OSG

shaping your dreams

SOMTA

World Class Cutting Tools

OSG GROUP COMPANY

SOMTA USER GUIDE

DISTRIBUTOR

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 CATALOGUE

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 over 7 000 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, Solid Carbide Tooling, Carbide Insert Tooling, Custom Tools and Surface Coatings

Profile

SOMTA TOOLS specialises in the design and manufacture of drills, cutters, reamers, threading tools, toolbits and custom tools for the industrial and "do it yourself" 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.

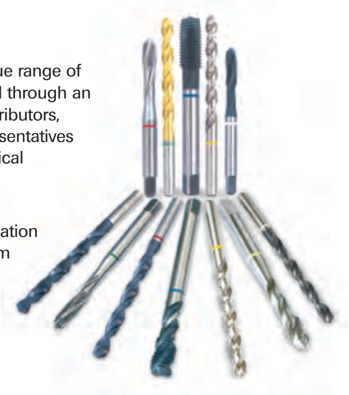
The Company

SOMTA TOOLS factory in Pietermaritzburg manufactures 7 000 standard items and a further 3 000 made-to-order items to serve local markets and export markets in over 70 countries worldwide.

The Product

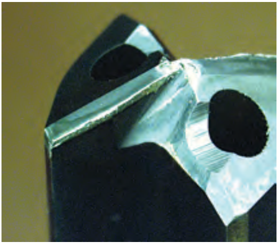
SOMTA TOOLS standard catalogue range of precision cutting tools are supplied through an extensive network of industrial distributors, backed up by technical sales representatives who are available to provide technical assistance where required.

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

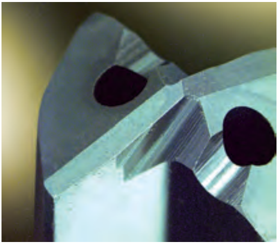


Manufacturers of precision industrial and
DIY coated cutting tools

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 re-sharpening of HSS and Solid Carbide 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.

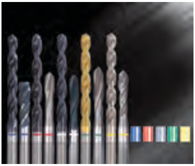
Contact our technical department on **Tel: +27 33 355 6600** or **e-mail: tech@somta.co.za** for technical assistance if experiencing a cutting tool problem.



State of the art . . .

Somta has an integrated **state-of-the art Balzers PVD Rapid Coating System** in its manufacturing programme, offering the innovative range of **Balzers BALINIT®** high performance coatings on all cutting tools.

Application
products



Colour Band Range

SOMTA Colour Band Application range of drills and taps is specially designed to optimize your machining performances. Each range has been designed with different cutting geometries and surface treatments to ensure optimum tool performance for each specific material category.

Solid Carbide End Mills for Aluminium

Somta's high efficiency carbide end mill range with optimal flute geometry provides all the required features for high performance machining of aluminium, with the added benefits of greater stock removal rates at high speeds and feeds, excellent surface finish quality and extended tool life.

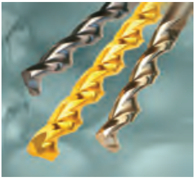
MTS Chipbreaker Drills

This outstanding development increases drill cutting efficiency by means of greatly improved chip control. A chipbreaker rib is positioned along the length of the flutes, which curls and breaks long chip forming material into small manageable chips for easier evacuation. There is no clogging of chips in the flutes, as small chips flow freely along the flutes. The chipbreaker drill thus cuts more freely than standard drills.

Solid Carbide VariCut End Mills

Somta's new "Vari-Style" End Mill, VariCut has a new patent pending tool design. This unique design uses a new core form and a new reinforced end geometry with unequal flute spacing which enables it to remove the most amount of material in the least amount of time with an excellent surface finish. On Stainless, or Titanium, it will match or outperform any other 4 flute "Vari-Style" End Mill in the market.





UD Ultra Parabolic Twist 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.

Reamer Tool Range

Established in 1954, Somta offers a range of high quality standard reamer products and custom reaming solutions. From closest tolerance precision machining of extremely accurate holes through to enlargement, alignment and deburring of holes for construction, assembly and general purpose applications. We have a product or a solution to satisfy your specific engineering requirements.

Solid Carbide Tooling (Standard Range)

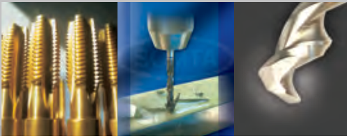
Somta's Solid Carbide Tooling Standard Range is a comprehensive family of solid carbide stub and jobber length drills, 2 and 4 flute end mills and ball nose end mills, in both regular and long series.

Solid Carbide Hard Material End Mills

A range of high performance finishing end mills for operation on hard materials up to 52HRC (512HB), plus Somta's Hi-Feed end mill with patent pending geometry that removes the most amount of material in the least amount of time combined with extended tool life, for use on hard and super hard steels up to 65HRC (880HV).

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

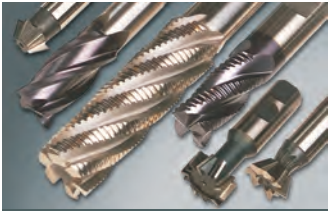
Standard
products




Straight Shank Drills


Morse Taper Shank Drills


Reamers, Countersinks
& Counterbores

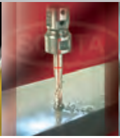

Shank Cutters

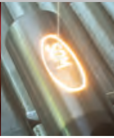

Bore Cutters

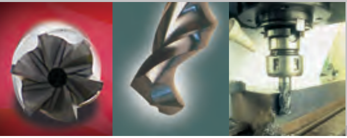

Threading Tools


Toolbits

Contents







Somta cutting tools are manufactured from the finest steels available.

Solid Carbide

Sub-micron or ultra fine cabide grade of European origin. High stock removal rates with excellant rigidity at high speeds and feeds and extended wear life are the major benefits of this material.

High Speed Steel

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. The heat treatment process is controlled by our Metallurgical laboratory using advanced computerised and electronic instrumentation.

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)
HSS	0.9	4	6	5	2	-	63 - 65
HSS-Co5	0.9	4	6	5	2	5	64 - 66
HSS-Co8	1.1	4	1.5	9.5	1	8	66 - 68.5(70)
HSSE-V3	1.2	3.9	7	5.2	2.7	-	64-66

HSS (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.

HSS-Co5 (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.

HSS-Co8 (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.

HSSE-V3 material is mainly used in the manufacture of machine taps because of its good wear resistance, good grinding capabilities, high hardness and excellent toughness.

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 Finish

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.

Carbide Coatings

The material and properties of the coating used is best matched to solid carbide cutting tool substrate material allowing the coated solid carbide tool to be successfully employed under the extreme conditions of hard machining and typically permits much faster and more economical machining of dies in hard and super hard steels.

SELECTING THE CORRECT DRILL

Straight Shank Drills

The following are available ex-stock from Somta.

Stub Drills

Solid Carbide for high production drilling.

Coolant Feed Solid Carbide for high production drilling.

A robust drill suited to portable drill application.

UDL for use on CNC machines where high productivity and accurate holes are required.

Jobber Drills

Solid Carbide for high production drilling.

Coolant Feed Solid Carbide for high production drilling.

For precision drilling.

For light industrial drilling.

For drilling high tensile steels and other difficult materials.

UDL for use on CNC machines where high productivity and accurate holes are required.

CBA - Yellow Band Quick Spiral for drilling Aluminium.

CBA - Blue Band RF for drilling Stainless Steel (VA).

CBA - Green Band NDX for drilling Carbon Steel.

CBA - Red Band UDS for drilling Tough Treatable Steel.

CBA - White Band UDC for drilling Cast Iron.

Double Ended Sheet Metal / Body Drills

Double ended self centering drill designed to produce accurate holes in thin materials.

Reduced Shank (Electricians) Drills

For general purpose drilling.

Long Series Drills

For general purpose long reach drilling.

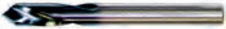
UDL for use on CNC machines where high productivity and accurate holes are required. High performance deep hole drilling.

Extra Length Drills

For extra deep hole drilling.

UDL for use on CNC machines where high productivity and accurate holes are required. High performance extra deep hole drilling.

NC Spotting Drills



For accurate positioning of holes. Ideal for CNC lathes. Alternative to using centre drills.

Centre Drills - Form A, B, R and American Standard

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

Masonry Drills

For drilling concrete, brick and tile.

Heavy Duty SDS Plus Drills

For drilling concrete, brick and tile.

Cutting tools may shatter
eye protection should be worn

Morse Taper Shank Drills

The following are available ex-stock from Somta.

Morse Taper Shank Drills

For general purpose drilling.

For general purpose drilling in difficult materials.

MTS Extra Length Drills

For extra deep hole drilling.

MTS Chipbreaker Drills

High performance production drilling.

MTS Oil Tube Chipbreaker Drills

High performance production drilling.

MTS Armour Piercing Drills

Heavy duty drilling in work hardening and heat treated steels.

MTS Rail Drills

For drilling manganese rails and other tough steels.

MTS Core Drills

For enlarging diameters of existing holes whether drilled, punched or cast.

Sorgers

A wood auger for drilling all types of wood.

Specials

Drills for special Applications.

Tanged Jobber Drills

Designed to fit tang drive sleeve.

Drill Reamers

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

Sheet Metal Drills

Self centering drill designed to produce accurate holes in thin materials.

UDC Jobber Drills

For use on NC and CNC machines where high productivity and accurate holes are required in Cast Iron.

Cotter Pin Drills

For heavy duty drilling using a tang drive sleeve.

Tin Tip Jobber Drill

For higher performance in general purpose drilling.

Subland Drills

For drilling stepped holes in one operation.

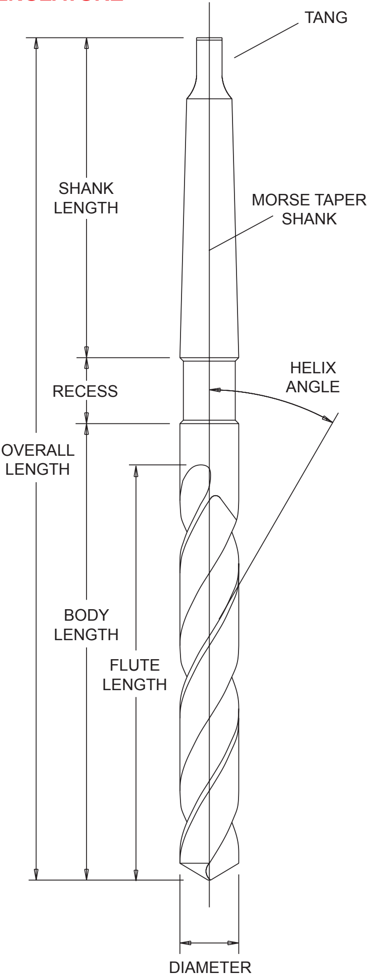
Left Hand Jobber Drills

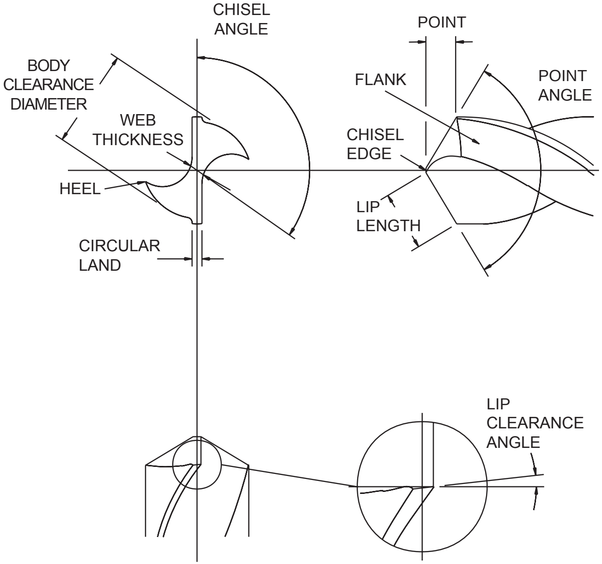
General purpose drilling in the left hand direction.

UDS Jobber Drills

For use on CNC where high productivity and accurate holes are required.

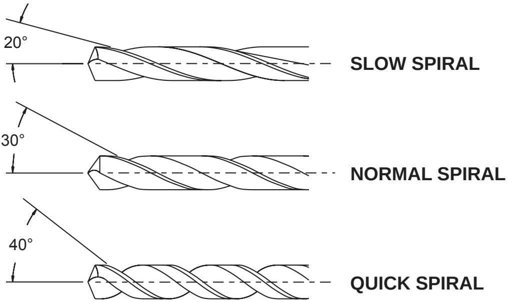
DRILL NOMENCLATURE





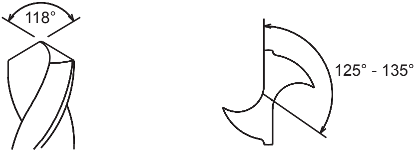
DETAILED VIEW OF LIP CLEARANCE

TYPES OF SPIRAL (OR HELIX) ANGLES



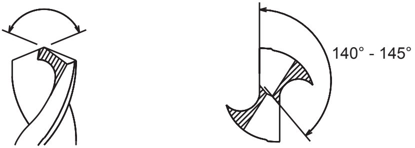
DRILL POINT STYLES

Standard Point



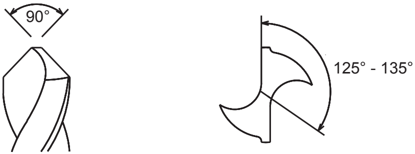
This point is suitable for general purpose drilling.

Split Point



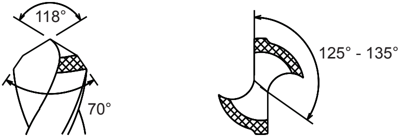
The split point minimises end thrust and is self centering.

Long Point



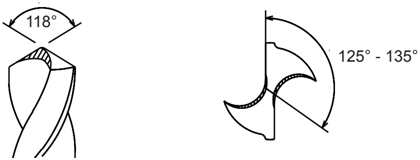
Used for wood, plastic, hard rubber, fibres etc.

Cast Iron Point ("DX" Point)



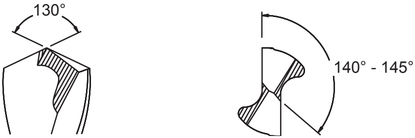
The secondary angle reduces wear on the outer corners.

Web Thinned Point



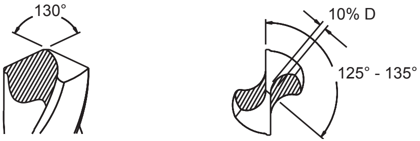
The web thinned point reduces end thrust and improves centre cutting efficiency.

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.

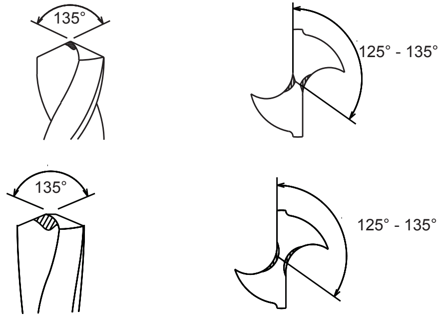
"UX" Point



The 130° special notched "UX" point style provides self centering, easier penetration, improved hole accuracy and improved load distribution.

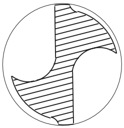
This special notch geometry gives a corrected rake angle of 15° which provides strong point for harder materials, as well as preventing snatching with materials such as Aluminium, Brass, Bronze and Plastics. Available on UDL and UDS drills.

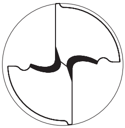
Notched Point and Heavy Duty Notched Point

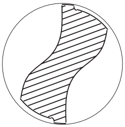


The notched point reduces end thrust and optimises centre cutting efficiency with chisel strength. It is recommended for hard and high strength materials.

FLUTE FORMS


• Conventional Web


• Chipbreaker


• Parabolic Flute Form
• Thicker Web

Benefits of the Parabolic Flute Form

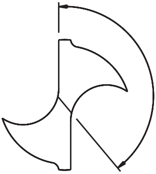
Heavy web construction increases rigidity under torsional load thus eliminating chatter at the cutting edges which cause edge break down and early failure. The Parabolic drill web is 50-90% thicker than the standard drill, depending on drill diameter.

Wider flute form, together with quicker spiral, promotes better chip removal while allowing easier coolant flow to the drill point.

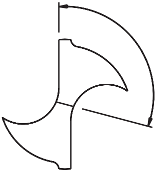
COMMON RE-SHARPENING ERRORS

LIP
CLEARANCE
ANGLE
TOO GREAT

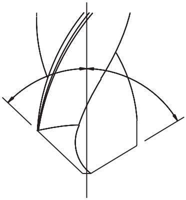
LIP
CLEARANCE
ANGLE
TOO SMALL



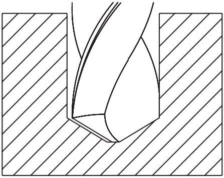
CHISEL ANGLE
TOO GREAT



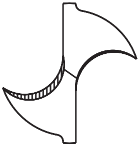
CHISEL ANGLE
TOO SMALL



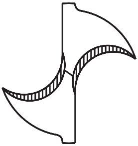
LIPS UNEQUAL LENGTH
(DIFFERENCE IN RELATIVE
LIP HEIGHT.)



DIFFERENCE IN RELATIVE
LIP HEIGHT WILL DRILL AN
OVERSIZE HOLE.



UNEVEN WEB
THINNING



WEB THINNING
TOO GREAT

Web thinning is recommended for:

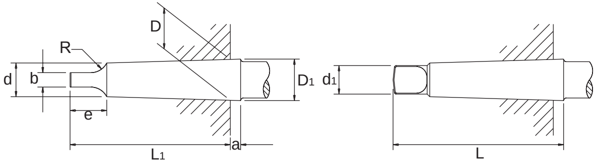
1. restoring chisel edge to the original length after several regrinds.
2. larger drills where the machine thrust is limited.
3. difficult materials.

Lip Clearance Angle

Drill Size (mm)	Lip Clearance Angle	Chisel Angle
Up to 3	18° - 24°	125° - 135°
3.1 - 6	14° - 18°	
6.1 - 12	10° - 14°	
12.1 - 20	8° - 12°	
Above 20	6° - 10°	

STANDARD MORSE TAPER SHANK

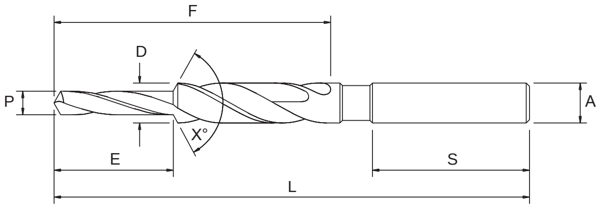
To ISO 296 DIN 228 BS 1660



No. of Taper	Fitting line Diameter D	Diameter d	Overall Length Max L	D_1	a	Max L_1	Max e	H13 b	Max d_1	Taper / mm on Dia	Max R
1	12.065	9.0	65.5	12.2	3.5	62.0	13.5	5.2	8.7	0.04998	5.0
2	17.780	14.0	80.0	18.0	5.0	75.0	16.0	6.3	13.5	0.04995	6.0
3	23.825	19.0	99.0	24.1	5.0	94.0	20.0	7.9	18.5	0.05020	7.0
4	31.267	25.0	124.0	31.6	6.5	117.5	24.0	11.9	24.5	0.05194	8.0
5	44.399	36.0	156.0	44.7	6.5	149.5	29.0	15.9	35.7	0.05263	10.0
6	63.348	52.0	218.0	63.8	8.0	210.0	40.0	19.0	51.0	0.05214	13.0

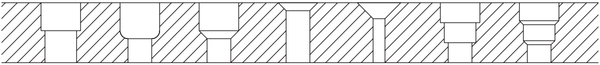
HOW TO ORDER SPECIALS

Multiple Diameter Drills



Specify whether drill is to be Step or Subland Type.
D = Diameter of large, fluted section.
P = Diameter of small, fluted section.
A = Shank Diameter.
L = Overall Length.
F = Flute Length.
E = Length of Small Diameter. This is measured from the extreme point to the bottom corner of the step angle.
X°= Included angle of the step angle.
S = Shank Length.

It is possible to drill two or more diameters in a hole on one operation with a correctly designed drill and these are often used in mass production engineering.



Some of the hole types that can be drilled in a single operation.

If you have any cutting tool problem, please feel free to contact our technical sales representatives.

Modified Standards

There are many instances when a special tool (a tool not found in the Somta catalogue or price list) can be manfactured from a standard product. We call this a ‘modified standard’.

Somta has both the capability and capacity to offer this service which, under normal circumstances, means a short delivery time.

The following are typical drill modifications:

Intermediate Diameters
Standard sizes can be ground down to special diameters and tolerances.

Reduced Overall Lengths
Standard drills can be cut to special lengths.

Drill Points
The standard drill point angle is 118° included. This can be modified to any angle required. Many special points are available which include web thinning, notch points, split points, double angle points, spur and brad points etc.

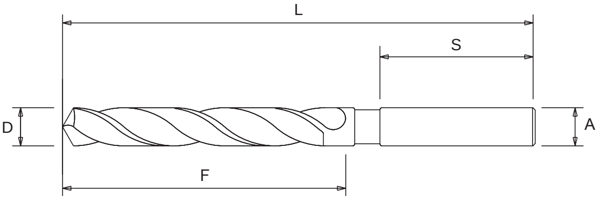
Tangs and Flats
Tangs can be produced to DIN, ASA and ISO, also special whistle notch flats on shanks.

Step Drills
Standard drills can be modified into step drills. (See drawing on page 18).

Surface Treatments
A full range of surface treatments including nitriding, stream oxide, chemical blackening, gold oxide and various PVD coatings are available.

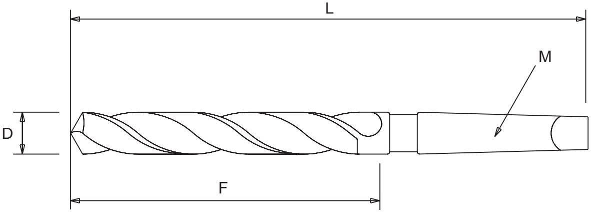
When an intermediate diameter or a non standard length of drill is required, the following diameters and lengths need to be specified.

Straight Shank Drills



D = Drill Diameter
A = Shank Diameter
L = Overall Length
F = Flute Length
S = Shank Length

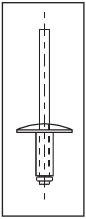
Morse Taper Shank Drills



D = Drill Diameter
L = Overall Length
F = Flute Length
M = Morse Taper Size

DRILL SIZES FOR ALUMINIUM & STEEL BLIND (POP) RIVETS

Rivet Body Diameter (mm)	Drill Size (mm)
2.4	2.5
3.2	3.3
4.0	4.1
4.8	4.9
6.4	6.5



Note: For grooved and peeled rivets add 0.2mm to the above drill sizes (up to 4.8mm diameter).

CUTTING DIAMETER TOLERANCE

SOMTA Twist Drills are manufactured to h8 tolerance.
2 Flute Drills

Cutting Diameter Tolerance on Twist Drills

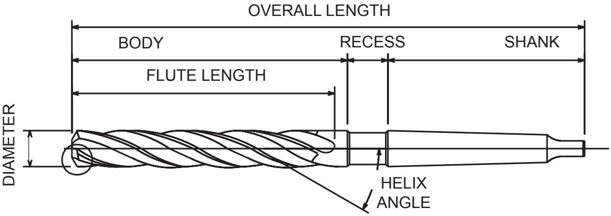
Drill Diameter (mm)		Diameter Tolerance (mm)	
Above	Up to	Plus	Minus
-	3	+ 0	-0,014
3	6	+ 0	-0,018
6	10	+ 0	-0,022
10	18	+ 0	-0,027
18	30	+ 0	-0,033
30	50	+ 0	-0,039
50	80	+ 0	-0,046

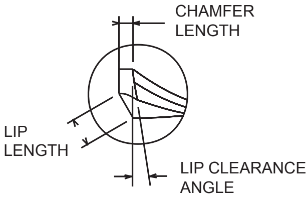
Back Taper on Diameter of Fluted Portion
The drill diameter is normally reduced over the fluted portion to prevent jamming. The amount of back taper is a maximum of: 0,08 mm on diameter per 100 mm length.

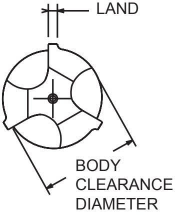
Back taper is usually only applied to sizes over 6 mm.

CORE DRILLING

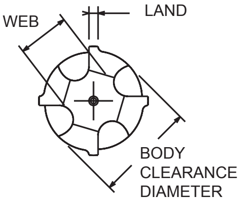
Core Drill Nomenclature







3 FLUTES



4 FLUTES

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

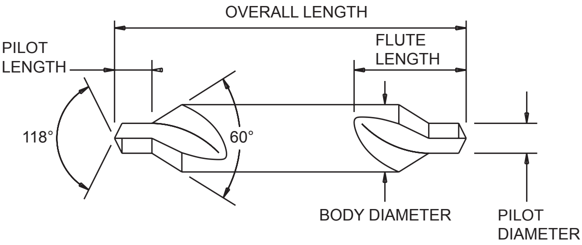
Speed - As for 2 flute drills

Feed - **3 Flute**
1 to 1,5 x 2 flute drill feed rate

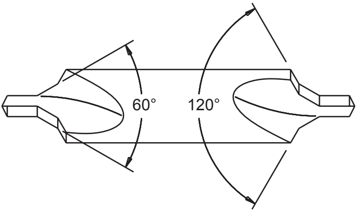
4 Flute
1,5 to 2 x 2 flute drill feed rate

CENTRE DRILLS

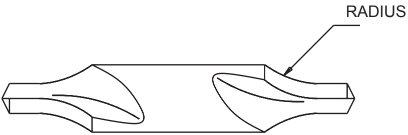
Centre Drill Nomenclature



TYPE 'A'



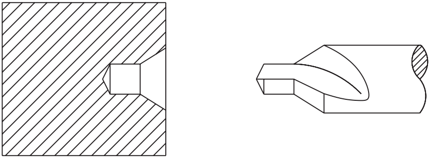
TYPE 'B'



TYPE 'R'

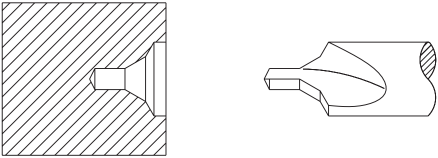
Selecting the correct Centre Drill

TYPE "A"



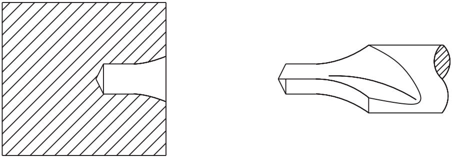
For general centering operations on workpieces requiring additional maching between centres.

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 34-35. 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 32-33, 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 16-17.

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 34-35).
- Do not allow chips to clog the drill flutes.
- When re-sharpening take care to achieve the correct point geometry (see page 13-15) 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 5-10).

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 28. 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/mm². Similar drills for softer materials such as aluminium, mild steel etc. with hardness up to 500 N/mm² 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.

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 32-33.

(b) **Possible Cause**

Insufficient lip clearance behind the cutting edge.

Solution

Check that the lip clearance is as per information on page 16-17.

(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.

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 16-17.

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.

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.

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.

Worn outer Corners

(a) **Possible Cause**
The peripheral speed is too high for the material being drilled.
Solution
Use the recommended speed - see page 34-35.

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 16-17.

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 5-10 for the correct drill to suit the application.

Rough hole finish

(a) **Possible Cause**
The drill is blunt.
Solution
Re-sharpen as per information on page 16-17.

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

If you have any cutting tool problem, please feel free to contact our technical sales representatives.

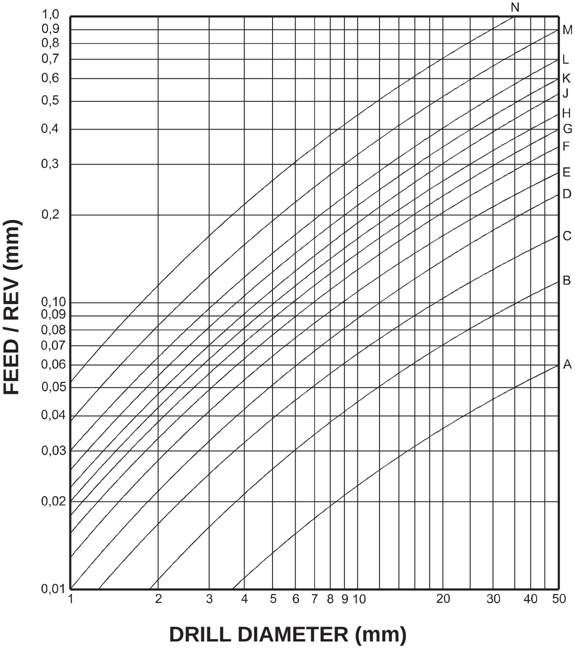
DRILL FEED CURVE CHART

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.

Drill Feeds (mm / rev.)



How to use the Drill Feed Chart

1. Locate Feed Curve (as given in the application data pages 35 & 37) 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.

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	150	35	540	140 141 151 184 185	1X1-1X6 101-102 1TT 112 177 175-176
	0.3 to 0.4% Carbon	170	40	620		
	0.3 to 0.4% Carbon	248	59	910		
	0.4 to 0.7% Carbon	206	47	720		
	0.4 to 0.7% Carbon	286	67	1030		
	Low Alloy Tool Steels	248	59	910		
	High Alloy Tool Steels	330	75	1150		
	Heat Treatable Steels					
	Die Steels	380	87	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		
CAST IRONS	Grey Irons	110 - 300	-	-	AS ABOVE	AS ABOVE
	Nodular Irons					
	Malleable Irons					
MANGANESE STEEL		AS SUPPLIED			AS ABOVE	AS ABOVE
ALUMINIUM	Wrought Alloys	AS SUPPLIED			AS ABOVE	AS ABOVE
	Cast Alloys					
	Silicon Alloys					
MAGNESIUM ALLOYS						
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
	Difficult to Machine Alloys	COMMERCIAL BRONZE 90% PHOSPHOR BRONZE 5 - 10% ALUMINIUM BRONZE				
PLASTICS	Soft	AS SUPPLIED			AS ABOVE	AS ABOVE
	Hard					
	Reinforced					

DRILL TECHNICAL DATA *(cont.)*

CODE TYPE				COOLANT	SPEED METRES / MIN	FEED CURVE See Page 32-33
LONG SERIES	EXTRA LENGTH	MORSE TAPER STANDARD	MORSE TAPER E/LENGTH			
116-117	121-126 132-136	2X1-2X4 201-205 211-214 208 261 279	242-245 252-255	SOLUBLE OIL OR SEMI-SYNTHETIC OIL	25 - 30	H
					15 - 20	
					10 - 15	
				SOLUBLE OIL	15 - 24	F
				SOLUBLE OIL EXTREME PRESSURE	10 - 15 4 - 8	H
AS ABOVE	AS ABOVE	AS ABOVE	AS ABOVE	SOLUBLE OIL EXTREME PRESSURE OR SULPHO- CHLORINATED	12 - 16	H
					6	C
AS ABOVE	AS ABOVE	AS ABOVE	AS ABOVE	SOLUBLE OIL EXTREME PRESSURE OR SULPHO-CHLORINATED	5 - 10	E
AS ABOVE	AS ABOVE	AS ABOVE	AS ABOVE	SOLUBLE OIL SULPHO-CHLORINATED EXTREME PRESSURE CHLORINATED OIL	15 - 25	F
					7 - 11	C
AS ABOVE	AS ABOVE	AS ABOVE	AS ABOVE	DRY OR DETERGENT WATER - SOLUBLE EMULSION	25 - 35	K
					15 - 30	
					25 - 30	
AS ABOVE	AS ABOVE	AS ABOVE	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	L
					30 - 35	
				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	L
					15 - 20	
AS ABOVE	AS ABOVE	AS ABOVE	AS ABOVE	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 32-33
extra long	UDL TiN	35 - 45 50 - 70	H J
middle/long	UDL TiN	25 - 35 40 - 50	H J
long	UDL TiN TiCN TiAIN	25 - 30 35 - 40	G I
long	UDL TiN TiCN TiAIN	25 - 30 35 - 40	G I
long	UDL TiN TiCN TiAIN	15 - 20 25 - 30	E G
long	UDL TiN TiCN TiAIN	15 - 20 20 - 25	E G
middle	UDL TiN TiCN TiAIN	18 - 21 27 - 32	E G
long	UDL TiN TiCN TiAIN	8 - 10 12 - 15	K M
long	UDL TiN TiCN TiAIN	10 - 15 16 - 22	E G
extra short	UDC TiAIN	30 - 35 45 - 55	G I
extra short	UDC TiAIN	25 - 30 35 - 45	G I
middle/short	UDC TiAIN	18 - 21 25 - 35	E G
middle/short	UDC TiAIN	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 TiAIN	12 - 16 20 - 25	G I
long	UDL TiCN TiAIN	6 - 8 10 - 12	G I
long	UDL TiCN TiAIN	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

CBA DRILL TECHNICAL DATA

TOOL MATERIAL / MACHINED MATERIALS	HARDNESS BRINELL	HARDNESS N/mm²	CUTTING SPEED Metres/Min.	FEED RATE FOR DIAMETERS							
				3mm	5mm	6mm	8mm	10mm	12mm	16mm	20mm
YELLOW BAND QUICK SPIRAL JOBBER DRILLS											
Aluminium Alloys											
Wrought & Extruded	< 150	541	50 - 60	0.120	0.150	0.170	0.220	0.260	0.280	0.320	0.360
Wrought & Treated	> 150	541	35 - 50	0.085	0.110	0.120	0.160	0.190	0.200	0.240	0.280
"Cast, Low Silicon <5%"	< 150	541	30 - 40	0.085	0.110	0.120	0.160	0.190	0.200	0.240	0.280
"Cast, High Silicon >10%"	> 150	541	23 - 35	0.085	0.110	0.120	0.160	0.190	0.200	0.240	0.280
Copper											
Pure Copper	< 100	-	35 - 55	0.062	0.080	0.095	0.120	0.140	0.150	0.160	0.210
"Brass, Soft"	< 200	717	40 - 50	0.150	0.190	0.210	0.280	0.330	0.350	0.400	0.450
"Brass, Bronze"	> 200	717	35 - 45	0.085	0.110	0.120	0.160	0.190	0.200	0.240	0.280
BLUE BAND RF JOBBER DRILLS											
Stainless Steels											
Free Cutting	< 250	861	12 - 22	0.085	0.110	0.120	0.160	0.190	0.200	0.240	0.280
Austenitic	< 250	861	10 - 15	0.085	0.110	0.120	0.160	0.190	0.200	0.240	0.280
"Martensitic, Ferritic"	> 300	971	12 - 18	0.062	0.080	0.095	0.120	0.140	0.150	0.160	0.210
Titanium											
"Pure Titanium, unalloyed"	< 200	758	20 - 32	0.062	0.080	0.095	0.120	0.140	0.150	0.160	0.210
Titanium Alloys	> 300	971	6 - 12	0.062	0.080	0.095	0.120	0.140	0.150	0.160	0.210
Nickel											
"Pure Nickel, Unalloyed"	< 300	971	10 - 15	0.085	0.110	0.120	0.160	0.190	0.200	0.240	0.280
GREEN BAND NDX JOBBER DRILLS											
Carbon Alloy Steels											
Free Cutting Mild Steel	< 120	420	40 - 50	0.120	0.150	0.170	0.220	0.260	0.280	0.320	0.360
Low Carbon Steel	< 200	758	30 - 40	0.085	0.110	0.120	0.160	0.190	0.200	0.240	0.280
Medium Carbon Steel	< 250	861	25 - 35	0.062	0.080	0.095	0.120	0.140	0.150	0.160	0.210
RED BAND UDS JOBBER DRILLS											
Copper											
High Tensile Bronze	< 350	1144	15 - 28	0.062	0.080	0.095	0.120	0.140	0.150	0.160	0.210
Carbon Alloy Steels											
Low Alloy Steel	> 250	861	25 - 30	0.062	0.080	0.095	0.120	0.140	0.150	0.160	0.210
"Alloyed, Heat Treated"	> 300	971	15 - 20	0.045	0.060	0.065	0.070	0.100	0.110	0.130	0.160
"Alloyed, Heat Treated"	> 350	1144	10 - 15	0.045	0.060	0.065	0.070	0.100	0.110	0.130	0.160
Nickel											
"Nickel, Nimonic 75"	> 300	971	6 -10	0.062	0.080	0.095	0.120	0.140	0.150	0.160	0.210
"Nickel, Inconel 718 Alloy"	< 350	1144	4 - 8	0.045	0.060	0.065	0.070	0.100	0.110	0.130	0.160
WHITE BAND UDC JOBBER DRILLS											
Cast Irons											
Plain Grey Irons	< 150	541	35 - 45	0.120	0.150	0.170	0.220	0.260	0.280	0.320	0.360
Plain 'SG' Iron	< 250	861	23 - 35	0.062	0.080	0.095	0.120	0.140	0.150	0.160	0.210
Alloy 'SG' Iron Nickel Hard	> 250	861	15 - 28	0.062	0.080	0.095	0.120	0.140	0.150	0.160	0.210

MTS CHIPBREAKER DRILL TECHNICAL DATA

RECOMMENDED SPEED, COOLANT AND LIP CLEARANCE ANGLE					
MATERIAL TO BE DRILLED	SPEED		COOLANT	POINT ANGLE	LIP CLEARANCE ANGLE
	Metres per Min.	Feet per Min.			
Aluminium & Aluminium Alloys	61-92	200-300	Soluble Oil or Paraffin	118°	12°
Brass	46-76	150-250	Dry or Soluble Oil	118°	15°
Brass - Leaded	61-92	200-300	Dry or Soluble Oil	118°	15°
Bronze	30-61	100-200	Soluble Oil	118°	15°
Bronze - High Tensile	22-30	70-100	Soluble Oil	118°	15°
Cast Iron - Soft	30-46	100-150	Dry or use air	90°	12° - 15°
Cast Iron - Malleable	22-24	70-80	Soluble Oil	118°	10° - 12°
Cast Iron - Hard	15-22	50-70	Dry or use air	118°	10° - 12°
Cast Iron - Chilled	8-11	25-35	Soluble Oil	118°	10° - 12°
Copper	30-61	100-200	Soluble Oil	100°	15°
Magnesium	Up to 122	Up to 400	Soluble Oil or Paraffin	118°	12°
Monel	12-15	40-50	Soluble Oil or Sulphurised Oil	125°	10° - 12°
Steel - Plate, Bar, Cast, Forged Free Cutting Mild	30-61	100-200	Soluble Oil or Sulphurised Oil	118°	10° - 12°
Steel - Up to 620 N/mm^2 (175 HB)	24-33	80-110	Soluble Oil or Sulphurised Oil	130°	10° - 12°
Steel - >620 N/mm^2 (175 HB), <910 N/mm^2 (250 HB)	14-22	45-70	Soluble Oil or Sulphurised Oil	130°	10°
Steel - >910 N/mm^2 (250 HB), <1220 N/mm^2 (350 HB)	9-14	30-45	Soluble Oil or Sulphurised Oil	125°	12°
Steel - >1220 N/mm^2 (350 HB)	5-8	15-25	Soluble Oil or Sulphurised Oil	130°	10°
Steel - Manganese (Low)	5-6	15-20	Sulphurised Oil	130°	10°
Steel - Stainless - Free Cutting	15-18	50-60	Soluble Oil or Sulphurised Oil	130°	10° - 12°
Steel - Tough Grades	6-15	20-50	Soluble Oil or Sulphurised Oil	130°-140°	6° - 12°

RECOMMENDED FEED RATES			
METRIC SIZES		IMPERIAL SIZES	
Drill Diameter (mm)	Feed per Rev. (mm)	Drill Diameter (inches)	Feed per Rev. (inches)
6 - 8	0.10 - 0.25	1/4" - 5/16"	0.004" - 0.010"
>8 - 11	0.15 - 0.30	>5/16" - 7/16"	0.006" - 0.012"
>11 - 14	0.20 - 0.35	>7/16" - 9/16"	0.008" - 0.014"
>14 - 17.5	0.25 - 0.40	>9/16" - 11/16"	0.010" - 0.015"
>17.5 - 20.5	0.30 - 0.45	>11/16" - 13/16"	0.012" - 0.018"
>20.5 - 28.5	0.30 - 0.50	>13/16" - 1.1/8"	0.012" - 0.020"
>28.5 - 38	0.35 - 0.75	>1.1/8" - 1.1/2"	0.014" - 0.030"
>38	0.40 - 0.90	>1.1/2"	0.016" - 0.035"

Feed is an important aspect in successful performance. When drilling a new material, start at the lower end of the recommended feed and increase until optimum results are obtained. Optimum results are the lowest cost in producing the hole, tool life being just one factor. Emphasis should be placed on production rate, with tool life considered a partial cost of production, rather than the end result.

SOLID CARBIDE DRILLS TECHNICAL DATA

Material Type Grade	Hardness HB	Tensile Strength N/mm^2	Recommended Surface Speed in m/min for Coated Tungsten Carbide Drills	
			min	max
Steel				
Free cutting steels	< 120	< 400	**120**	**150**
Structural steel	< 200	< 700	**120**	**150**
Plain carbon steel	< 250	< 850	**120**	**150**
Alloy steel	< 350	< 1200	**90**	**120**
Alloy steel, hardened & tempered steel	> 350	> 1200	**50**	**70**
Stainless Steel				
Free machining	< 250	< 850	**75**	**90**
Austenitic	< 250	< 850	**60**	**75**
Ferritic & martensitic	< 300	< 1000	**40**	**50**
Cast Iron				
Lamellar graphite	< 150	< 500	**90**	**150**
Lamellar graphite	< 300	< 1000	**75**	**90**
Nodular graphite, Malleable cast iron	< 200	< 700	**90**	**150**
Nodular graphite, Malleable cast iron	< 300	< 1000	**75**	**90**
Titanium				
Unalloyed	< 200	< 700	**50**	**60**
Alloyed	< 270	< 900	**30**	**40**
Alloyed	< 350	< 1200	**20**	**30**
Nickel				
Unalloyed	< 150	< 500	**55**	**70**
Alloyed	< 270	< 900	**35**	**50**
Alloyed	< 350	< 1200	**20**	**30**
Copper				
Copper	< 100	< 350	**50**	**80**
Beta brass, bronze	< 200	< 700	**50**	**70**
Alpha brass	< 200	< 700	**50**	**70**
High strength bronze	< 470	< 1500	**40**	**50**
Aluminium Alloys				
Wrought alloys	< 100	< 350	**150**	**200**
Cast alloys < 5% Si	< 150	< 500	**120**	**150**
Cast alloys > 5% Si < 10% Si	< 120	< 400	**75**	**90**
Cast alloys >10% Si	< 120	< 400	**45**	**60**
Synthetics				
Duroplastics (short chipping)	-	-	**100**	**300**
Thermoplastics (long chipping)	-	-	**100**	**300**
Fibre reinforced synthetics	-	-	**100**	**300**

TO CALCULATE:

RPM = (surface speed x 1000) / (π x d)
FEED RATE in mm/min = rpm x feed per revoloution

NOTE: For uncoated drills reduce surface speed by 35% to 50%.
For drill depths deeper then 3xD suggest peck drilling.

SOLID CARBIDE DRILLS TECHNICAL DATA *(cont.)*

Recommended feed in mm per revolution for Coated Tungsten Carbide Drills Drill Diameter in mm											
1	2	3	4	5	6	8	10	12	13	14	16
0.03 0.03 0.03 0.02 0.02	0.05 0.05 0.05 0.04 0.04	0.08 0.08 0.08 0.06 0.05	0.10 0.10 0.10 0.08 0.06	0.13 0.13 0.13 0.10 0.08	0.15 0.15 0.15 0.12 0.09	0.20 0.20 0.20 0.16 0.12	0.25 0.25 0.25 0.20 0.15	0.30 0.30 0.30 0.24 0.18	0.33 0.33 0.33 0.26 0.20	0.35 0.35 0.35 0.28 0.21	0.40 0.40 0.40 0.32 0.24
0.02 0.02 0.02	0.04 0.03 0.04	0.05 0.05 0.05	0.07 0.07 0.07	0.09 0.08 0.09	0.11 0.10 0.11	0.14 0.13 0.14	0.18 0.16 0.18	0.22 0.19 0.22	0.23 0.21 0.23	0.25 0.22 0.25	0.29 0.26 0.29
0.03 0.02 0.03 0.02	0.05 0.04 0.05 0.04	0.08 0.06 0.08 0.06	0.10 0.08 0.10 0.08	0.13 0.10 0.13 0.10	0.15 0.12 0.15 0.12	0.20 0.16 0.20 0.16	0.25 0.20 0.25 0.20	0.30 0.24 0.30 0.24	0.33 0.26 0.33 0.26	0.35 0.28 0.35 0.28	0.40 0.32 0.40 0.32
0.02 0.02 0.02	0.04 0.04 0.03	0.06 0.06 0.05	0.08 0.08 0.06	0.10 0.10 0.08	0.12 0.12 0.09	0.16 0.16 0.12	0.20 0.20 0.15	0.24 0.24 0.18	0.26 0.26 0.20	0.28 0.28 0.21	0.32 0.32 0.24
0.02 0.02 0.02	0.04 0.04 0.03	0.06 0.06 0.05	0.08 0.08 0.07	0.10 0.10 0.08	0.12 0.12 0.10	0.16 0.16 0.13	0.20 0.20 0.16	0.24 0.24 0.19	0.26 0.26 0.21	0.28 0.28 0.22	0.32 0.32 0.26
0.03 0.02 0.02 0.02	0.05 0.04 0.04 0.04	0.08 0.06 0.06 0.06	0.10 0.08 0.08 0.08	0.13 0.10 0.10 0.10	0.15 0.12 0.12 0.12	0.20 0.16 0.16 0.16	0.25 0.20 0.20 0.20	0.30 0.24 0.24 0.24	0.33 0.26 0.26 0.26	0.35 0.28 0.28 0.28	0.40 0.32 0.32 0.32
0.03 0.03 0.03 0.03	0.06 0.06 0.06 0.06	0.09 0.09 0.09 0.09	0.12 0.12 0.12 0.12	0.15 0.15 0.15 0.15	0.18 0.18 0.18 0.18	0.24 0.24 0.24 0.24	0.30 0.30 0.30 0.30	0.36 0.36 0.36 0.36	0.39 0.39 0.39 0.39	0.42 0.42 0.42 0.42	0.48 0.48 0.48 0.48
0.04 0.04 0.04	0.07 0.08 0.07	0.11 0.12 0.11	0.14 0.16 0.14	0.18 0.20 0.18	0.21 0.24 0.21	0.28 0.32 0.28	0.35 0.40 0.35	0.42 0.48 0.42	0.46 0.52 0.46	0.49 0.56 0.49	0.48 0.64 0.48

% feed reduction for deep hole drilling

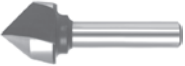
Hole Depth	% reduction
3 x drill diameter 4 x drill diameter 5 x drill diameter > 6 x drill diameter	10% 20% 30% 40%

SELECTING THE CORRECT REAMER, COUNTERSINK & COUNTERBORE

Standard Reamers, Countersinks & Counterbores

The following are available ex-stock from Somta.

Parallel Shank Countersinks



MTS Countersinks

To produce a countersink suitable for countersunk head screws, also used as a deburring tool.

Parallel Shank Counterbores

MTS Counterbores

For counterboring holes to suit capscrew heads.

Parallel Hand Reamers

General hand reaming.

MTS Parallel Machine Reamers

General machine reaming.

MTS Taper Bridge Machine Reamers

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

MTS Machine Chucking Reamers

General machine reaming.

Parallel Shank Machine Chucking Reamers

General machine reaming.

Hand Taper Pin Reamers

For reaming holes to suit standard taper pins.

Specials

Reamers for specific Applications.

MTS Taper Socket Finishing Reamers

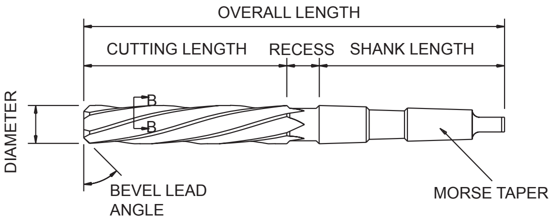
Finishing of Morse Taper holes.

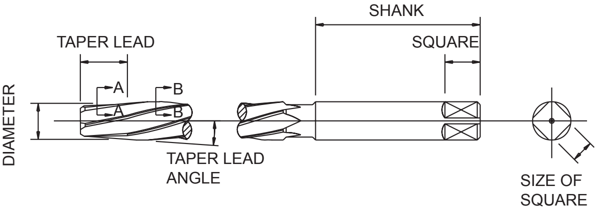
MTS Taper Socket Roughing Reamers

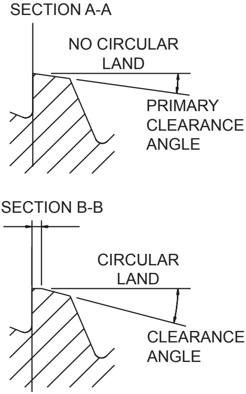
Rough reaming of Morse Taper holes.

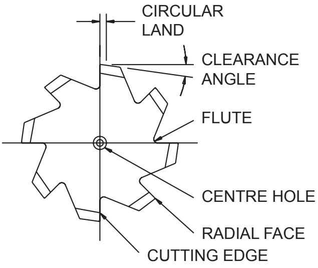
REAMERS

Reamer Nomenclature









FRONT END VIEW OF PARALLEL MACHINE REAMER

The correct use of Reamers

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 46-47 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

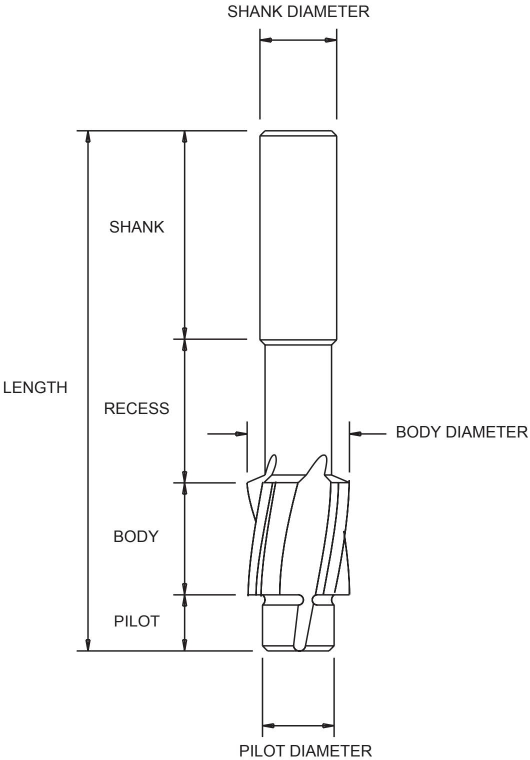
Other useful tolerances can be found on page 151.

Re-Sharpening Reamers

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.

COUNTERBORES

Counterbore Nomenclature



The correct use of Counterbores

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

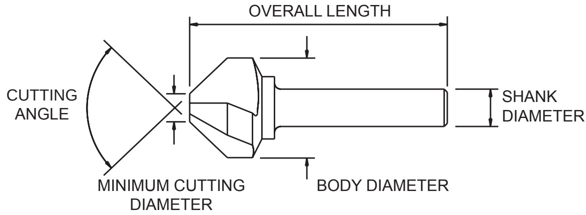
The speeds and feeds for counterbores are approximately 80% to 85% of those for drills as given on page 34-35.
The counterbore diameter given in the above table is used for this calculation.

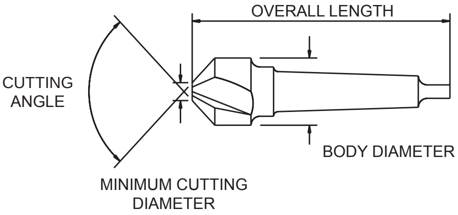
Re-Sharpening Counterbores

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

COUNTERSINKS

Countersink 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 34-35) and are based on the diameter midway between the largest and smallest diameter of the countersink.

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

REAMING PROBLEMS: CAUSES AND SOLUTIONS

Poor Surface Finish

(a) **Possible Cause**
Incorrect speed and/or feed.
Solution
Use the recommended speed/feed - see page 52-53.

(b) **Possible Cause**
A Worn reamer
Solution
Do not allow the reamer to become too blunt. See page 46-47 for re-sharpening details.

(c) **Possible Cause**
Insufficient or wrong type of lubricant.
Solution
Apply an adequate supply of the correct lubricant to the cutting area. See the drill table on page 34-35 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 52-53.

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 45-46 for recommended stock removal.

(b) **Possible Cause**
Speed too high or feed too low.
Solution
Use the recommended speed/feed - see page 52-53.

(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 page 45-46 for recommended stock removal.

(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.

REAMER TECHNICAL DATA

TYPE	GRADE	TYPICAL PHYSICAL PROPERTIES			† Speed m/min	*Type of Feed
		HARDNESS BRINELL	TONS PER SQ IN. (MAX)	N/mm² (MAX)		
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 152-153.
* See table on page 46.

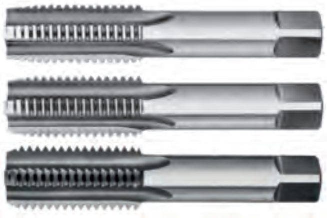
REAMER TECHNICAL DATA *(cont.)*

TYPE	GRADE	TYPICAL PHYSICAL PROPERTIES			† Speed m/min	*Type of Feed
		HARDNESS BRINELL	TONS PER SQ IN. (MAX)	N/mm² (MAX)		
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 152-153.
* See table on page 46.

SELECTING THE CORRECT TAP

Short Hand Taps



Taper

Second

Bottoming

This regular type is the basic tap designed as a general purpose tool for hand and machine operation.

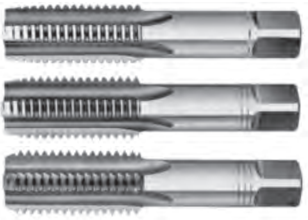
As this basic tap will give acceptable performance in most materials and for short production runs, it is usually the most economical tap to use. However, it performs best in materials where the cutting action results in chips which break up readily and do not present problems of chip disposal.

The regular hand tap has four flutes in sizes larger than 1/4 inch diameter. These taps may not be suitable because of inadequate chip space when deep or blind holes have to be tapped in soft stringy materials. This applies particularly to the coarser pitch threads such as BSW and UNC.

If a gun tap or spiral fluted tap cannot be used, a three fluted tap which permits extra chip space, is recommended.

Cutting tools may shatter
eye protection should be worn

Serial Hand Taps



Rougher

Intermediate

Finisher

Serial taps comprise of one or more undersized roughing taps which remove most of the material before final sizing with a finishing tap.

Some reasons for using serial taps are:

(a) The toughness of the material being tapped.
(b) The amount of material to be removed could cause swarf choking with a single tap.
(c) The very small tolerance on pitch diameter.
(d) An extremely good finish required.

Spiral Flute Short Machine Taps

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 15° or 35° right hand helix. The flute shape eliminates clogging and jamming, resulting in improved tap life.

These taps are designed primarily for machine tapping of blind holes, are used to the best advantage in materials which produce long stringy chips. The shearing action provided by the spiral flutes produces a better finish on difficult to machine metals and causes the chips to be drawn back, eliminating clogging at the cutting chamfer.

Gun Nose (Spiral Point) Short Machine Taps

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.

Gun nose taps have straight flutes supplemented by angular cutting faces at the point. These faces cut with a shearing action which propels the chips ahead of the tap leaving the flutes clear for the free flow of coolant to the point.

Primarily designed for use in through holes, these taps can be used in blind holes providing that there is ample clearance beyond the threaded section to accomodate the chips. The advantages of a gun nose tap are, the shearing action of the angular cutting faces which produce a fine finish on the threads and, shallower flutes which permit a stronger cross section throughout the tap.

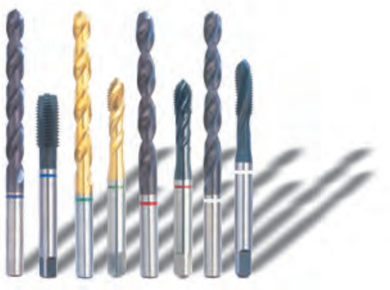
Pipe Taps

For machine use on pipe work for parallel threads.

For machine use on pipe work for tapered threads.

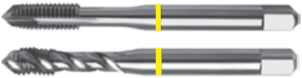
Pipe taps are supplied with PARALLEL threads or with TAPER threads. These taps are shorter than a similar size of regular hand tap, but the design features are the same. They are suitable for hand or machine use.

Colour Band Application (CBA) Taps



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 HSSE-V3 steel (High Vanadium) for greater wear resistance.

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 200HB, tensile strength up to 700N/mm².

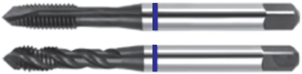
Wide flutes allow more efficient swarf removal which prevents clogging and excessive torque. High rake angle improves shear characteristic and reduces build-up on the cutting edge, allowing tap to cut more freely for longer periods. Spiral flute taps have 40° right hand helix, allowing ductile material swarf to be efficiently forced out of the hole. The yellow band tap is supplied as standard in bright condition.

Yellow Band 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.

These taps are designed for machine tapping in ductile materials, “Fluteless” taps have no flutes or cutting faces, but have special roll forming lobes with circular lands and have long or short taper leads for through or blind holes.

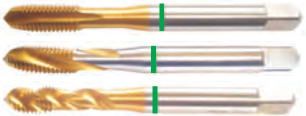
Blue Band



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 TiAIN coating.

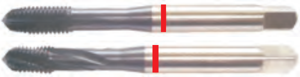
Green Band



The machinebility of different steels is just as varied as their properties. Soft-tough construction steels place completely different demands on the tools, and the green band combination of taps has been perfected for this range of steels.

Green Band characteristics include ability to machine materials with hardness up to 250HB, tensile strength up to 900N/mm². Surface finish - TiN Coating (standard) increases surface hardness of the tool to around 85RC, with excellent resistance to abrasion and cold welding. Thread and flute configuration design for free cutting and structural steels in the general purpose range of medium tensile strengths.

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.

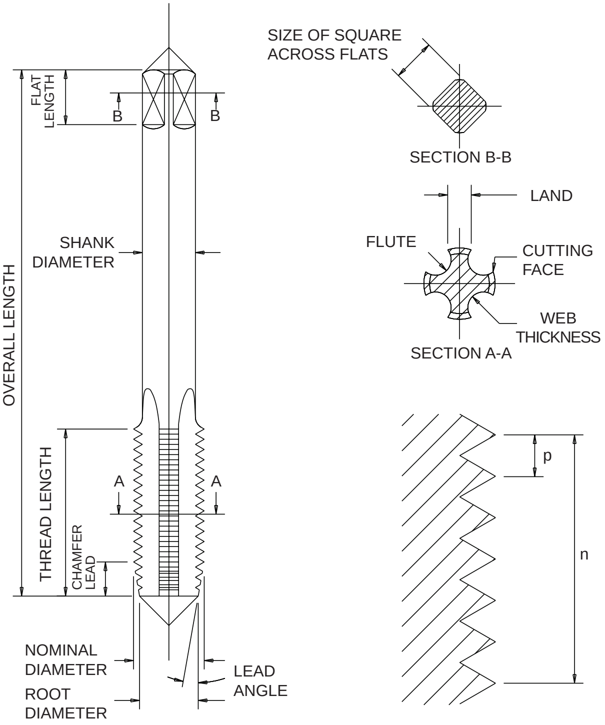
Used to tap materials with hardness up to 470HB, tensile strength up to 1500N/mm². Spiral flute taps have 15° right hand helix which efficiently forces high tensile material swarf up out of the hole, while still maintaining correct cutting geometry. The red band tap is supplied as standard with TiAlN coating.

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.

Special taps are available on request.

TAP NOMENCLATURE



n = No. OF THREADS PER INCH

p = PITCH

Abbreviations for standard thread forms

M - Metric Coarse
MF - Metric Fine

General Specifications: ISO 529
ISO 2283 (Long Series)
Threads ground to: ISO 2857 - 1973, Class 2

General Specifications: DIN 371 / DIN 374 / DIN 376
Threads ground to: DIN 802 CLASS 6H

UNC - Unified National Coarse
UNF - Unified National Fine

General Specifications: ISO 529
Threads ground to: ANSI B1.1 1982 2B

BSW - British Standard Whitworth
BSF - British Standard Whitworth Fine
BA - British Association
BSB - British Standard Brass

General Specifications: ISO 529
Threads ground to: BS 949: 1976 CLASS 2

BSP - British Standard Pipe (Fine) "G" Series
BSPT - British Standard Pipe Taper (Rc Series)

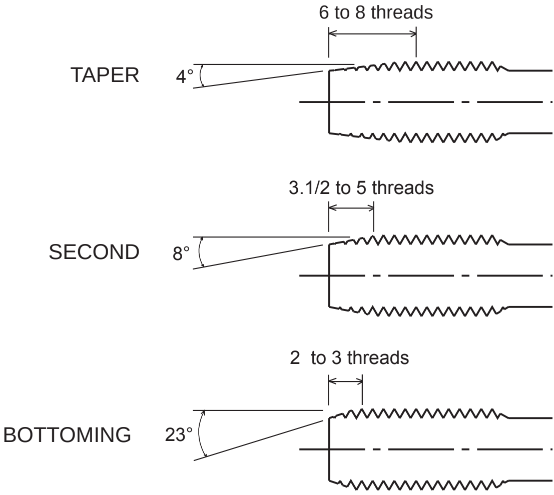
General Specifications: ISO 2284
Threads ground to: BS 949: 1976 G-Series

NPS - National Pipe Straight
NPT - National Pipe Taper

General Specifications: ANSI 94.9 1979
Threads ground to: ANSI 94.9 1979

UNEF - Unified Extra Fine

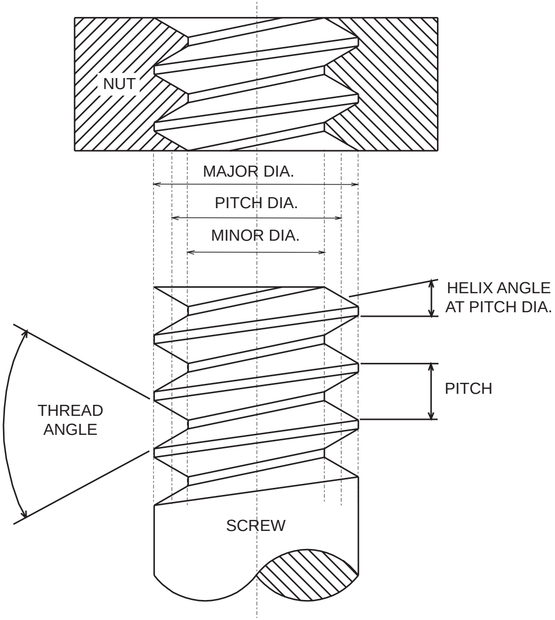
STANDARD LEAD (CHAMFER ANGLES)



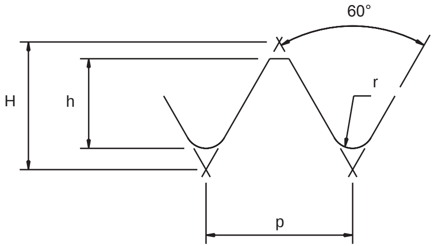
In some countries the name "PLUG" is commonly used to indicate a Bottoming tap. In America it is used to indicate a second tap. To avoid confusion with American terms, the terminology adopted by British Standard 949 1979 as shown above, should be used.

Short Hand and Machine, Long Shank Machine and Pipe Taps	4° (6-8 threads) Taper 8° (3.1/2-5 threads) Second 23° (2-3 threads) Bottoming
Spiral Point Taps	8° (4-5 threads)
Spiral Flute Taps	23° (2-3 threads)
Fluteless Taps	23° (2-3 threads)

COMPONENT ELEMENTS

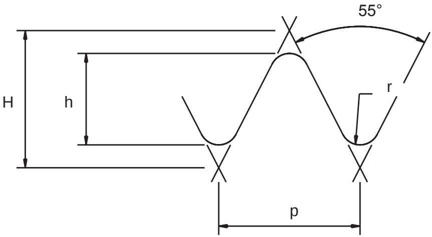


ISO METRIC



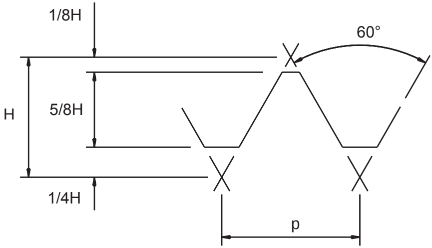
H = 0.866P
h = 0.61343P
r = 0.1443P

WHITWORTH



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

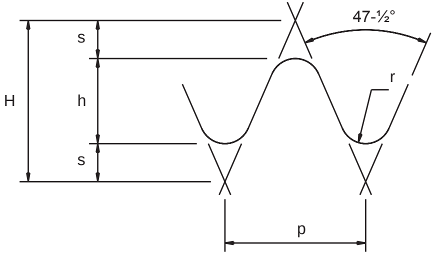
UNIFIED



H = 0.86603P
5/8H = 0.54127P

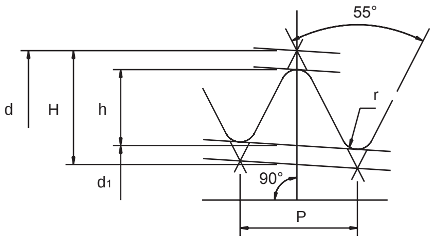
1/4H = 0.21651P
1/8H = 0.10825P

BA



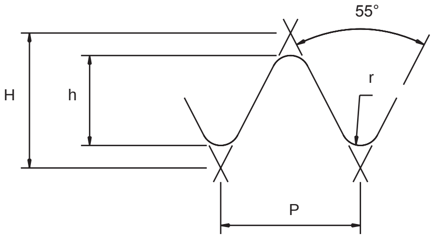
H = 1.1363365P
h = 0.6P
s = 0.26817P
r = 0.18083P

BSPT



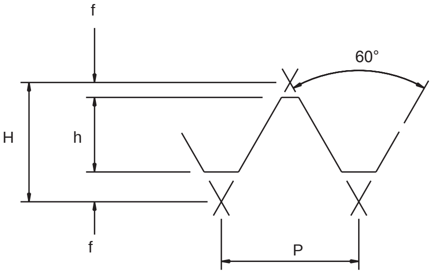
H = 0.960237P	d = MAJOR DIAMETER AT GAUGE PLANE
h = 0.640327P	d_1 = MINOR DIAMETER AT GAUGE PLANE
r = 0.137278P	TAPER = 1 IN 16 ON DIAMETER

BSB



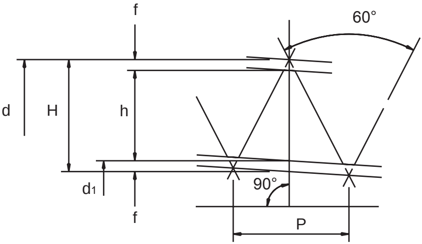
H = 0.86603P
h = 0.5237P
r = 0.1667P

NPS



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

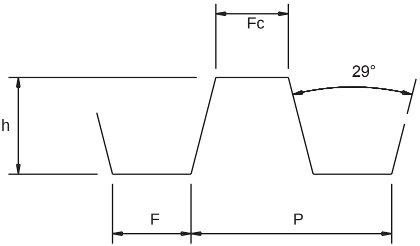
NPT



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

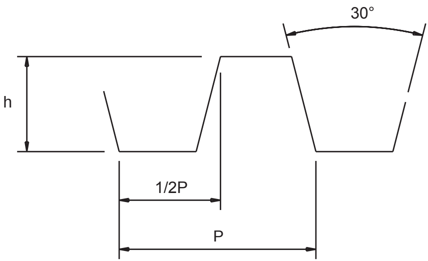
d = MAJOR DIAMETER AT GAUGE PLANE
d_1 = MINOR DIAMETER AT GAUGE PLANE
TAPER = 1 IN 16 ON DIAMETER

ACME



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

TRAPEZOIDAL



h = 0.5P + CLEARANCE

RECOMMENDED TAPPING DRILL SIZES

For 75% thread depth

Metric Coarse

60° included angle (Fluteless*)

Size	Pitch	Tapping Drill Size (mm)
M1	0.25	0.75 (0.9*)
M1.1	0.25	0.85
M1.2	0.25	0.95 (1.1*)
M1.4	0.30	1.1 (1.27*)
M1.6	0.35	1.25 (1.45*)
M1.8	0.35	1.45
M2	0.4	1.6 (1.85*)
M2.2	0.45	1.75
M2.5	0.45	2.05 (2.3*)
M3	0.5	2.5 (2.8*)
M3.5	0.6	2.9 (3.2*)
M4	0.7	3.3 (3.7*)
M4.5	0.75	3.7 (4.2*)
M5	0.8	4.2 (4.65*)
M6	1	5 (5.55*)
M7	1	6
M8	1.25	6.8 (7.4*)
M9	1.25	7.8
M10	1.5	8.5 (9.3*)
M11	1.5	9.5
M12	1.75	10.2 (11.2*)
M14	2	12 (13*)
M16	2	14 (15*)
M18	2.5	15.5 (16.8*)
M20	2.5	17.5 (18.8*)
M22	2.5	19.5 (20.8*)
M24	3	21 (22.5*)
M27	3	24 (25.5*)
M30	3.5	26.5 (28.2*)
M33	3.5	29.5 (31.2*)
M36	4	32 (33.9*)
M39	4	35 (36.9*)
M42	4.5	37.5 (39.6*)
M45	4.5	40.5 (42.6*)
M48	5	43

Metric Coarse (cont)

Size	Pitch	Tapping Drill Size (mm)
M52	5	47
M56	5.5	50.5
M60	5.5	54.5
M64	6	58
M68	6	62

Metric Fine
60° included angle (Fluteless*)

Size	Pitch	Tapping Drill Size (mm)
MF2	0.25	1.75
MF2.2	0.25	1.95
MF2.5	0.35	2.15
MF3	0.35	2.65
MF3.5	0.35	3.15
MF4	0.5	3.5
MF4.5	0.5	4
MF5	0.5	4.5
MF5	0.75	4.3
MF5.5	0.5	5
MF6	0.5	5.5
MF6	0.75	5.25
MF7	0.75	6.25
MF8	0.75	7.25
MF8	1	7 (7.55*)
MF9	0.75	8.3
MF9	1	8
MF10	0.75	9.3
MF10	1	9 (9.55*)
MF10	1.25	8.75 (9.45*)
MF11	1	10
MF11	1.25	9.8
MF12	1	11 (11.55*)
MF12	1.25	10.75 (11.45*)
MF12	1.5	10.5 (11.3*)
MF14	1	13
MF14	1.25	12.75 (13.4*)
MF14	1.5	12.5 (13.3*)
MF15	1	14

Metric Fine (cont)

Size	Pitch	Tapping Drill Size (mm)
MF15	1.5	13.5
MF16	1	15
MF16	1.5	14.5 (15.3*)
MF17	1.5	15.5
MF18	1	17
MF18	1.25	16.8
MF18	1.5	16.5 (17.3*)
MF18	2	16
MF20	1	19
MF20	1.5	18.5 (19.3*)
MF20	2	18
MF22	1	21
MF22	1.5	20.5 (21.3*)
MF22	2	20
MF24	1	23
MF24	1.5	22.5 (23.3)
MF24	2	22
MF25	1	24
MF25	1.5	23.5
MF25	2	23
MF27	1.5	25.5
MF27	2	25
MF28	1.5	26.5
MF28	2	26
MF30	1	29
MF30	1.5	28.5
MF30	2	28
MF30	3	27
MF32	1.5	30.5
MF32	2	30
MF33	1.5	31.5
MF33	2	31
MF33	3	30
MF35	1.5	33.5
MF36	1.5	34.5
MF36	2	34
MF36	3	33
MF38	1.5	36.5
MF38	2	36

Metric Fine (cont)

Size	Pitch	Tapping Drill Size (mm)
MF39	1.5	37.5
MF39	2	37
MF39	3	36
MF40	1.5	38.5
MF40	2	38
MF40	3	37
MF42	1.5	40.5
MF42	2	40
MF42	3	39
MF42	4	38
MF45	1.5	43.5
MF45	2	43
MF45	3	42
MF45	4	41
MF48	1.5	46.5
MF48	2	46
MF48	3	45
MF48	4	44
MF50	1.5	48.5
MF50	2	48
MF50	3	47
MF52	1.5	50.5
MF52	2	50
MF52	3	49
MF52	4	48

BSW
55° Included Angle

Size	TPI	Tapping Drill Size (mm)
1/16	60	1.2
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

BSW (cont)

Size	TPI	Tapping Drill Size (mm)
7/16	14	9.3
1/2	12	10.5
9/16	12	12.2
5/8	11	13.5
11/16	11	15.2
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"	4.5	45
2.1/4	4	51
2.1/2	4	57
2.3/4	3.5	63
3"	3.5	69
3.1/4	3.25	75
3.1/2	3.25	81
3.3/4	3	87
4"	3	93

BSF
55° included angle

Size	TPI	Tapping Drill Size (mm)
3/16	32	4
7/32	28	4.7
1/4	26	5.4
9/32	26	6.2
5/16	22	6.8
3/8	20	8.3
7/16	18	9.8
1/2	16	11
9/16	16	12.7
5/8	14	14
11/16	14	15.5
3/4	12	16.5
7/8	11	19.5

BSF (cont)

Size	TPI	Tapping Drill Size (mm)
1"	10	22.5
1.1/8	9	25.5
1.1/4	9	29
1.3/8	8	31.8
1.1/2	8	34.5
1.5/8	8	38
1.3/4	7	41
2"	7	47
2.1/4	6	53
2.1/2	6	59
2.3/4	6	66
3"	5	71
3.1/4	5	77
3.1/2	4.5	83
3.3/4	4.5	90
4"	4.5	96

UNC
60° included angle

Size	Nom. Dia.	TPI	Tapping Drill Size (mm)
No.1	1.85	64	1.55
No.2	2.18	56	1.85
No.3	2.51	48	2
No.4	2.84	40	2.25
No.5	3.18	40	2.6
No.6	3.51	32	2.75
No.8	4.17	32	3.4
No.10	4.83	24	3.8
No.12	5.49	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

UNC (cont)

Size	TPI	Tapping Drill Size (mm)
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"	4.5	45

UNF
60° included angle

Size	Nom. Dia.	TPI	Tapping Drill Size (mm)
No.1	1.85	72	1.55
No.2	2.18	64	1.9
No.3	2.51	56	2.1
No.4	2.84	48	2.35
No.5	3.18	44	2.65
No.6	3.51	40	2.9
No.8	4.17	36	3.5
No.10	4.83	32	4.1
No.12	5.49	28	4.6
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

Cutting tools may shatter
eye protection should be worn

BSP
55° included angle

Size	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
2.1/4	11	63.3
2.1/2	11	72.8
3"	11	85.5

BSPT
55° included angle

Size	TPI	Tapping Drill Size (mm)
1/8	28	8.6
1/4	19	11.5
3/8	19	15
1/2	14	18.5
3/4	14	24
1"	11	30.25
1.1/4	11	39
1.1/2	11	45
2"	11	56.5

NPS
60° included angle

Size	TPI	Tapping Drill Size (mm)
1/8	27	9.1
1/4	18	12
3/8	18	15.5
1/2	14	19

NPS (cont)

Size	TPI	Tapping Drill Size (mm)
3/4	14	24.5
1"	11.5	30.5
1.1/4	11.5	39.4
1.1/2	11.5	45.5
2"	11.5	57.5

NPT
60° included angle

Size	TPI	Tapping Drill Size (mm)
1/8	27	8.4
1/4	18	11
3/8	18	14.25
1/2	14	17.5
3/4	14	23
1"	11.5	29
1.1/4	11.5	37.5
1.1/2	11.5	43.5
2"	11.5	55.5

BA
47½° included angle

Size	Nom. Dia.	TPI	Tapping Drill Size (mm)
No.12	1.3	90.1	1.05
No.10	1.7	72.6	1.4
No.9	1.9	65.1	1.55
No.8	2.2	59.1	1.8
No.7	2.5	52.9	2.05
No.6	2.8	47.9	2.3
No.5	3.2	43.1	2.65
No.4	3.6	38.3	3
No.3	4.1	34.8	3.4
No.2	4.7	31.3	3.9
No.1	5.3	28.2	4.5
No.0	6	25.4	5.1

If you have any cutting tool problem, please feel free to contact our technical sales representatives.

BSB
55° included angle

Size	TPI	Tapping Drill Size (mm)
1/4"	26	5.3
5/16"	26	6.9
3/8"	26	8.4
7/16"	26	10
1/2"	26	11.7
5/8"	26	15
3/4"	26	18
1"	26	24.5

FLUTELESS TAPS

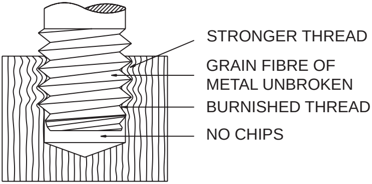
Fluteless taps are used for cold forming threads in ductile materials and have the following advantages.

(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.



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.

A guide to successful machine tapping

- Use the correct tap to suit the application (see page 54-59).
- Select the correct tapping drill size (see page 69-78).
- Direct an adequate supply of the recommended lubricant to the cutting area of the tap (see page 92-95).
- 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

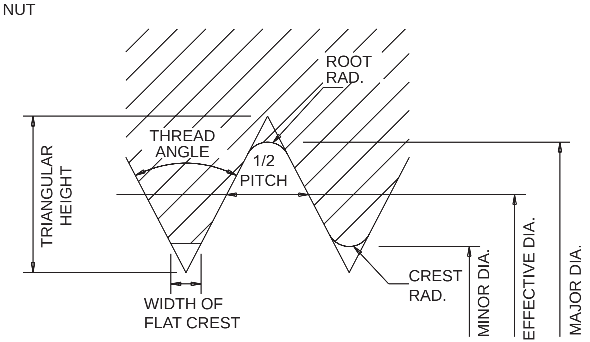
For general purpose work a thread depth of 75% is recommended. A drill size equal to the minor diameter of the tap produces a 100% thread depth. This practice is normally recommended for the following reasons.

(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.

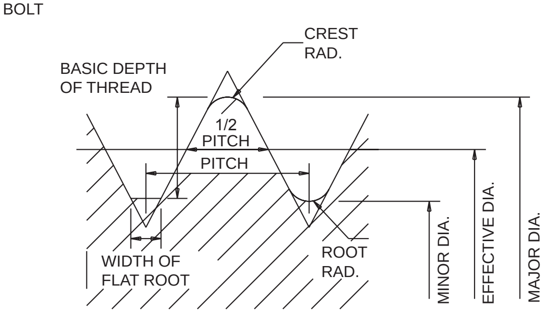
Basic sizes and tolerance classes

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



(b) the minimum major diameter.



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.

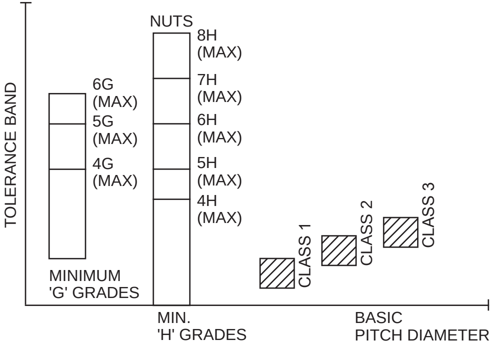
Major Diameter - The minimum major diameter is established by adding the oversize allowance to the basic major diameter (the nominal thread size). Therefore, on measurement, the major diameter of the tap is larger than the nominal thread size, and must not be used to judge the size of the tap.

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

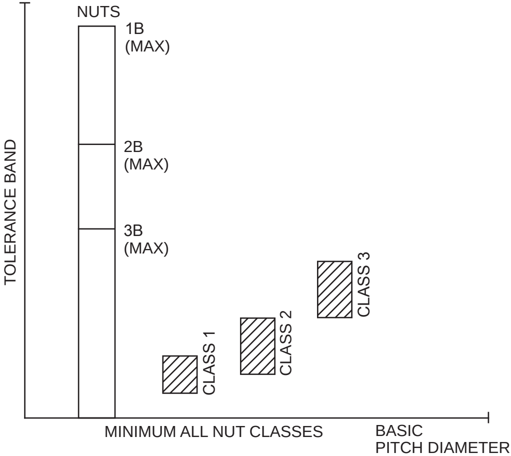
Tap Tolerance Classes

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 TAPS
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.

Damaged tap threads in the hole

(a) **Possible Cause**
Mis-alignment of the tap with the hole.
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 34-35. Use serial taps.

(d) **Possible Cause**
Incorrect rake angle
Solution
Use the recommended tap for the material. See page 54-59.

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 69-78.

(c) **Possible Cause**
The tap is too dull.
Solution
Use a tap which is in 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 with 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 54-59.

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 92-95.

(e) **Possible Cause**
Using the incorrect or unsuitable tap for the material.
Solution
Use the recommended tap for the material. See page 54-59.

(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 54-59.

Excessive Tap Wear

(a) **Possible Cause**
Mis-alignment of the tap with 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) **Possible Cause**
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 54-59.

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 good condition.

(c) **Possible Cause**
Using the incorrect tap.
Solution
Use the recommended tap. See page 54-59.

(d) **Possible Cause**
Excessive tapping speed is applied.
Solution
Use the recommended tapping speed. See page 92-95.

Bell-Mouthed Tapped Hole

(a) **Possible Cause**
Mis-alignment of the tap with 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 69-78.

Over-size tapped hole

(a) **Possible Cause**

Using the incorrect tap.
Solution
Use the recommended tap. See page 54-59.

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

(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.

(d) **Possible Cause**
Incorrect rake angle.
Solution
Use the recommended tap for the material. See page 54-59.

Tap binding in the hole

(a) **Possible Cause**
Using the incorrect tap.
Solution
Use the recommended tap.See page 54-59.

(b) **Possible Cause**
The drilled hole is too small.
Solution
Use the recommended drill size. See page 69-78.

(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 92-95.

(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 54-59.

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 54-59.

(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 drill size. See page 69-78.

(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 with 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 92-95.

(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.
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 54-59.

TAP TECHNICAL DATA

TYPE	GRADE	TYPICAL PHYSICAL PROPERTIES		
		HARD NESS 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		

TAP TECHNICAL DATA *(cont.)*

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 96-97.				2-4	Chlorinated oil
See CBA Tap section pages 96-97.				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

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

TAP TECHNICAL DATA *(cont.)*

TYPE	GRADE	TYPICAL PHYSICAL PROPERTIES		
		HARDNESS BRINELL	TONS PER SQ IN.	N/mm²
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	As Supplied		
	Bronze Manganese			
	Copper Free Machining Copper Electrolytic			
	Bronze Aluminium Bronze Commercial			
	Bronze Phosphor			
PLASTICS	Soft	As Supplied		
	Hard Reinforced			

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
Fluteless	Fluteless	Str/Flute	Str/Flute	15-20 25-30 10-12	Sol. oil or light mineral oil
Sp/Point	Str/Flute	Str/Flute	Str/Flute	3-5	
Fluteless	Fluteless	Sp/Point	Sp/Point	15-20 8-12	Chlorinated oil or soluble oil
Sp/Point	Str/Flute	Str/Flute	Str/Flute	10-12 3-5	
				3-5	Sol. oil or light mineral oil
Str/Flute	Str/Flute	Sp/Point	-	4-7 12-15	Dry

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

CBA TAP 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	-	-

CBA TAP TECHNICAL DATA *(cont.)*

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

TAP PERIPHERAL SPEED TO rpm CONVERSION CHART

Metres / Min		4	6	8	9	10	12
Tap Size		Revolutions per minute					
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

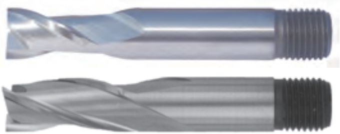
TAP PERIPHERAL SPEED TO rpm CONVERSION CHART *(cont.)*

15	18	21	25	27	30	36
Revolutions 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

SHANK CUTTERS

Selecting the correct Shank Cutter

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.

Two Flute Ball Nose 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.

Cutting tools may shatter
eye protection should be worn

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.

Corner Rounding Cutters

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

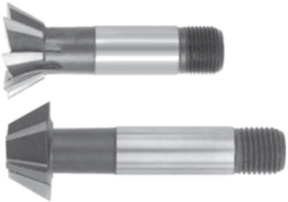
Woodruff Cutters

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.

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.

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.

Solid Carbide Tooling (Standard Range)

Somta's Solid Carbide Tooling Range is a comprehensive family of solid carbide stub and jobber length drills, 2 and 4 flute end mills and ball nose end mills, in both regular and long series.

Solid Carbide Roughing End Mills

A range of high performance roughing end mills for rapid stock removal, with new geometry designed for very low cutting forces. These new roughers are for general engineering applications such as side cutting, slotting and ramping and can be used on machines with medium to low rigidity.

Solid Carbide Finishing End Mills

A range of high performance finishing end mills for operation on hard materials. The finishers are designed for peripheral milling of contours and complex shapes, and are ideal for hardened mould and die steels up to 52HRC (512HB).

Solid Carbide VariCut End Mill

Somta's new “Vari-Style” End Mill, VariCut has a new patent pending tool design. This unique design uses a new core form and a new reinforced end geometry with unequal flute spacing which enables it to remove the most amount of material in the least amount of time with an excellent surface finish.

On Stainless, or Titanium, it will match or outperform any other 4 flute “Vari-Style” End Mill in the market.

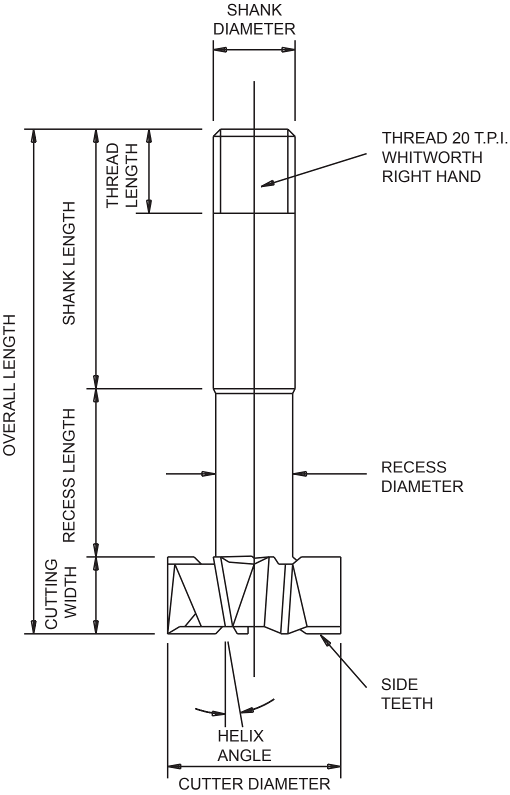
Solid Carbide Hi-Feed-End Mill

Somta’s Hi-Feed end mill with patent pending geometry removes the most amount of material in the least amount of time with extended tool life. It is excellent in 3 dimensional cutting of hard and super hard steels up to 65HRC (880HV).

Solid Carbide End Mills for Aluminium

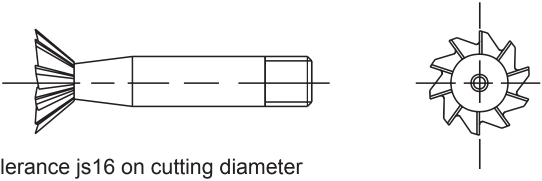
Somta's high efficiency carbide end mill range with optimal flute geometry provides all the required features for high performance machining of aluminium, with the added benefits of greater stock removal rates at high speeds and feeds, excellent surface finish quality and extended tool life.

Shank Cutter Nomenclature



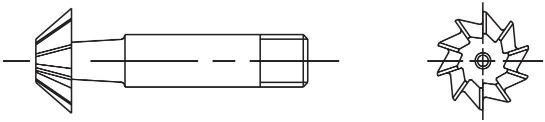
TYPES OF SHANK CUTTERS

Dovetail Cutter



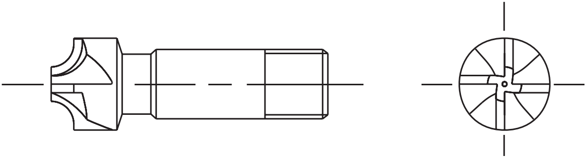
Tolerance js16 on cutting diameter
(see page 151 for tolerance tables)

Inverted Dovetail Cutter



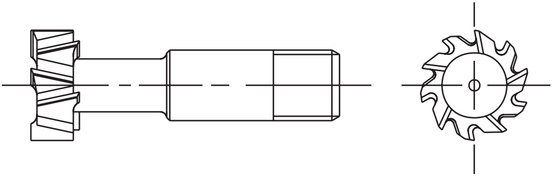
Tolerance js16 on cutting diameter
(see page 151 for tolerance tables)

Corner Rounding Cutter



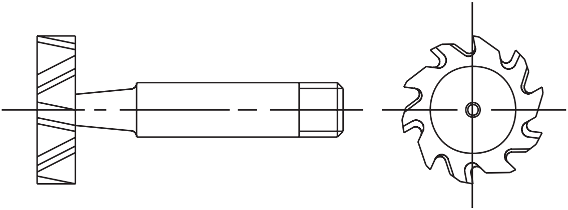
Tolerance H11 on radius and js14 on cutting tip
(see page 151 for tolerance tables)

T-Slot Cutter



Tolerance d11 on metric cutting diameter and width
Tolerance h12 on fractional cutting diameter and width
(see page 151 for tolerance tables)

Woodruff Cutter



Tolerance h11 on metric cutting diameter and e8 on width
(see page 151 for tolerance tables)

Tolerance on fractional diameter is

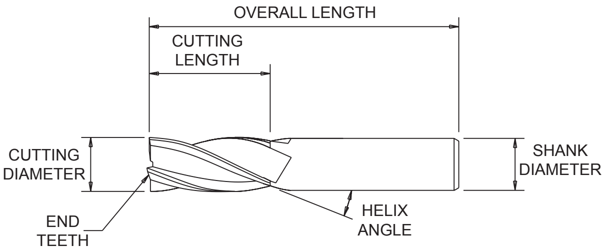
size	+0,381
	+0,127

and on width is

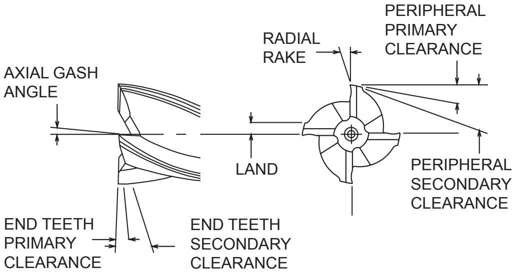
size	+0,000
	- 0,025

If you have any cutting tool problem, please feel free to contact our technical sales representatives.

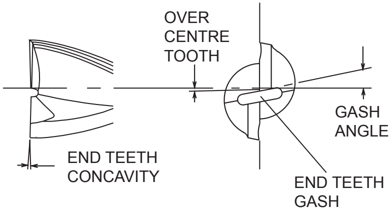
End Mills



OVERALL LENGTH
CUTTING LENGTH
CUTTING DIAMETER
SHANK DIAMETER
END TEETH
HELIX ANGLE



RADIAL RAKE
PERIPHERAL PRIMARY CLEARANCE
AXIAL GASH ANGLE
LAND
PERIPHERAL SECONDARY CLEARANCE
END TEETH PRIMARY CLEARANCE
END TEETH SECONDARY CLEARANCE



OVER CENTRE TOOTH
GASH ANGLE
END TEETH CONCAVITY
END TEETH GASH

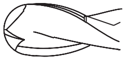
TYPICAL END MILL OPTIONS

Two Flute End Mill

Tolerance e8 on cutting diameter (see page 151 for tolerance tables)

Two Flute Ball Nose End Mill

Tolerance e8 on cutting diameter (see page 151 for tolerance tables)

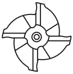


Three Flute End Mill

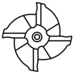
Tolerance e8 on cutting diameter (see page 151 for tolerance tables)

Multi-Flute End Mill

Tolerance k10 on cutting diameter (see page 151 for tolerance tables)



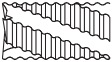
CENTRE CUT

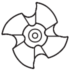


CENTRE HOLE

Roughing End Mill

Tolerance k12 on cutting diameter (see page 151 for tolerance tables)

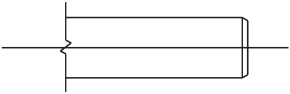




Shank Options

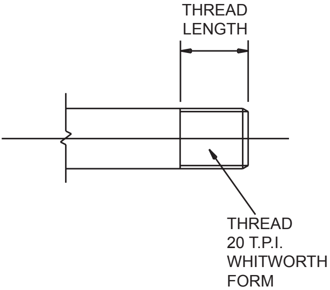
Plain Shank

Tolerance h7 on metric shank diameter (see page 151 for tolerance tables)



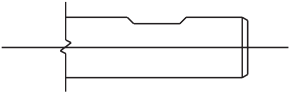
Threaded Shank

Tolerance h8 on metric/ fractional shank diameter (see page 151 for tolerance tables)



Flatted Shank

Tolerance h6 on metric shank diameter (see page 151 for tolerance tables)



Hints for successful Shank Cutter 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 collets regularly ensuring that they are in good condition. The most likely cause of cutter run-out is damaged chucks and collets. Always clean cutter shanks and collets prior to assembly. Check that cutters are running true. 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 129 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 138-141 for technical data.)

Selecting the correct Arbor Mounted Cutter

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 - Plain Form

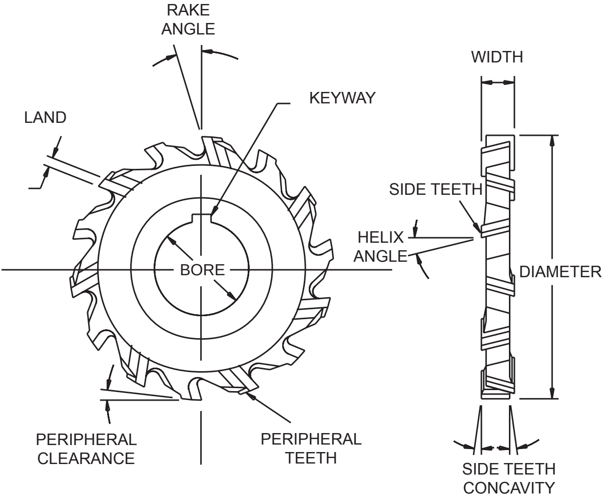
With helical peripheral teeth these cutters fill the gap between normal shank cutters and the much larger facing cutters,this cuttter is better suited to light/medium cuts in a facing or stepping operation with its plain bore.

Shell End Mills - Roughing Form

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.

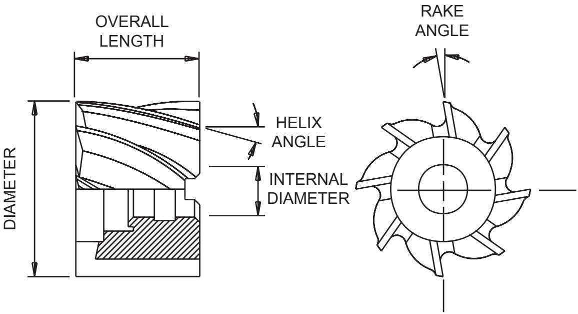
Arbor Mounted Cutter Nomenclature

Side and Face Cutter - (Staggered Tooth shown)



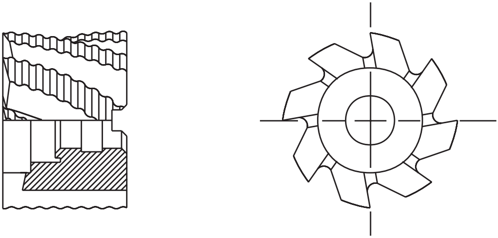
Tolerance js16 on metric cutting diameter and k11 on width
(see page 151 for tolerance tables)

Shell End Mills - Plain Form



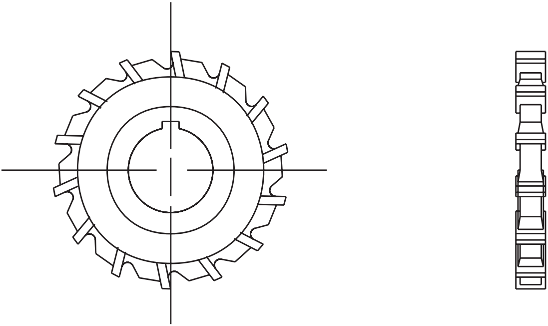
See page 114 for application.

Shell End Mills - Roughing Form



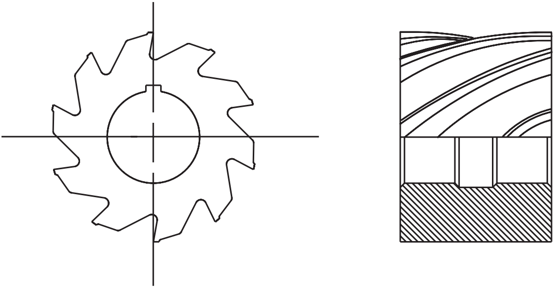
See page 114 for application.

Side and Face Cutter - Straight Tooth



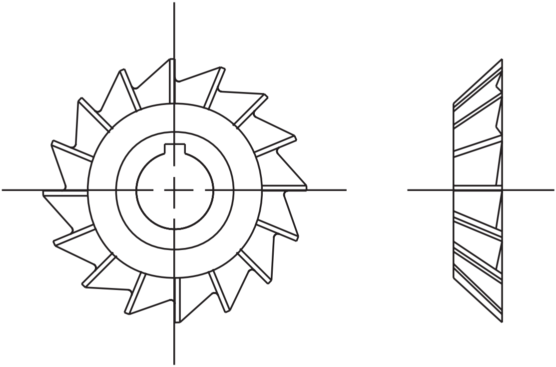
Tolerance js16 on metric/fractional cutting diameter and k11 on metric/fractional width
(see page 151 for tolerance tables)

Cylindrical Cutter



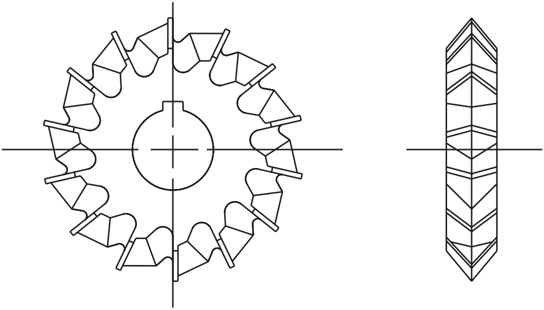
Tolerance js16 on cutting diameter and width
(see page 151 for tolerance tables)

Single Angle Cutter



Tolerance js16 on cutting diameter and js14 on width
(see page 151 for tolerance tables)

Double Angle Cutter



Tolerance js16 on cutting diameter and width
(see page 151 for tolerance tables)

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.

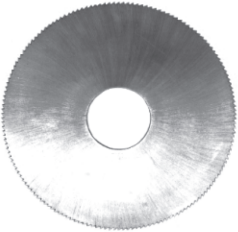
Cleanliness of cutters and arbors is essential.

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

SLITTING SAWS

Selecting the correct Slitting Saw

Slitting Saw - Plain



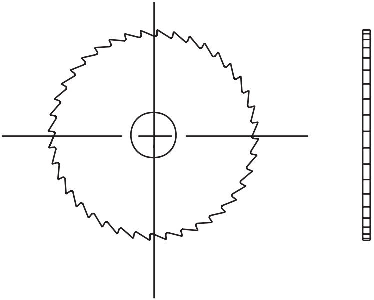
Intended for shallow cutting-off operations, these saws have straight teeth on the periphery and are tapered on width towards the bore to prevent binding. They are available in either coarse or fine pitch to suit the type and section of materials to be cut.

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.

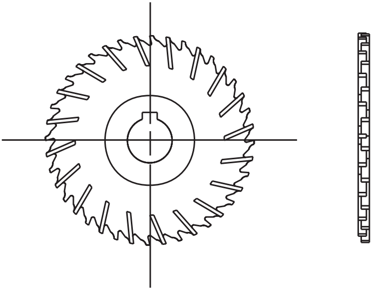
Slitting Saw Nomenclature

Slitting Saw - Plain



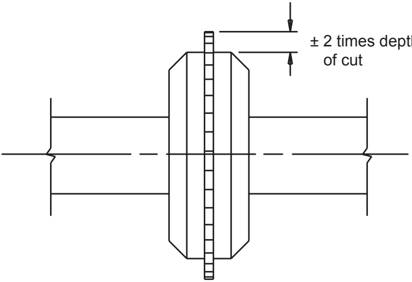
Tolerance js16 on cutting diameter and js10 on width
(see page 151 for tolerance tables)

Slitting Saw - Side Chip Clearance



Tolerance js16 on cutting diameter and js10 on width
(see page 151 for tolerance tables)

Hints for successful Slitting Saw usage



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

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.

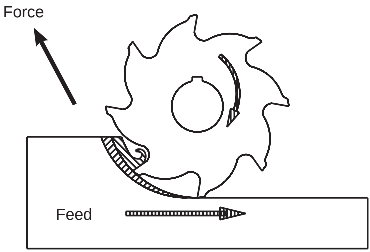
In conventional milling the end mill engages the workpiece at the bottom of the cut. The end mill teeth slide along until sufficient pressure builds up to break through the surface of the work. This sliding action under pressure tends to abrade the periphery of the end mill with resulting dulling.

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.

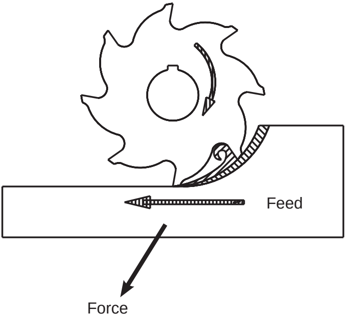
In climb milling the end mill rotates in the direction of the feed. The tooth meets the work at the top of the cut at the thickest portion of the chip. This provides instant engagement of the end mill with the workpiece producing a chip of definite thickness at the start of the cut without the rubbing 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.



Conventional Milling
Engage cut thin



Climb Milling
Engage cut thick

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 138-141 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 129.
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
Workholder 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

If you have any cutting tool problem, please feel free to contact our technical sales representatives.

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 be 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 higher 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.

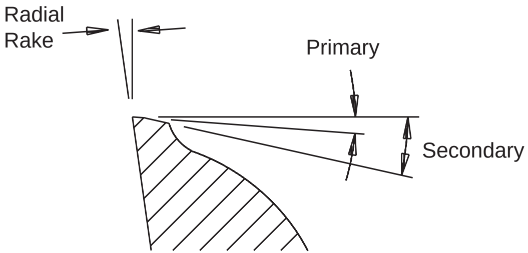
RE-SHARPENING AND CARE OF MILLING CUTTERS

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 138-141.



SOLID CARBIDE END MILLS (STANDARD

Material Type Grade	Hardness HB	Tensile Strength N/mm²	Recommended Surface Speed in m/min for Coated Tungsten Carbide End Mills	
			min	max
Steel				
Free cutting steels	< 120	< 400	**150**	**200**
Structural steel	< 200	< 700	**100**	**150**
Plain carbon steel	< 250	< 850	**80**	**120**
Alloy steel	< 350	< 1200	**50**	**80**
Alloy steel, hardened & tempered steel	> 350	> 1200	**30**	**50**
Stainless Steel				
Free machining	< 250	< 850	**50**	**80**
Austenitic	< 250	< 850	**40**	**70**
Ferritic & martensitic	< 300	< 1000	**35**	**60**
Cast Iron				
Lamellar graphite	< 150	< 500	**80**	**160**
Lamellar graphite	< 300	< 1000	**60**	**120**
Nodular graphite, Malleable cast iron	< 200	< 700	**80**	**160**
Nodular graphite, Malleable cast iron	< 300	< 1000	**60**	**120**
Titanium				
Unalloyed	< 200	< 700	**50**	**80**
Alloyed	< 270	< 900	**40**	**60**
Alloyed	< 350	< 1200	**25**	**40**
Nickel				
Unalloyed	< 150	< 500	**30**	**50**
Alloyed	< 270	< 900	**25**	**40**
Alloyed	< 350	< 1200	**15**	**20**
Copper				
Copper	< 100	< 350	**80**	**120**
Beta brass, bronze	< 200	< 700	**80**	**120**
Alpha brass	< 200	< 700	**80**	**120**
High strength bronze	< 470	< 1500	**50**	**100**
Aluminium Alloys				
Wrought alloys	< 100	< 350	**75**	**135**
Cast alloys < 5% Si	< 150	< 500	**75**	**100**
Cast alloys > 5% Si < 10% Si	< 120	< 400	**45**	**80**
Cast alloys >10% Si	< 120	< 400	**45**	**80**
Synthetics				
Duroplastics (short chipping)	-	-	**100**	**300**
Thermoplastics (long chipping)	-	-	**100**	**300**
Fibre reinforced synthetics	-	-	**100**	**300**

TO CALCULATE:

RPM = (surface speed x 1000) / (π x d)
FEED RATE in mm/min = rpm x feed per tooth x number of teeth

NOTE: For uncoated end mills reduce surface speed by 35% to 50%

RANGE) TECHNICAL DATA

Recommended feed in mm per tooth for Coated Tungsten Carbide End Mills based on full diameter cutting width and half diameter cutting depth (Use 50% of recommended feed rate for long series end mills) End Mill Diameter in mm								
2	4	6	8	10	12	16	20	25
0.050	0.050	0.060	0.080	0.080	0.100	0.100	0.100	0.100
0.050	0.050	0.060	0.080	0.080	0.100	0.100	0.100	0.100
0.050	0.050	0.060	0.080	0.080	0.100	0.100	0.100	0.100
0.050	0.050	0.050	0.050	0.060	0.060	0.070	0.080	0.080
0.050	0.050	0.050	0.050	0.060	0.060	0.070	0.080	0.080
0.008	0.010	0.015	0.020	0.030	0.040	0.050	0.060	0.070
0.008	0.010	0.015	0.020	0.030	0.040	0.050	0.060	0.070
0.006	0.007	0.010	0.015	0.020	0.030	0.040	0.045	0.050
0.050	0.050	0.060	0.080	0.080	0.100	0.100	0.100	0.100
0.050	0.050	0.050	0.060	0.060	0.060	0.070	0.080	0.080
0.050	0.050	0.060	0.080	0.080	0.100	0.100	1.100	0.100
0.050	0.050	0.050	0.060	0.060	0.060	0.070	0.080	0.080
0.080	0.080	0.080	0.080	0.090	0.090	0.100	0.120	0.120
0.050	0.050	0.050	0.050	0.060	0.060	0.070	0.080	0.080
0.050	0.050	0.050	0.050	0.060	0.060	0.070	0.080	0.080
0.030	0.030	0.045	0.045	0.060	0.080	0.080	0.090	0.090
0.020	0.020	0.030	0.030	0.040	0.050	0.050	0.060	0.060
0.020	0.020	0.030	0.030	0.040	0.050	0.050	0.060	0.060
0.050	0.050	0.070	0.070	0.080	0.080	0.090	0.100	0.100
0.050	0.050	0.070	0.070	0.080	0.080	0.090	0.100	0.100
0.050	0.050	0.070	0.070	0.080	0.080	0.090	0.100	0.100
0.025	0.025	0.035	0.035	0.040	0.040	0.045	0.050	0.050
0.050	0.050	0.100	0.200	0.200	0.200	0.300	0.500	0.500
0.050	0.050	0.100	0.200	0.200	0.200	0.300	0.500	0.500
0.050	0.050	0.100	0.200	0.200	0.200	0.300	0.500	0.500
0.050	0.050	0.100	0.200	0.200	0.200	0.300	0.500	0.500
0.050	0.050	0.070	0.080	0.090	0.100	0.120	0.150	0.150
0.050	0.050	0.070	0.080	0.090	0.100	0.120	0.150	0.150
0.050	0.050	0.070	0.080	0.090	0.100	0.120	0.150	0.150

SOLID CARBIDE ROUGHING END

Material Type	Hardness HB	Tensile Strength N/mm²	Recommended Surface Speed in m/min	
			min	max
03D Solid Carbide 3 Flute Roughing End Mill REGULAR LENGTH, FLAT CREST, **03F Solid Carbide 4 Flute Roughing End Mill** REGULAR LENGTH, FLAT CREST,				
Free Cutting Carbon Steel	< 150	< 540	150	200
0.3 to 0.4% Carbon Steel	< 170	< 620	140	190
0.3 to 0.4% Carbon Steel	< 248	< 910	120	160
Alloy Steel	< 330	< 1150	90	150
Hardened Alloy Steel	< 400	-	100	140
Stainless Steel - Martensitic (400 Series)	< 248	< 810	60	100
Stainless Steel - Austenitic (300 Series)	< 300	< 1000	80	100
Grey Cast Irons			120	160
Nodular Cast Irons	110-300	-	110	140
Malleable Cast Irons			100	130
Heat Resisting Alloys	< 350	< 1200	20	40
Commercially Pure Titanium	< 275	< 1000	50	80
Commercially Alloyed Titanium	< 350	< 1200	45	65
03E Solid Carbide 4 Flute Roughing End Mill REGULAR LENGTH, KNUCKLE				
Free Cutting Carbon Steel	< 150	< 540	150	200
0.3 to 0.4% Carbon Steel	< 170	< 620	140	190
0.3 to 0.4% Carbon Steel	< 248	< 910	120	160
Alloy Steel	< 330	< 1150	90	150
Hardened Alloy Steel	< 400	-	80	140
Stainless Steel - Martensitic (400 Series)	< 248	< 810	60	100
Stainless Steel - Austenitic (300 Series)	< 300	< 1000	80	100
Grey Cast Irons			120	160
Nodular Cast Irons	110-300	-	110	140
Malleable Cast Irons			100	130
Commercially Pure Titanium	< 275	< 1000	50	80

SOLID CARBIDE ROUGHING END MILL

Material Type	Hardness HB	Tensile Strength N/mm²	Recommended Surface Speed in m/min	
			min	max
03C Solid Carbide 3 Flute Roughing End Mill REGULAR LENGTH, KNUCKLE				
Aluminium wrought alloys	< 100	< 350	500	2000
Aluminium cast alloys > 5% Si < 10% Si	< 120	< 400	500	1500

MILLS TECHNICAL DATA

Recommended feed in mm per tooth for Carbide End Mills based on 1.0 x D cutting depth with 0.5 x D cutting width (Reduce depth to 0.75 x D for slotting) End Mill Diameter in mm					
6	8	10	12	16	20
COARSE PITCH, COATED FINE PITCH, COATED					
0.044	0.060	0.072	0.083	0.101	0.114
0.044	0.060	0.072	0.083	0.101	0.114
0.036	0.050	0.061	0.070	0.087	0.101
0.033	0.045	0.054	0.062	0.077	0.088
0.033	0.045	0.054	0.062	0.077	0.088
0.029	0.040	0.048	0.056	0.070	0.081
0.036	0.050	0.061	0.070	0.087	0.101
0.044	0.060	0.072	0.083	0.101	0.114
0.036	0.050	0.061	0.070	0.087	0.101
0.029	0.040	0.048	0.056	0.070	0.081
0.019	0.026	0.032	0.037	0.046	0.054
0.029	0.040	0.048	0.056	0.070	0.081
0.026	0.037	0.045	0.052	0.064	0.074
FORM, FINE PITCH, COATED					
0.036	0.049	0.059	0.072	0.087	0.098
0.036	0.049	0.059	0.072	0.087	0.098
0.030	0.041	0.049	0.061	0.075	0.087
0.027	0.037	0.044	0.054	0.066	0.076
0.027	0.037	0.044	0.054	0.066	0.076
0.024	0.033	0.039	0.049	0.060	0.070
0.030	0.041	0.049	0.061	0.075	0.087
0.036	0.049	0.059	0.072	0.087	0.098
0.030	0.041	0.049	0.061	0.075	0.087
0.024	0.033	0.039	0.049	0.060	0.070
0.024	0.033	0.039	0.049	0.060	0.070

FOR ALUMINIUM TECHNICAL DATA

Recommended feed in mm per tooth for Carbide End Mills based on 1.0 x D cutting depth with 0.5 x D cutting width (For slotting up to 1.0 x D, reduce by 30%) End Mill Diameter in mm					
6	8	10	12	16	20
FORM, COARSE PITCH, UNCOATED (FOR ALUMINIUM)					
0.066	0.088	0.110	0.132	0.176	0.220
0.059	0.079	0.099	0.119	0.158	0.198

SOLID CARBIDE END MILLS FOR

Material Type	Hardness HB	Tensile Strength N/mm²	Recommended Surface Speed in m/min	
			min	max
02A Solid Carbide 2 Flute End Mill REGULAR LENGTH, UNCOATED (FOR ALUMINIUM) **02R Solid Carbide 3 Flute End Mill** REGULAR LENGTH, UNCOATED (FOR ALUMINIUM)				
Aluminium wrought alloys **Aluminium cast alloys > 5% Si < 10% Si**	< 100 < 120	< 350 < 400	**500** **500**	**2000** **1500**
02S Solid Carbide 3 Flute Ball Nose End Mill REGULAR LENGTH, UNCOATED (FOR **02U Solid Carbide 3 Flute Toroidal End Mill with Neck** REGULAR LENGTH,				
Aluminium wrought alloys **Aluminium cast alloys > 5% Si < 10% Si**	< 100 < 120	< 350 < 400	**500** **500**	**2000** **1500**

Parameters based on ideal conditions.
For improved surface finish, reduce feed per tooth.

SOLID CARBIDE VARICUT END

Material Type	Hard-ness HB	Tensile Strength N/mm²	Side Milling		Slot-ting
			ap	ae	ap
Free Cutting Carbon Steel **0.3 to 0.4% Carbon Steel** **0.3 to 0.4% Carbon Steel** **Alloy Steel** **Hardened Alloy Steel**	< 150 < 170 < 248 < 330 < 400	< 540 < 620 < 910 < 1150 -	1.5xD 1.5xD 1.5xD 1.5xD 1.5xD	0.5xD 0.5xD 0.5xD 0.5xD 0.5xD	1.5xD 1.5xD 1.5xD 0.75xD 0.75xD
Stainless Steel - Martensitic (400 Series) **Stainless Steel - Austenitic (300 Series)** **Duplex Steel**	< 248 250-450 < 230 240-300 < 270	< 810 820-1350 < 700 710-1000 < 900	1.5xD 1.5xD 1.5xD 1.5xD 1.5xD	0.5xD 0.5xD 0.5xD 0.5xD 0.5xD	1.0xD 0.75xD 1.25xD 1.25xD 1.0xD
Grey Cast Irons **Nodular Cast Irons** **Malleable Cast Irons**	- - -	- - -	1.5xD 1.5xD 1.5xD	0.5xD 0.5xD 0.5xD	1.5xD 1.25xD 1.25xD
Heat Resisting Alloys	< 260 270-350	< 1200 < 1200	1.5xD 1.5xD	0.3xD 0.3xD	0.3xD 0.3xD
Commercially Pure Titanium **Commercially Alloyed Titanium**	< 275 < 350	< 1000 < 1200	1.5xD 1.5xD	0.5xD 0.5xD	1.25xD 1.0xD

Parameters based on ideal conditions.
Please adjust parameter accordingly to real applications.

ALUMINIUM TECHNICAL DATA

Recommended feed in mm per tooth for Carbide End Mills based on 1.0 x D cutting depth with 0.5 x D cutting width (For slotting up to 1.0 x D, reduce depth by 30%) End Mill Diameter in mm									
1	2	3	4	5	6	8	10	12	16
0.014 0.012	0.018 0.016	0.027 0.024	0.036 0.032	0.045 0.041	0.054 0.049	0.072 0.065	0.090 0.081	0.108 0.097	0.144 0.130
ALUMINIUM) **UNCOATED (FOR ALUMINIUM)**									
- -	- -	0.032 0.027	0.041 0.036	0.049 0.045	0.060 0.054	0.080 0.072	0.100 0.090	0.120 0.108	0.160 0.144

MILL TECHNICAL DATA

Recommended Surface Speed in m/min		Recommended feed in mm per tooth for Side Milling (For Slotting reduce by 10% - 20%) End Mill Diameter in mm							
min	max	5	6	8	10	12	16	20	25
150 **140** **120** **90** **80**	**200** **190** **160** **150** **140**	0.036 0.036 0.030 0.027 0.027	0.044 0.044 0.036 0.033 0.033	0.060 0.060 0.049 0.045 0.045	0.072 0.072 0.059 0.054 0.054	0.082 0.082 0.067 0.062 0.062	0.101 0.101 0.083 0.076 0.076	0.114 0.114 0.094 0.086 0.086	0.124 0.124 0.102 0.093 0.093
60 **50** **60** **90** **60**	**100** **75** **80** **115** **70**	0.024 0.021 0.024 0.030 0.021	0.030 0.025 0.030 0.036 0.025	0.041 0.034 0.041 0.049 0.034	0.049 0.041 0.049 0.059 0.041	0.056 0.047 0.056 0.067 0.047	0.069 0.058 0.069 0.083 0.058	0.077 0.065 0.077 0.094 0.065	0.084 0.071 0.084 0.102 0.071
120 **110** **100**	**150** **130** **130**	0.036 0.030 0.024	0.044 0.036 0.030	0.060 0.049 0.041	0.072 0.059 0.049	0.082 0.067 0.056	0.101 0.083 0.069	0.114 0.094 0.077	0.124 0.102 0.084
50 **25**	**90** **40**	0.030 0.017	0.036 0.020	0.049 0.028	0.059 0.033	0.067 0.038	0.083 0.047	0.094 0.053	0.102 0.058
60 **50**	**80** **60**	0.024 0.021	0.030 0.025	0.041 0.034	0.049 0.041	0.056 0.047	0.069 0.058	0.077 0.065	0.084 0.071

Material Type	Rockwell C HRC	Recommended Surface Speed in m/min	
		min	max
03G Solid Carbide 6 Flute Finishing End Mill REGULAR LENGTH, COATED			
Hardened Steels, Irons **Hardened Steels, Irons**	< 48 48-52	**120** **80**	**140** **130**

Material Type	Rockwell C HRC	Recommended Surface Speed in m/min	
		min	max
03I Solid Carbide 2 Flute Ball Nose Finishing End Mill REGULAR LENGTH, COATED			
Hardened Steels, Irons **Hardened Steels, Irons**	< 48 48-52	**290** **200**	**400** **350**
03J Solid Carbide 2 Flute Ball Nose Finishing End Mill LONG SERIES, COATED			
Hardened Steels, Irons **Hardened Steels, Irons**	< 48 48-52	**290** **200**	**400** **350**

Material Type	Rockwell C HRC	Recommended Surface Speed in m/min	
		min	max
03H Solid Carbide Hi-Feed End Mill REGULAR LENGTH, COATED			
Hardened Steels, Irons **Hardened Steels, Irons**	48-52 52-62	**100** **70**	**120** **100**

END MILLS TECHNICAL DATA

Recommended feed in mm per tooth for Coated Carbide End Mills based on 2.0 x D cutting depth with 0.15 x D cutting width End Mill Diameter in mm					
6	8	10	12	16	20
0.036 0.027	0.049 0.037	0.059 0.044	0.069 0.051	0.084 0.063	0.107 0.078

Recommended feed in mm per tooth for Coated Carbide End Mills based on 0.03 x D cutting depth with 0.03 x D cutting width End Mill Diameter in mm						
4	6	8	10	12	16	20
0.100 0.080	0.160 0.120	0.220 0.160	0.260 0.200	0.300 0.230	0.380 0.280	0.430 0.320
0.100 0.080	0.160 0.120	0.220 0.160	0.260 0.200	0.300 0.230	0.380 0.280	0.430 0.320

Recommended feed in mm per tooth for Coated Carbide End Mills based on Ap1 and Ap2 max (For Circular Interpolation note min and max circle diameter range) End Mill Diameter in mm					
6	8	10	12	16	20
0.200 0.150	0.250 0.200	0.300 0.250	0.400 0.300	0.500 0.400	0.600 0.500

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 Std. Grade	As Supplied	
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

CUTTER TECHNICAL DATA (cont.)

PERIPHERAL SPEED RANGE				† 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	12-16 5-10	8-15 4-8	10-20 5-10	8° - 20°	Add 10° to primary	9° - 14°
10-20 5-10	12-16 5-10	8-15 4-8	10-20 5-10			
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°

Explanatory Notes

***Cutter types**
A - Shank Cutters
B - Side and Face Cutters, Single and Double Angle Cutters, Slitting Saws
C - Shell End Mills - Plain Tooth
D - Shell End Mills - Roughing
Note: For Roughing End Mills see page 144-145.

† Cutting Angles
Use higher angles for smaller diameters, reducing proportionately for larger diameters.

CUTTER TECHNICAL DATA (cont.)

MATERIAL TYPE	GRADE	HARDNESS HB	TENSILE STRENGTH N/mm²
TOOL STEEL	HSS Standard Grades HSS Cobalt Grades Hot Working Steel Cold Working Steel	225 250 250 250	735 830 830 830
CAST IRONS	Grey, Malleable Hardened	240 330	800 1137
ALUMINIUM ALLOYS	Wrought Wrought Cast	55 110 100	
COPPER ALLOYS	Brass : Free Cutting Low Leaded Bronze: Silicon Manganese Aluminium Phosphor Copper	As Supplied	
PLASTICS		As Supplied	

CUTTER TECHNICAL DATA (cont.)

PERIPHERAL SPEED RANGE				† CUTTING ANGLES		
TYPE *A	TYPE *B	TYPE *C	TYPE *D	PRIMARY CLEARANCE	SECONDARY CLEARANCE	RADIAL RAKE
10-20	10-20	8-15	10-20	8° - 20°	Add 10° to primary	9° - 14°
10-16	10-20	8-13	10-16			
10-16	10-16	8-13	10-16			
10-16	10-16	8-13	10-16			
16-20	16-20	12-16	20-28	8° - 20°	Add 10° to primary	9° - 14°
12-16	10-14	10-12	16-22			
200-1500	120-180	50-180		10° - 20°	Add 10° to primary	20° - 28°
100-250	100-180	50-100				
40-100	50-70	30-80				
40-70	35-45	30-60		8° - 20°	Add 10° to primary	9° - 14°
50-80	45-70	40-65				
40-70	35-45	30-60				
25-45	20-40	20-35				
15-25	15-25	12-20		8° - 20°		9° - 14°
15-25	15-25	12-20				
40-70	35-45	30-60		10° - 20°		20° - 28°
50-200	50-200			10° - 20°	Add 10° to primary	9° - 14°

Explanatory Notes

***Cutter types**
A - Shank Cutters
B - Side and Face Cutters, Single and Double Angle Cutters, Slitting Saws
C - Shell End Mills - Plain Tooth
D - Shell End Mills - Roughing
Note: For Roughing End Mills see page 144-145.

† Cutting Angles
Use higher angles for smaller diameters, reducing proportionately for larger diameters.

FEEDS PER TOOTH Sz (mm):
End Mills

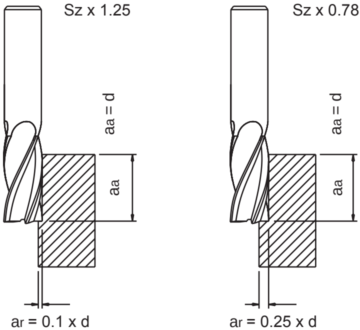
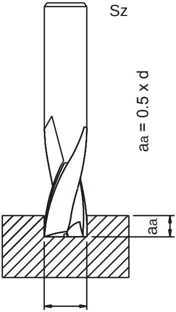


Table Shows Sz Values

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

For Peripheral Speed (m/min) see pages 138-141



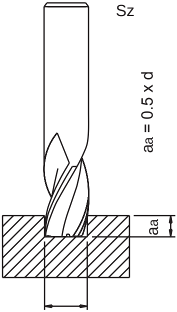


Table Shows Sz Values

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	0.117	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

FEEDS PER TOOTH Sz (mm): Roughing End Mills

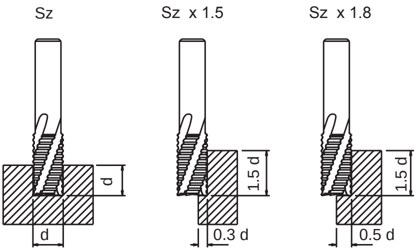


Table Shows Sz Values

End Mill Size	Steels up to 500N/mm² Malleble Cast Iron up to 120 HB	Steels of 500-800 N/mm² Non - Alloyed Tool Steels Pure Titanium	Steels of 800-1200 N/mm² Hot Working Steels Cast Iron of 120 - 180 HB
6	0.008	0.008	0.009
8	0.013	0.013	0.015
10	0.017	0.020	0.020
12	0.023	0.025	0.025
14	0.026	0.030	0.030
16	0.030	0.038	0.038
22	0.032	0.040	0.040
25	0.035	0.042	0.042
28	0.035	0.045	0.042
30	0.040	0.045	0.045
32	0.042	0.050	0.050
35	0.042	0.050	0.050
38	0.045	0.057	0.057
40	0.045	0.057	0.057
45	0.047	0.059	0.060
50	0.060	0.074	0.075
Peripheral Speed (m/min)	28 - 40	24 - 32	18 - 25

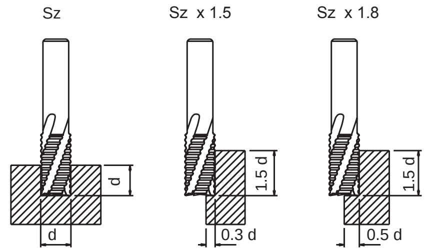


Table Shows Sz Values

Stainless Steels Titanium Alloys (Annealed) Cast Iron of more than 180 HB	Titanium Alloys (Hard-ened)	Brass and Bronze (Cast)	Brass and Bronze (Rolled)	Plastics and similar
0.010	0.013	0.008	0.006	0.006
0.015	0.020	0.012	0.009	0.009
0.021	0.030	0.017	0.013	0.012
0.033	0.037	0.024	0.016	0.013
0.037	0.047	0.026	0.021	0.015
0.044	0.053	0.033	0.024	0.019
0.048	0.060	0.038	0.025	0.022
0.050	0.063	0.040	0.028	0.025
0.050	0.065	0.040	0.028	0.025
0.056	0.068	0.040	0.030	0.028
0.064	0.080	0.044	0.036	0.035
0.064	0.080	0.044	0.036	0.035
0.070	0.086	0.048	0.040	0.035
0.070	0.090	0.048	0.040	0.038
0.075	0.094	0.048	0.042	0.040
0.090	0.119	0.060	0.052	0.047
12 - 18	7 - 12	35 - 45	45 - 70	200 - 250

FEEDS PER TOOTH Sz (mm):
Side and Face Cutters - Staggered Tooth

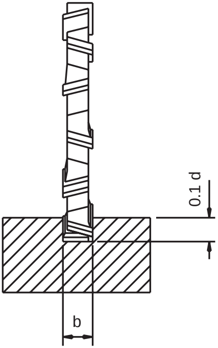


Table Shows Sz Values

Cutter Diameter	Cutter Width		Steels up to 500N/mm² Malleble Cast Iron up to 120 HB	Steels of 500-800 N/mm² Non - Alloyed Tool Steels Pure Titanium	Steels of 800-1200 N/mm² Hot Working Steels Cast Iron of 120 - 180 HB
	over	to			
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	0.100
200	8	18	0.093	0.093	0.102
	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

For Peripheral Speed (m/min) see pages 138-141

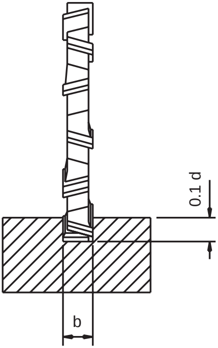


Table Shows Sz Values

Stainless Steels Titanium Alloys (Annealed) Cast Iron of more than 180 HB	Titanium Alloys (Hardened)	Brass and Bronze (Cast)	Brass and Bronze (Rolled)	Plastics and similar
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

FEEDS PER TOOTH Sz (mm):
Shell End Mills

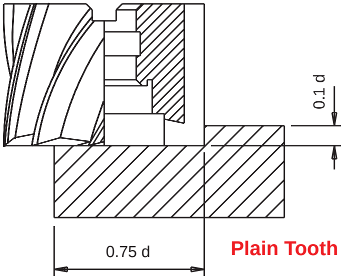
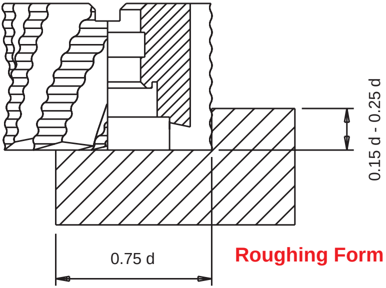


Table Shows Sz Values

Type	Cutter Diameter	Steels up to 500N/mm² Malleble Cast Iron up to 120 HB	Steels of 500-800 N/mm² Non - Alloyed Tool Steels Pure Titanium	Steels of 800-1200 N/mm² Hot Working Steels Cast Iron of 120 - 180 HB
Plain Tooth	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 Form	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

For Peripheral Speed (m/min) see pages 138-141



Roughing Form

Table Shows Sz Values

Stainless Steels Titanium Alloys (Annealed) Cast Iron of more than 180 HB	Titanium Alloys (Hard-ened)	Brass and Bronze (Cast)	Brass and Bronze (Rolled)	Plastics and similar
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)

TOLERANCES

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
K12	+100 -0	+120 -0	+150 -0	+180 -0	+210 -0	+250 -0	+300 -0	+350 -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

PERIPHERAL SPEED TO rpm CONVERSION CHART

METRES PER MIN	5	10	20	30	40
Dia. mm	Revolutions per Minute				
1	1591	3182	6364	9546	12728
2	795	1590	3182	4770	6360
3	530	1060	2120	3180	4240
4	398	795	1590	2385	3180
5	318	636	1272	1908	2544
6	265	530	1060	1590	2120
7	227	455	910	1365	1820
8	199	398	796	1194	1592
9	177	353	706	1059	1412
10	159	318	636	954	1272
11	145	289	578	867	1156
12	133	265	530	795	1060
13	122	245	490	735	980
14	114	227	454	681	908
15	106	212	424	636	848
16	100	199	398	597	796
18	89	177	354	531	708
20	80	159	318	477	636
22	73	145	290	435	580
24	67	133	266	399	532
26	61	122	244	366	488
28	57	113	228	342	456
30	53	106	212	318	424
35	45	91	182	273	364
40	40	80	160	240	320
45	35	70	140	210	280
50	32	64	128	192	256
63	25	50	100	150	200
75	21	42	84	126	168
100	16	32	64	96	128

PERIPHERAL SPEED TO rpm CONVERSION CHART *(cont.)*

Revolutions per Minute

50	60	70	80	90	100
15910 7950	19092 9540	22274 11130	25456 12720	28638 14310	31820 15900
5300 3975	6360 4770	7420 5565	8480 6360	9540 7155	10600 7950
3180 2650	3816 3180	4452 3710	5088 4240	5724 4770	6360 5300
2275 1990	2730 2388	3185 2786	3640 3184	4095 3582	4550 3980
1765 1590	2118 1908	2471 2226	2824 2544	3177 2862	3530 3180
1445 1325	1734 1590	2023 1855	2312 2120	2601 2385	2890 2650
1225 1135	1470 1362	1715 1589	1960 1816	2205 2043	2450 2270
1060 995	1272 1194	1484 1393	1696 1592	1908 1791	2120 1990
885 795	1062 954	1239 1113	1416 1272	1593 1431	1770 1590
725 665	870 798	1015 931	1160 1064	1305 1197	1450 1330
610 570	732 684	854 798	976 912	1098 1026	1220 1140
530 455	636 546	742 637	848 728	954 819	1060 910
400 350	480 420	560 490	640 560	720 630	800 700
320 250	384 300	448 350	512 400	576 450	640 500
210 160	252 192	294 224	336 256	378 288	420 320

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	93.266
11/16	17.462	42.862	68.262	93.662
45/64	17.859	43.859	68.859	94.059
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	48.419	73.819	99.219
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	450
75	—	140	130	31	480
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

HSS (M2)	823-876 HV30 - 63-65 HRC
HSS-Co5 (M35)	849-920 HV30 - 64-66 HRC
HSS-Co8 (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.

USEFUL FORMULAE

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

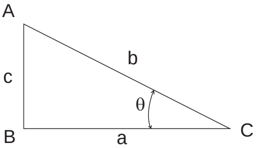
$$\text{Surface Speed (metres/min)} = \frac{\text{Dia (mm)} \times \pi}{1000} \times \text{rpm}$$

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

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

Trigonometry

Formulae for the solution of RIGHT ANGLED TRIANGLES

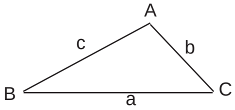


$$\text{Tan } \theta = \frac{\text{opposite}}{\text{adjacent}} = \frac{c}{a}$$

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

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

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:

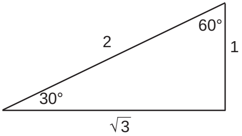
$a^2 = b^2 + c^2 - 2bc \quad \text{Cos A}$

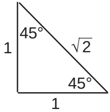
$b^2 = a^2 + c^2 - 2ac \quad \text{Cos B}$

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

USEFUL VALUES IN TRIGNOMETRICAL RATIOS

For right angled triangles





ANGLES 30° - 45° - 60°

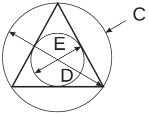
θ	Tan θ	Sin θ	Cos θ
30°	$\frac{1}{\sqrt{3}}$ = 0.577350	$\frac{1}{2}$ = 0.500000	$\frac{\sqrt{3}}{2}$ = 0.866025
45°	1	$\frac{1}{\sqrt{2}}$ = 0.707107	$\frac{1}{\sqrt{2}}$ = 0.707107
60°	$\sqrt{3}$ = 1.732051	$\frac{\sqrt{3}}{2}$ = 0.866025	$\frac{1}{2}$ = 0.500000

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.

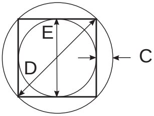
TRIANGLE

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



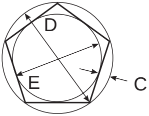
SQUARE

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



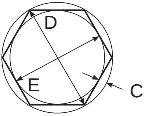
PENTAGON

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



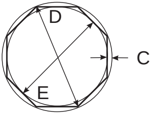
HEXAGON

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

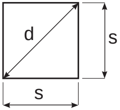
E	= side x 2.4142 = D x 0.9239
D	= side x 2.6131 = E x 1.0824
Side	= D x 0.3827
C	= D x 0.038



Areas of Plane Figures

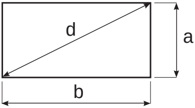
SQUARE

A = area
A = $S^2 = 1/2\ d^2$
S = $0.7071d = \sqrt{A}$
d = $1.414S = 1.414\sqrt{A}$



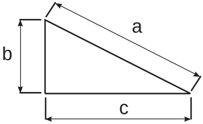
RECTANGLE

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
$A = \frac{\mathbf{bc}}{\mathbf{2}}$
$a = \sqrt{b^2 + c^2}$
$b = \sqrt{a^2 - c^2}$
$c = \sqrt{a^2 - b^2}$



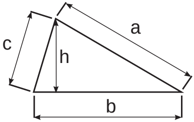
ACUTE ANGLED TRIANGLE

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,

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



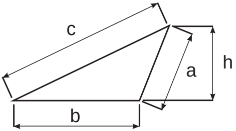
OBTUSE ANGLED TRIANGLE

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

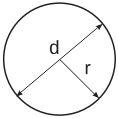
$A = \pi r^2 = 3.1416\, r^2$

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

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

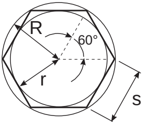
$r = C \div 6.2832 = \sqrt{A \div 3.1416} = 0.564\sqrt{A}$

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



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 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	

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)	

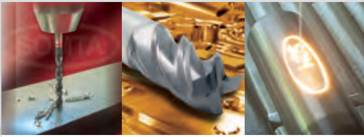
NUMBER AND LETTER DRILL SIZES

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		

NOTES

NOTES



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

Head Office and Surface Coating Division
Somta House, 290-294 Moses Mabhida (Edendale) Road, Pietermaritzburg, 3201
Private Bag X401, Pietermaritzburg, 3200
South Africa

Tel: Factory: +27 33 355 6600
Fax: Factory: +27 33 394 0564
Tel: Sales: +27 11 390 8700 (Local)
Fax: Sales: +27 11 397 6720/1 (Local)
Email: jhbsales@somta.co.za (Local)
Tel: Sales: +27 33 355 6600 (Exports)
Fax: Sales: +27 33 394 7509 (Exports)
Email: exports@somta.co.za (Exports)

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

Tel: +27 11 390 8700
Fax: +27 11 397 6720/1
Sharecall: 086 010 4367
Email: jhbsales@somta.co.za

Technical Information:
Email: tech@somta.co.za
Toll Free Number: 0800-331-399
Technical Representatives Cell Numbers:
082 801 8930, 082 800 5259, 082 906 3486
082 806 8202, 082 803 0505

http://app.somta.co.za

YEAR OF PRINT: 2017

SOMTA
World Class Cutting Tools
OSG GROUP COMPANY
www.somta.co.za

SABS ISO 9001
EST. 1954
oerlikon balzers