIQUE ANGLED TRIANGLES



The Sine rule:

$$\frac{a}{\text{Sin A}} = \frac{b}{\text{Sin B}} = \frac{c}{\text{Sin C}}$$

The Cosine rule:

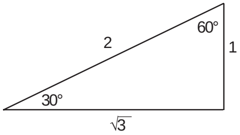
$a^2 = b^2 + c^2 - 2bc \quad \text{Cos A}$

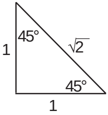
$b^2 = a^2 + c^2 - 2ac \quad \text{Cos B}$

$c^2 = a^2 + b^2 - 2ab \quad \text{Cos C}$

USEFUL VALUES IN TRIGNOMETRICAL RATIOS

For right angled triangles





ANGLES 30° - 45° - 60°

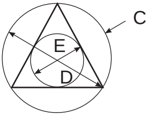
θ	Tan θ	Sin θ	Cos θ
30°	$\frac{1}{\sqrt{3}}$ = 0.577350	$\frac{1}{2}$ = 0.500000	$\frac{\sqrt{3}}{2}$ = 0.866025
45°	1	$\frac{1}{\sqrt{2}}$ = 0.707107	$\frac{1}{\sqrt{2}}$ = 0.707107
60°	$\sqrt{3}$ = 1.732051	$\frac{\sqrt{3}}{2}$ = 0.866025	$\frac{1}{2}$ = 0.500000

Useful formulae for Finding Dimensions of Circles, Squares, etc.

D is diameter of stock necessary to turn shape desired.
E is distance "across flats," or diameter of inscribed circle.
C is depth of cut into stock turned to correct diameter.

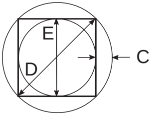
TRIANGLE

E	= side x 0.57735
D	= side x 1.1547 = 2E
Side	= D x 0.866
C	= E x 0.5 = D x 0.25



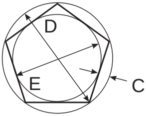
SQUARE

E	= side = D x 0.7071
D	= side x 1.4142 = diagonal
Side	= D x 0.7071
C	= D x 0.14645



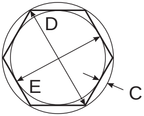
PENTAGON

E	= side x 1.3764 = D x 0.809
D	= side x 0.7013 = E x 1.2361
Side	= D x 0.5878
C	= D x 0.0955



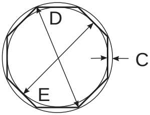
HEXAGON

E	= side x 1.7321 = D x 0.866
D	= side x 2 = E x 1.1547
Side	= D x 0.5
C	= D x 0.067



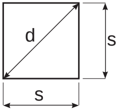
OCTAGON

E	= side x 2.4142 = D x 0.9239
D	= side x 2.6131 = E x 1.0824
Side	= D x 0.3827
C	= D x 0.038



Areas of Plane Figures

SQUARE



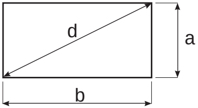
A = area

$A = \mathbf{S^2} = 1/2\, d^2$

$S = 0.7071d = \sqrt{A}$

$d = 1.414S = 1.414\sqrt{A}$

RECTANGLE



A = area

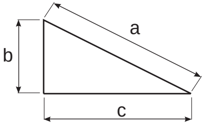
$A = ab = a\sqrt{d^2 - a^2} = b\sqrt{d^2 - b^2}$

$d = \sqrt{a^2 + b^2}$

$a = \sqrt{d^2 - b^2} = A \div b$

$b = \sqrt{d^2 - a^2} = A \div a$

RIGHT ANGLED TRIANGLE



A = area

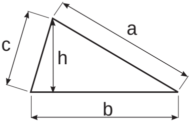
$A = \dfrac{\mathbf{bc}}{\mathbf{2}}$

$a = \sqrt{b^2 + c^2}$

$b = \sqrt{a^2 - c^2}$

$c = \sqrt{a^2 - b^2}$

ACUTE ANGLED TRIANGLE



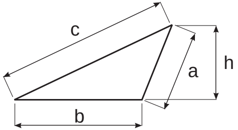
A = area

$$A = \frac{\mathbf{bh}}{\mathbf{2}} = \frac{b}{2}\sqrt{a^2 - \left(\frac{a^2 + b^2 - c^2}{2b}\right)^2}$$

if $S = \frac{1}{2}(a + b + c)$ then,

$$A = \sqrt{S(S - a)(S - b)(S - c)}$$

OBTUSE ANGLED TRIANGLE



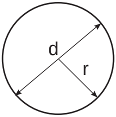
A = area

$$A = \frac{bh}{2} = \frac{b}{2}\sqrt{a^2 - \left(\frac{c^2 - a^2 - b^2}{2b}\right)^2}$$

if $S = \frac{1}{2}(a + b + c)$ then,

$$A = \sqrt{S(S-a)(S-b)(S-c)}$$

CIRCLE



A = area C = circumference

$A = \pi r^2 = 3.1416\ r^2$

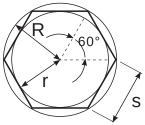
$A = \frac{\pi d^2}{4} = 0.7854\ d^2$

$C = 2\pi r = 6.2832r = 3.1416d$

$r = C \div 6.2832 = \sqrt{A \div 3.1416} = 0.564\sqrt{A}$

$d = C \div 3.1416 = \sqrt{A \div 0.7854} = 1.128\sqrt{A}$

REGULAR HEXAGON



A = area
R = radius of circumscribed circle
r = radius of inscribed circle
$A = 2.598S^2 = 2.598R^2 = 3.464r^2$
$R = S = 1.155r$
$r = 0.866S = 0.866R$

The construction of a regular hexagon forms six equilateral triangles,thus the area of the hexagon can also be found by calculating the area of the equilateral triangle and multiplying the result by six.

USEFUL FORMULAE

$$\text{rpm} = \text{Surface Speed (metres/min)} \div \frac{\text{Dia (mm)} \times \pi}{1000}$$

$$\text{Surface Speed (metres/min)} = \frac{\text{Dia (mm)} \times \pi}{1000} \times \text{rpm}$$

$$\text{Feed Rate (mm/rev)} = \frac{\text{Feed rate (mm/min}}{\text{rpm}}$$

Penetration rate (mm/min) = rpm X feed rate (mm/rev)

USEFUL TAPERS

Cone of	Included Angle			Angle with Centre Line		
1 in 2	28°	4′	20″	14°	2′	10″
2-1/2	22°	37′	12″	11°	18′	36″
1 in 3	18°	55′	28″	9°	27′	44″
3-1/2	16°	15′	38″	8°	7′	49″
1 in 4	14°	15′	0″	7°	7′	30″
4-1/2	12°	40′	50″	6°	20′	25″
1 in 5	11°	25′	16″	5°	42′	38″
5-1/2	10°	23′	20″	5°	11′	40″
1 in 6	9°	31′	36″	4°	45′	48″
6-1/2	8°	47′	52″	4°	23′	56″
1 in 7	8°	10′	16″	4°	5′	8″
7-1/2	7°	37′	43″	3°	48′	52″
1 in 8	7°	9′	10″	3°	34′	35″
8-1/2	6°	43′	58″	3°	21′	59″
1 in 9	6°	21′	34″	3°	10′	47″
9-1/2	6°	1′	32″	3°	0′	46″
1 in 10	5°	43′	31″	2°	51′	46″
1 in 11	5°	12′	18″	2°	36′	9″
1 in 12	4°	46′	19″	2°	23′	9″
1 in 13	4°	24′	16″	2°	12′	8″
1 in 14	4°	5′	26″	2°	2′	43″
1 in 15	3°	49′	6″	1°	54′	33″
1 in 16	3°	34′	48″	1°	47′	24″
1 in 17	3°	22′	9″	1°	41′	4″
1 in 18	3°	10′	58″	1°	35′	29″
1 in 19	3°	0′	54″	1°	30′	27″
1 in 20	2°	51′	52″	1°	25′	56″
1 in 25	2°	17′	31″	1°	8′	46″
1 in 30	1°	54′	36″		57′	18″
1 in 35	1°	38′	14″		49′	7″
1 in 40	1°	25′	56″		42′	58″
1 in 45	1°	16′	24″		38′	12″
1 in 48	1°	11′	37″		35′	48″
1 in 50	1°	8′	46″		34′	23″
1 in 55	1°	2′	29″		31′	14″
1 in 60		57′	17″		28′	39″

MORSE TAPERS AND BROWN & SHARPE TAPERS

Taper Number	Taper Per mm on dia.	Taper Per foot on dia.	Included Angle			Angle to Centre Line		
			Deg.	Mins.	Secs.	Deg.	Mins.	Secs.
Morse								
1	0.049881	0.59858	2	51	27	1	25	43
2	0.049951	0.59941	2	51	41	1	25	50
3	0.050196	0.60235	2	52	31	1	26	16
4	0.051938	0.62326	2	58	31	1	29	15
5	0.052626	0.63151	3	0	52	1	30	26
6	0.052137	0.62565	2	59	12	1	29	36
Brown & Sharpe								
4	0.041867	0.50240	2	23	54	1	11	57
5	0.041800	0.50160	2	23	41	1	11	50
7	0.041789	0.50147	2	23	39	1	11	49
9	0.041737	0.50085	2	23	28	1	11	44
10	0.051343	0.51612	2	27	50	1	13	55
11	0.041750	0.50100	2	23	30	1	11	45
12	0.041644	0.49973	2	23	08	1	11	34

Conversion Factors:
British - Metric

To convert	Multiply by
Inches to millimetres	25.40
Feet to metres	0.3048
Yards to metres	0.9144
Miles to kilometres	1.60934
Square inches to square centimetres	6.4516
Square feet to square metres	0.092903
Square yards to square metres	0.836127
Square miles to square kilometres	2.58999
Cubic inches to cubic centimetres	16.3871
Cubic feet to cubic metres	0.028317
Cubic yards to cubic metres	0.764555
Pints to litres	0.568261
Gallons to litres	4.54609
Ounces to grams	28.3495
Pounds to kilograms	0.453592
Tons to tonnes (1.000kg)	1.01605
Lb/sq.in. to kg/sq.m	703.070
Fahrenheit = 9/5°C+32	

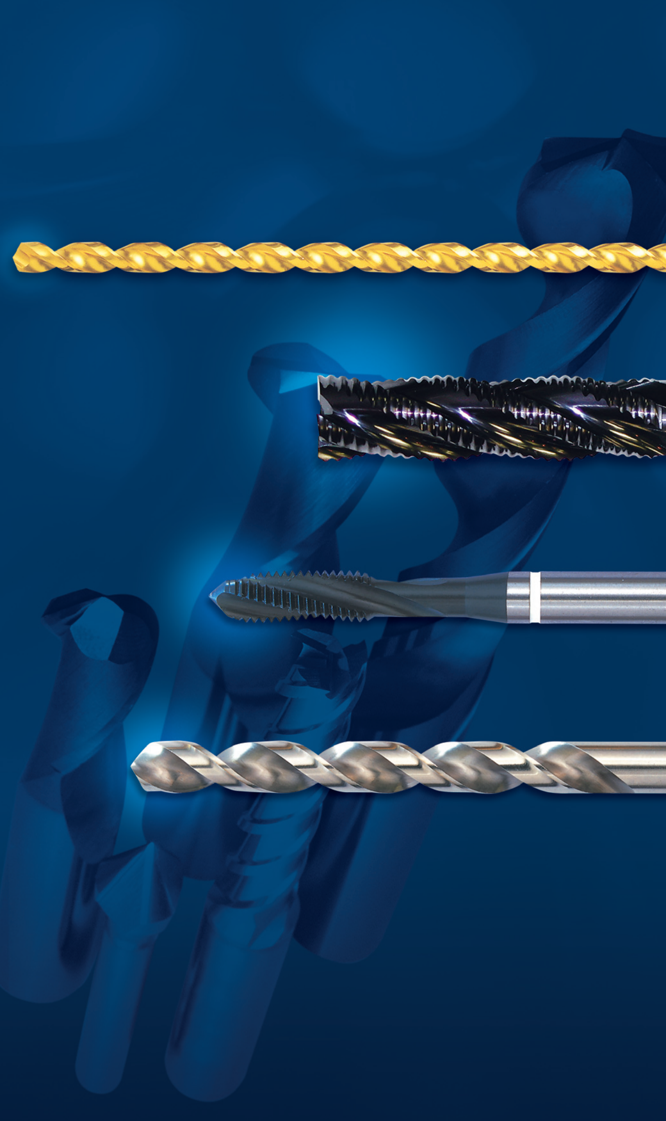
Conversion Factors:
Metric - British

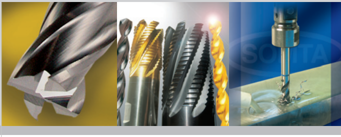
To convert	Multiply by
Millimetres to inches	0.0393701
Metres to feet	3.28084
Metres to yards	1.09361
Kilometres to miles	0.621371
Square centimetres to square inches	0.1550
Square metres to square feet	10.76391
Square metres to square yards	1.19599
Square kilometres to square miles	0.3861
Cubic centimetres to cubic inches	0.061024
Cubic metres to cubic feet	35.3147
Litres to pints	1.76
Litres to gallons	0.22
Grams to ounces	0.035274
Kilograms to pounds	2.20462
Tonnes to tons	0.984207
Kg/sq.mm to lb/sq.in.	0.001422
Centigrade (Celcius) = 5/9° (F-32)	

NUMBER AND LETTER DRILL SIZES

Decimal Equivalents

mm-Inch-Wire	Decimal Inch	mm-Inch-Wire	Decimal Inch	mm-Inch-Wire	Decimal Inch	mm-Inch-Wire	Decimal Inch
.1mm	.0039	45	.0820	5	.2055	29/64	.4531
.2mm	.0079	44	.0860	4	.2090	**15/32**	**.4688**
.3mm	.0118	43	.0890	3	.2130	12mm	.4724
80	.0135	42	.0935	**7/32**	.2188	31/64	.4844
79	.0145	**3/32**	.0938	2	.2210	**1/2**	**.5000**
1/64	.0156	41	.0960	1	.2280	13mm	.5118
.4mm	.0157	40	.0980	A	.2340	33/64	.5156
78	.0160	39	.0995	**15/64**	.2344	17/32	.5313
77	.0180	38	.1015	6mm	.2362	35/64	.5469
.5mm	.0197	37	.1040	B	.2380	14mm	.5512
76	.0200	36	.1060	C	.2420	**9/16**	**.5625**
75	.0210	**7/64**	.1094	D	.2460	37/64	.5781
74	.0225	35	.1100	**1/4 & E**	.2500	15mm	.5906
.6mm	.0236	34	.1110	F	.2570	19/32	.5938
73	.0240	33	.1130	G	.2610	39/64	.6094
72	.0250	32	.1160	**17/64**	.2656	**5/8**	**.6250**
71	.0260	3mm	.1181	H	.2660	16mm	.6299
.7mm	.0276	31	.1200	I	.2720	41/64	.6406
70	.0280	**1/8**	.1250	7mm	.2756	21/32	.6562
69	.0292	30	.1285	J	.2770	17mm	.6693
68	.0310	29	.1360	K	.2810	43/64	.6719
1/32	.0312	28	.1405	**9/32**	.2812	**11/16**	**.6875**
.8mm	.0315	**9/64**	.1406	L	.2900	45/64	.7031
67	.0320	27	.1440	M	.2950	18mm	.7087
66	.0330	26	.1470	**19/64**	.2969	23/32	.7188
65	.0350	25	.1495	N	.3020	47/64	.7344
.9mm	.0354	24	.1520	**5/16**	.3125	19mm	.7480
64	.0360	23	.1540	8mm	.3150	**3/4**	**.7500**
63	.0370	**5/32**	.1562	O	.3160	49/64	.7656
62	.0380	22	.1570	P	.3230	25/32	.7812
61	.0390	4mm	.1575	**21/64**	.3281	20mm	.7874
1mm	.0394	21	.1590	Q	.3320	51/64	.7969
60	.0400	20	.1610	R	.3390	**13/16**	**.8125**
59	.0410	19	.1660	**11/32**	.3438	21mm	.8268
58	.0420	18	.1695	S	.3480	53/64	.8281
57	.0430	**11/64**	.1719	9mm	.3543	27/32	.8438
56	.0465	17	.1730	T	.3580	55/64	.8594
3/64	.0469	16	.1770	**23/64**	.3594	22mm	.8661
55	.0520	15	.1800	U	.3680	**7/8**	**.8750**
54	.0550	14	.1820	**3/8**	.3750	57/64	.8906
53	.0595	13	.1850	V	.3770	23mm	.9055
1/16	.0625	**3/16**	.1875	W	.3860	29/32	.9062
52	.0635	12	.1890	**25/64**	.3906	59/64	.9219
51	.0670	11	.1910	10mm	.3937	**15/16**	**.9375**
50	.0700	10	.1935	X	.3970	24mm	.9449
49	.0730	9	.1960	Y	.4040	61/64	.9531
48	.0760	5mm	.1969	**13/32**	.4062	31/32	.9688
5/64	.0781	8	.1990	Z	.4130	25mm	.9843
47	.0785	7	.2010	**27/64**	.4219	63/64	.9844
2mm	.0787	**13/64**	.2031	11mm	.4331	**1"**	**1.0000**
46	.0810	6	.2040	7/16	.4375		





Manufacturers & Suppliers

of Drills, Reamers, End Mills,

Bore Cutters,

Taps & Dies, Toolbits,

Custom Tools and

Surface Coatings

Head Office And Surface Technology Division
Somta House, 290-294 Edendale Road, 3201
Private Bag X401, Pietermaritzburg 3200
South Africa

Tel: Factory: +27 33 3556600
Fax: Factory: +27 33 3940564
Tel: Sales: +27 33 3556798 (Local)
Fax: Sales: +27 33 3940564 (Local)
Tel: Sales: +27 33 3556600 (Exports)
Fax: Sales: +27 33 394 7509 (Exports)
Email: exports@somta.co.za

Technical Information:
Email: tech@somta.co.za
Toll Free Number: 0800-331-399
Technical Representative
Cell Number: 082 801 8930

Johannesburg Office (Local Sales)
43 Bisset Road, Hughes Ext. 7, Boksburg 1459
P.O. Box 14212, Witfield 1467
South Africa

Tel: +27 11 397 6696
Fax: +27 11 397 6720
Sharecall: 0860 104367
Email: jhbsales@somta.co.za
Technical Representatives Cell Numbers:
082 806 8263, 082 806 7021, 082 800 5259,
082 906 3486, 082 806 8202, 082 803 0505

Port Elizabeth & CapeTown Sales Office
13 Richard Road, Broadwood, Port Elizabeth
Tel: +27 41 368 2060, Fax: +27 41 368 2060
Technical Representative
Cell Number: 082 803 0506

SOMTA
World Class Cutting Tools

www.somta.co.za

EST 1954

balzers
Products coated with Balzers Technology

ISO 9001 SABS