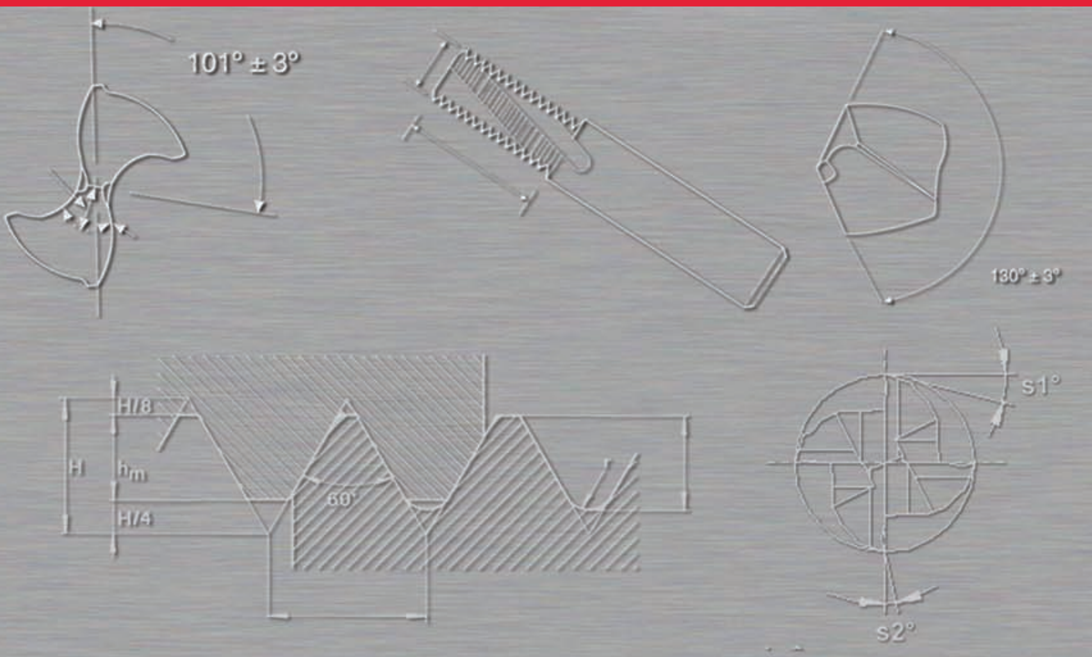
DORMER

Innovation starts here ™

Technical Handbook



DORMER

Dormer Tools
Shireoaks Road
Worksop
S80 3HB
UK
Phone: 0870 850 44 66
Fax: 0870 850 88 66
dormer.uk@dormertools.com

Dormer Tools Inc.
6835 Century Avenue
Mississauga, Ontario
L5N 2L2
CANADA
Phone: 905-542-3000
Fax: 905-542-7000
dormer.ca@dormertools.com

Dormer Tools LLC
2300 Global Way
Hebron, Kentucky
41048
USA
Phone: 800-243-4445
Fax: 800-356-9481
dormer.us@dormertools.com

Dormer Tools
Caixa Postal 413
CEP 01059-970
Sao Paulo SP
BRAZIL
Phone: 55 (0)11-56 60 30 00
Fax: 55 (0)11-56 67 22 67
dormer.br@dormertools.com

Dormer Tools International
Shireoaks Road
Worksop
S80 3HB
UK
Phone: 44 1909 534700
Fax: 44 1909 534703
dormer.int@dormertools.com

Dormer Tools
1-3 Fowler Road
Dandenong 3175, Victoria
AUSTRALIA
Phone: 1300 131 274
Fax: 61 3 9238 7105
dormer.int@dormertools.com

Dormer Tools
Sandvik New Zealand
269 Ti Rakau Drive
East Tamaki 1701
Auckland
NEW ZEALAND
Phone: 64 9 2735858
Fax: 64 9 2735857
dormer.int@dormertools.com

Dormer Tools
1120, Nippa-cho
Kohoku-ku, Yokohama 223-0057
JAPAN
Phone: 81 45 544 9280
Fax: 81 45 544 9284
dormer.int@dormertools.com

Dormer Tools
B.P 6209
45062 Orleans Cedex 2
FRANCE
Phone: 33 (0)2 38 41 40 15
Fax: 33 (0)2 38 41 40 30
dormer.fr@dormertools.com

Dormer Tools
Via Varesina 184
20156 Milano
ITALY
Phone: 39 02 38 04 51
Fax: 39 02 38 04 52 43
dormer.it@dormertools.com

Dormer Tools
Postbus 344
NL-3900 AH Veenendaal
NETHERLANDS
Phone: 31 318 569 569
Fax: 31 318 569 579
dormer.nl@dormertools.com

Dormer Tools
Steenweg op Leuven 1188
BE-1200 Brussel
BELGIUM
Phone: 32 3 440 59 01
Fax: 32 3 449 15 43
dormer.be@dormertools.com

Dormer Tools AB
Box 618
SE-301 16 Halmstad
SWEDEN
Phone: 46 (0) 35 16 52 00
Fax: 46 (0) 35 16 52 90
dormer.se@dormertools.com

Dormer Tools
Robert Huberintie 1
FI-01510 Vantaa
FINLAND
Phone: 358 9 870 66 341
Fax: 358 9 870 66 342
dormer.fi@dormertools.com

Dormer Tools
Sandvik A/S
Postboks 54
N-1306 Bærum
NORWAY
Phone: 47 67 17 56 00
Fax: 47 67 17 56 01
dormer.no@dormertools.com

Dormer Tools
Sandvik A/S
Postboks 160
DK-2605 Brøndby
DENMARK
Phone: +45 43 46 52 80
Fax: +45 43 46 52 81
dormer.dk@dormertools.com

Dormer Tools
Heerdter Landstraße 231
DE-40549 Düsseldorf
Postfach 10 21 62
DE-40012 Düsseldorf
GERMANY
Phone: 49 211 950 90 0
Fax: 49 211 950 90 52
dormer.de@dormertools.com

Dormer Tools
Av. Do Forte, No. 3,
Edificio SUECIA IV
Piso 0
2790-073 Carnaxide
PORTUGAL
Phone: 351 21 424 54 20
Fax: 351 21 424 54 25
dormer.pt@dormertools.com

Dormer Tools
C/Bigues, 15
ES-08140 Caldes de Montbui
(Barcelona)
SPAIN
Phone: 34 93 862 98 02
Fax: 34 93 862 98 00
dormer.es@dormertools.com

Dormer Tools
Sandvik
P.O. Box 25038
East Rand 1462
SOUTH AFRICA
Phone: 27-11-929-5300
Fax: 27-11-570-9709
dormer.int@dormertools.com

Dormer Tools International
Sandvik in Austria
Gesellschaft m.b.H.
Postfach 90
AT-1211 Wien
Street address:
Scheydgasse 44
AT-1211 Wien
AUSTRIA
Phone: 43 1 277 37 202
Fax: 43 1 277 37 203
dormer.int@dormertools.com

Dormer Tools
Sandvik Malaysia Sdn Bhd
No 5 Jalan 13/6
P.O. Box 213
46720 Petaling Jaya
Selangor
MALAYSIA
Phone: 60 3 7956 2136
Fax: 60 3 7956 2372
& 60 3 7957 3132
dormer.my@dormertools.com

Dormer Tools
Sandvik Thailand Ltd
4th Floor Chamnan
Phenjati Business Center Bldg
65/48 Rama 9 Rd
Huay-Kwang
Bangkok 10320
THAILAND
dormer.my@dormertools.com

Dormer Tools
Sandvik China Ltd
No 4555 Yin Du Road
Xin Zhuang Industrial Park
Shanghai 201108
CHINA
Phone: 86 21 5442 6866
Fax: 86 21 5442 6315
dormer.my@dormertools.com

Dormer Tools
Sandvik A.E.
294 Kifissias Avenue
152 32 Chalandri
Athens
GREECE
Phone: 30 210 6823604
Fax: 30 210 6823771
dormer.gr@dormertools.com

Dormer Tools
Sandvik Asia Ltd
Mumbai-Pune Road
Pune 411 012
INDIA
Phone 91 20 27 10 47 00
Fax 91 20 27 14 57 36
dormer.int@dormertools.com

Dormer Tools
P.T. Sandvik Indonesia
Mugi Griya Building
Lt. 3 Jl. M.T. Haryono Kav 10
12810 Jakarta
INDONESIA
Phone: 62 21 8308 411
Fax: 62 21 8308 410
dormer.my@dormertools.com

Dormer Tools
Sandvik Taiwan Ltd
Room 502 No.6 Lane 7
Wucyuan Rd
Wu-Ku Ind Park, zip code 248
Taipei Hsien
TAIWAN ROC
Phone: 886 2 2298 0999
Fax: 886 2 2299 2350
dormer.my@dormertools.com

Dormer Tools
Sandvik South East Asia Pte Ltd
111 Neythal Road
628598 SINGAPORE
Phone: 65 2 6319 0690
Fax: 65 2 6266 2849
dormer.my@dormertools.com

Dormer Tools
Sandvik South East Asia Pte Ltd
(Vietnam Branch)
190 Tran Quoc Thao Street
District 3, Ho Chi Minh City
VIETNAM
Phone: 84 8 9310 817
Fax: 84 8 8465 750
dormer.my@dormertools.com

Dormer Tools
Sandvik Philippines Inc
KM 20 East Service Rd,
South Superhighway
1770 Muntinlupa City
PHILIPPINES
Phone: 632 8076 372
Fax: 632 8076 383
dormer.my@dormertools.com

Dormer Tools
Sandvik Hong Kong Ltd
5th Floor, 9 Wing Hong Street
Cheung Sha Wan
Kowloon
HONG KONG
Phone: 852 27 43 33 31
Fax: 852 27 85 09 77
dormer.my@dormertools.com

Dormer Tools
Sandvik Korea Ltd
26-4 Yeouido-dong
Youngdeuongpo-ku Seoul
KOREA
dormer.my@dormertools.com

Advancement
For over 90 years, Dormer has been a leading name in cutting tool technology, constantly at the forefront of tool manufacturing, to meet the tough and ever-changing demands of engineering.

Innovation
As machine tools evolve and material technology develops, Dormer rises to the challenge every time with new products tailored to specific applications.

Commitment to Quality
Committed to providing cost effective and highly precise solutions in drilling, milling threading and reaming, Dormer's dedicated technical experts work together to design and produce tools exclusively for Dormer to stringent quality standards. Dormer has achieved ISO 9001 2000 and ISO 14000 in its main manufacturing units, which means that you can rest assured that quality and the Dormer brand are synonymous.

Research and Development
Industry-specific end user forums, customer visits to Dormer's manufacturing plants and day-to-day contact by Dormer sales engineers with customers allow for an information exchange, which shapes the company's research and development programmes.

Global Presence
Dormer's worldwide network of sales companies and distributors ensures that a Dormer representative is never far away from even the most remote customers. New sales companies and divisional units have recently opened in many parts of the world.

Next Day Deliveries
The European Distribution Centre has long been providing next day deliveries to most parts of Europe. Two new warehouses now supply America and the Asia Pacific regions, making 24-48 hour deliveries a reality for even more Dormer customers.

The concept of the new Dormer technical handbook projects innovations in tool materials and surface treatments, material and machine technology and tool design. It is for use by professional engineers and technicians. Use it independently for reference and information, or in conjunction with Dormer's latest catalogue, which shows the entire range of Dormer's standard stocked tools. Specials and tool modifications are of course all part of our service to you. In addition to the catalogue and the technical handbook, the Product Selector CD has been updated with the latest new products and will assist you in selecting the optimum tool for your particular application. You can also refer to our website at www.dormertools.com.

For ease of use, the information in the technical handbook is arranged according to the machining process you are carrying out, with a general information section and a regrinding section. At the end of each main section, a trouble-shooting table examines the causes of problems during machining. It suggests possible remedies and advises correct tool usage, to get you back on track to achieve your production goals and deadlines.

Of course there will be times when you prefer to seek the professional advice of your local Dormer sales engineer and you can be sure that they are only a phone call away. All are well qualified to advise you on your particular application and are backed up by an international group of Dormer product specialists, to give you the most favourable solution.

Table of Contents

Threading with taps (continued)

General Information

Formulae (Metric)

DRILLING

RPM

$$n = \frac{V_c * 1000}{\pi * D}$$

n = RPM

V_C = cutting speed (m/min.)

D = diameter (mm)

Table feed

$$V_f = n * fn$$

V_f = feed rate (mm/min.)

n = r/min (RPM)

fn = feed/rev

Thrust, Axial Force

$$T = 11.4 * K * D * (100 * fn)^{0.85}$$

Power

$$P = \frac{1.25 * D^2 * K * n * (0.056 + 1.5 * fn)}{100,000}$$

To convert to HP multiply by 1.341

P = Power (kW)

K = material factor

T = thrust (N)

D = diameter (mm)

V_f = rate of feed (mm/min.)

n = r/min (RPM)

fn = feed/rev

MILLING

RPM

$$n = \frac{V_c * 1000}{\pi * D}$$

n = RPM

V_c = cutting speed m/min.

D = diameter in mm

Table feed

$$V_f = n * f_Z * z$$

V_f = feed rate mm/min.

f_Z = feed/tooth

z = no. of teeth

Torque

$$M_c = \frac{a_p * a_e * v_f * k_c}{2 \pi * n}$$

Mc=Cutting Torque [Nm]

a_p = axial depth [mm]

a_e = radial depth [mm]

Power

$$P_c = \frac{a_p * a_e * v_f * k_c}{60 * 102 * 9{,}81}$$

Pc= Cutting Power [W]

n = RPM

kc=specific cutting force [N/mm²]

$$k_c = k_{c1} * h_m^{-z}$$

h_m = average chip thickness [mm or inch]

z = correction factor joined to average chip thickness

kc=specific cutting force [N/mm²]

kc1 =specific cutting force related to a 1 mm h_m

where

$$h_m = \frac{fz * ae * 360}{D * \pi * \arccos[1 - \frac{2 * a_e}{D}]}$$

THREADING

RPM

$$n = \frac{V_c * 1000}{\pi * D}$$

Torque Calculations

$$M_d = \frac{p^2 * D * k_c}{8000}$$

Power

$$P = \frac{M_d * 2 * \pi * n}{60}$$

M_d = Torque [Nm]

p = pitch [mm]

D = nominal diam. [mm]

k_C = specific cutting force [N/mm²]

n = RPM

P = Power (kW)

Formulae (Imperial)

DRILLING

RPM

$$n = \frac{12 * V_c}{\pi * D_c}$$

n = RPM

V_c = cutting speed (ft/min.)

D_c = Cutting diameter (inches)

Table feed

$$V_f = n * fn$$

V_f = feed rate (inch/min)

n = r/min (RPM)

fn = feed/rev (inch)

MILLING

RPM

$$n = \frac{12 * V_c}{\pi * D_c}$$

n = RPM

V_c = cutting speed (ft/min.)

D_c = Cutting diameter (inches)

Table feed

$$V_f = f_z * n * z$$

V_f = feed rate (inch/min)

f_z = feed per tooth (inches)

n = r/min (RPM)

z = no. of teeth

General Information

Specific Cutting Force (Kc Value)

			Drilling	Milling		Threading
			k	k_{C1}	z	k_C
Application Material Groups			Material factor	N/mm²	Correction factor	N/mm²
1. Steel	1.1	Magnetic soft steel	1,3	1400	0,18	2000
	1.2	Structural steel, case carburizing steel	1,4	1450	0,22	2100
	1.3	Plain Carbon steel	1,9	1500	0,20	2200
	1.4	Alloy steel	1,9	1550	0,20	2400
	1.5	Alloy steel, Hardened and tempered steel	2,7	1600	0,20	2500
	1.6	Alloy steel, Hardened and tempered steel	3,4	1700	0,20	2600
	1.7	Alloy steel, Heat treated	3,7	1900	0,20	2900
	1.8	Alloy steel, Hardened & Wear resistant steel	4,0	2300	0,20	2900
2. Stainless Steel	2.1	Free machining, Stainless Steel	1,9	1300	0,36	2300
	2.2.	Austenitic,	1,9	1500	0,32	2600
	2.3	Ferritic + Austenitic, Ferritic, Martensitic	2,7	1600	0,24	3000
3. Cast Iron	3.1	Lamellar graphite	1,0	900	0,26	1600
	3.2	Lamellar graphite	1,5	1100	0,26	1600
	3.3	Nodular graphite, Malleable Cast Iron	2,0	1150	0,24	1700
	3.4	Nodular graphite, Malleable Cast Iron	1,5	1450	0,24	2000
4. Titanium	4.1	Titanium, unalloyed	1,4	900	0,20	2000
	4.2	Titanium, alloyed	2,0	1200	0,22	2000
	4.3	Titanium, alloyed	2,7	1450	0,22	2300
5. Nickel	5.1	Nickel, unalloyed	1,3	1100	0,12	1300
	5.2	Nickel, alloyed	2,0	1450	0,22	2000
	5.3	Nickel, alloyed	2,7	1700	0,22	2000
6. Copper	6.1	Copper	0,6	450	0,20	800
	6.2	β-Brass, Bronze	0,7	500	0,30	1000
	6.3	α-Brass	0,7	600	0,32	1000
	6.4	High Strength Bronze	1,5	1600	0,36	1000
7. Aluminium Magnesium	7.1	Al, Mg, unalloyed	0,6	250	0,22	700
	7.2	Al alloyed, Si < 0.5%	0,6	450	0,18	700
	7.3	Al alloyed, Si > 0.5% < 10%	0,7	450	0,18	800
	7.4	Al alloyed, Si > 10% Whisker reinforced Al-alloys Mg-alloys	0,7	500	0,15	1000
8. Synthetic materials	8.1	Thermoplastics	0,6	1400	0,15	400
	8.2	Thermosetting plastics	0,6	1400	0,20	600
	8.3	Reinforced plastic materials	1,0	1600	0,30	800
9. Hard material	9,1	Cermets (metals-ceramics)	4,0	2600	0,38	>2800
10. Graphite	10.1	Graphite	-	200	0,30	600

Cutting Tool Material

HIGH SPEED STEEL MATERIALS

HSS **High Speed Steel**

A medium-alloyed high speed steel that has good machinability and good performance. HSS exhibits hardness, toughness and wear resistance characteristics that make it attractive in a wide range of applications, for example in drills and taps.

HSSV **Vanadium High Speed Steel**

A vanadium-based grade that offers excellent wear resistance and hardness and good performance. This makes it especially good for use in tapping applications.

HSCo **Cobalt High Speed Steel**

This high speed steel contains cobalt for increased hot hardness. The composition of HSCo is a good combination of toughness and hardness. It has good machinability and good wear resistance, which makes it usable for drills, taps, milling cutters and reamers.

HSS XS1 **Non Cobalt Powder Metallurgy Steel**

Has a finer and more consistent grain structure than HSCo resulting in a tougher product. Tool life and wear resistance is normally higher than HSCo and this grade has superior edge strength and rigidity. Mainly used for milling cutters and taps.

HSCo XP **Sintered Cobalt High Speed Steel**

HSCo-XP is a Cobalt high speed steel which has been produced using powder metallurgy technology. High speed steel produced by this method exhibits superior toughness and grindability. Taps and milling cutters find particular advantage when manufactured from XP grade steel.

CS **Chromium Steel**

Chromium steel is a tool steel in which the principal alloying element is Chromium. It is used only for the manufacture of taps and dies. This steel has lower hot hardness properties in comparison with high speed steels. Suited for hand tap applications.

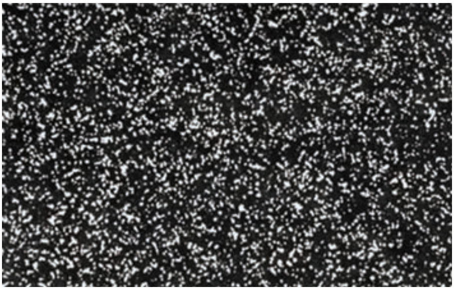
General Information

Material structure

Example on material structure for different HSS materials.

Steels produced with powder metallurgy technology (eg. HSCo-XP) will have a finer grain structure, resulting in a material with higher toughness and wear resistance.

HSS



HSCo-XP

The main steels used by Dormer include

	Grade	Hardness (HV10)	C %	W %	Mo %	Cr %	V %	Co %	ISO standard
HSS	M2	810-850	0,9	6,4	5,0	4,2	1,8	-	HSS
HSSV	M9V	830-870	1,25	3,5	8,5	4,2	2,7	-	HSS-E
HSCo	M35	830-870	0,93	6,4	5,0	4,2	1,8	4,8	HSS-E
	M42	870-960	1,08	1,5	9,4	3,9	1,2	8,0	HSS-E
HSS XS1	-	830-870	0,9	6,25	5,0	4,2	1,9	-	HSS-PM
HSCo XP	ASP 2017	860-900	0,8	3,0	3,0	4,0	1,0	8,0	HSS-E-PM
	ASP 2030	870-910	1,28	6,4	5,0	4,2	3,1	8,5	HSS-E-PM
	ASP 2052	870-910	1,6	10,5	2,0	4,8	5,0	8,0	HSS-E-PM
CS	-	775-825	1,03	-	-	1,5	-	-	-

CARBIDE MATERIALS

HM **Carbide Materials (or Hard Materials)**

A sintered powder metallurgy steel, consisting of a metallic carbide composite with binder metal. The most central raw material is tungsten carbide (WC). Tungsten carbide contributes to the hardness of the material. Tantalum carbide (TaC), titanium carbide (TiC) and niobium carbide (NbC) complements WC and adjusts the properties to what is desired. These three materials are called cubic carbides. Cobalt (Co) acts as a binder and keeps the material together.

Carbide materials are often characterised by high compression strength, high hardness and therefore high wear resistance, but also by limited flexural strength and toughness. Carbide is used in taps, reamers, milling cutters, drills and thread milling cutters.

Properties	**HSS materials**	**Carbide materials**	**K10/30F (often used for solid tools)**
Hardness (HV30)	800-950	1300-1800	1600
Density (g/cm^3)	8,0-9,0	7,2-15	14,45
Compressive strength (N/mm^2)	3000-4000	3000-8000	6250
Flexural strength, (bending) (N/mm^2)	2500-4000	1000-4700	4300
Heat resistance (°C)	550	1000	900
E-module (KN/mm^2)	260-300	460-630	580
Grain size (μm)	-	0,2-10	0,8

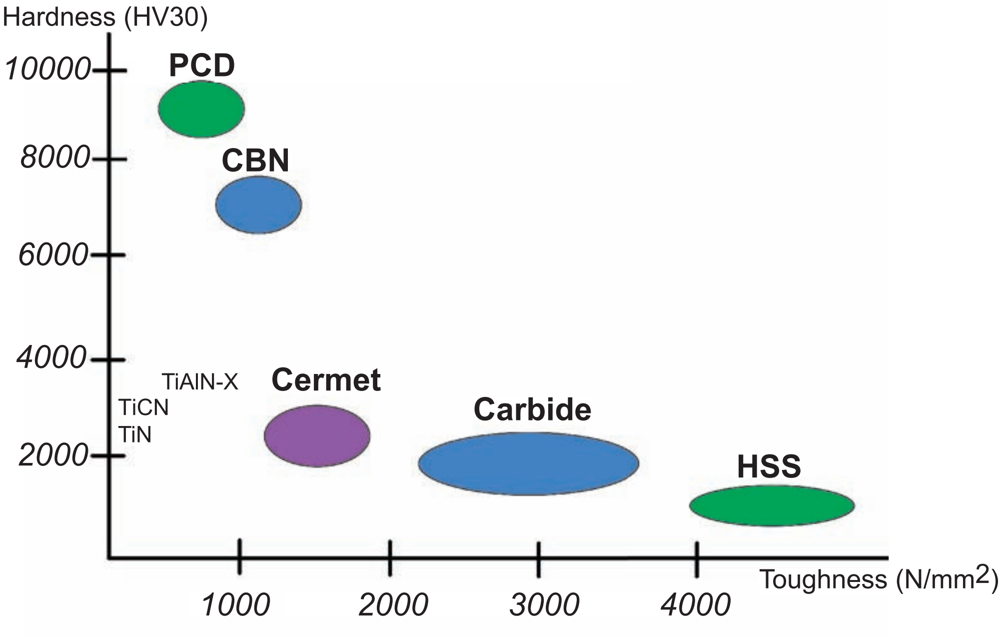
The combination of hard particle (WC) and binder metal (Co) give the following changes in characteristics.

Characteristic	**Higher WC content give**	**Higher Co content give**
Hardness	Higher hardness	Lower hardness
Compressive strength (CS)	Higher CS	Lower CS
Bending strength (BS)	Lower BS	Higher BS

Grain size also influences the material properties. Small grain sizes means higher hardness and coarse grains give more toughness.

General Information

Cutting Tool Material - Hardness In Relation To Toughness



Cermet = Ceramic Metal
CBN = Cubic Boron Nitride
PCD = Polycrystalline Diamond

Surface Treatments

Steam Tempering
Steam tempering gives a strongly adhering blue oxide surface that acts to retain cutting fluid and prevent chip to tool welding, thereby counteracting the formation of a built-up edge. Steam tempering can be applied to any bright tool but is most effective on drills and taps.

Bronze Finish
The bronze finish is a thin oxide layer formed on the tool surface and it is applied principally to Cobalt and Vanadium high speed steels.

Nitriding (FeN)
Nitriding is a process that is used to increase the hardness and wear resistance of the surface of a tool. It is particularly suitable for taps that are used on abrasive materials such as cast iron, bakelite, etc. Nitriding is used on twist drills when it is desirable to increase the strength and wear resistance of the cylindrical lands.

Hard Chromium Plating (Cr)
Hard chromium plating under specific conditions increases the surface hardness significantly, achieving values of up to 68Rc. It is especially suitable when tapping structural grade steels, carbon steels, copper, brass, etc.

Surface Coatings

Titanium Nitride Coating (TiN)
Titanium Nitride is a gold coloured ceramic coating applied by physical vapour deposition (PVD). High hardness combined with low friction properties ensures considerably longer tool life, or alternatively, better cutting performance from tools which have not been coated. TiN coating is used mainly for drills and taps.

Titanium Carbon Nitride Coating (TiCN)
Titanium Carbon Nitride is a ceramic coating applied by PVD coating technology. TiCN is harder than TiN and has a lower coefficient of friction. Its hardness and toughness in combination with good wear resistance ensures that it finds its principal application in the field of milling to enhance the performance of milling cutters.

Titanium Aluminium Nitride Coating (TiAlN)
Titanium Aluminium Nitride is a multi layer ceramic coating applied by PVD coating technology, which exhibits high toughness and oxidation stability. These properties make it ideal for higher speeds and feeds, whilst at the same time improving tool life TiAlN is suitable for drilling and tapping. It is recommended to use TiAlN when machining dry.

TiAlN - X
TiAlN – X is a Titanium Aluminium Nitride coating. The high aluminium content of the coating ensures a unique blend of high temperature resistance, hardness and toughness. This coating is ideal for milling cutters which are intended to run without coolant and when milling materials with high hardness.

Chromium Nitride Coating (CrN)
CrN is an excellent coating for aluminium alloys, copper alloys and low alloyed steel materials. CrN can also be used as an alternative on Titanium and Nickel alloys. This coating has a low tendency for built-up edges.

Super-R Coating (Ti, C, N)
SUPER-R is a coating specific to the milling operation. It exhibits low internal stress, high toughness and wear resistance, whilst demonstrating excellent oxidation resistance, thanks to the high oxidation temperature of the coating.

Super G coating (AlCrN)
Super G is an aluminium chromium nitride coating mostly used for milling cutters. The coatings hot hardness and high oxidation resistance are two unique properties. When machining applications involving heavy mechanical and thermal stresses, these properties translate into supreme wear resistance.

Zirconium Nitride Coating (ZrN)
Zirconium Nitride is a ceramic coating applied by PVD coating technology. It exhibits a combination of properties, such as high oxidation temperature and low friction coefficient, which make it attractive when tapping Aluminium and Aluminium alloys.

Dialub (Diamond-like Coating)
Dialub is an amorphous diamond coating with extremely low coefficient of friction and high hardness. This coating is specially developed for tapping in aluminium alloys with low Si-content, and drilling of stainless steels.

Super B Coating (TiAlN+WC/C)
Super B with its multi-layer coating is used for tough machining operations and gives high reliability. Its low coefficient of friction and hardness makes it ideal for tapping in tough to machine materials and materials with long chips, for example, stainless steel.

Diamond
A polycrystalline diamond coating is specially adapted to the performance demands of graphite and non ferrous materials processing. The properties of the crystalline structure dramatically improve the wear coefficient and the hardness. The coating is only used for cemented carbide tools and especially for milling cutters.

SURFACE TREATMENT / COATING PROPERTIES

Surface Treaments	Colour	Coating material	Hardness (HV)	Thick-ness (µm)	Coating structure	Frict. coeff. against steel	Max. appl. temp. (°C)
ST	Dark grey	Fe 304	400	Max. 5	Conversion into the surface	–	550
Bronze	Bronze	Fe 304	400	Max. 5	Conversion into the surface	–	550
N	Grey	FeN	1300	20	Diffusion zone	–	550
Cr	Silver	Cr	1100	Max. 5	Mono-layer	–	550
TiN	Gold	TiN	2300	1-4	Mono-layer	0,4	600
TiCN	Blue grey	TiCN	3000	1-4	Multi-layer gradient	0,4	500
TiAlN	Black grey	TiAlN	3300	3	Nano structured	0,3-0,35	900
TiAlN X	Violet grey	TiAlN	3500	1-3	Mono-layer	0,4	900
CrN	Silver grey	CrN	1750	3-4	Mono-layer	0,5	700
Super R	Copper	Ti, C, N	2900	3,5-3,7	Multi-layer	0,3-0,4	475
Super G	Blue grey	AlCrN	3200		Mono-layer	0,35	1100
Super B	Black	TiAlN+ WC/C	3000	2-6	Multi-layer lamellar	0,2	800
ZrN	Gold yellow	ZrN	2800	2-3	Mono-layer	0,2	800
Dialub	Black	a-C:H	6000	1,5-2	Mono-layer	0,1-0,2	600
Dia mond	Light grey	Polycrystal-line diamond	8000	6, 12, 20	Mono-layer	0,15-0,20	700

General Information

Workpiece Materials

Dormer classifies workpiece material in AMG (Application Material Groups) below. Tool recommendation is based on these AMGs.

APPLICATION MATERIAL GROUPS

Application Material Groups			Hardness HB	Tensile strength N/mm²
1. Steel	1.1	Magnetic soft steel	< 120	< 400
	1.2	Structural steel, case carburizing steel	< 200	< 700
	1.3	Plain Carbon steel	< 250	< 850
	1.4	Alloy steel	< 250	< 850
	1.5	Alloy steel, Hardened and tempered steel	> 250 < 350	> 850 < 1200
	1.6	Alloy steel, Hardened and tempered steel	> 350	> 1200 < 1620
	1.7	Alloy steel, Heat treated	49-55 HRc	> 1620
	1.8	Alloy steel, Hardened & Wear resistant steel	55-63 HRc	> 1980
2. Stainless Steel	2.1	Free machining, Stainless Steel	< 250	< 850
	2.2	Austenitic	< 250	< 850
	2.3	Ferritic + Austenitic, Ferritic, Martensitic	< 300	< 1000
3. Cast Iron	3.1	Lamellar graphite	> 150	> 500
	3.2	Lamellar graphite	> 150 ≤ 300	> 500 < 1000
	3.3	Nodular graphite, Malleable Cast Iron	< 200	< 700
	3.4	Nodular graphite, Malleable Cast Iron	> 200 <300	> 700 < 1000
4. Titanium	4.1	Titanium, unalloyed	< 200	< 700
	4.2	Titanium, alloyed	< 270	< 900
	4.3	Titanium, alloyed	> 270 <350	> 900 ≤ 1250
5. Nickel	5.1	Nickel, unalloyed	< 150	< 500
	5.2	Nickel, alloyed	> 270	> 900
	5.3	Nickel, alloyed	> 270 <350	> 900 < 1200
6. Copper	6.1	Copper	< 100	< 350
	6.2	β-Brass, Bronze	< 200	< 700
	6.3	α-Brass	< 200	< 700
	6.4	High Strength Bronze	< 470	< 1500
7. Aluminium Magnesium	7.1	Al, Mg, unalloyed	< 100	< 350
	7.2	Al alloyed, Si < 0.5%	<150	< 500
	7.3	Al alloyed, Si > 0.5% < 10%	<120	< 400
	7.4	Al alloyed, Si > 10% Whisker reinforced Al-alloys Mg-alloys	<120	< 400
8. Synthetic materials	8.1	Thermoplastics		
	8.2	Thermosetting plastics		
	8.3	Reinforced plastic materials	-	-
9. Hard material	9,1	Cermets (metals-ceramics)	< 550	< 1700
10. Graphite	10.1	Graphite		

EXAMPLES OF WORKPIECE MATERIALS FROM DIFFERENT STANDARDS

A full list of materials and comparisons between different standards can be found in Dormer Product Selector, available on CD or www.dormertools.com.

AMG	EN	W no.	DIN
1.1		1.1015, 1.1013	Rfe60, Rfe100
1.2	EN 10 025 – S235JRG2	1.1012, 1.1053, 1.7131	St37-2, 16MnCr5, St50-2
1.3	EN 10 025 – E295	1.1191, 1.0601	CK45, C60
1.4	EN 10 083-1 – 42 CrMo 4 EN 10 270-2	1.7225, 1.3505 1.6582, 1.3247	42CrMo4, 100Cr6 34CrNiMo6, S2-10-1-8
1.5	EN ISO 4957 – HS6-5-2 EN-ISO 4957 – HS6-5-2-5	1.2510, 1.2713 1.3247, 1.2080	100MnCrW12, 55NiCrMoV6 X210Cr12, S2-10-1-8
1.6	EN-ISO 4957 – HS2-9-1-8	1.2510, 1.2713 1.3247, 1.2080	100MnCrW12 X210Cr12, S2-10-1-8
1.7	EN-ISO 4957 – HS2-9-1-8	1.2510	100MnCrW4
1.8	EN-ISO 4957 – X40CrMoV5-1	1.3343, 1.2344	S6-5-2, GX40CrMoV5-1
2.1	EN 10 088-3 – X14CrMoS17	1.4305, 1.4104	X10CrNiS189, X12CrMoS17
2.2.	EN 10 088-2,0 -3 – 1.4301+AT	1.4301, 1.4541 1.4571	X5CrNi189 X10CrNiMoTi1810
2.3	EN 10 088-3 – 1.4460	1.4460, 1.4512 1.4582	XBCrNiMo275, X4CrNiMoN6257
3.1	EN 1561 – EN-JL1030	0.6010, 0.6040	GG10, GG40
3.2	EN 1561 – EN-JL1050	0.6025, 0.6040	GG25, GG40
3.3	EN 1561 – EN-JL2040	0.7040, 0.7070 0.8145, 0.8045	GGG40, GGG70 GTS45-06, GTW45-07
3.4	EN 1561 – EN-JL2050	0.7040, 0.7070 0.8145, 0.8045	GGG40, GGG70 GTS45-06, GTW45-07
4.1		3.7024LN	Ti99,8
4.2		3.7164LN, 3.7119LN	TiAl6V4, TiAl55n2
4.3		3.7164LN 3.7174LN, 3.7184LN	TiAl6V4, TiAl6V5Sn2 TiAl4MoSn2
5.1		2.4060, 2.4066	Nickel 200, 270, Ni99,6
5.2		2.4630LN, 2.4602 2.4650LN	Nimonic 75, Monel 400 Hastelloy C, Inconel 600
5.3		2.4668LN, 2.4631LN 2.6554LN	Inconel 718 Nimonic 80A, Waspaloy
6.1	EN 1652 – CW004A	2.0060, 2.0070	E-Cu57, SE-Cu
6.2	EN 1652 – CW612N	2.0380, 2.0360 2.1030, 2.1080	CuZn39Pb2, CuZn40 CuSn8, CuSn6Zn
6.3	EN 1652 – CW508L	2.0321, 2.0260	CuZn37, CuZn28
6.4			Ampco 18, Ampco 25
7.1	EN 485-2 – EN AW-1070A	3.0255	Al99,5
7.2	EN 755-2 – EN AW-5005	3.1355, 3.3525	AlCuMg2, AlMg2Mn0,8
7.3	EN 1706 – EN AC-42000	3.2162.05, 3.2341.01	GD-AlSi8Cu, G-AlSi5Mg
7.4	SS-EN 1706 – EN AC-47000	3.2581.01	G-AlSi18, G-AlSi12
8.1			
8.2			
8.3			
9,1			
10.1			

General Information

	BS	SS	USA	UNS
1.1	230Mo7, 050A12	1160	Leaded Steels	G12120
1.2	060A35, 080M40, 4360-50B	1312, 1412, 1914	135, 30	G10100
1.3	080M46, 080A62	1550, 2142, 2172	1024, 1060, 1061	G10600
1.4	708M40/42, 817M40 534A99, BM2, BT42	1672-04, 2090 2244-02, 2541-02	4140, A2, 4340 M42, M2	G41270, G41470 T30102, T11342
1.5	B01, BM2, BT42 826 M40, 830M31	2244-04, 2541-03 2550, 2722, 2723	01, L6, M42, D3, A2 M2, 4140, 8630	G86300, T30102 T11302, T30403 T11342
1.6	801 826 M40, 830M31	2244-05, 2541-05 HARDOX 400	01, L6, M42, D3 4140, 8130	T30403, G41400 J14047
1.7	BO1, BD3, BH13	HARDOX 500		
1.8	BM2, BH13	2242 HARDOX 600		
2.1	303 S21 416 S37	2301, 2312, 2314 2346, 2380	303, 416 430F	S30300, S41600 S43020
2.2.	304 S15, 321 S17 316 S, 320 S12	2310, 2333, 2337 2343, 2353, 2377	304, 321, 316	S30400, S32100 S31600
2.3	317 S16, 316 S16	2324, 2387, 2570	409, 430, 436	S40900, S4300, S43600
3.1	Grade150, Grade 400	0120, 0212, 0814	ASTM A48 class 20	F11401, F12801
3.2	Grade200, Grade 400	0125, 0130, 0140, 0217	ASTM A48 class 40 ASTM A48 class 60	F12801, F14101
3.3	420/12, P440/7 700/2, 30g/72	0219, 0717, 0727 0732, 0852	ASTM A220 grade 40010 ASTM A602 grade M4504	F22830 F20001
3.4	420/12, P440/7 700/2, 30g/72	0221, 0223 0737, 0854	ASTM A220 grade 90001 ASTM A602 grade M8501	F26230 F20005
4.1	TA1 to 9	Ti99,8	ASTM B265 grade 1	R50250
4.2	TA10 to 14, TA17	TiAl6V4, TiAl5Sn2	AMS4928	R54790
4.3	TA10 to 13, TA28	TiAl6V5Sn2	AMS4928, AMS4971	R56400, R54790
5.1	NA 11, NA12	Ni200, Ni270	Nickel 200, Nickel 230	N02200, N02230
5.2	HR203 3027-76		Nimonic 75,Monel400 Hastelloy, Inconel600	N06075, N10002 N04400, N06600
5.3	HR8 HR401, 601		Inconel 718, 625 Nimonic 80	N07718, N07080 N06625
6.1	C101	5010	101	C10100, C1020
6.2	CZ120, CZ109,PB104	5168		C28000, C37710
6.3	CZ108,CZ106	5150		C2600, C27200
6.4	AB1 type	5238, JM7-20		
7.1	LMO, 1 B (1050A)	4005	EC, 1060, 1100	A91060, A91100
7.2	LM5, 10, 12, N4 (5251)	4106, 4212	380, 520.0, 520.2, 2024, 6061	A03800, A05200, A92024
7.3	LM2,4,16,18,21,22, 24,25,26,27,L109	4244	319.0, 333.0 319.1, 356.0	A03190, A03330 C35600
7.4	LM6, 12,13, 20, 28, 29, 30	4260, 4261, 4262	4032, 222.1, A332.0	A94032, A02220, A13320
8.1	Polystyrene, Nylon, PVC Cellulose, Acetate & Nitrate		Polystyrene, Nylon PVC	
8.2	Ebonite, Tufnol, Bakelite		Bakelite	
8.3	Kevlar Printed Circuit boards		Kevlar	
9,1	Ferrotic Ferrotitanit			
10.1				

Machining Of Steel

ALLOY ELEMENTS

Steel materials can roughly be divided into carbon steels and alloyed steels.

Carbon steels or unalloyed steels are materials with carbon as the main alloying element. Carbon steels seldom have a carbon content above 1.3 %.

Alloyed steels are materials with essential alloying elements besides carbon and iron. The total content of alloying elements can vary for different reasons such as strength, wear resistance and heat treatment ability.

When classifying carbon steels and alloyed steels, the border between the two is not well-defined.

PRACTICAL USE

Steel materials can also be classified dependent upon their use. This classification is often made between construction steel and tool steel.

Construction steels are materials used for supporting constructions. These steels are often used in the same condition as they are delivered from the steel plant. Tensile strength for example is an important factor for this group. Construction steels are seldom heat-treated.

Tool steels are used for tool applications like cutting tools, knives and forming tools. Important factors for these materials are wear resistance, hardness and sometimes toughness. In many cases tool steels are hardened to various degrees depending on their application.

Also when classifying construction and tool steels, the border between the two is not well-defined.

IMPORTANT WHEN MACHINING STEEL MATERIALS

- The group of steel materials is extensive, which makes it important to find out the properties of the material to be machined. Use the Dormer Product Selector to find the correct AMG classification which in turn will help you to find the correct tool for the application.
- In general a non-alloyed or low-alloyed material is soft and sticky. Use sharp tools with positive geometries.
- A high alloyed steel can be abrasive or hard. To reduce rapid wear on the cutting surface, use coated tools and carbide tools.
- As mentioned above tool steels can be hardened to various degrees. It is important to be aware of both material grade and hardness in order to select the correct tool configuration for the application.

Machining Of Stainless Steels

Stainless steels are alloyed steels with a Chrome content normally above 12%. Resistance against corrosion generally increases with the Cr-content. Other alloying elements like Nickel and Molybdenum change the structure and mechanical properties of the steel.

Stainless steels can be divided into the following groups
Ferritic stainless steels – often have good strength. Good machinability.
Martensitic stainless steel – relatively good machinability
Austenitic stainless steel – characterised by high coefficient of elongation. Machinability medium to low.
Austenitic-ferritic stainless steel – often called duplex stainless steel. These steels have low machinability.

WHY ARE STAINLESS STEEL SEEN AS DIFFICULT TO MACHINE?

- Most stainless steel materials work harden during deformation, i.e the process of producing a chip. The work hardening decreases rapidly with an increasing distance from the surface. Hardness values close to the machined surface can increase by up to 100% of the original hardness value if using the incorrect tool.

- Stainless steels are poor heat conductors, which leads to high cutting edge temperatures compared to a steel, in for example, AMG 1.3 with similar levels of hardness.

- High toughness leads to high torque, which in turn results in a high work load for a tap or drill. When combined with the effects of work hardening and poor heat conductivity, the cutting tool has to perform in a relatively hostile environment.

- The materials have a tendency to smear the surface of the cutting tool.

- Chip breaking and swarf management problems, due to the high toughness of the stainless steel.

IMPORTANT WHEN MACHINING STAINLESS STEELS

- For drilling operations, use ADX or CDX drills with internal coolant capability. This will counter the work hardening that occurs when machining Stainless Steel. With internal cooling, the work hardening is kept to a minimum, about 10%.

- High feeds rates transfer more heat away from the machining area. This is a very important consideration for a troublefree machining operation.

- When it comes to choosing the correct cutting speed, always start in the lower region of Dormer recommendations.This is due to the fact that different material batches may require different cutting speeds. Also keep in mind that for deeper holes, cutting speed should be reduced by 10-20%, for the chosen application.

- When threading in DUPLEX or in high alloyed stainless steel, keep the cutting speed in the lower region of Dormer recommendations.

- Use preferably a neat cutting oil. If an emulsion is the only option for the operation, a minimum 8% concentration is recommended.

- First choice should always be a coated tool since they have a greater tendency to resist built-up edges.

- Avoid using tools with worn cutting edges, since this will increase work hardening.

Machining Of Cast Iron

Cast iron consists of three basic structural constituents:

Ferritic – Easy-to-machine, low strength and a hardness below 150 HBN. At low cutting speed the cast iron can be "sticky" and result in built-up edges.
Ferritic/pearlitic – Vary from low strength and low hardness of 150 HBN to high strength and a hardness of 290 HBN.
Pearlitic – Its strength and hardness is dependent on the roughness of its lamellar. With fine lamellar the cast iron is very hard and has high strength, causing it to smear and build up edges on the tool.

ALLOY ELEMENTS

Cast iron is an iron-carbon alloy with a carbon content of mostly 2-4% as well as other elements like silicon (Si), manganese (Mn), phosphorous (P) and sulphur (S). Depending mainly on the form in which carbon occurs, cast iron is divided into four main types: grey cast iron, nodular cast iron, malleable cast iron and alloy cast iron.

Using for example nickel, copper, molybdenum and chromium can effect the heat and corrosion resistance, rigidity and strength of the cast iron. The alloying elements can be divided into two groups: carbide forming and graphite-elements. The alloys greatly effect the machinability of the cast iron.

PRACTICAL USE

Cast iron components are used in a wide variety of applications like engine blocks, pumps and valves. The reason for using cast iron is a combination of complex shapes and the need for strength.

IMPORTANT WHEN MACHINING CAST IRON

- Most cast iron materials are easy to machine because of the short chipping properties. The reason is that graphite makes chip breaking easier and can improve lubrication.
- Tools with low rake angles are generally used in cast iron.
- Most materials are abrasive, so coatings improve tool life.
- Dry machining can be done in most applications.
- The most significant difficulties are due to irregular shapes of casting, the presence of hard skins and sand inclusions.

Machining Of aluminium alloys

Aluminium alloys offer many advantages when machining: high cutting speed, low cutting forces, minimal wear on the tools and relatively low machining temperature. When machining aluminium alloys, it is always best to utilise tools with geometric configurations specifically designed for these materials. Although the use of general purpose tooling can be satisfactory, it is difficult to obtain an acceptable surface finish and to avoid aluminium built-up edges on the tools.

ALLOY ELEMENTS

Most aluminium is in the form of an alloy and by using different types of alloys the aluminium can be produced to contain a wide range of characteristics, for example, tensile strengths, hardness and plastic malleability. The most common alloys are silicon (Si) magnesium (Mg), manganese (Mn), copper (Cu) and zinc (Zn). Alloys containing a maximum 1% iron and silicon in total are called pure or unalloyed aluminium. Aluminium alloys are usually divided into wrought and cast alloys. They are further divided into the groups heat-treated and non-heat treated and work hardenable.

Cast alloys may be both heat and non-heat treatable as well as die cast and sand cast. The most common cast alloy is aluminium-silicon with 7-12% silicon. The type of alloy chosen depends on the product requirements and the proposed cast method.

The wrought alloys are mostly heat and non-heat treatable. Age and work hardening with solution and precipitation treatments are methods used widely to improve properties to stronger, harder materials.

PRACTICAL USE

Aluminium is the second most used metal. The reason for this is the combination of attractive characteristics like low density, high conductivity, high strength and ease of recycling.

Aluminium is used almost everywhere:

- Transportation equipment: vehicles, trucks, buses and trains where aluminium gives the opportunity to reduce weight. Product examples are engine blocks, pistons and radiators.
- Mechanical industry: in a wide range of constructions and often in specially manufactured aluminium profiles.
- Aluminium alloys are also heavily used in the electro-mechanical industry, building industry and packing industry.

IMPORTANT WHEN MACHINING ALUMINIUM ALLOYS

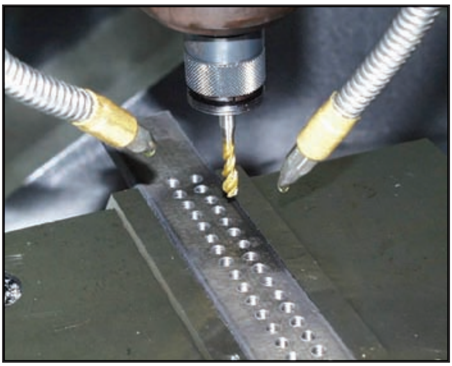
- Sharp edges and positive geometries are important for machining aluminium alloys with low Si-content.
- Correct cutting speed and feed are important to get rid of built-up edges and to improve chip breaking.
- For more abrasive aluminium alloys with higher Si-content, above 6%, coated tools are recommended.
- The use of lubrication is also important when machining aluminium alloys.

LUBRICANTS

Lubricants or coolants are used on cutting tools to reduce friction or to reduce heat.

Type of Lubricant	Description	Advantages	Disadvantages
Emulsion	Emulsions or water-soluble cutting oils give lubrication properties combined with good cooling property. The oil concentrate in emulsion contains additives that give different properties like lubricators, preservatives and EP additives to improve bearing strength.	Reduces heat. Flushes away chips.	Disposal cost. Environment.
Minimal lubrication	Minimal lubrication is a small amount of oil distributed with compressed air to lubricate the cutting or forming process.	Low cost. Good lubricator.	Bad chip removal. Requires good set up of nozzle positioning
Oil	Cutting oils have good lubrication properties but do not provide such good cooling as water-based cutting fluids.	Good lubricator.	High cost. Environment.
Dry / compressed air	Compressed air directed to the cutting process.	Clean process. Remove chips. Low cost.	Works in a limited no. of applications.

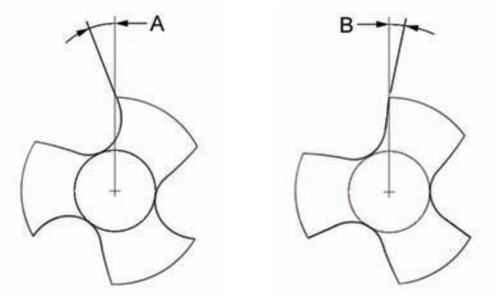

Emulsion


Minimal lubrication

General Information

Lubricant	Tools	Sub-group	AMG-Groups 1.1-1.4	1.5-1.8	2	3	4	5	6	7	8	9	10
Emulsion	Milling cutters	HM	•	•	•	•	•	•	•	•	•		
		HSS Slotting, Roughing, Finishing	•	•	•	•	•	•	•	•	•	•	
		HSS Finishing (coated only)	•	•	•	•	•	•	•	•	•	•	
	Thread milling cutters	HM	•	•	•	•	•	•	•	•	•		
	Drills	HM	•	•	•	•	•	•	•	•	•	•	
		HSS	•	•	•	•	•	•	•	•	•	•	
	Taps	HM Coated	•	•	•	•	•	•	•	•	•		
		HSS Bright	•	•	•	•			•	•	•		
		HSS Coated	•	•	•	•			•	•	•		
Minimum lubrication	Milling cutters	HM	•	•	•	•	•	•	•	•	•		
		HSS Slotting, Roughing, Finishing	•	•	•	•		•	•	•	•	•	
		HSS Finishing (coated only)	•	•	•	•		•	•	•	•	•	
	Thread milling cutters	HM	•	•	•	•		•	•	•	•		
	Drills	HM	•	•	•	•		•	•	•	•		
		HSS											
	Taps	HM Coated	•	•	•	•	•	•	•	•	•		
		HSS Blank							•	•	•		
		HSS Coated	•		•	•			•	•	•		
Oil	Taps	HM Coated											
		HSS Bright	•	•	•		•	•				•	
		HSS Coated	•	•	•		•	•				•	
Dry / comp. air	Milling cutters	HM	•	•		•			•		•		•
		HSS Slotting, Roughing, Finishing											
		HSS Finishing (coated only)	•			•			•				
	Thread milling cutters	HM	•	•		•			•	•	•		
	Drills	HM Coated				•							
	Taps	HSS											
		HM Coated	•	•		•			•		•		
		HSS Bright											
		HSS Coated				•					•		

General Geometry



A. Positive rake angle
B. Negative rake angle

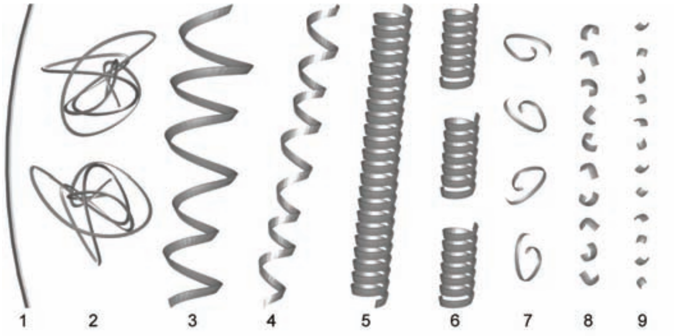
Rake Angle	Benefits / range of application	Disadvantages
Low or negative (-5° – 5°)	Strong geometry, strong edge. Works well in cast iron and hardened steel.	Doesn't work in soft or tensile material. High cutting forces.
Medium (8° – 14°)	Cuts well. Works well in most materials for example steel and stainless steel	
High (20° – 30°)	Low cutting force. Works best in aluminium and other soft materials.	Chipping often occurs because of the sharp edge.

Types Of Chips

Chip formation is mostly caused by plastic deformation. This process, due to the friction generated during machining, generates heat. Heat has the positive effect of increasing the plasticity of the workpiece material, but the negative effect of increasing the wear on the tool. When workpiece material reaches its breakage point, then the chip is generated. Its form and development depend on different factors, such as:

- Chemical-physical compatibility between tool and workpiece materials
- Cutting operation
- Cutting conditions (speed, feed, material removal rate)
- Tool geometry
- Friction coefficient (with or without coating)
- Lubrication

Depending on different combinations of the above mentioned factors, the chips can turn out in many different ways (see figure below).

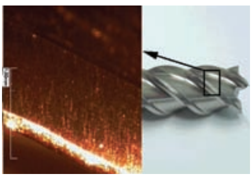
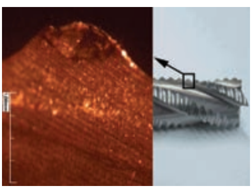


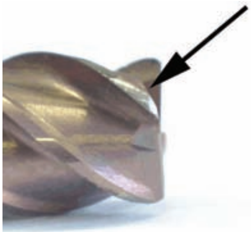
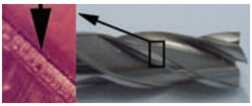
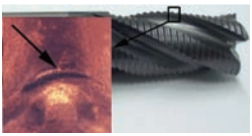
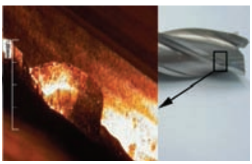
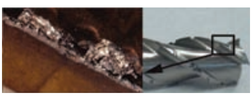
1 Ribbon chips
2-3 Snarled chips
4-6 Washer-type chips
7 Arc chips
8-9 Element chips

Types Of Wear

Wear is generated by mechanical abrasion, adhesion, chemical diffusion and oxidation. The most important factors to influence the different types of wear are the mechanical and chemical properties of the materials in contact, the working conditions, but mainly cutting speed and temperature. At low speeds, abrasion and adhesion wear are most important, whereas at high speeds, it is diffusion and plastic deformation. It is not easy to set up a mechanical model to foresee wear development on cutting tools.

Types of wear can be briefly summarised in nine different types (see table below).

TYPE	ORIGIN	CONSEQUENCE	REMEDY
Flank Wear 	Cutting speed too high.	High surface roughness, inconsistent tolerance, high friction.	Reduce cutting speed. Use a coated tool. Use more wear resistant tool material.
Crater Wear 	Generated by chemical diffusion due to high temperature on the cutting edge.	Cutting edge weakness, high surface roughness.	Choose a tool with positive geometry. Reduce cutting speed and then feed. Use coated tool.

TYPE	ORIGIN	CONSEQUENCE	REMEDY
Plastic Deformation 	High temperature and high pressure.	Bad chip control, high surface roughness, high flank wear.	Use a tool with bigger cross section. Reduce cutting speed and then feed.
Notch Wear 	Oxidation, friction.	High surface roughness, cutting edge breakage.	Reduce cutting speed. Use coated tool.
Thermal Micro-flaws 	Due to thermal variation, caused by interrupted cutting or low cooling.	Cracks through cutting edge, high surface roughness.	Increase cooling flow. Use cutting tool with high tensile strength.
Cracks 	Due to mechanical fatigue.	Tool breakage.	Reduce feed rate, Improve tool holder stability.
Chipping 	Due to weak tool geometry or built up edge.	High roughness, flank wear.	Choose a tool with stronger and positive geometry. Increase cutting speed to reduce BUE. Reduce feed rate at first removal pass. Improve machine stability.
Tool Breakage 	Too high load.	Tool breakage, work-piece breakage.	Reduce feed and/or speed. Choose a tool with stronger geometry. Improve machine stability.
Built up edge (BUE) 	Negative geometry. Low cutting speed. Work-piece material with welding tendency (as stainless steel or aluminium).	Workpiece material slides on tool edge and welds on it. High surface roughness, chipping.	Increase cutting speed. Choose a tool with positive geometry. Increase lubrication amount.

DORMER

General Information

Hardness and Tensile Strength

HV Vickers Hardness No.	HRC Rockwell C. Scale Hardness No.	HB Brinell Hardness No.	Tensile Strength Newton per sq. mm	Tensile Strength Tons per sq. in.
940	68			
900	67			
864	66			
829	65			
800	64			
773	63			
745	62			
720	61			
698	60			
675	59			
655	58		2200	142
650		618	2180	141
640		608	2145	139
639	57	607	2140	138
630		599	2105	136
620		589	2070	134
615	56	584	2050	133
610		580	2030	131
600		570	1995	129
596	55	567	1980	128
590		561	1955	126
580		551	1920	124
578	54	549	1910	124
570		542	1880	122
560	53	532	1845	119
550		523	1810	117
544	52	517	1790	116
540		513	1775	115
530		504	1740	113
527	51	501	1730	112
520		494	1700	110
514	50	488	1680	109
510		485	1665	108
500		475	1630	105
497	49	472	1620	105
490		466	1595	103
484	48	460	1570	102
480		456	1555	101
473	47	449	1530	99
470		447	1520	98
460		437	1485	96
458	46	435	1480	96
450		428	1455	94
446	45	424	1440	93
440		418	1420	92

HV Vickers Hardness No.	HRC Rockwell C. Scale Hardness No.	HB Brinell Hardness No.	Tensile Strength Newton per sq. mm	Tensile Strength Tons per sq. in.
434	44	413	1400	91
423	43	402	1360	88
413	42	393	1330	86
403	41	383	1300	84
392	40	372	1260	82
382	39	363	1230	80
373	38	354	1200	78
364	37	346	1170	76
355	36	337	1140	74
350		333	1125	73
345	35	328	1110	72
340		323	1095	71
336	34	319	1080	70
330		314	1060	69
327	33	311	1050	68
320		304	1030	67
317	32	301	1020	66
310	31	295	995	64
302	30	287	970	63
300		285	965	62
295		280	950	61
293	29	278	940	61
290		276	930	60
287	28	273	920	60
285		271	915	59
280	27	266	900	58
275		261	880	57
272	26	258	870	56
270		257	865	56
268	25	255	860	56
265		252	850	55
260	24	247	835	54
255	23	242	820	53
250	22	238	800	52
245		233	785	51
243	21	231	780	50
240		228	770	50
235		223	755	49
230		219	740	48
225		214	720	47
220		209	705	46
215		204	690	45
210		199	675	44
205		195	660	43
200		190	640	41

Useful Tolerances

Tolerance values in µm   1 µm = 0.001 mm / 0.000039 inches

Toler-ance	Diameter (mm)							
	> 1 ≤ 3	> 3 ≤ 6	> 6 ≤ 10	> 10 ≤ 18	> 18 ≤ 30	> 30 ≤ 50	> 50 ≤ 80	> 80 ≤ 120
	Diameter (inch)							
	> 0.039 ≤ 0.118	> 0.118 ≤ 0.236	> 0.236 ≤ 0.394	> 0.394 ≤ 0.709	> 0.709 ≤ 1.181	> 1.181 ≤1.968	>1.968 ≤ 3.149	> 3.149 ≤ 4.724
	Tolerance values (µm)							
e8	-14 / -28	-20 / -38	-25 / -47	-32 / -59	-40 / -73	-50 / -89	-60 / -106	-72 / -126
f6	-6 / -12	-10 / -18	-13 / -22	-16 / -27	-20 / -33	-25 / -41	-30 / -49	-36 / -58
f7	-6 / -16	-10 / -22	-13 / -28	-16 / -34	-20 / -41	-25 / -50	-30 / -60	-36 / -71
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
h9	0 / -25	0 / -30	0 / -36	0 / -43	0 / -52	0 / -62	0 / -74	0 / -87
h10	0 / -40	0 / -48	0 / -58	0 / -70	0 / -84	0 / -100	0 / -120	0 / -140
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
k10	+40 / 0	+48 / 0	+58 / 0	+70 / 0	+84 / 0	+100 / 0	+120 / 0	+140 / 0
k12	+100 / 0	+120 / 0	+150 / 0	+180 / 0	+210 / 0	+250 / 0	+300 / 0	+350 / 0
m7	+2 / +12	+4 / +16	+6 / +21	+7 / +25	+8 / +29	+9 / +34	+11 / +41	+13 / +48
js14	+/- 125	+/- 150	+/- 180	+/- 215	+/- 260	+/- 310	+/- 370	+/- 435
js16	+/- 300	+/- 375	+/- 450	+/- 550	+/- 650	+/- 800	+/- 950	+/- 1100
H7	+10 / 0	+12 / 0	+15 / 0	+18 / 0	+21 / 0	+25 / 0	+30 / 0	+35 / 0
H8	+14 / 0	+18 / 0	+22 / 0	+27 / 0	+ 33 / 0	+39 / 0	+46 / 0	+54 / 0
H9	+25 / 0	+30 / 0	+36 / 0	+43 / 0	+52 / 0	+62 / 0	+74 / 0	+87 / 0
H12	+100 / 0	+120 / 0	+150 / 0	+180 / 0	+210 / 0	+250 / 0	+300 / 0	+350 / 0
P9	-6 / -31	-12 / -42	-15 / -51	-18 / -61	-22 / -74	-26 / -86	-32 / -106	-37 / -124
S7	-13 / -22	-15 / -27	-17 / -32	-21 / -39	-27 / -48	-34 / -59	-42 / -72	-58 / -93

General Information

Decimal Equivalents

MM	Fract.	Gauge	Inch	MM	Fract.	Gauge	Inch	MM	Fract.	Gauge	Inch	MM	Fract.	Gauge	Inch
.3			.0118	1.55			.0610	3.797		25	.1495	6.909		I	.2720
.32			.0126	1.588	1/16		.0625	3.8			.1496	7.0			.2756
.343		80	.0135	1.6			.0630	3.861		24	.1520	7.036		J	.2770
.35			.0138	1.613		52	.0635	3.9			.1535	7.1			.2795
.368		79	.0145	1.65			.0650	3.912		23	.1540	7.137		K	.2810
.38			.0150	1.7			.0669	3.969	5/32		.1562	7.144	9/32		.2812
.397	2/64		.0156	1.702		51	.0670	3.988		22	.1570	7.2			.2835
.4			.0157	1.75			0.689	4.0			.1575	7.3			.2874
.406		78	.0160	1.778		50	.0700	4.039		21	.1590	7.366		L	.2900
.42			.0165	1.8			.0709	4.089		20	.1610	7.4			.2913
.45			.0177	1.85			.0728	4.1			.1614	7.493		M	.2950
.457		77	.0180	1.854		49	.0730	4.2			.1654	7.5			.2953
.48			.0189	1.9			.0748	4.216		19	.1660	7.541	19/64		.2969
.5			.0197	1.93		48	.0760	4.3			.1693	7.6			.2992
.508		76	.0200	1.95			.0768	4.305		18	.1695	7.671		N	.3020
.52			.0205	1.984	5/64		.0781	4.366	11/64		.1719	7.7			.3031
.533		75	.0210	1.994		47	.0785	4.394		17	.1730	7.8			.3071
.55			.0217	2.0			.0787	4.4			.1732	7.9			.3110
.572		74	.0225	2.05			.0807	4.496		16	.1770	7.938	5/16		.3125
.58			.0228	2.057		46	.0810	4.5			.1772	8.0			.3150
.6			.0236	2.083		45	.0820	4.572		15	.1800	8.026		O	.3160
.61		73	.0240	2.1			.0827	4.6			.1811	8.1			.3189
.62			.0244	2.15			.0846	4.623		14	.1820	8.2			.3228
.635		72	.0250	2.184		44	.0860	4.7		13	.1850	8.204		P	.3230
.65			.0256	2.2			.0866	4.762	3/16		.1875	8.3			.3268
.66		71	.0260	2.25			.0886	4.8		12	.1890	8.334	21/64		.3281
.68			.0268	2.261		43	.0890	4.851		11	.1910	8.4			.3307
.7			.0276	2.3			.0906	4.9			.1929	8.433		Q	.3320
.711		70	.0280	2.35			.0925	4.915		10	.1935	8.5			.3346
.72			.0283	2.375		42	.0935	4.978		9	.1960	8.6			.3386
.742		69	.0292	2.381	3/32		.0938	5.0			.1969	8.611		R	.3390
.75			.0295	2.4			.0945	5.055		8	.1990	8.7			.3425
.78			.0307	2.438		41	.0960	5.1			.2008	8.731	11/32		.3438
.787		68	.0310	2.45			.0965	5.105		7	.2010	8.8			.3465
.794	1/32		.0312	2.489		40	.0980	5.159	13/64		.2031	8.839		S	.3480
.8			.0315	2.5			.0984	5.182		6	.2040	8.9			.3504
.813		67	.0320	2.527		39	.0995	5.2			.2047	9.0			.3543
.82			.0323	2.55			.1004	5.22		5	.2055	9.093		T	.3580
.838		66	.0330	2.578		38	.1015	5.3			.2087	9.1			.3583
.85			.0335	2.6			.1024	5.309		4	.2090	9.128	23/64		.3594
.88			.0346	2.642		37	.1040	5.4			.2126	9.2			.3622
.889		65	.0350	2.65			.1043	5.41		3	.2130	9.3			.3661
.9			.0354	2.7			.1063	5.5			.2165	9.347		U	.3680
.914		64	.0360	2.705		36	.1065	5.556	7/32		.2188	9.4			.3701
.92			.0362	2.75			.1083	5.6			.2205	9.5			.3740
.94		63	.0370	2.778	7/64		.1094	5.613		2	.2210	9.525	3/8		.3750
.95			.0374	2.794		35	.1100	5.7			.2244	9.576		V	.3770
.965		62	.0380	2.8			.1102	5.791		1	.2280	9.6			.3780
.98			.0386	2.819		34	.1110	5.8			.2283	9.7			.3819
.991		61	.0390	2.85			.1122	5.9			.2323	9.8			.3858
1.0			.0394	2.87		33	.1130	5.944		A	.2340	9.804		W	.3860
1.016		60	.0400	2.9			.1142	5.953	15/64		.2344	9.9			.3898
1.041		59	.0410	2.946		32	.1160	6.0			.2362	9.922	25/64		.3906
1.05			.0413	2.95			.1161	6.045		B	.2380	10.0			.3937
1.067		58	.0420	3.0			.1181	6.1			.2402	10.084		X	.3970
1.092		57	.0430	3.048		31	.1200	6147		C	.2420	10.1			.3976
1.1			.0433	3.1			.1220	6.2			.2441	10.2			.4016
1.15			.0453	3.175	1/8		.1250	6.248		D	.2460	10.262		Y	.4040
1.181		56	.0465	3.2			.1260	6.3			.2480	10.3			.4055
1.191	3/64		.0469	3.264		30	.1285	6.35	1/4	E	.2500	10.319	13/32		.4063
1.2			.0472	3.3			.1299	6.4			.2520	10.4			.4094
1.25			.0492	3.4			.1339	6.5			.2559	10.49		Z	.4130
1.3			.0512	3.454		29	.1360	6.528		F	.2570	10.5			.4134
1.321		55	.0520	3.5			.1378	6.6			.2598	10.6			.4173
1.35			.0531	3.569		28	.1405	6.629		G	.2610	10.7			.4213
1.397		54	.0550	3.572	9/64		.1406	6.7			.2638	10.716	27/64		.4219
1.4			.0551	3.6			.1417	6.747	17/64		.2656	10.8			.4252
1.45			.0571	3.658		27	.1440	6.756		H	.2660	10.9			.4291
1.5			.0591	3.7			.1457	6.8			.2677	11.0			.4331
1.511		53	.0595	3.734		26	.1470	6.9			.2717				

DORMER

Decimal Equivalents

MM	FRACT.	INCH	MM	FRACT.	INCH	MM	FRACT.	INCH	MM	FRACT.	INCH	MM	FRACT.	INCH
11.11		.4370	19.05	3/4	.7500	29.75		1.1713	44.053	1 47/64	1.7344	68.00		2.6772
11.112	7/16	.4375	19.25		.7579	29.766	1 11/64	1.1719	44.45	1 3/4	1.7500	68.262	2 11/16	2.6875
11.2		.4409	19.447	49/64	.7656	30.0		1.1811	44.5		1.7520	69.0		2.7165
11.3		.4449	19.5		.7677	30.162	1 3/16	1.1875	44.847	1 49/64	1.7656	69.056	2 23/32	2.7188
11.4		.4488	19.75		.7776	30.25		1.1909	45.0		1.7717	69.85	2 3/4	2.7500
11.5		.4528	19.844	25/32	.7812	30.5		1.2008	45.244	1 25/32	1.7812	70.0		2.7559
11.509	29/64	.4531	20.0		.7874	30.559	1 13/64	1.2031	45.5		1.7913	70.644	2 25/32	2.7812
11.6		.4567	20.241	51/64	.7969	30.75		1.2106	45.641	1 51/64	1.7969	71.0		2.7953
11.7		.4606	20.25		.7972	30.956	1 7/32	1.2188	46.0		1.8110	71.438	2 13/16	2.8125
11.8		.4646	20.5		.8071	31.0		1.2205	46.038	1 13/16	1.8125	72.0		2.8346
11.9		.4685	20.638	13/16	.8125	31.25		1.2303	46.434	1 53/64	1.8281	72.231	2 27/32	2.8438
11.906	15/32	.4688	20.75		.8169	31.353	1 15/64	1.2344	46.5		1.8307	73.0		2.8740
12.0		.4724	21.0		.8268	31.5		1.2402	46.831	1 27/32	1.8438	73.025	2 7/8	2.8750
12.1		.4764	21.034	53/64	.8281	31.75	1 1/4	1.2500	47.0		1.8504	73.819	2 29/32	2.9062
12.2		.4803	21.25		.8366	32.0		1.2598	47.228	1 55/64	1.8594	74.0		2.9134
12.3		.4843	21.431	27/32	.8438	32.147	1 17/64	1.2656	47.5		1.8701	74.612	2 15/16	2.9375
12.303	31/64	.4844	21.5		.8465	32.5		1.2795	47.625	1 7/8	1.8750	75.0		2.9528
12.4		.4882	21.75		.8563	32.544	1 9/32	1.2812	48.0		1.8898	75.406	2 31/32	2.9688
12.5		.4921	21.828	55/64	.8594	32.941	1 19/64	1.2969	48.022	1 57/64	1.8906	76.0		2.9921
12.6		.4961	22.0		.8661	33.0		1.2992	48.419	1 29/32	1.9062	76.2	3	3.0000
12.7	1/2	.5000	22.225	7/8	.8750	33.338	1 5/16	1.3125	48.5		1.9094	76.994	3 1/32	3.0312
12.8		.5039	22.25		.8760	33.5		1.3189	48.816	1 59/64	1.9219	77.0		3.0315
12.9		.5079	22.5		.8858	33.734	1 21/64	1.3281	49.0		1.9291	77.788	3 1/16	3.0625
13.0		.5118	22.622	57/64	.8906	34.0		1.3386	49.212	1 15/16	1.9375	78.0		3.0709
13.097	33/64	.5156	22.75		.8957	34.131	1 11/32	1.3438	49.5		1.9488	78.581	3 3/32	3.0938
13.1		.5157	23.0		.9055	34.5		1.3583	49.609	1 61/64	1.9531	79.0		3.1102
13.2		.5197	23.019	29/32	.9062	34.528	1 23/64	1.3594	50.0		1.9685	79.375	3 1/8	3.1250
13.3		.5236	23.25		.9154	34.925	1 3/8	1.3750	50.006	1 31/32	1.9688	80.0		3.1496
13.4		.5276	32.416	59/64	.9219	35.0		1.3780	50.403	1 63/64	1.9844	80.169	3 5/32	3.1562
13.494	17/32	.5312	23.5		.9252	35.322	1 25/64	1.3906	50.5		1.9882	80.962	3 3/16	3.1875
13.5		.5315	23.75		.9350	35.5		1.3976	5038	2	2.0000	81.0		3.1890
13.6		.5354	23.812	15/16	.9375	35.719	1 13/32	1.4062	51.0		2.0079	81.756	3 7/32	3.2188
13.7		.5394	24.0		.9449	36.0		1.4173	51.594	2 1/32	2.0312	82.0		3.2283
13.8		.5433	24.209	61/64	.9531	36.116	1 27/64	1.4219	52.0		2.0472	82.55	3 1/4	3.2500
13.891	35/64	.5469	24.25		.9547	36.5		1.4370	52.388	2 1/16	2.0625	83.0		3.2677
13.9		.5472	24.5		.9646	36.512	1 7/16	1.4375	53.0		2.0866	83.344	3 9/32	3.2812
14.0		.5512	24.606	31/32	.9688	36.909	1 29/64	1.4531	53.181	2 3/32	2.0938	84.0		3.3071
14.25		.5610	24.75		.9744	37.0		1.4567	53.975	2 1/8	2.1250	84.138	3 5/16	3.3125
14.288	9/16	.5625	25.0		.9843	37.306	1 15/32	1.4688	54.0		2.1260	84.931	3 11/32	3.3438
14.5		.5709	25.003	63/64	.9844	37.5		1.4764	54.769	2 5/32	2.1562	85.0		3.3465
14.684	37/64	.5781	25.25		.9941	37.703	1 31/64	1.4844	55.0		2.1654	85.725	3 3/8	3.3750
14.75		.5807	25.4	1	1.0000	38.0		1.4961	55.562	2 3/16	2.1875	86.0		3.3858
15.0		.5906	253.5		1.0039	38.1	1 1/2	1.5000	56.0		2.2047	86.519	3 13/32	3.4062
15.081	19/32	.5938	25.75		1.0138	38.497	1 33/64	1.5156	56.356	2 7/32	2.2188	87.0		3.4252
15.25		.6004	35.797	1 1/64	1.0156	38.5		1.5157	57.0		2.2441	87.312	3 7/16	3.4375
15.478	39/64	.6094	26.0		1.0236	38.894	1 17/32	1.5312	57.15	2 1/4	2.2500	88.0		3.4646
15.5		.6102	26.194	1 1/32	1.0312	39.0		1.5354	57.944	2 9/32	2.2812	88.106	3 15/32	3.4688
15.75		.6201	26.25		1.0335	39.291	1 35/64	1.5469	58.0		2.2835	88.9	3 1/2	3.5000
15.875	5/8	.6250	26.5		1.0433	39.5		1.5551	58.738	2 5/16	2.3125	89.0		3.5039
16.0		.6299	26.591	1 3/64	1.0469	39.688	1 9/16	1.5625	59.0		2.3228	90.0		3.5433
16.25		.6398	26.75		1.0531	40.0		1.5748	593531	2 11/32	2.3438	910.488	3 9/16	3.5625
16.272	41/64	.6406	26.998	1 1/16	1.625	40.084	1 37/64	1.5781	60.0		2.3622	91.0		3.5827
16.5		.6496	27.0		1.0630	40.481	1 19/32	1.5938	60.325	2 3/8	2.3750	92.0		3.6220
16.669	21/32	.6562	27.25		1.0728	40.5		1.5945	61.0		2.4016	92.075	3 5/8	6.6250
16.75		.6594	27.384	1 5/64	1.0781	40.878	1 39/64	1.6094	61.119	2 13/32	2.4062	93.0		3.6614
17.0		.6693	27.5		1.0827	41.0		1.6142	61.912	2 7/16	2.4375	93.662	3 11.16	3.6875
17.066	43/64	.6719	27.75		1.0925	41.275	1 5/8	1.6250	62.0		2.4409	94.0		3.7008
17.25		.6791	27.781	1 3/32	1.0938	41.5		1.6339	62.706	2 15/32	2.4668	95.0		3.7402
17.462	11/16	.6875	28.0		1.1024	41.672	1 41/64	1.6406	63.0		2.4803	95.25	3 3/4	3.7500
17.5		.6890	28.178	1 7/64	1.1094	42.0		1.6535	63.5	2 1/2	2.5000	96.0		3.7795
17.75		.6988	28.25		1.1122	42.069	1 21/32	1.6562	64.0		2.5197	96.838	3 13/16	3.8125
17.859	45/64	.7031	28.5		1.1220	42.466	1 43/64	1.6719	64.294	2 17/32	2.5312	97.0		3.8189
18.0		.7087	28.575	1 1/8	1.1250	42.5		1.6732	65.0		2.5591	98.0		3.8583
18.25		.7185	28.75		1.1319	42.862	1 11/16	1.6875	65.088	2 9/16	2.5625	98.425	3 7/8	3.8750
18.256	23/32	.7188	28.972	1 9/64	1.1406	43.0		1.6929	65.881	2 19/32	2.5938	99.0		3.8976
18.5		.7283	29.0		1.1417	43.259	1 45/64	1.7031	66.0		2.5984	100.0		3.9370
18.653	47/64	.7344	29.25		1.1516	43.5		1.7126	66.675	2 5/8	2.6250	100.012	3 15/16	3.9375
18.75		.7382	29.369	1 5/32	1.1562	43.656	1 23/32	1.7188	67.0		2.6378	101.6	4	4.0000
19.0		.7480	29.5		1.1614	44.0		1.7323	67.469	2 21/32	2.6562			

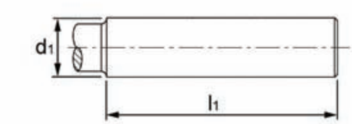
Table Of Cutting Speeds

PERIPHERAL CUTTING SPEED																	
Metres/Min		5	8	10	15	20	25	30	40	50	60	70	80	90	100	110	150
Feet/Min		16	26	32	50	66	82	98	130	165	197	230	262	296	330	362	495
Tool Diameter		REVOLUTIONS PER MINUTE (RPM)															
mm	inch																
1,00		1592	2546	3138	4775	6366	7958	9549	12732	15916	19099	22282	25465	28648	31831	35014	47747
1,50		1061	1698	2122	3183	4244	5305	6366	8488	10610	12732	14854	16977	19099	21221	23343	31831
2,00		796	1273	1592	2387	3183	3979	4775	6366	7958	9549	11141	12732	14324	15916	17507	23873
2,50		637	1019	1273	1910	2546	3183	3820	5093	6366	7639	8913	10186	11459	12732	14006	19099
3,00		531	849	1061	1592	2122	2653	3183	4244	5305	6366	7427	8488	9549	10610	11671	15916
3,18	1/8	500	801	1001	1501	2002	2502	3003	4004	5005	6006	7007	8008	9009	10010	11011	15015
3,50		455	728	909	1364	1819	2274	2728	3638	4547	5457	6366	7176	8185	9095	10004	13642
4,00		398	637	796	1194	1592	1989	2387	3183	3979	4775	5570	6366	7162	7958	8754	11937
4,50		354	566	707	1061	1415	1768	2122	2829	3537	4244	4951	5659	6366	7074	7781	10610
4,76	3/18	334	535	669	1003	1337	1672	2006	2675	3344	4012	4681	5350	6018	6687	7356	10031
5,00		318	509	637	955	1273	1592	1910	2546	3183	3820	4456	5093	5730	6366	7003	9549
6,00		265	424	531	796	1061	1326	1592	2122	2653	3183	3714	4244	4775	5305	5836	7958
6,35	1/4	251	401	501	752	1003	1253	1504	2005	2506	3008	3509	4010	4511	5013	5514	7519
7,00		227	364	455	682	909	1137	1364	1819	2274	2728	3183	3638	4093	4547	5002	6821
7,94	5/16	200	321	401	601	802	1002	1203	1604	2004	2405	2806	3207	3608	4009	4410	6013
8,00		199	318	398	597	796	995	1194	1592	1989	2387	2785	3183	3581	3979	4377	5968
9,00		177	283	354	531	707	884	1061	1415	1768	2122	2476	2829	3183	3537	3890	5305
9,53	3/8	167	267	334	501	668	835	1002	1336	1670	2004	2338	2672	3006	3340	3674	5010
10,00		159	255	318	477	637	796	955	1273	1592	1910	2228	2546	2865	3183	3501	4775
11,11	7/16	143	229	287	430	573	716	860	1146	1433	1719	2006	2292	2579	2865	3152	4298
12,00		133	212	265	398	531	663	796	1061	1326	1592	1857	2122	2387	2653	2918	3979
12,70	1/2	125	201	251	376	501	627	752	1003	1253	1504	1754	2005	2256	2506	2757	3760
14,00		114	182	227	341	455	568	682	909	1137	1364	1592	1819	2046	2274	2501	3410
14,29	9/16	111	178	223	334	446	557	668	891	1114	1337	1559	1782	2005	2228	2450	3341
15,00		106	170	212	318	424	531	637	849	1061	1273	1485	1698	1910	2122	2334	3183
15,88	5/8	100	160	200	301	401	501	601	802	1002	1203	1403	1604	1804	2004	2205	3007
16,00		99	159	199	298	398	497	597	796	995	1194	1393	1592	1790	1989	2188	2984
17,46	11/16	91	146	182	273	365	456	547	729	912	1094	1276	1458	1641	1823	2005	2735
18,00		88	141	177	265	354	442	531	707	884	1061	1238	1415	1592	1768	1945	2653
19,05	3/4	84	134	167	251	334	418	501	668	835	1003	1170	1337	1504	1671	1838	2506
20,00		80	127	159	239	318	398	477	637	796	955	1114	1273	1432	1592	1751	2387
24,00		66	106	133	199	265	332	398	531	663	796	928	1061	1194	1326	1459	1989
25,00		64	102	127	191	255	318	382	509	637	764	891	1019	1146	1273	1401	1910
27,00		59	94	118	177	236	295	354	472	589	707	825	943	1061	1179	1297	1768
30,00		53	85	106	159	212	265	318	424	531	637	743	849	955	1061	1167	1592
32,00		50	80	99	149	199	249	298	398	497	597	696	796	895	995	1094	1492
36,00		44	71	88	133	177	221	265	354	442	531	619	707	796	884	973	1326
40,00		40	64	80	119	159	199	239	318	398	477	557	637	716	796	875	1194
50,00		32	51	64	95	127	159	191	255	318	382	446	509	573	637	700	955

FOR PERIPHERAL SPEEDS NOT GIVEN, RPM CAN BE OBTAINED BY SIMPLE ADDITION OR SUBTRACTION e.g. For 120 metres/min. add 110+10 figures.

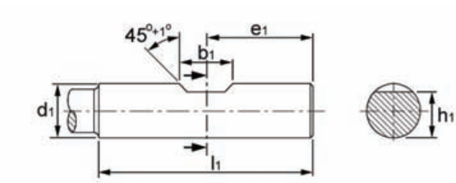
SHANK DESCRIPTIONS AND DIMENSIONS

STRAIGHT SHANKS TO DIN 6535 HA

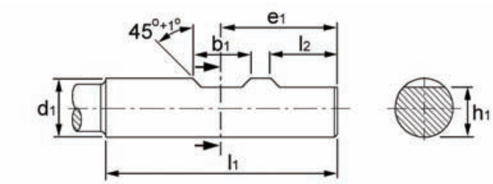


STRAIGHT SHANKS TO DIN 6535 HB

For d_1 = 6 to 20 mm



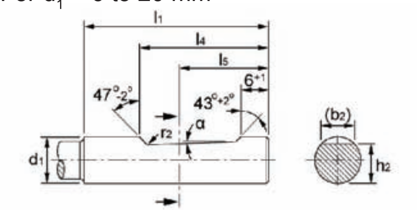
For d_1 = 25 to 32 mm



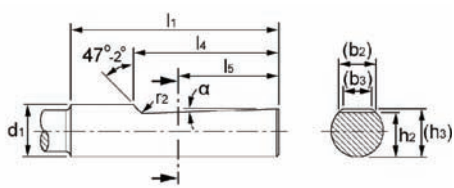
d_1 h6 mm	l_1 +2 mm	b_1 +0,05 mm	e_1 -1 mm	l_2 +1 mm	h_1 h11 mm
2	28	-	-	-	-
3	28	-	-	-	-
4	28	-	-	-	-
5	28	-	-	-	-
6	36	4,2	18	-	5,1
8	36	5,5	18	-	6,9
10	40	7	20	-	8,5
12	45	8	22,5	-	10,4
14	45	8	22,5	-	12,7
16	48	10	24	-	14,2
18	48	10	24	-	16,2
20	50	11	25	-	18,2
25	56	12	32	17	23,0
32	60	14	36	19	30,0

STRAIGHT SHANKS TO DIN 6535 HE

For d_1 = 6 to 20 mm



For d_1 = 25 to 32 mm

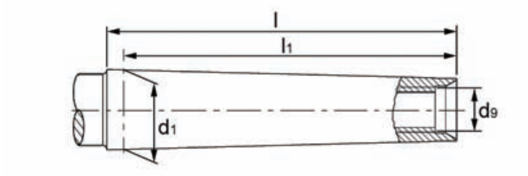


d_1 h6 mm	l_1 +2 mm	l_4 -1 mm	l_5 mm	r_2 mm	α -30´ °	(b_2) ≈ mm	(b_3) mm	h_2 h11 mm	(h_3) mm
6	36	25	18	1,2	2°	4,3	-	5,1	-
8	36	25	18	1,2	2°	5,5	-	6,9	-
10	40	28	20	1,2	2°	7,1	-	8,5	-
12	45	33	22,5	1,2	2°	8,2	-	10,4	-
14	45	33	22,5	1,2	2°	8,1	-	12,7	-
16	48	36	24	1,6	2°	10,1	-	14,2	-
18	48	36	24	1,6	2°	10,8	-	16,2	-
20	50	38	25	1,6	2°	11,4	-	18,2	-
25	56	44	32	1,6	2°	13,6	9,3	23,0	24,1
32	60	48	35	1,6	2°	15,5	9,9	30,0	31,2

STRAIGHT SHANKS TO DIN 1809

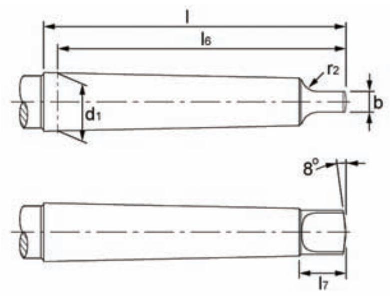
Diameter Range d mm				b h12 mm	l ± IT16 mm	r mm
	3,0	to	3,5	1,6	2,2	
Over	3,5	to	4,0	2	2,2	
Over	4,0	to	4,5	2,2	2,5	0,2
Over	4,5	to	5,5	2,5	2,5	
Over	5,5	to	6,5	3	3	
Over	6,5	to	8,0	3,5	3,5	
Over	8,0	to	9,5	4,5	4,5	
Over	9,5	to	11,0	5	5	
Over	11,0	to	13,0	6	6	0,4
Over	13,0	to	15,0	7	7	
Over	15,0	to	18,0	8	8	
Over	18,0	to	21,0	10	10	

MORSE TAPER SHANKS TO DIN 228 A



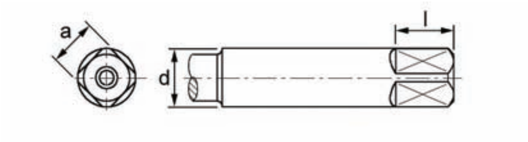
Morse Taper No.	d_1 mm	d_9	l_1 max. mm	l mm	Taper per mm on dia.
0	9,045	-	50	53	0,05205
1	12,065	M6	53,5	57	0,04988
2	17,780	M10	64	69	0,04995
3	23,825	M12	81	86	0,05020
4	31,267	M16	102,5	109	0,05194
5	44,399	M20	129,5	136	0,05263
6	63,348	M24	182	190	0,05214

MORSE TAPER SHANKS TO DIN 228 B



Morse Taper No.	d_1 mm	l_6 -1 mm	b h13 mm	r_2 mm	l_7 max. mm	l mm	Taper per mm on dia.
0	9,045	56,5	3,9	4	10,5	59,5	0,05205
1	12,065	62	5,2	5	13,5	65,5	0,04988
2	17,780	75	6,3	6	16	80	0,04995
3	23,825	94	7,9	7	20	99	0,05020
4	31,267	117,5	11,9	8	24	124	0,05194
5	44,399	149,5	15,9	10	29	156	0,05263
6	63,348	210	19	13	40	218	0,05214

STRAIGHT SHANKS TO DIN 10

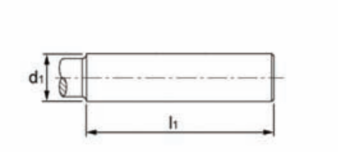


Diameter Range d h9 mm				a h11 mm	l mm
Over	1,32	to	1,50	1,12	4
Over	1,50	to	1,70	1,25	4
Over	1,70	to	1,90	1,40	4
Over	1,90	to	2,12	1,60	4
Over	2,12	to	2,36	1,80	4
Over	2,36	to	2,65	2,00	4
Over	2,65	to	3,00	2,24	5
Over	3,00	to	3,35	2,50	5
Over	3,35	to	3,75	2,80	5
Over	3,75	to	4,25	3,15	6
Over	4,25	to	4,75	3,55	6
Over	4,75	to	5,30	4,00	7
Over	5,30	to	6,00	4,50	7
Over	6,00	to	6,70	5,00	8
Over	6,70	to	7,50	5,60	8
Over	7,50	to	8,50	6,30	9

Diameter Range d h9 mm				a h11 mm	l mm
Over	8,50	to	9,50	7,10	10
Over	9,50	to	10,6	8,00	11
Over	10,6	to	11,8	9,00	12
Over	11,8	to	13,2	10,0	13
Over	13,2	to	15,0	11,2	14
Over	15,0	to	17,0	12,5	16
Over	17,0	to	19,0	14,0	18
Over	19,0	to	21,2	16,0	20
Over	21,2	to	23,6	18,0	22
Over	23,6	to	26,5	20,0	24
Over	26,5	to	30,0	22,4	26
Over	30,0	to	33,5	25,0	28
Over	33,5	to	37,5	28,0	31
Over	37,5	to	42,5	31,5	34
Over	42,5	to	47,5	35,5	38
Over	47,5	to	53,0	40,0	42

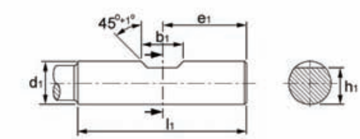
General Information

STRAIGHT SHANKS TO DIN 1835 A

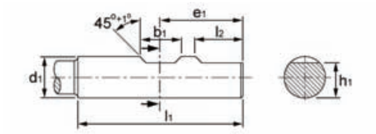


STRAIGHT SHANKS TO DIN 1835 B

FOR D_1 = 6 TO 20 MM

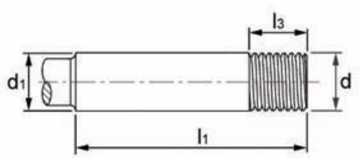


FOR D_1 = 25 TO 63 MM



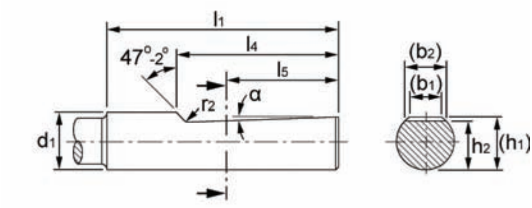
d_1 A=h8, B=h6 mm	l_1 +2 mm	b_1 +0,05 mm	e_1 -1 mm	l_2 +1 mm	h_1 h13 mm
3	28	-	-	-	-
4	28	-	-	-	-
5	28	-	-	-	-
6	36	4,2	18	-	4,8
8	36	5,5	18	-	6,6
10	40	7	20	-	8,4
12	45	8	22,5	-	10,4
16	48	10	24	-	14,2
20	50	11	25	-	18,2
25	56	12	32	17	23
32	60	14	36	19	30
40	70	14	40	19	38
50	80	18	45	23	47,8
63	90	18	50	23	60,8

STRAIGHT SHANKS TO DIN 1835 D



d_1 h6 mm	l_1 +2 mm	l_3 +2 mm	d dimension nominal size	d outside ∅ mm	d core ∅ mm
6	36	10	W 5,90-20	5,9	4,27
10	40	10	W 9,90-20	9,9	8,27
12	45	10	W 11,90-20	11,9	10,27
16	48	10	W 15,90-20	15,9	14,27
20	50	15	W 19,90-20	19,9	18,27
25	56	15	W 24,90-20	24,9	23,27
32	60	15	W 31,90-20	31,9	30,27

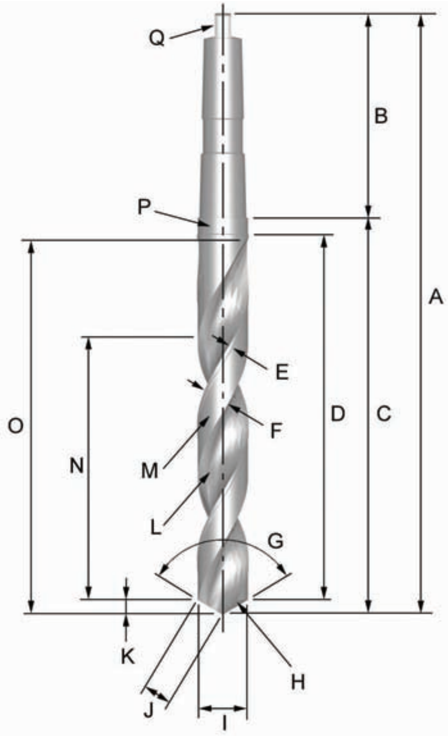
STRAIGHT SHANKS TO DIN 1835 E



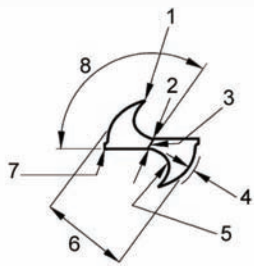
d_1 h6	l_1 +2	l_4 -1	l_5	r_2 min.	α -30´	(b_2)	(b_1) ≈	h_2 h13	(h_1)
6	36	25	18	1,2	2°	4,8	3,5	4,8	5,4
8	36	25	18	1,2	2°	6,1	4,7	6,6	7,2
10	40	28	20	1,2	2°	7,3	5,7	8,4	9,1
12	45	33	22,5	1,2	2°	8,2	6,0	10,4	11,2
16	48	36	24	1,6	2°	10,1	7,6	14,2	15,0
20	50	38	25	1,6	2°	11,5	8,4	18,2	19,1
25	56	44	32	1,6	2°	13,6	9,3	23,0	24,1
32	60	48	35	1,6	2°	15,5	9,9	30,0	31,2

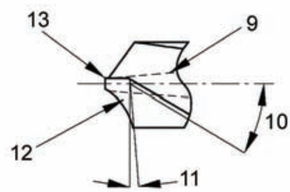
Drilling

Nomenclature



A Overall Length
B Shank
C Body
D Back Tapered Over This Length
E Width of Land
F Width of Fluted Land
G Point Angle
H Major Cutting Edge (Lip)
I Drill Diameter
J Major Cutting Edge (Lip) Length
K Point
L Face
M Body Clearance
N Lead of Helix
O Flute Length
P Silver Ring
Q Tang





1 Heel
2 Web Thickness
3 Chisel Edge
4 Depth of Body Clearance
5 Flute
6 Body Clearance Diameter
7 Outer Corner
8 Chisel Edge Angle

9 Web Taper (shown exaggerated)
10 Axial Rake Angle at Periphery
11 Lip Clearance Angle
12 Flank
13 Chisel Edge Corner

General Hints On Drilling

1. Select the most appropriate drill for the application, bearing in mind the material to be machined, the capability of the machine tool and the coolant to be used.

2. Flexibility within the component and machine tool spindle can cause damage to the drill as well as the component and machine - ensure maximum stability at all times. This can be improved by selecting the shortest possible drill for the application.

3. Tool holding is an important aspect of the drilling operation and the drill cannot be allowed to slip or move in the tool holder.

4. The correct use of Morse Taper Shank drills relies on an efficient fit between the taper surfaces of the tool and the tool holder. The use of a soft-faced hammer should be used to drive the drill into the holder.

5. The use of suitable coolants and lubricants are recommended as required by the particular drilling operation. When using coolants and lubricants, ensure a copious supply, especially at the drill point.

6. Swarf evacuation whilst drilling is essential in ensuring the correct drilling procedure. Never allow the swarf to become stationary in the flute.

7. When regrinding a drill, always makes sure that the correct point geometry is produced and that any wear has been removed.

SELECTION OF DRILL TYPES

Dormer offers an extensive range of standard and special drills with materials and geometry optimised to take into account the cutting behaviour of the workpiece. For example, slow helix drills are better for short chipping materials and quick helix drills are more appropriate for long chipping, ductile alloys.

The following factors have to be taken into consideration when selecting a suitable drill:

- MATERIAL BEING DRILLED
- DEPTH OF HOLE
- MACHINE TOOL CAPABILITY
- COOLANT USED
- CONDITION OF THE MACHINE
- PRODUCTIVITY REQUIREMENTS
- CHOICE OF TOOL HOLDING
- STABILITY OF WORK HOLDING
- HORIZONTAL OR VERTICAL DRILLING
- STATIONARY OR REVOLVING DRILL
- SWARF CONTROL
- HOLE SIZE REQUIREMENTS

Drilling

SELECTION OF DRILLS, FEEDS AND SPEEDS FOR DIFFERENT APPLICATION MATERIALS

The selection of the correct drill and its recommended operating conditions can be found in the Dormer catalogue or Product Selector. Besides the aforementioned considerations, several other factors will dictate a more pertinent selection:

Drill substrate – The materials used for the manufacture of drills could be HSS, HSCo or Solid Carbide. Each material offers certain benefits when drilling certain materials. HSS for example offers high toughness characteristics with relatively low hardness properties. Solid Carbide on the other hand has low impact resistance (Toughness), but very high hardness.

Drill geometry – With the different array of materials to be drilled, comes the need for different drill geometries. Some drills, which are classed as general purpose, will drill a wide range of materials. Application drills however are tools designed with a specific material in mind, i.e. Drills for stainless steel, aluminium or plastics..

Surface Coating – A selection of hard surface coatings are available e.g. Titanium Nitride, Titanium Aluminium Nitride. These are applied to further enhance drill performance, offering different levels of surface hardness, thermal properties and friction coefficient.

The combination of all or some of the above factors has generated a large and comprehensive range of products from which you can choose the most applicable. Ranging from a general purpose HSS drill with standard geometry and no hard surface coating through to a Solid Carbide high performance drill with enhanced geometry and Titanium Aluminium Nitride coating.

Hole Size

As geometric, substrate and coating configurations become more advanced, the ability of a drill to produce a more accurate hole size increases. In general, a standard geometry tool will achieve a hole size to H12. However as the configuration of the drill becomes more complex the achievable hole size, under favourable conditions, can be as good as H8.

To offer a better insight, listed below are the product types and their achievable hole tolerances:

HSS General Purpose drills – H12

HSS / HSCo Parabolic Flute Deep Hole Drills (PFX) – H10

HSS / HSCo High performance TiN/ TiALN coated (ADX) – H9

Solid Carbide High Performance TiN / TiALN coated (CDX) – H8

NOMINAL HOLE DIAMETER (MM)

∅ (mm)	H8	H9	H10	H12
≤ 3	0 / +0.014	0 / +0.025	0 / +0.040	0 / +0.100
> 3 ≤ 6	0 / +0.018	0 / +0.030	0 / +0.048	0 / +0.120
> 6 ≤ 10	0 / +0.022	0 / +0.036	0 / +0.058	0 / +0.150
> 10 ≤ 18	0 / +0.027	0 / +0.043	0 / +0.070	0 / +0.180
> 18 ≤ 30	0 / +0.033	0 / +0.052	0 / +0.084	0 / +0.210

NOMINAL HOLE DIAMETER (INCHES)

∅ (inch)	H8	H9	H10	H12
≤ .1181	0 / +0.0006	0 / +0.0010	0 / +0.0016	0 / +0.0040
>.1181≤.2362	0 / +0.0007	0 / +0.0012	0 / +0.0019	0 / +0.0048
>.2362 ≤.3937	0 / +0.0009	0 / +0.0015	0 / +0.0023	0 / +0.0059
>.3937≤.7087	0 / +0.0011	0 / +0.0017	0 / +0.0028	0 / +0.0071
>.7087≤1.1811	0 / +0.0013	0 / +0.0021	0 / +0.0033	0 / +0.0083

In view of the ability of some drills to produce a much tighter hole tolerance, due consideration should be given to drilled holes which are subject to secondary operations, eg. tapping, reaming. The diameter of the drill will need to be increased from what is recommended to account for the fact that the hole size produced will be smaller.

General Speed And Feed Guide For 2 Diameter Products

When calculating the speed and feed of two diameter drills like centre drills, step drills and subland drills, a compromise is used between the two diameters.
The largest cutting diameter is used to calculate the speed (RPM) and smallest diameter used to establish the feed (mm/rev).

Through Tool Coolant Pressure

The use of through tool coolant is to maintain a copious coolant flow direct to the point of the drill, therefore reducing the amount of heat generated and consequently increasing tool life. High performance drills require an increase in coolant pressure as the coolant flow not only keeps the cutting area cool, it helps with the efficient evacuation of swarf at high penetration rates. In short, the higher the coolant pressure, the more efficient the cooling and swarf evacuation processes are. For high performance and increased productivity, the coolant pressure should be at a minimum 20 bar.

Radial Run Out

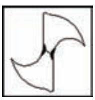
Radial run out is measured at the point of the tool whilst held in a collet on the shank.
The total indicator reading (TIR) is taken by rotating the tool.
For Solid Carbide Tools, 0.02mm max.
For High Performance HSS Tools, 0.11mm max.
For HSS Jobber Drills, use the algorithm 0.01mm x (Overall Length/Diameter)+0.03mm

Drilling

Flute Form

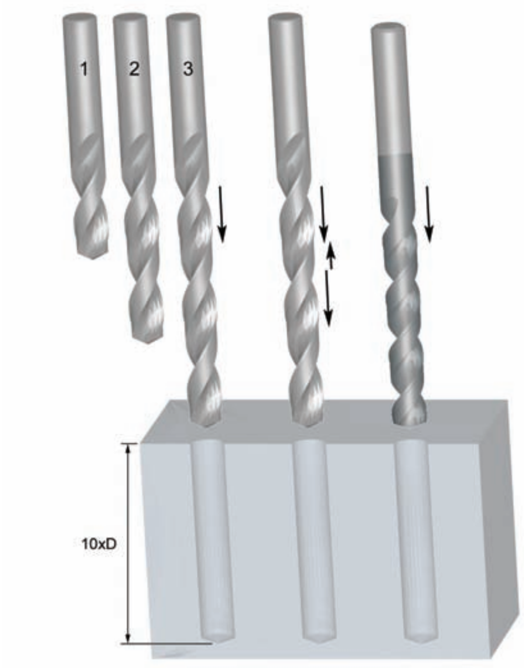
Description	Type of flute	Used for
H	Type H - Slow Spiral (10º to 20º Helix Angle)	Application drills for plastic and brass
N	Type N - Standard Spiral (21º to 34º Helix Angle)	General purpose drills
W	Type W - Quick Spiral (35º to 45º Helix Angle)	Application drills for Stainless Steel and Aluminium. General Purpose High Performance Drills
CTW™	CTW - Continuously Thinned Web	Type N flute with integral thinning for the total flute length

Point Type

Description	Type of point
	4 facet point
NAS 907	National Aerospace standard 907. A recognised standard with the Aerospace Industry
	Point thinned. Used on large diameter drills with a large chisel edge
PS	PS point is the point geometry of A001 / A002. It is a Dormer designation
S.P.	Special point. Again this is a Dormer designation to describe the point geometries of ADX and CDX drills.

Deep Hole Drilling

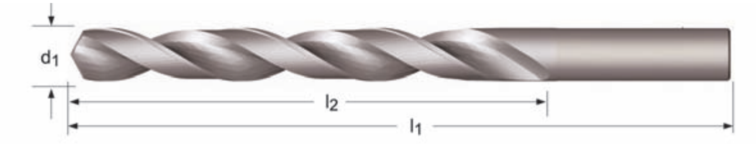
When drilling deep holes, several methods can be adopted to achieve the depth required. The example below shows four ways of drilling a hole with 10 x the diameter of the drill.



	Series Drilling	Series Drilling	Peck Drilling	Single Pass Drilling
No of drills	3 (2,5xD, 6xD,10xD)	2 (2,5xD,10xD)	1 (10xD)	1 (10xD)
Type of drill	Standard geometry, general purpose	2,5xD ADX or PFX 10xD PFX	Standard geometry, general purpose	PFX geometry and purpose specific tools
+ / -	Expensive Time consuming	More cost effective, quick	Time consuming	Cost effective Fast

Drilling

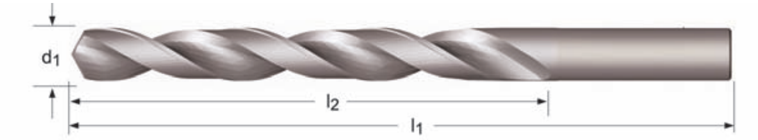
Standard Length and Flute Length - DIN



	DIN 1897		DIN 338		DIN 340		DIN 1869						DIN 6537				DIN 345	
d_1	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2
mm	mm		mm		mm		mm		mm		mm		mm		mm		mm	
							Series 1		Series 2		Series 3		K		L			
≤ 0,24	19	1,5	19	2,5														
≤ 0,30	19	1,5	19	3														
≤ 0,38	19	2	19	4														
≤ 0,48	19	2,5	20	5														
≤ 0,53	20	3	22	6	32	12												
≤ 0,60	21	3,5	24	7	35	15												
≤ 0,67	22	4	26	8	38	18												
≤ 0,75	23	4,5	28	9	42	21												
≤ 0,85	24	5	30	10	46	25												
≤ 0,95	25	5,5	32	11	51	29												
≤ 1,06	26	6	34	12	56	33												
≤ 1,18	28	7	36	14	60	37												
≤ 1,32	30	8	38	16	65	41												
≤ 1,50	32	9	40	18	70	45												
≤ 1,70	34	10	43	20	75	50	115	75										
≤ 1,90	36	11	46	22	80	53	115	75										
≤ 2,12	38	12	49	24	85	56	125	85	160	110	205	135						
≤ 2,36	40	13	53	27	90	59	135	90	160	110	215	145						
≤ 2,65	43	14	57	30	95	62	140	95	160	110	225	150						
≤ 3,00	46	16	61	33	100	66	150	100	190	130	240	160	62	20	66	28	114	33
≤ 3,20	49	18	65	36	106	69	155	105	200	135	240	170	62	20	66	28	117	36
≤ 3,35	49	18	65	36	106	69	155	105	200	135	240	170	62	20	66	28	120	39
≤ 3,75	52	20	70	39	112	73	165	115	210	145	265	180	62	20	66	28	120	39
≤ 4,25	55	22	75	43	119	78	175	120	220	150	280	190	66	24	74	36	124	43
≤ 4,75	58	24	80	47	126	82	185	125	235	160	295	200	66	24	74	36	128	47
≤ 5,30	62	26	86	52	132	87	195	135	245	170	315	210	66	28	82	44	133	52
≤ 6,00	66	28	93	57	139	91	205	140	260	180	330	225	66	28	82	44	138	57
≤ 6,70	70	31	101	63	148	97	215	150	275	190	350	235	79	34	91	53	144	63
≤ 7,50	74	34	109	69	156	102	225	155	290	200	370	250	79	36	91	53	150	69
≤ 8,50	79	37	117	75	165	109	240	165	305	210	390	265	89	40	103	61	156	75
≤ 9,50	84	40	125	81	175	115	250	175	320	220	410	280	89	40	103	61	162	81

Drilling

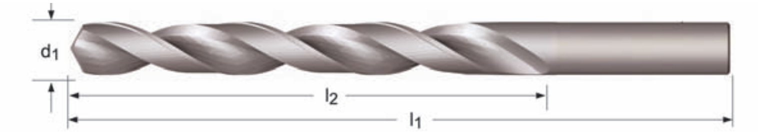
Standard Length and Flute Length - DIN



	DIN 1897		DIN 338		DIN 340		DIN 1869						DIN 6537				DIN 345	
d_1	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2
mm	mm		mm		mm		mm		mm		mm		mm		mm		mm	
							Series 1		Series 2		Series 3		K		L			
≤ 10,60	89	43	133	87	184	121	265	185	340	235	430	295	102	55	118	70	168	87
≤ 11,80	95	47	142	94	195	128	280	195	365	250			102	55	118	70	175	94
≤ 13,20	102	51	151	101	205	134	295	205	375	260			107	60	124	76	182	101
≤ 14,00	107	54	160	108	214	140							107	60	124	76	189	108
≤ 15,00	111	56	169	114	220	144							115	65	133	82	212	114
≤ 16,00	115	58	178	120	227	149							115	65	133	82	218	120
≤ 17,00	119	60	184	125	235	154							123	73	143	91	223	125
≤ 18,00	123	62	191	130	241	158							123	73	143	91	228	130
≤ 19,00	127	64	198	135	247	162							131	79	153	99	233	135
≤ 20,00	131	66	205	140	254	166							131	79	153	99	238	140
≤ 21,20	136	68			261	171											243	145
≤ 22,40	141	70			268	176											248	150
≤ 23,00	141	70			268	176											253	155
≤ 23,60	146	72			275	180											276	155
≤ 25,00	151	75			282	185											281	160
≤ 26,50	156	78			290	190											286	165
≤ 28,00	162	81			298	195											291	170
≤ 30,00	168	84			307	201											296	175
≤ 31,50	174	87			316	207											301	180
≤ 31,75	180	90															306	185
≤ 33,50	180	90															334	185
≤ 35,50	186	93															339	190
≤ 37,50	193	96															344	195
≤ 40,00	200	100															349	200
≤ 42,50	207	104															354	205
≤ 45,00	214	108															359	210
≤ 47,50	221	112															364	215
≤ 50,00	228	116															369	220

Drilling

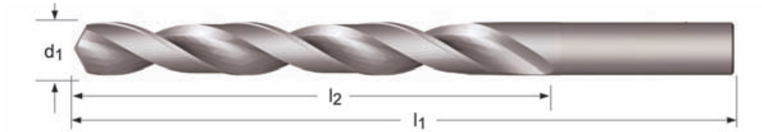
Standard Length And Flute Length - ANSI



Decimal Inch	Decimal Metric	Screw Machine		Jobber Length		Taper Length		Morse Taper shank	
d_1	d_1	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2
		inch	inch	inch	inch	inch	inch	inch	inch
0.0059-0.0079	0.150-0.200			3/4	1/16				
0.0083-0.0100	0.211-0.254			3/4	5/64				
0.0105-0.0130	0.267-0.330			3/4	3/32				
0.0135-0.0145	0.343-0.368			3/4	1/8				
0.0150-0.0157	0.380-0.400			3/4	3/16				
0.0160-0.0200	0.406-0.508			7/8	3/16				
0.0210-0.0225	0.533-0.572			1.	1/4				
0.0236-0.0250	0.600-0.635			1.1/8	5/16				
0.0256-0.0280	0.650-0.711			1.1/4	3/8				
0.0292-0.0330	0.742-0.838			1.3/8	1/2				
0.0335-0.0380	0.850-0.965			1.1/2	5/8				
0.0390-0.420	0.991-1.067	1.3/8	1/2	1.5/8	11/16	2.1/4	1.1/8		
0.0430-0.0469	1.092-1.191	1.3/8	1/2	1.3/4	3/4	2.1/4	1.1/8		
0.0472-0.0625	1.200-1.588	1.5/8	5/8	1.7/8	7/8	3.	1.3/4		
0.0630-0.0635	1.600-1.613	1.11/16	11/16	1.7/8	7/8	3.3/4	2.		
0.0650-0.0781	1.650-1.984	1.11/16	11/16	2.	1.	3.3/4	2.		
0.0785-0.0787	1.994-2.000	1.11/16	11/16	2.	1.	4.1/4	2.1/4		
0.0807-0.0860	2.050-2.184	1.3/4	3/4	2.1/8	1.1/8	4.1/4	2.1/4		
0.0866-0.0938	2.200-2.383	1.3/4	3/4	2.1/4	1.1/4	4.1/4	2.1/4		
0.0945-0.0995	2.400-2.527	1.13/16	13/16	2.3/8	1.3/8	4.5/8	2.1/2		
0.1015-0.1065	2.578-2.705	1.13/16	13/16	2.1/2	1.7/16	4.5/8	2.1/2		
0.1094	2.779	1.13/16	13/16	2.5/8	1.1/2	4.5/8	2.1/2		
0.1100.1130	2.794-2.870	1.7/8	7/8	2.5/8	1.1/2	5.1/8	2.3/4		
0.1142-0.1160	2.900-2.946	1.7/8	7/8	2.3/4	1.5/8	5.1/8	2.3/4		
0.1181-0.1250	3.000-3.175	1.7/8	7/8	2.3/4	1.5/8	5.1/8	2.3/4	5.1/8	1.7/8
0.1260-0.1285	3.200-3.264	1.15/16	15/16	2.3/4	1.5/8	5.3/8	3.	5.3/8	2.1/8
0.1299-0.1406	3.300-3.571	1.15/16	15/16	2.7/8	1.3/4	5.3/8	3	5.3/8	2.1/8
0.1417-0.1496	3.600-3.800	2.1/16	1.	3.	1.7/8	5.3/8	3	5.3/8	2.1/8
0.1520-0.1562	3.861-3.967	2.1/16	1.	3.1/8	2.	5.3/8	3	5.3/8	2.1/8
0.1570	3.988	2.1/8	1.1/16	3.1/8	2.	5.3/4	3.3/8		
0.1575-0.1719	4.000-4.366	2.1/8	1.1/16	3.1/4	2.1/8	5.3/4	3.3/8	5.3/4	2.1/2
0.1730-0.1820	4.394-4.623	2.3/16	1.1/8	3.3/8	2.3/16	5.3/4	3.3/8	5.3/4	2.1/2
0.1850-0.1875	4.700-4.762	2.3/16	1.1/8	3.1/2	2.5/16	5.3/4	3.3/8	5.3/4	2.1/2
0.1890-0.1910	4.800-4.851	2.1/4	1.3/16	3.1/2	2.5/16	6.	3.5/8	6.	2.3/4
0.1929-0.2031	4.900-5.159	2.1/4	1.3/16	3.5/8	2.7/16	6.	3.5/8	6.	2.3/4
0.2040-0.2188	5.182-5.558	2.3/8	1.1/4	3.3/4	2.1/2	6.	3.5/8	6.	2.3/4

For decimal equivalent chart, please see pages 30-31

Standard Length And Flute Length - ANSI

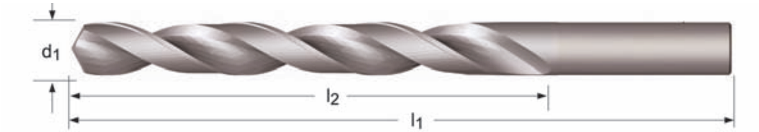


Decimal Inch	Decimal Metric	Screw Machine		Jobber Length		Taper Length		Morse Taper shank	
d_1	d_1	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2
		inch	inch	inch	inch	inch	inch	inch	inch
0.2205-0.2344	5.600-5.954	2.7/16	1.5/16	3.7/8	2.5/8	6.1/8	3.3/4	6.1/8	2.7/8
0.2362-0.2500	6.000-6.350	2.1/2	1.3/8	4.	2.3/4	6.1/8	3.3/4	6.1/8	2.7/8
0.2520	6.400	2.5/8	1.7/16	4.1/8	2.7/8	6.1/4	3.7/8		
0.2559-0.2656	6.500-6.746	2.5/8	1.7/16	4.1/8	2.7/8	6.1/4	3.7/8	6.1/4	3.
0.2660-0.2770	6.756-7.036	2.11/16	1.1/2	4.1/8	2.7/8	6.1/4	3.7/8	6.1/4	3.
0.2795-0.2812	7.100-7.142	2.11/16	1.1/2	4.1/4	2.15/16	6.1/4	3.7/8	6.1/4	3.
0.2835-0.2900	7.200-7.366	2.3/4	1.9/16	4.1/4	2.15/16	6.3/8	4.	6.3/8	3.1/8
0.2913-0.2969	7.400-7.541	2.3/4	1.9/16	4.3/8	3.1/16	6.3/8	4.	6.3/8	3.1/8
0.2992-0.3020	7.600-7.671	2.13/16	1.5/8	4.3/8	3.1/16			6.3/8	3.1/8
0.3031-0.3125	7.700-7.938	2.13/16	1.5/8	4.1/2	3.3/16	6.3/8	4.	6.3/8	3.1/8
0.3150-0.3160	8.000-8.026	2.15/16	1.11/16	4.1/2	3.3/16	6.1/2	4.1/8	6.1/2	3.1/4
0.3189-.03281	8.100-8.334	2.15/16	1.11/16	4.5/8	3.5/16	6.1/2	4.1/8	6.1/2	3.1/4
0.3307-0.3438	8.400-8.733	3.	1.11/16	4.3/4	3.7/16	6.1/2	4.1/8	6.1/2	3.1/4
0.3465-0.3594	8.800-9.129	3.1/16	1.3/4	4.7/8	3.1/2	6.3/4	4.1/4	6.3/4	3.1/2
0.3622-0.3750	9.200-9.525	3.1/8	1.13/16	5.	3.5/8	6.3/4	4.1/4	6.3/4	3.1/2
03770-0.3906	9.576-9.921	3.1/4	1.7/8	5.1/8	3.3/4	7.	4.3/8	7.	3.5/8
0.3937-0.3970	10.000-10.084	3.5/16	1.15/16	5.1/8	3.3/4	7.	4.3/8	7.	3.5/8
0.4016-0.4062	10.200-10.320	3.5/16	1.15/16	5.1/4	3.7/8	7.	4.3/8	7.	3.5/8
0.4130-0.4134	10.490-10.500	3.3/8	2.	5.1/4	3.7/8	7.1/4	4.5/8	7.1/4	3.7/8
0.4219	10.716	3.3/8	2.	5.3/8	3.15/16	7.1/4	4.5/8	7.1/4	3.7/8
0.4252-0.4375	10.800-11.112	3.7/16	2.1/16	5.1/2	4.1/16	7.1/4	4.5/8	7.1/4	3.7/8
0.4409-0.4531	11.200-11.509	3.9/16	2.1/8	5.5/8	4.3/16	7.1/2	4.3/4	7.1/2	4.1/8
0.4646-0.4688	11.800-11.908	3.5/8	2.1/8	5.3/4	4.5/16	7.1/2	4.3/4	7.1/2	4.1/8
0.4724-0.4844	12.000-12.304	3.11/16	2.3/16	5.7/8	4.3/8	7.3/4	4.3/4	8.1/4	4.3/8
0.4921-0.5000	12.500-12.700	3.3/4	2.1/4	6.	4.1/2	7.3/4	4.3/4	8.1/4	4.3/8
0.5039-0.5118	12.800-13.000	3.7/8	2.3/8	6.	4.1/2			8.1/2	4.5/8
0.5156-0.5315	13.096-13.500	3.7/8	2.3/8	6.5/8	4.13/16			8.1/2	4.5/8
0.5433-0.5781	13.800-14.684	4.1/8	2.5/8	6.5/8	4.13/16			8.3/4	4.7/8
0.5807-0.5938	14.750-15.083	4.1/8	2.5/8	7.1/8	5.3/16			8.3/4	4.7/8
0.6004-0.6250	15.250-15.875	4.1/4	2.3/4	7.1/8	5.3/16			8.3/4	4.7/8
0.6299-0.6562	16.000-16.669	4.1/2	2.7/8	7.1/8	5.3/16			9.	5.1/8
0.6594-0.6875	16.750-17.462	4.1/2	2.7/8	7.5/8	5.5/8			9.1/4	5.3/8
0.6890	17.500	4.3/4	3.	7.5/8	5.5/8			9.1/2	5.5/8
0.7031-0.7188	17.859-18.258	4.3/4	3.					9.1/2	5.5/8
0.7283-0.7500	18.500-19.050	5.	3.1/8					9.3/4	5.7/8
0.7656-0.7812	19.446-19.845	5.1/8	3.1/4					9.7/8	6.
0.7879-0.8125	20.000-20.638	5.1/4	3.3/8					10.3/4	6.1/8

For decimal equivalent chart, please see pages 30-31

Drilling

Standard Length And Flute Length - ANSI



Decimal Inch	Decimal Metric	Screw Machine		Jobber Length		Taper Length		Morse Taper shank	
d_1	d_1	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2
		inch	inch	inch	inch	inch	inch	inch	inch
0.8268-0.8750	21.000-22.225	5.3/8	3.1/2					10.3/4	6.1/8
0.8858-0.9062	22.500-23.017	5.5/8	3.5/8					10.3/4	6.1/8
0.9219-0.9375	23.416-23.812	5.3/4	3.3/4					10.3/4	6.1/8
0.9449-0.9688	24.000-24.608	5.7/8	3.7/8					11.	6.3/8
0.9843-1.000	25.000-25.400	6.	4.					11.	6.3/8
1.0039-1.0312	25.500-26.192							11.1/8	6.1/2
1.0433-1.0630	26.500-27.000							11.1/4	6.5/8
1.0781-1.0938	27.384-27.783							12.1/2	6.7/8
1.1024-1.1250	28.000-28.575							12.3/4	7.1/8
1.1406-1.562	28.971-29.367							12.7/8	7.1/4
1.1614-1.1875	29.500-30.162							13.	7.3/8
1.2008-1.2188	30.500-30.958							13.1/8	7.1/2
1.2205-1.2500	31.000-31.750							13.1/2	7.7/8
1.2598-1.2812	32.000-32.542							14.1/8	8.1/2
1.2969-1.3125	32.941-33.338							14.1/4	8.5/8
1.3189-1.3438	33.500-34.133							14.3/8	8.3/4
1.3583-1.3750	34.500-34.925							14.1/2	8.7/8
1.3780-1.4062	35.000-35.717							14.5/8	9.
1.4173-1.4375	36.000-36.512							14.3/4	9.1/8
1.4531-1.4688	36.909-37.308							14.7/8	9.1/4
1.4764-1.5000	37.500-38.100							15.	9.3/8
1.5312	38.892							16.3/8	9.3/8
1.5354-1.5625	39.000-39.688							16.5/8	9.5/8
1.5748-1.5938	40.000-40.483							16.7/8	9.7/8
1.6094-1.6250	40.879-41.275							17.	10.
1.6406-1.8438	41.671-46.833							17.1/8	10.1/8
1.8504-2.0312	47.000-51.592							17.3/8	10.3/8
2.0472-2.1875	52.000-55.563							17.3/8	10.1/4
2.2000-2.3750	56.000-60.325							17.3/8	10.1/8
2.4016-2.500	61.000-63.500							18.3/4	11.1/4
2.5197-2.6250	64.000-66.675							19.1/2	11.7/8
2.6378-2.7500	67.000-69.850							20.3/8	12.3/4
2.7559-2.8125	70.000-71.438							21.1/8	13.3/8

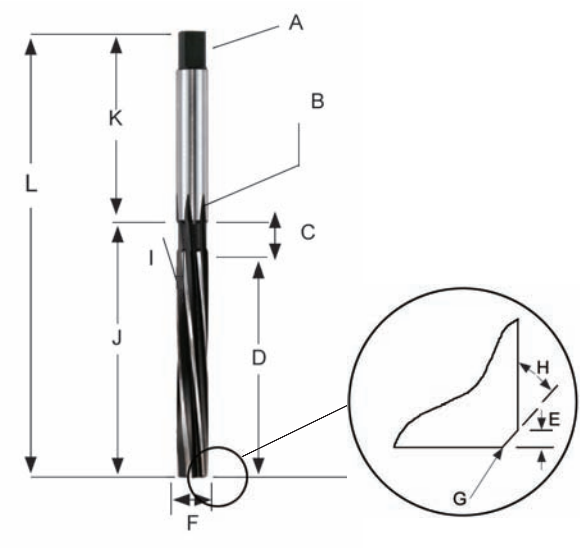
For decimal equivalent chart, please see pages 30-31

Trouble Shooting When Drilling

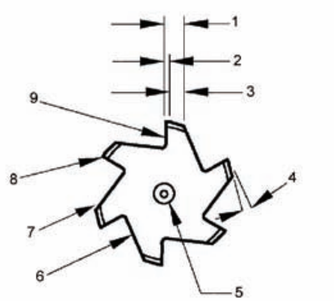
Problem	Cause	Remedy
Broken or twisted tangs	Bad fit between shank and socket	Ensure the shank and socket are clean and free from damage
Splitting of the web	Feed too high	Reduce feed to optimum rate
	Insufficient initial clearance	Regrind to correct specification
	Excessive web thinning	Regrind to correct specification
	Heavy impact at point of drill	Avoid impact at the point of drill. Take care with taper shank drills when inserting / ejecting them from a spindle
Worn outer corners	Excessive speed	Reduce speed to optimum - may be able to increase feed
Broken outer corners	Unstable component set up	Reduce movement in the component
Chipped cutting lips	Excessive initial clearance	Regrind to correct specification
Breakage at flute run out	Choking of flutes	Adopt a peck / series drilling concept
	Drill slipping	Ensure the drill is held securely in the chuck and spindle
Spiral finish in hole	Insufficient feed	Increase feed
	Bad positional accuracy	Use a spot drill before drilling
Hole size too large	Incorrect point geometry	Check point geometry in regrinding section
	Ineffective swarf clearance	Adjust speed, feed and peck length to achieve more manageable swarf

Reaming

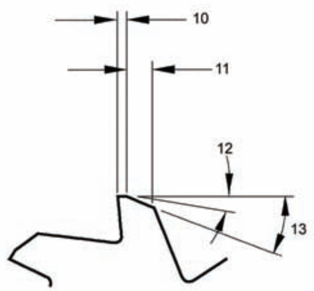
Nomenclature



A Tang
B Recess
C Recess Length
D Cut Length
E Bevel Lead Length
F Diameter
G Bevel Lead
H Bevel Lead Angle
I Helix Angle
J Body Length
K Shank Length
L Overall Length



1 Width of Land
2 Circular Land
3 Clearance
4 Clearance Angle
5 Centre Hole
6 Flute
7 Heel
8 Cutting Edge
9 Face



10 Width of Primary Clearance
11 Width of Secondary Clearance
12 Primary Clearance Angle
13 Secondary Clearance Angle

General Hints On Reaming

To obtain the best results when using reamers it is essential to make them 'work'. It is a common fault to prepare holes for reaming with too little stock left in. If insufficient stock is left in the hole before reaming, then the reamer will rub, quickly show wear and will result in loss of diameter. It is equally important for performance not to leave too much stock in the hole. (See Stock Removal on next page).

1. Select the optimum type of reamer and the optimum speeds and feeds for the application. Ensure that pre-drilled holes are the correct diameter.

2. The workpiece must be held rigid and the machine spindle should have no play.

3. The chuck in which a straight shank reamer is held must be good quality. If the reamer slips in the chuck and the feed is automatic, breakage of the reamer may occur.

4. When driving a Morse Taper Shank reamer into a socket, sleeve or machine spindle, always use a soft faced hammer. Make sure there is a good fit between the reamer shank and the sleeve or socket otherwise misalignment will occur and the reamer may cut oversize.

5. Keep tool overhang from machine spindle to a minimum.

6. Use recommended lubricants to enhance the life of the reamer and ensure the fluid reaches the cutting edges. As reaming is not a heavy cutting operation, soluble oil 40:1 dilution is normally satisfactory. Air blasting may be used with grey cast iron, if dry machining.

7. Do not allow the flutes of a reamer to become blocked with swarf.

8. Before the reamer is reground, check concentricity between centres. In most instances only the bevel lead will need regrinding.

9. Keep reamers sharp. Frequent regrinding is good economy, but it is important to understand that reamers cut only on the bevel and taper leads and not on the lands. Consequently only these leads need regrinding. Accuracy of regrinding is important to hole quality and tool life.

HAND / MACHINE REAMERS

Although both hand and machine reamers offer the same capability regarding finished hole size, the use of each must be considered according to application. A hand reamer, for reasons of alignment, has a long taper lead, whereas a machine reamer has only a 45 degree bevel lead. A machine reamer cuts only on the bevel lead, a hand reamer cuts on the bevel lead and also on the taper lead.

Reaming

APPLICATION REAMERS

As with most cutting tools, the substrate and geometric configuration of reamers differs, dependent on the material they are intended to cut. As such, care should be taken to ensure that the correct choice of reamer is made.

NC reamers are manufactured with a shank tolerance of h6. This enables the reamer to be used in hydraulic and heat shrink tool holding systems, offering enhanced accuracy and concentricity.

ADJUSTABLE REAMERS

Several types of adjustable reamers are available, all offering varying degrees of diameter adjustment. It is an important aspect of adjustable reamers to follow this set procedure:

- Adjust the reamer to the required diameter.
- Check the reamer between centres for concentricity and lip height variation.
- If required, grind the reamer to eliminate any eccentricity or lip height variation.
- Re-check the diameter.

STOCK REMOVAL

The recommended stock removal in reaming is dependent on the application material and the surface finish of the pre-drilled hole. General guidelines for stock removal are shown in the following tables:

Size of reamed hole (mm)	When pre-drilled	When pre-core-drilled		Size of reamed hole (inches)	When pre-drilled	When pre-core-drilled
Below 4	0.1	0.1		Below 3/16	0.004	0.004
Over 4 to 11	0.2	0.15		3/16 to ½	0.008	0.006
Over 11 to 39	0.3	0.2		½ to 1. ½	0.010	0.008
Over 39 to 50	0.4	0.3		1. ½ to 2	0.016	0.010

SELECTION OF REAMER TYPES

Reaming is a recognised method of producing dimensionally accurate holes of fine surface finish. Dormer offers a range of reamers for producing holes to H7 tolerance.

Reamers are classified into various types:

- Solid - available in two shank types, Straight (cylindrical) and Morse Taper.
- Shell - for use on arbors.
- Expanding - with adjustable HSS blades and used for light work.

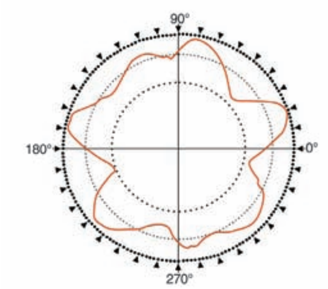
The most common types of reamers have a left-hand spiral because the main applications involve through holes requiring chips to be pushed forward. For blind holes, reamers with straight flutes or right hand spirals are recommended.

The most efficient reaming conditions depend on the application, material, quality of hole required, stock removal, lubrication and other factors. A general guide to surface speeds and feeds for machine reamers is shown in the reamer AMG and feed charts (see Dormer catalogue or Product Selector) and stock removal tables.

Extremely unequal spacing on reamers means that the divide is not the same for each tooth. As there are no two teeth diametrically opposite each other, the reamer produces a hole with a roundness variance of between 1 and 2 µm. This compared with a variance of up to 10µm with unequal spacing.

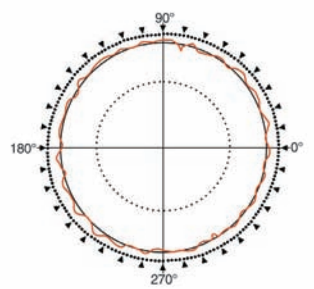
Carbide Reamers - Comparison spacing / EU spacing

unequal spacing
roundness error up to 10 µm



Results of roundness

extremely unequal spacing
roundness error up to 1 - 2 µm



Results of roundness

Reaming

Tolerance Limits

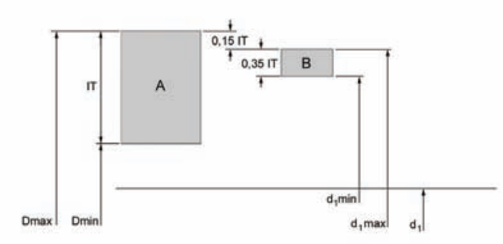
1. ON THE CUTTING DIAMETER OF STANDARD REAMERS
   The diameter (d_1) is measured across the circular land immediately behind the bevel or taper lead. The tolerance is in accordance with DIN 1420 and is intended to produce H7 holes.

REAMER TOLERANCE			
Diameter (mm)		Tolerance Limit (mm)	
Over	Up to and including	High +	Low +
	3	0.008	0.004
3	6	0.010	0.005
6	10	0.012	0.006
10	18	0.015	0.008
18	30	0.017	0.009
30	50	0.021	0.012
50	80	0.025	0.014

2. ON A H7 HOLE
   The most common tolerance on a finished hole is H7 (see table below). For any other tolerance the figure and table beneath point 3 can be used to calculate the reamers tolerance location and width.

HOLE TOLERANCE			
Diameter (mm)		Tolerance Limit (mm)	
Over	Up to and including	High +	Low +
	3	0.010	0
3	6	0.012	0
6	10	0.015	0
10	18	0.018	0
18	30	0.021	0
30	50	0.025	0
50	80	0.030	0

3. When it is necessary to define the dimensions of a special reamer intended to cut to a specific tolerance, e.g. D8, this well proven guide can be used.



A = Hole Tolerance
B = Reamer Tolerance
IT = Tolerance Width
Dmax = Max Diameter of Hole
Dmin = Min Diameter of Hole
d_1 = Nominal Diameter
$d_1$max = Max Diameter of Reamer
$d_1$min = Min Diameter of Reamer

	Diameter Tolerance Width							
Tolerance width	over 1 incl. 3	over 3 incl. 6	over 6 incl. 10	over 10 incl. 18	over 18 incl. 30	over 30 incl. 50	over 50 incl. 80	over 80 incl. 120
IT 5	4	5	6	8	9	11	13	15
IT 6	6	8	9	11	13	16	19	22
IT 7	10	12	15	18	21	25	30	35
IT 8	14	18	22	27	33	39	46	54
IT 9	25	30	36	43	52	62	74	87
IT 10	40	48	58	70	84	100	120	140
IT 11	60	75	90	110	130	160	190	220
IT 12	100	120	150	180	210	250	300	350

Example of a 10mm hole with tolerance D8

Maximum diameter of hole = 10.062
Minimum diameter of hole = 10.040
Hole tolerance (IT8) = 0.022

The maximum limit for the reamer is the maximum limit of the hole size reduced by 0.15 times the tolerance for the hole. The value is rounded up to the next higher multiple of 0.001mm

0.15 x hole tolerance (IT8) = 0.0033, rounded up = 0.004

The minimum limit for the reamer is the maximum limit of the reamer reduced by 0.35 times the tolerance for the hole. The value is rounded up to the next higher multiple 0.001mm.

0.35 x hole tolerance (IT8) = 0.0077, rounded up = 0.008

Maximum limit for reamer = 10.062 - 0.004 = 10.058
Minimum limit for reamer = 10.058 - 0.008 = 10.050

Selection Table for 0.01mm Increment Reamers

Example:

Required Fit: d = 4,25mm F8

Selection: Basic Diameter + Table Value for F8 = 1/100 reamer

4,25 + 0,02 = 4,27mm

Tool Required: 4,27mm Diameter Reamer

	A 9	A 11	B 8	B 9	B10	B11	C 8	C 9	C10	C11	D 7	D 8	D 9	D10	D11
1 - 3	-	+ 0,31	-	-	+ 0,17	+ 0,18	-	-	+ 0,09	+ 0,10	-	-	-	+ 0,05	+ 0,06
3 - 6	+ 0,29	+ 0,32	+ 0,15	+ 0,16	+ 0,17	+ 0,19	+ 0,08	+ 0,09	+ 0,10	+ 0,12	-	+ 0,04	+ 0,05	+ 0,06	+ 0,08
6 - 10	+ 0,30	+ 0,35	+ 0,16	+ 0,17	+ 0,19	+ 0,22	+ 0,09	+ 0,10	+ 0,12	+ 0,15	-	+ 0,05	+ 0,06	+ 0,08	+ 0,11
10 - 18	+ 0,32	+ 0,37	-	+ 0,18	+ 0,20	+ 0,23	+ 0,11	+ 0,12	+ 0,14	+ 0,18	+ 0,06	+ 0,06	+ 0,08	+ 0,10	+ 0,13
	E 7	**E 8**	**E 9**	**F 7**	**F 8**	**F 9**	**F 10**	**G 6**	**G 7**	**H 6**	**H 7**	**H 8**	**H 9**	**H10**	**H11**
1 - 3	-	+0,02	+ 0,03	+ 0,01	-	+ 0,02	-	-	-	-	-	-	-	+ 0,03	+ 0,04
3 - 6	-	+0,03	+ 0,04	-	+ 0,02	+ 0,03	+ 0,04	-	+ 0,01	-	-	+ 0,01	+ 0,02	+ 0,03	+ 0,05
6 - 10	-	-	+ 0,05	+ 0,02	-	+ 0,03	+ 0,05	-	-	-	-	+ 0,01	+ 0,02	+ 0,04	+ 0,07
10 - 18	+ 0,04	-	+ 0,06	-	+ 0,03	+ 0,04	+ 0,07	-	-	-	+ 0,01	-	+ 0,03	+ 0,05	+ 0,08
	H12	**H 13**	**J 6**	**J 7**	**J 8**	**JS 6**	**JS 7**	**JS 8**	**JS 9**	**K 7**	**K 8**	**M 6**	**M 7**	**M 8**	**N 6**
1 - 3	+ 0,08	+ 0,11	-	-	-	-	-	+ 0,00	+ 0,00	-	-	-	-	-	-
3 - 6	+ 0,09	+ 0,14	-	+ 0,00	+ 0,00	-	+ 0,00	+ 0,00	+ 0,00	-	-	-	-	-	-
6 - 10	+ 0,12	+ 0,18	-	+ 0,00	+ 0,00	-	+ 0,00	+ 0,00	+ 0,00	-	-	-	-	- 0,01	-
10 - 18	+ 0,14	+ 0,22	-	+ 0,00	+ 0,00	-	+ 0,00	+ 0,00	+ 0,01	-	-	- 0,01	- 0,01	- 0,01	-
	N 7	**N 8**	**N 9**	**N10**	**N11**	**P 6**	**P 7**	**R6**	**R 7**	**S 6**	**S 7**	**U 6**	**U 7**	**U10**	**Z10**
1 - 3	- 0,01	-	-	- 0,02	- 0,02	-	-	-	-	-	- 0,02	-	-	-	- 0,04
3 - 6	- 0,01	- 0,01	- 0,01	- 0,02	- 0,02	-	-	-	-	-	-	-	-	- 0,04	- 0,05
6 - 10	-	-	-	- 0,02	- 0,02	-	-	-	-	-	-	-	- 0,03	- 0,05	- 0,06
10 - 18	-	-	- 0,02	- 0,02	- 0,03	-	- 0,02	-	-	-	- 0,03	-	-	- 0,05	- 0,07

Notes for use with the above table

This table is formulated to allow the selection of reamers with diameters in 0,01mm increments.

The values given take into consideration the the basic manufacturing tolerances as standard. These are:

Up to Diameter 5,50mm + 0,004 / 0

Over 5.50mm + 0,005 / 0

All tolerances in blue are achievable with 0,01mm increment reamers as they correspond to the manufacturing tolerances for reamers according to DIN 1420.

Standard Length And Flute Length

	DIN 9		DIN 206		DIN 208		DIN 212		DIN 311		DIN 859		DIN 1895		DIN 2180	
d_1	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2
mm	mm		mm		mm		mm		mm		mm		mm		mm	
≤ 0,24																
≤ 0,30																
≤ 0,38																
≤ 0,48																
≤ 0,53																
≤ 0,60	38	20														
≤ 0,67																
≤ 0,75																
≤ 0,85	42	24														
≤ 0,95																
≤ 1,06	46	28														
≤ 1,18																
≤ 1,32	50	32					34	5.5								
≤ 1,50	57	37	41	20			40	8								
≤ 1,70			44	21			43	9								
≤ 1,90			47	23			46	10								
≤ 2,12	68	48	50	25			49	11								
≤ 2,36			54	27			53	12								
≤ 2,65	68	48	58	29			57	14								
≤ 3,00	80	58	62	31			61	15								
≤ 3,35			66	33			65	16								
≤ 3,75			71	35			70	18								
≤ 4,25	93	68	76	38			75	19			76	38				
≤ 4,75			81	41			80	21			81	41				
≤ 5,30	100	73	87	44	133	23	86	23			87	44			155	73
≤ 6,00	135	105	93	47	138	26	93	26			93	47			187	105
≤ 6,70			100	50	144	28	101	28	151	75	100	50	137	61		
≤ 7,50			107	54	150	31	109	31	156	80	107	54				

Reaming

	DIN 9		DIN 206		DIN 208		DIN 212		DIN 311		DIN 859		DIN 1895		DIN 2180	
d_1	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2	l_1	l_2
mm	mm		mm		mm		mm		mm		mm		mm		mm	
≤ 8,50	180	145	115	58	156	33	117	33	161	85	115	58			227	145
≤ 9,50			124	62	162	36	125	36	166	90	124	62				
≤ 10,60	215	175	133	66	168	38	133	38	171	95	133	66	142	66	257	175
≤ 11,80			142	71	175	41	142	41	176	100	142	71				
≤ 13,20	255	210	152	76	182	44	151	44	199	105	152	76			315	210
≤ 14,00					189	47	160	47	209	115						
≤ 15,00	280	230	163	81	204	50	162	50	219	125	163	81	173	79		
≤ 16,00					210	52	170	52	229	135					335	230
≤ 17,00			175	87	214	54	175	54	251	135	175	87				
≤ 18,00					219	56	182	56								
≤ 19,00			188	93	223	58	189	58	261	145	188	93				
≤ 20,00	310	250	201	100	228	60	195	60							377	250
≤ 21,20					232	62			271	155	201	100	212	96		
≤ 22,40			215	107	237	64										
≤ 23,60					241	66			281	165	215	107				
≤ 25,00	370	300			268	68									427	300
≤ 26,50			231	115	273	70			296	180	231	115	263	119		
≤ 28,00					277	71										
≤ 30,00	400	320	247	124	281	73			311	195	247	124			475	320
≤ 31,50					285	75			326	210						
≤ 33,50			265	133	317	77			354	210	265	133				
≤ 35,50					321	78										
≤ 37,50			284	142	325	79			364	220	284	142				
≤ 40,00	430	340			329	81			374	230			331	150	495	340
≤ 42,50			305	152	333	82					305	152				
≤ 45,00					336	83										
≤ 47,50			326	163	340	84			384	240	326	163				
≤ 50,00	460	360	347	174	344	86			394	250	347	174			550	360

Reamer Form and DIN Designation

DIN	Form	Description
212	A	Straight Flute ≤ 3.5mm diameter
	B	Spiral Flute ≤ 3.5mm diameter
	C	Straight Flute ≥ 4.0mm diameter
	D	Spiral Flute ≥ 4.0mm diameter
	E	Quick Spiral
208 219	A	Straight Flute
	B	Spiral Flute
	C	Quick Spiral
9, 205,206, 859, 8050, 8051, 8093, 8094	A	Straight Flute
	B	Spiral Flute
1895	C	Spiral Flute
	D	Quick Spiral
	E	Straight Flute

Spiral Flute = 7° left hand spiral
Quick Spiral = 45° left hand spiral

Reaming

Trouble Shooting When Reaming

PROBLEM	CAUSE	REMEDY
Broken or twisted tangs	Incorrect fit between shank and socket	Ensure the shank and the socket are clean and free from damage
Rapid Tool Wear	Insufficient stock to remove	Increase the amount of stock to be removed (See Page 52)
Oversize Hole	Excessive lip height variation	Regrind to correct specification
	Displacement in the machine spindle	Repair and rectify spindle displacement
	Defects on the tool holder	Replace tool holder
	Tool shank is damaged	Replace or regrind the shank
	Ovality of the tool	Replace or regrind the tool
	Asymmetric bevel lead angle	Regrind to correct specification
	Too high feed or cutting speed	Adjust cutting conditions in accordance with Catalogue or Product Selector
Undersize hole	Insufficient stock to remove	Increase the amount of stock to be removed (See Page 52)
	Too much heat generated while reaming. The hole widens and shrinks.	Increase coolant flow
	The tool diameter is worn and is undersize.	Regrind to correct specification.
	Too low feed or cutting speed	Adjust cutting conditions in accordance with the Dormer Product Selector.
	Pre-drilled hole is too small	Decrease the amount of stock to be removed. (See Page 52)
Oval and conical holes	Displacement in the machine spindle	Repair and rectify spindle displacement
	Misalignment between tool and hole	Use a bridge reamer
	Asymmetric bevel lead angle	Regrind to correct specification

Reaming

PROBLEM	CAUSE	REMEDY
Bad Hole finish	Excessive stock to remove	Decrease the amount of stock to be removed (See Page 52)
	Worn out tool	Regrind to specification
	Too small cutting rake angle	Regrind to specification
	Too diluted emulsion or cutting oil	Increase % concentration
	Feed and/or speed too low	Adjust cutting conditions in accordance with Catalogue/ Product Selector
	Cutting speed too high	Adjust cutting conditions in accordance with Catalogue/ Product Selector
The tool clamps and breaks	Worn out tool	Regrind to correct specification
	Back taper of the tool is too small	Check and replace / modify the tool
	The width of the land is too wide	Check and replace / modify the tool
	Workpiece material tend to squeeze	Use an adjustable reamer to compensate for the displacement
	Pre-drilled hole is too small	Decrease the amount of stock to be removed (See Page 52)
	Heterogeneous material with hard inclusions	Use solid carbide reamer

Counterboring and Countersinking

General Hints On Counterboring and Countersinking

COUNTERBORING

The counterbore is an end cutting tool which is used to enlarge a preformed hole when a flat bottom is required or to spotface when a machine finish is required. It may have a fixed pilot (solid pattern) **Fig.1** or be designed **Fig.2** for an interchangeable pilot **Fig. 3.**

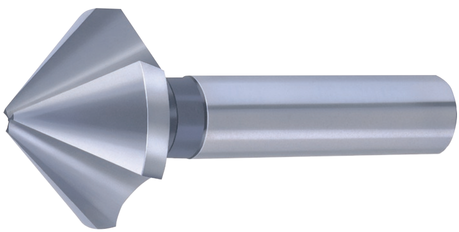
Fig.1

Fig.2

Fig. 3

COUNTERSINKING

The countersink is an conical cutting tool, usually made with angular relief, having one or more flutes with specific size angle cutting edges. It is used for chamfering and countersinking holes. The countersink may have a straight shank, tapered shank, bit stock shank or special shank requiring a special holder, for holding in a power or hand operated machine.



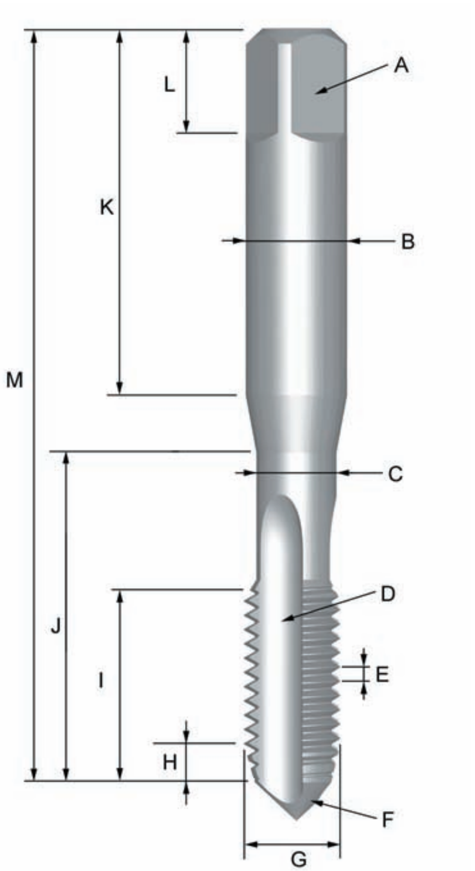
Counterboring and Countersinking

TROUBLE SHOOTING WHEN COUNTERBORING

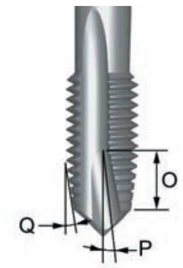
Problem	Cause	Remedy
Excessive Cutting Edge Wear	Incorrect feeds & speeds	Increase feed - especially when machining ductile or free machining materials. Also try reducing speed
	Rough cutting edge	Lightly hone cutting edge with fine grit diamond hone
	Insufficient coolant	Increase coolant flow - review type of coolant
Chipping	Poor chip removal	Use tool with larger flute space - larger diameter or fewer flutes
	Recutting work hardened chips	Increase coolant flow
	Vibration	Increase rigidity of set-up, especially worn tool holders
Short Tool Life	Excessive cratering	Increase speed or decrease feed
	Abrasive material	Decrease speed and increase feed Increase coolant flow
	Hard materials	Reduce speed - rigidity very important
	Insufficient chip room	Use larger diameter tool
	Delayed resharpening	Prompt resharpening to original geometry will increase tool life
Glazed Finish	Feed too light	Increase feed
	Dull cutting edge	Resharpen tool to original geometry
	Insufficient clearance	Resharpen tool with more clearance
Rough Finish	Dull cutting edge	Resharpen to original tool geometry
	Wrong feeds & speeds	Increase speed - also try reducing feed
Chattering	Insufficient machine horsepower	Use tool with fewer flutes as correct feeds & speeds must be maintained
	Vibration	Resharpen tool with more clearance

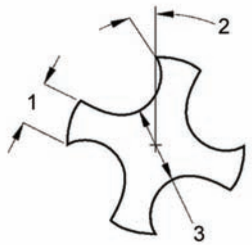
Threading with taps

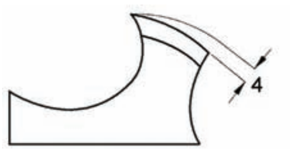
Nomenclature



A Square Across Flat
B Shank Diameter
C Recess Diameter
D Flute
E Pitch
F External Centre (Male)
G Thread Diameter (External)
H Chamfer Lead Length
I Thread Length
J Recess Length
K Shank Length
L Square Length
M Overall Length
N Helix angle
O Spiral Point Length
P Spiral Point Angle
Q Chamfer Lead Angle







1 Land Width
2. Cutting Rake Angle
3 Web Diameter
4 Radial Thread Relief

General Hints On Tapping

The success of any tapping operation depends on a number of factors, all of which effect the quality of the finished product.

1. Select the correct design of tap for the component material and type of hole, i.e. through or blind, from the Application Material Groups chart.

2. Ensure the component is securely clamped - lateral movement may cause tap breakage or poor quality threads.

3. Select the correct size of drill from the tapping drill charts (see pages 76 - 79). The correct sizes of drill are also shown in the tap pages of the catalogue. Remember the drill sizes for fluteless taps are different. Always ensure that work hardening of the component material is kept to a minimum, see stainless steel part in General Information section.

4. Select the correct cutting speed as shown in the Visual Index in the Catalogue and in the Product Selector.

5. Use appropriate cutting fluid for correct application.

6. In NC applications ensure that the feed value chosen for the program is correct. When using a tapping attachment, 95% to 97% of the pitch is recommended to allow the tap to generate its own pitch.

7. Where possible, hold the tap in a good quality torque limiting tapping attachment, which ensures free axial movement of the tap and presents it squarely to the hole. It also protects the tap from breakage if accidentally ‘bottomed’ in a blind hole.

8. Ensure smooth entry of the tap into the hole, as an uneven feed may cause ‘bell mouthing’.

Threading with taps

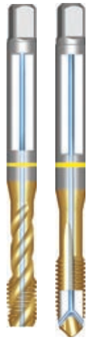
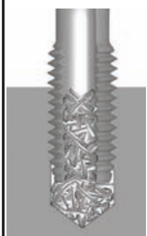
Tap Geometries and Tapping Process

Type	Variants	Process	Description	Chips
		 1,5xD	**Taps with straight flutes** Straight flutes are the most commonly used type of tap. Suitable for use on most materials, mainly short chipping steel and cast iron, they form the basis of the program.	
		 2xD	**Taps with interrupted thread** The interrupted thread ensures less friction and therefore less resistance, which is particularly important when threading material which is resilient and difficult to machine (e.g. aluminium, bronze). It is also easier for lubricant to penetrate to the cutting edges, thus helping to minimise the torque generated.	
		 2,5xD	**Spiral point taps** The tap has a straight fairly shallow flute and is often referred to as a gun nose or spiral point tap. The gun nose or spiral point is designed to drive the swarf forward. The relatively shallow flutes ensure that the sectional strength is maximised. They also act to allow lubricant to reach the cutting edges. This type of tap is recommended for threading through holes.	

Threading with taps

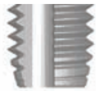
Type	Variants	Process	Description	Chips
		 1,5xD	**Taps with flutes only on the chamfer lead** The cutting part of the tap is formed by gun nosing in the same manner as for a spiral point tap, the function being to drive the swarf forward ahead of the cutting edges. This design is extremely rigid which facilitates good machining results. However, the short length of the gun nosing limits its application to a depth of hole less than about 1.5 x Ø.	
	   	 2xD 3xD	**Taps with spiral flutes** Taps with spiral flutes are intended primarily for threading in blind holes. The helical flute transports the swarf back away from the cutting edges and out of the hole, thus avoiding packing of swarf in the flutes or at the bottom of the hole. In this way, danger of breaking the tap or damaging the thread is minimised.	
		 2,5xD	**Cold forming taps** Cold forming taps differ from cutting taps in that the thread is produced by plastic deformation of the component material rather than by the traditional cutting action. This means that no swarf is produced by their action. The application range is materials with good formability. Tensile strength (Rm) should not exceed 1200 N/mm^2 and the elongation factor (A_5) should not be less than 10%. Cold forming taps without flutes are suitable for normal machining and are especially suitable when vertically tapping blind holes. They are also available with through coolant.	

Threading with taps

Type	Variants	Process	Description	Chips
		 2,5xD 3xD	**Through coolant taps** The performance of taps with through coolant holes is higher than the same taps used with external lubrication. These kinds of taps allow better evacuation of the chip, which is transported away from the cutting area itself. Wear on the cutting edge is reduced, since the cooling effect on the cutting zone is higher than the heat generation. Lubrication can be oil, emulsion or air pressurised with oil mist. Working pressure not less then 15 bar is required, but good results can be obtained with minimal lubrication.	
		 D18-20 C 2-3 2xD	**Nut taps** These taps are generally used to thread nuts but can be used also on deep through holes. They have a shank diameter smaller than the nominal and a longer overall length, because their function is to accumulate nuts. They are used on special machines designed to thread huge amounts of nuts. They can work in steel and stainless steel. The first serial tap has a very long chamfer, in order to spread the cutting load on almost two thirds of the thread length.	

Point/Chamfer Matrix

The type of point on taps is up to the producer to choose. Below is a chart showing the points and chamfers that are commonly used together on products from Dormer, sorted by the diameter of the tap.

Types of Point				
1	**2**	**3**	**4**	
				
Full Point	Reduced point	Internal point	Removed point	

Chamfer Form					
Tap ∅ mm	**A** 6 - 8	**B** 3,5 - 5	**C** 2 - 3	**D** 18 - 20	**E** 1,5 - 2
≤ 5	1	1	1	1	1
>5 ≤6	1	1	1, 2	1	1
>6 ≤10	1, 2	1	1, 2, 4	1, 2	1, 4
>10 ≤12	2, 3	2, 3	2, 3	2, 3	2, 3
>12	3	3	3	3	3
ANSI	Taper	Plug	Bottoming		

Thread (Or Cold) Forming Geometries And Process

Advantages compared with cutting taps

- Cold forming is faster than ordinary thread cutting.
- Cold forming taps often give a longer tool life.
- One type of tool can be used in different materials and for both through and blind holes.
- Cold forming taps have a stable design which gives lower risk of breakage.
- Threads to the correct tolerance are guaranteed.
- No chips.
- Stronger thread (higher stripping strength), compared to thread obtained by cutting (up to 100% more).
- Lower surface roughness on thread obtained by forming than by cutting.

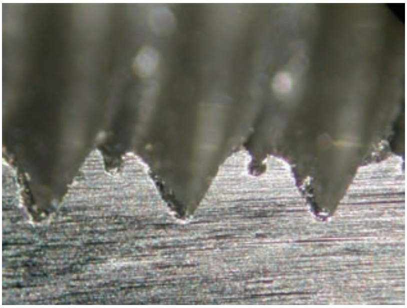
Pre-conditions for effective use

- Sufficent material elongation
  A_5>10 %
- Precise drilled hole diameter.
- Good lubrication is imperative.

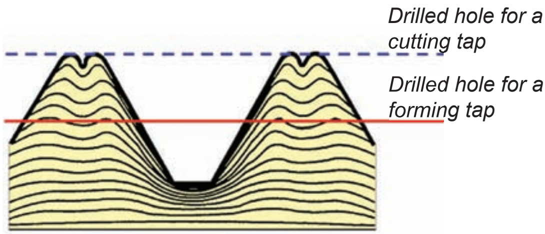
Threading with taps

FLOW OF MATERIAL WHEN FORMING A THREAD

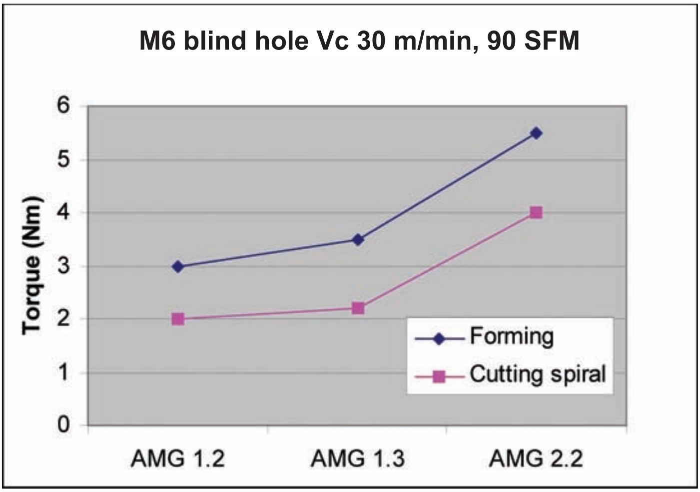
The tapping hole size depends upon the material being drilled, the cutting conditions selected and the condition of the equipment being used. If material is pushed up at the thread entry by the tap and/or the life of the tap is too short, select a slightly larger drill diameter. If on the other hand the profile of the thread formed is insufficient, then select a slightly smaller drill diameter.



Section of thread obtained by forming tap on steel C45



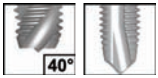
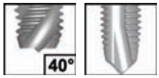
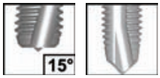
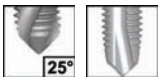
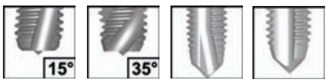
Cold forming taps require more power on the spindle, compared to a cutting tap of the same size, since it generates higher torque.



Torque comparison between forming and cutting taps in different material groups.

Threading with taps

VANGARD / SHARK COLOUR RINGS APPLICATION TAPS

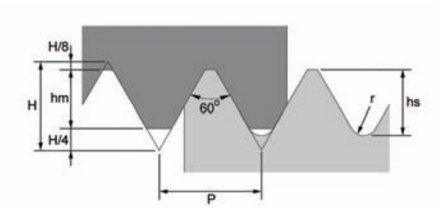
Colour	Material	Tool types available
	AMG 1.1 – AMG 1.4	
	AMG 1.1 – 1.5	
	AMG 1.4 – 1.6	
	AMG 1.5 – 1.6 AMG 4.2 – 4.3	
	AMG 2.1 – AMG 2.3	
	AMG 3.1 – AMG 3.4	
	AMG 5.1 – 5.3	
	AMG 7.1 – 7.4	

Threading with taps

Thread Profiles

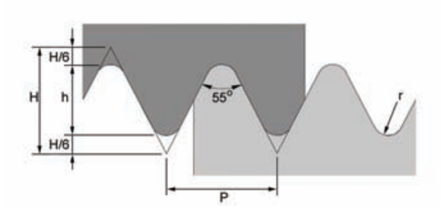
ISO-threads
Metric threads, M
Unified threads, UN

H	=	0.86603 P
Hm	=	5/8H = 0.54127 P
Hs	=	17/24H = 0.613343 P
H/8	=	0.10825 P
H/4	=	0.21651 P
R	=	H/6 = 0.14434P



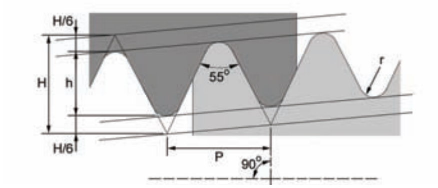
Whitworth W (BSW)
BSF, G, Rp, ADMF, Brass 1/4
BS Conduit, ME

H	=	0.96049 P
H	=	2/3H = 0.64033 P
H/6	=	0.16008 P
R	=	0.13733 P



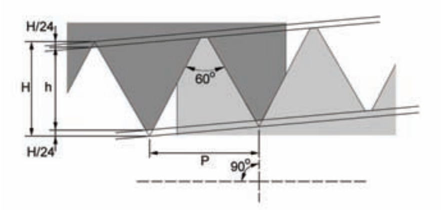
Whitworth conical pipe threads
Rc (BSPT), Conical 1:16

H	=	0.96024 P
H	=	2/3H = 0.64033 P
R	=	0.13728 P



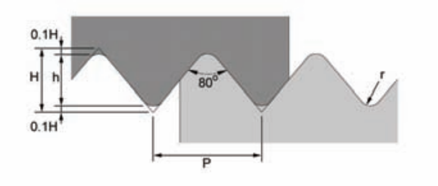
American conical pipe threads
NPT, Conical 1:16

H	=	0.8668 P
H	=	0.800 P
H/24	=	0.033 P (min. value)



Steel conduit threads
PG (Pr)

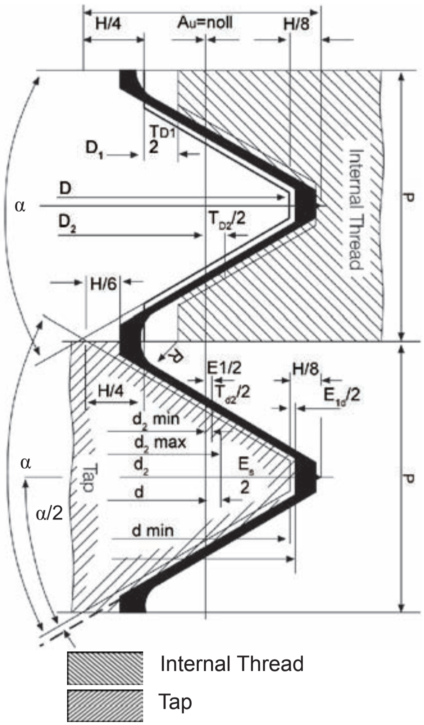
H	=	0.59588 P
H	=	0.4767 P
R	=	0.107 P



TOLERANCES

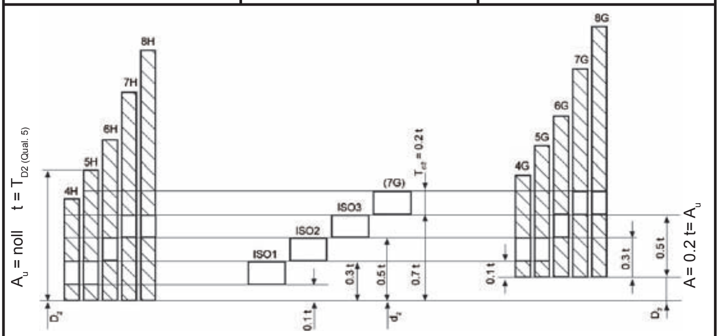
THREAD TOLERANCE WITH TAPS FOR METRIC ISO 60° THREAD PROFILE (M+UN)

Internal thread		**Tap**	
A_u	Base deviation	d	Basic major diameter (=D)
D	Basic major Diameter	d_{min}	Min major diameter
D_1	Basic minor diameter	d_2	Basic pitch diameter
D_2	Basic pitch diameter	d_{2max}	Max pitch diameter
H	Height of fundamental triangle	d_{2min}	Min pitch diameter
P	Pitch	E_1	Lower deviations on d_2
T_{d1}	Tolerance for D1	E_s	Upper deviations on d_2
T_{d2}	Tolerance for D2	E_{1d}	Lower deviations on d
α	Profile angle	P	Pitch
		R	Tap root radius
		T_{d2}	Tolerance on pitch diameter
		$T_{\alpha 2}$	Tolerance on half profile angle
		α	Profile angle
		α/2	Half profile angle



USUAL TOLERANCES FOR TAPS AND INTERNAL THREAD

Tolerance class H Internal Thread (Nut)	Tap Tolerance	Tolerance class G Internal Thread (Nut)



Threading with taps

TABLE OVER TAP TOLERANCE VS TOLERANCE ON INTERNAL THREAD (NUT)

Tolerance class, Tap			Tolerance, Internal thread (Nut)					Application
ISO	DIN	ANSI BS						
ISO 1	4 H	3 B	4 H	5 H				Fit without allowance
ISO 2	6 H	2 B	4 G	5 G	6 H			Normal fit
ISO 3	6 G	1 B			6 G	7 H	8 H	Fit with large allowance
-	7 G	-				7 G	8 G	Loose fit for following treatment or coating

Thread tolerances for taps are collected in standard reference DIN 13.

Normal tolerance is ISO 2 (6H) on taps, which generates an average quality fit between screw and nut. Lower tolerance (ISO 1) generates a fine fit without a gap on the flanks between screw and nut. Higher tolerance (ISO 3) generates a rough fit, with large gap. It is used in the case of a nut which will later be coated or if a loose fit is preferred.

Between tolerances 6H (ISO2) and 6G (ISO3), as well as between 6G and 7G, the tap manufacturer produces taps with tolerance 6HX and 6GX. “X” means the tolerance is outside standard and it is used for taps working high strength material or abrasive material such as cast iron. These materials do not cause oversize problems, so higher tolerance can be used in order to increase tool life. The width of the tolerance is equal between, for example, 6H and 6HX.

Forming taps are usually produced with a 6HX or 6GX tolerance.

The tolerance icon for BSW and BSF is medium. This refers to BS 84 “medium fit”.

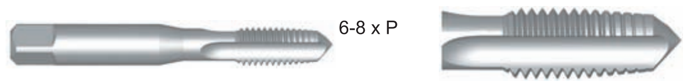
Pipe threads with the tolerance icon “Normal” refer to the following standards:

G threads to ISO 228-1. One class for internal thread (tap), and class A and B for external thread (die).
R, Rc and R threads to ISO 7-1.
NPT and NPSM to ANSI B1.20.1.
NPTF and NPSF to ANSI B1.20.3.
PG to DIN 40 430.

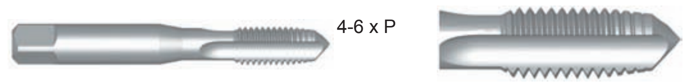
Chamfer Lengths And Serial Taps

The first group (No. 1, No. 2, No. 3) includes taps with complete thread profile and the difference is in the chamfer length. The second group (No. 4, No. 5) includes taps with incomplete thread profile. They have lower pitch and outer diameter, compared to the complete standard, and longer chamfer. After using them, a finishing tap No. 3, must be used.

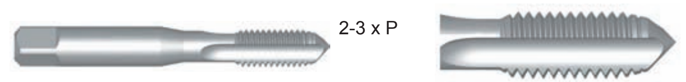
No. 1 =



No. 2 =

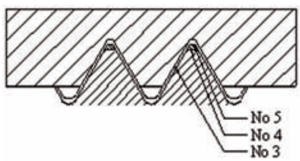


No. 3 =

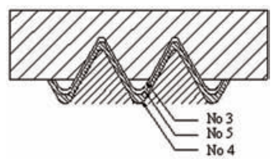


No. 4 =

No. 5 =



Ø<=M10



Ø>=M12

ISO	**Set code number**	**Including tap number**
	No. 6	No. 1 + No. 2 + No. 3
	No. 7	No. 2 + No. 3
	No. 8	No. 4 + No. 5 + No. 3
	No. 9	No. 5 + No. 3

DIN	**Set code number**	**Including tap number**
	No. 8	No.3 (form C) + No.4 (form A) + No.5 (form B)
	No. 9	No.3 (form C) + No.5 (form B)

ANSI	**Set code number**	**Including tap number**
	Hand Tap (No. 6)	Taper(No.1) + Plug(No.2) + Bottoming(No.3)

Threading with taps

Drill Diameters For Cutting Taps – Recommendation Tables

Drill diameter can be calculated from:

$$D = D_{nom} - P$$

D = Drill diameter (mm)
D_{nom} = Tap nominal diameter (mm)
P = Tap pitch (mm)

ISO METRIC COARSE THREAD

TAP M	Pitch mm	Max. Internal Diam. mm	DRILL Diam. mm	DRILL Diam. inch
1.6	0.35	1.321	1.25	3/64
1.8	0.35	1.521	1.45	54
2	0.4	1.679	1.6	1/16
2.2	0.45	1.833	1.75	50
2.5	0.45	2.138	2.05	46
3	0.5	2.599	2.5	40
3.5	0.6	3.010	2.9	33
4	0.7	3.422	3.3	30
4.5	0.75	3.878	3.8	27
5	0.8	4.334	4.2	19
6	1	5.153	5	9
7	1	6.153	6	15/64
8	1.25	6.912	6.8	H
9	1.25	7.912	7.8	5/16
10	1.5	8.676	8.5	Q
11	1.5	9.676	9.5	3/8
12	1.75	10.441	10.3	Y
14	2	12.210	12	15/32
16	2	14.210	14	35/64
18	2.5	15.744	15.5	39/64
20	2.5	17.744	17.5	11/16
22	2.5	19.744	19.5	49/64
24	3	21.252	21	53/64
27	3	24.252	24	61/64
30	3.5	26.771	26.5	1.3/64
33	3.5	29.771	29.5	1.5/32
36	4	32.270	32	1.1/4
39	4	35.270	35	1.3/8
42	4.2	37.799	37.5	
45	4.5	40.799	40.5	
48	5	43.297	43	
52	5	47.297	47	

ISO METRIC FINE THREAD

TAP MF	Max. Internal Diam. mm	DRILL Diam. mm	DRILL Diam inch
3x0.35	1.321	2.65	37
3.5x0.35	3.221	3.2	1/8
4x0.5	3.599	3.5	29
5x0.5	4.599	4.5	16
5.5x0.50	5.099	5	9
6x0.75	5.378	5.3	5
7x0.75	6.378	6.3	D
8x0.75	7.378	7.3	9/32
8x1	7.153	7	J
9x1	8.153	8	O
10x0.75	9.378	9.3	U
10x1	9.153	9	T
10x1.25	8.912	8.8	11/32
11x1	10.153	10	X
12x1	11.153	11	7/16
12x1.25	10.912	10.8	27/64
12x1.5	10.676	10.5	Z
14x1	13.153	13	17/32
14x1.25	12.912	12.8	1/2
14x1.5	12.676	12.5	31/64
15x1	14.153	14	35/64
15x1.5	13.676	13.5	17/32
16x1	15.153	15	19/32
16x1.5	14.676	14.5	9/16
18X1	17.153	17	43/64
18X1.5	16.676	16.5	41/64
18X2	16.210	16	5/8
20X1	19.153	19	3/4
20X1.5	18.676	18.5	47/64
20X2	18.210	18	45/64
22X1	21.153	21	53/64
22X1.5	20.676	20.5	13/16
22X2	20.210	20	25/32
24X1	23.153	23	29/32
24X1.5	22.676	22.5	7/8
24X2	22.210	22	55/64

TAP MF	Max. Internal Diam. mm	DRILL Diam. mm
25X1	24.153	24
25X1.5	23.676	23.5
25x2	23.210	23
26x1.5	24.676	24.5
27x1.5	25.676	25.5
27x2	25.210	25
28x1.5	26.676	26.5
28x2	26.210	26
30x1.5	28.676	28.5
30x2	28.210	28
32x1.5	30.676	30.5
32x2	30.210	30
33x2	31.210	31
35x1.5	33.676	33.5
36x1.5	34.676	34.5
36x2	34.210	34
36x3	33.252	33
38x1.5	36.676	36.5
39x3	36.252	36
40x1.5	38.676	38.5
40x2	38.210	38
40x3	37.252	37
42x1.5	40.676	40.5
42x2	40.210	40
42x3	39.252	39
45x1.5	43.676	43.5
45X2	43.210	43
45X3	45.252	42
48X1.5	46.676	46.5
48X2	46.210	46
48X3	45.252	45
50X1.5	48.686	48.2
50X2	48.210	48
50X3	47.252	47

RECOMMENDED DIAMETERS WHEN USING DORMER ADX AND CDX DRILLS

These tables for drill diameters refer to ordinary standard drills. Modern drills such as Dormer ADX and CDX produce a smaller and more accurate hole which makes it necessary to increase the diameter of the drill in order to avoid breakage of the tap. Please see the small table to the right.

ISO METRIC COARSE THREAD FOR ADX/CDX

TAP M	Pitch mm	DRILL Diameter mm
4	0.70	3.40
5	0.80	4.30
6	1.00	5.10
8	1.25	6.90

TAP M	Pitch mm	DRILL Diameter mm
10	1.50	8.70
12	1.75	10.40
14	2.00	12.25
16	2.00	14.25

DORMER

Drill Diameters For Cutting Taps – Recommendation Tables

ISO UNIFIED COARSE THREAD

TAP UNC	Max. Internal Diam. mm	DRILL Diam. mm	DRILL Diam Inch
nr 2-56	1.872	1.85	50
nr 3-48	2.146	2.1	47
nr 4-40	2.385	2.35	43
nr 5-40	2.697	2.65	38
nr 6-32	2.896	2.85	36
nr 8-32	3.513	3.5	29
nr 10-24	3.962	3.9	25
nr 12-24	4.597	4.5	16
1/4-20	5.268	5.1	7
5/16-18	6.734	6.6	F
3/8-16	8.164	8	5/16
7/16-14	9.550	9.4	U
1/2-13	11.013	10.8	27/64
9/16-12	12.456	12.2	31/64
5/8-11	13.868	13.5	17/32
3/4-10	16.833	16.5	21/32
7/8-9	19.748	19.5	49/64
1-8	22.598	22.25	7/8
1.1/8-7	25.349	25	63/64
1.1/4-7	28.524	28	1.7/64
1.3/8-6	31.120	30.75	1.7/32
1.1/2-6	34.295	34	1.11/32
1.3/4-5	39.814	39.5	1.9/16
2-41/2	45.595	45	1.25/32

ISO UNIFIED FINE THREAD

TAP UNF	Max. Internal Diam. mm	DRILL Diam. mm	DRILL Diam Inch
nr 2-64	1.913	1.9	50
nr 3-56	2.197	2.15	45
nr 4-48	2.459	2.4	42
nr 5-44	2.741	2.7	37
nr 6-40	3.023	2.95	33
nr 8-36	3.607	3.5	29
nr 10-32	4.166	4.1	21
nr 12-28	4.724	4.7	14
1/4-28	5.580	5.5	3
5/16-24	7.038	6.9	I
3/8-24	8.626	8.5	Q
7/16-20	10.030	9.9	25/64
1/2-20	11.618	11.5	29/64
9/16-18	13.084	12.9	33/64
5/8-18	14.671	14.5	37/64
3/4-16	17.689	17.5	11/16
7/8-14	20.663	20.4	13/16
1-12	23.569	23.25	59/64
1.1/8-12	26.744	26.5	1.3/64
1.1/4-12	29.919	29.5	1.11/64
1.3/8-12	33.094	32.75	1.19/64
1.1/2-12	36.269	36	1.27/64

WHITWORTH COARSE THREAD

TAP BSW	Number of t.p.i.	Max. Internal Diam. mm	DRILL Diam. mm
3/32	48	1.910	1.85
1/8	40	2.590	2.55
5/32	32	3.211	3.2
3/16	24	3.744	3.7
7/32	24	4.538	4.5
1/4	20	5.224	5.1
5/16	18	6.661	6.5
3/8	16	8.052	7.9
7/16	14	9.379	9.2
1/2	12	10.610	10.5
9/16	12	12.176	12
5/8	11	13.598	13.5
3/4	10	16.538	16.5
7/8	9	19.411	19.25
1	8	22.185	22
1.1/8	7	24.879	24.75
1.1/4	7	28.054	28
1.3/8	6	30.555	30.5
1.1/2	6	33.730	33.5
1.5/8	5	35.921	35.5
1.3/4	5	39.098	39
1.7/8	4.1/2	41.648	41.5
2	4.1/2	44.823	44.5

CYLINDRICAL WHITWORTH PIPE THREAD

TAP G	Number of t.p.i.	Max. Internal Diam. mm	DRILL Diam. mm
1/8	28	8.848	8.8
1/4	19	11.890	11.8
3/8	19	15.395	15.25
1/2	14	19.172	19
5/8	14	21.128	21
3/4	14	24.658	24.5
7/8	14	28.418	28.25
1	11	30.931	30.75
1.1/4	11	39.592	39.5
1.1/2	11	45.485	45
1.3/4	11	51.428	51
2	11	57.296	57
2.1/4	11	63.342	63
2.1/2	11	72.866	72.5
2.3/4	11	79.216	79
3	11	85.566	85.5

ISO METRIC COARSE INSERT THREAD

TAP EG M	DRILL Diameter mm
2.5	2.6
3	3.2
3.5	3.7
4	4.2
5	5.2
6	6.3
8	8.4
10	10.5
12	12.5
14	14.5
16	16.5
18	18.75
20	20.75
22	22.75
24	24.75

ISO UNIFIED COARSE INSERT THREAD

TAP EG UNC	DRILL Diam. mm
nr 2-56	2.3
nr 3-48	2.7
nr 4-40	3
nr 5-40	3.4
nr 6-32	3.7
nr 8-32	4.4
nr 10-24	5.1
nr 12-24	5.8
1/4-20	6.7
5/16-18	8.4
3/8-16	10
7/16-14	11.7
1/2-13	13.3

Threading with taps

Drill Diameters For Cutting Taps – Recommendation Tables

CYLINDRICAL AMERICAN PIPE THREAD

TAP NPSM	Min Internal Diam mm	Max. Internal Diam. mm	Rec. Drill Diam. mm	Rec. Drill Diam. Inch
1/8”-27	9.039	9.246	9.10	23/64
1/4”-18	11.887	12.217	12.00	15/32
3/8”-18	15.316	15.545	15.50	39/64
1/2”-14	18.974	19.279	19.00	3/4
3/4”-14	24.333	24.638	24.50	31/32
1”-11.1/2	30.506	303.759	30.50	1.13/64
1.1/4”-.11.1/2	39.268	39.497	39.50	1. 9/16
1.1/2”-.11.1/2	45.339	45.568	45.50	1.51/64
2”-11.1/2	57.379	57.607	57.50	2. 1/4
2.1/2”-8	68.783	69.266	69.00	2.23/32
3”-8	84.684	85.166	85.00	3.3/8

CYLINDRICAL AMERICAN PIPE THREAD “DRYSEAL”

TAP NPSF	Min Internal Diam mm	Recommended Drill Diam. mm
1/8”-27	8.651	8.70
1/4”-18	11.232	11.30
3/8”-18	14.671	14.75
1/2”-14	18.118	18.25
3/4”-14	23.465	23.50
1”-11.1/2”	29.464	29.50

TAPERED WHITWORTH PIPE THREAD

TAP Rc	Number of t.p.i.	DRILL Diam. mm
1/8	28	8.4
1/4	19	11.2
3/8	19	14.75
1/2	14	18.25
5/8	14	20.25
3/4	14	23.75
7/8	14	27.5
1	11	30
1.1/8	11	34.5
1.1/4	11	38.5
1.3/8	11	41
1.1/2	11	44.5
1.3/4	11	50
2	11	56
2.1/4	11	62
2.1/2	11	71.5
2.3/4	11	78
3	11	84

TAPERED AMERICAN PIPE THREAD

TAP NPT	Number of t.p.i.	DRILL Diam. mm	DRILL Diam. Inch
1/16	27	6.3	D
1/8	27	8.5	R
1/4	18	11	7/16
3/8	18	14.5	37/64
1/2	14	18	23/32
3/4	14	23	59/64
1	14	29	1.5/32
1.1/4	11.1/2	38	1.1/2
1.1/2	11.1/2	44	1.47/64
2	11.1/2	56	2.7/32
2.1/2	8	67	2.5/8
3	8	83	3.1/4

TAPERED AMERICAN PIPE THREAD “DRYSEAL”

TAP NPTF	Number of t.p.i.	DRILL Diam. mm
1/8	27	8.4
1/4	18	10.9
3/8	18	14.25
1/2	14	17.75
3/4	14	23
1	11.1/2	29
1.1/4	11.1/2	37.75
1.1/2	11.1/2	43.75
2	11.1/2	55.75
2.1/2	8	66.5
3	8	82.5

ARMOUR PIPE THREAD

TAP PG	Number of t.p.i.	Max. Internal Diam. mm	DRILL Diam. mm
7	20	11.45	11.4
9	18	14.01	13.9
11	18	17.41	17.25
13.5	18	19.21	19
16	18	21.31	21.25
21	16	27.03	27
29	16	35.73	35.5
36	16	45.73	45.5
42	16	52.73	52.5
48	16	58.03	58

Drill Diameters For Cold Forming Taps – Recommendation Tables

Drill diameter can be calculated from:

$$D = D_{nom} - 0,0068 * P * 65$$

D = Drill diameter (mm)
D_{nom} = Tap nominal diameter (mm)
P = Tap pitch (mm)
65 in the formula stands for desired thread height in %

ISO METRIC COARSE THREAD

TAP M	Max. Internal Diam. mm	DRILL Diameter mm	DRILL Diameter Inch
2	1.679	1.8	
2.5	2.138	2.3	
3	2.599	2.8	35
3.5	3.010	3.2	30
4	3.422	3.7	
5	4.334	4.6	14
6	5.153	5.5	7/32
8	6.912	7.4	
10	8.676	9.3	
12	10.441	11.2	7/16
14	12.210	13.0	
16	14.210	15.0	

ISO METRIC FINE THREAD

TAP MF	Max. Internal Diam. mm	DRILL Diameter mm
4x0.50	3.599	3.8
5x0.50	4.599	4.8
6x0.75	5.378	5.7
8x0.75	7.378	7.7
8x1.00	7.158	7.5
10x1.00	9.153	9.5
10x1.25	8.912	9.4
12x1.00	11.153	11.5
12x1.25	10.9912	11.4
12x1.50	10.676	11.3
14x1.00	13.153	13.5
14x1.25	12.912	13.4
14x1.50	12.676	13.3
16x1.00	15.153	15.5
16x1.50	14.676	15.25

ISO UNIFIED COARSE THREAD

TAP UNC	Max. Internal Diam. mm	DRILL Diam. mm	DRILL Diam. Inch
nr 1-64	1.582	1.7	51
nr 2-56	1.872	2	47
nr 3-48	2.148	2.3	
nr 4-40	2.385	2.6	39
nr 5-40	2.697	2.9	33
nr 6-32	2.896	3.2	1/8
nr 8-32	3.513	3.8	25
nr 10-24	3.962	4.4	11/64
nr 12-24	4.597	5	9
1/4-20	5.268	5.8	
5/16-18	6.734	7.3	
3/8-16	8.164	8.8	11/32
7/16-14	9.550	10.3	Y
1/2-13	11.013	11.9	.463

ISO UNIFIED FINE THREAD

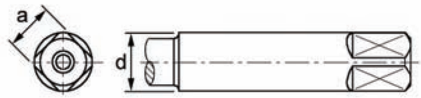
TAP UNF	Max. Internal Diam. mm	DRILL Diam. mm	DRILL Diam. Inch
nr 1-72	1.613	1.7	51
nr 2-64	1.913	2.0	
nr 3-56	2.197	2.3	
nr 4-48	2.459	2.6	37
nr 5-44	2.741	2.9	33
nr 6-10	3.023	3.2	1/8
nr 8-36	3.607	3.9	24
nr 10-32	4.166	4.5	16
nr 12-28	4.724	5.1	7
1/4-28	5.588	6	A
5/16-24	7.038	7.5	.293
3/8-24	8.626	9.1	
7/16-20	10.030	10.6	Z
1/2-20	11.618	12.1	.476

Threading with taps

Shank Description

ISO SHANK AND SQUARE DIMENSIONS

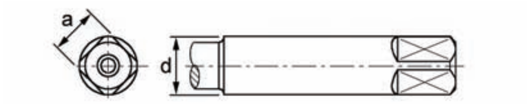
Shank diameter mm	Square mm	ISO 529 Metric	ISO 529 UNC/UNF BSW/BSF	ISO2283 Metric	ISO2284 G	ISO2284 Rc
2,50	2,00	M1				
		M1,2				
		M1,4				
		M1,6	No. 0			
		M1,8				
		M2	No. 1			
2,80	2,24	M2,2	No. 2			
		M2,5	No. 3			
3,15	2,50	M3	No. 4 No. 5	M3		
3,55	2,80	M3,5	No. 6	M3,5 M4		
4,00	3,15	M4		M5		
4,50	3,55	M4,5	No. 8	M6		
5,00	4,00	M5	No. 10 3/16			
5,60	4,50	M5,5	No. 12 7/32	M7		
6,30	5,0	M6	¼	M8		
7,10	5,60	M7	9/32			
8,00	6,30	M8	5/16	M10	G 1/8	Rc 1/8
9,00	7,10	M9		M12		
10,00	8,00	M10	3/8		G ¼	Rc ¼
8,00	6,30	M11	7/16			
9,00	7,10	M12	½			
11,20	9,00	M14	9/16	M14		
12,50	10,00	M16	5/8	M16	G 3/8	Rc 3/8
14,00	11,20	M18 M20	11/16 ¾	M18 M20		
16,00	12,50	M22	7/8	M22		
18,00	14,00	M24	1”	M24	G 5/8	Rc 5/8
20,00	16,00	M27 M30	1 1/8	M27 M30	G ¾	Rc ¾
22,40	18,00	M33	1 ¼		G 7/8	Rc 7/8
25,00	20,00	M36	1 3/8		G 1”	Rc 1”
28,00	22,40	M39 M42	1 ½			



DIN SHANK AND SQUARE DIMENSIONS

Shank diameter mm	Square mm	DIN 352	DIN 371	DIN 376	DIN 374	DIN 2182	DIN 2183	DIN 353 DIN 374
2,5	2,1	M1	M1					
		M1,1	M1,1					
		M1,2	M1,2	M3,5	M3,5	1/16		
		M1,4	M1,4					
		M1,6	M1,6					
		M1,8	M1,8					
2,8	2,1	M2	M2					
		M2,2	M2,2	M4	M4	3/32	5/32	
		M2,5	M2,5					
3,20	2,4						3/16	
3,50	2,70	M3	M3	M5	M5			
4,00	3,00	M3,5	M3,5			1/8		
4,50	3,40	M4	M4	M6	M5,5 M6	5/32	¼	
6,00	4,90	M5 M6 M8	M5 M6	M8	M8	3/16	5/16	
7,00	5,50	M10		M10	M9 M10	¼	3/8	G 1/8
8,00	6,20		M8			5/16	7/16	
9,00	7,00	M12		M12	M12	3/8	½	
10,00	8,00		M10					
11,00	9,00	M14		M14	M14		9/16	G ¼
12,00	9,00	M16		M16	M16		5/8	G 3/8
14,00	11,00	M18		M18	M18		¾	
16,00	12,00	M20		M20	M20			G ½
18,00	14,50	M22 M24		M22 M24	M22 M24		7/8	G 5/8
20,00	16,00	M27		M27	M27 M28		1”	G ¾
22,00	18,00	M30		M30	M30		1 1/8	G 7/8
25,00	20,00	M33		M33	M33		1 ¼	G 1”
28,00	22,00	M36		M36	M36		1 3/8	G 1 1/8
32,00	24,00	M39 M42		M39 M42	M39 M42		1 ½ 1 5/8	G 1 ¼
36,00	29,00	M45 M48		M45 M48	M45 M48		1 ¾ 1 7/8	G 1 ½
40,00	32,00	M52		M52			2	G 1 ¾
45,00	35,00							G 2”
50,00	39,00							G 2 ¼ G 2 ½ G 2 ¾ G 3”

Threading with taps



ANSI SHANK AND SQUARE DIMENSIONS

Shank diameter inch	Square inch	ASME B94.9 machine screw sizes	ASME B94.9 fract. sizes	ASME B94.9 metric sizes
0,141	0,11	No 0		M 1.6
		No 1		M 1.8
		No 2		M 2
		No 3		M 2.5
		No 4		
		No 5		M 3
		No 6		M 3.5
0,168	0,131	No 8		M 4
0,194	0,152	No 10		M 5
0,22	0,165	No 12		
0,255	0,191		¼	M 6
0,318	0,238		5/16	M 7
				M 8
0,381	0,286		3/8	M 10
0,323	0,242		7/16	
0,367	0,275		½	M 12
0,429	0,322		9/16	M14
0,48	0,36		5/8	M16
0,542	0,406		11/16	M18
0,59	0,442		¾	
0,652	0,489		13/16	M20
0,697	0,523		7/8	M22
0,76	0,57		15/16	M24
0,8	0,6		1	M 25
0,896	0,672		1 1/16	M27
			1 1/8	
1,021	0,766		1 3/16	M30
			1 ¼	
1,108	0,831		1 5/16	M33
			1 3/8	
1,233	0,925		1 7/16	M36
			1 ½	
1,305	0,979		1 5/8	M39
1,43	1,072		1 ¾	M42
1,519	1,139		1 7/8	
1,644	1,233		2	M48

Trouble Shooting When Tapping

Problem	Cause	Remedy
Oversize	Incorrect tolerance	Choose a tap with lower thread tolerance
	Incorrect axial feed rate	Reduce feed rate by 5-10% or increase compression of tap holder
	Wrong type of tap for application	Use spiral point for through hole or spiral flute for blind hole. Use coated tool to prevent built up edge. Check Catalogue or Product Selector for correct tool alternative
	Tap not centered on the hole	Check tap holder and position tap centre on the hole
	Lack of lubrication	Use good lubrication in order to prevent built up edge. See lubricant section
	Tap speed too slow	Follow recommendation in Catalogue / Product Selector
Undersize	Wrong type of tap for application	Use spiral point for through hole or spiral flute for blind hole. Use coated tool to prevent built up edge. Use tap with higher rake angle. Check Catalogue or Product Selector for correct tool alternative
	Incorrect tolerance	Choose a tap with higher tolerance, especially on material with low oversize tendency, such as cast iron, stainless steel
	Incorrect or lack of lubricant	Use good lubrication in order ot prevent chip blockage inside the hole. See lubricant section
	Tap drill hole too small	Increase drill diameter to the maximum value. See tap drill tables
	Material closing in after tapping	See recommendation in Catalogue / Product Selector for correct tool alternative
Chipping	Wrong type of tap for application	Choose a tap with lower rake angle. Choose a tap with longer chamfer. Use spiral point taps for through hole and spiral flute for blind holes, in order to avoid chip blockage. Check Catalogue or Product Selector for correct tool alternative
	Incorrect or lack of lubricant	Use good lubrication in order to prevent built up edge. See lubricant section
	Taps hit bottom of hole	Increase depth of drilling or decrease depth of tapping
	Work hardening surface	Reduce speed, use coated tool, use good lubrication. See section for machining of stainless steel
	Swarf trapping on reversal	Avoid sudden return of tap on reversal motion
	Chamfer hits hole entrance	Check axial position and reduce axial error of tap point on hole centre.
	Tap drill hole too small	Increase drill diameter to maximum value. See tap drill tables

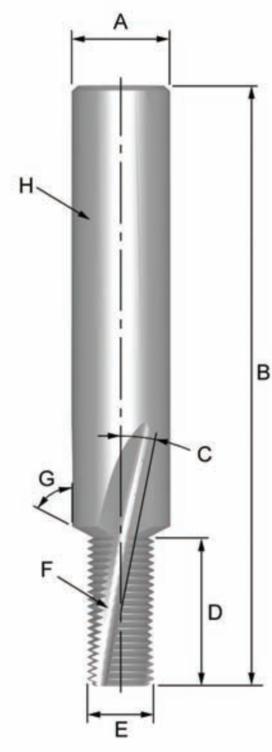
DORMER

Threading with taps

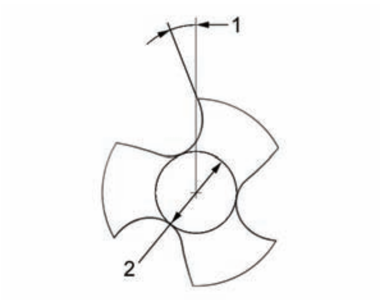
Problem	Cause	Remedy
Breakage	Tap worn out	Use a new tap or regrind the old one
	Lack of lubricant	Use good lubrication in order to prevent built up edge and chip blockage. See lubricant section
	Taps hit bottom of hole	Increase depth of drilling or decrease depth of tapping
	Tap speed too high	Reduce cutting speed. Follow recommendation in Catalogue / Product Selector
	Work hardening surface	Reduce speed. Use coated tool Use good lubrication. See section for machining of stainless steel
	Tap drill hole too small	Increase drill diameter up to maximum value. See tap drill tables
	Too high torque	Use tapping attachment with torque adjustment clutch
	Material closing in after tapping	See recommendation in Catalogue / Product Selector for correct tool alternative
Rapid wear	Wrong type of tap for application	Use tap with lower rake angle and/or higher relief and/or longer chamfer. Use coated tool. Check Catalogue or Product Selector for correct tool alternative
	Lack of lubricant	Use good lubrication in order to prevent built up edge and thermal stress on cutting edge. See lubricant section
	Tap speed too high	Reduce cutting speed. Follow recommendation in Catalogue / Product Selector
Built up edge	Wrong type of tap for application	Use tap with lower rake angle and/or higher relief. Check Catalogue or Product Selector for correct tool alternative
	Lack of lubricant	Use good lubrication in order to prevent built up edge. See lubricant section
	Surface treatment not suitable	See section for surface treatment recommendations
	Tap speed too low	Follow recommendation in Catalogue / Product Selector

Thread milling

Nomenclature



A Shank Diameter
B Overall Length
C Helix Angle
D Cutting Length
E Thread Diameter
F Flute
G Chamfer Angle
H Shank



1 Cutting Rake Angle
2 Web Diameter
3 Radial Thread Relief

General Hints On Thread Milling

Thread milling is a method of producing a thread with a milling operation.
To make this possible, you need a CNC-machine which can make helical paths, (run in 3 directions). Consult your manual or contact your machine supplier for information. You can also write your own sub-program for the thread milling operation.

1. Run the application in the Product Selector and you will get a suggestion of the thread milling cutter best suited for the application, with cutting data and a CNC program optimised for the application. The CNC program can be produced according to the most common systems, including DIN66025(ISO), Heidenhain, Fanuc and Siemens.

2. Use recommended drill sizes for the thread diameter, as for conventional taps.

3. For easy adjustments of the thread tolerance, always program with radius correction. Start value Rprg is printed on the cutter. If a tolerance is selected in the Product Selector you will also get a recommendation of how much to adjust the Rprg value.

4. Use a gauge to check the tolerance on the first thread and then regularly to get an indication if the radius needs to be corrected. The radius can normally be corrected 2-3 times before the thread milling cutter is worn out.

5. When dry machining is used, it is recommended to blow away the chips with compressed air.

6. When working with more difficult materials, it is recommended that the thread milling operation be done in 2 or 3 passes. The Product Selector gives you the choice of generating a CNC-program dividing the cut by a half or 1/3 (2 or 3 passes).

ADVANTAGES OF THREAD MILLING COMPARED TO CONVENTIONAL THREADING

- Thread milling gives increased reliability, namely:
  - Smaller chips.
  - Tolerance adjustments can be made using exact calculations.
  - Full thread to the bottom of the hole.
- Longer tool life.
- Suitable for most materials.
- The same cutter can be used for many diameters, as long as the pitch is the same.
- The same tool can be used both for left and right hand internal threads, and G can be used for both internal and external threads.
- Enables dry machining.
- The countersink on the metric tool makes it possible to chamfer.
- Conical threads with the possibility to chamfer with a superior quality and accuracy compared to conventional taps.

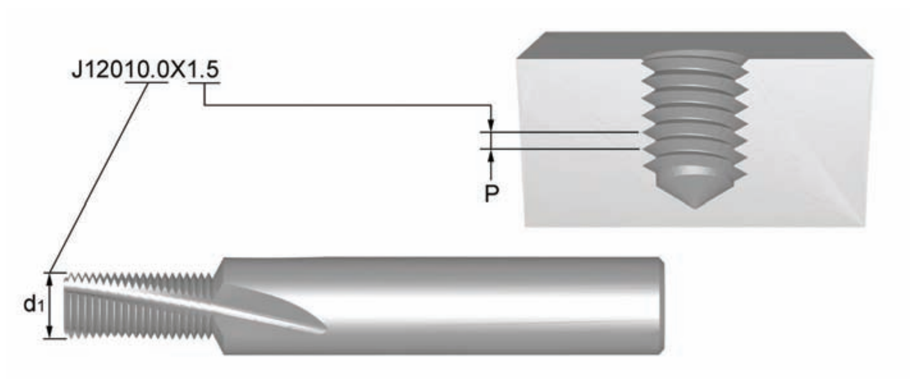
Thread milling

OTHER FACTS ABOUT THREAD MILLING

- Thread milling is a slow process and time savings are apparent on larger dimensions. However, the quality of finish and accuracy achieved can greatly compensate for the speed of the production process.
- Thread depth is limited to 2 x Diameter for metric and 1.5 x Diameter on Metric Fine and G.
- It is possible to regrind the cutter on the rake side (see regrinding section).

CHOOSING YOUR TOOL

All thread milling cutters have an Item Code based on the type, diameter (d_1) and the pitch (P). The item code is the number to use when ordering your tool. Always consult the Catalogue/Product Selector for correct thread dimension of tool.

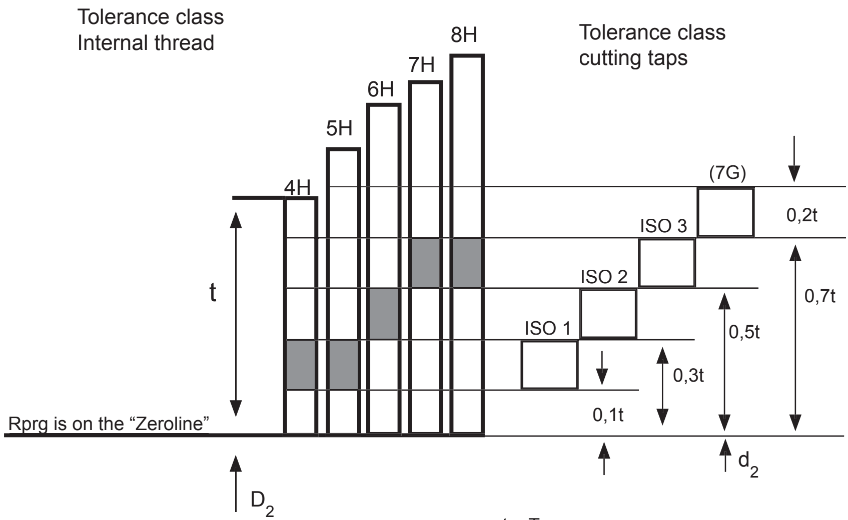


This thread milling cutter can be used for threads ≥ M12x1.5 (M14x1.5, M16x1.5 etc)

PROGRAMMING WITH RPRG

For easy adjustment of the thread tolerance, always program with radius correction. The Rprg value is printed on the cutter and it is normally entered in the tool memory offset. The Rprg is a start value for new cutters.

Rprg is based on the theoretical Zero-line of the thread. This means that when you work with Rprg the thread is never too large but very tight, normally too tight. You have to add a small amount afterwards in order to find out the correct tolerance for your Nominal thread Diameter. Check with a gauge. If you use the Product Selector to generate the CNC-program, you will get a recommendation of how much to adjust the Rprg value to get the selected tolerance. Remember a smaller Rprg value gives a bigger nominal thread diameter.



$t = T_{D2} =$ Tolerance class DIN 13 Part 15
$D_2 = d_2 =$ Basic pitch diameter

Thread milling

Trouble Shooting When Thread Milling

Problem	Cause	Remedy
Short tool life	Wrong cutting data	Reduce speed/feed
	Instability	Check tool holder
	Rapid wear	Reduce speed
Broken cutter	Bad chip evacuation	Use compressed air, emulsion or internal coolant
	Load too big	Divide cut into 2 or 3 passes
		Reduce feed
	Instability	Check/change tool holder
Chipping	Instability	Check/change tool holder
	Wrong cutting data	Reduce speed/feed
	Load too big	Divide cut into 2 or 3 passes
		Reduce feed

For best tool life always use recommended CNC-program with correct soft entry into material. Check with the Product Selector.

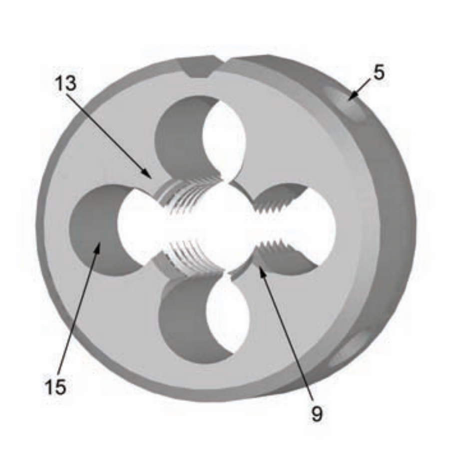
Programming:

"I can't find the program language for my CNC-machine in the Selector."
– Many control systems can be switched over to DIN/ISO when you are doing the thread milling path. Check with the manual.

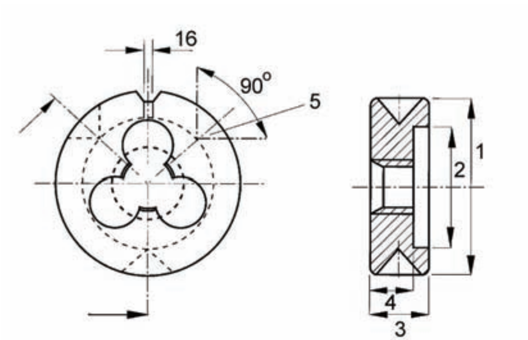
"It is the first time I have used a thread milling cutter and when I run the thread mill above the workpiece, it looks like it will create a much too large a thread."
- The control system does not read that you have moved the centre of the tool to the contour (M41). Check that correct Rprg-value is in the tool-memory offset, and that the correct tool is linked to this Rprg-value.

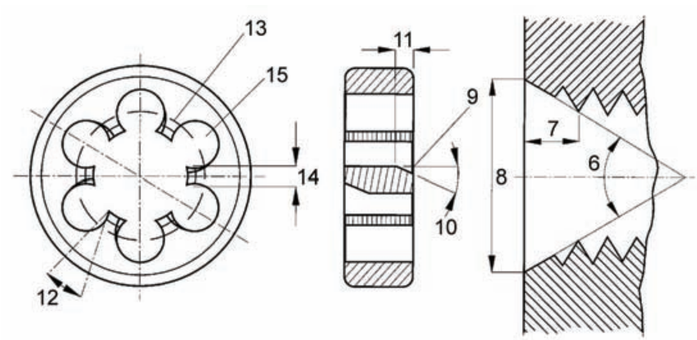
Threading with Dies

Nomenclature



1 Outside Diameter
2 Recess Diameter
3 Thickness
4 Thread Length
5 Conical Hole for Fixing Screw
6 Chamfer Angle
7 Chamfer Length
8 Chamfer Diameter
9 Gun-nose
10 Spiral Angle
11 Spiral Length
12 Rake Angle
13 Land
14 Width of Land
15 Clearance Hole
16 Split of Adjustment





General Hints On Threading With Dies

1. Before starting the die or dienut, chamfer the end of the bar at an angle of 45 degrees to eliminate sudden loading of the leading edges. Ensure the die or dienut is presented to the bolt squarely.

2. Make use of the large tolerances associated with the major diameter of the bolt, by reducing the diameter of the bar (see below). This will reduce the cutting force to a minimum.

3. Use the gun nose type of die, as this ensures the swarf is directed away from the cutting area.

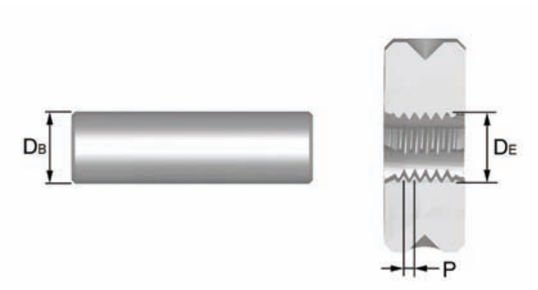
4. Ensure a good supply of the correct lubricant is aimed at the cutting area.

5. When adjusting split dies, avoid opening out as this will cause rubbing. Split dies may be closed down by approximately 0.15mm, by turning the adjustment screws equally. Pressure on one side of the die only may cause breakage.

6. Generally speaking, dienuts are used for reclaiming or cleaning out existing threads by hand. They tend to be of a more robust construction and should only be used in exceptional circumstances to cut a thread from solid.

Pre-machining Dimensions

The diameter of the bolt blank must be smaller than the max. external diameter of the screw thread.



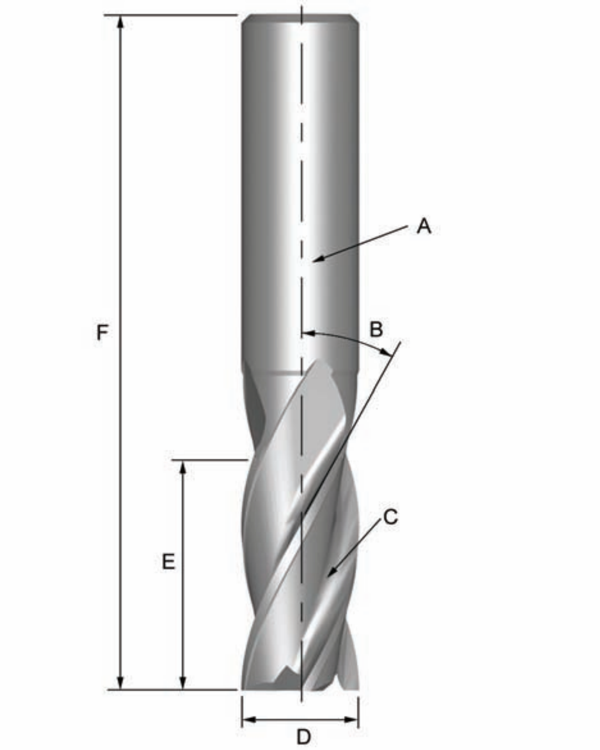
$$D_B = D_E - (0,1 * P)$$

Threading with Dies

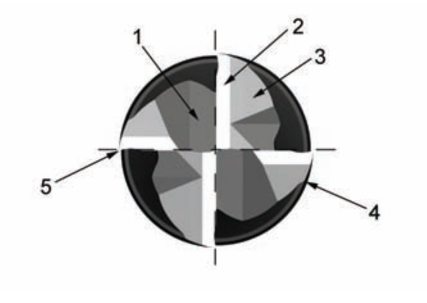
Trouble Shooting When Threading With Dies

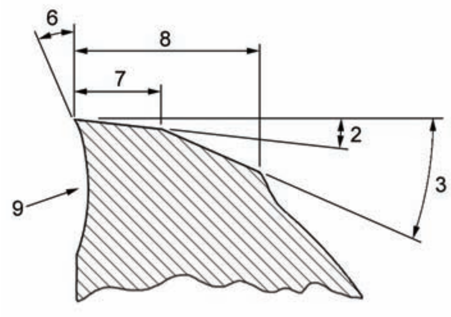
Problem	Cause	Remedy
Oversize / Undersize	Misalignment	Correct alignment, ensure cleanliness
	Incorrect axial feed rate	Ensure axial feed rate is controlled accurately
Poor finish	Incorrect rake angle for the material	Try alternative dies or special die
	Incorrect/lack of lubricant	See lubricants section
	Incorrect speed	Follow recommendations in Catalogue
	Bar diameter too large	Reduce to appropriate size
	Bar end not chamfered	Ensure bar end is chamfered
Chipping / Breakage	Wrong type of die	Follow recommendations in Catalogue
	Speed too high	Follow recommendations in Catalogue
	Bar diameter too large	Reduce to appropriate size
	Bar end not chamfered	Ensure bar end is chamfered
	Misalignment	Correct alignment, ensure cleanliness
Rapid wear	Incorrect/lack of lubricant	See lubricants section
	Speed too high	Follow recommendations in Catalogue
Built up edge	Incorrect/lack of lubricant	See section lubricants
	Bar diameter too large	Reduce to appropriate size
	Speed too low	Follow recommendations in Catalogue

NOMENCLATURE



A Shank
B Helix Angle
C Flute
D Outside Diameter
E Cutting Length
F Overall Length





1 Gash
2 Primary Relief Angle
3 Secondary Relief Angle
4 Heel
5 Cutting Edge

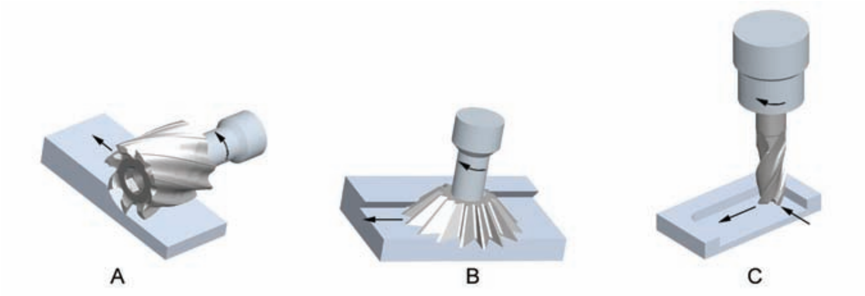
6 Rake Angle
7 Width of Primary Relief Land
8 Width of Secondary Relief Land
9 Undercut Face

General Hints On Milling

Milling is a process of generating machined surfaces by progressively removing a predetermined amount of material or stock from the workpiece at a relatively slow rate of movement or feed by a milling cutter rotating at a comparatively high speed.
The characteristic feature of the milling process is that each milling cutter tooth removes its share of the stock in the form of small individual chips.

TYPE OF MILLING CUTTERS

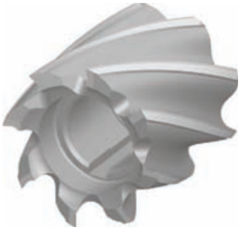
The three basic milling operations are shown below: (A) peripheral milling, (B) face milling and (C) end milling.



In peripheral milling (also called slab milling), the axis of cutter rotation is parallel to the workpiece surface to be machined. The cutter has a number of teeth along its circumference, each tooth acting like a single-point cutting tool called a plain mill.
Cutters used in peripheral milling may have straight or helical teeth generating an orthogonal or oblique cutting action.
In face milling, the cutter is mounted on a spindle with an axis of rotation perpendicular to the workpiece surface. The milled surface results from the action of cutting edges located on the periphery and face of the cutter.
In end milling, the cutter generally rotates on an axis vertical to the workpiece. It can be tilted to machine tapered surfaces. Cutting teeth are located on both the end face of the cutter and the periphery of the cutter body.

Milling

PERIPHERAL AND FACE MILLING CUTTERS

Shell End Mills	Side and Face Cutters	Single and Double Angle Cutters
		 
Has peripheral cutting edge plus face cutting edges on one face. It has a keyway through it to secure it to the spindle.	Has a cutting edge on the sides as well as on the periphery. The teeth are staggered so that every other tooth cuts on a given side of the slot. This allows deep, heavy duty cuts to be taken.	On angle cutters, the peripheral cutting edges lie on a cone rather than on a cylinder. A single or double angle may be created.

END MILLING CUTTERS

Flat End Mills	Ball-nose End Mills	Corner Radius End Mill	Miniature Cutters
			
This end mill has a square angle at the end of the mill.	The shape of the end mill is a semisphere.	This end mill has a small radius instead of the square end.	End Mills with cutting diameter up to 1 mm

SELECTING THE END MILL AND THE MILLING PARAMETERS

Before any milling job is attempted, several decisions must be made to determine:

- the most appropriate end mill to be used
- the correct cutting speed and feed rate to provide good balance between rapid metal removal and long tool life.

Determining the most appropriate end mill:

- identify the type of the end milling to be carried out:-
  1. type of end mill
  2. type of centre.

- consider the condition and the age of the machine tool.

- select the best end mill dimensions in order to minimize the deflection and bending stress:-
  1. the highest rigidity
  2. the largest mill diameter
  3. avoid excessive overhang of tool from tool holder.

- choose the number of flutes
  1. more flutes – decreased space for chips – increased rigidity – allows faster table feed
  2. less flutes – increased space for chips – decreased rigidity – easy chip ejection.

Determining the correct cutting speed and feed rate can only be done when the following factors are known:

- type of material to be machined
- end mill material
- power available at the spindle
- type of finish.

FEATURES OF THE END MILL – END CUTTING EDGES

End cutting edges are divided into:

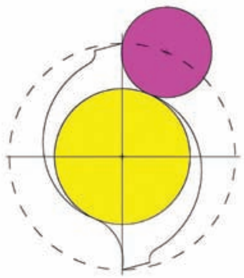
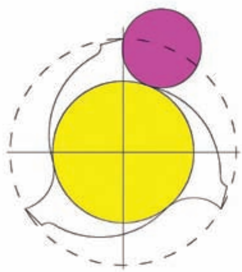
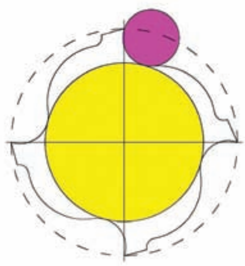
Centre Cutting Type	**Non-Centre Cutting Type**
	
Allows drilling and plunging operations. Two edges reach the centre in the case of an even number of flutes (i.e. 2-4-6, etc). Only one edge in the case of an odd number (i.e. 3-5, etc).	Used only for profiling and open slotting. Allows the regrinding between centres.

Milling

Features Of The End Mill - Choosing The Number Of Flutes

Number of flutes should be determined by:

- Milled material
- Dimension of workpiece
- Milling conditions

2 Flutes	3 Flutes	4 Flutes (or multiflutes)
		
Flexural strength — Low	⟷	High
Chip space — Big	⟷	Small
• Large chip space. • Easy chip ejection. • Good for slot milling. • Good for heavy duty milling. • Less rigidity due to small section area. • Lower quality surface finish.	• Chip space almost as large as for 2 flutes. • Larger section area – higher rigidity than 2 flutes. • Improved surface finish.	• Highest rigidity. • Largest section area – small chip space. • Gives best surface finish. • Recommended for profiling, side milling and shallow slotting.

Features Of The End Mill – Helix Angle

Increasing the number of flutes makes the load on the single tooth more homogeneous and consequently, this allows for a better finish. But with a high helix angle, the load (FV) along the cutter axis is increased too. A high FV can give:

- Load problems on the bearings
- Cutter movement along the spindle axis. To avoid this problem it is necessary to use Weldon or screwed shanks.

Features Of The End Mill – Cutter Type

The DIN 1836 defines the different types of cutter profiles:

N	Cutter type for steel, low to high resistance.
W	Cutter type for soft malleable materials.

The DIN 1836 also defines the chip breakers:

NR	**Coarse pitch rounded profile chip breaker** Suitable for heavy duty cutting on steels and non-ferrous materials with tensile strength up to 800 N/mm^2.
HR	**Fine pitch rounded profile chip breaker** Suitable for rough milling on hard steels and non-ferrous with tensile strength more than 800 N/mm^2.
FS	**Semi-finishing chip breaker** Suitable for the roughing of light alloys and for the semi-finishing of steels and non-ferrous materials.
NF	**Coarse pitch flat profile chip breaker** Has the same application as the NR, obtaining, however, a good finishing surface and for this reason, it is placed between roughing and finishing, also called semi-finishing.

Dormer has introduced two types of roughing cutters, with **asymmetrical chip breaker**:

HRA	Fine pitch asymmetrical rounded profile chip breaker. The asymmetry of the chip breaker reduces vibration and increases tool life.
NRA	Coarse pitch asymmetrical rounded profile chip breaker. The asymmetry of the chip breaker reduces vibration and increases tool life.

End Milling Types

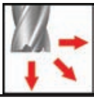
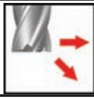
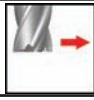
There are many different operations that come under the term “end milling”.
For each operation, there is an optimal cutter type. Three parameters influence the choice of the type of cutter:

- Direction of use of the cutter
- MRR (Material Removal Rate)
- Application

Milling

DIRECTION OF USE OF THE CUTTER

We can split the range of the cutters in relationship to the possible working directions to the workpiece surface. There are three different types:

3 Directions	2 Directions	1 Direction
		

Please note that the axial direction is possible only with centre cutting end mills.

MRR (MATERIAL REMOVAL RATE) Q

We can calculate material removal rate Q as the volume of material removed divided by the time taken to cut. The volume removed is the initial volume of the workpiece minus the final volume. The cutting time is the time needed for the tool to move through the length of the workpiece. This parameter strongly influences the finishing grade of the workpiece.

$$Q = \frac{a_p * a_e * v_f}{1000}$$

Q = MRR (cm^3/min)

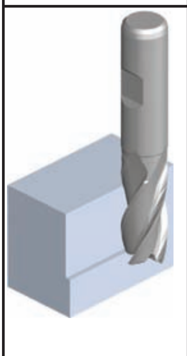
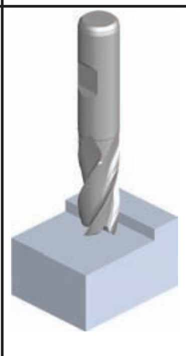
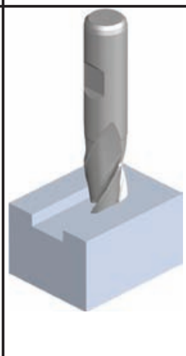
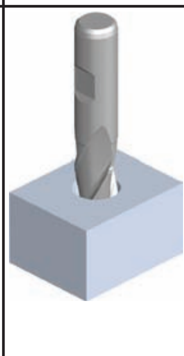
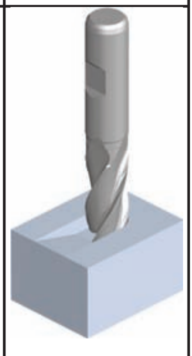
a_p = axial depth (mm)

a_e = radial depth (mm)

v_f = feed rate mm/min

APPLICATIONS

The MRR and the applications are strongly related. For each different application we have a different MRR that increases with the engagement section of the cutter on the workpiece. The recent Dormer Catalogue was produced with simple icons that show the different applications.

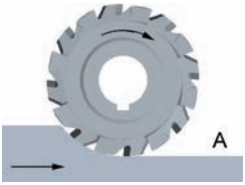
Side Milling	Face Milling	Slot Milling	Plunge Milling	Ramping
				
The radial depth of cut should be less than 0.25 of the diameter of the end mill.	The radial depth of cut should be no more than 0.9 of the diameter, axial depth of cut less than 0.1 of the diameter.	Machining of a slot for keyways. The radial depth of cut is equal to the diameter on the end mill.	It is possible to drill the work-piece with an end mill only with the cutting centre. In this operation the feed has to be halved.	Both axial and radial entering into the workpiece.

P9 Slotting

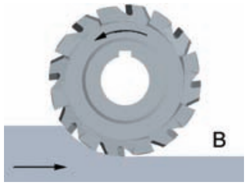
It is important to underline the capacity to make slots with P9 tolerance (please see the table on page 29 of General Information). Our cutters capable of slotting to this tolerance have the P9 icon.

Milling – Conventional vs Climb

The cutting action occurs either by conventional milling or climb milling.



Conventional Milling



Climb Milling

CONVENTIONAL MILLING

In conventional milling, also called *up milling*, the maximum chip thickness is at the end of the cut. The feed movement is opposite to the tool rotation.

Pros:

- Tooth engagement is not a function of workpiece surface characteristics.
- Contamination or scale on the surface does not affect tool life.
- The cutting process is smooth, provided that the cutter teeth are sharp.

Cons:

- The tool has the tendency to chatter.
- The workpiece has the tendency to be pulled up, thus proper clamping is important.
- Faster wear on tool than climb milling.
- Chips fall in front of the cutter – chip disposal difficult.
- Upward force tends to lift up workpiece.
- More power required due to increased friction caused by the chip beginning at the minimum width.
- Surface finish marred due to the chips being carried upward by tooth.

CLIMB MILLING

In climb milling, also called *down milling*, cutting starts with the chip at its thickest location. The feed movement and the tool rotation have the same direction.

Pros:
- The downward component of cutting forces holds the workpiece in place, particularly for slender parts.
- Easier chip disposal - chips removed behind cutter.
- Less wear - increases tool life up to 50%.
- Improved surface finish - chips less likely to be carried by the tooth.
- Less power required - cutter with high rake angle can be used.
- Climb milling exerts a downward force on workpiece - fixtures simple and less costly.

Cons:
- Because of the resulting high impact forces when the teeth engage the workpiece, this operation must have a rigid setup, and backlash must be eliminated in the table feed mechanism.
- Climb milling is not suitable for machining workpieces having surface scale, such as hot-worked metals, forgings and castings. The scale is hard and abrasive and causes excessive wear and damage to the cutter teeth, thus reducing tool life.

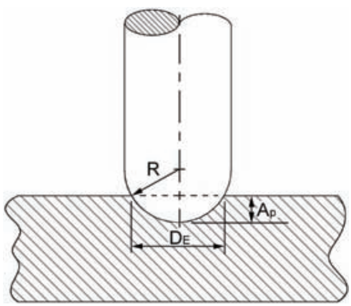
Ball Nose End Mills

A ball nose end mill, also known as a spherical end mill or ball end mill, has a semi-sphere at the tool end. Ball nose end mills are used extensively in the machining of dies, moulds, and on workpieces with complex surfaces in the automotive, aerospace, and defence industries.

Effective diameter is the main factor used in the calculation of the required spindle speed. Effective diameter is defined as the actual diameter of the cutter at the axial depth-of-cut line. The effective diameter is affected by two parameters: tool radius and axial depth of cut.

$$D_E = 2 * \sqrt{R^2 - (R - A_p)^2}$$

D_E = Effective diameter
R = Tool radius
A_p = Axial depth of cut

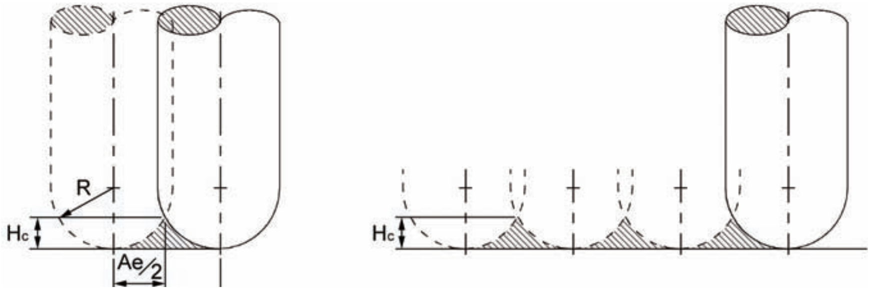


The effective diameter replaces the cutter diameter when calculating the effective cutting speed V_C for ball nose end milling. The formula becomes:

$$Vc = \frac{\pi * D_E * n}{1000}$$

Vc = Cutting speed (m/min)
D_E = Effective diameter (mm)
n = Rotation speed (rpm)

When a cutter with a non-flat end, such as a ball nose end mill, is used to cut a surface in a zigzag pattern, an uncut strip is created between the two cutting passes. The height of these undesirable strips is called cusp height.



The cusp height can be calculated from

$$Hc = R - \sqrt{R^2 - \left(\frac{Ae}{2}\right)^2}$$

or

$$Ae = 2\sqrt{R^2 - (R - Hc)^2}$$

H_c = Cusp height
R = Tool nose radius
Ae = Step over value between two cutting passes

The correlation between H_C and R_A (surface roughness) is approximately:

H_C (µm)	0,2	0,4	0,7	1,25	2,2	4	8	12,5	25	32	50	63	100
R_A (µm)	0,03	0,05	0,1	0,2	0,4	0,8	1,6	3,2	6,3	8	12,5	16	25

R_A is appr. 25 % of H_C

Milling

BALL NOSE END MILLS IN HARDENED STEEL

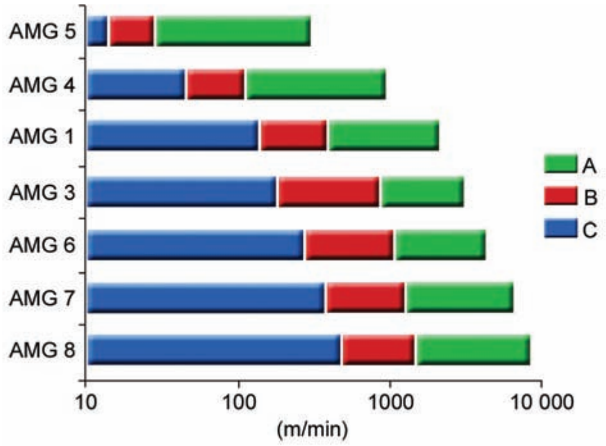
The following guidelines can be used for axial depth when machining hardened steel.

Hardness (HRC)	**Axial depth = A_P**
30 ≤ 40	0,10 x D
40 ≤ 50	0,05 x D
50 ≤ 60	0,04 x D

HIGH SPEED MACHINING

High Speed Machining (HSM) may be defined in various ways. With regard to attainable cutting speeds, it is suggested that operating at cutting speeds significantly higher than those typically utilised for a particular material may be termed HSM.

A = HSM Range, B = Transition Range, C = Normal Range



DEFINITION OF HSM

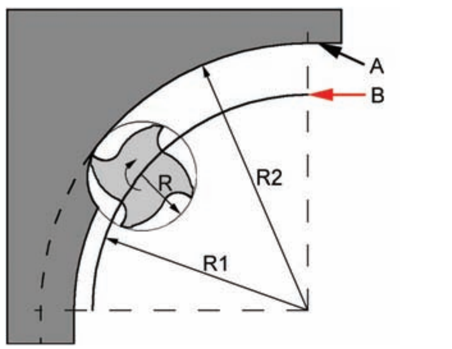
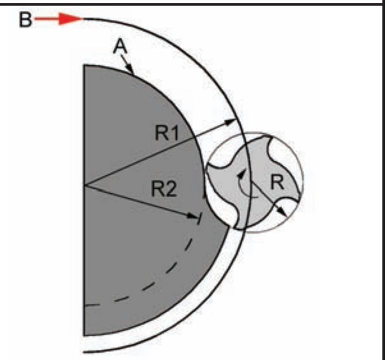
At a certain cutting speed (5-10 times higher than in conventional machining), the chip removal temperature at the cutting edge starts to decrease.

ADVANTAGES OF HSM

• Increased utilisation of the machine tool • Improved part quality • Reduced machining time • Decreased manpower • Reduced costs • Low tool temperature • Minimal tool wear at high speeds • Use of fewer tools	• Cutting forces are low (due to reduced chip load) • Low power and stiffness requirements • Smaller deflection of tools • Improved accuracy and finish obtainable • Ability to machine thin webs • Reduced process sequence time • Possibility of higher stability in cutting against chatter vibration cutting force

Milling Strategies

FEED CORRECTION WHEN MILLING INSIDE AND OUTSIDE CONTOURS

Inside contour	**Outside contour**
 	
$v_f prog = v_f * \frac{R2 - R}{R2}$	$v_f prog = v_f * \frac{R2 + R}{R2}$

A = Path followed on workpiece
B = Movement of centre point of mill
R = Mill radius
R1 = Radius for the mills movement path
R2 = Radius to be milled on workpiece

Important: Some machine control systems have automatic correction, M-function.

RAMP-TYPE FEEDING

Recommendation for maximum ramping angle (α) for HM end mills.

Number of teeth on end mill	**2**	**3**	**≥4**
For steel and cast iron	≤ 15	≤ 10	≤ 5
For aluminium, copper and plastics	≤ 30	≤ 20	≤ 10
For hardened steel	≤ 4	≤ 3	≤ 2

Milling

SPIRAL-TYPE FEEDING

Recommendation for spiral type feeding in different materials.

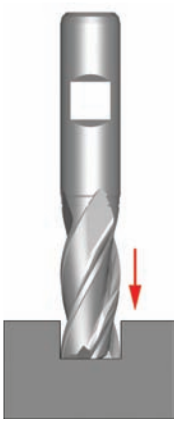
Material	Recommended ap
Steel	< 0,10 x D
Aluminium	< 0,20 x D
Hardened steel	< 0,05 X D

$$D_{bmax} = 2 * (D - R)$$

D_{bmax} = Maximum possible bore diameter
D = Mill diameter
R = Corner radius of the mill

Use maximum bore diameter (near D_{bmax}) for good chip evacuation.

AXIAL PLUNGING



In this operation, the feed rate has to be divided by the number of teeth. Please consider that it is not advisable to carry out axial plunging with an end mill with more than four teeth.

Trouble Shooting When Milling

Problem	Cause	Remedy
Breakage	Too high stock removal	Decrease feed per tooth
	Feed too fast	Slow down feed
	Flute length or overall length too long	Hold shank deeper, use shorter end mill
Wear	Workpiece material too hard	Check Catalogue or Selector for correct tool with higher grade material and/or proper coating
	Improper feed and speed	Check Catalogue or Selector for correct cutting parameters
	Poor chip evacuation	Reposition coolant lines
	Conventional milling	Climb milling
	Improper cutter helix	See recommendation in Catalogue/Selector for correct tool alternative
Chipping	Feed rate too high	Reduce feed rate
	Chattering	Reduce the RPM
	Low cutting speed	Increase the RPM
	Conventional milling	Climb milling
	Tool rigidity	Choose a shorter tool and/or place shank further up holder
	Workpiece rigidity	Hold workpiece tightly
Short Tool Life	Tough work material	Check Catalogue or Selector for correct tool alternative
	Improper cutting angle and primary relief	Change to correct cutting angle
	Cutter/workpiece friction	Use coated tool
Bad Surface finish	Feed too fast	Slow down to correct speed
	Speed too slow	Increase the speed
	Chip biting	Decrease stock removal
	Tool wear	Replace or regrind the tool
	Edge build up	Change to higher helix tool
	Chip welding	Increase coolant quantity
Workpiece inaccuracy	Tool deflection	Choose a shorter tool and/or place shank further up holder
	Insufficient number of flutes	Use a tool with more flutes
	Loose/worn tool holder	Repair or replace it
	Poor tool holder rigidity	Replace with shorter/more rigid tool holder
	Poor spindle rigidity	Use larger spindle
Chattering	Feed and speed too high	Correct feed and speed with the help of the Catalogue/Selector
	Flute or overall length too long	Hold shank deeper and use shorter end mill
	Cutting too deep	Decrease depth of cut
	Not enough rigidity (machine and holder)	Check the tool holder and change it if necessary
	Workpiece rigidity	Hold workpiece tightly

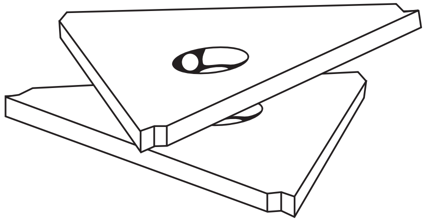
Parting Off Tools

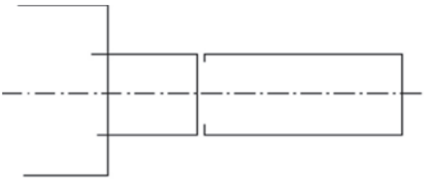
General Hints On Parting Off Tools

Dormer's parting off tools are indexable-type inserts with three edges. Manufactured in cobalt alloyed high speed steel, they are available in bright, TiN coated or TiAlN coated. TiAlN is harder than TiN and can withstand higher temperatures.
The sides of the inserts are hollow ground, which means that the clearance will be correct radially as well as axially.
A chip breaker has been shaped in the cutting surface of the edge in order to obtain the best possible type of chips when working in long-chipping materials.

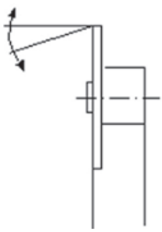
INDEXABLE INSERTS IN TWO SIZES

Indexable inserts are available in two sizes with straight edges and with 8º and 15º lead angle in both left hand and right hand versions.
Inserts for standard retaining ring size grooves, with widths of 1.1, 1.3, 1.6, 1.85 and 2.15 mm, are also available.

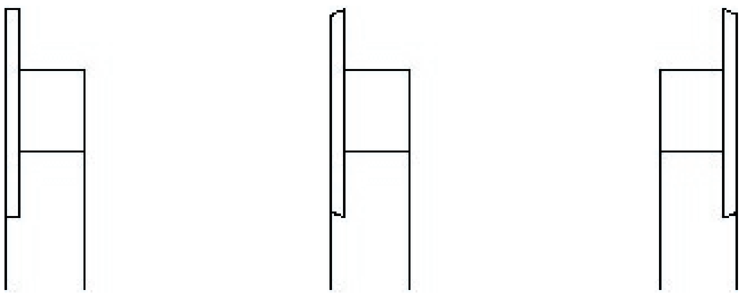




lead angle



straight right hand left hand



right hand holder left hand holder

Toolholding

General Hints On Tool Holders

INTRODUCTION

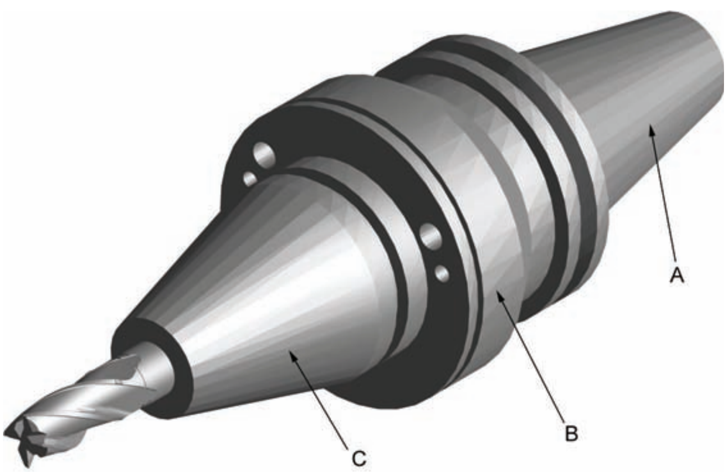
To define tool holder quality, one must first consider the function of a tool holder. A tool holder can be defined as follows:

> *A device that acts as an interchangeable interface between a machine tool spindle and a cutting tool such that the efficiency of either element is not diminished.*

To hold with this definition, four separate elements are essential:

1. **Concentricity** - The rotational axis of the machine spindle and of the cutting tool must be maintained concentrically.
2. **Holding Strength** - The cutting tool must be held securely to withstand rotation within the tool holder.
3. **Gauges** - The tool holder must be consistent. The application of proper gauges assures consistency from holder to holder.
4. **Balancing** – Tool holders must be balanced as finely as the spindles in which they are installed.

As you can see it is possible to split the holder into three separate parts: the interface with spindle (**taper, A**), the **balancing device (B)** and the part to clamp the tool (**holding mechanism, C**).



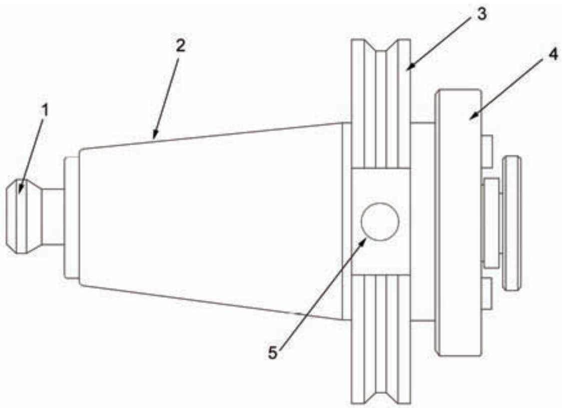
Types of Taper

- **Steep Taper** (CAT, BT, TC, ISO)
- **HSK** (Hollow Shank Taper). For more information please see the HSM (High Speed Machining) section
- **Floating holders** (only for tapping and reaming)
- **Other** (Morse Taper, Automotive Shank, Cylindrical 1835 A, Cylindrical B+E, ABS, Wohlhaupter)

Large manual machines and CNC machines use tool holders that have been precisely ground with a male taper that mates with the machine's specific female taper. There is also a way to secure the tool holder in place with a pull stud or a draw bar thread. With CNC machines, the pull stud is more popular because it allows for easier automatic tool changing.

A tool holder consists of five basic components (see figure below):

1. Pull Stud
2. Tapered Shank
3. Flange
4. Adapter
5. Opposed Slot



TAPERED SHANK

The tapered shank fits the tool holder to the spindle.
The standard defines six basic taper shank sizes including #30, #35, #40, #45, #50, and #60. Larger machines use tool holders that have larger shank taper numbers. The taper of the shank is made to 3.5 in./ft (or a ratio of 7:24).

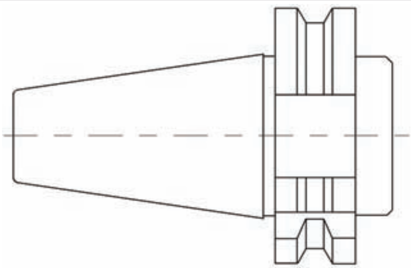
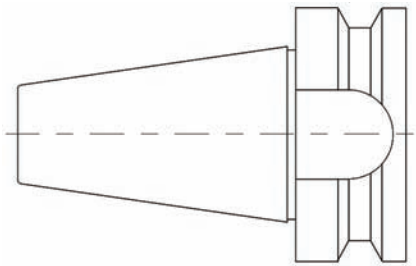
The proper Taper Shank for the Type of Machine

#60 Very large machines
#50 Medium size machines
#40 Small size machines
#30 Very small machines

FLANGE TYPE

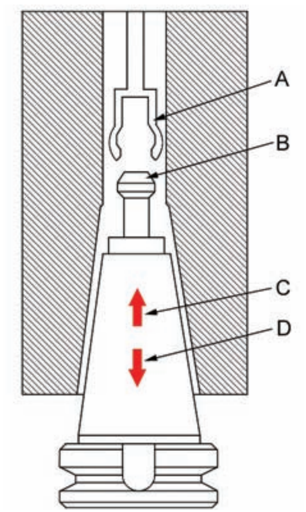
The flange allows the tool holder to be grabbed by the tool gripper or the machine spindle. There are two commonly used flange types: V-flange and BT-flange.

BT-flange holders have metric threads for the pull stud, but their adapters can be designed to accommodate a wide range of inch-dimensioned cutting tools. BT-flange holders are widely used in Japanese and European-made machining centres.

DIN 69871 V-Flange	MAS/BT-Flange
	

PULL STUD

The pull stud allows the locking drawbar (A) of the spindle to pull the tool holder firmly into the spindle and to release the tool holder automatically. Pull studs (B) are made in various styles and sizes. They are not necessarily interchangeable. Only use the pull studs that are specified by the machine tool manufacturer.

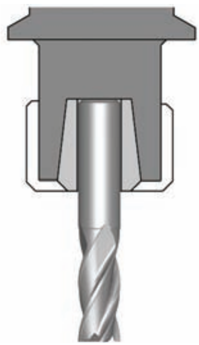
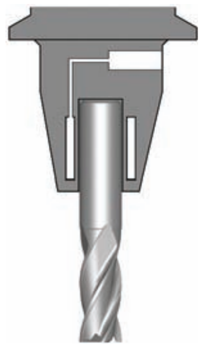
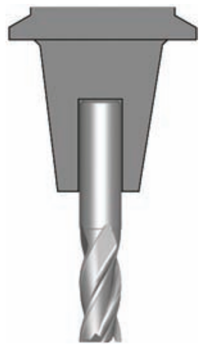


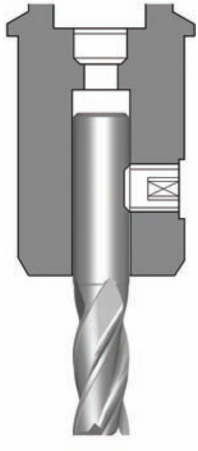
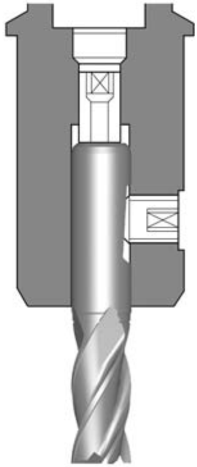
A Locking Drawbar
B Pull stud
C Clamping
D Unclamping

CLAMPING SYSTEMS

There are four different types of clamping systems:

1. Collet DIN 6388 and DIN 6499
2. Hydraulic Chuck
3. Shrink Fit
4. Weldon and Whistle Notch

Collet DIN 6388, DIN 6499	**Hydraulic Chuck**	**Shrink Fit**
		
A metal collet around the cylindrical shank is tightened with a nut.	A hydraulic tool holder uses a reservoir of oil to equalise clamping pressure around the tool. Turning a screw increases the pressure on this oil, causing an expanding sleeve to grip the tool shank.	A shrink fit tool holder works in conjunction with a specialised heater. The tool holder takes advantage of thermal expansion and contraction to clamp the tool. At normal shop temperature, the bore in which the tool locates is slightly undersize compared to the tool shank. Heating the tool holder opens up this bore, allowing the tool to be inserted. As the tool holder cools, the bore shrinks around the tool to create a concentric and rigid clamp.

Weldon, DIN 1835 B	Whistle Notch, DIN 1835 E
	
For weldon and whistle notch holders, a radial screw is in contact with the tool and holds it in place. The tool needs to have a flat ground onto the shank.	

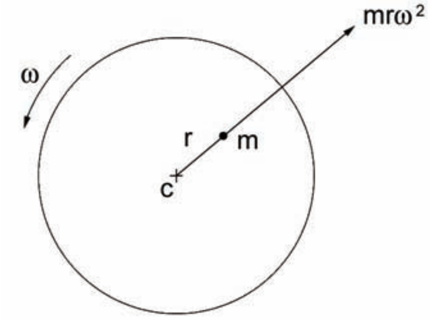
Characteristics	Collet	Weldon Whistle Notch	Hydraulic	Shrink Fit
Machining	Milling (Tapping) Drilling Reaming Boring	Milling (Tapping) Drilling Reaming Boring	Milling Tapping Drilling Reaming Boring	Milling Drilling Reaming Boring
End Mill Shank	Plain Shank HSS (DIN 1835A) Carbide (DIN 6535HA) Screwed Shank HSS (DIN 1835D)	Weldon Shank HSS (DIN 1835B) Carbide (DIN 6535HB) Whistle Notch HSS (DIN 1835E) Carbide (DIN 6535HE)	Plain Shank HSS (DIN 1835A) Carbide (DIN 6535HA)	Plain Shank HSS (DIN 1835A) Carbide (DIN 6535HA)
Runout	About 25 microns for a quality holder and collet	Around 10 microns	Around 5 microns	Around 4 microns
Rigidity	Good	Very Good	Fair	Excellent

Characteristics	Collet	Weldon Whistle Notch	Hydraulic	Shrink Fit
Balance	Different types of collets exist in relationship to the concentricity	Asymmetric design creates unbalance, but tool holder can be manufactured to remove weight where appropriate to compensate for this	Asymmetric design creates unbalance, but tool holder can be manufactured to remove weight where appropriate to compensate for this	Best – With no screws or other asymmetrical features, holder is inherently well-balanced
Vibration	No advantage	No advantage	The fluid reservoir may offer some capacity to damp vibration	No advantage
Ease of use	Low – the accuracy is dependent on the operator	Good	Better – the accuracy is consistent but clamping mechanism is easy to damage	High – low skill operators can use effectively
Cost	Normal	Normal	More expensive	Holders are cheap, but the need for a heater means there is a high start-up investment

Balancing Of The System Tool holder/Cutter

Unbalance occurs when the mass centre and the geometric centre do not coincide with each other.

Unbalance amount is expressed as



$$U = m * r$$

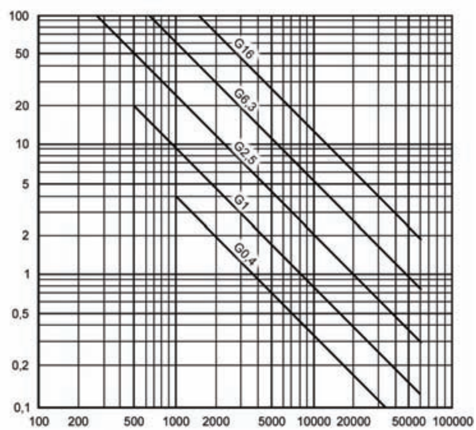
$$e = \frac{U}{M} = \frac{m * r}{M}$$

$$G = \frac{e * 2 * \pi * n}{60.000}$$

Quantity	Symbol	Unit
Specific permissible unbalance	e	gmm/Kg
Balance Grade Code	G	mm/s
Unbalance mass	m	g
Constant angular speed	ω	rad/s
Rotor mass	M	Kg
Distance from unbalance mass to centreline	r	mm
Total permissible unbalance	U	gmm
Rotation speed	n	rpm

Balance Quality Based on Standardised Tables

G Quality Grade (the inclined lines in the diagram below) relates max. surface rotational speed (X-axis) to the specific permissible unbalance *e* (Y-axis).



For a specific grade, as the rotational speed of the cutter increases, the permissible unbalance *e* decreases.

Balance quality grades are separated from each other by a factor 2.5.
0,4x2,5=1 x2,5=2,5 x2,5=6,25 x2,5=15,625.
Some standards about this have been produced.

ISO 1940-1:2003 gives specifications for rotors in a constant (rigid) state. It specifies balance tolerances, the necessary number of correction planes, and methods for verifying the residual unbalance.

Recommendations are also given concerning the balance quality requirements for rotors in a constant (rigid) state, according to their machinery type and maximum surface speed. These recommendations are based on worldwide experience.

ISO 1940-1:2003 is also intended to facilitate the relationship between the manufacturer and user of rotating machines, by stating acceptance criteria for the verification of residual unbalance.

Detailed consideration of errors associated with balancing and verification of residual unbalance are given in ISO 1940-2.

Usually the balancing of the tool holder is carried out without the tool and is verified with it.

It is necessary to know what “G” rating the tool holder is balanced to and at what speed (rpm). These two components define the maximum permissible vibration displacement of the centre of mass. The higher the speed, the smaller the vibration displacement must be for a given “G” grade.

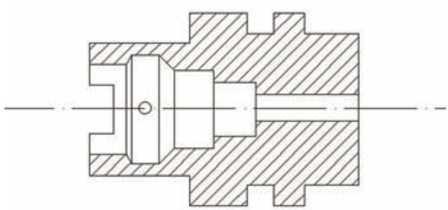
Some tool holders are advertised as “production balanced tool holders” for speeds up to 20,000 rpm without being actually specified to the ISO 1940 tolerance grade. When tested, many of these tool holders are found to fail to meet quality G6.3 standards, much less the more stringent G2.5 grade often specified for tool holders.

HSK

The German consortium of machining centre manufacturers, end users and tooling manufacturers, in conjunction with the Machine Tool Laboratory at the University of Aachen, developed the revolutionary HSK, Hollow Shank Kegel (German word for taper) tool holder connection.

In total, six separate standards were created for HSK Shanks DIN 69893 and six standards for matching Spindle Receivers DIN 69063.

DIN 69893-1. HOLLOW TAPER SHANKS – HSK WITH FLAT CONTACT SURFACE; TYPE A AND C



Form A

- Standard type for machining centres and milling machines
- For automatic tool change
- Coolant supply through centre via coolant tube
- Drive keys at the end of HSK taper
- Hole for data carrier DIN STD 69873 in the flange.

Form C

- For transfer lines, special machines and modular tooling systems
- For manual tool change
- Coolant supply through centre
- Drive keys at the end of HSK taper
- Since all Form A holders are equipped with side holes for manual tool change, they can also be used as Form C holders.

DIN 69893-2. HOLLOW TAPER SHANKS – HSK WITH FLAT CONTACT SURFACE; TYPE B AND D

Form B

- For machining centres, milling and turning machines
- With enlarged flange size for rigid machining
- For automatic tool change
- Coolant supply through the flange
- Drive keys at the flange
- Hole for data carrier DIN STD 69873 at the flange.

Form D
- For special machines
- With enlarged flange size for rigid machining
- For manual tool change
- Coolant supply through the flange
- Drive keys at the flange.

DIN V 69893-5. HOLLOW TAPER SHANKS – HSK WITH FLAT CONTACT SURFACE; TYPE E

Form E
- For high-speed applications
- For automatic tool change
- Coolant supply through centre via coolant tube is possible
- Without any drive keys for absolute symmetry.

DIN V 69893-6. HOLLOW TAPER SHANKS – HSK WITH FLAT CONTACT SURFACE; TYPE F

Form F
- For high-speed applications mainly in woodworking industries
- With enlarged flange size for rigid machining
- For automatic tool change
- Coolant supply through centre via coolant tube is possible
- Without any drive keys for absolute symmetry.

- DIN 69063-1. Tool Receiver for Hollow Taper Shanks - HSK Type A and C
- DIN 69063-2. Tool Receiver for Hollow Taper Shanks - HSK Type B and D
- DIN 69063-5. Tool Receiver for Hollow Taper Shanks - HSK Type E
- DIN 69063-6. Tool Receiver for Hollow Taper Shanks - HSK Type F

HSK benefits to the user include:

- High static and dynamic rigidity. Bending load is 30% to 200% greater than steep taper tool holders.
- High precision axial and radial reproducibility. The tool holder does not have the tendency to “suck in” like a steep taper holder.
- Low mass, low stroke length when tool changing.
- Centered clamping with twice the force.

Toolholding

Tapping Attachments

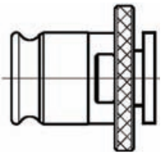
Typically a tapping attachment has to resolve the following problems:

1. Simple holding of the tap with quick tool change
2. Limit the maximum torque in relationship with the thread size
3. Compensate for the pitch errors of the machine tool

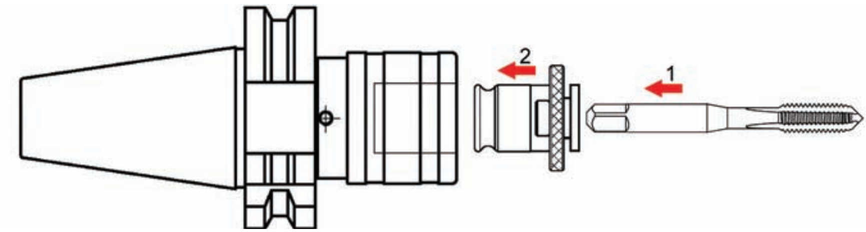
So there are different devices that supply these functions.

QUICK TOOL CHANGE DEVICES

• **Tap holder without clutch**

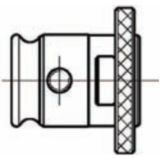


Sequence of operations

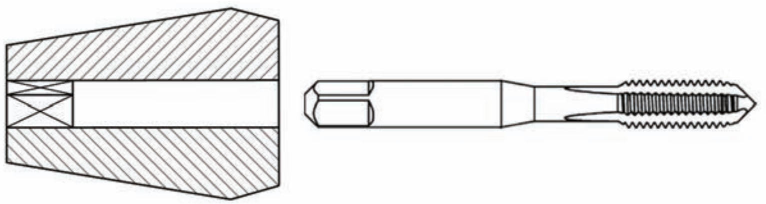


1. Insert the tap in the tap holder
2. Insert the tap holder in the end part of the tool holder

• **Tap holders without clutch and with threaded grain**



• **Tap holder collet with back square**

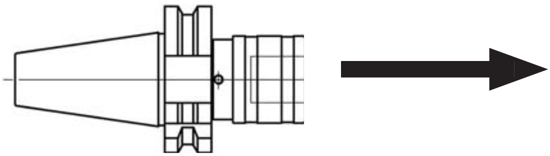


Tapping Attachments

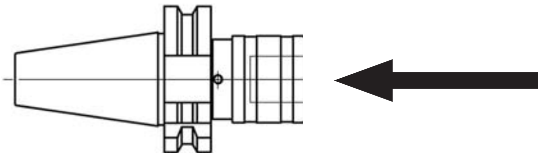
The process of tapping is a complex balance of rotational and axial movements of the tool. It is sometimes necessary to restrict the axial movements of the tool.

If the axial movement is not accurately controlled, the leading or trailing flanks of the tap may be forced to progressively "shave" one flank of the component thread, thus producing a thin and oversize thread in the component.

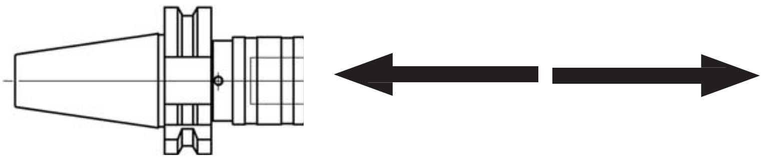
Tension – forward float capability allows the tap to progress into the component without interference from the axial feed of the machine spindle.



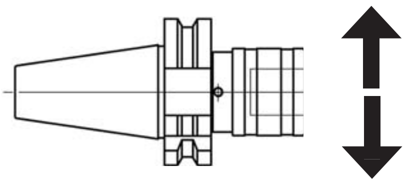
Compression – backward float capability, acts as a cushion and allows the tap to commence cutting at its owns axial feed independent of the machine spindle.



Compression / Tension – float is designed to negate any external forces during the machining operation.



Radial float – allows for slight misalignment of the machine spindle axis and hole axis prior to tapping. This is not recommended manufacturing practice and should be avoided.



Toolholding

SETTING VALUES FOR TAP HOLDERS WITH A SAFETY CLUTCH

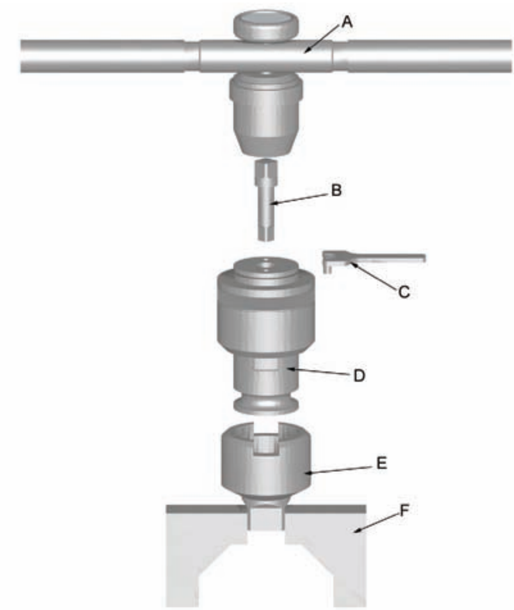
Tap holders with a safety clutch are preset to the following values dependent upon the recommended thread size.

Thread size	Torque setting (Nm)
M3	0,50
M3,5	0,8
M4	1,20
M4,5	1,60
M5	2,0
M6	4,0
M8	8,0
M10	16,0
M12	22,0
M14	36,0

Thread size	Torque setting (Nm)
M16	40,0
M18	63,0
M20	70,0
M22	80,0
M24	125,0
M30	220,0
M33	240,0
M39	320,0
M45	480,0
M48	630,0

Setting of torque on tap-holder with safety clutch

Note: Clockwise setting increases the torque
Counter clockwise setting decreases the torque



A Torque wrench
B Setting shank adaptor
C Key
D Tap holder with clutch
E Hexagonal socket shank
F Vice

Torque Calculations

$$M_d = \frac{p^2 * D * kc}{8000}$$

Md = Torque
P = Pitch

D = Nominal diameter in mm
Kc = specific cutting force

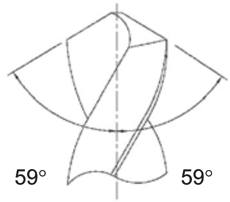
Values from this formula are valid for new cutting taps. A worn-out tap gives approximately a double torque value.
When using a forming tap the torque calculation has to be multiplied by 1.8.

Application Material Groups			***Kc*** **Specific cutting force N/mm²**
1. Steel	1.1	Magnetic soft steel	2000
	1.2	Structural steel, case carburizing steel	2100
	1.3	Plain Carbon steel	2200
	1.4	Alloy steel	2400
	1.5	Alloy steel, Hardened and tempered steel	2500
	1.6	Alloy steel, Hardened and tempered steel	2600
	1.7	Alloy steel, Heat treated	2900
	1.8	Alloy steel, Hardened & Wear resistant steel	2900
2. Stainless Steel	2.1	Free machining, Stainless Steel	2300
	2.2	Austenitic	2600
	2.3	Ferritic + Austenitic, Ferritic, Martensitic	3000
3. Cast Iron	3.1	Lamellar graphite	1600
	3.2	Lamellar graphite	1600
	3.3	Nodular graphite, Malleable Cast Iron	1700
	3.4	Nodular graphite, Malleable Cast Iron	2000
4. Titanium	4.1	Titanium, unalloyed	2000
	4.2	Titanium, alloyed	2000
	4.3	Titanium, alloyed	2300
5. Nickel	5.1	Nickel, unalloyed	1300
	5.2	Nickel, alloyed	2000
	5.3	Nickel, alloyed	2000
6. Copper	6.1	Copper	800
	6.2	β-Brass, Bronze	1000
	6.3	α-Brass	1000
	6.4	High Strength Bronze	1000
7. Aluminium Magnesium	7.1	Al, Mg, unalloyed	700
	7.2	Al alloyed, Si < 0.5%	700
	7.3	Al alloyed, Si > 0.5% < 10%	800
	7.4	Al alloyed, Si > 10% Whisker reinforced Al-alloys Mg-alloys	1000
8. Synthetic materials	8.1	Thermoplastics	400
	8.2	Thermosetting plastics	600
	8.3	Reinforced plastic materials	800
9. Hard material	9,1	Cermets (metals-ceramics)	>2800
10. Graphite	10.1	Graphite	600

DRILL POINT AND REGRINDING

To produce a perfect drill point, the following must be correct:

1. Point Angle
2. Chisel Edge angle
3. Initial clearance
4. Total clearance



Standard 118° Point Angle

Standard drills are ground with an included point angle of 118°. They are established as the most suitable for general purpose work.
If the correct initial clearance is produced and increased gradually towards the centre to produce a chisel edge angle of approximately 130°, the correct clearance will be achieved along the whole of the cutting lips.

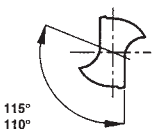
Drill Diameter mm	Initial Clearance Angle at Periphery
Up to and Including 1	21° - 27°
Over 1 to 6	12° - 18°
Over 6 to 10	10° - 14°
Over 10 to 18	8° - 12°
Over 18	6° - 12°

The two cutting lips lengths should be equal and at a similar angle to the drill axis, to provide correct balance and concentricity.

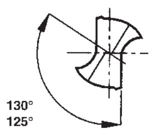
Split point geometry, DIN 1412 Type C

Due to the relatively thick web, it is necessary to grind away the flank of both cutting lips in two stages:

- Grind with required point angle (normally 118° or 135°) and a chisel edge angle of 110°–115°.
- Using the corner of the wheel, grind the secondary clearance (normally 35°– 45° to the drill axis) to produce a cutting edge at the chisel, leaving 0.1 to 0.25 mm of the original chisel edge remaining.



Chisel Edge Angle



Thinning Angle

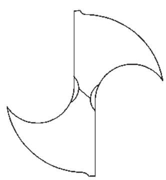
Note: If in any doubt we suggest you take an unused split point A120 drill, over 2.9mm, and use this as a pattern for regrinding worn drills.

WEB (CORE) THINNING, DIN 1412 TYPE A

Generally, drills are designed so that the thickness of the web gradually increases from the point to the run out of the flutes providing added strength and rigidity. It is not usually necessary to thin the chisel edge of a new drill. When about a third of the useful length has been ground away, the chisel edge has widened to such an extent that it needs to be thinned. If this is not done, the drilling thrust is considerably increased. Holes that are out of roundness or oversized may result, since the drill will not self-centre.

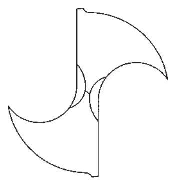
Web thinning must be carefully carried out and where possible this should be done on a point-thinning machine. If a machine is not available then a shaped grinding wheel half the width of the flute is most effective. Equal amounts of material must be ground from each side of the chisel, which must be thinned, to approximately 10% of the drill diameter.

Correct Web Thinning



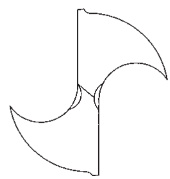
Note how thinning is blended evenly into the flutes. An equal amount of material has been removed from each side and the chisel edge has not been excessively reduced.

Excessive Web Thinning



An equal but excessive amount of material had been removed from the chisel edge. This has weakened the drill point and can cause the web to split.

Uneven Web Thinning



Excessive material has been removed on one side of the chisel edge causing an unbalanced drill. The result will be oversized holes and may cause the drill to break.

TOLERANCE LIMITS ON THE CUTTING DIAMETER OF STANDARD DRILLS

Dormer produces standard drills in accordance with the appropriate national or international standards.
The standard tolerance on diameter measured across the outer corner, immediately adjacent to the point, is h8 in accordance with British Standard, ISO and DIN as specified below.

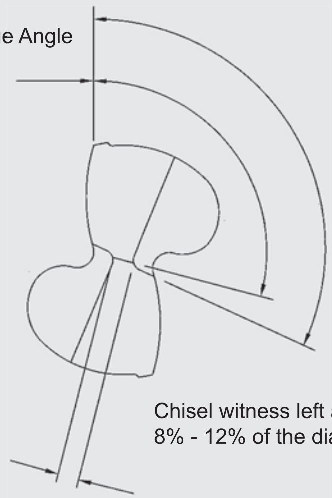
MILLIMETRES			
Diameter		Tolerance Limit	
Over	Up to and including	High +	Low -
	3	0	0.014
3	6	0	0.018
6	10	0	0.022
10	18	0	0.027
18	30	0	0.033
30	50	0	0.039
50	80	0	0.046

INCHES			
Diameter		Tolerance Limit	
Over	Up to and including	High +	Low -
	0.1181	0	0.0006
0.1181	0.2362	0	0.0007
0.2362	0.3937	0	0.0009
0.3937	0.7087	0	0.0011
0.7087	1.1811	0	0.0013
1.1811	1.9685	0	0.0015
1.9585	3.1496	0	0.0018

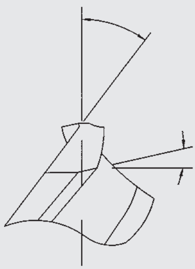
Regrinding

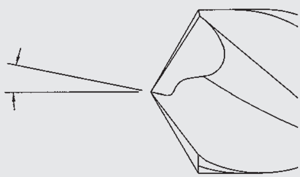
HEAVY DUTY DRILLS // PFX

POINT REGRINDING DETAILS



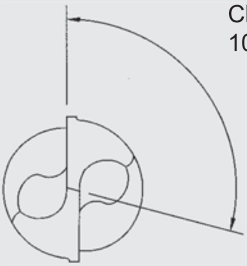
WEB THINNING DETAILS





EXTRA LONG SERIES DRILLS // PFX

POINT REGRINDING DETAILS

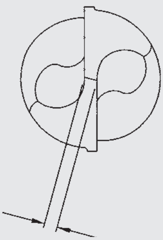


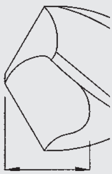
Chisel Edge Angle
105° +/- 3°

Point Angle
130° +/- 3°

Lip Clearance Angle
Sizes up to 6.00mm Inc.
12°+/- 2°
Sizes over 6.00:10 +/- 2°

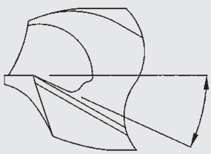
WEB THINNING DETAILS





Web thinning length
50%-75% Nominal diameter
The thinning must extend to the outer corner

Width of Chisel Edge
8% - 12% Nominal diameter



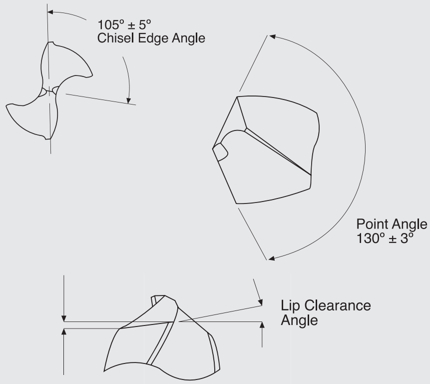
Thinning Rake Angle
20° - 30° Positive

Regrinding

POINT REGRINDING INFORMATION

A510 // A520

Chisel Centre Error		
0.05 TIV, MAX (mm)		
Lip Height Variation (mm)		
Sizes	3.0 -13.0	0.025 Max
Over	13.0 -14.0	0.050 Max
Lip Clearance Angle		
Sizes	3.0 - 6.0 inc.	11˚ - 15˚
Over	6.0 - 10.0 inc.	10˚ - 14˚
Over	10.0 - 13.0 inc.	8˚ - 12˚
Over	13.0 - 14.0 inc.	6˚ - 10˚
Total Point Clearance (mm)		
(should be continuous clearance)		
Sizes	3.0	0.20 - 0.40
Over	3.0 - 4.0 inc.	0.25 - 0.45
Over	4.0 - 6.0 inc.	0.25 - 0.50
Over	6.0 - 8.0 inc.	0.30 - 0.55
Over	8.0 - 10.0 inc.	0.35 - 0.60
Over	10.0 - 13.0 inc.	0.40 - 0.80
Over	13.0 - 14.0 inc.	0.50 - 1.20



Total Point Clearance (continuous clearance)

A551 // A552 // A553 // A554

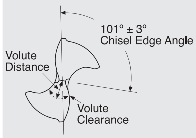
Chisel Centre Error		
0.05 TIV, MAX (mm)		
Lip Height Variation (mm)		
Sizes	5.0 - 13.0 inc.	0.025 Max
Over	13.0 - 20.0 inc.	0.050 Max

Volute Clearance

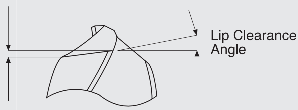
The value of the Volute Clearance should be:
50% - 75% of the resultant total point clearance.
(ie. resultant total point clearance; 0.60mm.
volute clearance: 0.30mm - 0.45mm)

The position for measuring the Volute Clearance should be as stated in the table opposite and measured from the centre as shown.

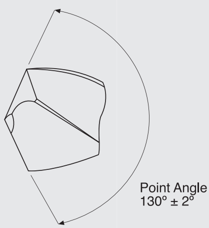
Lip Clearance Angle		
Sizes	3.0 - 6.0 inc.	11˚ - 15˚
Over	6.0 - 10.0 inc.	10˚ - 14˚
Over	10.0 - 13.0 inc.	8˚ - 12˚
Over	13.0 - 30.0 inc.	6˚ - 10˚
Total Point Clearance (mm)		
(should be continuous clearance)		
Sizes	5.0 - 8.0 inc.	0.20 - 0.45
Over	8.0 - 10.0 inc.	0.25 - 0.45
Over	10.0 - 13.0 inc.	0.40 - 0.60
Over	13.0 - 20.0 inc.	0.50 - 0.70
Over	20.0 - 30.0 inc.	0.70 - 1.10



Nominal Diameter mm	Volute Distance mm
5.00 - 6.00	0.80 - 1.00
6.01 - 8.00	0.65 - 1.15
8.01 - 10.00	0.90 - 1.40
10.01 - 12.00	1.15 - 1.65
12.01 - 14.00	1.50 - 2.00
14.01 - 16.00	1.75 - 2.25
16.01 - 18.00	2.00 - 2.50
18.01 - 20.00	2.25 - 2.75
20.01 - 25.00	3.10 - 3.60
25.01 - 30.00	4.00 - 4.50

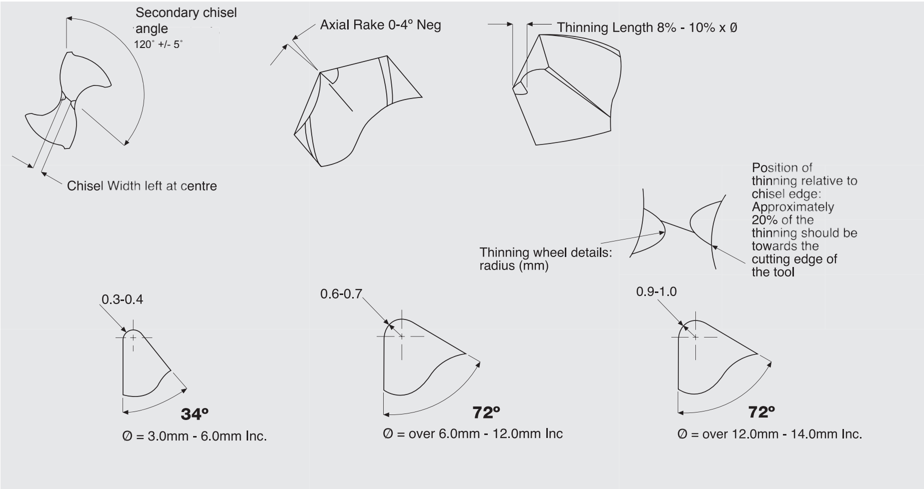


Total Point Clearance (continuous clearance)

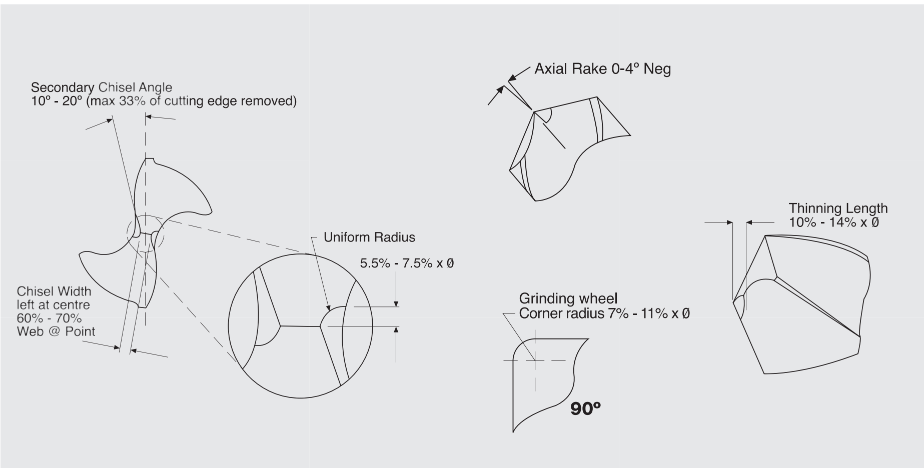


Web Thinning Details

A510 // A520



A551 // A552 // A553 // A554



CDX REGRINDING INFORMATION

Recommendations

Follow these recommendations with the drawing of the CDX point geometry as a reference.

•Regrind so that the coating in the flutes and on the lands is undamaged.

•The variations in web thinning should be <0.025mm.

•Use a diamond grinding wheel and a liberal supply of cutting compound.

•Use a stable grinding machine.

•If in any doubt we suggest you take an unused CDX drill and use this as a pattern for regrinding worn drills.

Avoid

Do not use the drills for too long before regrinding. Do not grind by hand.

Procedure

In order to restore the drills to a condition for best possible results after regrinding, we recommend the following three-stage process:-

I. GRINDING PRIMARY AND SECONDARY CLEARANCES

(SEE DRAWINGS OPPOSITE)

1. Set the machine to a 130° point angle.
2. Set the secondary clearance angle to 17-25°.
3. Grind the secondary clearance angle until it is in a position beyond the centre line on the cutting edge.
4. Set the primary clearance angle to 6-10°
5. Grind until the junction of primary and secondary clearances is over the centre line of the tool to the heel side to result in a chisel angle of 102-110°.

Point angle 130° +/- 2°

Clearance	6-10°, ground over the center line shown in fig.1	
	Diameter mm	Dimensions A and B mm
	3.0 - 8.0	0.10 - 0.25
	8.1 - 12.0	0.15 - 0.30
	12.1 - 16.0	0.20 - 0.35
	16.1 - 20.0	0.25 - 0.45

II. WEB THINNING

1. Use a 60° diamond-grinding wheel with a corner radius. We recommend the following

Diameter mm	Grinding Wheel radius	Length of web thinning
3.0	0.25	0.50 - 0.80
4.0	0.25	0.60 - 0.90
5.0	0.25	0.70 - 1.00
6.0	0.25	0.95 - 1.25
7.0	0.35	1.10 - 1.50
8.0	0.35	1.20 - 1.60
9.0	0.55	1.30 - 1.70
10.0	0.55	1.40 - 1.80
11.0	0.55	1.40 - 2.00
12.0 - 13.0	0.55	1.50 - 2.10
14.0 - 15.0	0.70	1.70 - 2.30
16.0	0.70	1.95 - 2.55
17.0 - 18.0	0.90	2.10 - 2.90

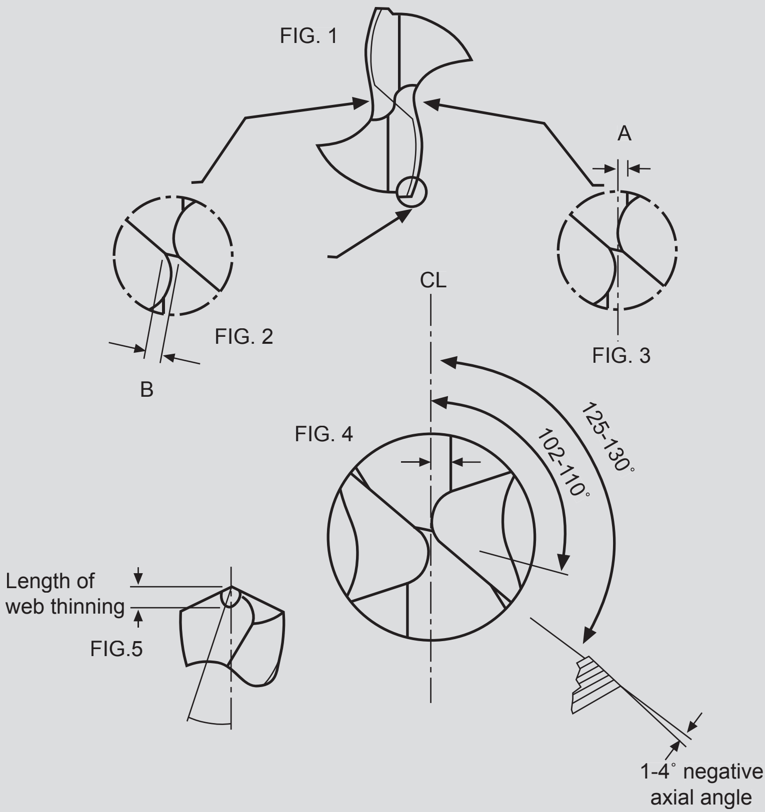
2. Set the machine so that the axial rake at the secondary cutting edge by thinning is between -1° and -4°.
3. For best results, grind until the amount specified in the regrinding figure (dimensions A and B) is achieved.
4. The web thinning must never go beyond the centre line, (Fig.3)

III. NEGATIVE RAKE

A negative rake edge with an angle of 20-35° to drill axis along the full cutting lips, with a width acc. to the table below in the direction of grinding, up to the secondary cutting lips is recommended. This should be produced by grinding or with a diamond finishing stick, to produce a smooth surface finish.

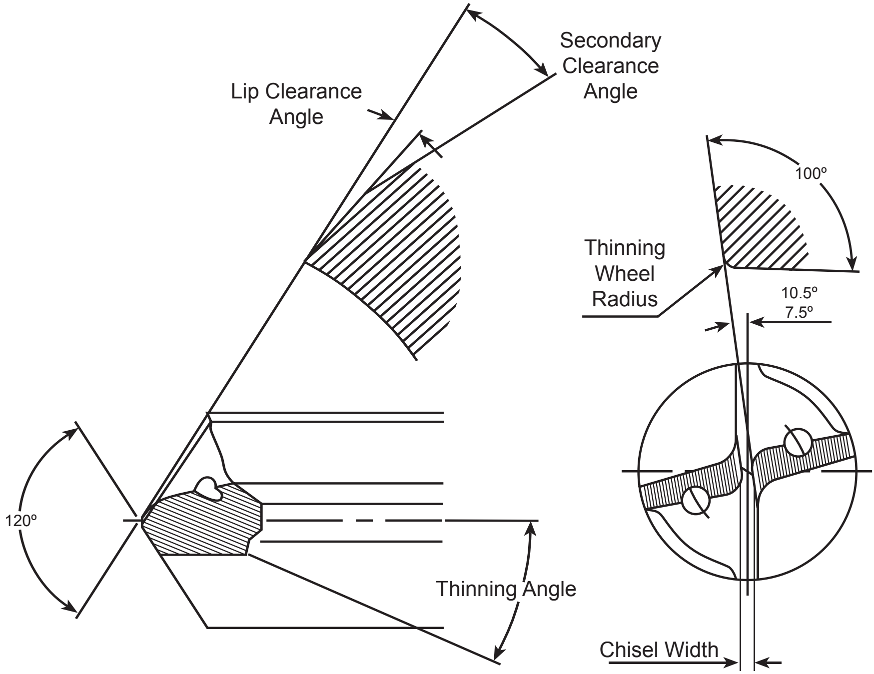
Negative rake on cutting lip, dim.F Diameter mm	20-35 degree negative Width of rake mm(Axial)
3.0 - 6.0	0.03 - 0.07
6.1 - 10.0	0.03 - 0.10
10.1 - 14.0	0.03 - 0.12
14.1 - 20.0	0.03 - 0.15

Regrinding of CDX Drills



A = Primary clearance over centre
B = Width of chisel edge
C = Secondary cutting edge angle
D = Chisel edge angle
E = Axial rake of secondary cutting edge
F = Negative rake edge

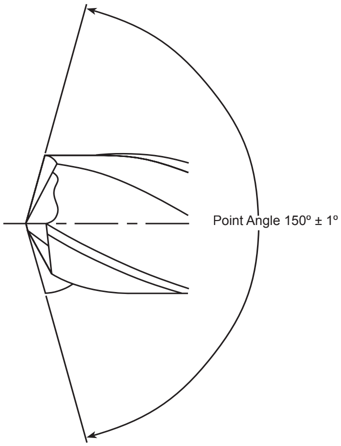
R210/R220 Regrinding Information

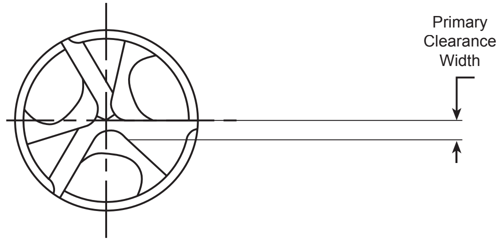


Diameter	Lip Clearance Angle	Secondary Clearance Angle	Chisel Width	Thinning Wheel Radius	Thinning Angle
5	13°	25°	0.3	0.5	24° - 26°
6	12°	25°	0.36	0.6	24° - 26°
8	11°	25°	0.48	0.8	24° - 26°
10	10°	25°	0.6	1.0	24° - 26°
12	9°	25°	0.72	1.2	24° - 26°
14	8°	25°	0.84	1.4	24° - 26°
16	7°	25°	0.96	1.6	24° - 26°
18	7°	25°	1.08	1.8	24° - 26°
20	6°	25°	1.2	2.0	24° - 26°

R325 REGRINDING INFORMATION

POINT GRINDING DETAILS

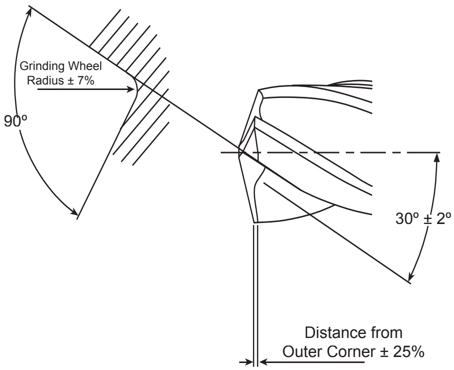




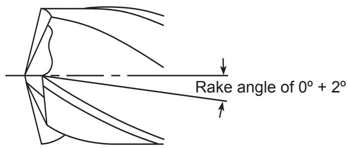
Drill Diameter	Primary Clearance Angle	Secondary Clearance Angle
3.0 - 4.2	10° +/- 1°	N/A
4.5 - 7.8	9° +/- 1°	16° +/- 2°
8.0 - 9.8	8° +/- 1°	16° +/- 2°
10.0 - 15.8	7° +/- 1°	16° +/- 2°
16.0	6° +/- 1°	16° +/- 2°

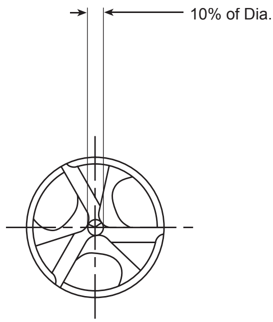
Drill Diameter	Primary Clearance Width +/- 14%
3.0 - 4.0	N/A
4.2 - 4.8	0.31
5.0 - 5.8	0.35
6.0 - 7.8	0.42
8.0 - 9.8	0.56
10.0 - 11.8	0.70
12.0 - 13.8	0.84
14.0 - 15.8	0.98
16.0	1.12

POINT THINNING DETAILS



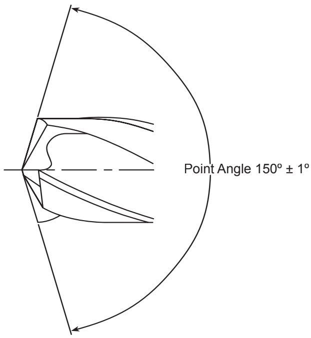
Drill Diameter	Distance from Outer Corner	Grinding Wheel Radius
3.0 - 3.8	0.14	0.40
4.0 - 4.8	0.17	0.55
5.0 - 5.8	0.20	0.65
6.0 - 7.8	0.23	0.75
8.0 - 9.8	0.29	1.05
10.0 - 11.8	0.35	1.30
12.0 - 13.8	0.39	1.55
14.0 - 15.8	0.43	1.85
16.0	0.49	2.05

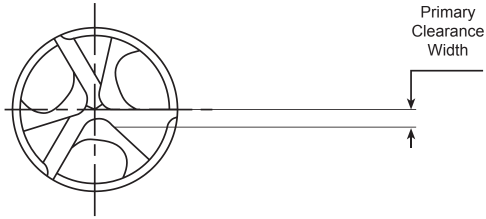




R330 REGRINDING INFORMATION

POINT GRINDING DETAILS

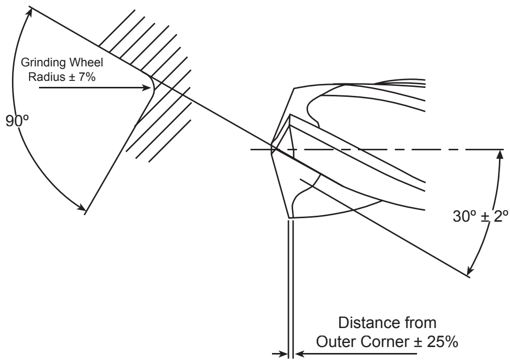




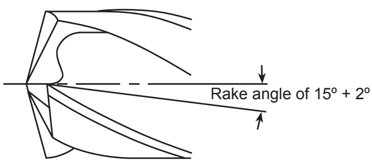
Drill Diameter	Primary Clearance Angle	Secondary Clearance Angle
3.0 - 4.2	10˚ +/- 1˚	N/A
4.5 - 7.8	9˚ +/- 1˚	16˚ +/- 2˚
8.0 - 9.8	8˚ +/- 1˚	16˚ +/- 2˚
10.0 - 15.8	7˚ +/- 1˚	16˚ +/- 2˚
16.0	6˚ +/- 1˚	16˚ +/- 2˚

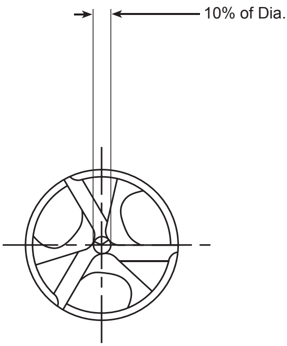
Drill Diameter	Primary Clearance Width +/- 14%
3.0 - 4.0	N/A
4.2 - 4.8	0.31
5.0 - 5.8	0.35
6.0 - 7.8	0.42
8.0 - 9.8	0.56
10.0 - 11.8	0.70
12.0 - 13.8	0.84
14.0 - 15.8	0.98
16.0	1.12

POINT THINNING DETAILS



Drill Diameter	Distance from Outer Corner	Grinding Wheel Radius
3.0 - 3.8	0.27	0.40
4.0 - 4.8	0.33	0.55
5.0 - 5.8	0.39	0.65
6.0 - 7.8	0.44	0.75
8.0 - 9.8	0.55	1.05
10.0 - 11.8	0.65	1.30
12.0 - 13.8	0.75	1.55
14.0 - 15.8	0.84	1.85
16.0	0.93	2.05





Reamers are precision tools, which have to be very exact with regard to dimensional accuracy and geometrical style. For this reason the tools are manufactured between centres.
Before regrinding, the tools have to be checked between centres for concentricity. The centres have to be undamaged. If the tools are found to be eccentric, they have to be straightened at the soft part of the shank.
It is very important to regrind the reamer before too much wear or damage has occurred. When the cutting chamfer is dull, the cutting edges and faces are stressed too much and the circular grinding chamfer gets used up. This means that even after resharpening the holes do not have exact size. When regrinding without coolant, avoid intense heating, otherwise HSS cutting edges burn out and carbide tips get grinding cracks.

REGRINDING OF CUTTING CHAMFER
Hand reamers, machine reamers and shell reamers for cylindrical bore holes are resharpened at the cutting chamfer. **The relief angle α of the first cut should be 5 - 8°.** It can be reached by changing the height of the support finger. With tool grinding machines with a rotating grinding spindle, the support finger has to be adjusted to the height of the centers and the grinding spindle has to be rotated to the required relief.

SUITABLE GRINDING WHEELS:

HSS tools:	potters wheel, special fused alumna, grain size 60, grade K - L
Carbide tools:	diamond wheel, resinoid bond, concentration 75, grade D, grain size 90

REGRINDING FACE
On reamers used for cylindrical bore holes it is sometimes also necessary to regrind the face. During this operation, a part of the circular land has to be preserved. The tool back rake angle should not be changed. Press the reamer slightly by hand towards the grinding wheel and move the reamers to left and right direction. If the pressure of the tool towards the grinding wheel is too high, the grinding wheel becomes deformed. This leads to a rounded minor cutting edge. The tool back rake is 3 - 6° positive.

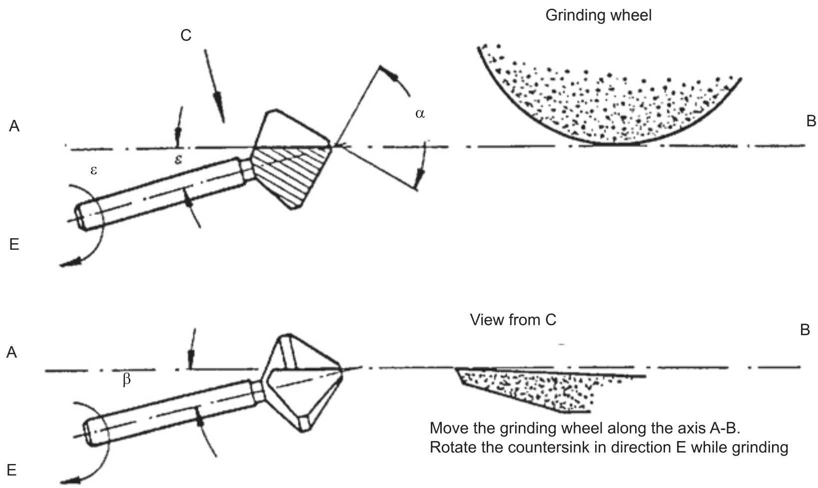
SUITABLE GRINDING WHEELS:
Diamond wheel, resinoid bond, grade D, grain size 30

At relief grind of the circular land, the support finger has to be mounted at the column of the machine. The support finger and this part of the grinding wheel, which cuts, have to be at the same point so a symmetrical land can be produced. The reamer clamped between centres is held by hand. Press the face of the tool slightly against the support finger. By moving the supporting table to the left and right direction, the support finger will work as a guide. The relief angle can be adjusted by putting the finger higher or lower. Spiral fluted reamers can be ground the same way.

SUITABLE GRINDING WHEELS:
Diamond wheel, resinoid bond, concentration 75, grade D, grain size 90

NOMINAL ∅	CHAMFER WIDTH	RELIEF ANGLE
2	0,15 – 0,20	≈ 25°
4		16 – 18 °
6		12 – 14 °
10		11 – 13 °
10 – 20	0,15 – 0,25	10 – 20 °
> 20	0,20 – 0,30	8 – 10 °

REGRINDING OF 3 FLUTE COUNTERSINKS



Shank	Diameter from up to and Incl.	α	ß	ε
Cyl.	6.3 - 25.0	60°	10.5°	22°
MK	16.0 - 31.5			
	40.0 - 80.0			12°
Cyl.	4.3 - 6.3	90°	12.5°	29°
	7.0 - 13.4			
	15.0 - 31.0			
MK	15.0 - 31.0			
	34.0 - 37.0		14°	15°
	40.0 - 80.0			

Regrinding

A worn tap has a tendency to chip or break, cut oversize or produce rough, poor quality threads. It requires higher power on the spindle and needs more time for the tapping cycle.

Generally, a tap needs to be reground when the roundness on the cutting edge has the same or even greater thickness than the chip. The remedy is to regrind, and it is cost effective mainly for big sizes > M12.

Regrinding of taps should, if possible, take place on a tap grinder and not be done by hand. The most important thing is to regrind with the original lead angle and to keep the same rake angle and relief for each flute. This can only be achieved using a tap grinding machine.

Wear on a tap extends on the cutting edge and on the outer diameter, but generally the bigger part is on the chamfer length. This portion removes the majority of material and withstands the higher loads during tapping. Generally it is sufficient to regrind just this part, removing the worn portion, in order to resharpen the tap.

The chamfer and accompanying relief on the top of the thread should be identical on all of the lands of the tap. If the chamfer is uneven, the result will be holes much larger than the tap size, torn and misshapen threads, uneven wear and eventual breakage of the taps.

When the edges of the thread begin to dull or become nicked, the flutes should be reground. A flute grinder providing accurate indexing of the cutting edges should be used. Unless the indexing is accurate, the tap is liable to break or cut oversize.

Regrinding of the flutes can also be undertaken in cases where there is a lack of suitable equipment for grinding the chamfer.

Note when regrinding:

- Grind the tap between centres and check it does not have radial run out.
- Grind the chamfer land following the original relief, using the outer face of a cup wheel or a disk wheel (see left fig. on next page).
- Grind the chamfer with a wheel having the chamfer angle β or incline the tap by the same amount if using a flat wheel (see left fig. on next page).
- The equal division of the cutting edges must be maintained.
- Grind the flute using a disk wheel, sharpened with the same flute profile of the tap (see right fig. on next page).
- The correct rake angle must be maintained – see rake angles table.
- The diameter of the tap will be reduced.
- The lands will be reduced and therefore will be weaker.
- Avoid forming burrs on the flanks of the thread form.

The chamfer angle (β) must be calculated in order to keep the same chamfer length as the original tap.

When regrinding the flute, the flank wheel is displaced in relation to the tap axis: the distance (X) is linked to rake angle (μ), see figure below. In this operation, it is very important to ensure a correct indexing on the machine, in order to keep the same wheel position for each flute.

Never regrind a damaged tap or a tap with build-up on the thread flanks.

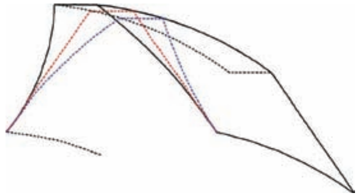
Rake angles (μ) for taps

Material to be tapped	Rake angle (approx.) in degrees
Cast Iron	4-6
Malleable Cast Iron	5-10
Steel, up to 500 N/mm^2 Tensile Strength	12-15
Steel, up to 1000 N/mm^2 Tensile Strength	10-12
Steel, over 1100 N/mm^2 Tensile Strength	7-10
Stainless Steel	8-12
Brass, cast	0-5
Aluminium	15-25

Chamfer Regrinding	Flute Regrinding
	
	Offset calculation $x = \frac{d * \sin(u)}{2}$

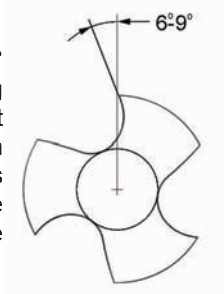
Dormer thread milling cutters are tooth form corrected (head depth, tooth depth) and thread angle corrected. The cutters are form relieved to allow several regrinding operations with no loss of profile.

With the tooth form correction, the profile distortion is counteracted through the movement of the screw. Here, the relationships between the diameter and the pitch are decisive factors. The correction of the thread angle is dependent upon rake angle, relief and spiral angle of the thread milling cutter.



Distortion of the flank with form relieved thread milling cutters with varying rake angles

As the thread milling cutters are only reground on the cutting face, the following requirements are imperative for the regrind:



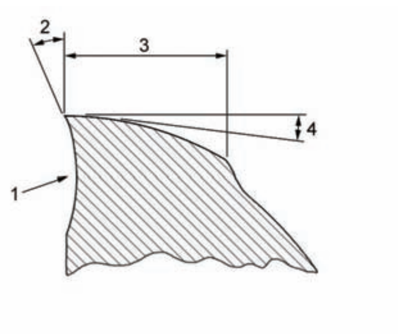
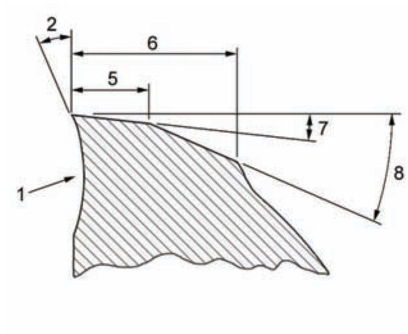
- Firstly the rake angle must be strictly adhered to from 6° to 9° - measured at the profile height. It must be noted that the cutting face up to the area under the “web” of the profile is as straight as possible. Deviations in the rake angle result in a change in the thread angle, which means that the thread milling cutter is no longer able to produce a thread to standard. (Moreover in the case of metric threads, the profile height is around 60% of the pitch.)
- The spiral angle must also be considered. Thread milling cutters in the Dormer standard range have a spiral angle of 10°. On special tools this can be different. Please contact Dormer for more information.

For each tool there is an economically optimum time for regrinding. This time depends on the land wear, which in turn depends on the period of use and on the cutting parameters. The period of use is often determined by size of the workpiece.

Cutter wear is usually seen on the primary clearance. It causes the component surface finish to worsen and cutter vibration to increase.

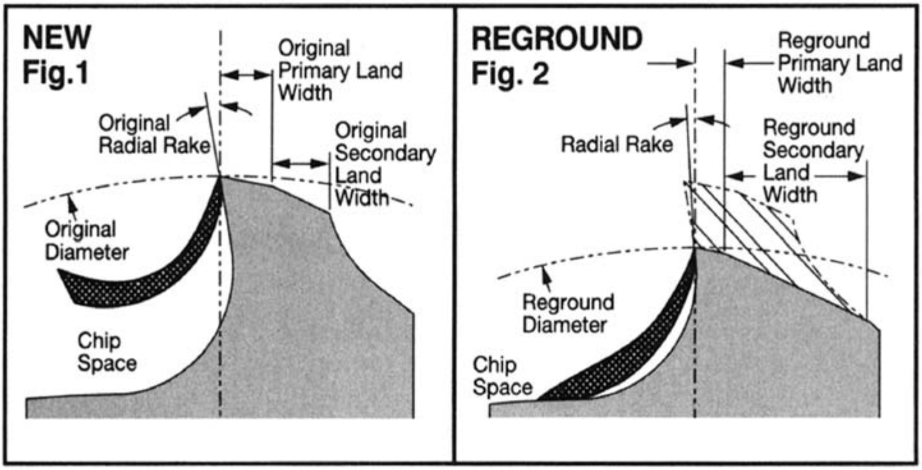
A useful aid in this case is a power input gauge on the milling machine. If the instrument registers a rise, this generally indicates increased tool wear. Exceeding the specified permissible land wear results in a rapid increase in the cutting force due to the dull cutting edge, which, if left unchecked, may lead to tool fracture.

There are two types of cutter profiles:

Archimede profile	**Double-end profile**
	
1. Undercut face 2. Rake angle 3. Width of land 4. Relief angle	5. Width of primary relief land 6. Width of secondary relief land 7. Primary relief angle 8. Secondary relief angle
With this profile it is possible to regrind only the undercut face. If the land is too badly damaged, then it is also necessary to regrind the land by creating a double-end profile (see right column).	With the double-end profile the regrinding will begin with the primary land and will continue with the secondary land.

RESULTS OF REGRINDING

Reduction in Diameter
A loss in diameter occurs when grinding the periphery of the primary land. This progressively impacts the end mill's deflection capacity when under load. Compare Figs. 1 and 2.



Reduction In Radial Rake Angle
An end mill must possess a rake angle that is suitable for the material being machined. After each regrind there is not only a reduction in diameter, but also a subsequent reduction in the radial rake angle. This, together with the corresponding, if slight change, in helix angle significantly affects the efficiency of the end mill. Compare Figs. 1 and 2. Face rake angles can be re-established by regrinding the flute face of the end mill

Increase In Secondary Land Width
The secondary land width increases substantially as a result of regrinding, which in turn increases regrinding time and cost. Compare Figs. 1 and 2.

Reduction In Flute Depth
As a consequence of reducing diameter, there is a corresponding reduction in the flute depth. Because of the subsequent impact on chip evacuation capabilities it can force the utilisation of feed rates that would be considered far less efficient. Compare Figs. 1 and 2.

Please contact your local Dormer sales company for information on our Regrinding Service.