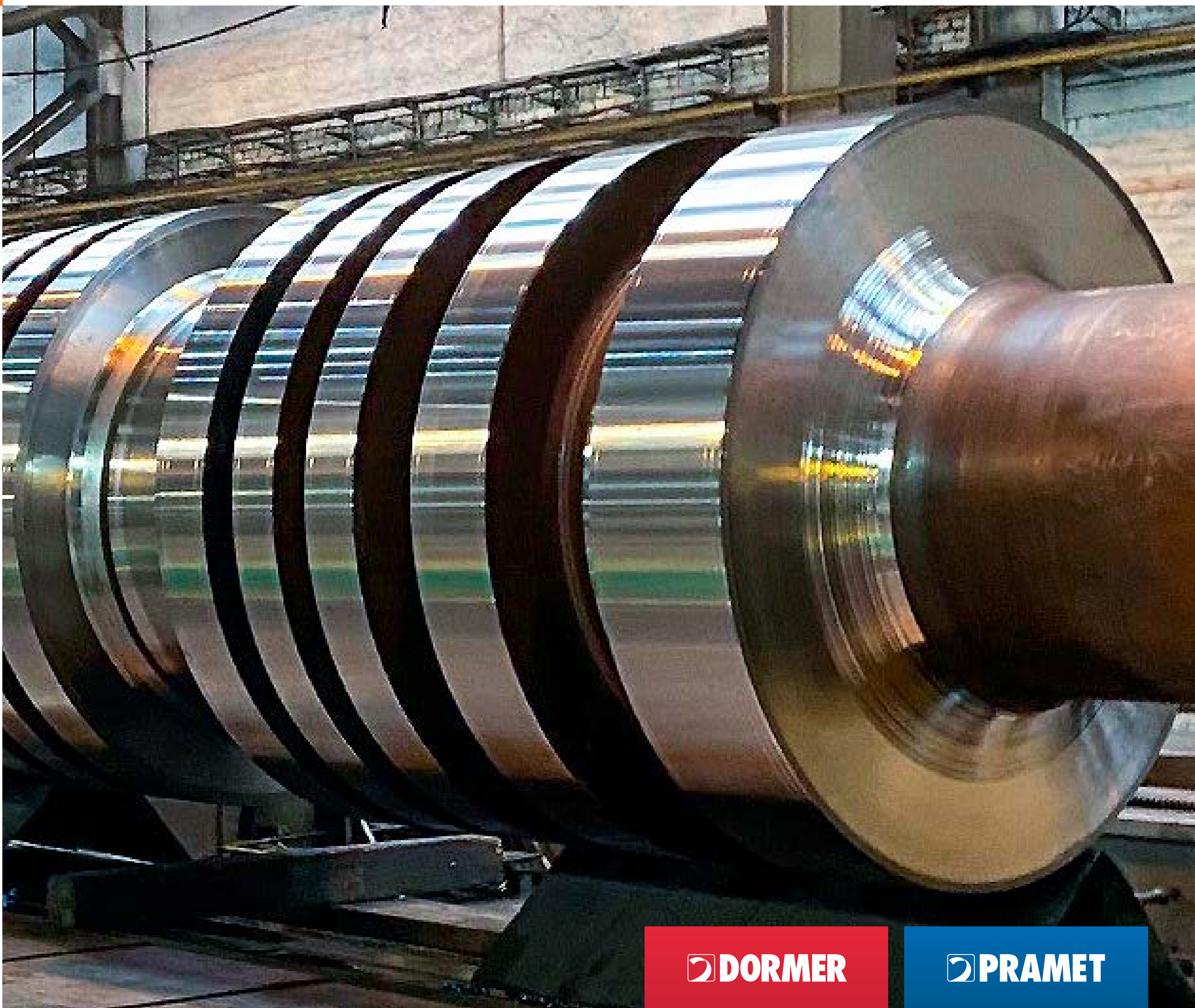
DORMER PRAMET

HEAVY MACHINING

2023

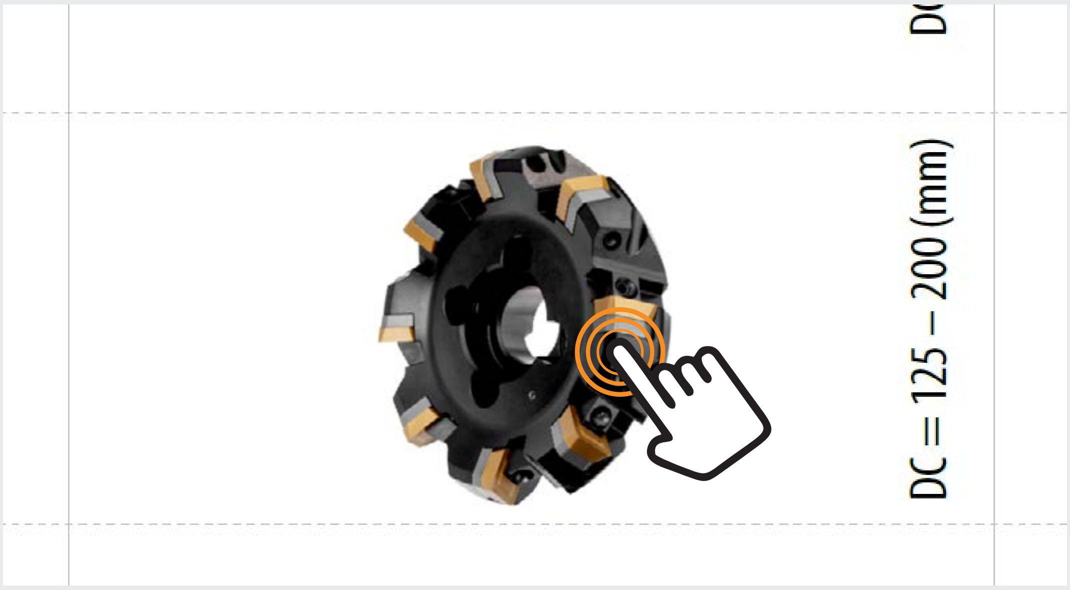


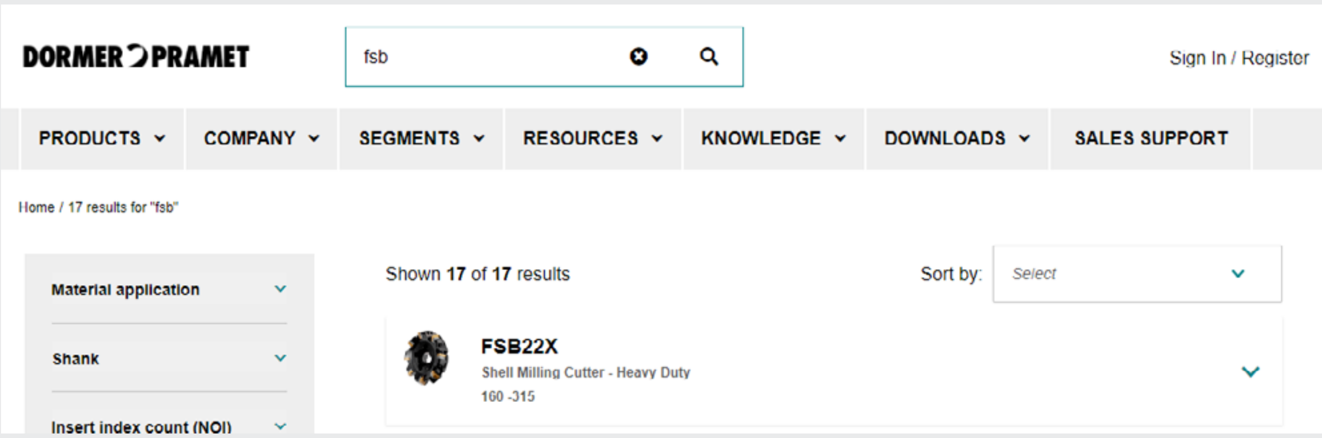
DORMER PRAMET

CONTENT

INTRODUCTION

This catalogue presents a selection of Dormer Pramet products most suitable for the machining of large-scale components, which require high levels of material to be removed. These applications require tools with high performance and operational reliability. Included are turning, milling, hole making and threading tools for machining large shafts, forgings, castings, and bright steel round bars.
Also included are special cutters for machining specimens with groove shape U or V for Charpy impact testing.
Technical section including the relevant information to help optimize the machining processes for each group of tools is a part of this catalogue as well.
The catalogue is designed as an interactive navigator through the roughing assortment, where by clicking on the image of the tool you will be directed to the webpage of the tool, which provides detailed information about the tool and the associated inserts and accessories.





PRAMET

250C09R-F60SB22X

Shell Milling Cutter - Heavy Duty
Material ID: 6759710

Face mill with positve/negative geometry which utilises 9 off SBKX 2207DZ.. or SBMR 2207DZ.. single sided inserts, cutting diameter 250 to maximum 267.4mm with 60° approach angle and 60mm connection bore according to DIN 8030-C, 63mm protrusion length and differential pitch for face milling with a maximum depth of cut of 15.0mm

If you want to see the prices or stock information, please sign in or request access.

Contact Sales

Related Parts

INSERT | SPARE PART

Grade	Chipbreaker manufacturer'...	Material application
Select	Select	Select

SBKX 22
Milling Insert - Heavy Duty

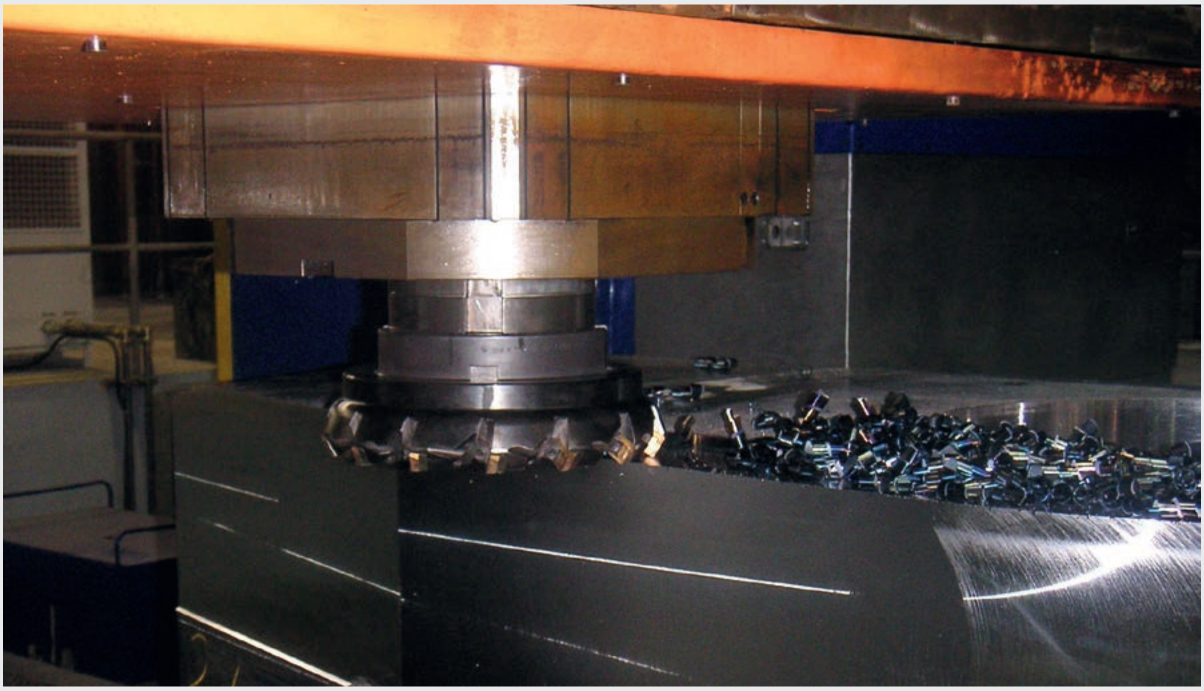
SBMR 22
Milling Insert - Heavy Duty

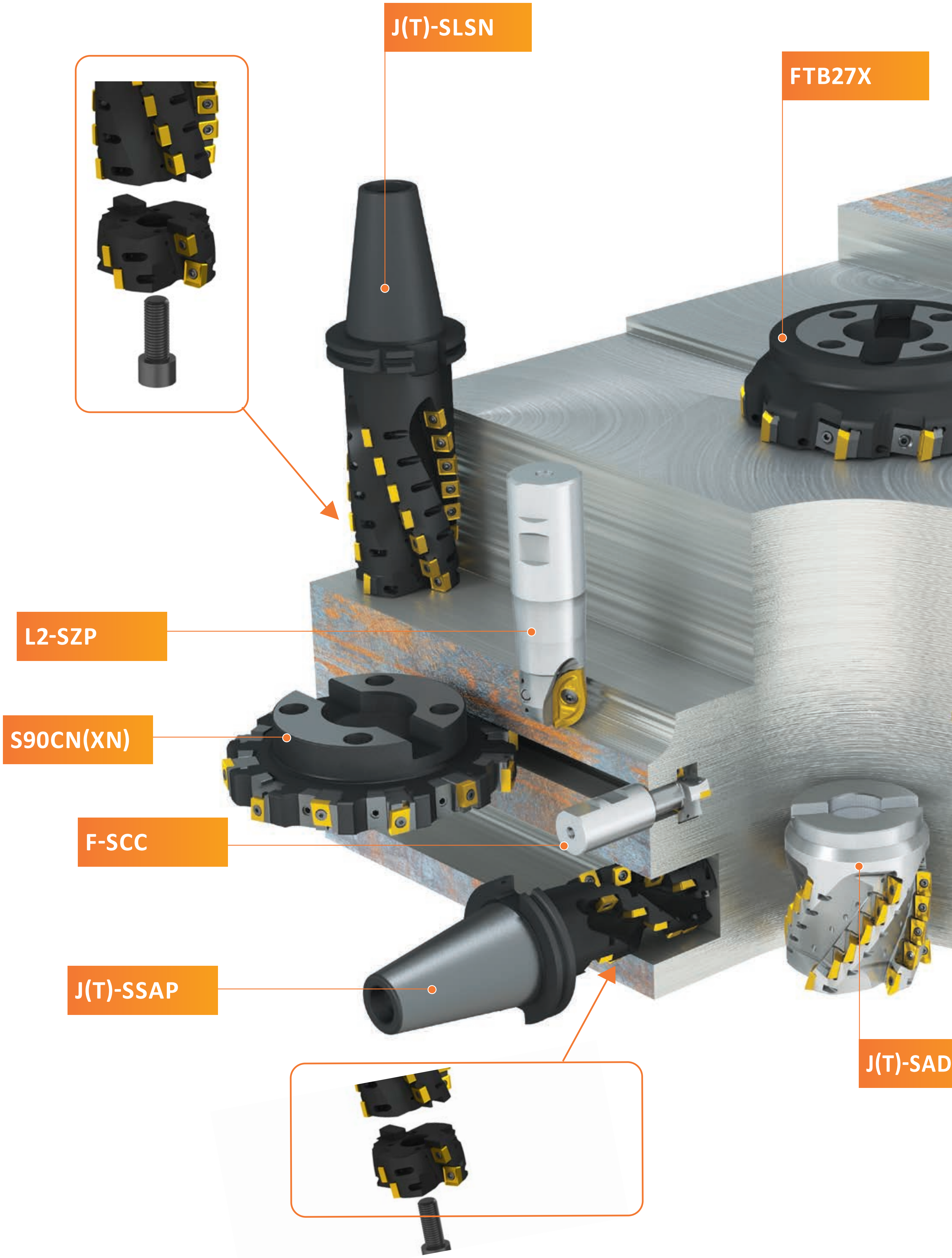
Downloads

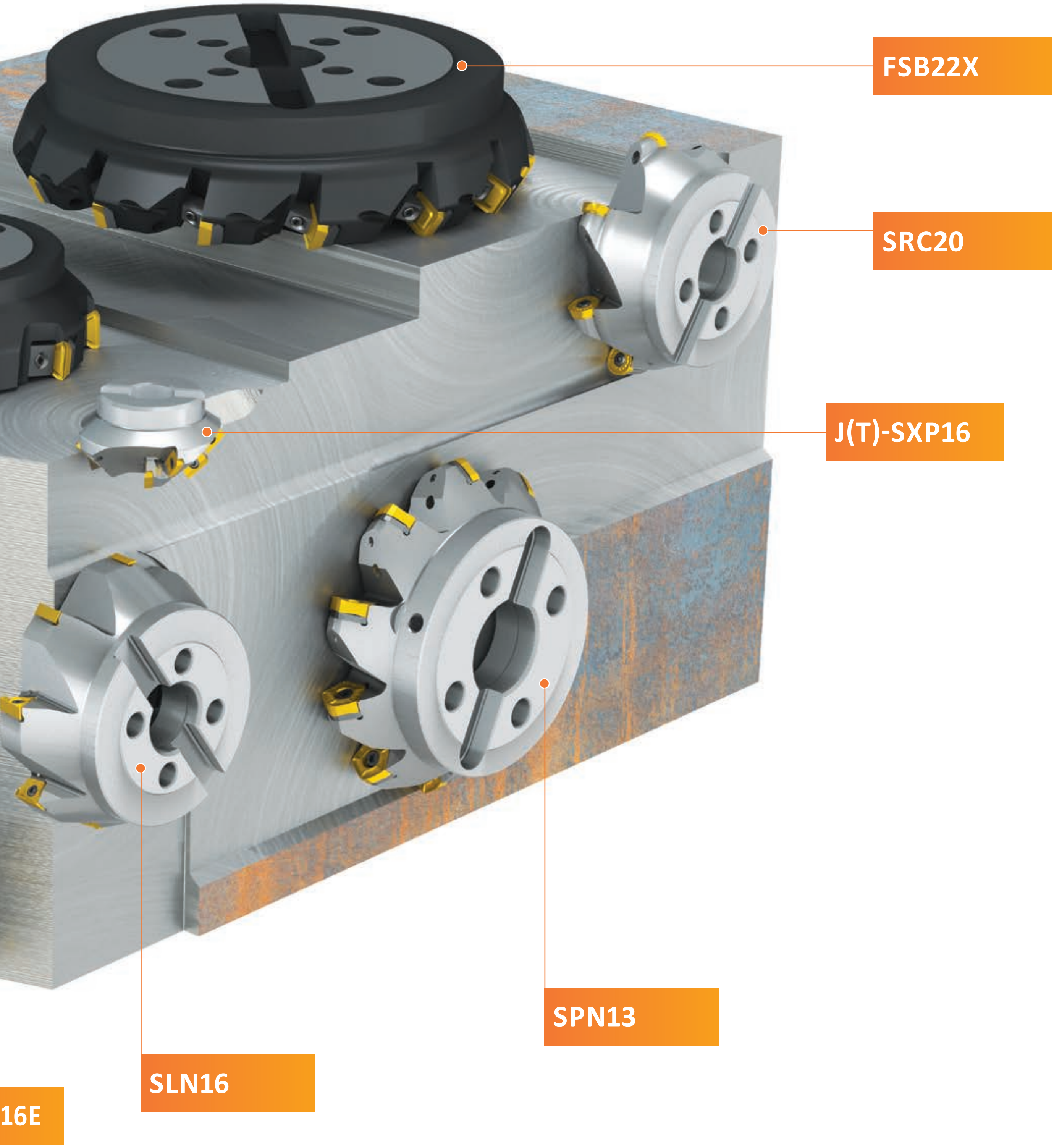
HEAVY MILLING

HEAVY MILLING

Milling large-size components is inherently associated with the removal of large material allowances. These are often castings with inclusions of molding sand or free-forged steel forgings with a thick layer of scale. These are extremely unfavorable conditions for the operation of cutting tools. Therefore, our indexable inserts and tool bodies are particularly wear-resistant and operationally reliable and withstand those conditions. The Dormer Pramet range of dedicated tools for heavy milling ensure effective milling of surfaces, offsets, grooves, chamfers, edge rounding and multi-axis shaping. The standard range is complemented by tools made to order.

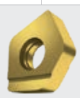



J(T)-SLSN
FTB27X
L2-SZP
S90CN(XN)
F-SCC
J(T)-SSAP
J(T)-SAD


FSB22X
SRC20
J(T)-SXP16
SPN13
SLN16
16E

INDEXABLE FACE MILLS

FACE MILLING

	SPN13	FSB22X	
	57°	60°	
	APMX (mm) 10.0	APMX (mm) 15.0	
	DC (mm) 100 – 315	DC (mm) 125 – 250	
Cylindrical shank			
Weldon			
Shell mill (≤ 125 mm)	 DC = 100 – 125 (mm)	 DC = 125 (mm)	
Shell mill (> 125 mm)	 DC = 160 – 315 (mm)	 DC = 160 – 250 (mm)	
Page			
ISO	P M K S H	P M K	
Insert shape	  	 	
Inserts	PNM. 1308, PNMQ 13, XN.. 1308	SB.. 2207, SB.. 2207	
No. of cutting edges	10 / 1	4 / 1	
Face milling	■	■	
Chamfer milling			
Helical interpolation			
Progressive plunging			
Ramping			
Shape surfaces milling (copy milling)			
Shallow shoulder milling			
Shallow slot milling			
Plunge milling			

■ Primary use ◩ Possible use

INDEXABLE SQUARE SHOULDER MILLS

SQUARE SHOULDER MILLING

	SAD16E	SLN16	FTB27X
	90°	90°	90°
	APMX (mm) 13.0	APMX (mm) 13.0	APMX (mm) 18.0
	DC (mm) 25 – 175	DC (mm) 63 – 175	DC (mm) 140 – 260
Cylindrical / Weldon	 DC = 25 – 32 (mm)  DC = 25 – 40 (mm)		
Modular / Morse	 DC = 32 – 40 (mm)  DC = 25 – 40 (mm)		
Shell mill (≤ 140 mm)	 40 – 140 (mm)	 DC = 63 – 140 (mm)	 DC = 140 (mm)
Shell mill	 DC = 160 – 175 (mm)	 DC = 160 – 175 (mm)	 DC = 175 – 260 (mm)
Page			
ISO	P M K N S H	P K N H	P M K
Insert shape		 	
Inserts	AD.X 1606	LN.U 1607, LNGU 16	TBMR 2707
No. of cutting edges	2	4	3
Shallow shoulder milling	■	■	■
Helical interpolation	■		
Shallow slot milling	■	■	◩
Plunge milling	■	■	
Progressive plunging	■		
Ramping	■		
Face milling	◩		◩
Shape surfaces milling (copy milling)	■	◩	

■ Primary use ◩ Possible use

INDEXABLE DEEP SHOULDER MILLS

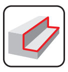
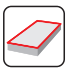
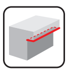
DEEP SHOULDER MILLING

	J(T)-SAD16E	J(T)-SLSN	J(T)-SSAP
	90°	90°	90°
	APMX (mm) 40.0 – 108.0	APMX (mm) 104.0 – 134.0	APMX (mm) 58.0– 95.0
	DC (mm) 50 – 100	DC (mm) 63 – 80	DC (mm) 50 – 80
Arbor (ISO/DIS 7388-1) (DIN 69871-1)	 DC = 50 – 80 (mm)	 DC = 63 – 80 (mm)	 DC = 50 – 80 (mm)
Arbor (ISO 297) (DIN 2080)	 DC = 50 – 80 (mm)	 DC = 63 – 80 (mm)	
Arbor (JIS B 6339)	 DC = 50 – 80 (mm)	 DC = 50 – 80 (mm)	 DC = 50 – 80 (mm)
Shell mill	 DC = 50 – 100 (mm)		
Page			
ISO	P M K N S H	P K	P M K N S H
Insert shape			
Inserts	AD.. 1606	LNET 1606, SN.. 1305, SNET 13	APEW 15, APET 15, SPET 12, SPEW 12 AD, SPET 12 AD
No. of cutting edges	2	2 / 8	2 / 4
Deep shoulder milling	■	■	■
Deep slot milling	■	■	■
Face milling	◩	◩	◩
Plunge milling	◩	◩	◩

■ Primary use ◩ Possible use

INDEXABLE SLOT MILLS

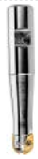
SLOT MILLING

	S90CN(XN)		
	90°		
	APMX (mm) 14.0 – 30.5		
	DC (mm) 125 – 315		
Disc	 DC = 125 – 315 (mm)		
Disc (200 mm)	 DC = 200 (mm)		
Shell mill	 DC = 125 – 200 (mm)		
Page			
ISO	P M K		
Insert shape	 		
Inserts	CNHQ 1005 / XNHQ 1205, XNHQ 1606		
No. of cutting edges	2		
Deep slot milling 	■		
Deep shoulder milling 	◩		
Face milling 	◩		
Rear face milling 	◩		

■ Primary use ◩ Possible use

INDEXABLE COPY MILLS

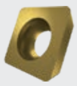
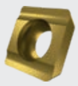
COPY MILLING

	SRC20	L2-SZP			
	–	–	–	–	–
	APMX (mm) 10.0	APMX (mm) 8.9 – 44.7	APMX (mm) 8.9 – 44.7	APMX (mm) 8.9 – 44.7	APMX (mm) 8.9 – 44.7
	DCX (mm) 80 – 125	DCX (mm) 25	DCX (mm) 32	DCX (mm) 40	DCX (mm) 50
Cylindrical shank					
Weldon					
Morse					
Shell mill / Modular					
Page					
	P M K S H	P M K S H	P M K S H	P M K S H	P M K S H
Insert shape					
Inserts	RC 2006	ZP	ZP	ZP	ZP
No. of cutting edges	–	2	2	2	2
Shape surfaces milling (copy milling)	■	■	■	■	■
Face milling	■				
Helical interpolation	■				
Progressive plunging	■				
Ramping	■				
Shallow slot milling					
Deep shoulder milling					
Chamfer milling					
Plunge milling					

■ Primary use ◩ Possible use

INDEXABLE CHAMFER & T-SLOTS MILLS

CHAMFER, T-SLOT MILLING

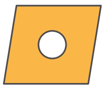
	J(T)-SXP16	SSD09	2516	F-SCC
	15°– 75°	45°	45°	90°
	APMX (mm) 7.0 – 28.0	APMX (mm) 4.5	APMX (mm) 8.5	APMX (mm) 11.0 – 18.0
	DC (mm) 35 – 45	DC (mm) 10 – 25	DC (mm) 11 – 19	DC (mm) 25 – 40
Cylindrical shank				
Weldon				
Morse				
Shell mill				
Page				
ISO	P M K N	P M K S H	P M K S	P M K
Insert shape		 		
Inserts	XPHT 1604	SDE. 0903 / SDE. 0903	TCMT 16T3	CCMX
No. of cutting edges	2	4	3	2
Chamfer milling	■	■	■	
Rear face milling				■
T-slot milling				■
Shallow shoulder milling				◩
Shallow slot milling				◩

■ Primary use ◩ Possible use

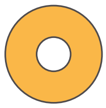
OTHER MILLING INSERTS

OTHER MILLING INSERTS – NAVIGATOR

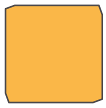
CNM 563



RDHX 20

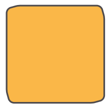


SPGN 25 DZ

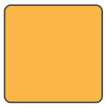


SPKN 15

SPKR 15

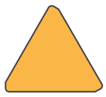


SPUN 25



TPKN 22

TPKR 22



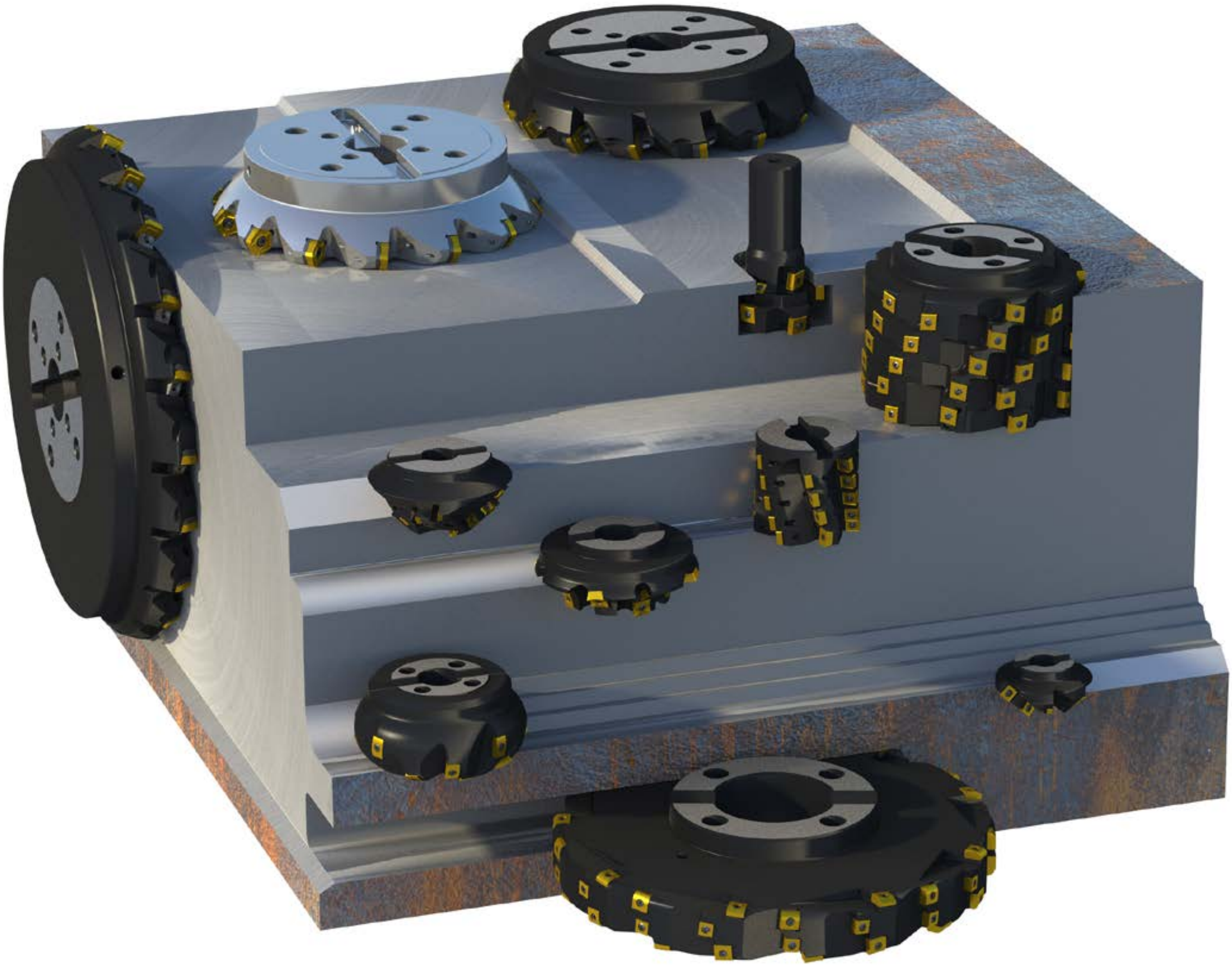
TPUN 22

TOOLS MADE ON REQUEST

Dormer Pramet also offers specialized tools for heavy machining. We understand that fabricating pieces occasionally requires a specialized tool configuration. Or you might need a very limited number of tools for a specific purpose. This section highlights our Heavy Machining tools that are less common, but no less important, to your heavy machining needs.

Some of our tools are already available in various models including left-handed cutting configuration or custom cutter bodies. Please contact your local Dormer Pramet sales office for further information and advice.

EXAMPLES OF TOOLS MADE ON REQUEST



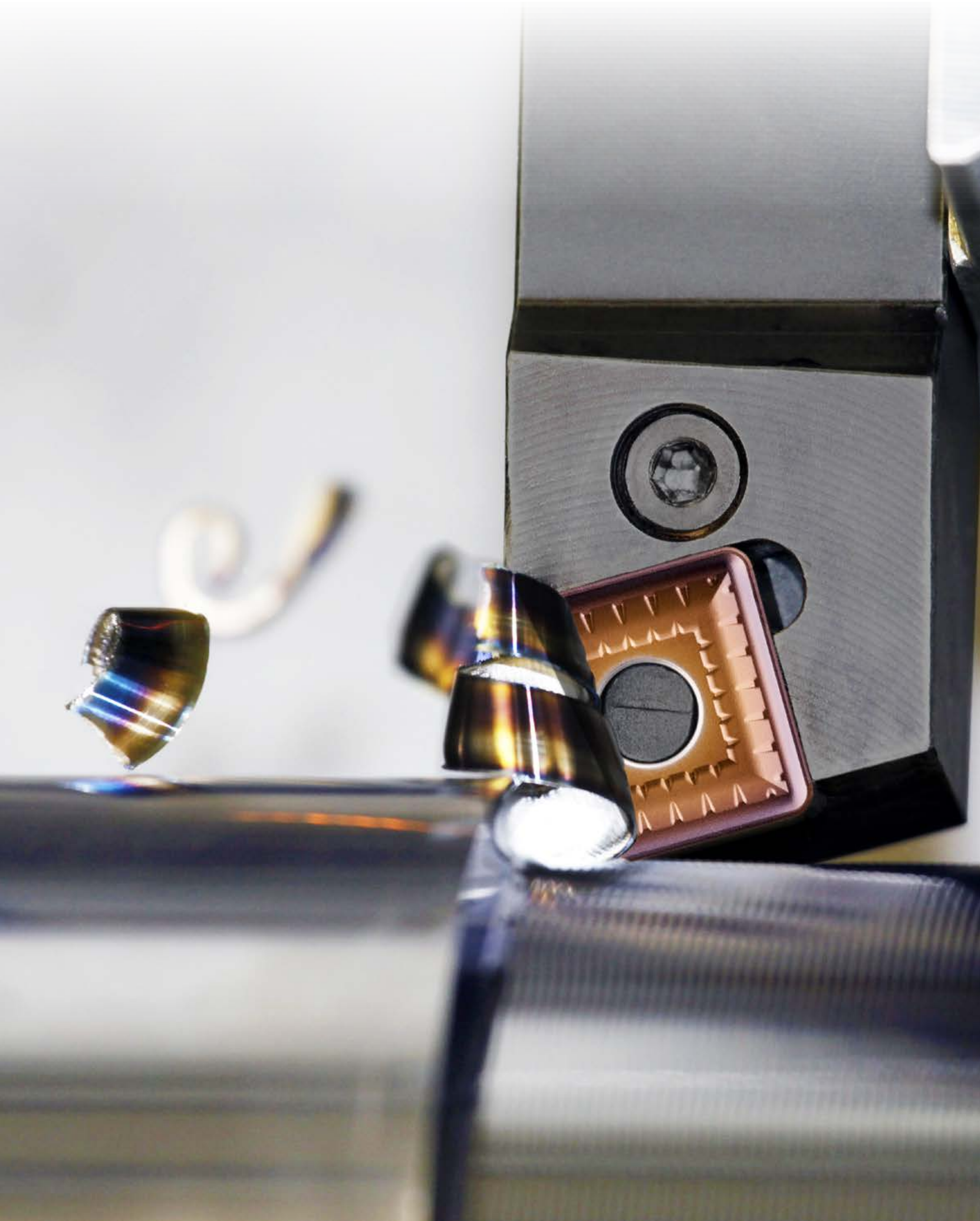
DORMER PRAMET

ALWAYS CONNECT

No wifi or internet connection? The machining calculator works perfectly even when you are offline, making sure it‘s always available when you need it. **Simply Reliable.**

Download on the App Store | GET IT ON Google Play | Download on AppGallery

HEAVY TURNING



HEAVY TURNING

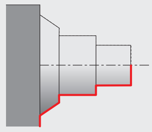
Heavy Turning of large shafts with scale from the forging or casting processes with interrupted cutting caused by uneven surfaces require extremely strong and operationally reliable tools. Within the range of Dormer Pramet indexable tools are inserts with cutting edge lengths from 25 – 50 mm and tool holders which cover all common applications and are available in sizes that ensure the required stability of the cutting process. Cutting efficiency is achieved by applying modern and reliable grades and insert geometries designed to withstand heavy cutting loads. To support quick tool change we offer the unique exchangeable cassettes system KH which allows the user to replace the head of the tool within 30 seconds.


KHP-LBN(RL)
KHP-RSC(RL)
KHS-SBC(RL)
KHP-SBN(RL)
PLBN(RL) EXT
PRSC(RL) EXT
PSBN(RL) EXT
SSBC(RL) EXT

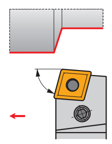

DKH(RL)
KHP-SSN(RL)
KHP-CBN(RL)
KHP-CLN(RL)
KHP-SSN(RL)
DWLN(RL) EXT
PCBN(RL) EXT
PSKN(RL) EXT
DSDNN EXT
PCLN(RL) EXT

ISO TURNING – HEAVY ROUGHING – EXTERNAL

FIXED TOOL HOLDERS (NEGATIVE INSERTS)

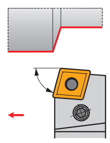


PCBN(RL) EXT



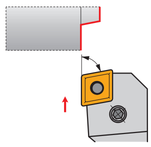
Angle	Insert	Size	Shank
75°	CN..	19 25	40×40

PCBN(RL) EXT



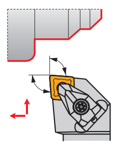
Angle	Insert	Size	Shank
75°	CN..	25	50×50

PCKN(RL) EXT



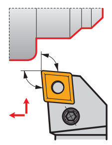
Angle	Insert	Size	Shank
75°	CN..	19	40×40

DCLN(RL) EXT



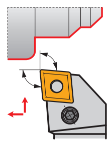
Angle	Insert	Size	Shank
95°	CN..	19	40×40

PCLN(RL) EXT



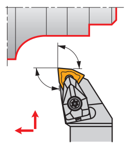
Angle	Insert	Size	Shank
95°	CN..	19 25	40×40

PCLN(RL) EXT



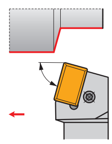
Angle	Insert	Size	Shank
95°	CN..	25	50×50

DWLN(RL) EXT



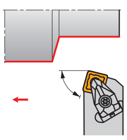
Angle	Insert	Size	Shank
95°	WN..	13	40×40

PLBN(RL) EXT



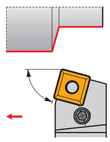
Angle	Insert	Size	Shank
75°	LN..	40 50	60×60

DSBN(RL) EXT



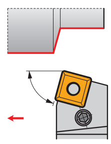
Angle	Insert	Size	Shank
75°	SN..	19	40×40

PSBN(RL) EXT



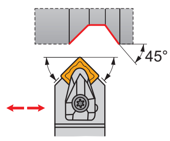
Angle	Insert	Size	Shank
75°	SN..	19 25	40×40

PSBN(RL) EXT



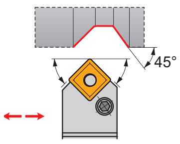
Angle	Insert	Size	Shank
75°	SN..	19 25	50×50

DSDNN EXT



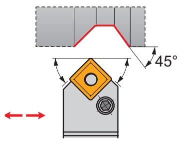
Angle	Insert	Size	Shank
45°	SN..	19 25	40×40

PSDNN EXT



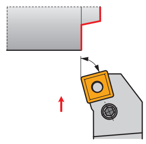
Angle	Insert	Size	Shank
45°	SN..	19 25	40×40

PSDNN EXT



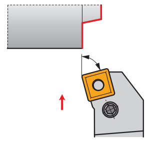
Angle	Insert	Size	Shank
45°	SN..	25	50×50

PSKN(RL) EXT



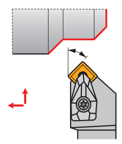
Angle	Insert	Size	Shank
75°	SN..	19 25	40×40

PSKN(RL) EXT

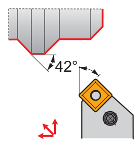


Angle	Insert	Size	Shank
75°	SN..	19 25	50×50

DSSN(RL) EXT

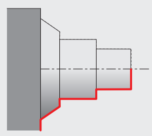
45°	SN..
	19
	40×40

PSSN(RL) EXT

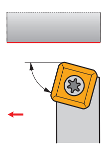
45°	SN..
	25
	50×50

ISO TURNING – HEAVY ROUGHING – EXTERNAL

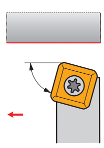
FIXED TOOL HOLDERS (POSITIVE INSERTS)



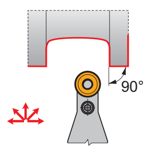
SSBC(RL) EXT

75°	SC..
	25 38
	40×40

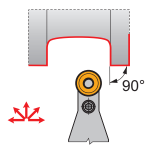
SSBC(RL) EXT

75°	SC..
	25 38
	60×60

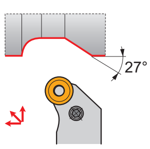
PRDCN EXT

	RC..
	20 25
	40×40

PRDCN EXT

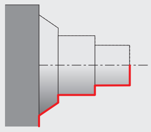
	RC..
	20 25
	50×50

PRSC(RL) EXT

	RC..
	20 25
	40×40

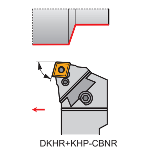
ISO TURNING – HEAVY ROUGHING – EXTERNAL

HEAD (KH)



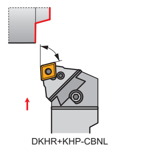
KHP-CBNR + DKH(RL)

75°	CN..
	25
40×50 / 60×80	



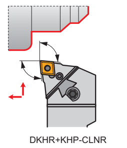
KHP-CBNL + DKH(RL)

75°	CN..
	25
40×50 / 60×80	



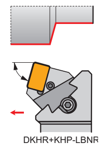
KHP-CLNR/L + DKH(RL)

95°	CN..
	19 25
40×50 / 60×80	



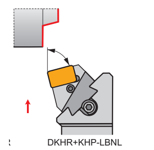
KHP-LBNR + DKH(RL)

75°	LN..
	40
40×50 / 60×80	



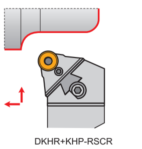
KHP-LBNL + DKH(RL)

75°	LN..
	40
40×50 / 60×80	



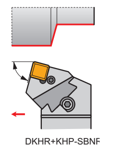
KHP-RSCR/L + DKH(RL)

	RC..
	20
40×50 / 60×80	



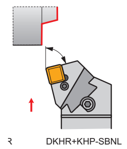
KHP-SBNR + DKH(RL)

75°	SN..
	19 25
40×50 / 60×80	



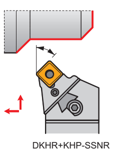
KHP-SBNL + DKH(RL)

75°	SN..
	25
40×50 / 60×80	



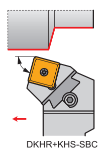
KHP-SSNR/L + DKH(RL)

45°	SN..
	25
40×50 / 60×80	



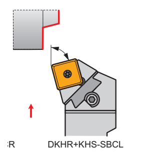
KHS-SBCR + DKH(RL)

75°	SC..
	25 38
40×50 / 60×80	



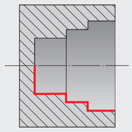
KHS-SBCL + DKH(RL)

75°	SC..
	25 38
40×50 / 60×80	

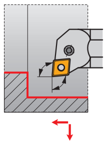


ISO TURNING – INTERNAL

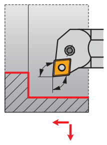
SHORT AND STABLE COMPONENTS (NEGATIVE INSERTS)



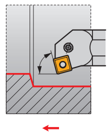
PCLN(RL) INT

95°	CN..
	19
ØDmin	50

PCLN(RL) INT

95°	CN..
	19
ØDmin	60

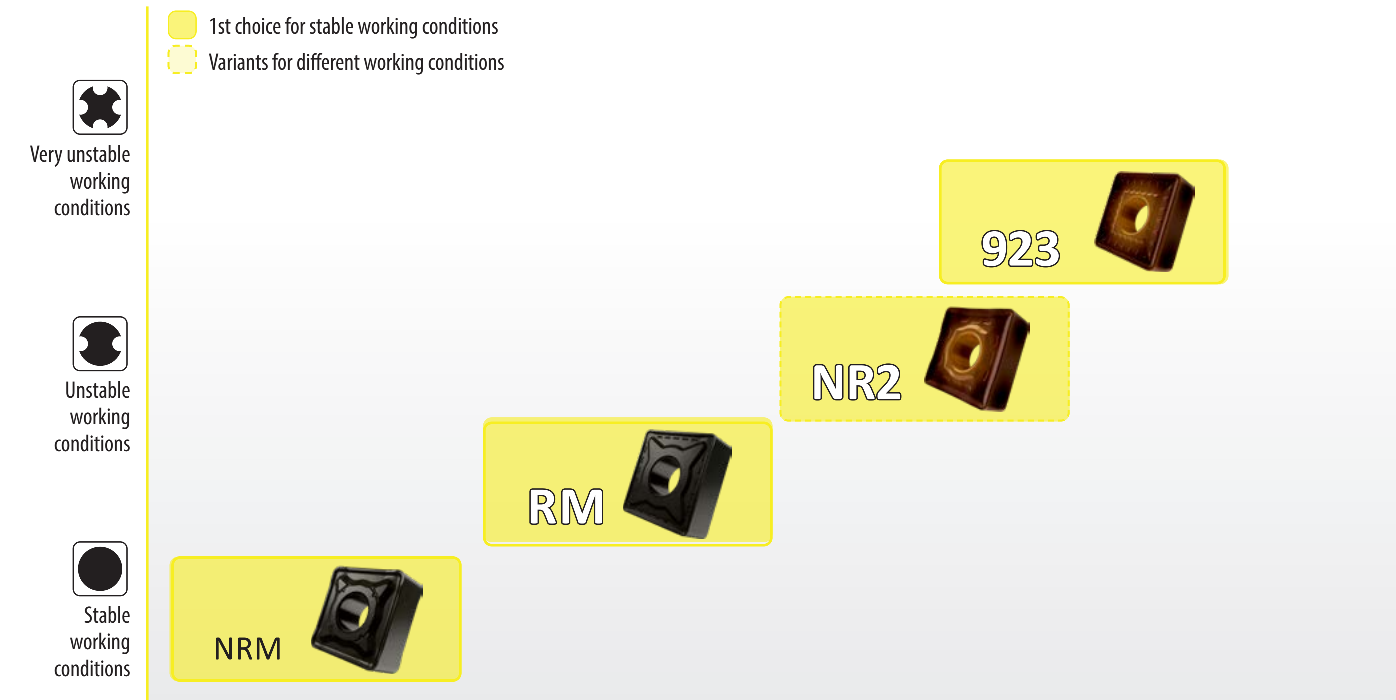
PSKN(RL) INT

93°	SN..
	19
ØDmin	50

P

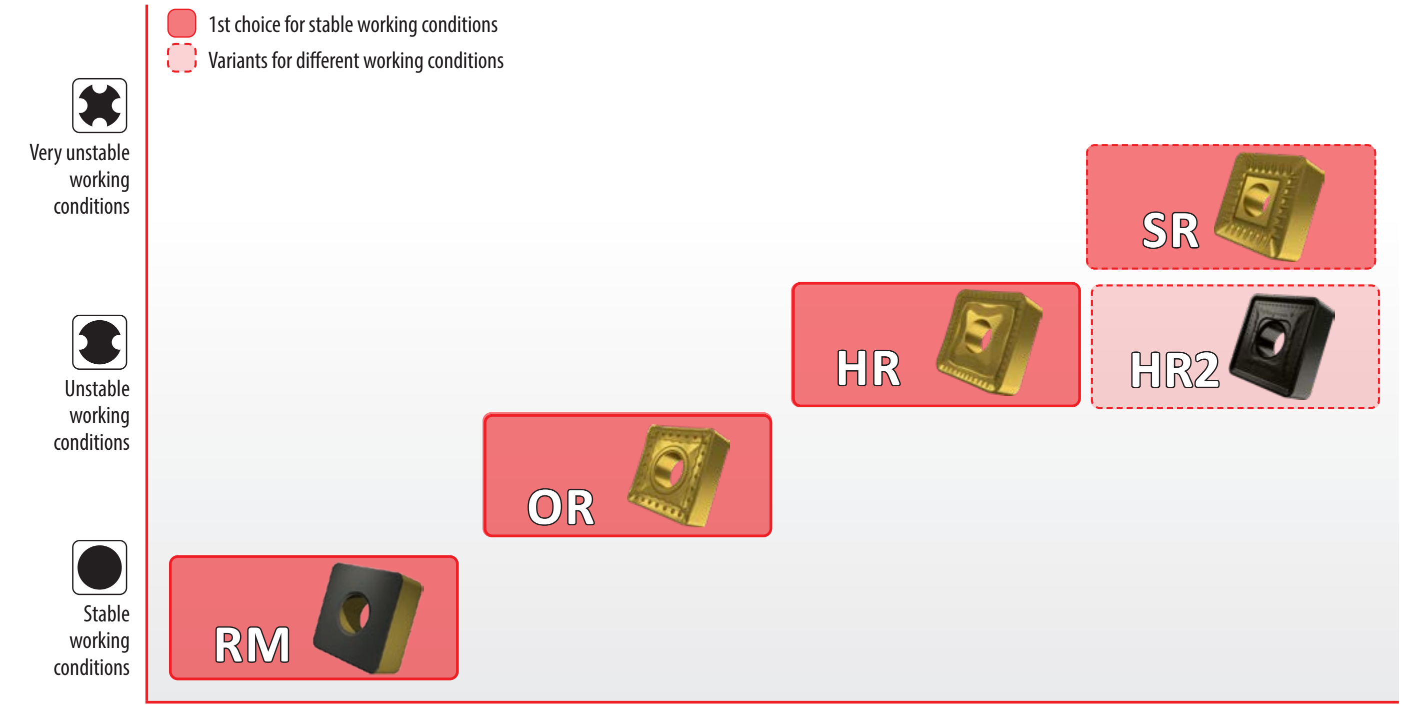
	R	HR	HR	HR
f	0.35 – 1.2 mm/rev	0.40 – 1.3 mm/rev	0.7 – 1.5 mm/rev	1.2 – 2.6 mm/rev
a_p	2.4 – 15 mm	3.0 – 16 mm	5.0 – 16 mm	10.0 – 36 mm

M



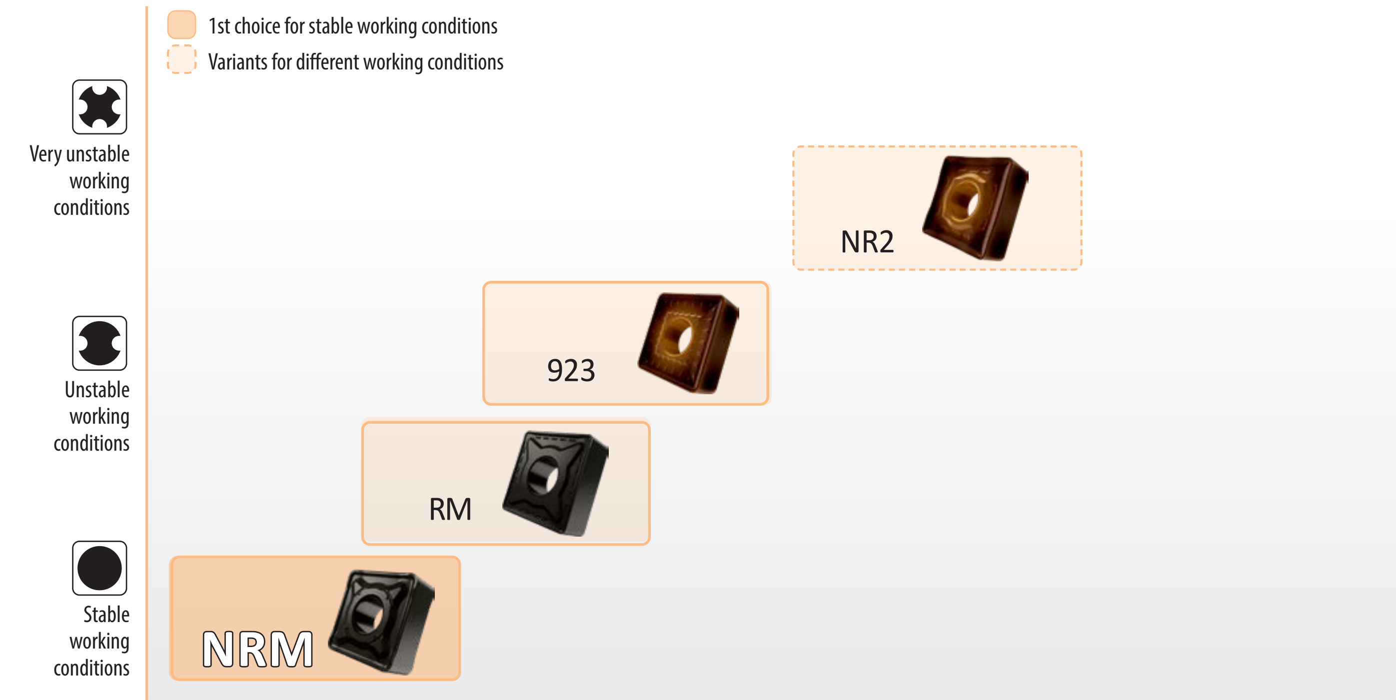
	R	R	HR	
f	0.4 – 1.0 mm/rev	0.40 – 1.0 mm/rev	0.5 – 1.2 mm/rev	
a_p	2.4 – 15 mm	3.0 – 13 mm	3.0 – 16 mm	

K



	R	R	HR	HR
f	0.1 – 1.1 mm/rev	0.40 – 1.3 mm/rev	0.5 – 1.4 mm/rev	0.7 – 1.3 mm/rev
a_p	2.4 – 12 mm	4.0 – 16 mm	5.0 – 16 mm	10.0 – 36 mm

S



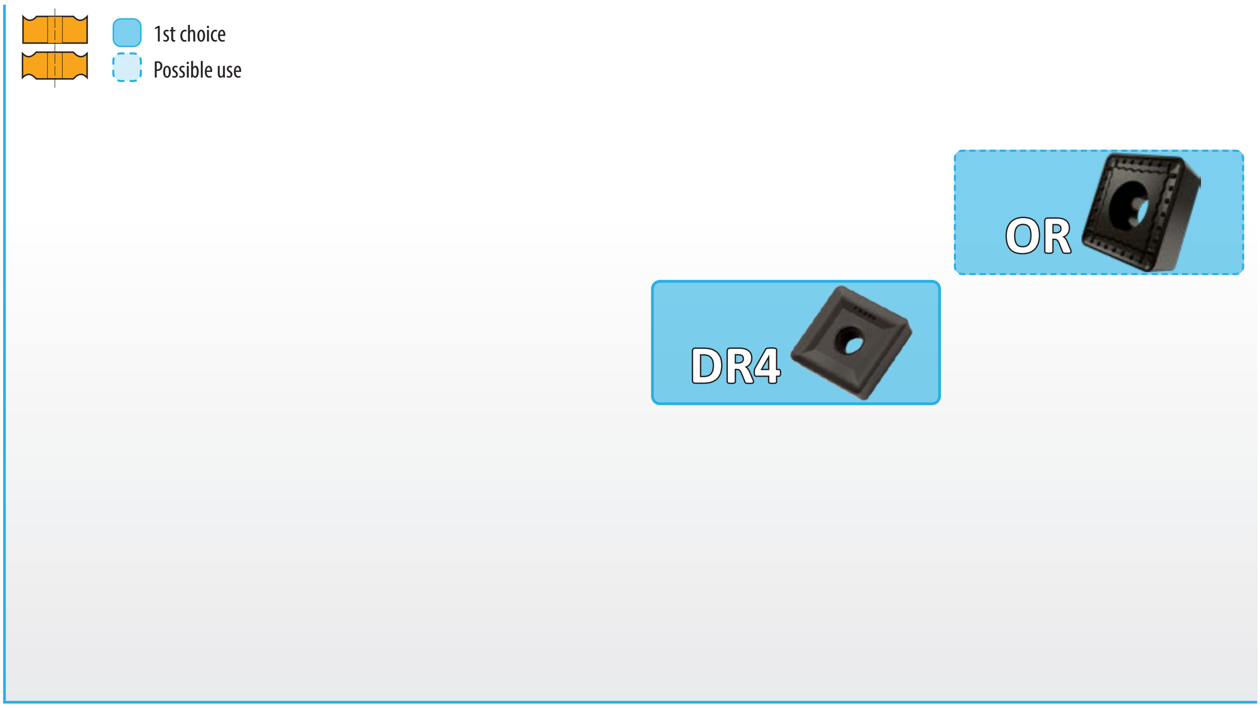
	R	R	HR	
f	0.35 – 0.8mm/rev	0.40 – 1.0 mm/rev	0.5 – 1.5 mm/rev	
a_p	2.4 – 12 mm	3.0 – 11 mm	3.0 – 13 mm	

P

Very unstable working conditions

Unstable working conditions

Stable working conditions



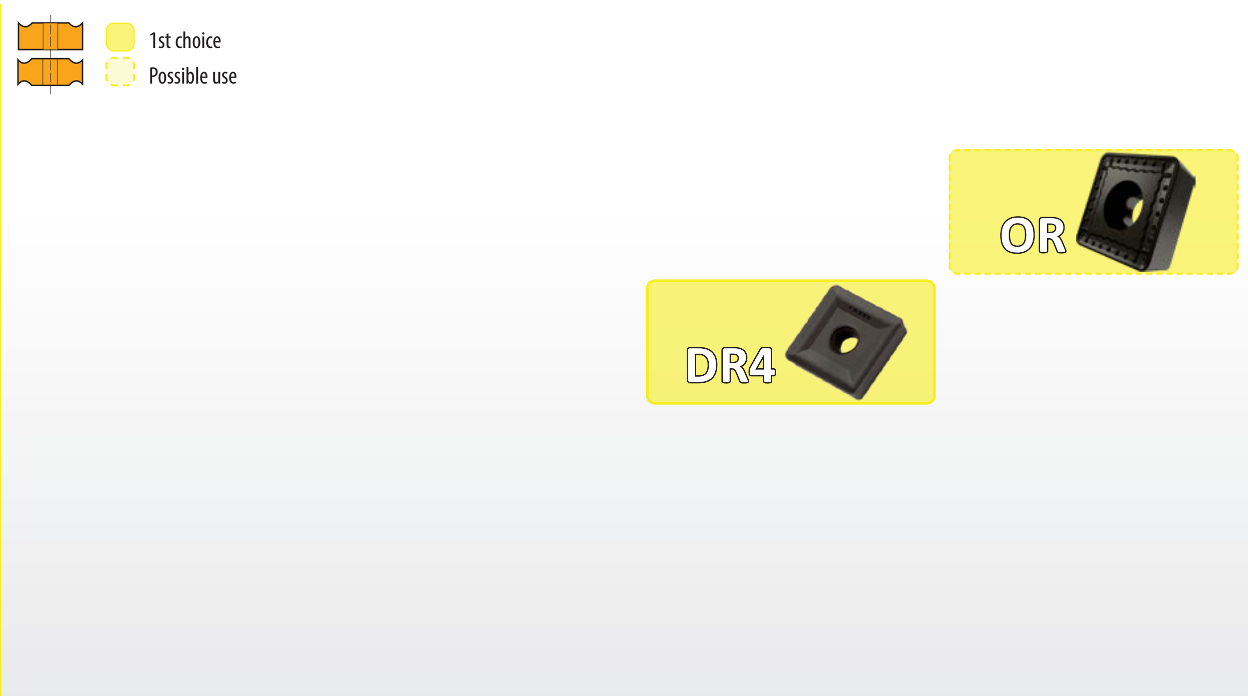
			HR	HR
f			0.7 – 1.4 mm/rev	1.0 – 1.8 mm/rev
a_p			4.0 – 18 mm	4.0 – 24 mm

M

Very unstable working conditions

Unstable working conditions

Stable working conditions



			HR	HR
f			0.7 – 1.3 mm/rev	1.0 – 1.6 mm/rev
a_p			4.0 – 18 mm	4.0 – 24 mm

S

1st choice

Possible use

Very unstable working conditions

Unstable working conditions

Stable working conditions

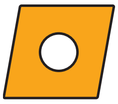
OR

				HR
f				1.1 – 1.6 mm/rev
a_p				4.0 – 13 mm

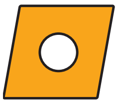
NEGATIVE INSERTS

ISO INSERTS NEGATIVE – NAVIGATOR

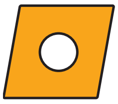
CNMA 19



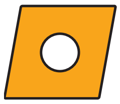
CNMG 19



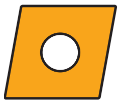
CNMG 25



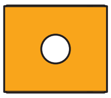
CNMM 19



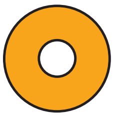
CNMM 25



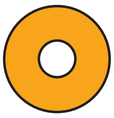
LNUX 40, LN.X 50



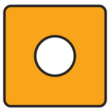
RNMG 19



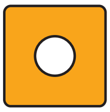
RNMG 25



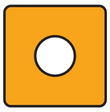
SNMA 19



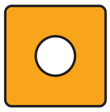
SNMA 25



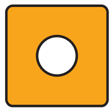
SNMG 19



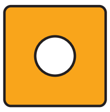
SNMG 25



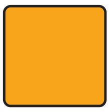
SNMM 19



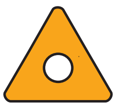
SNMM 25



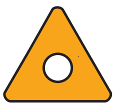
SNMX 25



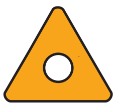
TNMG 27



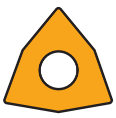
TNMG 33



TNMM 27

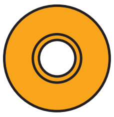


WNMM 13

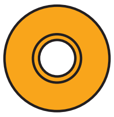


POSITIVE INSERTS

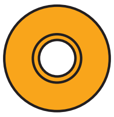
RCMX 20



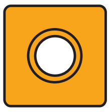
RCMX 25



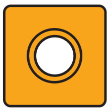
RCMX 32



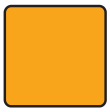
SCMT 25



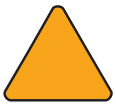
SCMT 38



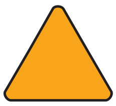
SPUN-IT 25



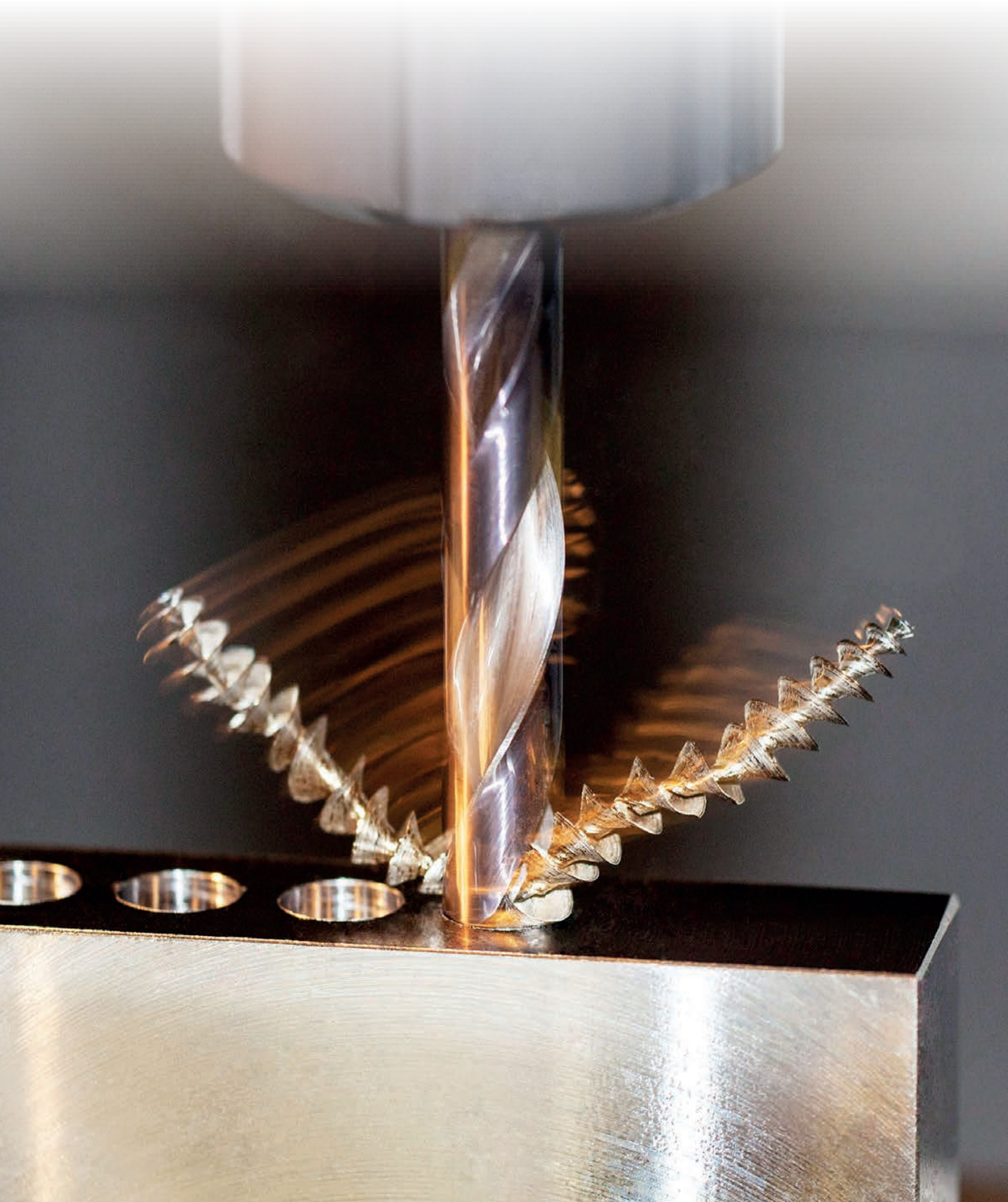
TPUN-IT 27



TPUN-IT 33



HOLEMAKING



HSS DRILLS

Material code (BMC)	HSS	HSS
Basic standard group (BSG)	DIN 345	DIN 1870(2)
Usable length (ULDR)	4×D	20×D
Application angle	118°	130°
Coating	ST	Bright ST
Shank		
Spiral form	λ20-35°	λ>35°
Hand (Cutting direction)	R	R
Cooling (CSP)		
		
Product Family Code	A130	A952
	3.00 – 50.80	8.00 – 40.00

		A130	A952
P	P1	■	■
	P2	■	■
	P3	■	◩
	P4	◩	◩
M	M1	◩	◩
	M2	◩	◩
	M3	◩	◩
	M4	◩	◩
K	K1	■	◩
	K2	◩	◩
	K3	◩	◩
	K4	◩	◩
	K5	◩	◩
N	N1	◩	◩
	N2	◩	◩
	N3	◩	◩
	N4	◩	◩
	N5		
S	S1	◩	◩
	S2	◩	◩
	S3	◩	◩
	S4	◩	◩
H	H1		
	H2		
	H3		
	H4		

■ Primary use ◩ Possible use

HYDRA DRILLS

	R950	R960	H851	H853	H855	H858	H8512	H860	H861
Material code (BMC)	HM	HM	HSS	HSS	HSS	HSS	HSS		
Basic standard group (BSG)	DORMER	DORMER	DORMER	DORMER	DORMER	DORMER	DORMER		
Usable length (ULDR)			1.5×D	3×D	5×D	8×D	12×D		
Application angle	140°	140°							
Coating	Ti-phon	Ti-phon	Bright Ni	Bright Ni	Bright Ni	Bright Ni	Bright Ni		
Shank			ISO 9766	DIN 6535HB DIN 6535HE	DIN 6535HB DIN 6535HE	DIN 6535HB DIN 6535HE			
Hand (Cutting direction)	R	R	R	R	R	R	R		
Cooling (CSP)									
	HYDRA	HYDRA	HYDRA NEW	HYDRA	HYDRA	HYDRA	HYDRA NEW	HYDRA	HYDRA
Product Family Code	R950	R960	H851	H853	H855	H858	H8512	H860	H861
	12.00 – 42.00, 15/32 – 1.5/8	12.00 – 30.50, 15/32 – 1.3/16	12.00 – 30.50, 15/32 – 1.3/16	12.00 – 42.50, 15/32 – 1.5/8	12.00 – 42.50, 15/32 – 1.5/8	13.50 – 42.50, 35/64 – 1.5/8	13.50 – 25.65, 35/64 – 1.1/64	N1 - N7	N1 - N6
P – P1	◩	■							
P – P2	■	◩							
P – P3	■								
P – P4	■								
M – M1		■							
M – M2		■							
M – M3		■							
M – M4		■							
K – K1		■	■						
K – K2	■	◩	■						
K – K3	■	◩	■						
K – K4	■	◩	■						
K – K5	■	◩	■						
N – N1									
N – N2									
N – N3									
N – N4									
N – N5									
S – S1		◩							
S – S2		◩							
S – S3		◩							
S – S4		◩							
H – H1									
H – H2									
H – H3									
H – H4									

■ Primary use ◩ Possible use

INDEXABLE DRILLS

Working length	2×D	3×D	4×D	5×D	XPET..AP	SCET..-UD	XPET..AP-SD	SCET..-SD
Picture								
Coolant					–	–	–	–
Drill type	802D	803D	804D	805D	–	–	–	–
Drill tolerance	± 0.05	± 0.05	± 0.05	± 0.05	–	–	–	–
Hole tolerance *	0 /+0.2	0 /+0.3	0 /+0.4	0 /+0.5	–	–	–	–
Surface finish *	R_a 2 – 6 µm	R_a 2 – 6 µm	R_a 2 – 6 µm	R_a 2 – 6 µm	–	–	–	–
Diameter range	15.0 – 40.0	15.0 – 58.0	17.0 – 58.0	19.0 – 31.0	–	–	–	–
P	P1				■	■	■	■
	P2				■	■	■	■
	P3				■	■	■	■
	P4				■	■	■	■
M	M1						■	■
	M2						■	■
	M3						■	■
	M4						■	■
K	K1				◪	■	◪	◪
	K2				◪	■	◪	◪
	K3				◪	■	◪	◪
	K4				◪	■	◪	◪
	K5				◪	■	◪	◪
S	S1						◪	◪
	S2						◪	◪
	S3						◪	◪
	S4						◪	◪

* The tolerance of drilled hole and surface finish are heavily dependent on machining conditions.

REAMERS AND COUNTERSINKS

Material code (BMC)	HSS-E	HSS	HSS	HSS
Coating	Bright ST	Bright ST	Bright	TiN
Basic standard group (BSG)	BS 328	DIN 311	DIN 335D	DIN 335D
Hand (Cutting direction)	R	R	R	R
Shank				
Application angle			90°	90°
Reamer form	B			
Achievable hole tolerance (TCHA)	H7	k11		
Taper gradient - millimeter (Rate of taper)				
				
Product Family Code	B101	B121	G138	G338
	3.00 – 2"	10.00 – 30.00	25.00 – 80.00	25.00 – 63.00

		B101	B121	G138	G338
P	P1	■	■	■	■
	P2	■	■	■	■
	P3	■	■	◩	■
	P4	◩	◩	◩	■
M	M1	◩		◩	◩
	M2			◩	◩
	M3				
	M4				
K	K1	■	■	◩	■
	K2	■	■	◩	■
	K3	◩	◩	◩	■
	K4				◩
	K5			◩	■
N	N1	■	■	◩	■
	N2	■	■	◩	■
	N3	■	■	■	■
	N4	◩	◩	◩	◩
	N5				
S	S1				
	S2				
	S3				
	S4				
H	H1				
	H2				
	H3				
	H4				

■ Primary use ◩ Possible use

THREADING HSS TAPS

Product Family Code		E100	E105	E108	E111	E500	E513	E515	E524	EP016H	EP11	EP21	EP31	EX016H
		M1.6 – M52	M2.5 – M50	No.5 – 1″	No.5 – 1″	M1 – M56	M3 – M50	No.1 – 2″	No.0 – 1.1/2	M2 – M30	M4 – M30	No.4 – 1″	No.8 – 1″	M2 – M64
Thread form (THFT)		M	MF	UNC	UNF	M	MF	UNC	UNF	M	MF	UNC	UNF	M
Basic standard group (BSG)		DIN 352	DIN 2181	DIN 352	DIN 2181	ISO 529	ISO 529	ISO 529	ISO 529	DIN 371/376	DIN 374	DIN 2184-1	DIN 2184-1	DIN 371/376
Thread tolerance class (TCTR)		6H	6H	2B	2B	6H	6H	2B	2B	6H	6H	2B	2B	6H
Threading application														
Usable length (ULDR)		1.5×D	1.5×D	1.5×D	1.5×D	1.5×D	1.5×D	1.5×D	1.5×D	2.5×D	2.5×D	2.5×D	2.5×D	2.5×D
Material code (BMC)		HSS	HSS	HSS	HSS	HSS	HSS	HSS	HSS	HSS-E PM	HSS-E PM	HSS-E PM	HSS-E PM	HSS-E PM
Tap chamfer style (TCS)		C 2-3	C 2-3	C 2-3	C 2-3					B 3.5-5	B 3.5-5	B 3.5-5	C 2-3	C 2-3
Flute Geometry (FDC)														
Flute helix angle (FHA)														λ 45°
Hand (Cutting direction)		R	R	R	R	R	R	R	R	R	R	R	R	R
Coating		Bright	Bright	Bright	Bright	Bright	Bright	Bright	Bright	ST	ST	ST	ST	ST
Coolant exit style (CXSC)														
P	P1	■	■	■	■	■	■	■	■	◪	◪	◪	◪	◪
	P2	■	■	■	◪	■	■	■	■	■	◪	■	■	■
	P3	◪	◪	◪	◪	◪	◪	◪	◪	■	■	■	■	■
	P4	◪	◪	◪	◪	◪	◪	◪	◪		◪	■	■	■
M	M1									◪	◪	◪	◪	◪
	M2									◪	◪	◪	◪	◪
	M3									◪	◪	◪	◪	◪
	M4									◪	◪	◪	◪	◪
K	K1	◪	◪	◪	◪	◪	◪	◪	◪	◪	◪	◪	◪	
	K2	◪	◪	◪	◪	◪	◪	◪	◪	◪	◪	◪	◪	
	K3	◪	◪	◪	◪	◪	◪	◪	◪	◪	◪	◪	◪	
	K4	◪	◪	◪	◪	◪	◪	◪	◪	◪	◪	◪	◪	
	K5			◪	◪	◪	◪	◪	◪	◪	◪	◪	◪	
N	N1	■	■	■	■	◪	◪	◪	◪					
	N2	■	■	■	■	◪	◪	◪	◪					
	N3	■	■	■	■	◪	■	■	■					
	N4	■	■	◪	■	◪	◪	◪	◪					
	N5													
S	S1													
	S2													
	S3													
	S4													
H	H1													
	H2													
	H3													
	H4													

■ Primary use ◪ Possible use

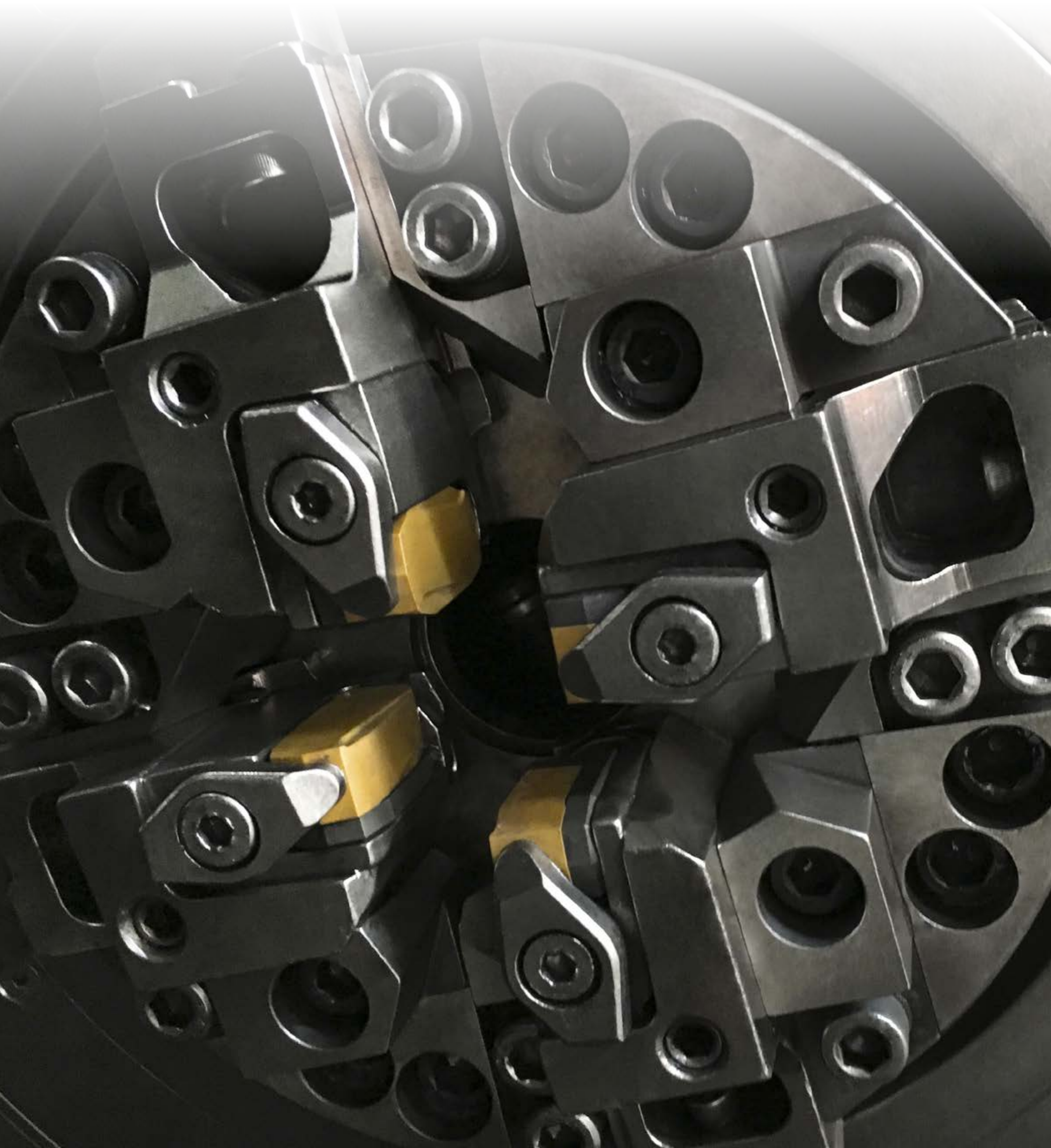
Thread form (THFT)	MF	UNC	UNF
Basic standard group (BSG)	DIN 374	DIN 2184-1	DIN 2184-1
Thread tolerance class (TCTR)	6H	2B	2B
Threading application			
Usable length (ULDR)	2.5×D	2.5×D	2.5×D
Material code (BMC)	HSS-E PM	HSS-E PM	HSS-E PM
Tap chamfer style (TCS)	C 2-3	C 2-3	C 2-3
Flute Geometry (FDC)			
Flute helix angle (FHA)	λ 45°	λ 45°	λ 45°
Hand (Cutting direction)	R	R	R
Coating	ST	ST	ST
Coolant exit style (CXSC)			

Product Family Code		EX11	EX21	EX31
		M4 – M30	No.4 – 1″	No.8 – 1″
P	P1	◩	◩	◩
	P2	■	■	■
	P3	■	■	■
	P4	■	■	■
M	M1	◩	◩	◩
	M2	◩	◩	◩
	M3	◩	◩	◩
	M4	◩	◩	◩
K	K1			◩
	K2			◩
	K3			◩
	K4			◩
	K5			◩
N	N1			
	N2			
	N3			
	N4			
	N5			
S	S1			
	S2			
	S3			
	S4			
H	H1			
	H2			
	H3			
	H4			

■ Primary use ◩ Possible use

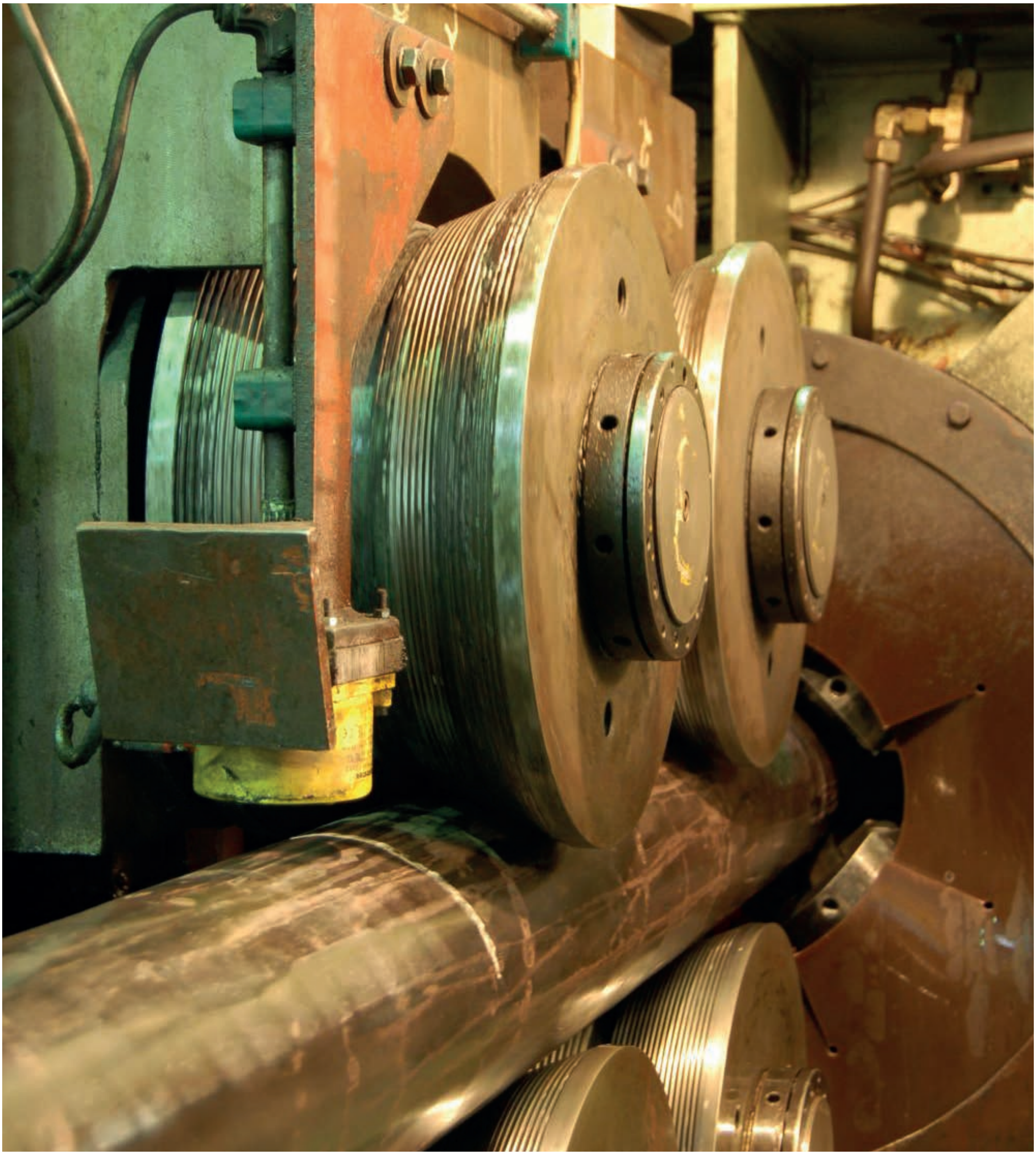
⏮

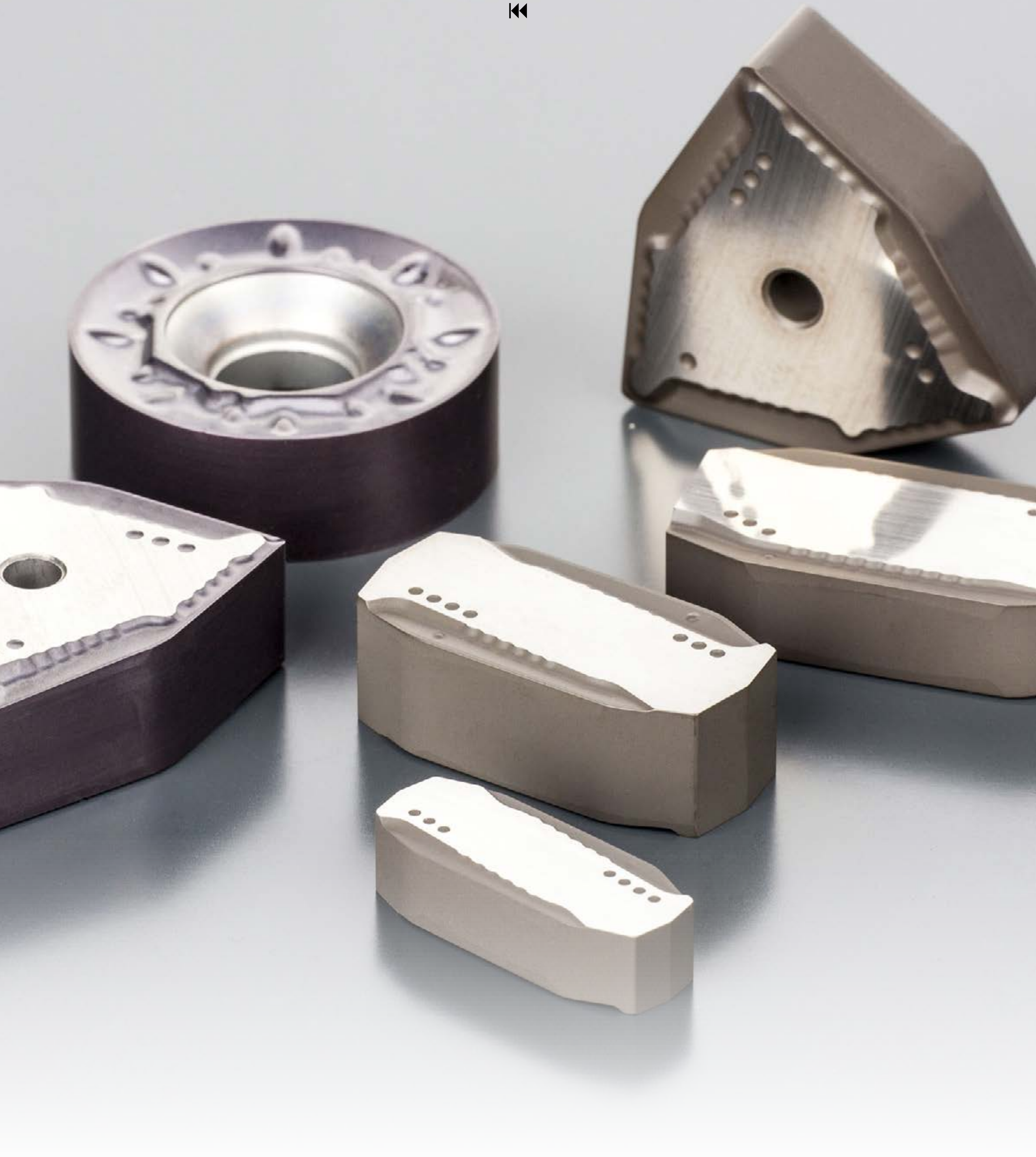
BAR PEELING



INTRODUCTION

In Bar Peeling applications, the most important criteria are process stability, high productivity, dimensional accuracy of machined bars, and excellent surface quality. With many years of experience, we can offer a wide range of inserts with specific geometries suitable for steel and stainless steel as well as difficult to machine materials. Dormer Pramet state-of-the-art production methods for cemented carbide grades and MT-CVD and PVD coating layers ensures our peeling inserts provide the required tool life to meet customer expectation. Specific cassettes that are used on different manufacturers machine tools can be provided on request.





BAR PEELING INSERTS

LNGF 30

12

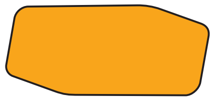
LNGF 36

12

LNGF 40

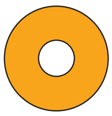
13

LNXR



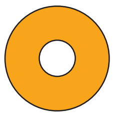
14

RNGH 38



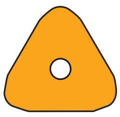
15

RNGH 50



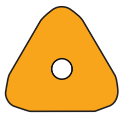
15

TNGJ 22



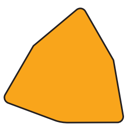
16

TNGJ 28



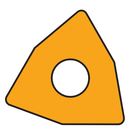
16

WNGF



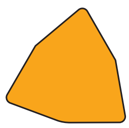
17

WNGU



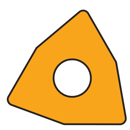
17

WNMF



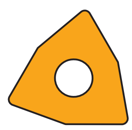
18

WNMJ

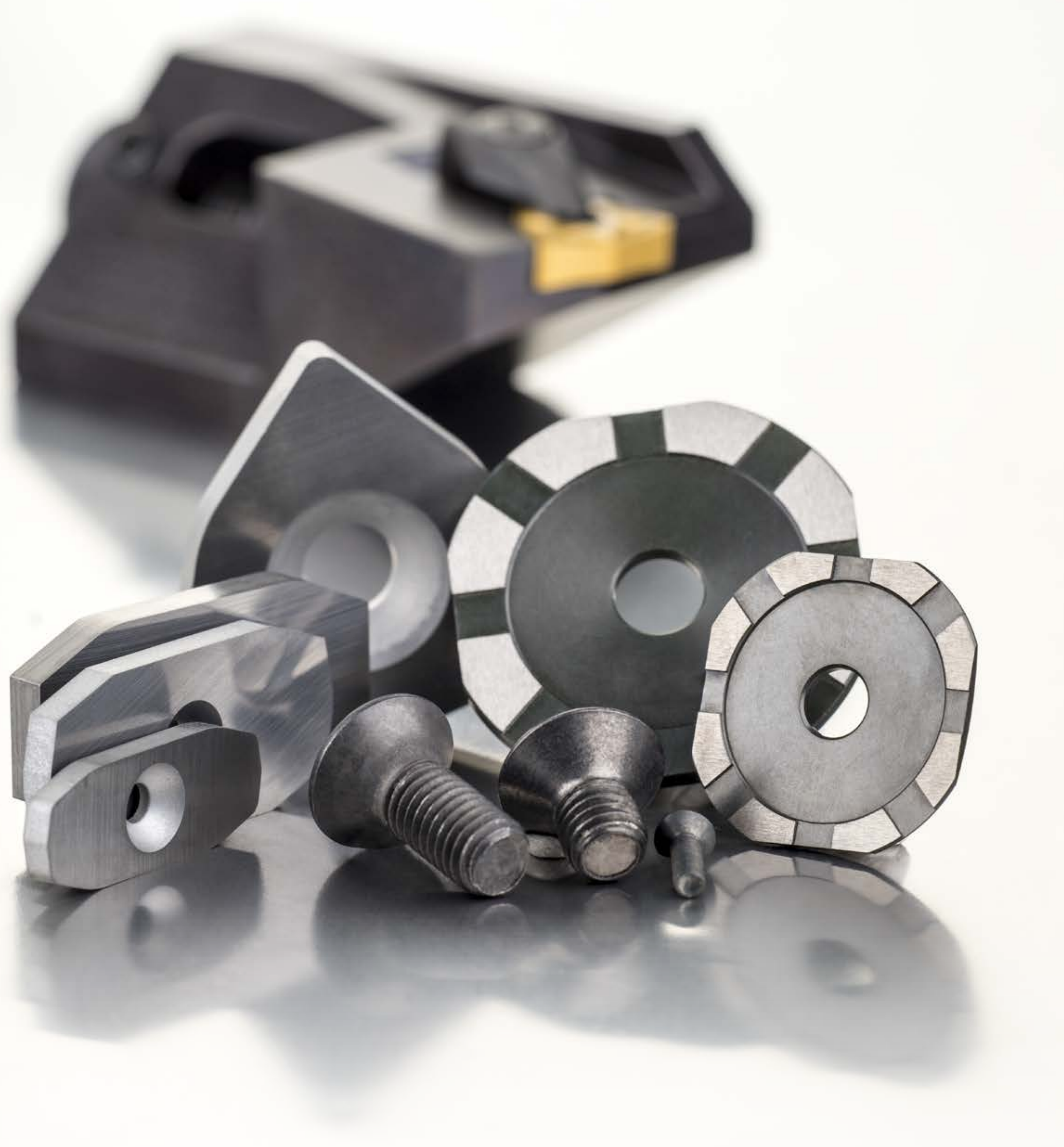


18

WNXJ



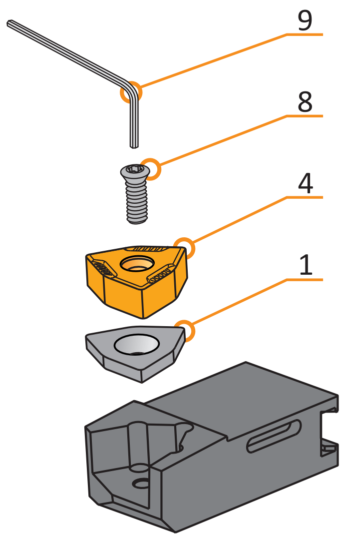
19

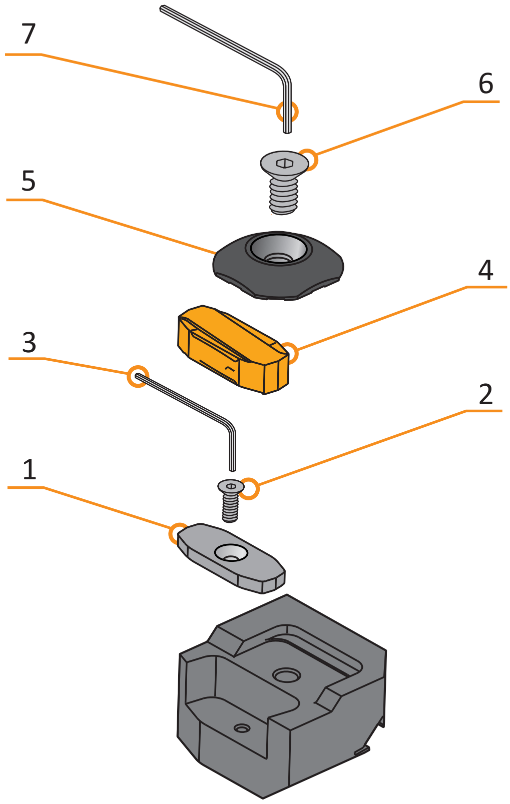


ACCESSORIES AND SPECIAL PRODUCTS

Bar Peeling inserts are generally mounted in two ways, either using a screw through the centre hole of the indexable insert or via a top clamp mechanism.

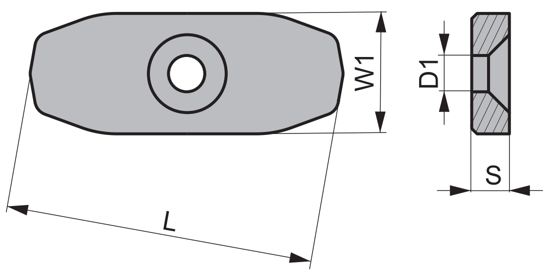
Most of our cassettes are equipped with a cemented carbide shim (washer) to protect the pocket and extend the life of the cassettes. The spare parts list for individual fastening systems is shown in the diagrams below and in the table of individual components.



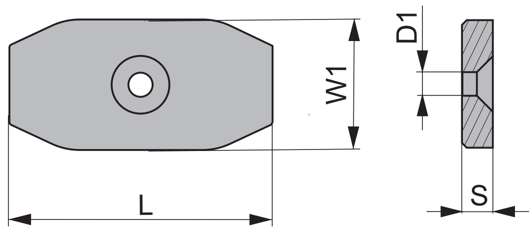


4	1	2	3	5	6	7	8	9
LNGF 3007..	LNW 300310	HCS 0308	HXK 2	UP 3005	HCS 0612	HXK 4	–	–
LNGF 3612..	LNW 360310	HCS 0308	HXK 2	UP 3005	HCS 0612	HXK 4	–	–
LNGF 4010..	LNW 400410	HCS 0310	HXK 2	UP 3005	HCS 0612	HXK 4	–	–
RNGH 3812..	RNX 380700	–	–	–	–	–	HCS 1030	HXK 6
WN.J 2013..	WNW 200615	–	–	–	–	–	US 8025-T30P	SDR T30P
WN.F 2013..	WNW 200615	HCS 0816	HXK 5	UP 4107	HCS 0820	HXK 5		–

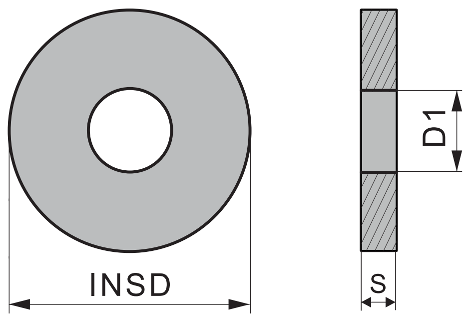
LNW 300310
LNW 360310



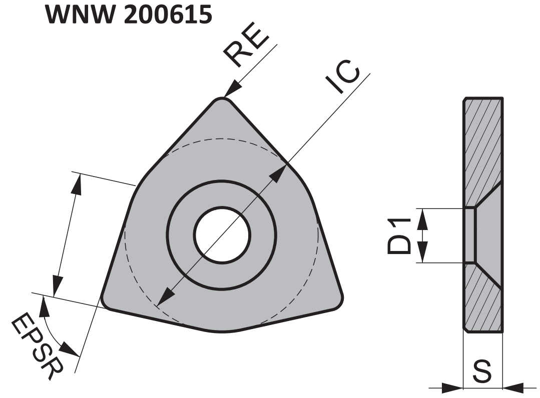
LNW 400410



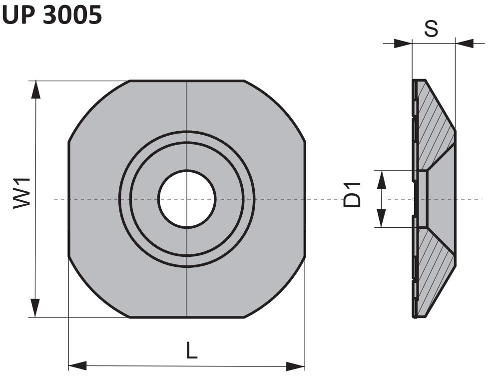
RNW 380700



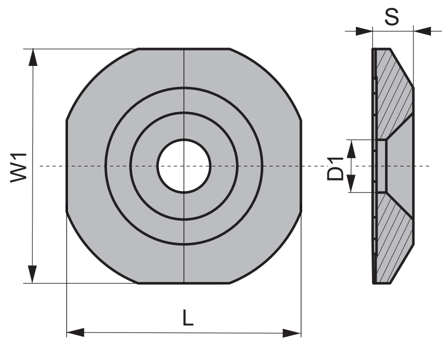
WNW 200615



UP 3005



UP 4107



	L	W1	S	D1	IC / INSD	EPSR
LNW 300310	29.75	11.60	3.50	3.50	–	–
LNW 360310	36.10	17.60	3.50	3.50	–	–
LNW 400410	39.70	19.70	4.75	3.50	–	–
WNW 200615	20.00	–	6.00	9.00	31.40	85
RNW 380700	–	–	7.00	11.15	37.75	–
UP 3005	27.00	27.00	4.70	6.50	–	–
UP 4107	38.20	38.20	6.40	8.60	–	–

SPECIAL TOOLS FOR SPECIMENS IN CHARPY TESTS

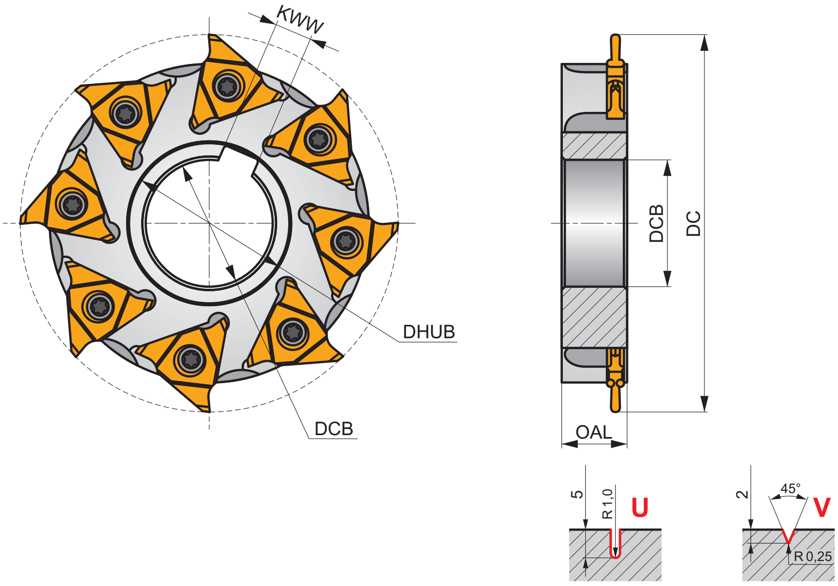
ST22N-1517-U+V

P M K

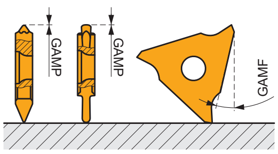
PRAMET S

Special Milling Cutter

Disc cutter utilizing tangentially clamped special inserts S-TNEW22- for machining specimens with groove shape U or V for Charpy impact testing. Available as a made-on request product.



KAPR	90°



h_m	0.03 - 0.05

	DC (mm)	OAL (mm)	DCB (mm)	DHUB (mm)	DCX (mm)	KWW (mm)	(mm)	GAMF (°)	GAMP (°)			max.		kg			
ISO 6462 DIN 8030 80X08R-STN22-1517	80	14.2	27	34	5	8	–	- 11	0	8	–	9000	–	0.28	S-TNEW 22-2501812 S-TNEW22-2501813	SPEC	–

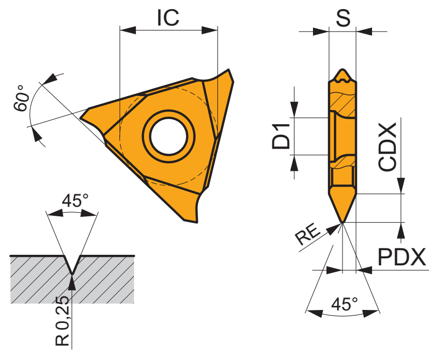
	V	U
	S-TNEW 22-2501812	S-TNEW22-2501813

		Nm			
	US 4011-T15P	3.5	M 4	10.6	SDR T15P

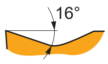
S-TNEW 22-V

PRAMET

	IC (mm)	D1 (mm)	S (mm)	RE (mm)	CDX (mm)	PDX (mm)
22	12.700	4.90	3.50	0.25	2.3	1.75



		RE (mm)	P vc (m/min)	P f (mm/tooth)	P ap (mm)	M vc (m/min)	M f (mm/tooth)	M ap (mm)	K vc (m/min)	K f (mm/tooth)	K ap (mm)	N vc (m/min)	N f (mm/tooth)	N ap (mm)	S vc (m/min)	S f (mm/tooth)	S ap (mm)	H vc (m/min)	H f (mm/tooth)	H ap (mm)
S-TNEW 22-2501812	**M8330**	0.25	■ 210	0.20	–	■ 125	0.20	–	■ 195	0.20	–	–	–	–	–	–	–	–	–	–



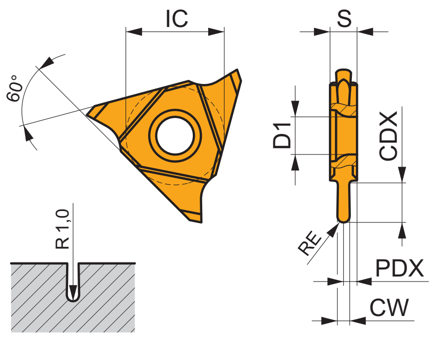
U E

Geometry dedicated for V shape of groove in specimens for Charpy impact tests.

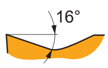
S-TNEW 22-U

PRAMET

	IC (mm)	D1 (mm)	S (mm)	RE (mm)	CW (mm)	CDX (mm)	PDX (mm)
22	12.700	4.90	3.50	1.00	2.00	5.10	1.75



		RE (mm)	P vc (m/min)	P f (mm/tooth)	P ap (mm)	M vc (m/min)	M f (mm/tooth)	M ap (mm)	K vc (m/min)	K f (mm/tooth)	K ap (mm)	N vc (m/min)	N f (mm/tooth)	N ap (mm)	S vc (m/min)	S f (mm/tooth)	S ap (mm)	H vc (m/min)	H f (mm/tooth)	H ap (mm)
S-TNEW22-2501813	**M8330**	1.00	■ 210	0.15	–	■ 125	0.15	–	■ 195	0.15	–	–	–	–	–	–	–	–	–	–



U E

Geometry dedicated for U shape of groove in specimens for Charpy impact tests.

ST22N-1504-U+V

P M K

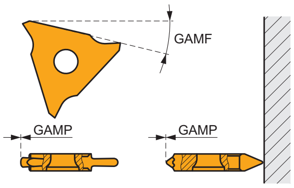
PRAMET

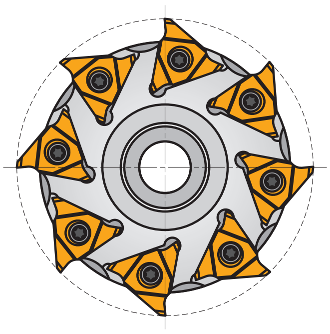
S

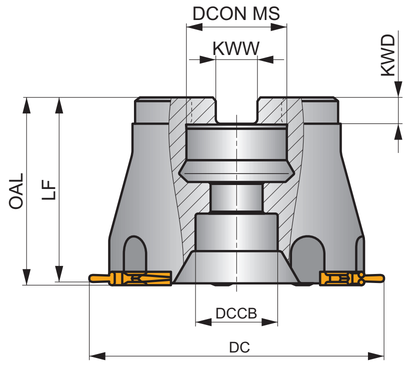
Special Milling Cutter

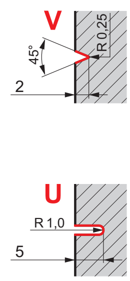
Arbor style cutter utilizing tangentially clamped special inserts S-TNEW22- for machining specimens with groove shape U or V for Charpy impact testing. Available as a made-on request product.

KAPR	90°









h_m	0.03 - 0.05

	DC (mm)	OAL (mm)	DCON MS (mm)	DCCB (mm)	LF (mm)	KWW	KWD	GAMF (°)	GAMP (°)			max.		kg			
80A08R-STN22-1504	80	50	27	38	49.2	12.4	7	-11	0	8	–	9000		0.99	S-TNEW 22-2501812 S-TNEW22-2501813	SPEC	AC001

	V	U
	S-TNEW 22-2501812	S-TNEW22-2501813

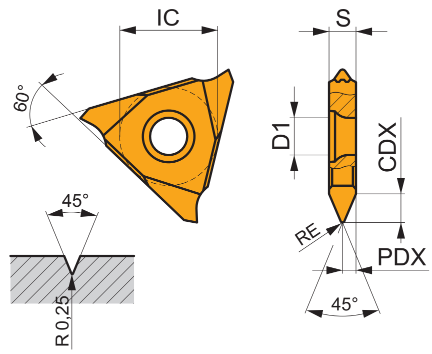
		Nm			
	US 4011-T15P	3.5	M 4	10.6	SDR T15P

AC001	KS 1230	K.FMH27

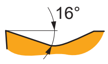
S-TNEW 22-V

PRAMET

☐	IC (mm)	D1 (mm)	S (mm)	RE (mm)	CDX (mm)	PDX (mm)
22	12.700	4.90	3.50	0.25	2.3	1.75



		RE (mm)	P vc (m/min)	P f (mm/tooth)	P ap (mm)	M vc (m/min)	M f (mm/tooth)	M ap (mm)	K vc (m/min)	K f (mm/tooth)	K ap (mm)	N vc (m/min)	N f (mm/tooth)	N ap (mm)	S vc (m/min)	S f (mm/tooth)	S ap (mm)	H vc (m/min)	H f (mm/tooth)	H ap (mm)
S-TNEW 22-2501812	M8330	0.25	■ 210	0.20	–	■ 125	0.20	–	■ 195	0.20	–	–	–	–	–	–	–	–	–	–

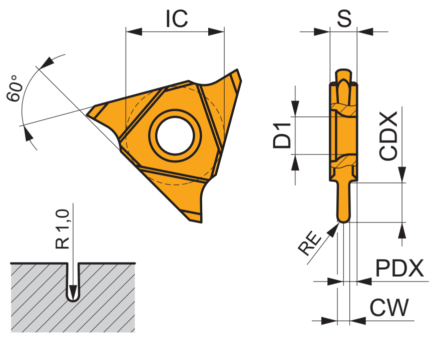


Geometry dedicated for V shape of groove in specimens for Charpy impact tests.

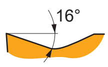
S-TNEW 22-U

PRAMET

☐	IC (mm)	D1 (mm)	S (mm)	RE (mm)	CW (mm)	CDX (mm)	PDX (mm)
22	12.700	4.90	3.50	1.00	2.00	5.10	1.75



		RE (mm)	P vc (m/min)	P f (mm/tooth)	P ap (mm)	M vc (m/min)	M f (mm/tooth)	M ap (mm)	K vc (m/min)	K f (mm/tooth)	K ap (mm)	N vc (m/min)	N f (mm/tooth)	N ap (mm)	S vc (m/min)	S f (mm/tooth)	S ap (mm)	H vc (m/min)	H f (mm/tooth)	H ap (mm)
S-TNEW22-2501813	M8330	1.00	■ 210	0.15	–	■ 125	0.15	–	■ 195	0.15	–	–	–	–	–	–	–	–	–	–

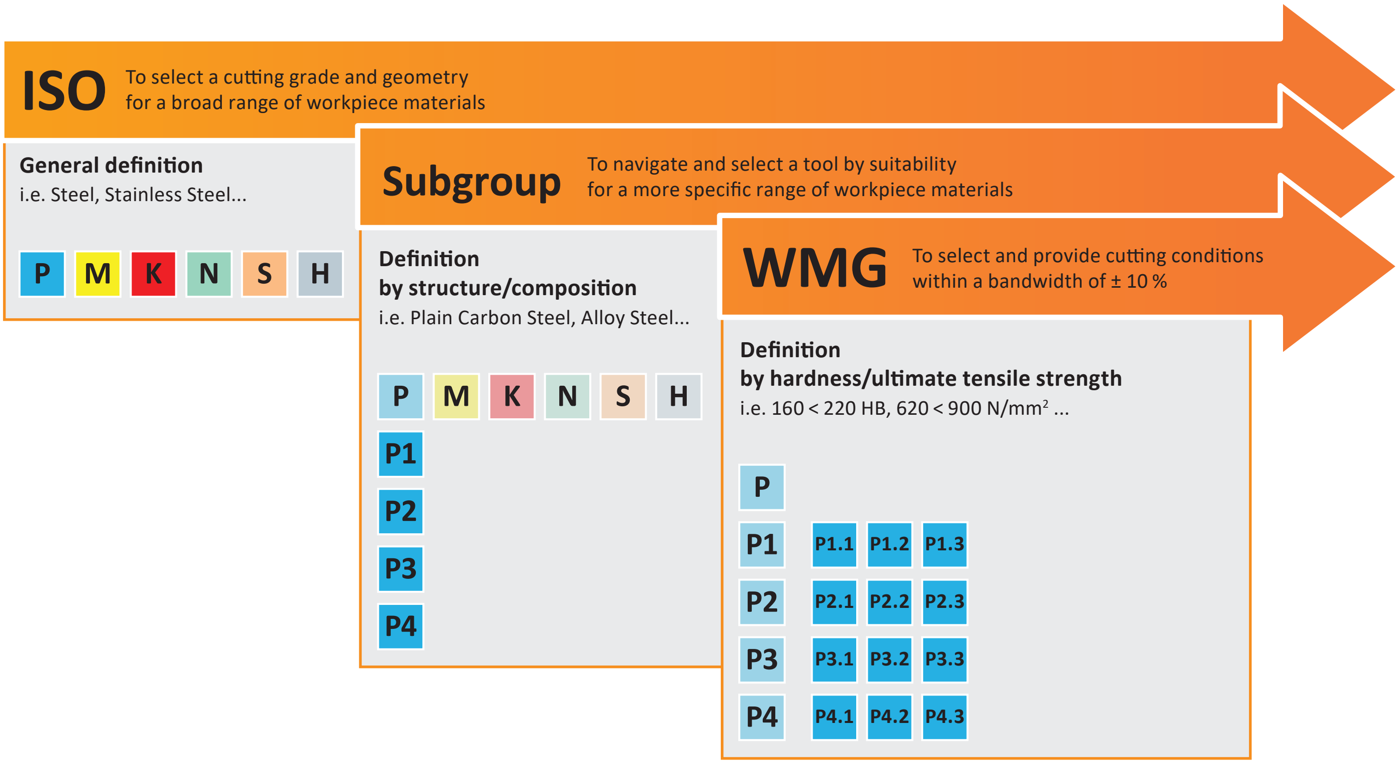


Geometry dedicated for U shape of groove in specimens for Charpy impact tests.

TECHNICAL INFORMATION

CONTENT OF TECHNICAL INFORMATION

Workpiece material groups (WMG)



About Dormer Pramet's workpiece material classification

Workpiece **M**aterial **G**roups (**WMG**) are used to support easy and reliable selection of the right cutting tool and starting values for machining conditions in particular applications.
Dormer Pramet classifies workpiece materials into six different coloured groups;

- **Blue**: Steel and cast steel (P-group)
- **Yellow**: Stainless steel (M-group)
- **Red**: Cast iron (K-group)
- **Green**: Non-ferrous metals (N-group)
- **Brown**: High-temperature alloys (S-group)
- **Grey**: Hardened materials (H-group)

Each of these are divided into subgroups on the basis of their structure and/or composition. For example, P-group steel and cast steel is split into four subgroups, namely;

- **P1** – Free machining steel
- **P2** – Plain carbon steel
- **P3** – Alloy steel
- **P4** – Tool steel

A final division includes material properties, such as hardness and ultimate tensile strength. This is to provide our customers with a complete tool recommendation, including starting values for cutting speed and feed.
The table on the next page includes a description of each workpiece material group, as well as examples of commonly used designations.

WMG (Work Material Group)

ISO group			WMG (Work Material Group)		Hardness (HB or HRC)	Ultimate Tensile Strength (MPa)
P	P1	P1.1	Free machining steel (carbon steels with increased machinability)	Sulfurized	< 240 HB	≤ 830
		P1.2		Sulfurized and phosphorized	< 180 HB	≤ 620
		P1.3		Sulfurized/phosphorized and leaded	< 180 HB	≤ 620
	P2	P2.1	Plain carbon steel (steels comprised of mainly iron and carbon)	Containing <0.25 % C	< 180 HB	≤ 620
		P2.2		Containing <0.55 % C	< 240 HB	≤ 830
		P2.3		Containing >0.55 % C	< 300 HB	≤ 1030
	P3	P3.1	Alloy steel (carbon steels with an alloying content ≤ 10%)	Annealed	< 180 HB	≤ 620
		P3.2		Hardened and tempered	180 – 260 HB	> 620 ≤ 900
		P3.3			260 – 360 HB	> 900 ≤ 1240
	P4	P4.1	Tool steel (special alloy steel for tools, dies and molds)	Annealed	< 26 HRC	≤ 900
		P4.2		Hardened and tempered	26 – 39 HRC	> 900 ≤ 1240
		P4.3			39 – 45 HRC	> 1240 ≤ 1450
M	M1	M1.1	Ferritic stainless steel (straight chromium non-hardenable alloys)		< 160 HB	≤ 520
		M1.2			160 – 220 HB	> 520 ≤ 700
	M2	M2.1	Martensitic stainless steel (straight chromium hardenable alloys)	Annealed	< 200 HB	≤ 670
		M2.2		Quenched and tempered	200 – 280 HB	> 670 ≤ 950
		M2.3		Precipitation-hardened	280 – 380 HB	> 950 ≤ 1300
	M3	M3.1	Austenitic stainless steel (chromium-nickel and chromium-nickel-manganese alloys)		< 200 HB	≤ 750
		M3.2			200 – 260 HB	> 750 ≤ 870
		M3.3			260 – 300 HB	> 870 ≤ 1040
	M4	M4.1	Austenitic-ferritic (DUPLEX) or super-austenitic stainless steel		< 300 HB	≤ 990
		M4.2	Precipitation hardening austenitic stainless steel		300 – 380 HB	≤ 1320
K	K1	K1.1	Gray iron or Automotive Gray iron (GG) (iron-carbon castings with a lamellar graphite microstructure)	Ferritic or ferritic-pearlitic	< 180 HB	≤ 190
		K1.2		Ferritic-pearlictic or pearlitic	180 – 240 HB	> 190 ≤ 310
		K1.3		Pearlitic	240 – 280 HB	> 310 ≤ 390
	K2	K2.1	Malleable iron (GTS/GTW) (iron-carbon castings with a graphite-free microstructure)	Ferritic	< 160 HB	≤ 400
		K2.2		Ferritic or pearlitic	160 – 200 HB	> 400 ≤ 550
		K2.3		Pearlitic	200 – 240 HB	> 550 ≤ 660
	K3	K3.1	Ductile iron (GGG) (iron-carbon castings with a nodular graphite microstructure)	Ferritic	< 180 HB	≤ 560
		K3.2		Ferritic or pearlitic	180 – 220 HB	> 560 ≤ 680
		K3.3		Pearlitic	220 – 260 HB	> 680 ≤ 800
	K4	K4.1	Austenitic gray iron (ASTM A436) (iron-carbon alloy castings with an austenitic lamellar graphite microstructure)		< 180 HB	≤ 190
		K4.2	Austenitic ductile iron (ASTM A439 or ASTM A571) (iron-carbon alloy castings with an austenitic nodular graphite microstructure)		< 240 HB	≤ 740
		K4.3	Austempered ductile iron (ASTM A897) (iron-carbon alloy castings with an ausferrite microstructure)		< 280 HB	> 840 ≤ 980
		K4.4			280 – 320 HB	> 980 ≤ 1130
		K4.5			320 – 360 HB	> 1130 ≤ 1280
	K5	K5.1	Compacted graphite iron CGI (ASTM A842) (iron-carbon castings with a vermicular graphite structure)	Ferritic	< 180 HB	≤ 400
		K5.2		Ferritic-pearlitic	180 – 220 HB	> 400 ≤ 450
		K5.3		Pearlitic	220 – 260 HB	> 450 ≤ 500
N	N1	N1.1	Commercially pure wrought aluminium		< 60 HB	≤ 240
		N1.2	Wrought aluminium alloys	Half hard tempered	60 – 100 HB	> 240 ≤ 400
		N1.3		Full hard tempered	100 – 150 HB	> 400 ≤ 590
	N2	N2.1	Cast aluminium alloys		< 75 HB	≤ 240
		N2.2			75 – 90 HB	> 240 ≤ 270
		N2.3			90 – 140 HB	> 270 ≤ 440
	N3	N3.1	Free-cutting copper-alloys materials with excellent machining properties		–	–
		N3.2	Short-chip copper-alloys with good to moderate machining properties		–	–
		N3.3	Electrolytic copper and long-chip copper-alloys with moderate to poor machining properties		–	–
	N4	N4.1	Thermoplastic polymers		–	–
		N4.2	Thermosetting polymers		–	–
		N4.3	Reinforced polymers or composites		–	–
	N5	N5.1	Graphite		–	–
S	S1	S1.1	Titanium or titanium alloys		< 200 HB	≤ 660
		S1.2			200 – 280 HB	> 660 ≤ 950
		S1.3			280 – 360 HB	> 950 ≤ 1200
	S2	S2.1	Fe-based high-temperature alloys		< 200 HB	≤ 690
		S2.2			200 – 280 HB	> 690 ≤ 970
	S3	S3.1	Ni-based high-temperature alloys		< 280 HB	≤ 940
		S3.2			280 – 360 HB	> 940 ≤ 1200
	S4	S4.1	Co-based high-temperature alloys		< 240 HB	≤ 800
		S4.2			240 – 320 HB	> 800 ≤ 1070
H	H1	H1.1	Chilled cast iron		< 440 HB	–
	H2	H2.1	Hardened cast iron		< 55 HRC	–
		H2.2			> 55 HRC	–
	H3	H3.1	Hardened steel < 55 HRC		< 51 HRC	–
		H3.2			51 – 55 HRC	–
	H4	H4.1	Hardened steel > 55 HRC		55 – 59 HRC	–
		H4.2			> 59 HRC	–

ISO group		Subgroup		WMG (Work Material Group)		k_{vG}	Examples of material (AISI, EN, DIN, ČSN, GB, SS, STN, BS, UNE, AFNOR, ASTM, GOST, UNS, UNI, ...)
P	**Steel and cast steel** (steels with alloy content ≤ 10 % and a hardness of < 45HRC)	P1	Free machining steel (carbon steels with increased machinability)	P1.1	Free machining sulfurized carbon steel with a hardness of < 240 HB	1.33	AISI **1108,** EN **15S22,** DIN **1.0723,** SS **1922,** ČSN **11120,** BS **210A15,** UNE **F.210F,** GB **Y15,** AFNOR **10F1,** GOST **A30,** UNI **CF10S20**
				P1.2	Free machining sulfurized and phosphorized carbon steel with a hardness of < 180 HB	1.49	AISI **1211,** EN **11SMn30,** DIN **1.0715,** SS **1912,** ČSN **11109,** BS **230M7,** UNE **F.2111,** GB **Y15,** AFNOR **S250,** GOST **A40G,** UNI **CF9SMn28**
				P1.3	Free machining sulfurized/phosphorized and leaded carbon steel with a hardness of < 180 HB	1.53	AISI **12L13,** EN **11SMnPb30,** DIN **1.0718,** SS **1914,** ČSN **12110,** BS **210M16,** UNE **F.2114,** GB **Y15Pb,** AFNOR **S250Pb,** GOST **AS35G2,** UNI **CF10SPb20**
		P2	Plain carbon steel (steels comprised of mainly iron and carbon)	P2.1	Plain low carbon steel containing < 0.25 % C with a hardness of < 180 HB	1.14	AISI **1015,** EN **C15,** DIN **1.0401,** SS **1350,** ČSN **11301 ,** BS **080A15,** UNE **F.111,** GB **15,** AFNOR **C18RR,** GOST **St2ps,** UNI **Fe360**
				P2.2	Plain medium carbon steel containing < 0.55 %C with a hardness of < 240 HB	1.00	AISI **1030,** EN **C30,** DIN **1.0528,** SS **1550,** ČSN **12031,** BS **080M32,** UNE **F.1130,** GB **30,** AFNOR **AF50C30,** GOST **30G,** UNI **Fe590**
				P2.3	Plain high carbon steel containing > 0.55 % C, with a hardness of < 300 HB	0.89	AISI **1060,** EN **C60,** DIN **1.0601,** SS **1655,** ČSN **12061,** BS **080A62,** UNE **F513,** GB **60,** AFNOR **1C60,** GOST **60G,** UNI **C60**
		P3	Alloy steel (carbon steels with an alloying content ≤ 10 %)	P3.1	Alloy steel with a hardness of < 180 HB	0.92	AISI **5015,** EN **16Mo3,** DIN **1.5415,** SS **2912,** ČSN **15020,** BS **1501-240,** UNE **F.2601,** GB **16Mo,** AFNOR **15D3,** GOST **15M,** UNI **16Mo3KW**
				P3.2	Alloy steel with a hardness of 180 – 260 HB	0.74	AISI **4140,** EN **42CrMo4,** DIN **1.7225,** SS **2244,** ČSN **15142,** BS **708M40,** UNE **F.8232,** GB **42CrMo,** AFNOR **42CD4,** GOST **40ChFA,** UNI **42CrMo4**
				P3.3	Alloy steel with a hardness of 260 – 360 HB	0.63	AISI **4140,** EN **42CrMo4,** DIN **1.7225,** SS **2244,** ČSN **15142,** BS **708M40,** UNE **F.8232,** GB **42CrMo,** AFNOR **42CD4,** GOST **40ChFA,** UNI **42CrMo4**
		P4	Tool steel (special alloy steel for tools, dies and molds)	P4.1	Tool steel with a hardness of < 26 HRC	0.55	AISI **D2,** EN **X155CrVMo12-1,** DIN **1.2370,** SS **2736,** ČSN **19573,** BS **BD2,** UNE **F.520A,** GB **Cr12Mo1V1,** AFNOR **Z160CDV12,** GOST **Ch12MF,** UNI **X155CrVMo121KU**
				P4.2	Tool steel with a hardness of 26 – 39 HRC	0.47	AISI **D2,** EN **X155CrVMo12-1,** DIN **1.2370,** SS **2736,** ČSN **19573,** BS **BD2,** UNE **F.520A,** GB **Cr12Mo1V1,** AFNOR **Z160CDV12,** GOST **Ch12MF,** UNI **X155CrVMo121KU**
				P4.3	Tool steel with a hardness of 39 – 45 HRC	0.38	AISI **D2,** EN **X155CrVMo12-1,** DIN **1.2370,** SS **2736,** ČSN **19573,** BS **BD2,** UNE **F.520A,** GB **Cr12Mo1V1,** AFNOR **Z160CDV12,** GOST **Ch12MF,** UNI **X155CrVMo121KU**

ISO group		Subgroup		WMG (Work Material Group)		k_{vG}	Examples of material (AISI, EN, DIN, ČSN, GB, SS, STN, BS, UNE, AFNOR, ASTM, GOST, UNS, UNI, ...)
M	**Stainless steel** (corrosion resistant steels with ≥ 11 % chromium content)	**M1**	Ferritic stainless steel (straight chromium non-hardenable alloys)	**M1.1**	Stainless steel, ferritic with a hardness of < 160 HB	1.22	AISI **S429,** EN **X7Cr14,** DIN **1.4001,** SS **2326,** BS **434S17,** UNE **F.3401,** AFNOR **Z8C12,** GOST **08Ch13,** UNI **X6CrTi12**
				M1.2	Stainless steel, ferritic with a hardness of 160 – 220 HB	1.03	AISI **446,** EN **X10CrAl24,** DIN **1.4762,** SS **2322,** ČSN **17113,** BS **430S17,** UNE **F.3154,** GB **10Cr17,** AFNOR **Z10CAS24,** GOST **12Ch17,** UNI **X16Cr26**
		M2	Martensitic stainless steel (straight chromium hardenable alloys)	**M2.1**	Stainless steel, martensitic with a hardness of < 200 HB	1.08	AISI **430F,** EN **X14CrMoS17,** DIN **1.4104,** SS **2383,** ČSN **17140,** BS **410S21,** UNE **F.3117,** AFNOR **Z10CF17,** UNI **X10CrS17**
				M2.2	Stainless steel, martensitic with a hardness of 200 – 280 HB	0.89	AISI **440C,** EN **X105CrMo17,** DIN **1.4125,** SS **2385,** ČSN **17023,** BS **425C11,** UNE **F.3402,** GB **102Cr17Mo,** AFNOR **Z100CD17,** GOST **95Ch18,** UNI **GX6CrNi 13 04**
				M2.3	Stainless steel, martensitic with a hardness of 280 – 380 HB	0.75	AISI **420,** EN **X45Cr13,** DIN **1.4034,** ČSN **17029,** BS **425C11,** UNE **F.3405,** AFNOR **Z44C14,** GOST **20X17H12,** UNI **X30Cr13**
		M3	Austenitic stainless steel (chromium-nickel and chromium-nickel-manganese alloys)	**M3.1**	Stainless steel, austenitic with a hardness of < 200 HB	1.00	AISI **304,** EN **X5CrNi18-12,** DIN **1.4303,** SS **2352,** ČSN **17249,** BS **305S17,** UNE **F.3513,** GB **10Cr18Ni12,** AFNOR **Z8CN18.12,** UNI **X7CrNi18 10**
				M3.2	Stainless steel, austenitic with a hardness of 200 – 260 HB	0.86	AISI **309,** EN **X15CrNiSi20-12,** DIN **1.4828,** ČSN **17251,** BS **309S24,** UNE **F.3312,** GB **1Cr23Ni13,** AFNOR **Z15CNS20.12,** GOST **20Ch20N14S2,** UNI **16CrNi23 14**
				M3.3	Stainless steel, austenitic with a hardness of 260 – 300 HB	0.77	AISI **5848,** EN **X45CrNiW18-9,** DIN **1.4873,** BS **331S40,** UNE **F.3211,** AFNOR **Z35CNWS14-4,** UNI **X45CrNiW 18 9**
		M4	Super-austenitic, Duplex or Precipitation Hardening stainless steel (austenitic alloys with > 20 % Ni, austenitic-ferritic microstructure or precipitation hardened)	**M4.1**	Stainless steel, austenitic-ferritic or super-austenitic with a hardness of < 300 HB	0.75	AISI **329,** EN **X1-NiCrMoCU25-20-5,** DIN **1.4539,** SS **2562,** ČSN **17265,** BS **318S13,** UNE **F.3552,** GB **022Cr25NiMo2N,** AFNOR **Z1NCDU25.20**
				M4.2	Stainless steel, precipitation hardening austenitic with a hardness of 300 – 380 HB	0.64	AISI **631 (17-7PH),** EN **X7CrNiAL17-7,** DIN **1.4568,** SS **2388,** ČSN **17465 ,** BS **301S13,** UNE **F.3217,** GB **07Cr17Ni7Al,** AFNOR **29CNA17-07,** GOST **09Ch17N7Ju1,** UNI **X53CrMnNiN21 9**

ISO group		Subgroup		WMG (Work Material Group)		k_{vG}	Examples of material (AISI, EN, DIN, ČSN, GB, SS, STN, BS, UNE, AFNOR, ASTM, GOST, UNS, UNI, ...)
K	**Cast Iron** (castings of iron and carbon alloys with > 2 % carbon content)	K1	Gray iron (GG) (iron-carbon castings with a lamellar graphite microstructure)	K1.1	Gray iron, ferritic or ferritic-pearlitic with a hardness of < 180 HB	1.35	ASTM **A48 Grade 20 (F11401)**, EN-**JL-100**, DIN **GG-10 (0.6010)**, SS **0110**, STN **422410**, BS **Grade 150**, UNE **FG10**, GB **HAT 100**, AFNOR **Ft10D**, GOST **SC 10**, UNI **G10**
				K1.2	Gray iron, ferritic-pearlitic or pearlitic with a hardness of 180 – 240 HB	1.00	ASTM **A48 Grade 30 (F12101)**, EN-**JL-1030**, DIN **GG-20 (0.6020)**, SS **0120**, STN **422420**, BS **Grade 220**, UNE **FG20**, GB **HT200**, AFNOR **Ft20D**, GOST **СЧ20**, UNI **G20**
				K1.3	Gray iron, pearlitic with a hardness of 240 – 280 HB	0.75	ASTM **A48 Grade 50 (F13501)**, EN-**JL-1060**, DIN **GG-35 (0.6035)**, SS **0135**, STN **422435**, BS **Grade 350**, UNE **FG35**, GB **HAT300**, AFNOR **Ft35D**, GOST **SC35**, UNI **G35**
		K2	Malleable iron (GTS/GTW) (heat-treated iron-carbon castings with a graphite-free microstructure)	K2.1	Malleable iron, ferritic with a hardness of < 160 HB	1.39	ASTM **A602 Grade M3210 (F20000)**, EN-**JM-1130**, DIN **GTS-35 (0.8135)**, SS **0815**, BS **B340/12**, UNE **Type A**, AFNOR **MN 35-10**, GOST **КЧ35-10**
				K2.2	Malleable iron, ferritic or pearlitic with a hardness of 160 – 200 HB	1.13	ASTM **A602 Grade M4504 (F20001)**, EN-**JM-1040**, DIN **GTS-50-05 (0.8045)**, BS **P50-05**, AFNOR **MB 45-7**
				K2.3	Malleable iron, pearlitic with a hardness of 200 – 240 HB	0.90	ASTM **A602 Grade M7002 (F20004)**, EN-**JM-1140**, DIN **GTS-45 (0.8145)**, SS **0854**, STN **422540**, BS **P 45-06**, UNE **Typ B**, AFNOR **MP 50-5**, GOST **КЧ45-7**, UNI **GMN 45**
		K3	Ductile iron (GGG) (iron-carbon castings with a nodular graphite microsturcture)	K3.1	Ductile (nodular/spheriodal) iron, ferritic with a hardness of < 180 HB	1.23	ASTM **A536 Grade 60-40-18 (F32800)**, EN-**JS-1030**, DIN **GGG-40 (0.7040)**, SS **0717**, STN **422304**, BS **420/12**, UNE **FGE 42-12**, GB **QT 400**, AFNOR **FGS 400-12**, GOST **ВЧ40**
				K3.2	Ductile (nodular/spheriodal) iron, ferritic or pearlitic with a hardness of 180 – 220 HB	0.94	ASTM **A536 Grade 80-55-06 (F33800)**, EN-**JS-1050**, DIN **GGG-50 (0.7050)**, SS **0727**, STN **422305**, BS **500/7**, UNE **FGE 50-7**, GB **QT 500-7**, AFNOR **FGS 500-7**, GOST **ВЧ50**
				K3.3	Ductile (nodular/spheriodal) iron, pearlitic with a hardness of 220 – 260 HB	0.76	ASTM **A536 Grade 100-70-03 (F34800)**, EN-**JS-1060**, DIN **GGG-60 (0.7060)**, SS **0732**, STN **422306**, BS **600/3**, UNE **FG70-2**, GB **QT 600-3**, AFNOR **FGS 600-3**, GOST **ВЧ60**
		K4	Austenitic or austempered ductile iron (Ni-Resist/ADI) (iron-carbon alloy castings with an austenitic or ausferrite microstructure)	K4.1	Austenitic cast iron with a hardness of < 180 HB	1.14	ASTM **A436 Type 1 (L-NiCuCr 15 6 2, F41000)**, EN-**JL-3011**, DIN **GGL-NiMn 13 7 (0.6652)**, SS **0523**, BS **Grade F1**, AFNOR **FGL-Ni13Mn7**, GOST **S-NiMn 13 7**
				K4.2	Austenitic cast iron with a hardness of 180 – 240 HB	0.86	ASTM **A439 Type D-2B (S-NiCr 20 3, F43001)**, EN-**JS-3021**, DIN **GGG-NiMn 23 4**, SS **0776**, BS **Grade S2M**, AFNOR **FGS Ni23 Mn4**, GOST **ЧН19Х3Ш**
				K4.3	Austempered ductile iron with a hardness of 240 – 280 HB	0.63	ASTM **A897 Grade 110-70-11**
				K4.4	Austempered ductile iron with a hardness of 280 – 320 HB	0.54	ASTM **A897 Grade 125-80-10**, EN-**JS-1100**, DIN **GGG-90 (5.3400)**
				K4.5	Austempered ductile iron with a hardness of 320 – 360 HB	0.45	ASTM **A897 Grade 2 (150-110-07)**, EN-**JS-1110**, DIN **GGG-100 (5.3403)**
		K5	Compacted graphite iron (CGI) (iron-carbon castings with a vermicular graphite structure)	K5.1	Vermicular, compacted graphite iron with a hardness of < 180 HB	1.29	ASTM **A842 Grade 300**, EN-**GJV-300**, DIN **GGV 30**, GOST **ЧВГ30**,
				K5.2	Vermicular, compacted graphite iron with a hardness of 180 – 220 HB	0.97	ASTM **A842 Grade 350**, EN-**GJV-350**, DIN **GGV 35 (5.2200)**, GOST **ЧВГ30**,
				K5.3	Vermicular, compacted graphite iron with a hardness of 220 – 260 HB	0.75	ASTM **A842 Grade 450**, EN-**GJV-450**, DIN **GGV 45**, GOST **ЧВГ45**,

ISO group		Subgroup		WMG (Work Material Group)		k_{vG}	Examples of material (AISI, EN, DIN, ČSN, GB, SS, STN, BS, UNE, AFNOR, ASTM, GOST, UNS, UNI, ...)
N	**Non-ferrous metals** (metals including alloys without an appreciable amount of iron)	N1	Wrought aluminium	N1.1	Pure aluminium and wrought aluminium alloys with a hardness of < 60 HB	1.33	UNS **A91200,** EN **AL99.6,** DIN **3.0205,** SS **4010,** STN **424009,** BS **1C,** UNE **L-3001,** GB **L5,** AFNOR **A4,** GOST **АДС,** UNI **3567**
				N1.2	Wrought aluminium alloys with a hardness of 60 – 100 HB	1.00	UNS **A93004,** EN **AlMn0.5Mg0.5,** DIN **3.0505,** SS **4054,** STN **424432,** BS **N31,** UNE **L-3831,** GB **LF2,** AFNOR **A-M1,** GOST **АМц,** UNI **3568**
				N1.3	Wrought aluminium alloys with a hardness of 100 – 150 HB	0.67	UNS **A95083,** EN **AlMg4.5Mn0.7,** DIN **3.3547,** SS **4140,** STN **424415,** BS **N8,** UNE **L-3321,** GB **AlMg4.5Mn,** AFNOR **A-G4.5Mn,** GOST **Amg 4.5,** UNI **P-AlMg4.4**
		N2	Cast aluminium	N2.1	Cast aluminium alloys with a hardness of < 75 HB	0.67	UNS **A02080,** EN **AlCu4S,** BS **LM11,** STN **424331,** UNE **Al Si1Cu,** GOST **AMg5K,** UNI **G-AlSi7Mg**
				N2.2	Cast aluminium alloys with a hardness of 75 – 90 HB	0.60	UNS **A02420,** EN **AlCu4Ni2Mg2,** SS **AlSi7MgFe,** BS **LM6,** STN **424519,** UNE **Al-7SiMg,** AFNOR **A-S7G,** GOST **AK7,** UNI **G-AlSi7Mg**
				N2.3	Cast aluminium alloys with a hardness of 90 < 140 HB	0.43	UNS **A03360,** EN **G-ALCu4NiMg2,** SS **ALSi10Mg,** STN **424336,** BS **LM 30,** AFNOR **A-S10G,** UNI **G-AlSi9Mg**
		N3	Copper or copper alloys	N3.1	Free-cutting copper-alloys materials with excellent machining properties	0.70	UNS **C14700,** EN **CuPb1P,** DIN **2.1498,** STN **423214,** BS **C111,** AFNOR **CuZn35Pb2,** GOST **L63-3,** UNI **CuS(P0.01)**
				N3.2	Short-chip copper-alloys with good to moderate machining properties	0.41	UNS **C81540,** EN **CuNi2SiCr,** DIN **2.0857,** STN **423220,** BS **NS113,** UNE **CuSn12,** AFNOR **CuZn40,** GOST **L60,** UNI **P-CuZn-40**
				N3.3	Electrolytic copper and long-chip copper-alloys with moderate to poor machining properties	0.21	UNS **C10100,** EN **CuAg0.1,** DIN **2.1203,** SS **5010,** UNE **CUSi3Mn1,** AFNOR **Cu-C2,** GOST **M1f,** UNI **Cu-OF**
		N4	Polymers (synthetic or semi-synthetic materials)	N4.1	Thermoplastic polymers	0.70	ABS, Acryl, Duraplast, Elastomer, EP, Epoxid, FEP, Fluor, Gummi, Kautschuk, Latex, MF, MPF, PA, PAI, PC, PE, PEEK, PEI, PES, PET, PF, Phenolharze, PI, PMMA, Polyamide, Polyester, Polyolefine, Polysulfon, POM, PP, PPE, PPS, PS, PSU, PTFE, PU, PUR, PVDF, SAN, SI, Styrol, UF, Ureol
				N4.2	Thermosetting polymers	0.27	Aramid, Epoxy, Fluoropolymer, Mehacrylate, Melamine, Phenolic, Polyester, Polyimide, Polymethacrylimide, Polyurethane
				N4.3	Reinforced polymers or composites	0.29	CFK, GFK, GMT, Honeycomb, Kevlar, LFT, Organo, SMC
		N5	Graphite	N5.1		1.0	CGM-1, CM-00, GM-10, GM-11, GR030, GR030PI, GR060, GR060PI, GR125, MC-01, MC-01RO, MC-03, MC-03M, IG11, IG-15, IG-32, IG-43, IG-45, IG-70, ISEM-1, ISEM-2, ISEM-3, R8340, R8500X, Technograph 15, Technograph 30, ISO-63, EDM C-3, EDM1, EDM3, ISO-90, ISO-93, ISO-95, R8510, R8650,

ISO group		Subgroup		WMG (Work Material Group)		k_{vG}	Examples of material (AISI, EN, DIN, ČSN, GB, SS, STN, BS, UNE, AFNOR, ASTM, GOST, UNS, UNI, ...)
S	**High-temperature alloys** (superalloys with high temperature stength and corrosion resistant surpassing that of stainless steel)	S1	Titanium or titanium alloys	S1.1	Titanium or titanium alloys, with a hardness of <200 HB	1.94	UNS **R50250 (Grade 1),** EN **Ti 99.6,** DIN **3.7035,** BS **TA.2,** UNE **Ti-Po2,** AFNOR **T-40,** GOST **BT1-00,** AISI **R50250, 3.7025, T35, 2TA1, R50400, 3.7035, 2TA2,**
				S1.2	Titanium alloys, with a hardness of 200 – 280 HB	1.72	UNS **R56404 (Grade 29),** EN **Ti2Cu,** DIN **3.7124,** BS **TA.21,** UNE **Ti-P11,** AFNOR **T-U2 ,** AISI **TA6V, Ti-6Al-4V, Ti 10.2.3, Ti5553**
				S1.3	Titanium alloys, a hardness of 280 – 360 HB	1.44	UNS **R54250 (Grade 38),** EN **TiAl6V4,** DIN **3.7165,** ČSN **TiAl6VELI,** BS **TA. 13,** UNE **Ti-P63,** AFNOR **T-A6V,** GOST **BT6,** AISI **TA6V, Ti-6Al-4V, Ti 10.2.3, Ti5553**
		S2	Fe-based high-temperature alloys	S2.1	High-temperature Fe-based alloys with a hardness of <200 HB	1.33	UNS **N08801 (Incoloy 801),** EN **X8 NiCrAlTi31-21,** DIN **1.4959,** BS **NA 15,** AFNOR **Z8NC33-21,** AISI **A-286, Discaloy, Haynes 556, Iconel 909, Greek Ascolloy**
				S2.2	High-temperature Fe-based alloys with a hardness of 200 – 280 HB	1.17	UNS **N19907,** EN **X6NiCrTiMoVB25-15-2,** DIN **1.4980,** SS **2570,** BS **HR52,** AFNOR **Z6NCTDV25.15B,** GOST **36НХТЮ,** AISI **A-286, Discaloy, Haynes 556, Iconel 909, Greek Ascolloy**
		S3	Ni-based high-temperature alloys	S3.1	High-temperature Ni-based alloys with a hardness of <280 HB	1.00	UNS **A09706 (Inconel 706),** EN **NiCr25FeAl,** DIN **2.4856,** BS **HR 6,** ČSN **Inconel 625,** UNE **F.3313,** GB **1Cr16Ni35,** AFNOR **NC22FeDNB,** GOST **XH38BT,** AISI **Inconel 718, 706 Waspalloy, Udimet 720, Inconel 625**
				S3.2	High-temperature Ni-based alloys with a hardness of 280 – 360 HB	0.83	UNS **N07001,** EN **NiCr20Co13Mo4Ti3Al,** DIN **2.4654,** BS **HR 2,** ČSN **Waspaloy,** AFNOR **NCKD 20ATV,** GOST **XH80ТБЮ,** AISI **Inconel 718, 706 Waspalloy, Udimet 720, Inconel 625**
		S4	Co-based high-temperature alloys	S4.1	High-temperature Co-based alloys with a hardness of <240 HB	0.78	UNS **R30016 (Stellite 6b),** EN **CoCr20W15Ni,** DIN **2.4964,** AFNOR **KC 20 WN,** GOST **ЛК52,** AISI **Haynes 25, Stellite 21, 31**
				S4.2	High-temperature Co-based alloys with a hardness of 240 – 320 HB	0.67	UNS **R30016 (Stellite 6b),** EN **CoCr20W15Ni,** DIN **2.4964,** AFNOR **KC 20 WN,** GOST **ЛК52,** AISI **Haynes 25, Stellite 21, 31**

ISO group		Subgroup		WMG (Work Material Group)		k_{vG}	Examples of material (AISI, EN, DIN, ČSN, GB, SS, STN, BS, UNE, AFNOR, ASTM, GOST, UNS, UNI, ...)
H	**Hardened materials** (any engineering metal with a hardness > 45 HRC)	**H1**	Chilled cast iron	**H1.1**	Chilled cast iron with a hardness of < 440 HB	1.52	UNS **F45001,** EN-**GJS-1050-6,** DIN **5.3406,** SS **0512,** BS **Grade 2A**
		H2	Hardened cast iron	**H2.1**	Hardened cast iron with a hardness < 55 HRC	0.90	UNS **F45003,** EN-**GJS-1400-1,** DIN **5.3405,** SS **0457,** BS **Grade 3D**
				H2.2	Hardened cast iron with a hardness > 55 HRC	0.77	UNS **F45003,** EN **G-X260NiCr4-2,** DIN **0.9620,** SS **0466,** BS **Grade S**
		H3	Hardened steel < 55 HRC	**H3.1**	Hardened steel with a hardness of < 51 HRC	1.00	AISI **4135,** EN **34CrMo4,** DIN **1.7220,** SS **2234,** STN **415131,** BS **198,** UNE **F.1250,** GB **35CrMo,** AFNOR **35CD4,** GOST **AC38XГM,** UNI **35CrMo4KB**
				H3.2	Hardened steel with a hardness of 51 – 55 HRC	0.82	AISI **4135,** EN **34CrMo4,** DIN **1.7220,** SS **2234,** STN **415131,** BS **198,** UNE **F.1250,** GB **35CrMo,** AFNOR **35CD4,** GOST **AC38XГM,** UNI **35CrMo4KB**
		H4	Hardened steel > 55 HRC	**H4.1**	Hardened steel with a hardness of 55 – 59 HRC	0.64	UNS **T31501,** EN **100MnCrW4,** DIN **1.2510,** SS **2140,** STN **419413,** BS **B01,** UNE **F.5220,** GB **9CrWMn,** AFNOR **90MWCrV5,** GOST **9XBГ,** UNI **95MNWCr5KU**
				H4.2	Hardened steel with a hardness of > 59 HRC	0.54	UNS **T31501,** EN **100MnCrW4,** DIN **1.2510,** SS **2140,** STN **419413,** BS **B01,** UNE **F.5220,** GB **9CrWMn,** AFNOR **90MWCrV5,** GOST **9XBГ,** UNI **95MNWCr5KU**

INDEXABLE MILLS – TECHNICAL INFO

Group	Cemented carbide with MTCVD	Cemented carbide with PVD	Uncoated
P01		M8310	
P05	M9315	M8310	
P10	M9315, M9325	M8310, 8215	
P15	M9315, M9325	8215	
P20	M9315, M9325	8215, M8330	
P25	M9315, M9325	M8330, M8340	
P30	M9325	M8330, M8340	
P35		M8330, M8340, M8345	
P40		M8330, M8340, M8345	
P45		M8340, M8345	
P50		M8340, M8345	

Group	Cemented carbide with MTCVD	Cemented carbide with PVD	Uncoated
M01			
M05			
M10			
M15			
M20		M6330, M8340	
M25		M6330, M8340	
M30	M9340	M6330, M8340, M8345	
M35	M9340	M6330, M8340, M8345	
M40	M9340	M8340, M8345	
M45			
M50			

Group	Cemented carbide with MTCVD	Cemented carbide with PVD	Uncoated
K01		M8310	
K05	M5315	M8310	
K10	M5315	8215	
K15	M5315	8215	
K20	M5315	8215, M8330	
K25	M5315	8215, M8330	
K30		M8330	
K35		M8330	
K40		M8330	
K45			
K50			

Group	Cemented carbide with MTCVD	Cemented carbide with PVD	Uncoated
N01			
N05			
N10		8215	
N15		8215	
N20		8215	
N25		8215	
N30			
N35			
N40			
N45			
N50			

Group	Cemented carbide with MTCVD	Cemented carbide with PVD	Uncoated
S01			
S05			
S10			
S15	M9340		
S20	M9340	M6330, M8340, M8345	
S25		M6330, M8340, M8345	
S30		M6330, M8340, M8345	
S35			
S40			
S45			
S50			

Group	Cemented carbide with MTCVD	Cemented carbide with PVD	Uncoated
H01			
H05	M5315	M8310	
H10	M5315	M8310, 8215	
H15	M5315	M8310, 8215	
H20	M5315	8215	
H25			
H30			
H35			
H40			
H45			
H50			

Grade Identification	Area of Application	Application	Feed	Cutting speed	Resistance to adverse Working Conditions	Coating	Colour	Substrate	Coolant benefit	Grade description
M5315	P05 – P20 K05 – K25 H05 – H20	◩ ■ ■				MT-CVD		H	- - -	One of the most abrasion-resistant milling grades which should be used under stable conditions. Its main advantage is the extremely high resistance to thermal stress and abrasive K05 – K25 wear. It is mainly used for machining hard and very hard materials, particularly cast iron.
M5326	P05 – P25 K10 – K30 H10 – H20	◩ ■ ◩				MT-CVD		H	- - -	Milling grade with high abrasion resistance even at high thermal loads, main application area is grey and ductile cast iron with relatively high cutting speeds with medium depths of cut.
M9315	P05 – P25 K10 – K30 H10 – H20	■ ■ ◩				MT-CVD		H	- - -	Milling grade with high abrasion resistance even at high thermal loads, main application area is higher cutting speeds with medium or small depths of cut.
M9325	P10 – P30 K10 – K30 H15 – H20	■ ■ ◩				MT-CVD		H	- - -	This grade has an ideal balance between wear resistance and toughness, it is mainly designed for roughing operations. Advantages are excellent wear resistance even at relatively high cutting speeds with excellent reliability, this grade is more suitable for applications using higher speeds and lower feed rates.
M9340	P35 – P50 M30 – M40 S15 – S20	■ ■ ■				MT-CVD		H	- - -	A very tough grade, where the main advantage is the high strength of the cutting edge and resistance to adverse cutting conditions. Although this material has an MT-CVD M30 – M40 coating, it is possible to use emulsion cooling for its application, especially in optimum cutting conditions.
M6330	P20 – P35 M20 – M35 S20 – S30	■ ■ ■				PVD		H	+ / -	Milling grade with extraordinary service reliability. Especially suitable for machining of hard to machine materials. Powerful in applications where unfavourable conditions and heavy cuts dominate.
M8310	P01 – P10 M01 – M10 K01 – K10 H05 – H15	■ ◩ ■ ◩				PVD		ultra submicron H	-	Grade specially developed for copy milling, featuring high resistance to abrasion. It is suitable for machining at higher cutting speeds under stable cutting conditions, and for machining virtually all groups of machined materials (particularly stronger and harder materials).
8215	P10 – P20 M10 – M20 K10 – K25 N10 – N25 S10 – S15 H10 – H15	■ ◩ ■ ■ ◩ ■				PVD		submicron H	+ / -	One of the most versatile milling grades, in terms of both the range of workpiece materials and the range of possible applications. It is characterised by high wear resistance and operational reliability. Its other advantages include excellent resistance to cracking induced by temperature shock. With its unique properties, this material is undoubtedly one of the pillars of the milling range.
M8325	P20 – P40 M15 – M30	■ ◩				PVD		S	-	The main application area of this grade is machining all kinds of steels (including stainless) in the "soft state". It can also be used for machining softer cast irons. Suitable for M15 – M30 machining at medium speeds under average cutting conditions.
M8326	P20 – P40 M15 – M30	■ ◩				PVD		H	-	Special grade for heavy duty. The main application area of this grade is machining all kinds of steels (including stainless) in the „soft state". It can also be used for machining softer cast irons. Suitable for M15 – M30 machining at medium speeds under average cutting conditions.
M8330	P20 – P40 M20 – M35 K20 – K40 N15 – N30 S15 – S25 H15 – H25	■ ■ ■ ◩ ◩ ◩				PVD		submicron H	+ / -	This grade is universal and can be used for machining various types of materials. However, it's priority application area lies within steels and ductile cast irons. It is recommended for milling at medium speeds under unstable cutting conditions.

⏮

Grade Identification	Area of Application	Application	Feed	Cutting speed	Resistance to adverse Working Conditions	Coating	Colour	Substrate	Coolant benefit	Grade description
M8340	P25 – P50	■	5	2	5+	PVD		submicron H	+ / -	One of the toughest grade dedicated for machining with lower cutting speed and unfavorable conditions. This grade is ideal for all operations where the main requirement is for a tough cutting edge.
	M20 – M40	■								
	K20 – K40	◩								
	S20 – S30	■								
M8345	P30 – P50	■	5	2	5+	PVD		H	-	This grade has exceptional operational reliability and is designed for heavy cuts in unfavourable conditions in difficult and tough materials.
	M30 – M40	■								
M8346	P30 – P50	■	5	2	5+	PVD		H	-	This grade has exceptional operational reliability and is designed for heavy cuts in unfavourable conditions in difficult and tough materials with low to medium cutting speed
	M30 – M40	◩								
S26	P15 – P30	■	5	1	4	–		S	+ +	Uncoated milling grade with excellent resistance to erosion of the cutting face. It is intended solely for machining carbon and alloy steels at low cutting speeds.

Substrate	
H	WC-Co based substrate
submicron H	WC-Co based substrate, fine-grained (< 1 µm)
ultra submicron H	WC-Co based substrate, very fine-grained (< 0.5 µm)
S	Substrate with cubic carbides

Coating	
MT-CVD	Medium-temperature chemical method of coating
PVD	Low-temperature physical method of coating
-	Uncoated grade

Coolant Benefit	
- - -	Very negative effect on tool life – cooling is not recommended
-	Slightly negative effect on tool life
+ / -	Influence of cooling may be both positive and negative – decisive factor is specific working conditions
+ +	Positive effect on tool life – cooling is recommended

Level of influence	
	Level 1 – 5

TECHNICAL INFORMATION – INDEXABLE MILLING– CORRECTION FACTORS

Correction factors for specific type of cutter and operation C_{VcO}

x.V	●	●	✖
Face mills with *KAPR* 45° – 60° and negative inserts (SHN06C, SHN09C, CHN09, …)	1.15	1.00	0.85
Face mills with *KAPR* 45° and positive inserts (SOE06Z, SOE09Z, SOD05,...)	1.15	1.00	0.85
Shoulder mills with *KAPR* 90° (SAD07D, SAD11E, SAD16E, SLN12, SLN16..)	1.10	1.00	0.90
Copy face mills (SRC10 – SRC20, SRD05 – SRD16, …)	1.10	1.00	0.90
Copy end mills (K2-PPH, K2-SLC, K2-SRC, K3-CXP…)	1.10	1.00	0.90
Disc mills (S90CN(XN), S90SN...)	1.10	1.00	0.90
Shoulder mills with extended flute J(T)-CSD12X, J(T)-SAD11E, J(T)-SAD16E...)	1.25	1.00	0.80
Face mills for heavy duty (FSB22X, SPN13..)	1.30	1.00	0.85
Shoulder mills for heavy duty (FTB27X..)	1.25	1.00	0.85

Correction factors for required durability C_{VcT}

minutes	15	20	30	45	60	90	120
General machining operations (fine finishing up to roughing)	1.23	1.13	1.00	0.89	0.81	0.72	–
Heavy machining operations (heavy roughing)	–	–	1.23	1.13	1.00	0.89	0.81

Additional correction factors C_{VcA}

Machining environment	C_{VcA}
Condition of the work-material (hard skin due to forging or casting)	0.70
Unstable machining conditions	0.85
Common machining conditions	1.00
Stable machining conditions	1.20

Correction factors for cutting speed when face and shoulder milling with < 100 % radial immersion C_{VcRCT}

a_e/DC	5 %	10 %	15 %	20 %	25 %	30 %	40 %	50 %	60 %	70 %	75 %	80 %	90 %	100 %
x.V	**1.48**	**1.35**	**1.27**	**1.22**	**1.19**	**1.16**	1.11	1.08	1.05	1.03	**1.00**	**1.00**	**1.00**	**1.00**

Correction factors to compensate for chip-thinning when face and shoulder milling with < 100 % radial immersion Cf_{zRCT}

a_e/DC	5 %	10 %	15 %	20 %	25 %	30 %	40 %	50 %	60 %	70 %	75 %	80 %	90 %	100 %
⇨x.f	**2.20**	**1.60**	**1.35**	**1.20**	**1.10**	0.95	0.85	0.75	0.85	0.95	**1.00**	**1.00**	**1.00**	**1.00**
⇨x.f	0.64	0.64	0.64	0.64	0.64	0.65	0.65	0.67	**0.68**	**0.71**	**0.72**	**0.74**	**0.79**	**1.00**

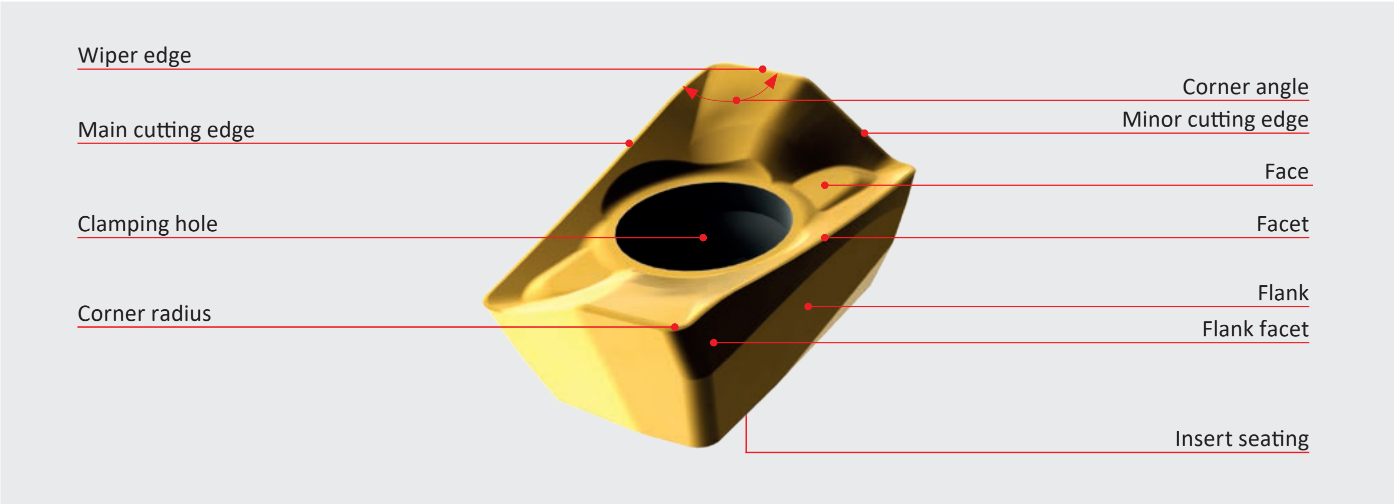
Resulting corrected cutting speed *vcc*

$vcc = v_c \times kvG \times C_{VcO} \times CvcT \times C_{VcA} \times C_{VcRCT} \times Cf_{zRCT}$

k_{vG} – coefficient of used material

v_c – starting speed from catalogue page

Parts of an Indexable Insert



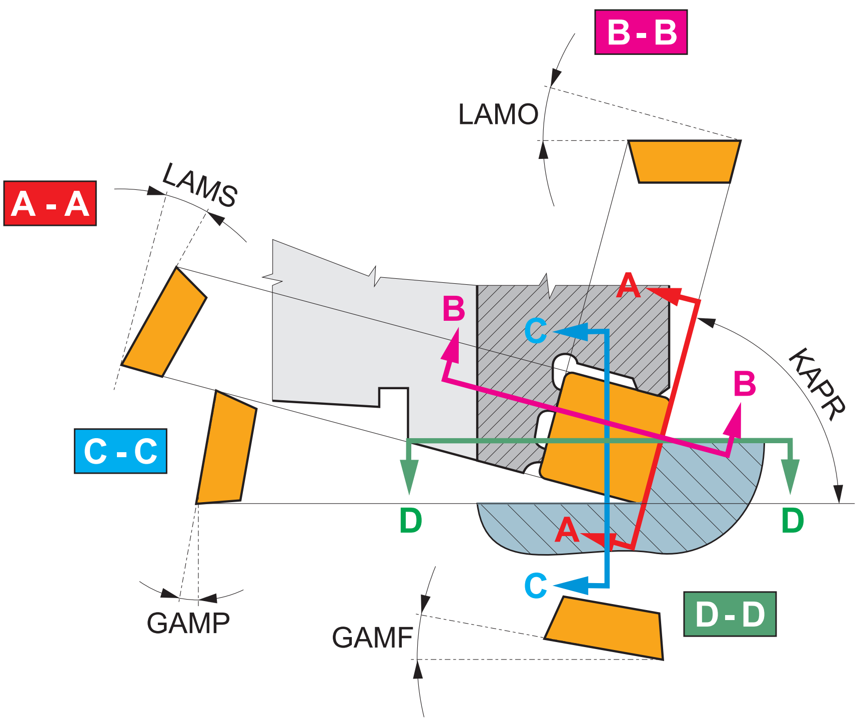
Geometry of milling tool

Constructional angles determine the basic orientation of the seat position that the cutting insert is clamped in and are therefore important for the design of the milling cutter body. There are two angles: axial face angle *GAMP* – γ_p (tool back rake) and radial face angle *GAMF* – γ_f (tool side rake) – see picture below.
Working angles are the setting angle *KAPR* – κ_r, the orthogonal face angle *GAMO* γ_o and the rake angle of the cutting edge *LAMS* – λ_s.

- **Orthogonal face angle** *GAMO* – γ_o affects not only the extent of plastic deformation of the cut chip but also the cutting force and temperature. The bigger the rake angle *GAMO* – γ_o, the lower the cutting force and power demand of the spindle motor (and vice versa).
- **Setting angle** *KAPR* – κ_r determines the thickness of the chip at a specific feed per tooth f_z and axial depth of cut a_p. It therefore affects cutting forces, specifically load, wear and tool service life. Reducing the setting angle *KAPR* – κ_r at a constant feed f_z causes a decrease in the chip thickness h.
- **Rake angle of cutting edge** *LAMS* – λ_s together with setting angle *KAPR* – κ_r and face angle *GAMO* – γ_o, this determines the point of first contact between the edge and work piece. That is why it affects the resistance of the edge to chipping during interrupted cut. At the same time, it affects the direction of chip evacuation.

Working angles of the tool you can determine the bed using the formulas or diagrams below.

Working and constructional angles of milling tool



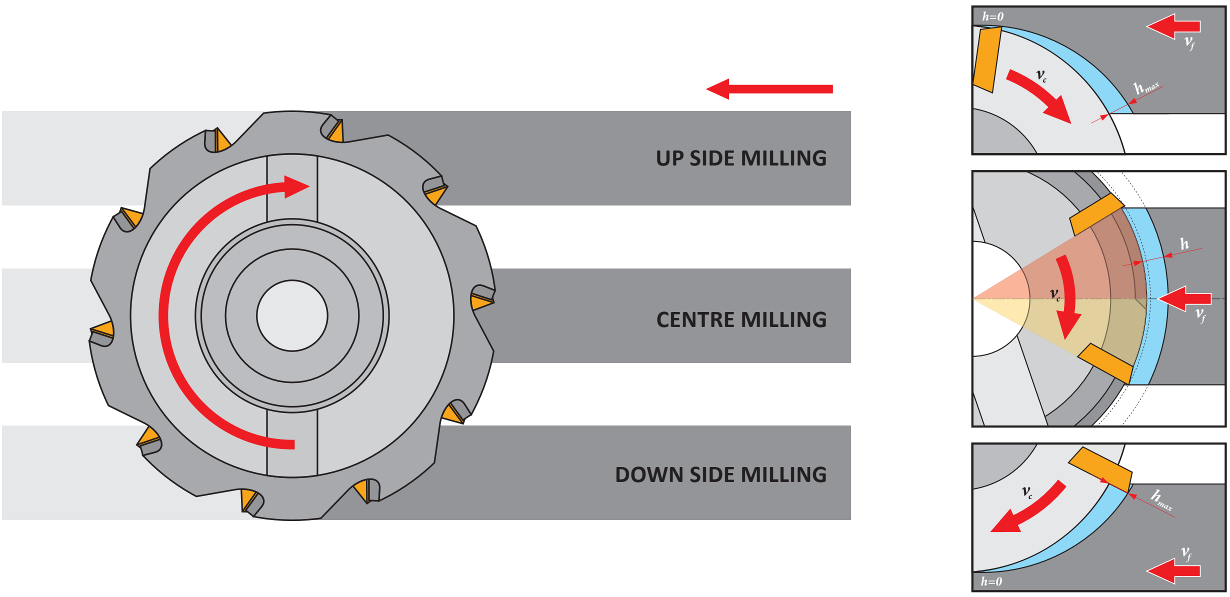
TECHNICAL INFORMATION – WORKING CONDITION WHEN MILLING

When performing a milling operation, the edge of the milling cutter almost always makes interrupted (intermittent) cuts. Each edge enters and exits the workpiece at least once within a single revolution of the tool.

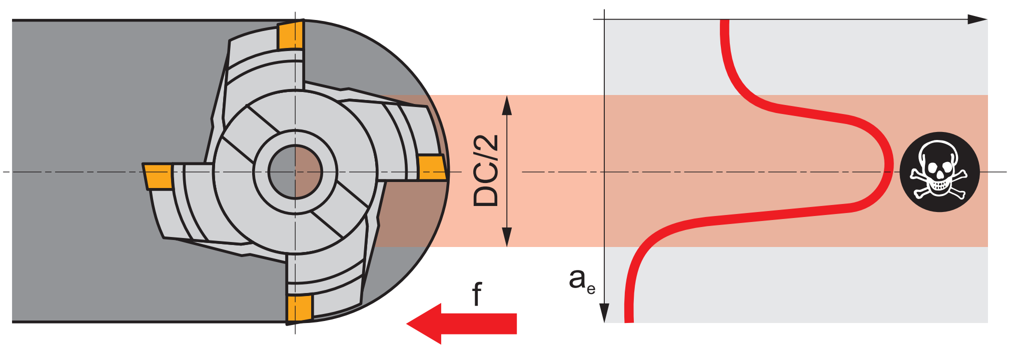
In addition, a periodic change in chip thickness takes place during each revolution of the milling cutter. This results in fluctuations in the size and direction of the tangential component of the cutting force. The edge of the milling cutter is thus subjected to cyclic stress which results in specific wear. The durability of the milling cutter edge is therefore dependent on the conditions in which the edge enters and exits the workpiece. Proper choice of these conditions significantly affects the milling process and its results in terms of cutting power and quality of the machined surface.

At the moment the edge enters or exits the workpiece, the edge is subjected to more or less intense mechanical shock which causes mechanical stress in the immediate vicinity of the cutting edge. If engagement conditions are chosen incorrectly, this shock can cause brittle damage to the edge, in the form of either fracturing or crumbling of the edge.

Position of the milling cutter relative to the workpiece is thus a very important factor. There are essentially three possible milling cutter positions: side up milling, centre milling and side down milling. For indexable tools, we recommend using co-directional engagement (so that the cutter forms thick chips on entry and thin chips on exit). However, there are notable exceptions (workpieces with surface skin, machines with worn feed screws...).

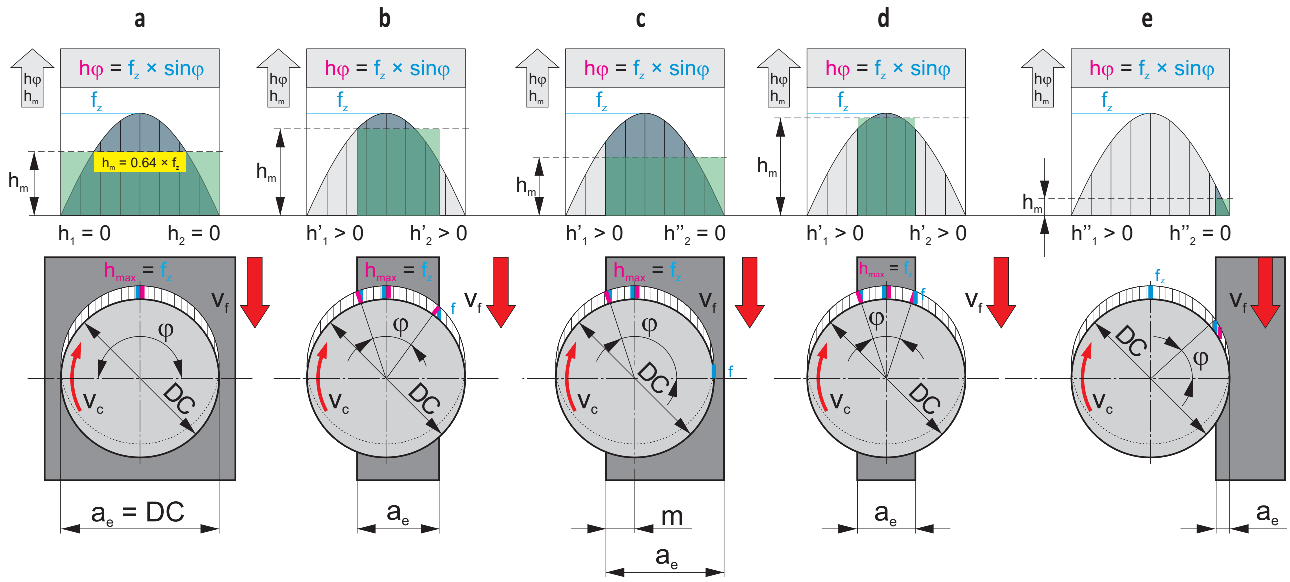


During face milling, where the width of the milled surface a_e is equal to the diameter of the milling cutter, follow the values recommended specifically for the inserts. If the engagement width is less than the diameter of the milling cutter, then the key factor is whether we machine with the centre or the side of the milling cutter, as mentioned above. In both cases, corrections in feed and cutting speed should be made (see correction tables on page XXX). Either way, we should try to ensure that the tool does not enter or exit the cut in an area close to the centre of the milling cutter (so-called dead zone).



When the edge exits from the cut, this is accompanied by both stressing of the edge due to rapid cooling of the surface layers of the insert near the cutting edge and by mechanical shock caused by the release of flexible deformations, particularly in the surface layers of the workpiece after a rapid decrease in cutting force.

As stated above, chip thickness h changes during a single revolution depending on the angle φ in line with the formula $h\varphi = f_z \times \sin\varphi$. Maximum chip thickness with steady f_z is reached within the axis of the milling cutter. The average thickness of a chip h_m removed by one tooth during one revolution is calculated as the height of a rectangle with the same area as the area under a sine curve relative to the radial depth of cut a_e. Average chip thickness h_m is dependent on the type of milling cutter and on engagement conditions, particularly the ratio of a_e/DC, feed per tooth f_z and naturally also on the entering angle $KAPR - \kappa_r$. The following figure shows illustrative examples.



Average chip thickness h_m for milling (with the centre) in accordance with figure a, b, d is calculated based on the formula:

$$h_m = f_z \times \sin\kappa_r \times \left(57.3 \frac{a_e}{DC \times \arcsin \times \left(\frac{a_e}{DC}\right)} \right)$$

Average chip thickness h_m for machining with the side of the milling cutter (figure c, e) is calculated based on the formula:

$$h_m = f_z \times \sin\kappa_r \times 114.6 \times \left(\frac{a_e}{DC \times \arccos \times \left(1 - \frac{2a_e}{DC}\right)} \right)$$

For milling with the side of the cutter in line with figure e, where the a_e/DC ratio is very low (< 0.2), average chip thickness h_m can be calculated using the simplified formula:

$$h_m = f_z \times \sin\kappa_r \times \sqrt{\frac{a_e}{DC}}$$

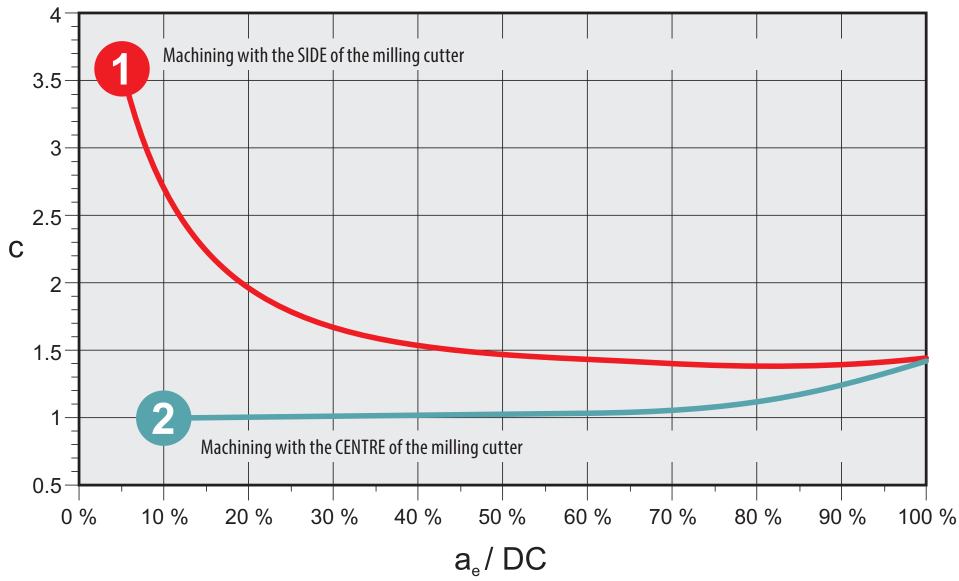
Where:

h_m	Is average chip thickness (mm)
f_z	Feed per tooth (mm/tooth)
a_e	Radial depth of cut (mm)
DC	Diameter of the milling cutter (mm)
κ_r	Entering angle of the main cutting edge *KAPR* (°)

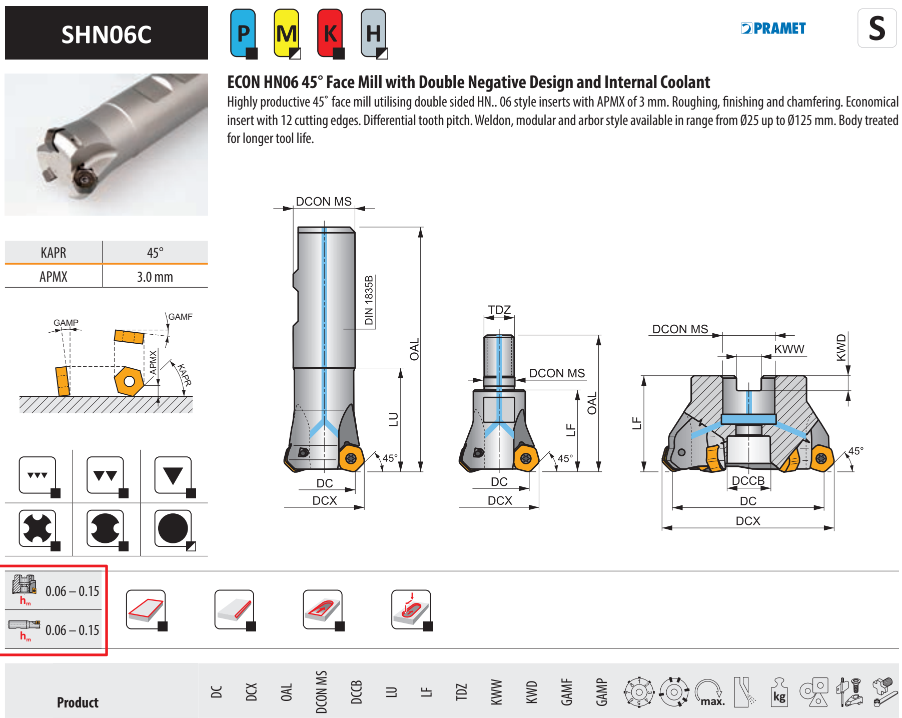
TECHNICAL INFORMATION – WORKING CONDITION WHEN MILLING

For optimal application of any milling tool, we recommend checking chip thickness, or rather, using the recommended h_m range to choose (calculate) the proper feed rate.
It is, of course, also necessary to take into account the geometry of the indexable insert itself. To calculate f_z, you can use the formulae provided above or use the following formula. The values of coefficient *c* can be derived from the following chart:

$$f_z = \frac{h_m}{sin \times \kappa_r} \times c$$



Each tool type listed in this catalogue has its own optimum range of average chip thickness. Using values lower than listed in this range may prevent the tool from cutting or, rather, may subject the insert to excessive wear and, in extreme cases, may even destroy it in the process. Similarly, exceeding the recommended values may destroy the insert by overloading the tool. The ranges of recommended average chip thickness are listed directly by each tool family.
The full range of chip thickness can only be used for groups P and K. The lower limit of chip thickness must be adjusted (taken as higher than listed) for groups M and S and for tougher materials from group N. The upper limit must be lowered for groups H, S and slightly also for tougher materials from group M. On the contrary, it is possible to increase the upper limit of recommended average chip thickness by approx. 10 – 15 % when machining soft materials from group N.

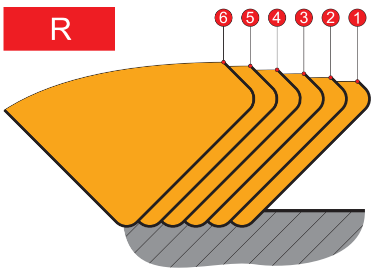


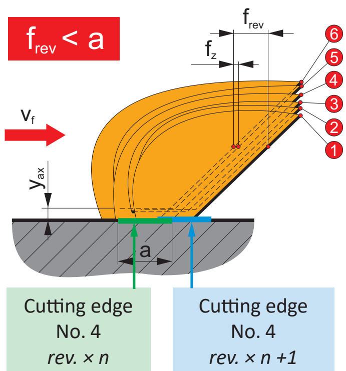
Optimum range
of average chip thickness
(mm)

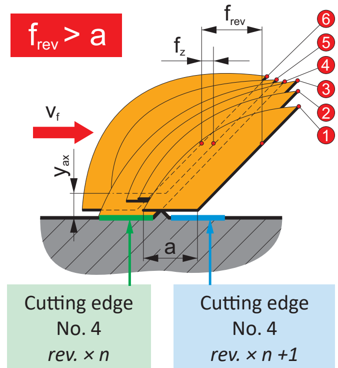
One of the key criteria in finishing operations is the resulting roughness of the machined surface. The following article will therefore provide several tips on how to approach this issue.

Face Milling

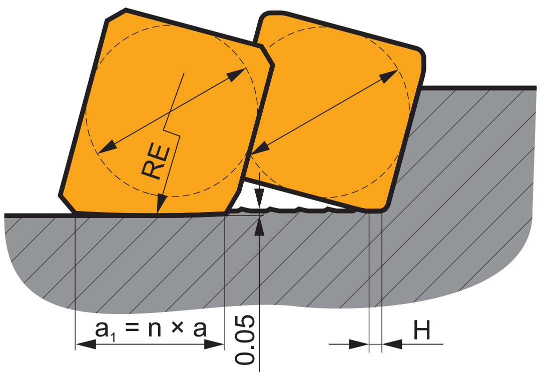
When performing any milling operation, the machined surface is shaped by multiple edges. The microgeometry of the surface is thus dependent on the axial runout of the individual edges of the milling cutter. The most axially protruding edges are the ones that shape the machined surface. The resulting roughness of the milled surface is, to a large extent, influenced by the design of the tip of the indexable insert. If the tip of the indexable insert has a radius, it creates imperfections on the surface. The size of these imperfections is dependent on the corner radius and feed speed. For inserts with smoothing segments, the rule of thumb is that the feed per revolution must be less than 80 % of the size of the smoothing segment. In larger (multi-tooth) cutters, fulfilling this condition can sometimes be problematic, since the maximum feed value $f_z = 0.8 \times a / z$ may approach the lower limit recommended for certain types of insert geometry (the feed speed is lower than the width of the facet in the feed direction). Using lower feed speeds usually results in an increase in cutting resistance, leading to reduced tool life.



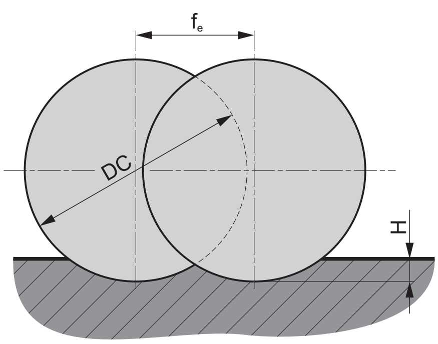




In that case, the best solution is to use a milling cutter with fewer teeth or to reduce the number of teeth on the milling cutter (only fitting an insert onto every other tooth of milling cutters with an even number of teeth). There is, however, a risk of reduced productivity. Another alternative is the use of so-called wiper inserts (if such inserts are available for the given type of tool). Even this solution has its drawbacks, however. For milling cutters with a small diameter (approx. 63 mm and less) the speed gradient is too high and there is a risk of tearing or smearing of the surface (edge build-up) towards the centre of the milling cutter when machining tough materials.
Information about the size of smoothing segments can be found at the beginning of technical information in the catalogue section.



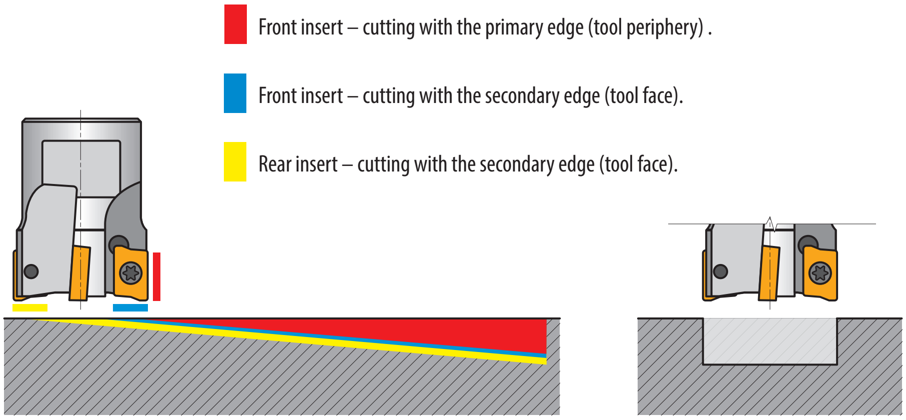
As regards the majority of other types of milling operations, the approximate maximum surface roughness can again be calculated. To do so, we can use the following formula, here accompanied by a graphical explanation.



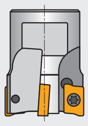
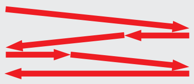
$$H = \frac{f_e^2}{4 \times DC} \rightarrow f_e = \sqrt{4 \times DC \times H}$$

Ramping

Ramping is a technology that simultaneously applies three different cutting methods:



An important parameter here is the ramp angle, i.e. the descent in the Z axis across the given stretch. Some tools (HFC) allow descending at a lower angle but with a higher feed, or allow a higher ramp angle with lower feed to be used. These angles or descents across the given section are listed in technical recommendations.

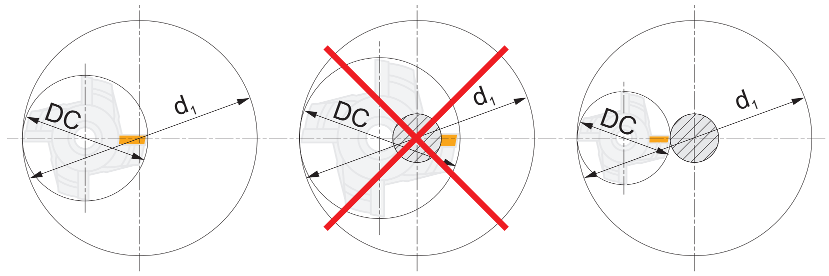
	
	Down at max. angle and horizontally back and down again at max. angle and horizontally back...
	There and back at a smaller (half) angle and last exit horizontally.
	Down at max. angle, back horizontally by length *DC* and then down at max. angle, repeat straight...
	Down at max. angle, then up by length *X* and down again at max. angle. $X = tg\,\alpha\,(DC - W1)$

When choosing the feed speed, we advise following the recommendation given for slot milling. If the slot is deeper (i.e. first pass at an angle, second to level off), you must select one of four basic programme variants for the consecutive steps.

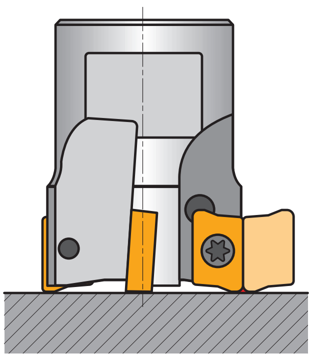
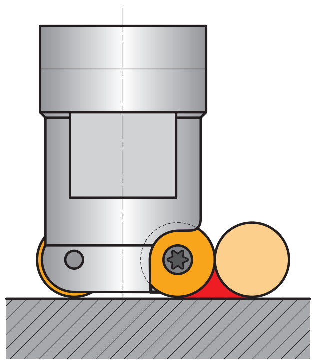
Where:
- *X* Offset (mm)
- α Ramp angle (°)
- *DC* Diameter of the milling cutter (mm)
- *W1* Insert width (mm)

Milling using Circular or Helical Interpolation

This method is analogous to ramping, except it is performed along a circular path. In this case, one of the most important factors is the milling cutter diameter or minimum and maximum diameter of the hole we are able to machine with the given milling cutter type (this information is vital only when using milling cutters without central cutting edges). If the milling cutter diameter is too large, the path of the insert will not pass through the axis of the hole, resulting in a protrusion which will collide with the face of the tool and may potentially destroy the tool completely. On the other hand, if the diameter of the milling cutter is too small, the core will remain inside the hole axis and must then be milled off separately.

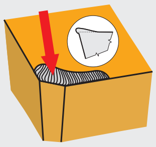


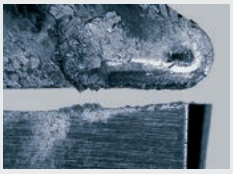
DC = cca 0.5 × d_1 DC > 0.5 × d_1 DC < 0.5 × d_1

	D_{max} – Hole diameter DC – Milling cutter diameter $INSD$ – Insert diameter RE – Insert corner radius BS – WIPER edge length b – Max. a_e for grooving	
Maximum hole diameter For blind holes, you can achieve a flat bottom by having the tool pass over the bottom's centre.		
For through hole: $D_{max} = 2 \times DC$		For through hole: $D_{max} = 2 \times DC$
Minimum hole diameter		
For through hole: $D_{min} = (DC - b) \times 2$		For through hole: $D_{min} = (DC - 0.8\ INSD) \times 2$
For flat bottom: $D_{min} = (DC - (RE + BS)) \times 2$		For flat bottom: $D_{min} = (DC - 0.5\ INSD) \times 2$

Recommendations include tables listing the minimum hole diameter, maximum hole diameter and in-axis descent angle values for these diameters (in some cases there will be two tables: one for standard insert geometry and another for HFC).

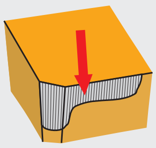
BUILT–UP EDGE

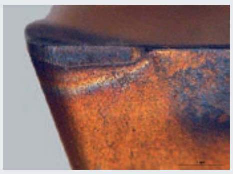




		It has no influence.
(MT)CVD PVD	++	Any coating (decisive factor is anti-adhesion effect).
f	↑	The higher the feed rate the less probability of built-up edge creation.
V	↓↑	Change (generally increase) the cutting speed.
a_p		It has no influence.
	↓↑	Use more positive geometry (built up edge is not created when the rake angle is more than 40°).
	-	Use a coolant with more effective anti-sticking properties (we do not recommend to use coolant for milling).

FLANK WEAR



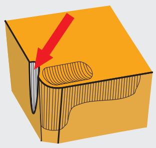


	↑	Use a more wear resistant substrate (H).
(MT)CVD PVD	++	Any coating (decisive factor is hardness – TiC, TiCN).
f	↑	Increase feed (especially if it is under 0.1 mm).
V	↓	Decrease cutting speed.
a_p		It has no influence.
	↑	Increase the clearance angle.
	+	It can help, but only with ideal working conditions.

CRATERING

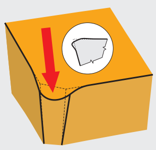
	↑	Use a more wear resistant substrate (S).
(MT)CVD PVD	++	CVD coating (decisive factor is oxidation resistance – α Al_2O_3).
f	↑	Feed has infuence on shape and position of crater.
V	↓	Decrease cutting speed.
a_p	↓	Minimal effect.
	↑	Use more positive cutting geometry.
	++	It can help, but only with ideal working conditions.

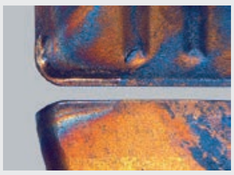
OXIDATION GROOVE ON THE MINOR EDGE



	↑	Use a more wear resistant substrate (S).
(MT)CVD PVD	++	CVD coating (decisive factor is oxidation resistance – α Al_2O_3).
f	↓	Feed has infuence on shape and position of groove.
v	↓	Decrease cutting speed.
a_p	↓	Minimal effect.
	↑	Use another (more positive) cutting geometry.
	++	It can help, but only with ideal working conditions.

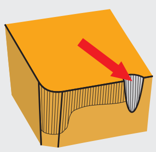
PLASTIC DEFORMATION





	↑	Using a more wear resistant substrate (decisive factor is content of Co).
(MT)CVD PVD	+	Any coating (decisive factor is friction).
f	↓	Decrease feed rate.
v	↓	Decrease cutting speed.
a_p	↓	Minimal effect.
	↑	Use another (more positive) cutting geometry.
	++	It can help, but only with ideal working conditions.

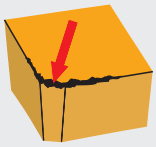
NOTCH WEAR



	↑↓	It depends on the character of the damage (abrasive – use more wear resistant substrate; breaking – use tougher substrate).
(MT)CVD PVD	++	CVD coating (decisive factor is oxidation resistance – α Al_2O_3).
f	↓	Feed has infuence on intensity, but less than the cutting speed.
v	↓	Decrease cutting speed.
a_p	↑↓	Use unequal depth of cut.
	↓	Use less positive cutting geometry.
	+	It can help, but only with ideal working conditions.
		Use tool with smaller setting angle.

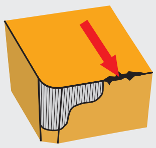
TYPES OF WEAR ON MILLING INSERTS

BRITTLE CRACKS AT THE CUTTING EDGE



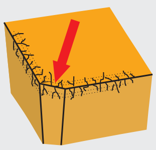
	↓	(H) grain has a great influence.
(MT)CVD PVD	+	PVD coating recommended.
f	↓	Feed has infuence on intensity, but less than the cutting speed.
v	↑↓	It is about vibrations.
a_p		It has no influence.
	↑	Increase the rake angle to reduce cutting forces.
	-	No coolant (it is possible to use air to remove chips from cutting area).
		Use better working condition (a_e / DC).

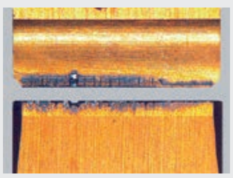
FAILURE OF CUTTING EDGE



	↓	(H) grain has a great influence.
(MT)CVD PVD	+	PVD coating recommended.
f	↑↓	Good swarf control is very important.
v	↑↓	It is about swarf control and vibration.
a_p	↑↓	Reduces the force load (important for machining with long overhangs).
	↓	Use less positive cutting geometry.
		It has no influence.
		Use better working conditions, reduce feed rate until insert is in cut.

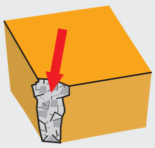
CREATION OF RACK CRACKS

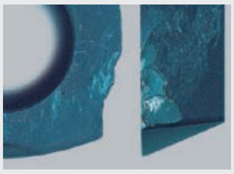




	↓	(H) grain has a great influence.
(MT)CVD PVD	+ +	PVD coating recommended.
f	↓	Feed has infuence on intensity, but less than the cutting speed.
v	↓	Lower speed means lower temperature.
a_p		It has no influence.
	↑	Use another (more positive) cutting geometry.
	- - -	No coolant (it is possible to use air to remove chips from cutting area).
		Use better working condition (a_e / DC).

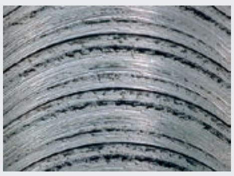
INSERT FRACTURE





	↓	(H) grain has a great influence.
(MT)CVD PVD	+	PVD coating recommended.
f	↓	Very important to reduce cutting force.
v	↑↓	It is about swarf control and vibration.
a_p	↓	Reduces the force load.
	↓	Use less positive cutting geometry.
		It has no influence.
		Use better working conditions (a_e / DC).

POOR SURFACE QUALITY



Description and cause:

Numerous causes depending on the workpiece material, cutting conditons (feed rate and cutting speed), the condition of the cutting edge, the extent and type of wear, and the condition and rigidity of the machine – tool – workpiece assembly.

- Incorrect tool chosen
- Incorrect chip thickness
- Incorrect cutting speed
- Coolant is needed
- High feed rate

Corrective measures:

- Use a finishing insert, or an insert with finishing segment
- Use an insert with suitable cutting geometry
- Reduce the feed rate
- Adjust (usually increase) the cutting speed
- Use coolant or lubrication (MQL)
- Eliminate vibrations
- Use a tool with which the position of the individual inserts can be adjusted more accurately
- Change the chip thickness (modify the machining conditions)

VIBRATIONS

Description and cause:

This is a very common problem, which is mainly caused by an unbalanced workpiece or tool, unstable fixing of the machined part and high cutting forces.

- Low rigidity of machine-tool-workpiece assembly
- Excessive chip depth (both axial and radial)
- Run-out – poor workpiece or tool balance
- Large tool overhang

Corrective measures:

- Check the stability of the workpiece fixing
- Check the stability of the tool fixing
- Reduce the cutting depth
- Use a tool with smaller overhang
- Modify the cutting speed
- Reduce the chip thickness (change the cutting or machining conditions)
- Choose a suitable cutting geometry and tool material to minimize the cutting process force balance (as sharp and as positive as possible), i.e. use a tool with a lower cutting resistance
- When milling, use a tool with a smaller setting angle

BURRS

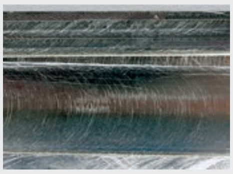
Description and cause:

This usually occurs on soft steels and plastic materials.

Corrective measures:

- Use a cutting insert with a sharp cutting edge
- Use a cutting insert with positive geometry
- Use a tool with a smaller setting angle

ERRORS IN DIMENSIONS AND SHAPE OF WORKPIECE



Description and cause:

Depends on a number of factors.

Corrective measures:

- Use a wear-resistant cutting insert
- Improve the stability of the cutter and workpiece
- Minimize tool overhang
- Use a workpiece with a suitable machining allowance

INADEQUATE CHIP FORMATION

Description and cause:

Producing a chip with a suitable shape is very important to insert durability and service life of the tool. The workpiece material, the feed rate, the depth of cut and the cutting geometry all have an effect on chip forming. A chip that is too long is unacceptable for various reasons, while a chip that is too short is undesirable as it overloads the cutting edge and causes vibrations.

Corrective measures:

- Change the feed rate and depth of cut
- Use a more suitable cutting geometry
- Change the cutting conditions

CHECK THE SEAT CONDITION OF THE CUTTING INSERT

Before clamping a new cutting insert or changing the edge, it is necessary to clean the seat and check its condition or the condition of the anvil and wedge (especially the damage under the corner of the cutting insert).

CHECK AND SERVICE THE CLAMPING PARTS

It is also important to check the clamping parts, including clamping levers, screws, wedges and clamps. Only use original, undamaged parts (found in the catalogue). Regularly lubricate the threads and the binding surface of screws using, for example, heat-resistant lubricant (MOLYKOTE). For assembly and disassembly, only use screwdrivers and wrenches specified in our catalogue or recommended by the tool manufacturer. Be careful not to over-tighten. To avoid this, we advise using a pre-set torque wrench.

CHECK THE TIGHTENING

Before tightening, check the fit of the cutting insert on the whole of the binding surface and in the radial and axial directions. Cutting inserts and tools must always be clean and undamaged.

Value	Unit	Formula
Number of revolutions	(rev/min)	$n = \dfrac{v_c \times 1000}{DC \times \pi}$
Cutting speed	(m/min)	$v_c = \dfrac{\pi \times DC \times n}{1000}$
Feed per revolution	(mm/rev)	$f_{rev} = \dfrac{f_{min}}{n} = f_z \times z$
Feed per minute (speed of feed)	(mm/min)	$f_{min} = v_f = f_{rev} \times n = f_z \times z \times n$
Feed per tooth	(mm/tooth)	$f_z = \dfrac{f_{rev}}{z} = \dfrac{f_{min}}{n \times z}$
Chip cross section	(mm^2)	$A = f_z \times a_p$
Chip thickness (for inserts with a straight edge)	(mm)	$h = f_z \times \sin KAPR$
Chip thickness (for round cutting inserts)	(mm)	$h = f_z \times \sqrt{\dfrac{a_p}{INSD}}$
Metal removal rate	(cm^3/min)	$Q = \dfrac{a_p \times a_e \times f_{min}}{1000}$
Power demand	(kW)	$P_c = \dfrac{a_p \times a_e \times f_{min}}{60 \times 10^6 \times \eta} \times k_c \times k_\gamma$
Approximate power demand	(kW)	$P_c = \dfrac{a_p \times a_e \times f_{min}}{x}$

Note:

	Quantity	Unit
n	Number of revolutions	(rev/min)
DC	Diameter (of tool or work piece)	(mm)
v_c	Cutting speed	(m/min)
f_{rev}	Feed per revolution	(mm/rev)
A	Chip cross section	(mm^2)
a_p	Axial depth of cut (depth of cut)	(mm)
a_e	Radial depth of cut (width of cut)	(mm)
$KAPR$	Setting angle	(°)
f_{min}	Feed per minute (sometimes called speed of feed)	(mm/min)
f_z	Feed per tooth	(mm/tooth)
z	Number of teeth	(-)
$INSD$	Diameter of insert	(mm)

	Quantity	Unit
h	Chip thickness	(mm)
Q	Material removal rate per minute	(cm^3/min)
P_c	Power demand	(kW)
k_c	Cutting force per mm^2	(MPa)
k_γ	Coefficient of influence of angle γ_0	(°)
η	Machine efficiency usually $\eta = 0.75$	(-)
x	Coefficient of influence of work piece material	(-)

Material	Steel	Cast iron	Al
Coefficient x	24 000	30 000	120 000

INDEXABLE TURNING – TECHNICAL INFO

TECHNICAL INFORMATION – TURNING GRADES – NAVIGATOR

Group	Cemented carbide with MTCVD	Cemented carbide with PVD	Cemented carbide	CERMET
P01	T9310	T6310		TT010
P05	T9310, T9315	T6310		TT010
P10	T9310, T9315	T6310		TT010, TT310
P15	T9310, T9315, T9325	T6310		TT310
P20	T9315, T9325, T9335	T8430		TT310
P25	T9315, T9325, T9335	T8430		TT310
P30	T9325, T9335	T8430		
P35	T9325, T9335	T8430		
P40	T9335	T8430		
P45	T9335			
P50				

Group	Cemented carbide with MTCVD	Cemented carbide with PVD	Cemented carbide	CERMET
M01		T6310		
M05		T6310		
M10	T7325, T9325	T6310		
M15	T7325, T9325	T6310		
M20	T7325, T7335, T9325	T8430		
M25	T7325, T7335, T9325	T8430		
M30	T7335, T9325	T8430		
M35	T7335	T8430		
M40	T7335			

Group	Cemented carbide with MTCVD	Cemented carbide with PVD	Cemented carbide	CERMET
K01	T5305			
K05	T5305			
K10	T5305, T5315			
K15	T5305, T5315			
K20	T5315			
K25	T5315	T8430		
K30		T8430		
K35		T8430		
K40		T8430		

Group	Cemented carbide with MTCVD	Cemented carbide with PVD	Cemented carbide	CERMET
N01				
N05				
N10				
N15				
N20				
N25				
N30				

Group	Cemented carbide with MTCVD	Cemented carbide with PVD	Cemented carbide	CERMET
S01		T6310		
S05		T6310		
S10	T7325	T6310		
S15	T7325, T7335	T6310		
S20	T7325, T7335			
S25	T7325, T7335			
S30				

Group	Cemented carbide with MTCVD	Cemented carbide with PVD	Cemented carbide	CERMET
H01		T6310		
H05	T5305	T6310		
H10	T5305	T6310		
H15	T5305, T9315	T6310		
H20	T9315			
H25				
H30				

Grade Identification	Area of Application	Application	Feed	Cutting speed	Resistance to adverse Working Conditions	Coating	Colour	Substrate	Coolant benefit	Grade description
T9226	P15 - P35	■				MT-CVD		FGM	+ + +	Grade designed for heave roughing applications. A versatile grade with high resistance to mechanical damage and retains very good wear resistance. Usable at lower cutting speeds.
	M10 - M30	■								
	K15 - K25	◩								
	S15 - S25	◩								
T9310	P01 - P15	■				MT-CVD		FGM	+ +	Grade with high abrasion resistance which can be used for slightly interrupted cutting. It will be used for finishing or semi-roughing operations. This material can also be used for roughing operations provided the machine-tool-workpiece configuration is sufficiently rigid.
	K05 - K20	◩								
	H10 - H20	◩								
T9315	P05 - P25	■				MT-CVD		FGM	+ +	A versatile grade with excellent wear resistance properties even under intense cutting conditions. It can also be used for operations with interrupted cuts. With it's well balanced properties this grade can be first choice for a wide range of turning operations. Not suited to low cutting speeds.
	K05 - K25	◩								
	H10 - H20	◩								
T9316	P10 - P20	■				MT-CVD		FGM	+ + +	Grade designed for railway applications. A versatile grade with excellent wear resistance properties. Usable at lower and high cutting speeds.
	M05 - M15	◩								
	K10 - K30	■								
	H15 - H25	◩								
T9325	P15 - P35	■				MT-CVD		FGM	+ +	From a technological perspective this is an extremely versatile grade with high resistance to mechanical damage in adverse cutting conditions and retains excellent wear resistance. The correct application of this material requires high cutting speeds.
	M10 - M30	■								
	K15 - K35	◩								
	S10 - S20	◩								
T9415 NEW	P05 - P30	■				MT-CVD		M	+ +	Highly wear-resistant material designed primarily for finish turning of common carbon and alloy steels. Despite its high abrasion resistance, it is also suitable for interrupted cutting operations. We recommend this material as the first choice for most turning operations, especially in high production applications.
	K05 - K25	◩								
	H10 - H20	◩								
T9335	P20 - P45	■				MT-CVD		FGM	+ + +	One of the toughest grades which is especially suitable for adverse cutting conditions at medium to high feed rates and medium cutting speeds. Compared to its predecessors, M15 – M40 it is not only tougher, but also more abrasion resistant which will be useful when using intensive cutting conditions.
	M15 - M40	■								
	S15 - S25	◩								
T7325	P15 - P35	◩				MT-CVD		FGM	+ + +	One of the most universal turning grades. Especially designed for stainless steel machining. Optimal balance between wear resistance and performance reliability. Suitable for broad variety of application in turning operations.
	M10 - M25	■								
	S10 - S25	■								
T7335	P20 - P40	◩				MT-CVD		FGM	+ + +	Grade with functionally graded substrate, featuring very high operational reliability and very good wear-resistance. It is best suited to use in the machining of very tough M20 – M40 materials.
	M20 - M40	■								
	S15 - S25	■								
T5305	P05 - P15	◩				MT-CVD		H	+	Grade with very high resistance to chemical wear; suitable for finishing operations using high cutting speeds. With its high abrasion resistance, it is also suitable for productive K01 – K15, machining of hardened and treated materials.
	K01 - K15	■								
	H05 - H15	◩								
T5315	P10 - P25	◩				MT-CVD		H	+	Grade intended primarily for productive machining which has high abrasion resistance and good operational reliability. Due to its properties, this material is particularly suitable for roughing and finishing operations for good or slightly adverse cutting conditions.
	K10 - K25	■								
	H15 - H25	◩								

TECHNICAL INFORMATION – TURNING GRADES

Grade Identification	Area of Application	Application	Feed	Cutting speed	Resistance to adverse Working Conditions	Coating	Colour	Substrate	Coolant benefit	Grade description
6630	P15 - P35	■				MT-CVD		FMG	+ + +	A versatile grade with high resistance to mechanical damage and retains very good wear resistance. Usable at lower cutting speeds for heavy roughing applications.
	M10 - M30	■								
	K20 - K30	◪								
	S15 - S25	□								
6640	P20 - P40	■				MT-CVD		H	+ + +	One of the toughest turning materials which can be used especially in roughing operations, or where operational reliability under adverse cutting conditions is a priority. Another ideal choice for machines working with low to medium cutting speeds and medium to high feed rates.
	M20 - M35	■								
	K25 - K40	■								
T6310	P01 - P15	■				PVD		ultra submicron H	+ + +	High wear resistant turning grade with top PVD coating. Suitable for finishing operation and applications, where sharp cutting edge together with high flank wear resistance is of high importance
	M01 - M15	■								
	K05 - K20	■								
	N05 - N20	■								
	S01 - S15	■								
	H01 - H15	■								
T8330	P25 - P40	■				PVD		submicron H	+ + +	Undoubtedly the most versatile grade it is suitable for machining all types of materials and can be applied in almost all turning operations. It's main benefits are high operational reliability and excellent frictional properties; it is therefore suited to applications at medium to low cutting speeds.
	M20 - M35	■								
	K20 - K40	■								
	N15 - N30	◪								
	S15 - S25	◪								
	H15 - H25	◪								
T8345	P30 - P50	■				PVD		submicron H	+ + +	This is the toughest turning grade, which is intended mainly for machining under the worst cutting conditions and in applications with the highest requirements for operating reliability. Because of these properties, this material is recommended for lower cutting speeds.
	M20 - M40	◪								
	K30 - K40	◪								
	S20 - S30	◪								
T8430 NEW	P20 - P40	■				PVD		submicron H	+ + +	Undoubtedly the most versatile cutting material, this is useful for machining of all types of machined materials and is practically applicable in almost all types of turning operations. Its main benefits are its high operational reliability and very good frictional properties; it is therefore suitable for applications at medium and lower cutting speeds.
	M20 - M35	■								
	K25 - K40	◪								
	N15 - N30	◪								
	S15 - S25	◪								
	H15 - H25	◪								

Substrat	
H	WC-Co based substrate
submicron H	WC-Co based substrate fine grained (< 1 µm)
ultra submicron H	WC-Co based substrate very fine grained (< 0,5 µm)
FGM	Functionally graded substrate
Cermet	Cemented carbide without WC
ceramics	Cutting ceramics
PCD	Polycrystalline Diamond
CBN	Cubic Boron Nitride
HSS	High speed steel

Coating	
MT-CVD	Medium-temperature chemical method of coating
PVD	Low-temperature physical method of coating
×	Uncoated grade

Benefits of cutting fluid	
+ + +	Use of coolant is essential
+ +	Highly recommended
+	Recommended
+ / –	Optional
– -	Do not use coolant
–	Coolant not recommended

Correction factors for specific type of operation C_{VcO}

	FF			F			M			R			HR		
a_p	0.5			1.5			2.5			5.0			12.0		
	I	II	III	I	II	III	I	II	III	I	II	III	I	II	III
f	0.05	0.08	0.10	0.10	0.15	0.20	0.20	0.30	0.40	0.40	0.60	0.80	0.80	1.00	1.30
Chip-breakers for fine finishing (FF, FF2…)	1.15	1.00	0.95	0.85	0.80	–	–	–	–	–	–	–	–	–	–
Chip-breakers for finishing (NF, SF..)	–	–	1.20	1.05	1.00	1.05	1.00	0.90	–	–	–	–	–	–	–
Chip-breakers for medium machining (FM, M, NM, NMR, SM…)	–	–	–	–	–	1.15	1.10	1.00	0.95	0.85	–	–	–	–	–
Chip-breakers for roughing (RM, NRM, NR, R…)	–	–	–	–	–	–	–	–	1.25	1.10	1.00	0.95	0.65	–	–
Chip-breakers for heavy roughing (HR, HR2, NR2, OR…) for 45 min durability	–	–	–	–	–	–	–	–	–	1.25	1.20	1.15	1.05	1.00	0.95

Correction factors for required durability C_{VcT}

minutes	10	15	20	30	45	60
General machining operations (fine finishing up to roughing)	1.13	1.00	0.93	0.84	0.76	0.71
Heavy machining operations (heavy roughing)	–	–	–	1.10	1.00	0.93

Additional correction factors C_{VcA}

Machining environment	C_{VcA}
Condition of the work-material (hard skin due to forging or casting)	0.70
Internal turning	0.75
Parting and grooving (radial)	0.88
Face grooving	0.80
Interrupted cut	0.80
Unstable machining conditions	0.85
Common machining conditions	1.00
Stable machining conditions	1.20

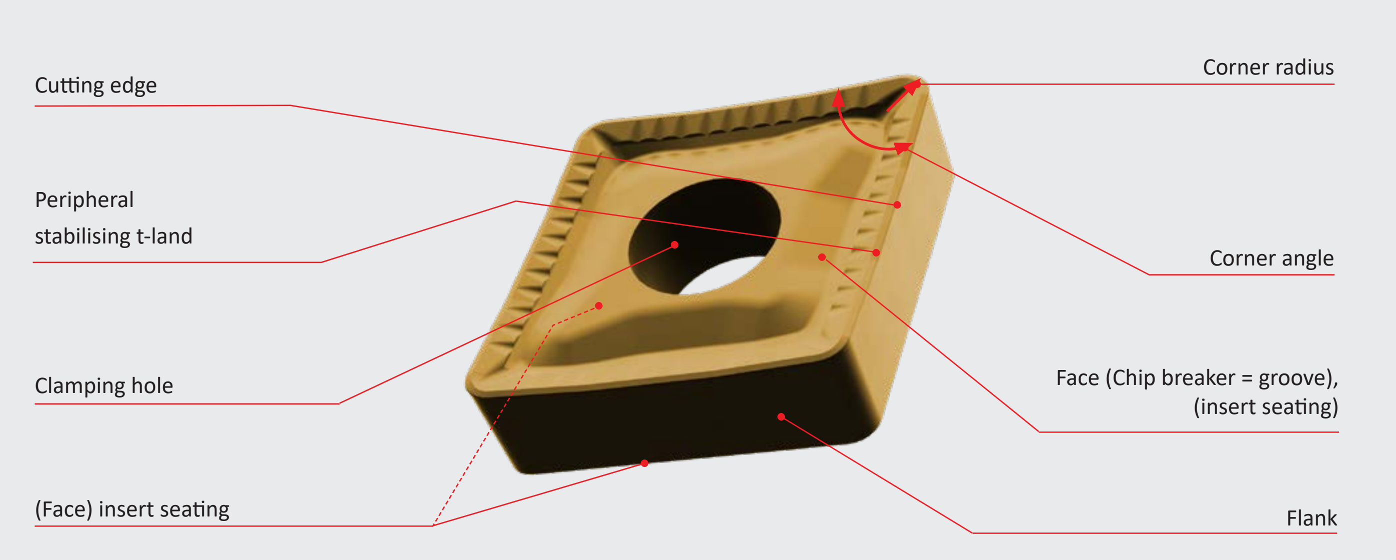
Resulting corrected cutting speed v_{cC}

$$v_{cC} = v_c \cdot k_{vG} \cdot C_{VcO} \cdot C_{VcT} \cdot C_{VcA}$$

k_{vG} – coefficient of used material
v_c – starting speed from catalogue page

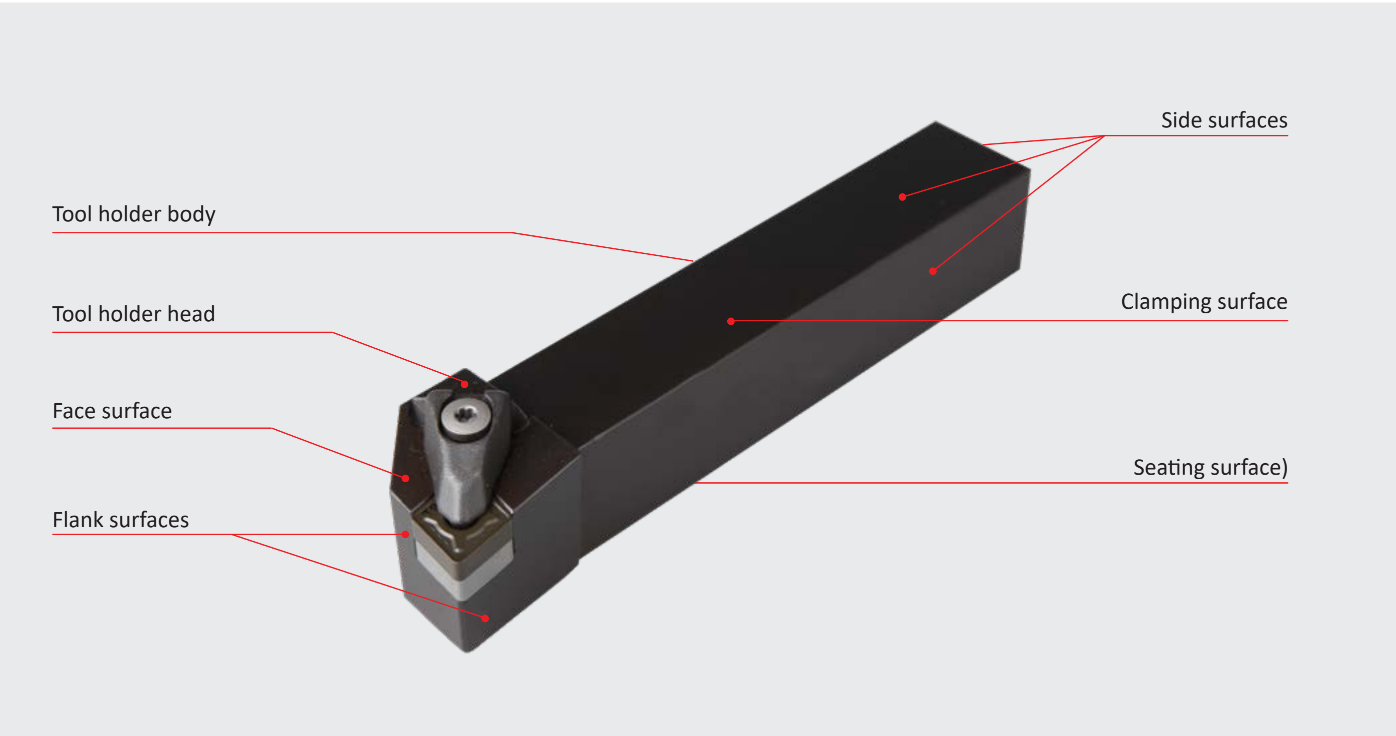
TECHNICAL INFORMATION – INDEXABLE TURNING– DEFINITION OF BASIC TERMS

Insert parts



Corner radius – determines in most cases the recommended minimum depth of cut and, together with the feed, also determines the achieved roughness.
The **Cutting edge** is the intersection of face and flank surfaces. Its longitudinal roughness is one of the first evaluation criteria when assessing an insert.
The **insert corner angle** is very important with regard to the usable cutting edge length, resistance in the interrupted cut, heat dissipation from the cutting point, etc.
The **peripheral stabilising t-land** is an area located after the cutting edge. Its width is very often variable and its angle also changes regularly. In most cases, the width of the t-land, together with the adjustment angle at which the insert works, is a limiting factor for specifying the minimum feed.
The **chip breaker** – together with the t-land determines the application area (narrow grooves – finishing and materials with short chip, wide grooves – mostly roughing operations and tough materials).
The **clamping hole** – if there is no hole, the insert will definitely be designed for the ISO C clamping system. If the hole is cylindrical, the insert is designed for the ISO P, M, D + clamping systems (in nearly all cases the flank angle is 0°. If the hole is trumpet-shaped and the flank angle is positive, the insert is single-sided and is designed for the ISO S clamping system. If the hole is conical and is the same on both sides of the insert, then it is most likely a tangential insert (double-sided).
The **insert seating** – if it is formed by the same relief as the face surface, the insert is double-sided, if it is different, the insert is single-sided. It must be assessed with regard to the planned load or the type of cut. (the size and distance of the radius and the cutting edges).

Tool holder parts



TECHNICAL INFORMATION – INDEXABLE TURNING – DEFINITION OF BASIC TERMS

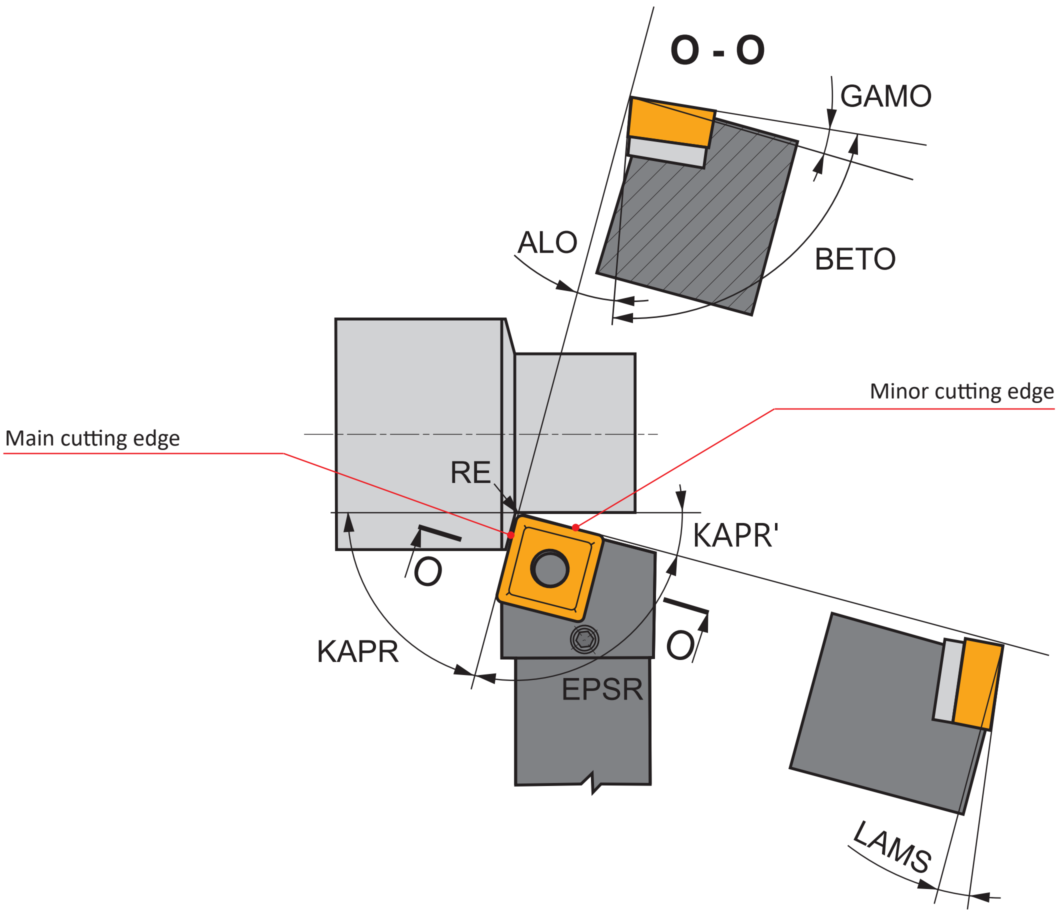
The turning tool consists of two basic parts:
1) the body consisting of:
seating surface
clamping surface
side surfaces (which can further be provided with adjusting screws)
Note: For external turning, the tool holder body is usually a square cross section (square or rectangle). For internal turning, the cross section of the holder body is circular and, for larger cross sections, it is provided with adjusting surfaces. But the holder body can also be formed by a special type of shank, such as CAPTO (PSC) or HSK

2) the head with:
face
flank surfaces
The tool holder head also includes a clamping system into which the inserts are inserted
From the face side (for radial inserts) or from the flank surface side (for tangential inserts)
Note: the types of heads are:
straight – allows turning in both directions
side – distinguish between right and left design
bent – distinguish between right and left design (allows better access when turning more complex surfaces)

Working and construction angles of turning tools

The position and orientation of the cutting edge in relation to the workpiece and its geometric shape determine the cutting angle characteristics.
The angles on the cutting edge are determined by a two coordinate system:
a) design
b) working

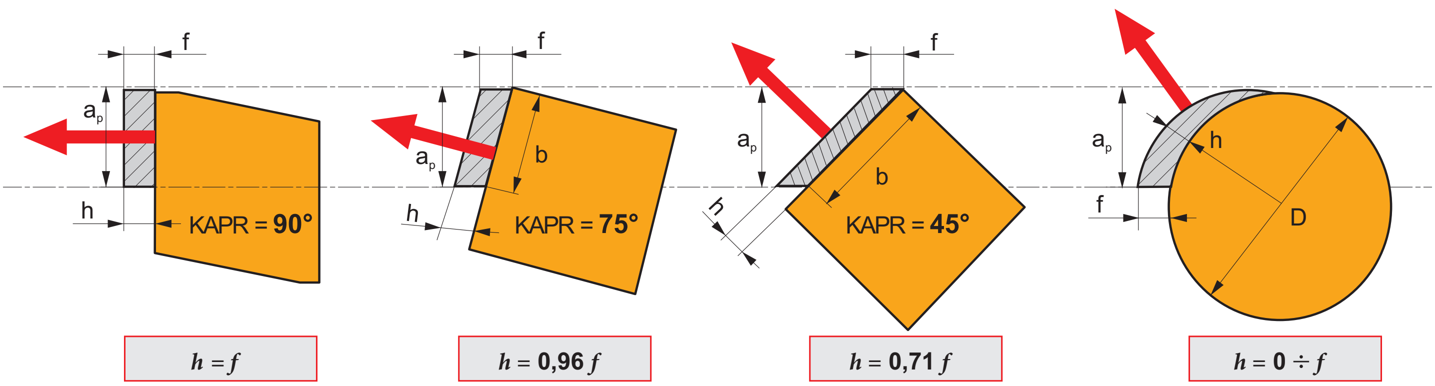
a) tool coordinate system (stationary), which is used to determine the cutting edge geometry during design, production and checking. All angles defined in this system are called tool cutting angles. All angles defined by ISO standards according to the insert shape belong in this group.



b) working coordinate system, used to determine the cutting edge geometry during the machining process. These angles are called working angles and they depend on the position of the insert clamped into the tool holder. For example, the cutting insert SNUN ..... has a tool clearance angle *AN* = 0° and a rake angle *GAMP* = 0°, however the insert is clamped in the tool holder to give a working clearance angle *ALO* = 6° and a working rake angle *GAMO* = -6°. The working angles affect the tool angles with pre-formed chip breakers. However the most important are the working angles for the cutting process.

The basic tool angles are indicated in the picture in the basic tool plane (interlaid by the bearing surface of the tool holder) and in the normal tool plane (interlaid across to cutting edge – cut O-O).
We are concerned with the following angles:
The rake angle *GAMO* – substantially affects the cutting process. Its size determines the progress and the intensity of plastic deformation during chip forming; it also determines the value of the cutting forces and the thermal stress on the cutting edge. The range of rake angles is wide, from *GAMO* = +25° to -15° for cutting tools with indexable cutting inserts for milling and turning. A positive rake angle improves the chip forming conditions, reduces the cutting forces and reduces the cutting temperature level. A negative rake angle improves the strength of the cutting edge, however it increases plastic deformation during chip forming and thereby also the cutting forces and temperatures.

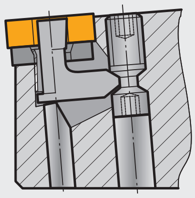
Clearance angle *ALO* affects the value of friction between the flank and the machined surface. Increasing the clearance angle *ALO* reduces this friction and thereby flank wear as well.
Wedge angle *BETO* is the angle of the cutting insert's wedge. Increasing angle *BETO* increases the strength of the cutting edge (resistance against shock), however it also increases the cutting resistance.
Inclination angle of main cutting edge *LAMS* – determines the point of first contact between the cutting edge and the workpiece, which is important for interrupted cut. If *LAMS* is positive, the point of contact is close to the nose of the cutting insert. The negative angle *LAMS* moves the point of first contact far from the nose and thereby affects the resistance of the cutting edge against mechanical stress. Furthermore, *LAMS* affects the direction of chip evacuation. If *LAMS* is negative, the direction of chip evacuation is towards the machined surface. Whereas if *LAMS* is positive, the direction of chip evacuation is away from the machined surface.
Setting angle of main cutting edge *KAPR* has main influence on the values of cutting forces and the cross section shape of the chip. Reducing angle *KAPR* makes the chip thinner at a given feed f and depth of cut a_p. Whereas if *KAPR* = 90° the chip thickness $h = f$ and the chip width $b = a_p$ becomes wider. Regarding the decreasing setting angle the function width of the T-land is increasing and the rake angle of insert is decreasing.
Setting angle of minor cutting edge *KAPR´* together with corner radius *RE* define the final surface quality.



TECHNICAL INFORMATION – INDEXABLE TURNING – CHOICE OF CUTTING TOOL

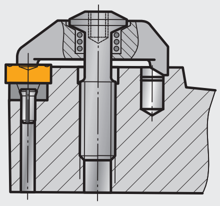
Tool holder choice with regard to the clamping technique

The PRAMET TOOLS offer includes tool holders, adjustable holders, turret heads and adjustable holders for external longitudinal, facing, copy turning, and naturally also for internal turning. Tool holders are classified according to the inserts clamping system into six groups that are schematically illustrated in the following passage.



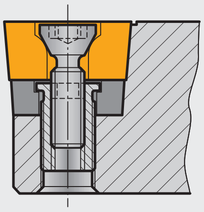
ISO P

This system serves for the clamping of negative inserts with cylindrical hole, both with chip formers and/or without them. The insert clamping is achieved as a result of an angle lever that after tightening the screw presses the insert down to the holder bed. Tool holders with this clamping system of inserts ensure a reliable and exact clamping of an insert. They perform the best and also the most frequent use at external turning operations, namely both finishing and roughing ones. Alternatively this type of clamping can be also used for holders intended for internal turning of holes with larger diameters.



ISO D

This system serves for the clamping of both negative and positive inserts without holes, namely with both chip formers (pre-pressed, ground and side-pressed ones) and without them. The insert is fixed in the bed of a tool holder by a screw-held clamp, under which there is still embedded a side-pressed chip former at some insert types. Holders with this clamping system are used for both the external and internal surface machining. At present the clamping system C loses its importance. Especially at tools for internal turning it is replaced by the system S with benefit.



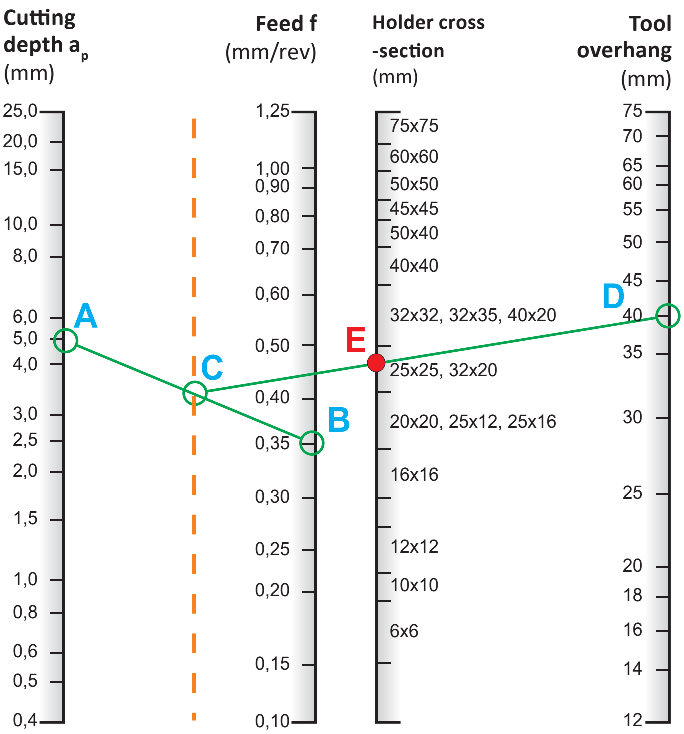
ISO S

This clamping system is mainly used for small cross-section tools, designed for both external and internal turning (drilling). In this case a special screw, going through an insert cone hole, achieves the clamping. By tightening this screw an insert is fixed in the tool bed. This solution is especially convenient because there is no obstacle for chip flow.

TECHNICAL INFORMATION – INDEXABLE TURNING – CHOICE OF CUTTING TOOL

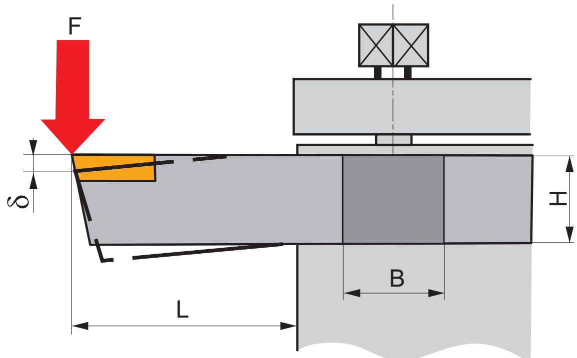
With reference to square cutting tool

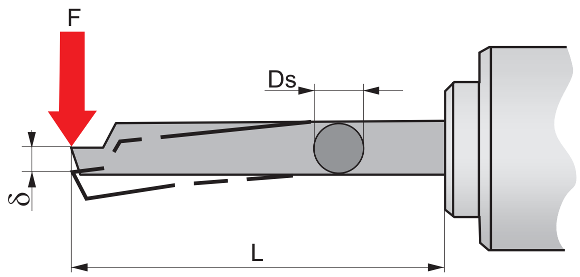
External cutting tools (square cross section)



Unfortunately, we cannot offer you a similar diagram for choosing the diameter of the inner cutting tool as the situation in internal turning is complicated by the chip. Due to the larger overhang, a holder with the largest possible diameter should be chosen, but if the diameter of the holder is close to the diameter of the hole to be machined, problems can occur with the chip evacuation. It usually gets between the hole wall and the holder body damaging the surface being formed. In general, if you use tools with a steel body, the overhang should not exceed 4xD, and if you have tools with a carbide or heavy metal body, the maximum overhang should be 6xD. Remember that for both types of tools, the portion for clamping the tool should be at least 3xD.

Use the **maximum possible cross section** with regard to clamping options and process limitations.





Bending stress

$$\underset{(MPa)}{\sigma} = \frac{6.F.L}{B.H^2}$$

Tool holder deflection

$$\underset{(mm)}{\delta} = \frac{4.F.L^3}{E.B.H^3}$$

Bending stress

$$\underset{(MPa)}{\sigma} = \frac{32.F.L}{\pi.D_s^3}$$

Tool holder deflection

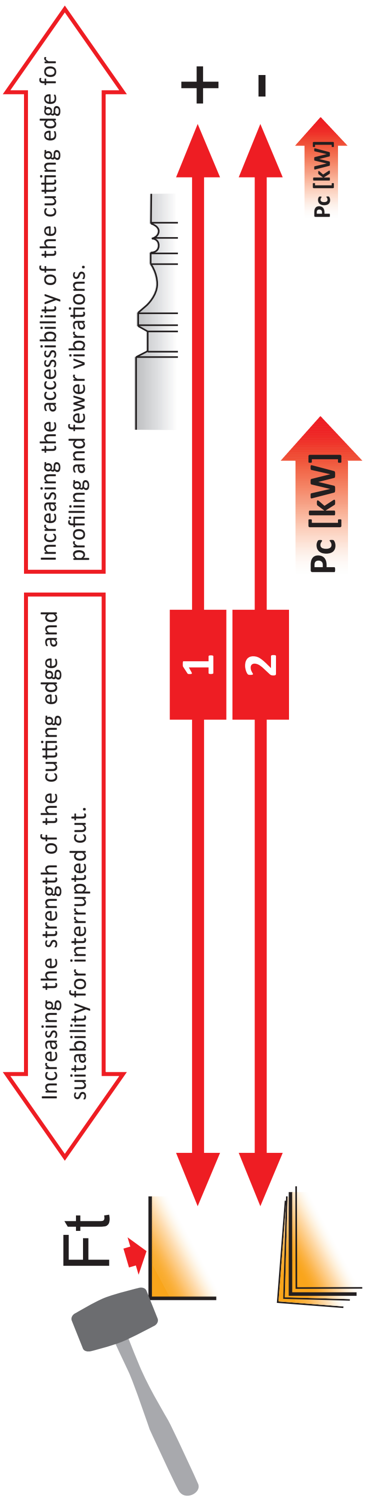
$$\underset{(mm)}{\delta} = \frac{64.F.L^3}{3.\pi.E.D_s^3}$$

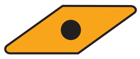
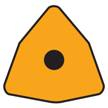
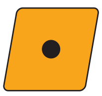
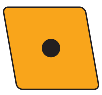
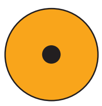
σ	Bending stress in the body (MPa)
F	Cutting force (N)
L	Tool overhang (mm)
B	Body width (mm)
H	Body height (mm)
Ds	Body diameter (mm)
E	Body material elastic modulus (MPa)

Material	MPa (N/mm²)	(kgf/mm²)
Steel	210.000	21.000
Sintered carbide	560.000 – 620.00	56.000 – 62.00

50% reduction in overhang reduces deflection by **88%.**
1/3 increase in cross section reduces bending by **68%.**

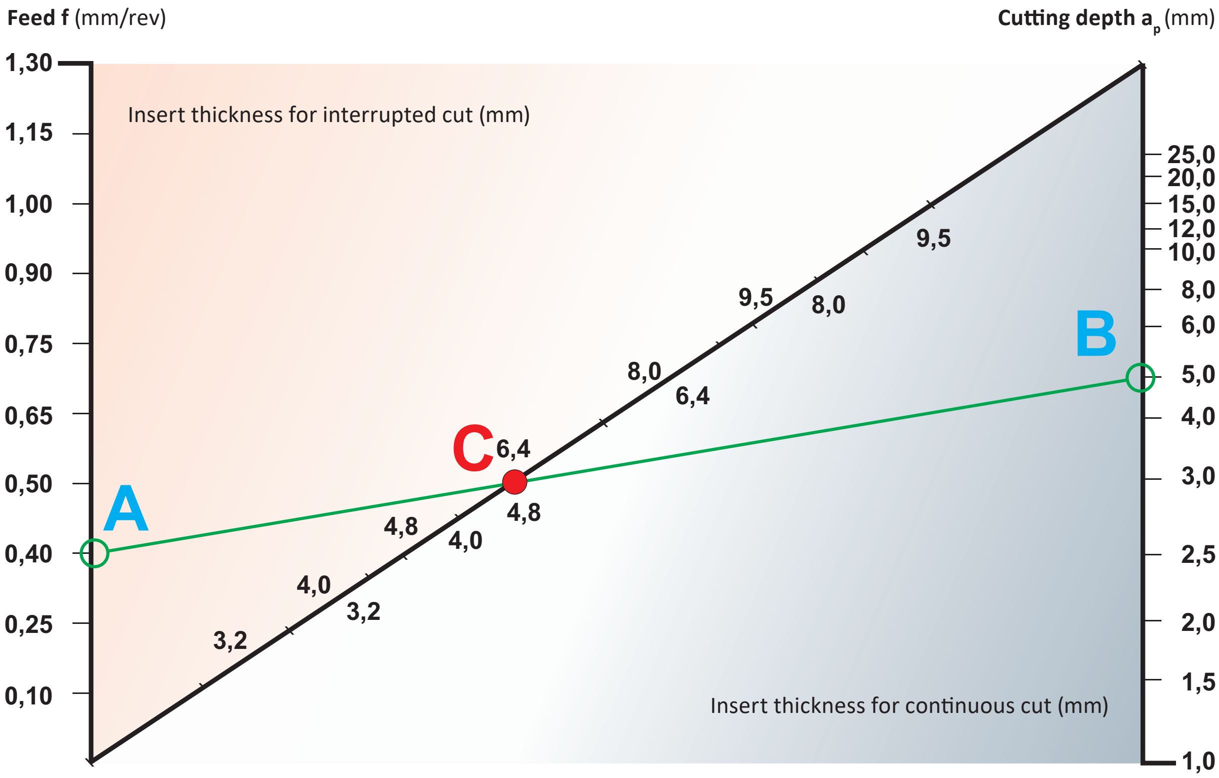
Choosing the shape and size of the insert



Insert shape		Nose angle	Insert size ISO	Insert size ANSI	Maximum length of cutting edge Lmax	(mm)	(")	Roughing	Light roughing	Finishing	Profile turning	Face turning	Versatile applications	Tendency to vibrate	Hard material	Interrupted cut
	V	35°	11	2	0.25L	2.80	.110"			■	■		◩	■		
			13			3.30	.130"									
			16	3		4.20	.165"									
	D	55°	07	2	0.25L	2.00	.078"		■	■	■	◩	■	■		
			11	3		2.90	.114"									
			15	4		3.90	.153"									
	T	60°	11	2	0.33L	3.60	.141"	◩	■	■	◩	◩	◩	■	◩	
			16	3		5.50	.216"									
			22	4		7.30	.287"									
			27	5		9.10	.358"									
	W	80°	06	3	0.50L	3.30	.129"	◩	■	■	◩	■	◩	◩	◩	◩
			08	4		4.40	.173"									
	C	80°	06	2	0.66L	4.20	.165"	■	■	◩	◩	■	■	◩	■	■
			09	3		6.40	.251"									
			12	4		8.50	.334"									
			16	5		10.60	.417"									
			19	6		12.70	.500"									
			25	8		16.50	.649"									
	S	90°	09	3	0.66L	6.30	.248"	■	◩			■			■	■
			12	4		8.40	.330"									
			15	5		10.40	.409"									
			19	6		12.60	.496"									
			25	8		16.80	.661"									
	C	100°	12	4	0.66L	8.50	.334"	■				■			■	■
			19	6		12.70	.500"									
			25	8		16.50	.649"									
	R		06		0.40D	2.40	.094"	■				◩	◩		■	■
			08			3.20	.125"									
			10			4.00	.157"									
			12			4.80	.188"									
			15			6.00	.236"									
			16			6.40	.251"									
			19			7.60	.299"									
			20			8.00	.315"									
			25			10.00	.393"									
			32			12.80	.503"									

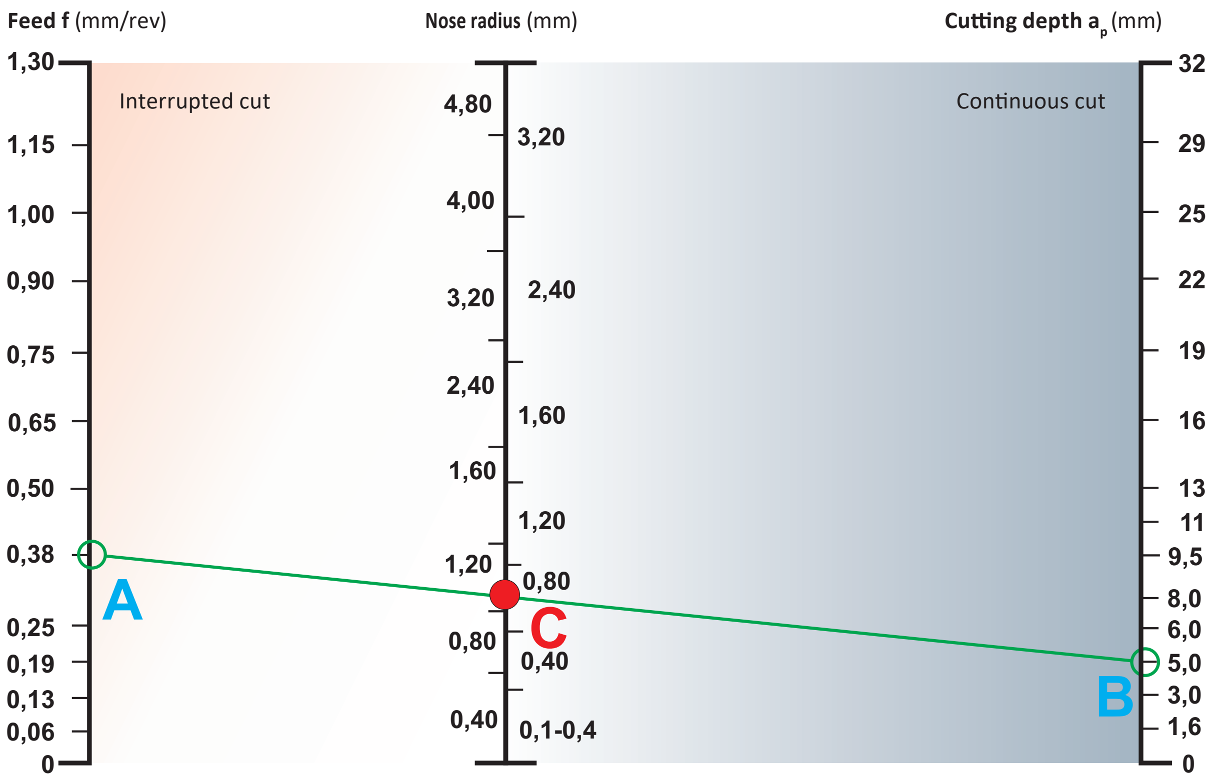
Choosing the optimum thickness of the insert

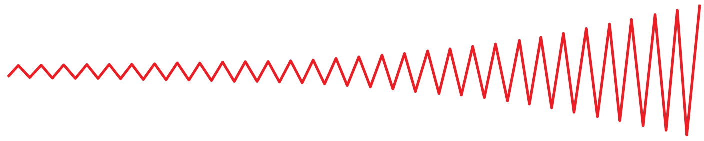
Based on practice, we recommend performing this only for interrupted cut and at a maximum load of inserts.



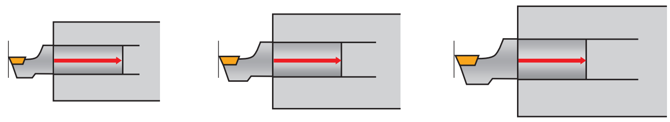
Choosing the nose radius of the insert

Based on practice, we recommend performing this only for interrupted cut and at a maximum load of inserts.

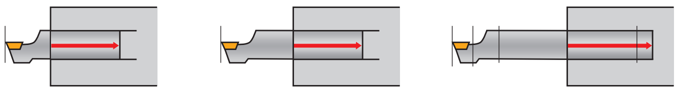




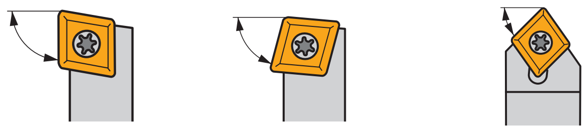
Hardness of chip formation – great attention must be paid to proper chip formation (the chip must be easily transportable from the cutting point and, at the same time, it should have the smallest possible plastic deformation, i.e. the lowest possible cutting forces).



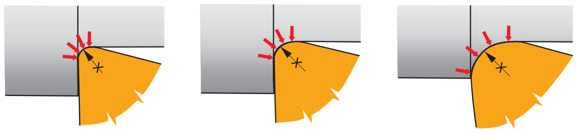
Tool body cross section (clamping rigidity) – the rigidity of the clamping itself is important. Therefore, we recommend using cutting tools with the largest possible body cross section that can be clamped. We also prefer to use monoblocks (PSC).



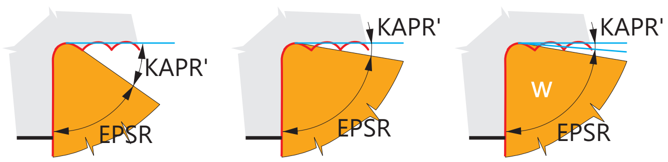
Overhang (clamping rigidity) – the tool overhang, or the rigidity of the clamping itself, is of great importance as well. Therefore, we recommend minimising the overhang.



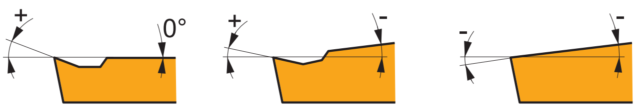
Main cutting edge angle – ideally, the tool adjustment angle should be close to 90°, i.e. the forces should be directed as much as possible to the axis of the machine spindle.



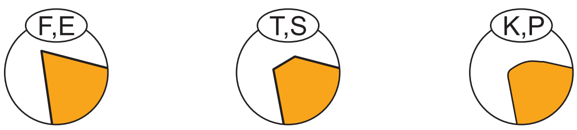
Insert radius – if there is a risk of vibration, we recommend selecting an insert with the smallest possible tip radius.



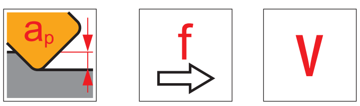
Corner angle and minor cutting edge angle – in this case it is recommended to choose an insert with the smallest possible tip angle, i.e. V (35°), D or K (55°), or T (60°).C or W shapes, or inserts with WIPER geometry, are not recommended. Please note this does not always apply (if torsional vibrations also occur, the application of these inserts can be beneficial).



Geometry – when turning thin-walled and slim parts, it is recommended to opt for positive inserts with positive geometry, then negative inserts with positive geometry, and only if there is no other choice, then negative inserts with neutral to negative geometry should be chosen.



Micro-geometry (cutting edge design) – to reduce the risk of vibration, it is necessary to select inserts with the sharpest possible cutting edge geometry. If you use inserts with t-landing, these should be as narrow and positive as possible. It is very important that the tool generates as little cutting resistance as possible.

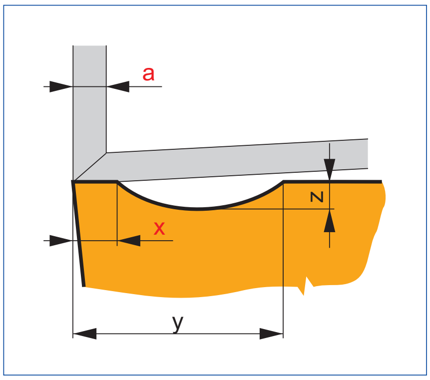


Cutting conditions:
1) When choosing the depth of cut, always make sure that the depth of cut is greater than the radius of the insert.
2) When choosing the feed, take into account that the specific cutting resistance increases with decreasing chip thickness, i.e. do not use extremely low feeds (below 0.1 mm).
3) If there are vibrations, cutting speed change (+/-) can also help – this is related to the inherent frequency of the machine.
NOTE: It is often advantageous to reduce the depth of cut (not below the radius) and increase the feed.

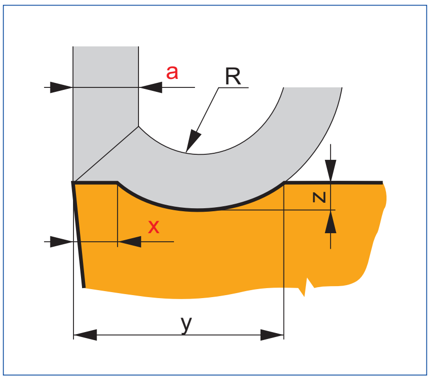
Do not use worn inserts – abrasion of the flank causes an increase in cutting forces, and thus the risk of vibration.
If tool clamping allows it, move the cutting edge above the axis (in external turning) by approx. 2% of the diameter.

Choosing the chip breaker

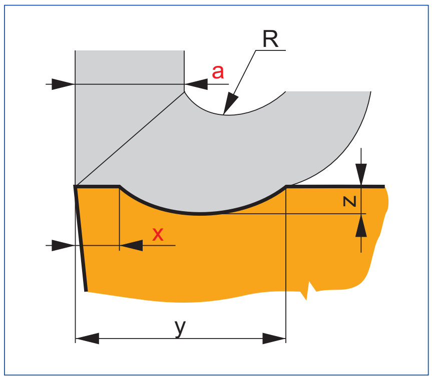
The shape of the chip depends on several factors – the properties of the machined component, material strength, toughness and micro-structure, properties of the insert grade, especially the frictional properties (on the rake face), geometry of cutting edge, cutting conditions and the type of chip breaker, also static and dynamic properties of the machine.
Virtually all of these factors in the cutting process work to combine and determine the shape of the chip (shearing action, flow of the chip, or curled chip – which can gather and clog the machining area). Each chip breaker works in a defined range of feed and depth of cut.
The minimum feed at which the chip breaker functions depends on the width of Top Land „x" and it's angle. The maximum feed depends on the distance from the cutting edge to the end of the chip breaker *y* and the depth of the chip breaker *z*.



If the thickness of layer „*a*" cut away (at setting angle *KAPR* = 90°, equal to the feed) is significantly smaller than the T–land „*x*", the chip is only in contact with the chamfer. It cannot enter the chip breaker and therefore it cannot be broken (see picture).

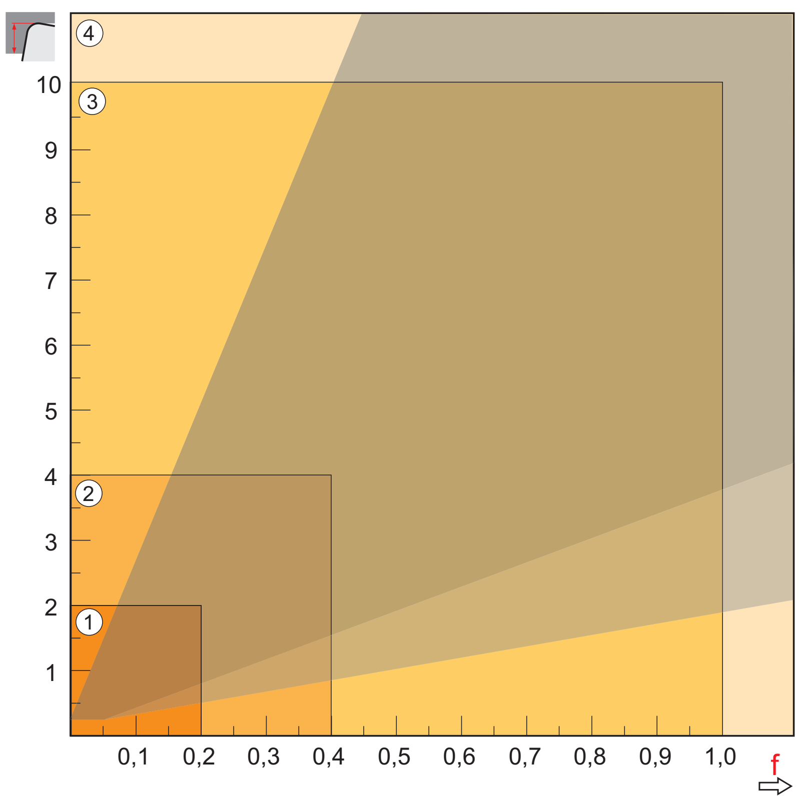


If the feed „*f*" is greater (thickness greater than the depth of „*a*" and *x* < *a* (*f*), the chip enters the chip breaker and is curved at specific values of radius *R* (see picture).



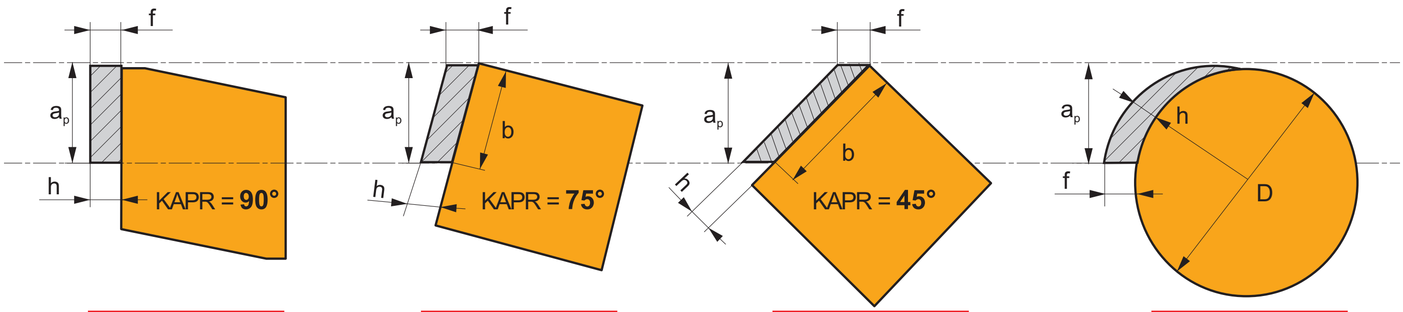
If *x* << *a* (see picture) the chip is excessively deformed (chip is crushed). If the chip misses the chip breaker it will not be broken.

All chip breakers work in a defined range of cutting conditions. This is why the chip breaking area is shown as a continuous range in order to define the most commonly used depth of cut and feed combinations (see following picture). The chip breaker application ranges also overlap.



	Chip breaker	f	a_p
1	FF, F	0.05 – 0.2	0.05 – 2
2	M	0.2 – 0.4	2 – 4
3	R	0.4 – 1.0	4 – 10
4	HR	> 1.0	> 10

TECHNICAL INFORMATION – INDEXABLE TURNING – CHOICE OF CUTTING TOOL



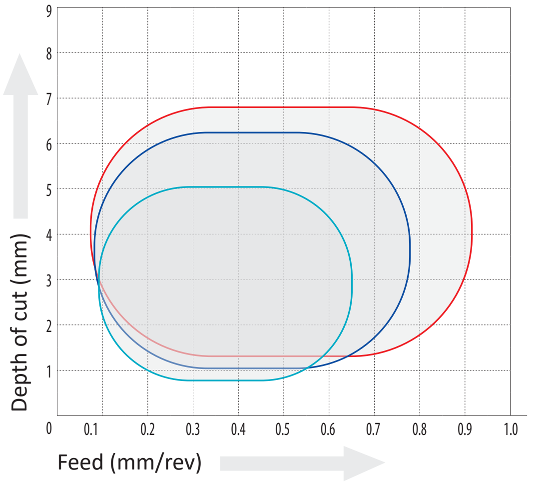
$h = f$ | $h = 0.96 \, . \, f$ | $h = 0.71 \, . \, f$ | $h = 0 - f$

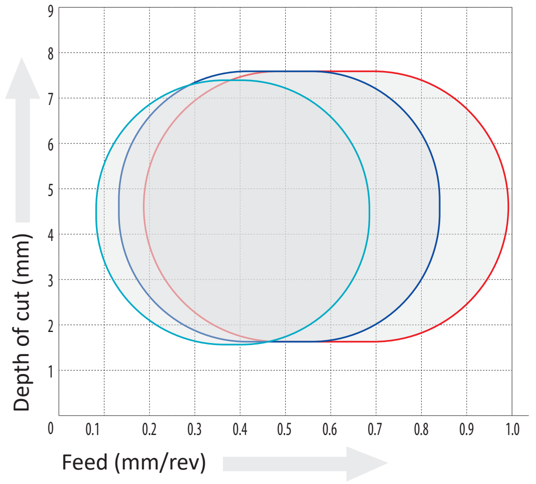
Optimal combination of depth of cut and feed varies for each material. The following table shows ranges of the optimal ratios ***b*** (chip width) to ***h*** (chip thickness). For adjustment angles close to 90°, this is essentially the ratio of depth of cut to feed. See picture.

Material		min b/h	max b/h	
P		5	15	
M		8	12	
K		3	30	
N		9	11	
S	Square chip b=h	8	12	Ribbon chip b/ h > 30
H		5	20	

As follows from the table, when choosing cutting conditions, you should avoid the so-called square chip, i.e. values where the width is close to the thickness of the chip and, on the other hand, the ribbon chip, i.e. high depths of cut in combination with low feed.
The above table shows that the most problematic chip formation relates to non-ferrous metal alloys, in particular aluminium alloys with or without a low silicon content. This is followed by superalloys and stainless steels (especially austenitic and duplex steels). Next are steels, and the best situation is with hardened materials and cast irons.

Also, keep in mind that the chip formation diagram moves slightly towards higher depths of cut (up), with increasing cutting edge length (insert size), and towards higher feeds (to the right), with increasing radius.





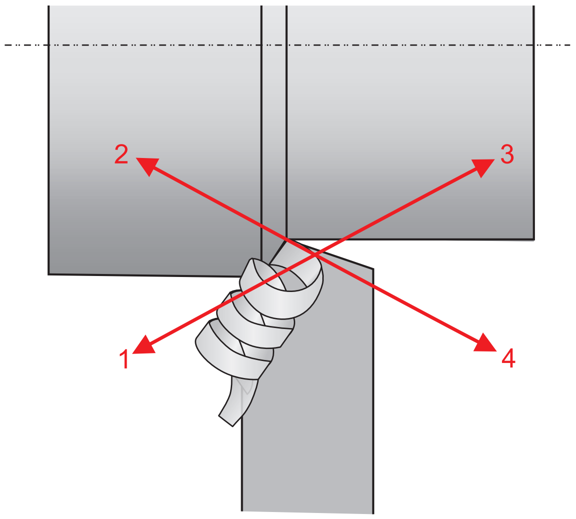
- An insert with the diameter of an inscribed circle IC = 19.050 (mm)
- An insert with the diameter of an inscribed circle IC = 15.875 (mm)
- An insert with the diameter of an inscribed circle IC = 12.700 (mm)

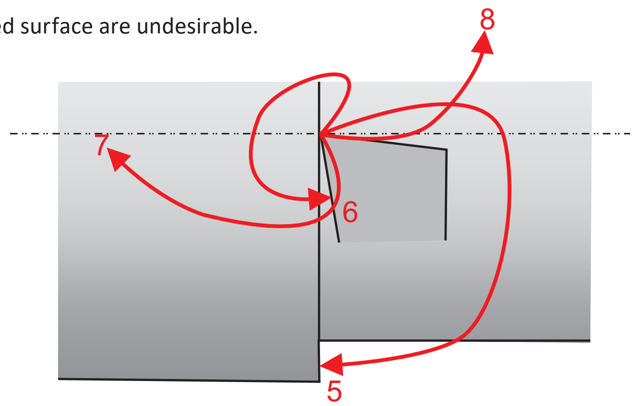
- An insert with the tip radius RE = 1.6 (mm)
- An insert with the tip radius RE = 1.2 (mm)
- An insert with the tip radius RE = 0.8 (mm)

Aside from shape of the chip, the direction of its evacuation is also very important. The following figure shows the basic directions of chip evacuation:

1 – from the workpiece in the feed direction,
2 – to the workpiece in the feed direction
3 – to the workpiece against the feed,
4 – from the workpiece against the feed,
5 – broken against the cutting area surface,
6 – broken against the side of the tool,
7 – broken against the surface being machined,
8 – broken against the machined surface,

Obviously, directions that can cause damage or scratching of the machined surface are undesirable.

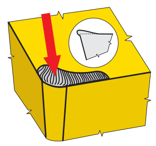


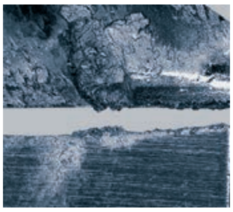


The following section clearly specifies all the geometries we offer you in structured groups. These tables should give you an optimal and more accurate choice.

TYPES OF WEAR ON TURNING INSERTS

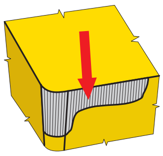
BUILT–UP EDGE

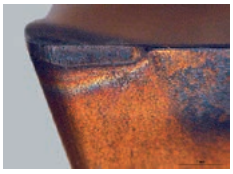




		It has no influence
(MT)CVD / PVD	++	Any coating (decisive factor is anti-adhesion effect)
f	↑	The higher the feed rate the less probability of built-up edge creation
v	↓↑	Change (generally increase) the cutting speed
a_p		It has no influence
	↓↑	Use more positive geometry (built up edge is not created when the rake angle is more than 40°)
	-	Use a coolant with more effective anti-sticking properties (or no coolant at all)

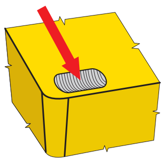
FLANK WEAR





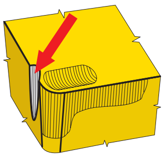
	↑	Use a more wear resistant substrate (S)
(MT)CVD / PVD	++	Any coating (decisive factor is oxidation resistance – α Al_2O_3)
f	↑	Feed has influence on shape and position of groove
v	↓	Decrease cutting speed
	↑	Minimal effect
a_p	+	Use another (more positive) cutting geometry
	+	Use coolant or increase its intensity

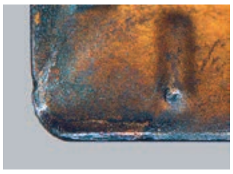
CRATERING



	↑	Use a more wear resistant substrate (S)
(MT)CVD / PVD	++	Any coating (decisive factor is thermal resistance – α Al_2o_3)
f	↑	Feed has influence on shape and position of crater
v	↓	Decrease cutting speed
a_p	↓	Minimal effect
	↑	Use more positive cutting geometry
	++	Use coolant or increase its intensity

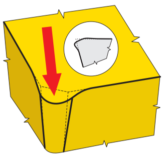
OXIDATION GROOVE ON THE MINOR EDGE

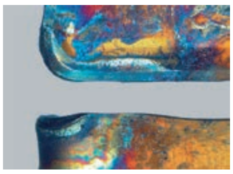




	↑	Use a more wear resistant substrate (H)
(MT)CVD PVD	+ +	Any coating (decisive factor is hardness – TIC, TICN)
f	↓	Increase feed (especially if it is under 0.1 mm)
v	↓	Decrease cutting speed
a_p	↓	It has no influence
	↑	Increase the clearance angle
	+ +	Use a coolant or increase its intensity

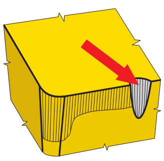
PLASTIC DEFORMATION

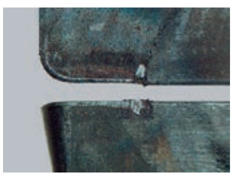




	↑	Use a more wear resistant grade (decisive factor is content of Co)
(MT)CVD PVD	+	Any coating (decisive factor is friction)
f	↓	Decrease feed rate
v	↓	Decrease cutting speed
a_p	↓	Minimal effect
	↑	Use another (more positive) cutting geometry
	+ +	Use coolant or increase its intensity

SIDE FLANK NOTCH – REMEDY

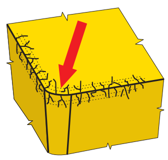


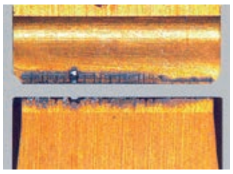


	↑ ↓	It depends on the character of the damage (abrasive – use more wear resistant substrate; breaking – use tougher substrate)
(MT)CVD PVD	+ +	CVD coating (decisive factor is oxidation resistance – α Al_2O_3)
f	↓	Feed has influence on intensity, but less than the cutting speed
v	↓	Decrease cutting speed
a_p	↑↓	Use unequal depth of cut
	↓	Use less positive cutting geometry
	+	Use coolant or increase its intensity
		Use tool with smaller setting angle

TYPES OF WEAR ON TURNING INSERTS

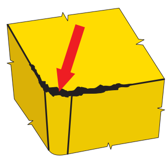
CREATION OF RACK CRACKS

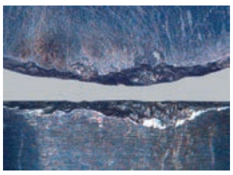




	↓	(H) grain has a great influence
(MT)CVD PVD	+ +	PVD coating recommended
f	↓	Feed has influence on intensity, but less than the cutting speed
V	↓	Lower speed means lower temperature
a_p		It has no influence
	↓	Use less positive cutting geometry
	- - -	No coolant (it is possible to use air to remove chips from cutting area)

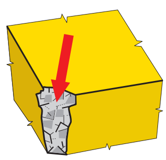
BRITTLE CRACKS AT THE CUTTING EDGE

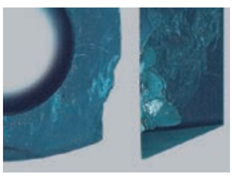




	↓	(H) grain has a great influence
(MT)CVD PVD	+	PVD coating recommended
f	↓	Good swarf control is very important
V	↑↓	It is about swarf control and vibration
a_p	↓	Reduces the force load (important for machining with long overhangs)
	↓	Use less positive cutting geometry
		It has no influence
		Use better working conditions, reduce feed rate until insert is in cut

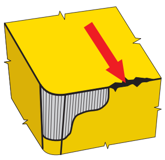
INSERT FRACTURE





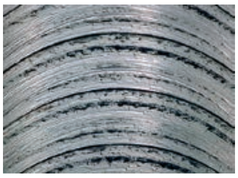
	↓	(H) grain has a great influence
(MT)CVD PVD	+	PVD coating recommended
f	↓	Reduces the force load
V	↑↓	It is about swarf control and vibration
a_p	↓	Reduces the force load
	↓	Use less positive cutting geometry
		It has no influence
		Use better working conditions

FAILURE OF CUTTING EDGE



	↓	(H) grain has a great influence
(MT)CVD / PVD	+	PVD coating recommended
f	↑↓	Good swarf control is very important
v	↑↓	It is about swarf control and vibration
a_p	↑↓	Good swarf control is very important
	↓	Use less positive cutting geometry
		It has no influence
		Problem is poor swarf control or evacuation of chips

POOR SURFACE QUALITY



Description and cause:

Numerous causes depending on the workpiece material, cutting conditions (feed rate and cutting speed), the condition of the cutting edge, the extent and type of wear, and the condition and rigidity of the machine–tool–workpiece assembly.

- incorrect tool chosen
- incorrect chip thickness
- incorrect cutting speed
- coolant is needed
- high feed rate

Corrective measures:

- use a wiper insert
- use a cutting insert with the right geometry
- reduce the feed rate
- change (usually increase) the cutting speed
- use a coolant
- improve the stability of the tool and workpiece
- change the chip cross section
- select a more easy–cutting chip breaker
- increase the nose radius

VIBRATIONS

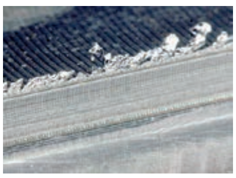
Description and cause:

This is a very common problem, which is mainly caused by an unbalanced workpiece or tool, unstable fixing of the workpiece, high cutting forces or tool overhang.

Corrective measures:

- improve the stability of the tool and workpiece
- reduce the depth of cut
- minimize tool overhang
- reduce the cutting speed
- use a tool with smaller setting angle
- reduce the chip cross section
- use a tool with a low cutting resistance
- increase the feed rate
- select a more easy–cutting chip breaker
- increase the nose radius

BURRS



Description and cause:

This usually occurs on soft steels and plastic materials.

Corrective measures:

- use a cutting insert with a sharp cutting edge
- use a cutting insert with positive geometry
- use a tool with a smaller setting angle

ERRORS IN DIMENSIONS AND SHAPE OF WORKPIECE

Description and cause:

Depends on a number of factors.

Corrective measures:

- use a wear–resistant cutting insert
- improve the stability of the cutter and workpiece
- minimize tool overhang
- use a workpiece with a suitable machining allowance

INADEQUATE CHIP FORMATION

Description and cause:

Producing a chip with a suitable shape is very important to insert durability and service life of the tool. The workpiece material, the feed rate, the depth of cut and the cutting geometry all have an effect on chip forming. A chip that is too long is unacceptable for various reasons, while a chip that is too short is undesirable as it overloads the cutting edge and causes vibrations.

Corrective measures:

- change the feed rate and depth of cut
- use a more suitable cutting geometry
- change the cutting conditions

CHECK THE SEAT CONDITION OF THE CUTTING INSERT

Before clamping a new cutting insert or changing the edge, it is necessary to clean the seat and check its condition or the condition of the anvil and wedge (especially the damage under the corner of the cutting insert).

CHECK AND SERVICE THE CLAMPING PARTS

It is also important to check the clamping parts, including clamping levers, screws, wedges and clamps. Only use original, undamaged parts (found in the catalogue). Regularly lubricate the threads and the binding surface of screws, for example using heat–resistant lubricant (Molykote G.). For assembly and disassembly, only use screwdrivers and wrenches specified in our catalogue or recommended by the tool manufacturer. Pay attention to the correct tightening (proportional) – it is advisable to use a torque wrench.

CHECK THE TIGHTENING

Before tightening, check the fit of the cutting insert on the whole of the binding surface and in the radial and axial directions. Cutting inserts and tools must always be clean and undamaged.

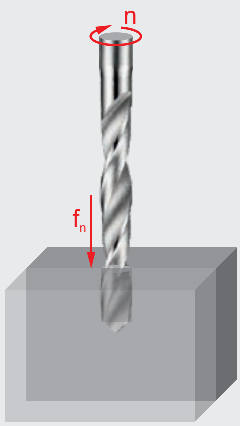
TECHNICAL INFORMATION – INDEXABLE TURNING – FORMULA FOR CALCULATING CUTTING DATA

Value	Formula	Unit	Note
Number of revolutions	$n = \frac{v_c \cdot 1000}{D \cdot \pi}$	(1/min)	n Number of revolutions (1/min) D Diameter (of tool or workpiece) (mm) v_c Cutting speed (m/min) f_{rev} Feed per revolution (mm/rev) f_{min} Feed per minute (Linear Feedrate) (mm/min)
Cutting speed	$v_c = \frac{\pi \cdot D \cdot n}{1000}$	(m/min)	
Feed per revolution	$f_{rev} = \frac{f_{min}}{n}$	(mm/rev)	
Feed per minute (Linear Feedrate)	$f_{min} = v_f = f_{rev} \cdot n$	(mm/min)	
Max. height of profile R_{max}	$R_{max} = \frac{125 \cdot f_{rev}^{\ 2}}{RE}$	(µm)	R_{max} max. height of profile (mm) R_a surface finish (mm) f_{rev} feed per revolution (mm/rev) RE nose radius (mm)
Surface finish R_a	$R_a = \frac{43{,}9 \cdot f_{rev}^{\ 1{,}88}}{RE^{\ 0{,}97}}$	(µm)	
Chip cross section	$A = f_{rev} \cdot a_p$	(mm^2)	A Chip cross section (mm^2) f_{rev} Feed per revolution (mm/rev) a_p Axial depth of cut (mm) κ_r Primary edge setting angle (°) h Chip thickness (mm) v_c Cutting speed (m/min) f_{min} Feed per minute (Linear Feedrate) (mm/min) Q Material removal rate per minute (cm^3/min) $INSD$ Insert diameter (mm)
Chip thickness (For insert with straight edge)	$h = f_{rev} \cdot \sin \kappa_r$	(mm)	
Chip thickness (For round cutting insert)	$h = f_{rev} \cdot \sqrt{\frac{a_p}{INSD}}$	(mm)	
Metal removal rate	$Q = a_p \cdot f_{rev} \cdot v_c$	(cm^3/min)	
Power demand	$P_c = \frac{a_p \cdot f_{rev}^{\ 1-c} \cdot k_{c1} \cdot v_c \cdot k\kappa_r}{6 \cdot 10^4 \cdot \eta}$	(kW)	P_c Power demand (kW) a_p Depth of cut (mm) f_{rev} Feed (mm/rev) c Constant KTV (1) k_c Specific cutting force (MPa) $k_{\kappa r}$ κ_r angle constant (1) η Efficiency (usually $\eta = 0{,}75$) (1) x Machined material constant (1)
Approximate power demand	$P_c = \frac{a_p \cdot f_{rev} \cdot v_c}{x}$	(kW)	

Material	Steel	Cast iron	Al
Coefficient *x*	20	25	100

HOLEMAKING – TECHNICAL INFO

TECHNICAL INFORMATION – DRILLING FEED RATE CHART



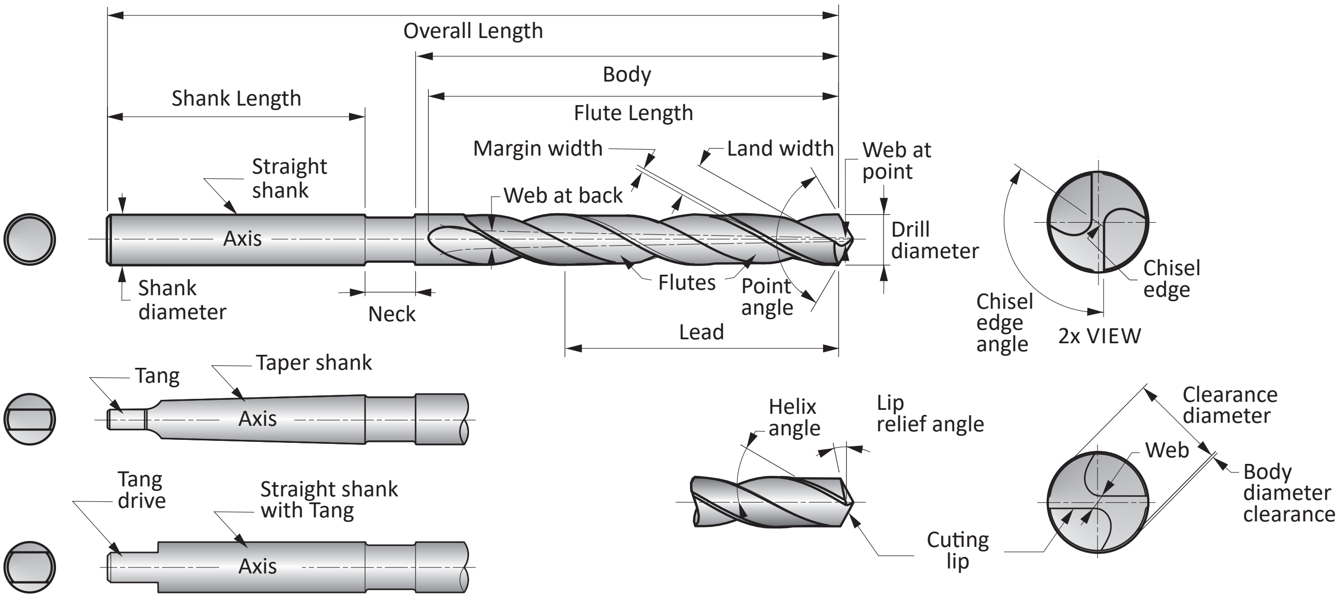
Feed per revolution (f_n in mm/rev)
Depending on the working conditions it might be necessary to adjust these values ± 25 %.

How to use this table to find the feed per revolution (f_n):

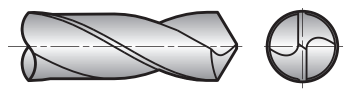
1. Find your Alpha Code on the product page (example: 46J, "J" is the Alpha Code).
2. Find the closest diameter for your cutting application in the top row of the table.
3. Find your Alpha Code in the left column of the table.
4. The intersection (cell) of the Diameter and Alpha Code is the feed per revolution (f_n).

Feed rates	ø DC (mm)																		
	0.15	**0.50**	**1.00**	**2.00**	**3.00**	**4.00**	**5.00**	**6.00**	**8.00**	**10.00**	**12.00**	**15.00**	**16.00**	**20.00**	**25.00**	**30.00**	**40.00**	**50.00**	**100.00**
A	0.003	0.006	0.012	0.023	0.029	0.032	0.036	0.042	0.054	0.062	0.069	0.082	0.086	0.110	0.125	0.135	0.155	0.175	0.263
B	0.004	0.007	0.014	0.028	0.037	0.041	0.046	0.053	0.067	0.080	0.090	0.103	0.108	0.135	0.153	0.165	0.188	0.208	0.312
C	0.004	0.008	0.015	0.032	0.044	0.050	0.056	0.064	0.080	0.098	0.110	0.125	0.130	0.160	0.180	0.195	0.220	0.240	0.360
D	0.004	0.008	0.016	0.038	0.053	0.060	0.068	0.078	0.098	0.119	0.130	0.149	0.155	0.188	0.210	0.228	0.253	0.275	0.413
E	0.004	0.009	0.017	0.043	0.062	0.071	0.080	0.092	0.115	0.140	0.150	0.173	0.180	0.215	0.240	0.260	0.285	0.310	0.465
F	0.005	0.009	0.018	0.050	0.073	0.084	0.095	0.109	0.138	0.165	0.178	0.202	0.210	0.248	0.275	0.295	0.320	0.343	0.515
G	0.005	0.010	0.019	0.056	0.084	0.096	0.109	0.126	0.160	0.190	0.205	0.231	0.240	0.280	0.310	0.330	0.355	0.375	0.563
H	0.005	0.010	0.020	0.066	0.102	0.116	0.130	0.150	0.190	0.228	0.243	0.271	0.280	0.320	0.355	0.375	0.398	0.418	0.627
I	0.005	0.011	0.021	0.076	0.119	0.134	0.150	0.173	0.220	0.265	0.280	0.310	0.320	0.360	0.400	0.420	0.440	0.460	0.690
J	0.006	0.012	0.024	0.084	0.135	0.152	0.170	0.197	0.250	0.298	0.315	0.349	0.360	0.405	0.445	0.465	0.485	0.503	0.755
K	0.007	0.013	0.026	0.092	0.150	0.170	0.190	0.220	0.280	0.330	0.350	0.388	0.400	0.450	0.490	0.510	0.530	0.545	0.818
L	0.007	0.014	0.028	0.101	0.165	0.186	0.208	0.240	0.305	0.360	0.385	0.419	0.430	0.485	0.525	0.545	0.568	0.588	0.882
M	0.008	0.015	0.030	0.110	0.180	0.202	0.225	0.260	0.330	0.390	0.420	0.450	0.460	0.520	0.560	0.580	0.605	0.630	0.945
N	0.008	0.016	0.032	0.119	0.195	0.218	0.242	0.280	0.355	0.420	0.455	0.481	0.490	0.555	0.595	0.615	0.642	0.672	1.008
S	0.002	0.004	0.008	0.014	0.020	0.025	0.030	0.037	0.050	0.080	0.100	0.123	0.130	0.150	0.170	0.190	0.220	0.240	–
T	0.004	0.008	0.015	0.028	0.040	0.050	0.060	0.070	0.090	0.110	0.130	0.160	0.170	0.190	0.210	0.230	0.260	0.275	–
U	0.007	0.013	0.026	0.048	0.070	0.080	0.090	0.107	0.140	0.170	0.200	0.223	0.230	0.240	0.270	0.300	0.360	0.375	–
V	0.010	0.019	0.038	0.069	0.100	0.115	0.130	0.153	0.200	0.250	0.280	0.310	0.320	0.340	0.400	0.440	0.510	0.530	–
W	0.012	0.025	0.049	0.089	0.130	0.150	0.170	0.200	0.260	0.330	0.380	0.418	0.430	0.450	0.470	0.490	0.520	0.540	–
X	0.014	0.028	0.056	0.103	0.150	0.180	0.210	0.250	0.330	0.420	0.480	0.533	0.550	0.580	–	–	–	–	–
Y	0.017	0.034	0.068	0.124	0.180	0.220	0.260	0.317	0.430	0.550	0.700	0.700	0.700	0.740	–	–	–	–	–
Z	0.024	0.047	0.094	0.172	0.250	0.325	0.400	0.533	0.800	1.000	1.100	1.175	1.200	1.200	–	–	–	–	–

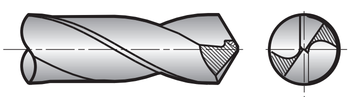
Drill Nomenclature



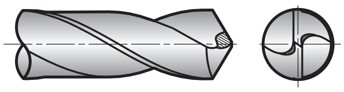
- **Axis** — The imaginary straight line which forms the longitudinal centre line of a drill.
- **Backtaper** — A slight decrease in diameter from front to back in the body of a drill.
- **Body** — The portion of a drill extending from the shank or neck to the outer corners of the cutting lips.
- **Body Clearance Diameter** — The portion of the land that has been cut away so it will not bind against the walls of the hole.
- **Chisel-Edge** — The edge at the end of the web that connects the cutting lips.
- **Chisel-Edge Angle** — The included angle between the chisel-edge and cutting lip, as viewed from the end of a drill.
- **Clearance Diameter** — The diameter over the cut away portion of the drill lands.
- **Drill** — A rotary end cutting tool having one or more cutting lips, and having one or more helical or straight flutes for the passage of chips and the admission of a cutting fluid.
- **Drill Diameter** — The diameter over the margins of a drill measured at the point.
- **Flute Length** — The length from the outer corners of the cutting lips to the extreme back of the flutes. Includes the sweep of the tool used to generate the flutes and therefore does not indicate the usable length of flutes.
- **Flutes** — Helical or straight grooves cut or formed in the body of a drill to provide cutting lips, permit removal of chips, and allow cutting fluid to reach the cutting lips.
- **Helix Angle** — The angle formed by the leading edge of the land with a plane containing the axis of a drill.
- **Land** — The peripheral portion of the body between adjacent flutes.
- **Land Width** — The distance between the leading edge and heel of the land; measured at a right angle to the leading edge.
- **Lead** — The axial advance of a leading edge of the land in one turn around the circumference.
- **Lip Relief Angle** — The axial relief angle at the outer corner of the lip; measured by projection to a plane tangent to the periphery at the outer corner of the lip.
- **Lips** — The cutting edges of a two flute drill extending from the chisel-edge to the periphery.
- **Margin** — The cylindrical portion of the land, which is not cut away, to provide clearance.
- **Neck** — The section of reduced diameter between the body and the shank of a drill.
- **Overall Length** — The length from the extreme end of the shank to the outer corners of the cutting lip. It does not include the conical shank end often used on straight shank drills, nor the conical cutting point used on both straight and taper shank drills.
- **Point** — The cutting end of a drill, made up of the ends of the lands and the web. In form, it resembles a cone, but departs from a true cone to furnish clearance behind the cutting lips.
- **Conventional** — Conventional Points with 118° included point angles are the most commonly used because they provide satisfactory results in a wide variety of materials. A possible limitation is that the straight chisel edge contributes to wandering at the drill point, often making it necessary to spot the hole for improved accuracy.



- **Split** — Split-Points (commonly called Crankshaft Points) were originally developed for use on drills designed for deep oil holes in automotive crankshafts. Since its inception, the split-point has gained widespread use and is applied to both 118° and 135° included point angles. Its main advantages are the ability to reduce thrust and eliminate wandering at the drill point. This is a distinct advantage when the drill is used in a portable drill or in drilling applications where bushings cannot be used. The split-point also has two positive rake cutting edges extending to the centre of the drill, which can assist as a chipbreaker to produce small chips which can readily be ejected.



- **Notched** — Notched Points were developed for drilling tough alloys. Commonly incorporated on heavy web drills, which allow the point to withstand the higher thrust loads required in drilling these materials. As with the split-point, the Notched Point contains two additional positive rake cutting edges extending toward the centre of the drill. These secondary cutting lips, which extend no further than half the original cutting lip, can assist in chip control and reduce the torque required in drilling tough materials. Notched Points can be incorporated on both 118° and 135° included point angles, making them suitable for drilling a wide variety of materials.



- **Point Angle** — The included angle between the cutting lips projected upon a plane parallel to the drill axis and parallel to the two cutting lips.
- **Relative Lip Height** — The difference in indicator reading between the cutting lips of a drill. Measured at a right angle to the cutting lip at a specific distance from the axis of the tool.
- **Shank** — The part of a drill by which it is held and driven.
- **Tang** — The flattened end of a taper shank, intended to fit into a driving slot in a socket.
- **Tang Drive** — Two opposite parallel driving flats on the extreme end of a straight shank.
- **Taper Shank** — Drills having conical shanks suitable for direct fitting in machine spindles, driving sleeves, or sockets. Tapered shanks generally have a tang.
- **Web** — The central portion of the body that joins the lands. The extreme end of the web forms the chisel-edge on a two flute drill.
- **Web Thickness** — The thickness of the web at the point, unless another specific location is indicated.

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.

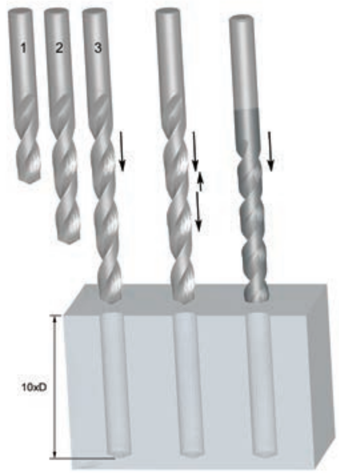
5. The use of suitable coolants and lubricants are recommended as required by the particular drilling operation. When using coolants and lubricants, ensure a copious supply, especially at the drill point.

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 make sure that the correct point geometry is produced and that any wear has been removed.

Deep hole drilling strategy

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× the diameter of the drill.



	Series Drilling	Series Drilling
No of drills	3 (2.5×D, 6×D,10×D)	2 (2.5×D,10×D)
Type of drill	Standard geometry, general purpose	Standard geometry, general purpose
+ / -	Expensive Time consuming	More cost effective Quick

	Peck Drilling	Single Pass Drilling
No of drills	1 (10×D)	1 (10×D)
Type of drill	Standard geometry, general purpose	Purpose specific tools
+ / -	Time consuming	Cost effective Fast

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 from spindle
Worn outer corner	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
	Ineffective swarf clearance	Adjust speed, feed and peck length to achieve more manageable swarf

Hole Size / Achievable Hole Tolerances

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 favorable 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) – H10
Solid Carbide High Performance TiN / TiALN coated (CDX, Force) – H8/H9

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.

Optimizing the Drilling Operation / Troubleshooting

Drill Selection
Use the shortest drill the application will permit in order to achieve maximum tool rigidity.

Holders
Tool holders and collets must provide good concentricity between the drill and the machine spindle. Use a positive back stop to prevent the tool from backing up into the holder. Never clamp the tool over the flutes or over-tighten the holder. Static runout in the tool assembly must be accurately checked and maintained.

Workpiece
A secure and rigid workpiece to minimize deflection is needed, particularly on through-hole applications.

Coolants
Coolants are recommended when drilling mild steel and high temperature alloys. The purpose of the coolant media is to direct the chips away from the cutting tool and workpiece. Excessive coolant pressure and/or too much volume can negatively affect performance. When using coolant fed drills, the coolant pressure that is required should be higher than normal. Suggested pressure for coolant fed drills is minimally 10.3 bar or 150 PSI. As the diameter of the drill is reduced, the higher the pressure. This is to assist the chip in evacuating from a more confined area.

Drilling Troubleshooting Guide

Problem	Solution
Wear on Outer Corners	Reduce cutting speed
	Increase feed (IPR)
	Improve direction of coolant flow
	Increase coolant pressure
	Add corner break
Chipping of Chisel Edge	Check accuracy of drill runout
	Check workpiece clamping accuracy and movement
	Check point centrality and lip height
	Increase feed rate
Chipping of Cutting Lips	Check accuracy of drill runout
	Check workpiece clamping accuracy and movement
	Reduce speed
	Reduce point clearance
	Increase hone
Cracking of Lands	Check movement of workpiece
	Increase back taper
	Check accuracy of drill runout
	Chip packing; increase flute form opening or peck drill (HSS or HSCO only)
	Slow down helix, horizontal drilling
	Increase feed
	When spot drilling, reduce feed
	Improve direction of coolant flow
	Increase coolant pressure
Oversize Hole	Increase speed, reduce feed
	Check workpiece clamping accuracy and movement
	Check accuracy of drill runout
	Chip packing, increase flute form opening or peck drill (HSS or HSCO only)
	Check point centrality and lip height
Undersize Hole	Improve direction of coolant flow
	Reduce cutting speed, increase feed
	Check drill diameter
Hole Not Round	Check accuracy of drill runout
	Check workpiece clamping accuracy and movement
	Check point centrality and lip height
	Chip packing, increase flute form opening or peck drill (HSS or HSCO only)
Drill Breakage	Chip packing, increase flute form opening or peck drill (HSS or HSCO only)
	Check workpiece clamping accuracy and movement
	Check accuracy of drill runout
	Reduce feed rate, increase feed rate
	Improve direction of coolant flow
	Increase coolant pressure

	Grade	Hardness (HV10)	C %	W %	Mo %	Cr %	V %	Co %	Tool Material
HSS	**M2**	810 – 850	0.9	6.4	5.0	4.2	1.8	–	HSS
HSS-E	**M35**	830 – 870	0.93	6.4	5.0	4.2	1.8	4.8	HSCO
	M42	870 – 960	1.08	1.5	9.4	3.9	1.2	8.0	

Properties	HSS materials	Carbide materials	K10/30F (often used for solid tools)
Hardness (HV30)	800-950	1300 – 1800	1600
Density (g/cm³)	8.0 – 9.0	7.2 – 15	14.45
Compressive strength (N/mm²(	3000 – 4000	3000 – 8000	6250
Flexural strength, (bending) (N/mm²(	2500 – 4000	1000 – 4700	4300
Heat resistance (°C)	550	1000	900
E-module (KN/mm²)	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.

Surface treatment / Coating properties examples

Surface Treaments	Colour	Coating material	Hardness (HV)	Thickness (µm)	Coating structure	Frict. coeff. against steel	Max. appl. temp. (°C)
ST	Dark grey	Fe_3O_4	400	Max. 5	Conversion into the surface	–	550
Bronze	Bronze	Fe_3O_4	400	Max. 5	Conversion into the surface	–	550
TiN	Gold	TiN	2300	1 – 4	Mono-layer	0.4	600
TiAlN	Black grey	TiAlN	3300	3	Nano structured	0.3 – 0.35	900

Tool materials

High Speed Steel	HSS	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.

Carbide materials

Carbide Materials (or Hard Materials)	HM	A sintered powder metallurgy substrate, 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.

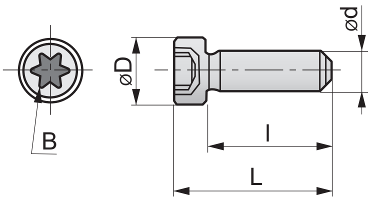
Surface Coatings

Bright Nickel Plating	Bright	Bright Nickel Plated surface protects hardened steel body from rust, corrosion and also improves chip evacuation.
Ti-phon (TiAlCrSiN)	Ti-phon	Ti-phon Coating is a coating similar to TiAIN but with the addition of Chromium (Cr) and Silicon (Si) which is specially formulated for Hydra Heads to prevent edge build-up and greatly improve chip flow. This coating exhibits high hot hardness, high oxidation resistance and superior lubricity when used on tools for machining applications involving heavy mechanical and thermal stresses, high speeds and high feed rates. These coating properties translate into superior wear resistance and edge strength.

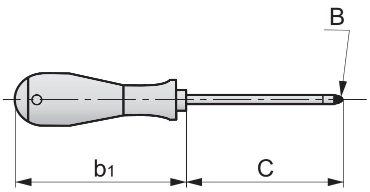
Torque table

 H860	 H861	 Hydra Head ø Metric Range	 Hydra Head ø Fractional Range	 Hydra Head ø Decimal Size Range (min. / max.)	Torque Values Nm (Metric System)	Torque Values in/lbs (Inch System)
H860N1	**H861N1**	12.0 mm – 15.5 mm	15/32“ – 39/64“	0.4688“ – 0.6102“	0.75 – 0.99	6.6 – 8.8
H860N2	**H861N2**	15.6 mm – 18.5 mm	5/8“ – 23/32“	0.6142“ – 0.7283v	0.93 – 1.24	8.2 – 11.0
H860N3	**H861N3**	18.6 mm – 21.5 mm	47/64“ – 27/32“	0.7323“ – 0.8465“	1.84 – 2.44	16.3 – 21.6
H860N4	**H861N3**	22.0 mm – 24.5 mm	55/64“ – 31/32“	0.8594“ – 0.9688“	2.73 – 3.72	24.2 – 32.9
H860N5	**H861N4**	25.0 mm – 27.5 mm	63/64“ – 1-3/32“	0.9843“ – 1.0938“	4.14 – 5.52	36.6 – 48.8
H860N6	**H861N5**	28.0 mm – 33.5 mm	1-7/64“ – 1-19/64“	1.1024“ – 1.3189“	4.97 – 6.63	44.0 – 58.7
H860N7	**H861N6**	34.0 mm – 42.0 mm	1-11/32“ – 1-5/8“	1.3386“ – 1.6535“	7.2	63.7

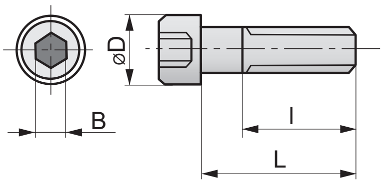
Screws and screw-driver data



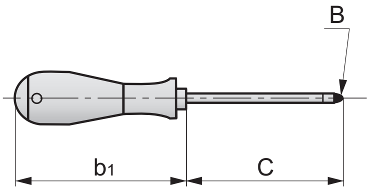
e-code	d	Pitch	L (mm)	l (mm)	D (mm)	B
H860N1	M2.2	0.45	7.5	5.7	3.5	8IP
H860N2	M2.5	0.45	9.0	7.0	4.1	10IP
H860N3	M3.0	0.50	10.5	8.0	4.9	15IP
H860N4	M3.5	0.60	11.5	8.8	5.5	15IP
H860N5	M4.0	0.70	12.5	9.5	6.0	20IP
H860N6	M4.5	0.75	14.3	10.8	6.8	25IP



e-code	B	C	b_1
H861N1	8IP	60	104
H861N2	10IP	80	111
H861N3	15IP	80	111
H861N4	20IP	100	118
H861N5	25IP	100	118

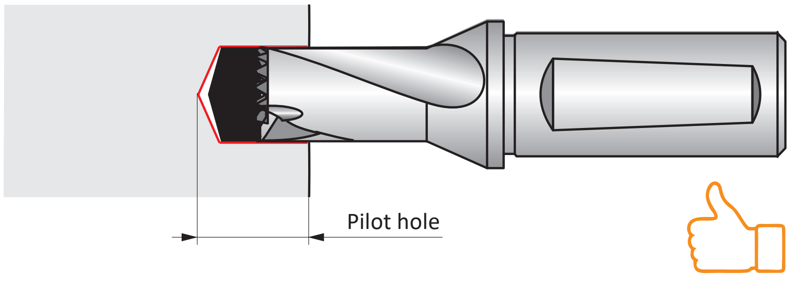


e-code	d	Pitch	L (mm)	l (mm)	D (mm)	B
H860N7	M5.0	0.8	15	full	8.5	4

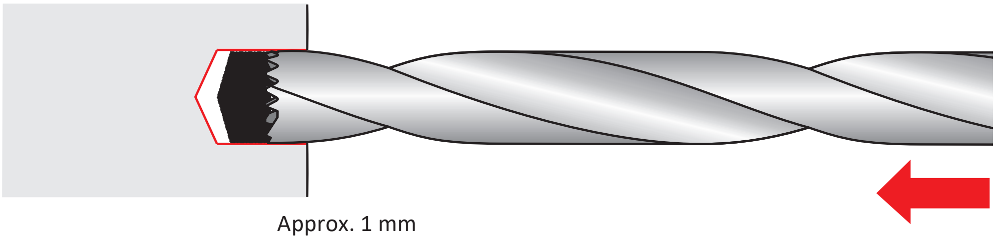


e-code	B	C	b_1
H861N6	4	75	111

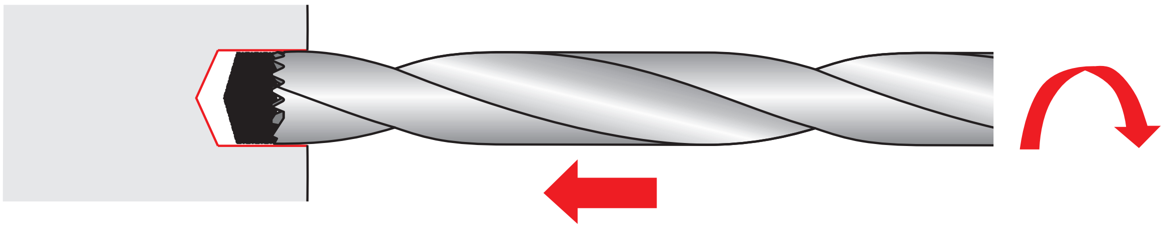
Apply special programming for 8×D and 12xD drilling



Drill a pilot hole (1.5×D to 3×D depth) with the same HYDRA head diameter (if needed check the runout of the drill max. +/ 0.05 mm).

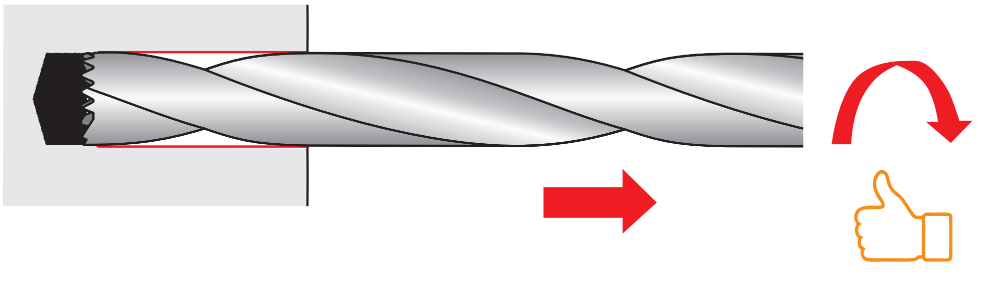


Enter the pilot hole with the 8×D or 12×D Body running a maximum of 500 rpm, to approximately 1mm above the pre-drilled pilot hole depth.



Start coolant flow and increase the rotational speed up to the recommended RPM.
Note: Apply a short dwell time don't start the feed before recommended RPM is reached.

Drill without pecking to the required depth.



When the required depth is reached, retract the drill by approximately 0.1 mm to 0.5 mm and reduce to 500 rpm followed by a complete retraction with normal feed.**Note: retracting the drill with a higher spindle speed may cause a shoulder damage from run out or destroy the hole surface and tolerance.**

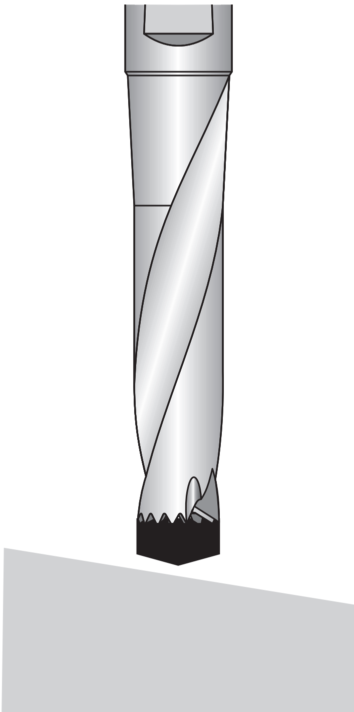
Drilling hints & tips with the hydra drill

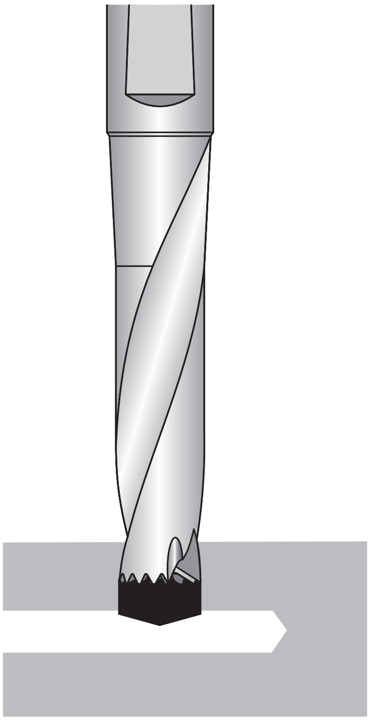
Coolants
For maximum chip evacuation and tool performance, coolant use is recommended. Emulsion coolant concentration of 6 – 8% is recommended for most applications, with a coolant pressure of 20 bar (290 PSI) or higher. For high strength steel, stainless steels and tougher drilling applications, use a higher concentration of 10 – 12%. In these applications, particularly in stainless steels, it is recommended to use the maximum coolant pressure on the machine. The Hydra-drill coolant holes provide improved web strength and reduce heat at the cutting edges for increased productivity and longer tool life.

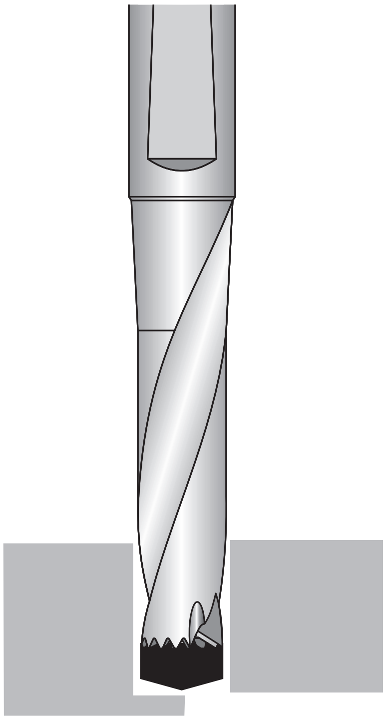
Holders
Always use tool holders and collets that provide good concentricity between the drill and the machine spindle. Use a positive stop to prevent the tool from backing up into the holder. Radial runout in the tool assembly must be accurately checked and maintained.

Workpiece
A secure and rigid workpiece will minimise deflection, and allow for better accuracy and true position of the hole.

Feeds
It is important not to underfeed the drill which will cause it to dwell and dull. This is particularly true in work hardening materials. Feed rates should be high enough for proper chip formation.


Entrance or exit on an angular surface


Drilling through cross holes


Interrupted at exit

In these drilling scenarios, reducing feed rate to 1/3 (33%) is generally recommended. Drilling into an entry angle of more than 10° is NOT recommended – surface should be milled flat first.

	Grade	Hardness (HV10)	C %	W %	Mo %	Cr %	V %	Co %	Tool Material
HSS	M2	810 – 850	0.9	6.4	5.0	4.2	1.8	–	HSS

Properties	HSS materials	Carbide materials	K10/30F (often used for solid tools)
Hardness (HV30)	800 – 950	1300 – 1800	1600
Density (g/cm³)	8.0 – 9.0	7.2 – 15	14.45
Compressive strength (N/mm²)	3000 – 4000	3000 – 8000	6250
Flexural strength, (bending) (N/mm²)	2500 – 4000	1000 – 4700	4300
Heat resistance (°C)	550	1000	900
E-module (KN/mm²)	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.

802D, 803D (XPET..AP, SCET..-UD)

	D9335	D8330	D8345	⌀ 15	⌀ 20	⌀ 25	⌀ 30	⌀ 40	⌀ 58
P1	■	■	■	0.07	0.08	0.09	0.10	0.12	0.16
P2	■	■	■	0.11	0.13	0.15	0.17	0.21	0.28
P3	■	■	■	0.13	0.15	0.18	0.20	0.24	0.32
P4	■	■	■	0.12	0.14	0.16	0.18	0.22	0.30
K1	■	■	■	0.14	0.16	0.19	0.21	0.26	0.34
K2	■	■	■	0.14	0.16	0.19	0.21	0.26	0.34
K3	■	■	■	0.14	0.16	0.19	0.21	0.26	0.34
K4	■	■	■	0.14	0.16	0.19	0.21	0.26	0.34
K5	■	■	■	0.14	0.16	0.19	0.21	0.26	0.34

802D, 803D (XPET..AP-SD, SCET..-SD)

	D9335	D8330	D8345	⌀ 15	⌀ 20	⌀ 25	⌀ 30	⌀ 40	⌀ 58
P1	■	■	■	0.08	0.09	0.10	0.11	0.14	0.18
P2	■	■	■	0.11	0.13	0.15	0.17	0.21	0.28
P3	■	■	■	0.13	0.15	0.18	0.20	0.24	0.32
P4	■	■	■	–	–	–	–	–	–
K1	◩	◩	◩	0.08	0.09	0.10	0.11	0.14	0.18
K2	◩	◩	◩	0.11	0.13	0.15	0.17	0.21	0.28
K3	◩	◩	◩	0.12	0.14	0.16	0.18	0.22	0.24
K4	◩	◩	◩	0.13	0.15	0.18	0.20	0.24	0.32
K5	◩	◩	◩	0.14	0.16	0.19	0.21	0.25	0.33
M1	■	■	■	0.12	0.14	0.16	0.18	0.22	0.30
M2	■	■	■	0.11	0.13	0.15	0.17	0.21	0.28
M3	■	■	■	0.07	0.08	0.09	0.10	0.12	0.16
M4	■	■	■	0.07	0.08	0.09	0.10	0.12	0.16
S1	◩	◩	◩	0.08	0.09	0.10	0.11	0.14	0.18
S2	◩	◩	◩	0.08	0.09	0.10	0.11	0.14	0.18
S3	◩	◩	◩	0.07	0.08	0.09	0.10	0.12	0.16
S4	◩	◩	◩	0.07	0.08	0.09	0.10	0.12	0.16

804D (XPET..AP, SCET..-UD)

	D9335	D8330	D8345	⌀ 15	⌀ 20	⌀ 25	⌀ 30	⌀ 40	⌀ 58
P1	■	■	■	0.06	0.07	0.08	0.09	0.10	0.14
P2	■	■	■	0.10	0.12	0.14	0.16	0.19	0.25
P3	■	■	■	0.12	0.14	0.16	0.18	0.22	0.30
P4	■	■	■	0.11	0.13	0.15	0.17	0.21	0.28
K1	■	■	■	0.13	0.15	0.18	0.20	0.24	0.32
K2	■	■	■	0.13	0.15	0.18	0.20	0.24	0.32
K3	■	■	■	0.13	0.15	0.18	0.20	0.24	0.32
K4	■	■	■	0.13	0.15	0.18	0.20	0.24	0.32
K5	■	■	■	0.13	0.15	0.18	0.20	0.24	0.32

804D (XPET..AP-SD, SCET..-SD)

f ⇨

	D9335	D8330	D8345	ø 15	ø 20	ø 25	ø 30	ø 40	ø 58
P1	■	■	■	0.07	0.08	0.09	0.10	0.12	0.16
P2	■	■	■	0.10	0.12	0.14	0.16	0.19	0.25
P3	■	■	■	0.12	0.14	0.16	0.18	0.22	0.30
P4	■	■	■	–	–	–	–	–	–
K1	◩	◩	◩	0.07	0.08	0.09	0.10	0.12	0.16
K2	◩	◩	◩	0.10	0.12	0.14	0.16	0.19	0.25
K3	◩	◩	◩	0.11	0.13	0.15	0.17	0.20	0.27
K4	◩	◩	◩	0.12	0.14	0.16	0.18	0.22	0.30
K5	◩	◩	◩	0.14	0.16	0.19	0.21	0.25	0.33
M1	■	■	■	0.11	0.13	0.15	0.17	0.21	0.28
M2	■	■	■	0.10	0.12	0.14	0.16	0.19	0.25
M3	■	■	■	0.06	0.07	0.08	0.09	0.10	0.14
M4	■	■	■	0.06	0.07	0.08	0.09	0.10	0.14
S1	◩	◩	◩	0.07	0.08	0.09	0.10	0.12	0.16
S2	◩	◩	◩	0.07	0.08	0.09	0.10	0.12	0.16
S3	◩	◩	◩	0.06	0.07	0.08	0.09	0.10	0.14
S4	◩	◩	◩	0.06	0.07	0.08	0.09	0.10	0.14

805D (XPET..AP, SCET..-UD)

f ⇨

	D9335	D8330	D8345	ø 15	ø 20	ø 25	ø 30	ø 40	ø 58
P1	■	■	■	0.06	0.07	0.08	0.09	0.10	0.14
P2	■	■	■	0.10	0.12	0.14	0.16	0.19	0.25
P3	■	■	■	0.12	0.14	0.16	0.18	0.22	0.30
P4	■	■	■	0.11	0.13	0.15	0.17	0.21	0.28
K1	■	■	■	0.13	0.15	0.18	0.20	0.24	0.32
K2	■	■	■	0.13	0.15	0.18	0.20	0.24	0.32
K3	■	■	■	0.13	0.15	0.18	0.20	0.24	0.32
K4	■	■	■	0.13	0.15	0.18	0.20	0.24	0.32
K5	■	■	■	0.13	0.15	0.18	0.20	0.24	0.32

805D (XPET..AP-SD, SCET..-SD)

f ⇨

	D9335	D8330	D8345	ø 15	ø 20	ø 25	ø 30	ø 40	ø 58
P1	■	■	■	0.07	0.08	0.09	0.10	0.12	0.16
P2	■	■	■	0.10	0.12	0.14	0.16	0.19	0.25
P3	■	■	■	0.12	0.14	0.16	0.18	0.22	0.30
P4	■	■	■	–	–	–	–	–	–
K1	◩	◩	◩	0.07	0.08	0.09	0.10	0.12	0.16
K2	◩	◩	◩	0.10	0.12	0.14	0.16	0.19	0.25
K3	◩	◩	◩	0.11	0.13	0.15	0.17	0.20	0.27
K4	◩	◩	◩	0.12	0.14	0.16	0.18	0.22	0.30
K5	◩	◩	◩	0.12	0.14	0.16	0.18	0.22	0.30
M1	■	■	■	0.11	0.13	0.15	0.17	0.21	0.28
M2	■	■	■	0.10	0.12	0.14	0.16	0.19	0.25
M3	■	■	■	0.06	0.07	0.08	0.09	0.10	0.14
M4	■	■	■	0.06	0.07	0.08	0.09	0.10	0.14
S1	◩	◩	◩	0.07	0.08	0.09	0.10	0.12	0.16
S2	◩	◩	◩	0.07	0.08	0.09	0.10	0.12	0.16
S3	◩	◩	◩	0.06	0.07	0.08	0.09	0.10	0.14
S4	◩	◩	◩	0.06	0.07	0.08	0.09	0.10	0.14

FORMULA FOR CALCULATION OF CUTTING PARAMETERS

Nomenclature and formula		
Parameter	**Formula**	**Unit**
RPM	$n = \frac{v_c \cdot 1000}{DC \cdot \pi}$	(rev/min)
Cutting speed	$v_c = \frac{\pi \cdot DC \cdot n}{1000}$	(m/min)
Table feed	$v_f = n \cdot f$	(mm/min)
Cross section area of the hole	$A = \frac{\pi \cdot DC^2}{4}$	(mm^2)
Metal removal rate	$Q = \frac{v_f \cdot A}{1000}$	(cm^3/min)
Machining time	$T_c = \frac{L + h}{v_f}$	(min/pcs)

DC	Diameter of drill	(mm)
f	Feed per revolution	(mm/rev)
h	Distance from drill point to workpiece before feeding	(mm)
L	Depth of hole	(mm)

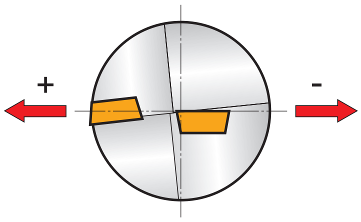
RECOMMENDED TIGHTENING TORQUES FOR SCREWS

	Nm					
US 2245-T07P	0.9	FLAG T07P	M 2.2	5.3	D-T7P	MR-0.8-2.0 vario
US 2205-T07P	0.9	FLAG T07P	M2.2	5.4	D-T7P	MR-0.8-2.0 vario
US 2506-T07P	1.2	FLAG T07P	M 2.5	6	D-T7P	MR-0.8-2.0 vario
US 2507-T08P	1.2	FLAG T08P	M 2.5	7	D-T8P	MR-0.8-2.0 vario
US 3007-T08P	2.0	FLAG T08P	M 3	7	D-T8P	MR-1.0-5.0 vario
US 3007-T09P	2.0	FLAG T09P	M 3	7.4	D-T9P	MR-1.0-5.0 vario
US 3009-T09P	2.0	FLAG T09P	M 3	8.7	D-T9P	MR-1.0-5.0 vario
US 3508-T15P	3.0	FLAG T15P	M 3.5	8.3	D-T15P	MR-1.0-5.0 vario
US 3510-T15P	3.0	FLAG T15P	M 3.5	10.6	D-T15P	MR-1.0-5.0 vario
US 4011-T15P	3.5	FLAG T15P	M 4	10.7	D-T15P	MR-1.0-5.0 vario
US 5012-T15P	5.0	FLAG T15P	M 5	12.2	D-T15P	MR-1.0-5.0 vario

Radial adjustment

Hole diameter adjustment and set-up recommendation
Radial adjustment is possible with indexable drills to achieve a smaller or larger hole diameter than the actual drill.
Radial adjustment values are available in the main drill data tables.

Rotating tool
For drilling holes with accuracy IT10 and higher, adjustable holders are recommended when using 802D, 803D, 804D and 805D drills.
Stationary tool
When mounting the drill make sure the drill centre line and workpiece centre are aligned. To achieve a larger hole diameter displace the drill so that the peripheral insert moves in a + away from the workpiece centre line (see diagram below).



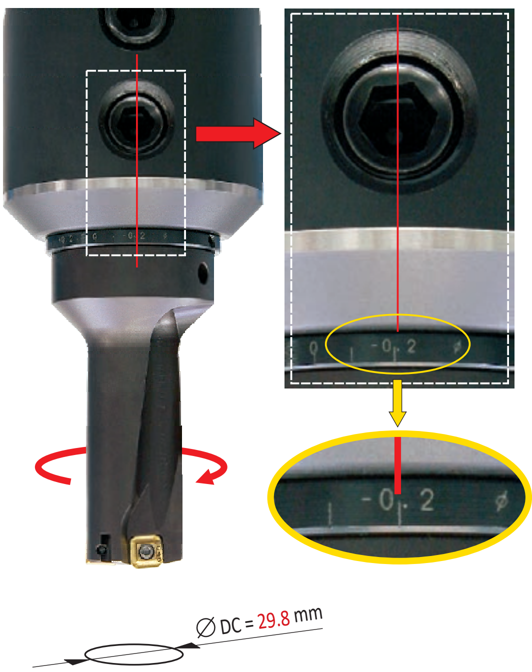
Tool life
Inserts should be changed when flank wear measures 0.2 – 0.4 mm at the largest point.
Cutting data recommendations in this catalogue are aimed at achieving tool life of 7 metres drilling depth on the peripheral insert. (20 – 30 mins contact).

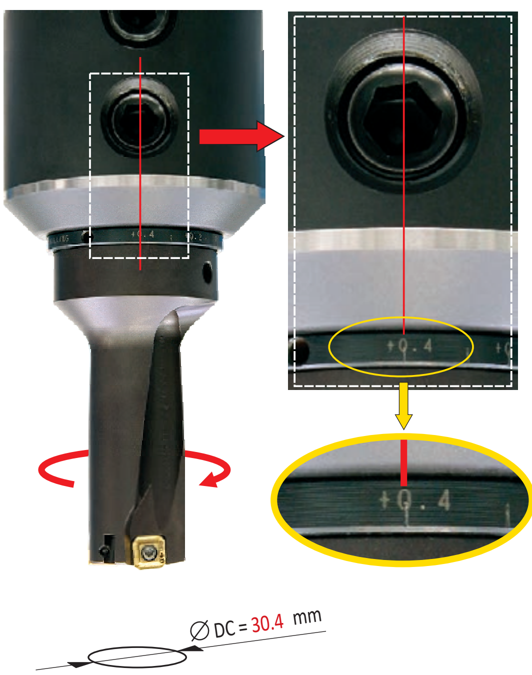
EP ADJUSTABLE SLEEVE

Shank diameter	Drill diameter	Range
25	15 – 24	+0.4 – -0.2
32	24.5 – 40	+0.4 – -0.2

For Milling Machines

Diameter adjustment range

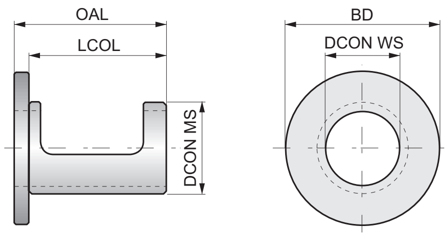




PRAMET

EP - Indexable Insert Drill Adjustment Sleeve

Sleeve to adjust indexable insert drill diameter. Can be used in Ø32 or Ø40 mm weldon tool holders. The outside drill diameter is adjusted by rotating the sleeve.



Diameter adjustment range is 0.4 – -0.2; center height adjustment range is 0.2 – -0.15.

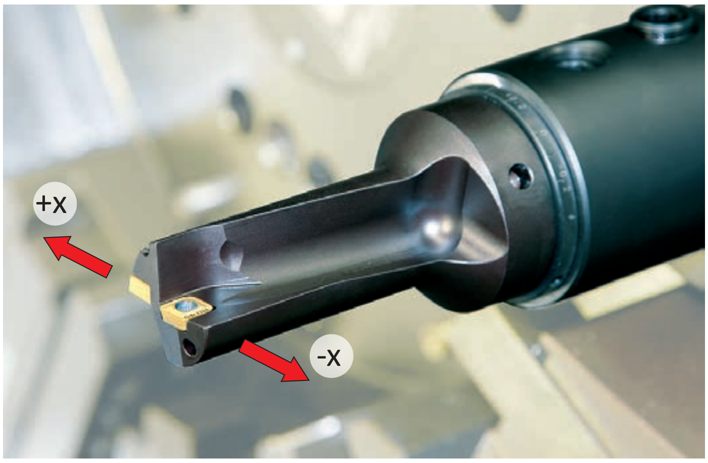
Product	DCON WS (mm)	DCON MS (mm)	BD (mm)	OAL (mm)	LCOL (mm)	kg
EP253253	25.00	32.00	53.00	53.0	48	0.15
EP324058	32.00	40.00	58.00	58.0	53	0.20

EP TECHNICAL INFO – INDEXABLE DRILLS – ADJUSTABLE SLEEVE

Shank diameter	Drill diameter	Range
25	15 – 24	+0.2 – -0.15
32	24.5 – 40	+0.2 – -0.15

Centre height adjustment
– for lathe operation

Centre height adjustment range



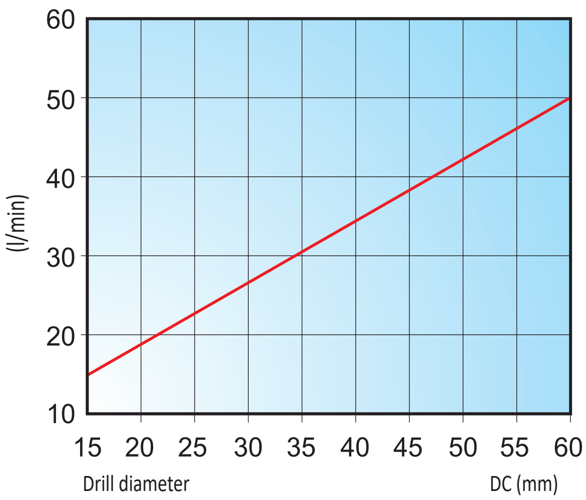
MACHINING DATA FOR INDEXABLE DRILLS

Recommended pressure of supplied cutting fluid

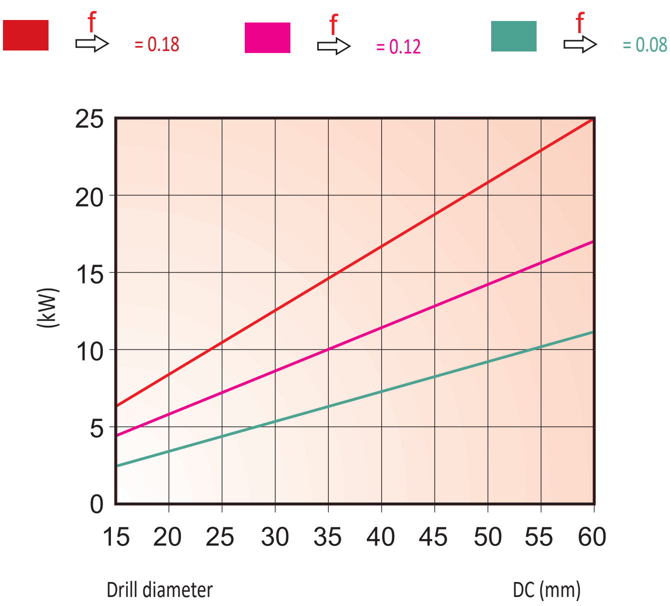
Drill diameter DC (mm)	Pressure of cutting fluid	
	Drill length	
	2.0 – 2.5 DC	3.0 – 5.0 DC
15 – 25	6 bar	12 bar
26 – 40	4.5 bar	9 bar
> 40	3 bar	6 bar

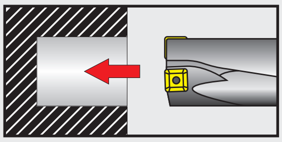
Coolant volume requirement

DRY DRILLING
Pressurised air through the drill is recommended when drilling without coolant in cast iron and steel

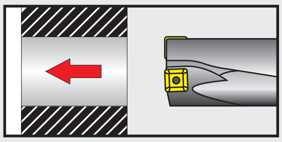


Net power consumption

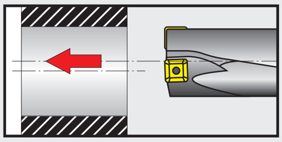




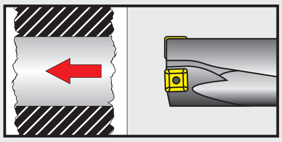
BLIND HOLE DRILLING
For drilling holes deeper than 1×DC internal cooling is necessary.



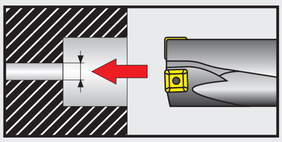
THROUGH HOLE DRILLING
A disc can be produced when the indexable drill exits the material. This disc can be ejected at high speed when the workpiece is rotating. It is essential that the machine is adequately guarded to ensure operator safety



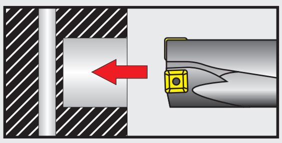
OFF-CENTRE DRILLING
Decrease the feed to lower recommended values for particular inserts. See inserts description pages for indexable drills.
Do not exceed radial adjustment values.



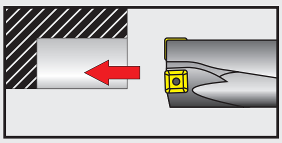
STARTING ON UNEVEN AND CAST SURFACES
Decrease the feed by 50% on entrance for indexable drills until both inserts are engaged.



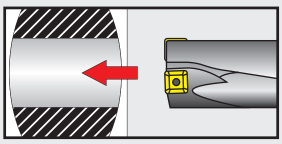
BORING AND DRILLING INTO PILOT HOLES
If a pre-drilled hole is larger than 1/4 drill diameter, decrease the feed.



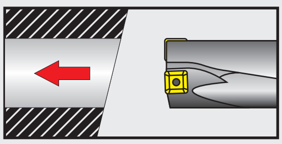
DRILLING CROSS HOLES
Decrease the feed by 50% when drilling across an existing hole.
The diameter of existing hole should not be larger than 0.25 x DC.



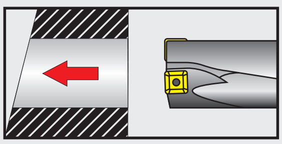
INTERRUPTED CUT AND PLUNGING
Decrease the feed to lower recommended feed values for particular insert. See inserts description site for indexable drills.



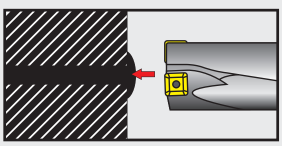
DRILLING ON CURVED SURFACE
Drilling on the centre line can be done with reduced feed rate down to 50 % during entrance and exit.



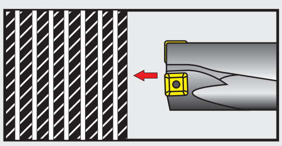
DRILLING ON ANGLED SURFACES
Decrease the feed by 50% on entrance for indexable drills until both inserts are engaged if the angle of entry is more than 5°.



EXIT ON ANGLED SURFACE
Decrease the feed by 50% on exit if angle of exit is more than 5°.



STARTING ON A WELDED SEAM
Facing is recommended before drilling.
Decrease the feed by 50 % during drilling of the welded material.



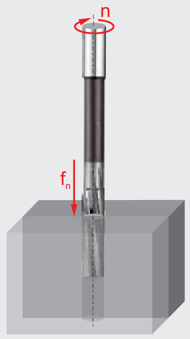
DRILLING OF STACKED MATERIALS
Avoid spaces larger than 0.2 mm between layers. The component must be securely fixed. If necessary reduce the feed.

TROUBLESHOOTING FOR INDEXABLE DRILLS

LOW PERFORMANCE OF DRIVING MOTOR (LOW SPINDLE POWER)	a) reduce cutting speed = reduction of spindle RPM b) reduce feed rate
EXCESSIVE WEAR OF PERIPHERAL INSERT	a) reduce cutting speed = reduction of spindle RPM b) choose a more wear resistant grade c) increase coolant volume and pressure
CHIPPING OF PERIPHERAL INSERT	a) reduce feed rate until peripheral insert is fully engaged b) choose a tougher insert grade c) reduce cutting speed
CHIPPING OF CENTRE INSERT	a) reduce feed rate during entry b) check the drill and workpiece clamping
CONTINUOUS, BADLY FORMED CHIP	a) adjust feed rate b) increase cutting speed and simultaneously reduce feed rate
SWARF CONGESTION IN THE FLUTES	a) increase coolant volume and pressure b) reduce cutting speed c) adjust feed rate

REAMERS – TECHNICAL INFO

REAMERS FEED RATE CHART

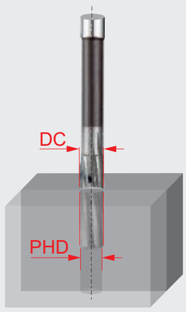


Feed per revolution (f_n in mm/rev)
Depending on the working conditions it might be necessary to adjust these values ± 15 %.

How to use this table to find the feed per revolution (f_n):

1. Find your Alpha Code on the product page (example: 21C, "C" is the Alpha Code).
2. Find the closest diameter for your cutting application in the top row of the table.
3. Find your Alpha Code in the left column of the table.
4. The intersection (cell) of the Diameter and Alpha Code is the feed per revolution (f_n).

		ø DC (mm)																		
		1.00	1.50	2.00	3.00	4.00	5.00	6.00	7.00	8.00	10.00	12.00	15.00	16.00	20.00	25.00	30.00	40.00	50.00	80.00
Feed rates	A	0.030	0.045	0.055	0.078	0.090	0.100	0.125	0.137	0.150	0.170	0.185	0.210	0.220	0.250	0.280	0.320	0.390	0.440	0.500
	B	0.035	0.055	0.072	0.110	0.130	0.150	0.165	0.172	0.180	0.210	0.240	0.270	0.280	0.310	0.360	0.400	0.500	0.550	0.600
	C	0.040	0.065	0.085	0.135	0.160	0.185	0.200	0.210	0.220	0.260	0.285	0.325	0.335	0.390	0.440	0.480	0.600	0.680	0.750
	D	0.050	0.080	0.110	0.160	0.180	0.200	0.235	0.253	0.270	0.320	0.360	0.400	0.410	0.470	0.540	0.600	0.730	0.850	0.950
	E	0.065	0.100	0.140	0.180	0.215	0.250	0.300	0.325	0.350	0.390	0.430	0.485	0.500	0.530	0.640	0.750	0.910	1.100	1.200
	F	0.090	0.140	0.180	0.260	0.305	0.350	0.395	0.417	0.440	0.500	0.550	0.610	0.630	0.700	0.800	0.930	1.200	1.500	1.650



Machining allowance when using a **machine reamer** (MA in mm)
Premachined hole diameter
PHD = DC – MA.

How to use this table to get to the right premachined hole diameter (PHD):

1. Find the diameter range for your cutting application in the top row of the table.
2. Find your ISO Group Code in the left column of the table (example: For Stainless Steel the ISO Group Code is "M")
3. The intersection (cell) of the Diameter Range and ISO Group Code is the Machining Allowance (MA)
4. Subtract the Machining Allowance from the reaming diameter to get to the premachined hole diameter (PHD).

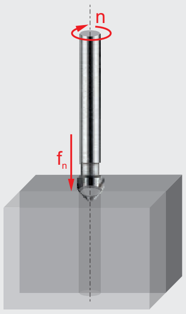
(example: for a 6mm hole in steel (P) the PHD is 5.85mm)

		ø DC (mm)											
		1.00	5.00	5.00	8.00	8.00	12.00	12.00	16.00	16.00	30.00	30.00	80.00
ISO group	P	0.10		0.15		0.20		0.20		0.30		0.30	
	M	0.08		0.10		0.10		0.20		0.20		0.30	
	K	0.10		0.15		0.20		0.20		0.30		0.30	
	N	0.10		0.15		0.20		0.20		0.30		0.30	
	S	0.05		0.10		0.10		0.15		0.20		0.20	
	H	0.05		0.05		0.10		0.10		0.15		0.20	

Be cautious with the machining tolerances of drills, the tool diameter is not the same as the hole diameter produced!

Note: The recommended allowance when using a hand reamer is 0.05 to 0.10 mm.

COUNTERSINKS FEED RATE CHART



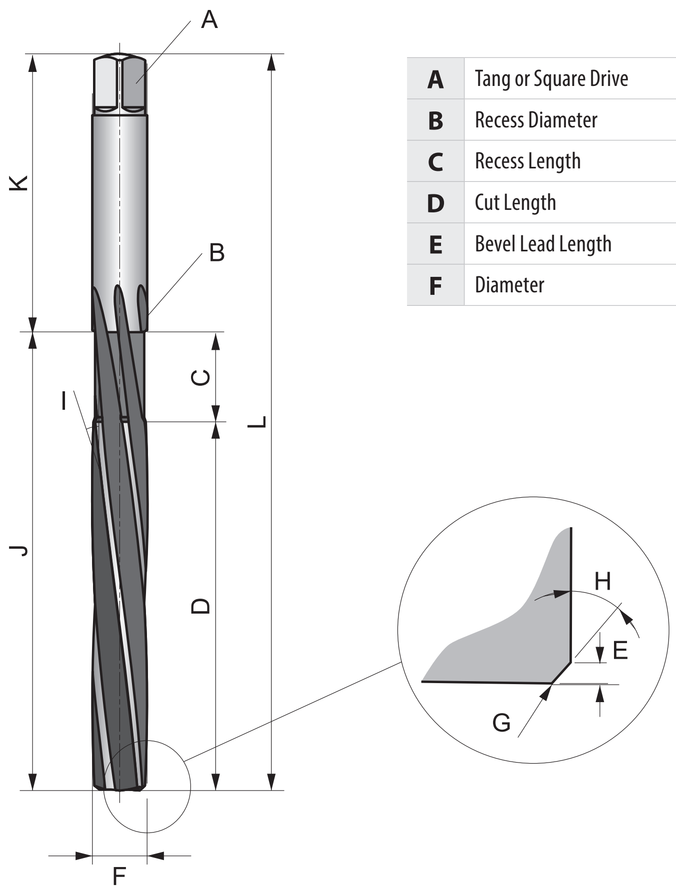
Feed per revolution (f_n in mm/rev)
Depending on the working conditions it might be necessary to adjust these values ± 15 %.

How to use this table to find the feed per revolution (f_n):

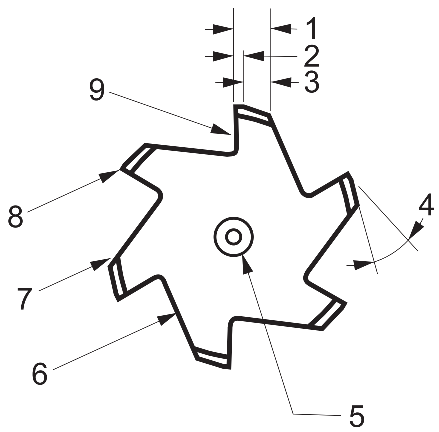
1. Find your Alpha Code on the product page (example: 23E, "E" is the Alpha Code).
2. Find the closest diameter for your cutting application in the top row of the table.
3. Find your Alpha Code in the left column of the table.
4. The intersection (cell) of the Diameter and Alpha Code is the feed per revolution (fn).

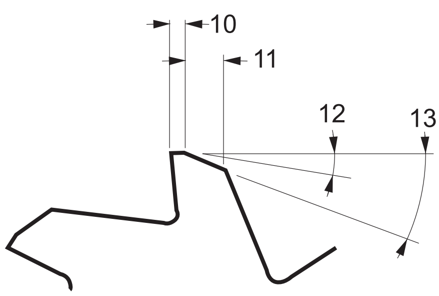
		ø DC (mm)									
		6.00	**8.00**	**10.00**	**16.00**	**20.00**	**25.00**	**32.00**	**40.00**	**60.00**	**80.00**
Feed rates	**A**	0.030	0.040	0.050	0.060	0.080	0.090	0.100	0.120	0.140	0.160
	B	0.040	0.050	0.060	0.080	0.100	0.120	0.140	0.160	0.180	0.200
	C	0.050	0.060	0.080	0.100	0.120	0.140	0.160	0.180	0.200	0.220
	D	0.060	0.080	0.100	0.120	0.150	0.180	0.200	0.220	0.250	0.280
	E	0.080	0.100	0.120	0.150	0.180	0.200	0.250	0.270	0.300	0.320
	F	0.090	0.110	0.130	0.160	0.190	0.210	0.260	0.290	0.330	0.360
	G	0.100	0.120	0.150	0.180	0.200	0.220	0.280	0.320	0.360	0.400
	H	0.120	0.150	0.180	0.200	0.220	0.250	0.300	0.350	0.400	0.450

Reamer Definitions / Nomenclature



A	Tang or Square Drive		**G**	Bevel Lead
B	Recess Diameter		**H**	Bevel Lead Angle
C	Recess Length		**I**	Helix Angle
D	Cut Length		**J**	Body Length
E	Bevel Lead Length		**K**	Shank Length
F	Diameter		**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

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 the starting hole diameter. If insufficient stock is left in the hole before reaming, 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 below).

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 for straight shank reamers must be of good quality and in good working condition. If the reamer slips in the chuck and the feed is automatic, breakage of the reamer may occur.
4. Keep tool overhang from machine spindle to a minimum.
5. 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.
6. Do not allow the flutes of a reamer to become blocked with chips. Retract if necessary to empty the flutes, this can help to prevent poor hole quality and breakage of the tool.
7. Before the reamer is reground, check concentricity between centres. In most instances only the bevel lead will need regrinding.
8. 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.

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:

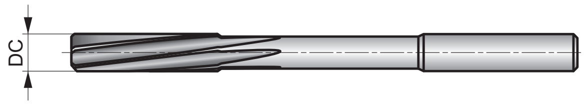
Size of reamed hole (mm)	When pre-drilled	When pre-core-drilled
Below 4	0.1	0.1
Over 4 to 11	0.2	0.15
Over 11 to 39	0.3	0.2
Over 39 to 50	0.4	0.3

Size of reamed hole (inches)	When pre-drilled	When pre-core-drilled
Below 3/16"	0.004"	0.004"
3/16" to ½"	0.008"	0.006"
½" to 1.½"	0.010"	0.008"
1.½" to 2"	0.016"	0.010"

Hand/Machine reaming

Although both hand and machine reamers offer the same capability regarding finished hole size, the use of each must be considered according to the 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 while a hand reamer cuts on the bevel lead as well as the taper lead.

Tolerance limits



1. On the cutting diameter of standard reamers
   The diameter (DC) 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

Reamer tolerance			
Diameter (mm)		**Tolerance Limit** (mm)	
Over	Up to and including	High +	Low +
18	30	0.017	0.009
30	50	0.021	0.012
50	80	0.025	0.014

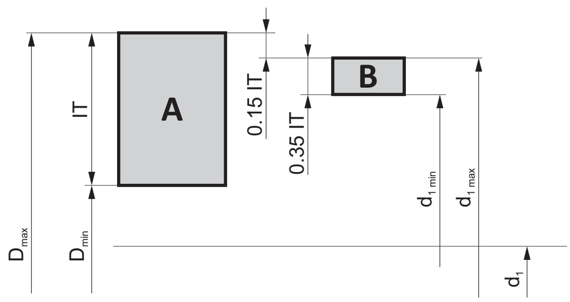
2. H7 hole tolerance
   The most common tolerance on a finished hole is H7 (see table below). For any other tolerance the figure and table below (in Note 3) can be used to calculate the reamers tolerance location and width

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

Hole tolerance			
Diameter (mm)		**Tolerance Limit** (mm)	
Over	Up to and including	High +	Low +
18	30	0.021	0
30	50	0.025	0
50	80	0.030	0

3. Other hole tolerances 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.

Diameter tolerance width (µm)								
Tolerance width (microns)	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
IT5	4	5	6	8	9	11	13	15
IT6	6	8	9	11	13	16	19	22
IT7	10	12	15	18	21	25	30	35
IT8	14	18	22	27	33	39	46	54
IT9	25	30	36	43	52	62	74	87
IT10	40	48	58	70	84	100	120	140
IT11	60	75	90	110	130	160	190	220
IT12	100	120	150	180	210	250	300	350



A = Hole tolerance
B = Reamer tolerance
IT = Tolerance width
D_{max} = Max. diameter of hole
D_{min} = Min. diameter of hole
d_1 = Nominal diameter
$d_{1\,max}$ = Max. diameter of reamer
$d_{1\,min}$ = Min. diameter of reamer

e.g. 10 mm hole with tolerance D8, Max dia = 10.062, Min dia = 10.040, Hole tolerance (IT8) = 0.022

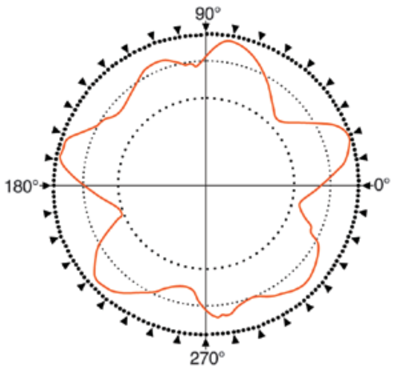
Maximum limit: 0.15 × hole tolerance (IT8) = 0.0033, rounded up = 0.004
Minimum limit: 0.35 × hole tolerance (IT8) = 0.0077, rounded up = 0.008
Maximum limit for reamer = 10.062 – 0.004 = 10.058
Minimum limit for reamer = 10.058 – 0.008 = 10.050

Applications – Reamer Selection

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 WMG and feed charts (see 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 conventional unequal spacing.

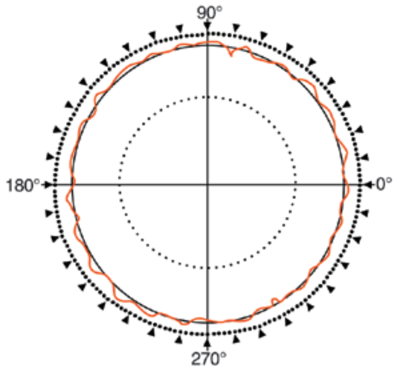
Carbide Reamers – Comparison spacing / EU spacing

Unequal spacing
Roundness error up to 10 µm



Results of roundness

Extremely unequal spacing
Roundness error up to 1 – 2 µm



Results of roundness

Trouble shooting when reaming

Problem	Cause	Remedy
Broken or twisted tangs	Incorrect fit between shank and socket	Ensure the shank and socket are clean and free from damage
Rapid tool wear	Insufficient stock to remove	Increase the amount of stock to be removed (smaller hole)
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
Undersize hole	Insufficient stock to remove	Increase the amount of stock to be removed (smaller hole)
	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 or replace tool
	Too low feed or cutting speed	Adjust cutting conditions in accordance with the Catalogue
	Pre-drilled hole is too small	Decrease the amount of stock to be removed (larger hole)
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
Bad hole finish	Excessive stock to remove	Decrease the amount of stock to be removed (larger hole)
	Worn out tool	Regind to correct specification
	Undersize cutting rake angle	Regind to correct specification
	Too diluted emulsion or cutting oil	Increase % concentration
	Feed and/or speed too low	Adjust cutting conditions in accordance with Catalogue
	Cutting speed too high	Adjust cutting conditions in accordance with Catalogue
The tool clamps and breaks	Worn out tool	Regind 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 (larger hole)
	Heterogeneous material with hard inclusions	Use solid carbide reamer

	Grade	Hardness (HV10)	C %	W %	Mo %	Cr %	V %	Co %	Tool Material
HSS	M2	810 – 850	0.9	6.4	5.0	4.2	1.8	–	HSS
HSS-E	M35	830 – 870	0.93	6.4	5.0	4.2	1.8	4.8	HSCO
	M42	870 – 960	1.08	1.5	9.4	3.9	1.2	8.0	

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.

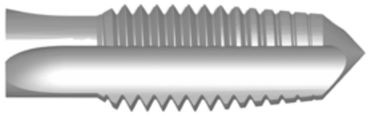
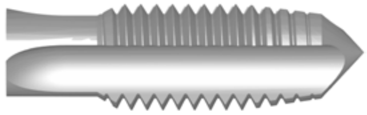
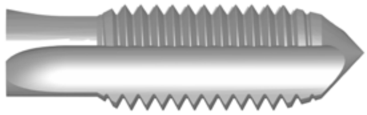
Surface treatment / Coating properties examples

Surface Treaments	Colour	Coating material	Hardness (HV)	Thickness (µm)	Coating structure	Frict. coeff. against steel	Max. appl. temp. (°C)
TiN	Gold	TiN	2300	1-4	Mono-layer	0.4	600
TiAlN	Black grey	TiAlN	3300	3	Nano structured	0.3-0.35	900

THREADING – TECHNICAL INFO

THEARDING – TECHNICAL INFO – TAP NO1 – NO9

Hand taps (ISO standard) with different chamfer lengths each producing a full thread profile.

NO1 =	 Taper lead	**A** 6-8	
NO2 =	 Plug lead	**B** 4-6	
NO3 =	 Bottoming lead	**C** 2-3	

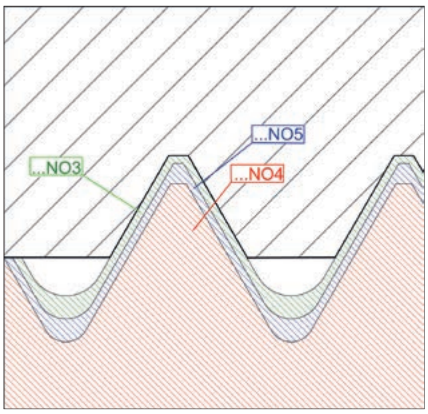
ISO	**NO6 =**	NO1 + NO2 + NO3
	NO7 =	NO2 + NO3 *

ANSI	**NO6 =**	NO1 (taper) + NO2 (plug) + NO3 (bottoming)

*** E550, E710** **NO7 =** NO3 (truncated) + NO3

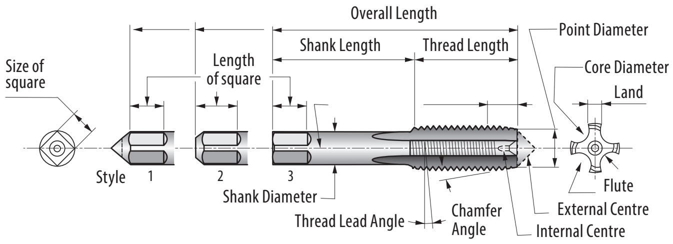
Serial taps (DIN standard) with each sequencing tap cutting a part of the profile, the NO3 tap is needed to complete a full thread profile.

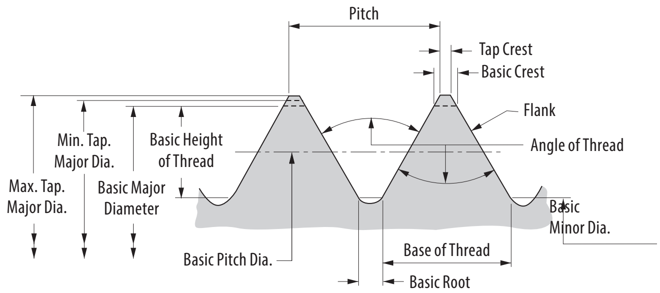
NO4 =	 Starter tap	**A** 6-8
NO5 =	 Seconding tap	**B** 3.5-5
NO3 =	 Finishing (bottoming lead) tap	**C** 2-3

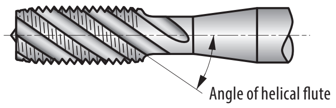


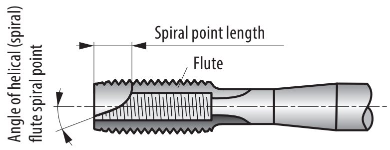
DIN ISO	**NO8 =**	NO3 + NO4 + NO5
	NO9 =	NO3 + NO5

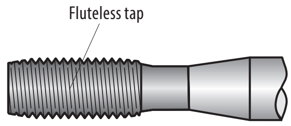
THREADING – GENERAL TECHNICAL INFORMATION

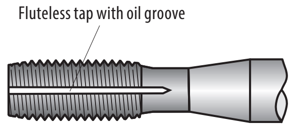












Allowance: The minimum clearance or maximum interference which is intended between mating parts.

Angle of Thread: The angle included between the flanks of a thread measured in an axial plane.

Back Taper: A slight taper on the threaded portion of the tap making the pitch diameter near the shank smaller than that at the chamfer.

Basic: The theoretical or nominal standard size from which all variations are made.

Chamfer: The tapered and relieved cutting teeth at the front end of the threaded section. Common types of chamfer are taper, 8 to 10 pitches long, plug, 3 to 5 pitches and bottoming, 1 to 2 pitches.

Crest: The top surface joining the two sides or flanks of a thread.

Cutting Face: The leading side of the land.

Flute: The longitudinal channels formed on a tap to create cutting edges on the thread profile.

Heel: The following side of the land.

Height of Thread: In profile, distance between crest and bottom section of thread measured normal to the axis.

Hook Face: A concave cutting face of the land. This may be varied for different materials and conditions.

Interrupted Thread: Alternate teeth are removed in the thread helix on a tap; usually restricted to those having an odd number of flutes.

Land: One of the threaded sections between the flutes of a tap.

Lead of Thread: The distance a screw thread advances axially in one turn.

Major Diameter: The largest diameter of the screw or nut on a straight screw thread.

Minor Diameter: The smallest diameter of the screw or nut on a straight screw thread.

Neck: The reduced diameter, on some taps, between the threaded portion and the shank.

Pitch: The distance from a point on one thread to a corresponding point on the next thread, measured parallel to the axis.

Pitch Diameter: On a straight screw thread, the diameter of an imaginary cylinder where the width of the thread and the width of the space between threads is equal.

Point Diameter: The diameter at the leading end of the chamfered portion.

Radial: The straight face of a land, the plane of which passes through the axis of the tap.

Rake: The angle of the cutting face of the land in relation to an axial plane intersecting the cutting face at the major diameter.

Relief: The removal of metal behind the cutting edge to provide clearance between the part being threaded and a portion of the threaded land. Also, see back taper.

Chamfer relief: The gradual decrease in land height from cutting edge to heel on the chamfered portion of the tap land to provide radial clearance for the cutting edge.

Concentric relief: Radial relief in the thread form starting at the back of a concentric margin.

Eccentric thread relief: Radial relief in the thread form starting at the cutting edge and continuing to the heel.

Root: The bottom surface joining the flanks of two adjacent threads.

Side or flank of thread: The surface of the thread which connects the crest with the root.

Shank: The portion of the tap by which it is held and driven.

Spiral Point: An oblique cutting edge ground into the lands to provide a shear cutting action on the first few threads.

Square: The squared end of the tap shank.

Thread: The helical formed tooth of the tap which produces the thread in a tapped hole.

Thread Lead Angle: The angle made by the helix of the thread at the pitch diameter, with a plane perpendicular to the axis.

Threads Per Inch: The number of threads in one inch of length.

THREAD: **Single:** A thread in which lead is equal to pitch.
Double: A thread in which lead is equal to twice the pitch.
Triple: A thread in which lead is equal to triple the pitch.

General hints on tapping

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

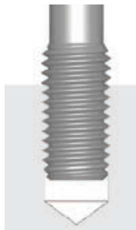
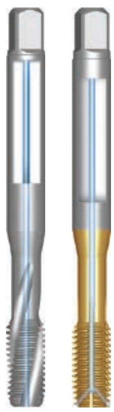
1. Select the correct design of tap for the component material and type of hole, i.e. through or blind, from the Materials Classification chart.
2. Ensure the component is securely clamped – lateral movement may cause tap breakage or poor quality threads.
3. Select the correct size of drill from the relevant catalogue page. Always ensure that work hardening of the component material is kept to a minimum.
4. Select the correct cutting speed as shown on the catalogue product page.
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'.

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

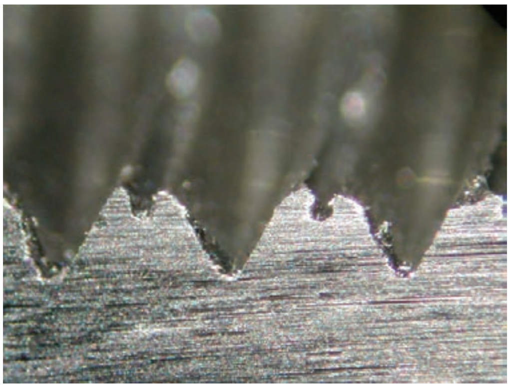
Tap Geometries & Applications

Description	Chips
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.	
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 minimize the torque generated.	
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 chips 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.	
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.	

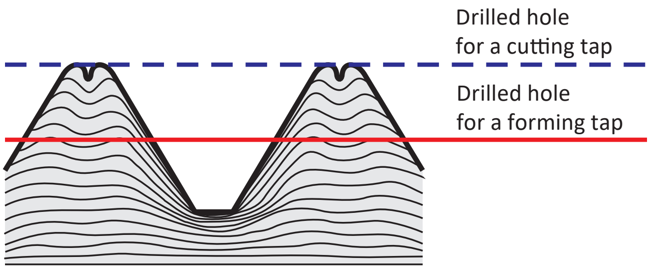
Description	Chips
Taps with flutes only on the chamfer lead The cutting part of the tap is formed by gun nosing in the same manner as for a spiral point tap, the function being to drive the chips forward ahead of the cutting edges. This design is extremely rigid which facilitates good machining results. However, the short length of the gun nosing limits its application to a depth of hole less than about 1.5 × *TDZ*.	
Taps with spiral flutes Taps with spiral flutes are intended primarily for threading in blind holes. The helical flute transports the chips back away from the cutting edges and out of the hole, thus avoiding packing of chips in the flutes or at the bottom of the hole. In this way, danger of breaking the tap or damaging the thread is minimised.	
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 chips are 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.	
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.	

Flow Of Material When Forming A Thread

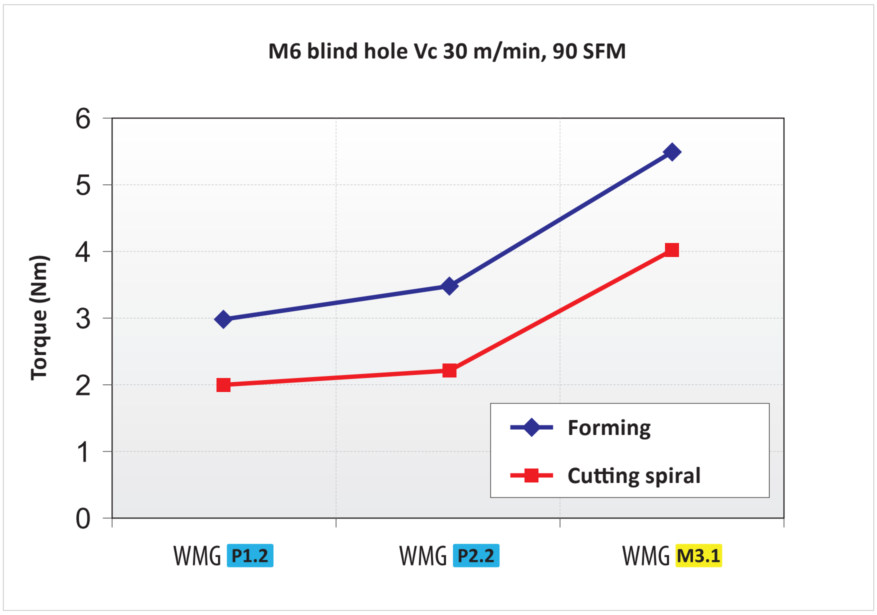
The tapping hole size depends upon the material being drilled, the cutting conditions selected and the condition of the equipment being used. If material is pushed up at the thread entry by the tap and/or the life of the tap is too short, select a slightly larger drill diameter. If on the other hand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

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 in technical handbook.
	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 to prevent chip blockage inside the hole. See lubricant section in technical handbook.
	Tap drill hole too small.	Increase drill diameter to the maximum value. Check tapping size drill.
	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 in technical handbook.
	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 in technical handbook.
	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. Check tapping size drill.

Trouble Shooting When Tapping

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 in technical handbook.
	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 in technical handbook.
	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 in technical handbook.
	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 in technical handbook.
	Surface treatment not suitable.	Choose a tap with the recommended surface treatment.
	Tap speed too low.	Follow recommendation in Catalogue/Product Selector.

BAR PEELING – TECHNICAL INFO

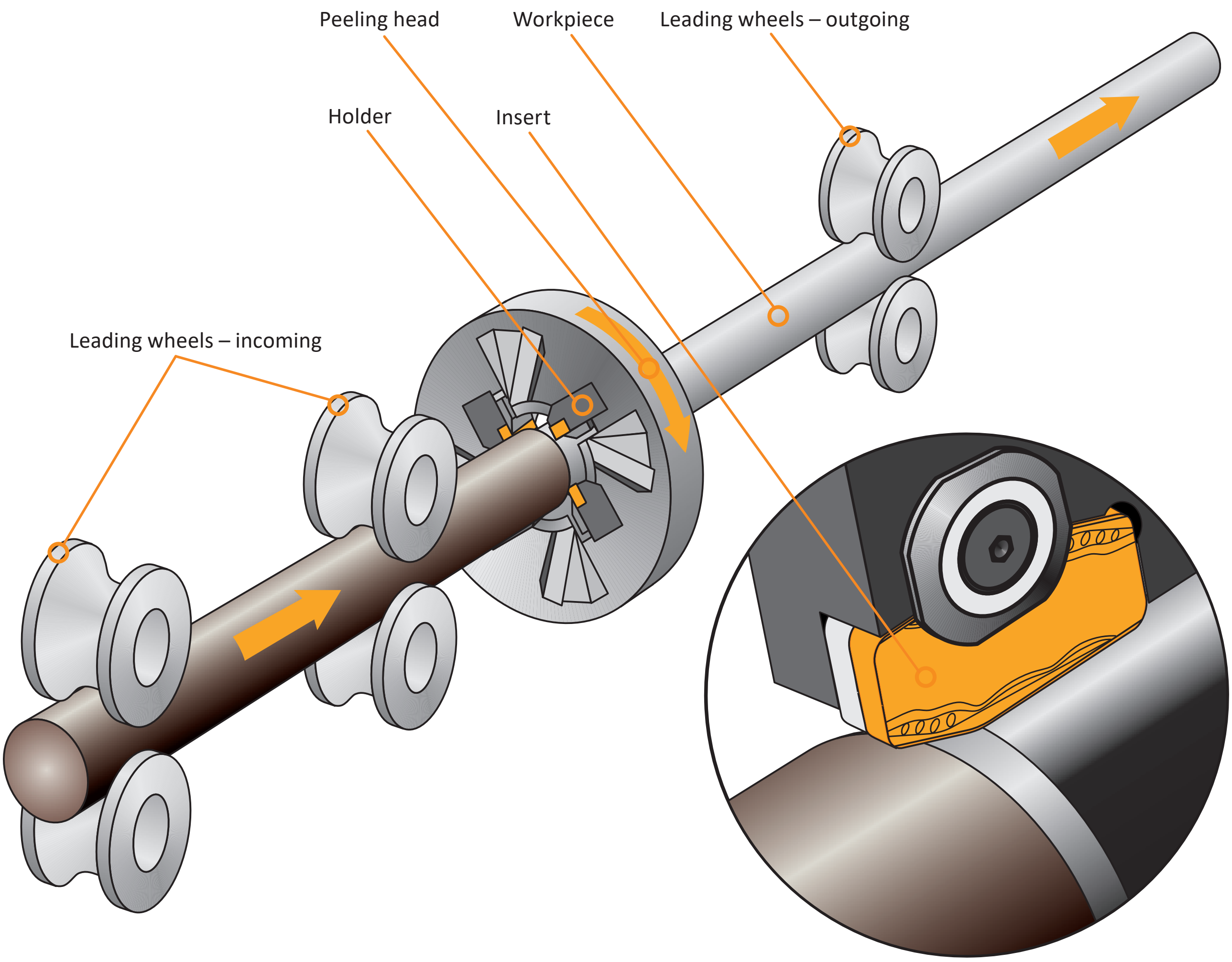
BAR PEELING

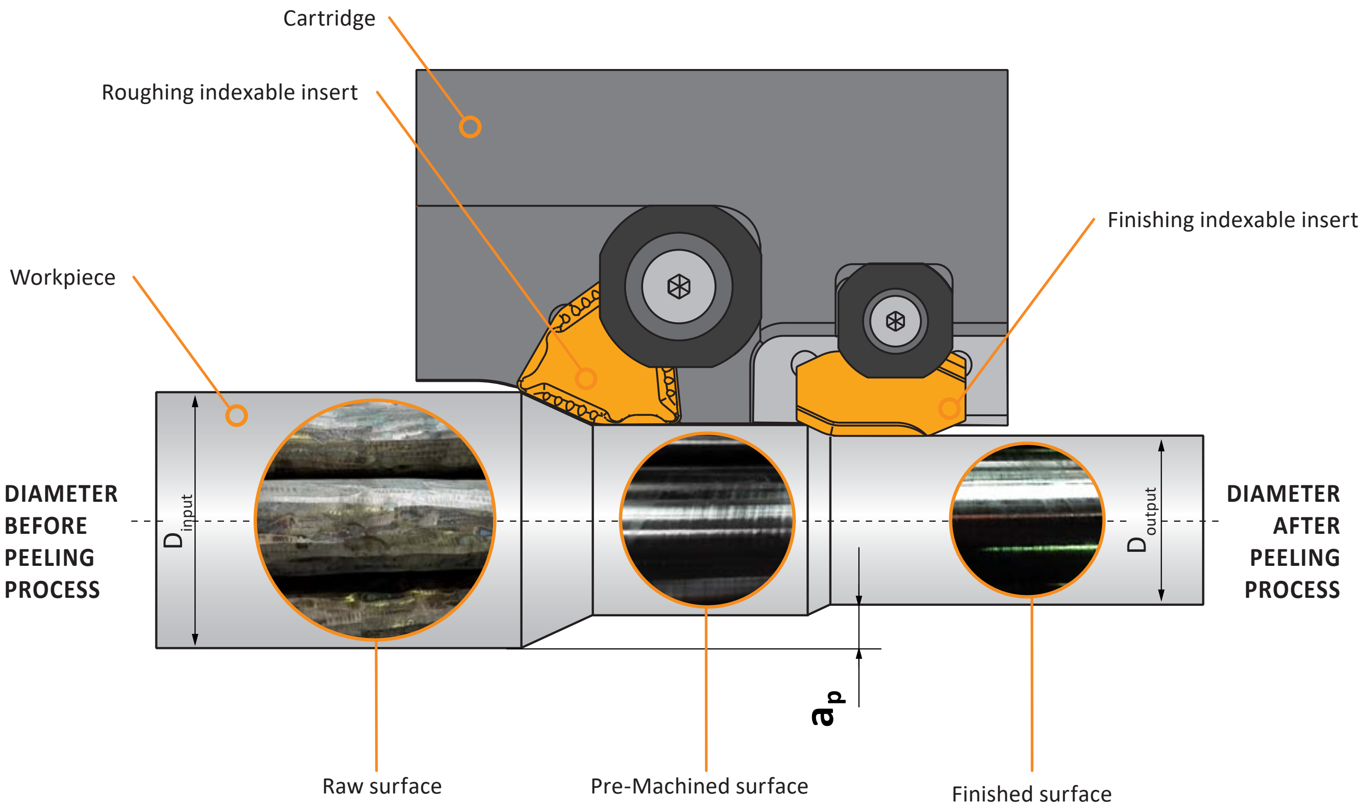
The outstanding feature of this specific operation is relatively high feed rates and small depth of cut applied to round bars and thick walled tubes. Peeling operations remove surface layers of oxides, rolled contaminants and cracks caused by hot forging or rolling.

Peeled materials are mostly carbon steel, alloy steel for heat treating, tool steel, stainless steel and also heat–resistant alloys based on Ni, Co, Fe and Ti.

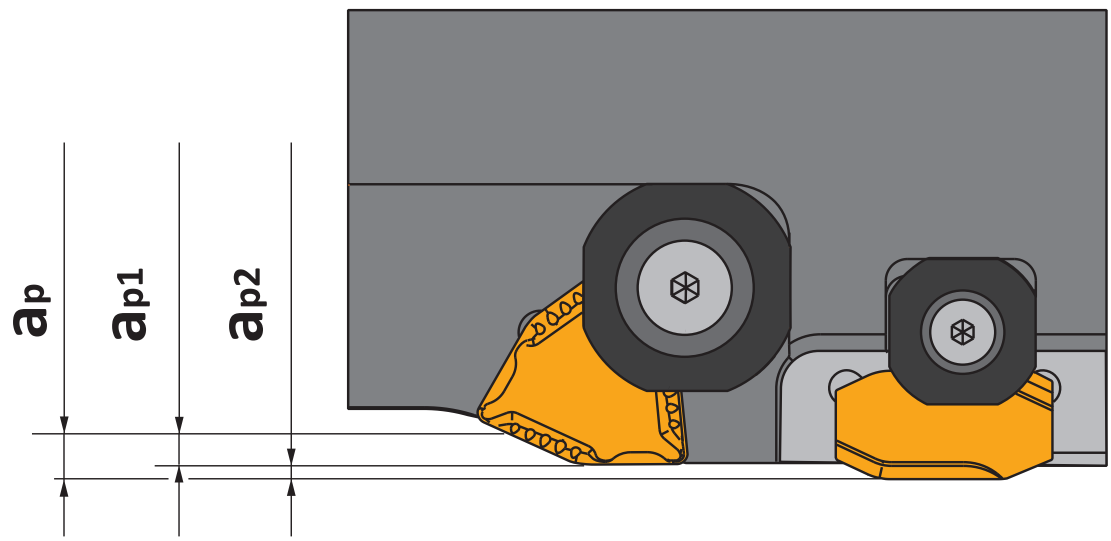
The advantages of peeling technology in comparison with turning are:

- Machining at higher feed rates
- Higher productivity
- Less inserts consumption
- Excellent roughness quality
- High dimensional accuracy



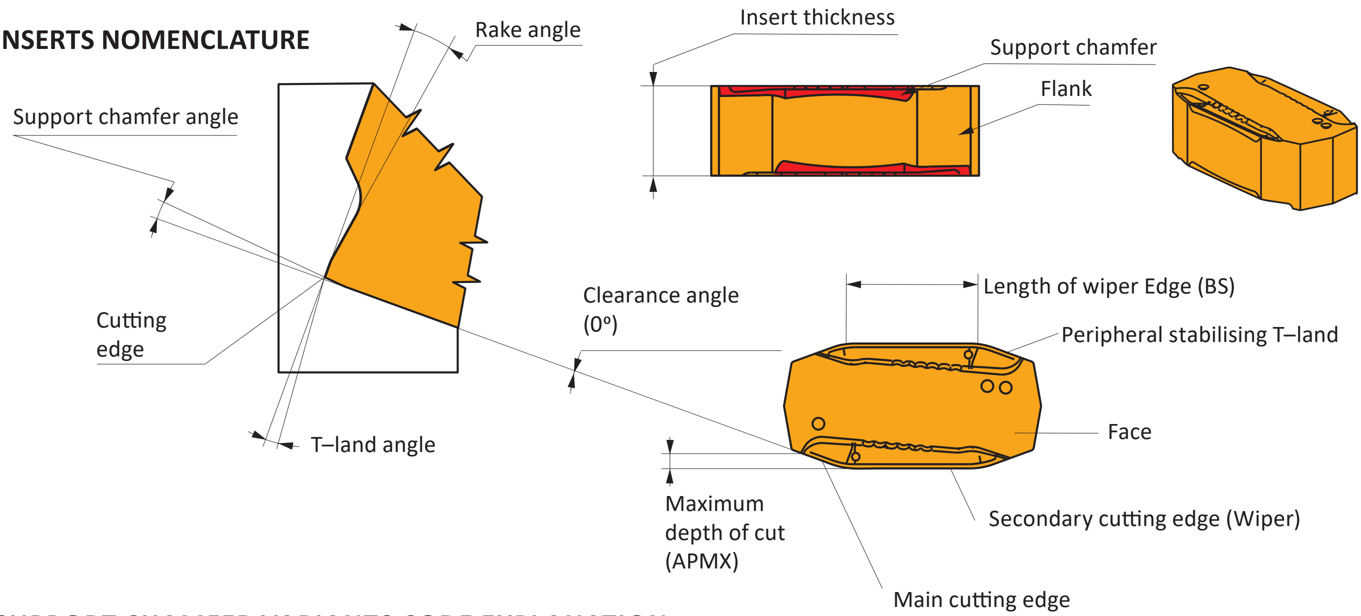


The total depth of cut a_p is the difference between the input diameter and output diameter of the workpiece divided by two.

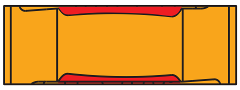
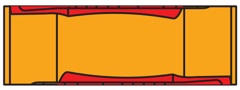
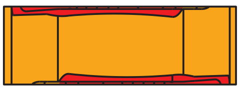


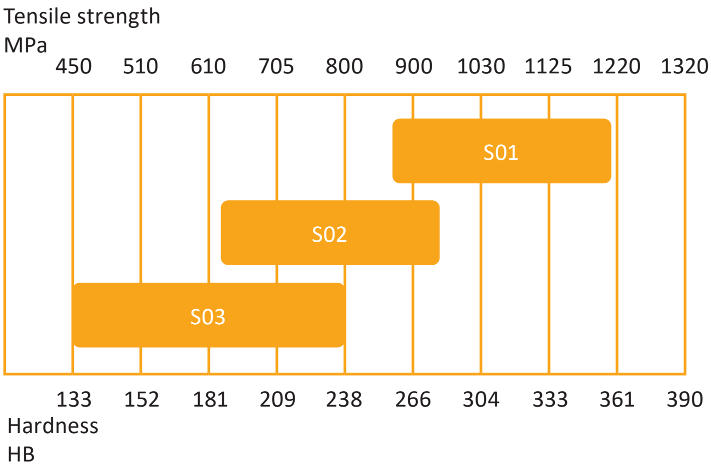
Depth of cut ap in cassettes with more than one insert is divided into the partial depths of cut for each insert (a_{p1}; a_{p2}). Those values should be taken into consideration during detailed analyses of the cutting conditions of the roughing and finishing inserts.

INSERTS NOMENCLATURE

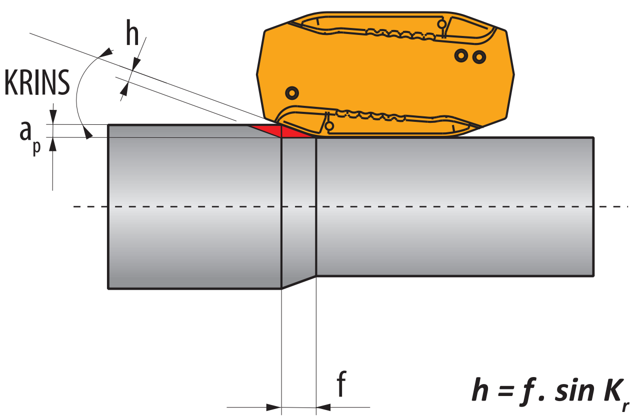


SUPPORT CHAMFER VARIANTS CODE EXPLANATION

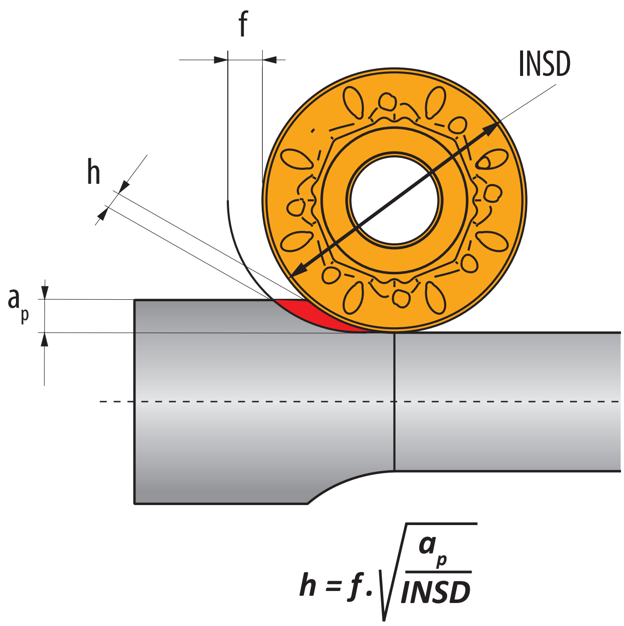
Support chamfer variant	Sketch	Main cutting edge – angle	Wiper cutting edge – angle	Workpiece material properties
S01		0°	5°	850 — 1200 MPa 123 — 174 kPsi 250 — 360 HB Tempered
S02		3°	5°	600 — 950 MPa 87 — 137 kPsi 180 — 260 HB Basic hardness
S03		5°	5°	450 — 800 MPa 65 — 116 kPsi 150 — 230 HB Annealed



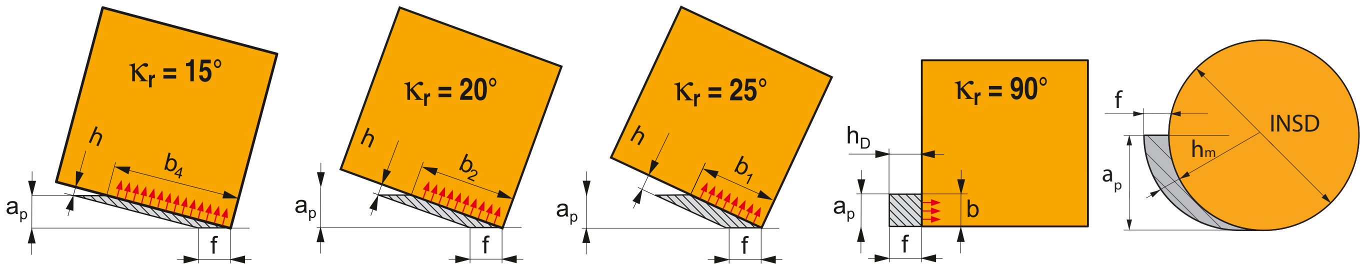
The setting angle of the main cutting edge KRINS, has the most influence on the cutting forces and cross-section shape of the chip. Reducing angle KRINS makes the chip thinner at a given feed ***f*** and depth of cut a_p. Whereas if KRINS = 90° the chip thickness ***h = f*** and the chip width $b = a_p$ becomes wider. Regarding the decreasing setting angle, the function width of the T-land is increasing and the rake angle of insert is decreasing. For round inserts, the chip thickness **h** varies from **0** to ***f*** depending on the depth of cut a_p. For that reason we use the average chip thickness value **hm** which is based on the relation a_p/INSD, where **INSD** is the external diameter of the round insert.
Dependence of chip thickness **h** on setting angle **KRINS**



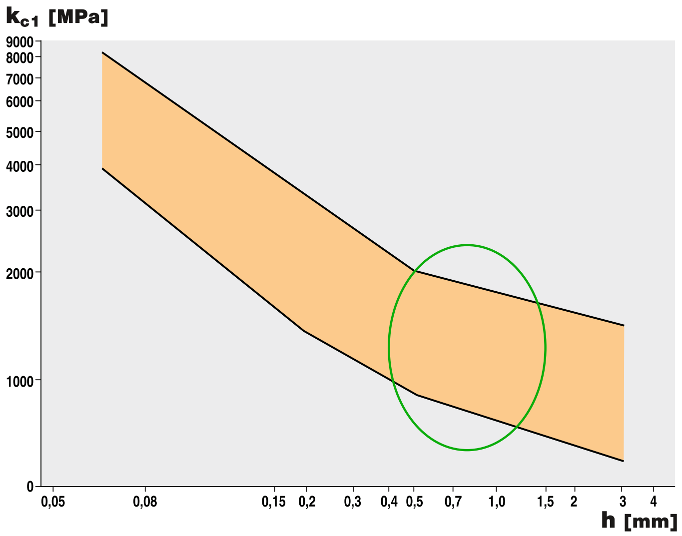
$$h = f \cdot \sin K_r$$



$$h = f \cdot \sqrt{\frac{a_p}{INSD}}$$



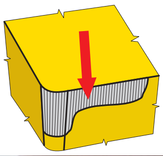
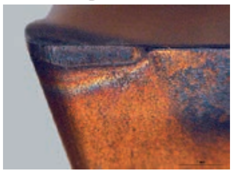
Dependence of specific cutting resistance kc1 on chip thickness



With decreasing chip thickness, the specific cutting resistance increases! Optimal chip thickness range is marked green on the graph.
We recommend using feeds in the range specified in the product section of this catalogue, which are also available on the insert box.
K_{c1} values for various materials are listed in the table on page 47.

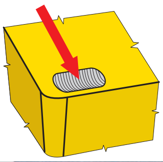
TYPES OF WEAR ON PEELING INSERTS & TROUBLESHOOTING

FLANK WEAR

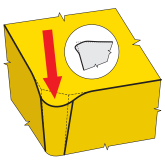
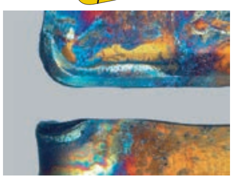
	↑	Use a more wear resistant substrate (s)
(MT)CVD PVD	+ +	Any coating (decisive factor is oxidation resistance – α Al_2O_3)
f	↑	Feed has influence on shape and position of groove
v	↓	Decrease cutting speed
a_p	+	It has no influence
	↑	Increase the clearance angle
		Use coolant or increase its intensity

CRATERING




	↑	Use a more wear resistant substrate (s)
(MT)CVD PVD	+ +	Any coating (decisive factor is thermal resistance – α Al_2O_3)
f	↑	Feed has influence on shape and position of crater
v	↓	Decrease cutting speed
a_p	↓	Minimal effect
	↑	Use more positive cutting geometry
	+ +	Use coolant or increase its intensity

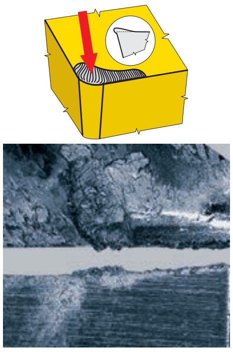
PLASTIC DEFORMATION

	↑	Use a more wear resistant grade (decisive factor is content of Co)
(MT)CVD PVD	+	Any coating (decisive factor is friction)
f	↓	Decrease feed rate
v	↓	Decrease cutting speed
a_p	↓	Minimal effect
	↑	Use another (more positive) cutting geometry
	+ +	Use coolant or increase its intensity

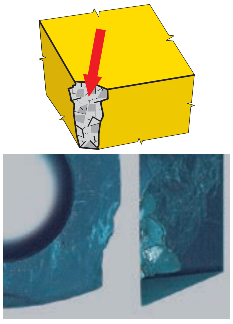
TYPES OF WEAR ON PEELING INSERTS & TROUBLESHOOTING

BUILT–UP EDGE



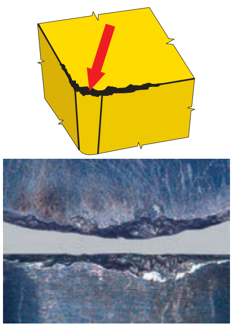
		It has no influence
(MT)CVD / PVD	+ +	Any coating (decisive factor is anti-adhesion effect)
f	↑	The higher the feed rate the less probability of built-up edge creation.
V	↓↑	Change (generally increase) the cutting speed.
a_p		It has no influence
	↓↑	Use more positive geometry
	-	Use a coolant with more effective anti-sticking properties (or no coolant at all)

INSERT FRACTURE



	↓	(H) grain has a great influence
(MT)CVD / PVD	+	PVD coating recommended
f	↓	Reduces the force load
V	↑↓	It is about swarf control and vibration
a_p	↓	Reduces the force load
	↓	Use less positive cutting geometry
		It has no influence
		Use better working conditions

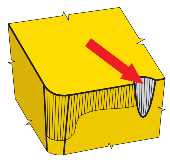
BRITTLE CRACKS AT THE CUTTING EDGE

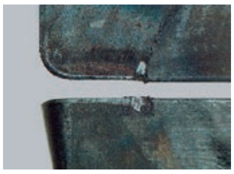


	↓	(H) grain has a great influence
(MT)CVD / PVD	+	PVD coating recommended
f	↓	Good swarf control is very important
V	↑↓	It is about swarf control and vibration
a_p	↓	Reduces the force load (important for machining with long overhangs)
	↓	Use less positive cutting geometry; Use insert with wider T-land
		It has no influence

TYPES OF WEAR ON PEELING INSERTS & TROUBLESHOOTING

SIDE FLANK NOTCH – REMEDY





	↑ ↓	It depends on the character of the damage (abrasive – use more wear resistant substrate; breaking – use tougher substrate)
(MT)CVD PVD	+ +	CVD coating (decisive factor is oxidation resistance – α Al_2O_3)
f	↓	Feed has influence on intensity, but less than the cutting speed
V	↓	Decrease cutting speed
	↓	Minimal effect
	+	Use another (more positive) cutting geometry
		Use tool with smaller setting angle

NON – CIRCULAR BAR CROSS SECTION

Description:	Troubleshooting:
– uneven bar surface (unstable depth of cut)	– check value of cutting depth–(noncircular raw) product = (noncircular final bar)
– non adjusted tool (incorrectly fixed inserts)	– check inserts clamping and slide of cartridge or toolholder
– bars are not brought into peeling head by coaxial way	– check entry rollers adjustment
	– check outgoing rollers adjustment

VIBRATIONS

- guide rollers are adjusted incorrectly
- smoothing edge is too sharp
- small damping facet on smoothing edge
- cutting edge is under axis
- too thin chips (insufficient feed rate)
- uneven or too high wear of inserts

- check leading rollers adjustment
- increase cutting edge rounding
- increase support facet on flank surface facet
- check cutting edge position (to axis or above axis)
- increase feed rate „f“ (mm/rev)
- check insert adjustment

POOR SURFACE (HELICAL TRACE)

- insert clamping is incorrect, worn insert pocket
- feed „f“ (mm/rev) is bigger than length of smoothing edge
- smoothing edge is not parallel to bar axis
- check adjustment and wear of insert (change insert)

- decrease feed rate „f“ (mm/rev)
- check insert adjustment

BAD CHIP FORMATION

- too low feed per insert
- not enough coolant
- incorrect geometry of insert
- increase the feed per insert

- increase coolant efficiency
- change insert geometry

UNEVEN WEAR BETWEEN INDEXABLE INSERTS

- different depth of cut for each indexable insert
- tool holder with damaged insert pocket
- insert clamped incorrectly
- check the tool-holders pre-adjustment

- use only tool-holder in good condition (change the shims if applied)
- clean the inset pocket properly before clamping of new insert

BAR PEELING – TECHNICAL INFO – FORMULAS

Value	Unit	Formula
Number of revolutions	[rev/min]	$n = \frac{v_c \,.\, 1000}{DC \,.\, p}$
Cutting speed	[m/min]	$v_c = \frac{\pi \,.\, DC \,.\, n}{1000}$
Feed per revolution	[mm/rev]	$f_{rev} = \frac{f_{min}}{n} = f_z \,.\, z$
Feed per minute (speed of feed)	[mm/min]	$f_{min} = v_f = f_{rev} \,.\, n = f_z \,.\, z \,.\, n$
Feed per one tool-holder in peeling head	[mm/tooth]	$f_z = \frac{f_{rev}}{z} = \frac{f_{min}}{n \,.\, z}$
Chip cross section	[mm²]	$A = f_z \,.\, a_p$
Chip thickness (for inserts with a straight cutting edge)	[mm]	$h = f_z \,.\, \sin k_r$
Chip thickness (for round cutting inserts)	[mm]	$h = f_z \cdot \sqrt{\frac{a_p}{INSD}}$
Metal removal rate	[cm³/min]	$Q = a_p \cdot f_{rev} \cdot v_c$
Power demand	[kW]	$P_c = \frac{a_p \cdot f_z \cdot v_c \cdot \frac{k_{c1}}{h^{mc}}}{60000\,\eta} \,.\, Z$

Note:

	Quantity	Unit
n	Number of revolutions	[rev/min]
DC	Diameter (of work piece)	[mm]
v_c	Cutting speed	[m/min]
f_{rev}	Feed per revolution of peeling head	[mm/rev]
A	Chip cross section (per one tool-holder / cassette)	[mm²]
a_p	Axial depth of cut (depth of cut)	[mm]
KRINS	Setting angle of insert main cutting edge	[°]
f_{min}	Feed per minute (sometimes called speed of feed)	[mm/min]
f_z	Feed per tooth (one tool-holder)	[mm/tooth]
z	Number of teeth (tool-holder)	[-]
INSD	Diameter of insert	[mm]

	Quantity	Unit
h	Chip thickness	[mm]
Q	Material removal rate per minute	[cm³/min]
Pc	Power demand	[kW]
kc1	Specific cutting force per 1 mm2 chip cross-section (see the table at page 47)	[MPa]
kc	Specific cutting force according to chip cross-section and thickness	[MPa]
η	Machine efficiency usually η = 0,65	[-]
mc	Exponent related to work piece material – (see the table at page 47)	[-]

BAR PEELING – TECHNICAL INFO – SPECIFIC CUTTING FORCE TABLE

TABLE

				Ultimate tensile strength Mpa (N/mm²)	Specific Cutting force kc1 N/mm2	Increase Value mc
P	**P1**	P1.1	Free machining sulfurized carbon steel with a hardness of < 240 HB	≤ 830	1500	0.24
		P1.2	Free machining sulfurized and phosphorized carbon steel with a hardness of < 180 HB	≤ 620	1250	0.24
		P1.3	Free machining sulfurized/phosphorized and leaded carbon steel with a hardness of <180 HB	≤ 620	1250	0.24
	P2	P2.1	Plain low carbon steel containing < 0.25 %C with a hardness of < 180 HB	≤ 620	1250	0.24
		P2.2	Plain medium carbon steel containing < 0.55%C with a hardness of < 240 HB	≤ 830	1500	0.24
		P2.3	Plain high carbon steel containing > 0.55%C, with a hardness of < 300HB	≤ 1030	1650	0.24
	P3	P3.1	Alloy steel with a hardness of < 190 HB	≤ 620	1550	0.24
		P3.2	Alloy steel with a hardness of 180–260 HB	> 620 ≤ 900	1650	0.24
		P3.3	Alloy steel with a hardness of 260–360 HB	> 900 ≤ 1240	1750	0.24
	P4	P4.1	Tool steel with a hardness of < 26 HRC	≤ 900	1800	0.24
		P4.2	Tool steel with a hardness of 26-39 RC	> 900 ≤ 1240	2000	0.24
		P4.3	Tool steel with a hardness of 39-45 HRC	> 1250 ≤ 1450	2300	0.24
M	**M1**	M1.1	Stainless steel, ferritic with a hardness of < 160 HB	≤ 520	1750	0.20
		M1.2	Staintess steel, ferritic with a hardness of 160–220 HB	> 520 ≤ 700	1950	0.20
	M2	M2.1	Stainless steel, martensitic with a hardness of < 200 HB	> 670	2100	0.20
		M2.2	Stainiess steel, martensitic with a hardness of 200–280 HB	> 670 ≤ 950	2200	0.20
		M2.3	Stainless steel, martensitic with a hardness of 280–380 HB	> 950 ≤ 1300	2450	0.20
	M3	M3.1	Stainless steel, austenitic with a hardness of < 200 HB	≤ 730	1900	0.20
		M3.2	Stainiess steel, austenitic with a hardness of 200–260 HB	> 750 ≤ 870	2100	0.20
		M3.3	Stainiess steel, austenitic with a hardness of 260-300 HB	> 870 ≤ 1040	2200	0.20
	M4	M4.1	Stainless steel, austenitic-ferritic or super-austenitic with a hardness of < 300 HB	≤ 990	2350	0.20
		M4.2	Stainless steel, precipitation hardening austenitic with a hardness of 300–380 HB	≤ 1320	2500	0.20
S	**S1**	S1.1	Titanium or titanium alloys, with a hardness of < 200 HB	≤ 660	1400	0.22
		S1.2	Titanium alloys, with a hardness of 200–280 HB	> 660 ≤ 950	1500	0.22
		S1.3	Titanium alloys, a hardness of 280–360 HB	> 950 ≤ 1200	1600	0.22
	S2	S2.1	High-temperature Fe-based alloys with a hardness of < 200 HB	≤ 690	2450	0.24
		S2.2	High-temperature Fe-based alloys with a hardness of 200–280 HB	> 690 ≤ 970	2550	0.24
	S3	S3.1	High-temperature Ni-based alloys with a hardness of < 260 HB	≤ 940	2850	0.24
		S3.2	High-temperature Ni-based alloys with a hardness of 280–360 HB	> 940 ≤ 1200	3100	0.24
	S4	S4.1	High-temperature Co-based alloys with a hardness of < 240HB	≤ 800	2880	0.24
		S4.2	High-temperature Co-based alloys with a hardness of 240–320 HB	>800 ≤ 1070	3100	0.24

RECOMMENDED SCREWS TORQUE AND USEFUL TABLES

RECOMMENDED TORQUE OF CLAMPING SCREWS

Clamping screw	Torque (Nm)	Thread –	Length (mm)
US 20	0.9	M 2	3
US 2205-T07P	0.9	M 2.2	5
US 25	1.2	M 2.5	5
US 2505-T08P	1.2	M 2.5	5
US 2506-T07P	1.2	M 2.5	6
US 3006-T09P	2	M 3	6
US 3007-T09P	2	M 3	7
US 3504-T09P	3	M 3.5	4
US 3507-T15	3	M 3.5	7
US 3509-T15	3	M 3.5	9
US 3511-T15	3	M 3.5	11
US 3512-T15P	3	M 3.5	12
US 4008-T15P	3.5	M 4	8
US 4011-T15P	3.5	M 4	11
US 4511-T20	5	M 4.5	11
US 5012-T15P	5	M 5	12
US 70	5	M 4	5
US 71	5	M 4	7
US 72	5	M 4	9
US 73	5	M 4	11
CS 3007-T08P	1.2	M 3	7
CS 4008-T15P	3	M 4	8
CS 42506-T07P	1	M 2.5	6
CS 43008-T08P	1.2	M 3	8
CS 43509-T10P	2	M 3.5	9
CS 44013-T15P	3	M 4	13
CS 45016-T20P	5	M 5	16
CS 46020-T25P	7.5	M 6	20
CS 48025-T40P	15	M 8	25
CS 5009-T20P	5	M 5	9
CS 5013-T20P	5	M 5	13
CS 5015-T20P	5	M 5	15
CS 6020-T20P	7.5	M 6	20
CS 8025-T30P	15	M 8	25
US 2505-T07P	1.2	M 2.5	5
US 2506-T07P	1.2	M 2.5	6
US 3007-T09P	2	M 3	7
US 3505-T09P	3	M 3.5	5
US 4011A-T15P	3.5	M 4	11
US 4011-T15P	3.5	M 4	11
US 44010-T15P	3.5	M 4	10
US 44012-T15P	3.5	M 4	12
US 45011-T20P	5	M 5	11
US 45012-T20P	5	M 5	12
US 5011-T20P	5	M 5	11
US 5018-T20P	5	M 5	18
US 52506-T07P	0.8	M 2.5	6
US 54511-T15P	5	M 4.5	11
US 62003A-T06P	0.6	M 2	3
US 62004A-T06P	0.6	M 2	4
US 62004-T06P	0.6	M 2	4
US 62505-T07P	1.2	M 2.5	5
US 62506-T07P	1.2	M 2.5	6
US 62506-T08P	1.2	M 2.5	6
US 62508-T08P	1.2	M 2.5	7
US 63009-T09P	1.2	M 3	9
US 63509-T15P	3	M 3.5	10
US 63510-T10P	2	M 3.5	9
US 63511D-T15P	3	M 3.5	11

Clamping screw	Torque (Nm)	Thread –	Length (mm)
US 63513-T15P	3	M 3.5	12
US 64014-T15P	3.5	M 4	14
US 65013-T20	5	M 5	13
US 65014-T20P	5	M 5	14
US 65017-T20P	5	M 5	17
US 66015-T25P	7.5	M 6	15
US 68020-T30P	15	M 8	20
US 68026-T30P	15	M 8	26
US 74016-T15P	3.5	M 4	16

Torque screwdrivers

Torque handle	Torque (Nm)	Clamping screw thread
MR-0.8-2.0 Vario	0.5 – 2.0	M 2 – M 3
MR-1.0-5.0 Vario	0.8 – 5.0	M 2.5 – M 5
MR-0.9 fix	0.9	M 2
MR-2.0 fix	2.0	M 3
MR-3.0 fix	3.0	M 3.5
MR-3.5 fix	3.5	M 4
MR-5.0 fix	5.0	M 5

Replaceable shanks

Replaceable shanks
D-T6
D-T6P
D-T7
D-T7P
D-T8
D-T8P
D-T9
D-T9P
D-T15
D-T15P
D-T20
D-T20P

Screw lubrication

Insert clamping screws are subject to high thermal stresses. It is recommended that all screws be lubricated with a high quality paste such as MOLYKOTE 1000.

RECOMMENDED SCREW TORQUES

CLAMPING SCREW

Screw designation	Screwdriver	Torque (Nm)
28588	MA2-8304	0.8
28992	MA2-8304	0.8
416.1-832	PT-8002	3.6
5513 020-01	PT-8004	3.6
5513 020-03	PT-8001	0.8
5513 020-04	PT-8003	1.5
5513 020-05	PT-8001	0.8
5513 020-14	TX 225PLUS	8.5
5513 020-24	PT-8002	1.5
5513 020-27	PT-8000	0.6
5513 020-28	PT-8000	0.6
5513 021-03	DMN 3124	13
CS 8601-T09P	SDR T09P	1.7
CS 8601-T15P	SDR T15P	3.9
CS 8601-T20P	SDR T20P	6.4
CS 8601-T25P	SDR T25P	9.5
DVF 0573	PT-8002	1.5
DVF 2260	TX 215PLUS	3.6
DVF 3584	DMD 1650	0.6
DVF 3593	TX 207PLUS	0.8
HS 0408	HXK 3	5
HS 0520C	HXK 4	5
HS 0616C	HXK 5	8
HS 0620	HXK 5	6
HS 0620C	HXK 5	6
HS 0625	HXK 5	6
HS 0625C	HXK 5	6
HS 0630	HXK 5	6
HS 0825	HXK 6	10
HS 0830	HXK 6	10
HS 0835	HXK 6	10
HS 0840	HXK 8	11
HS 1030	HXK 8	8
HS 1060	HXK 6	10
HS 93	HXK 5	8
HS 94	HXK 5	8
HSI 1020	HXK 6	8
PS 0512	HXK 2	2
PS 0512-A	HXK 2	2
PS 0616	HXK 2,5	4
PS 12040	HXK 5	8
PS 6026-709P	SRD T09P	2

CLAMPING SCREW

Screw designation	Screwdriver	Torque (Nm)
PS 8290	HXK 2	2
SR 14	HXK 10	10
SR 85011-T15P	SDR T15P	5
SR 85017-T09P	SDR T09P	2
SR 85020-T15P	SDR T15P	3
SR 86025-T20P	SRD T20P	5
T20.037	DMD 1650	0.6
UP 0909-T09P	SRD T09P	2
UP 1515-T15P	SDR T15P	8
US 2505-T07P	SDR T07P	0.9
US 2506-T07P	SDR T07P	0.9
US 3007-T09P	SDR T09P	2
US 34	HXK 3	5
US 35	HXK 4	6
US 3508-T15P	SDR T15P	3
US 3510A-T15P	SDR T15P	3
US 3510-T15P	SDR T15P	3
US 3512A-T15P	SDR T15P	3
US 3512-T15P	SDR T15P	3
US 36	HXK 4	6
US 38	HXK 5	8
US 39	HXK 5	8
US 40	HXK 4	6
US 4008-T15P	SDR T15P	3.5
US 4011-T15P	SDR T15P	3.5
US 41	HXK 4	6
US 42	HXK 4	6
US 45013-T20P	SDR T20P	5
US 4512-T15P	SDR T15P	5
US 4514A-T20	SDR T20	5
US 46	HXK 3	5
US 46017-T20P	SDR T20P	5
US 47	HXK 5	8
US 5012-T15P	SDR T15P	5
US 5015-T20P	SDR T20P	5
US 5018-T20P	SDR T20P	5
US 6020-T25P	SDR T25P	6
US 64518-T15P	SDR T15P	5
US 8025-T30P	SDR T20P	13
US 83	HXK 4	6
US 95	HXK 4	10

TORQUE SCREWDRIVERS

Torque handle	Torque (Nm)	Clamping screw thread
MR-0.8-2.0 vario	0.5 – 2.0	M 2 – M 3
MR-1.0-5.0 vario	0.8 – 5.0	M 2.5 – M 5
MR-0.9 fix	0.9	M 2
MR-2.0 fix	2.0	M 3
MR-3.0 fix	3.0	M 3.5
MR-3.5 fix	3.5	M 4
MR-5.0 fix	5.0	M 5

REPLACEABLE SHANKS

Replaceable shanks		
D-T6	**D-T8**	**D-T15**
D-T6P	**D-T8P**	**D-T15P**
D-T7	**D-T9**	**D-T20**
D-T7P	**D-T9P**	**D-T20P**

SCREW LUBRICATION

Insert clamping screws are subject to high thermal stresses. It is recommended that all screws be lubricated with a high quality paste such as MOLYKOTE 1000.

HARDNESS CONVERSION TABLE

Strength (MPa)	Hardness			
	BRINELL	VICKERS	ROCKWELL	ROCKWELL
R_m	HB	HV	HRB	HRC
285	86	**90**	1190	–
320	95	**100**	56.2	–
350	105	**110**	62.3	–
385	114	**120**	66.7	–
415	124	**130**	71.2	–
450	133	**140**	75.0	–
480	143	**150**	78.7	–
510	152	**160**	81.7	–
545	162	**170**	85.8	–
575	171	**180**	87.1	–
610	181	**190**	89.5	–
640	190	**200**	91.5	–
675	199	**210**	93.5	–
705	209	**220**	95	–
740	219	**230**	96.7	–
770	228	**240**	98.1	–
800	238	**250**	99.5	–
820	242	**255**	–	23.1
850	252	**265**	–	24.8
880	261	**275**	–	26.4
900	266	**280**	–	27.1
930	276	**290**	–	28.5
950	280	**295**	–	29.2
995	295	**310**	–	31.0
1030	304	**320**	–	32.2
1060	314	**330**	–	33.3
1095	323	**340**	–	34.4
1125	333	**350**	–	35.5
1155	342	**360**	–	36.6
1190	352	**370**	–	37.7
1220	361	**380**	–	38.8
1255	371	**390**	–	39.8
1290	380	**400**	–	40.8
1320	390	**410**	–	41.8
1350	399	**420**	–	42.7
1385	409	**430**	–	43.6
1420	418	**440**	–	44.5
1455	428	**450**	–	45.3
1485	437	**460**	–	46.1
1520	447	**470**	–	46.9
1555	456	**480**	–	47.7
1595	466	**490**	–	48.4
1630	475	**500**	–	49.1
1665	485	**510**	–	49.8
1700	494	**520**	–	50.5
1740	504	**530**	–	51.1
1775	513	**540**	–	51.7
1810	523	**550**	–	52.3
1845	532	**560**	–	53.0
1880	542	**570**	–	53.6
1920	551	**580**	–	54.1
1955	561	**590**	–	54.7
1995	570	**600**	–	55.2
2030	580	**610**	–	55.7
2070	589	**620**	–	56.3
2105	599	**630**	–	56.8
2145	608	**640**	–	57.3
2180	618	**650**	–	57.8

Industry Standard tolerancess For Shafts & Holes

Tolerance values are shown in Microns (µm)

Formula for Microns ...1 µm = 0.001 mm / 0.000039 ″

Tolerance	**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

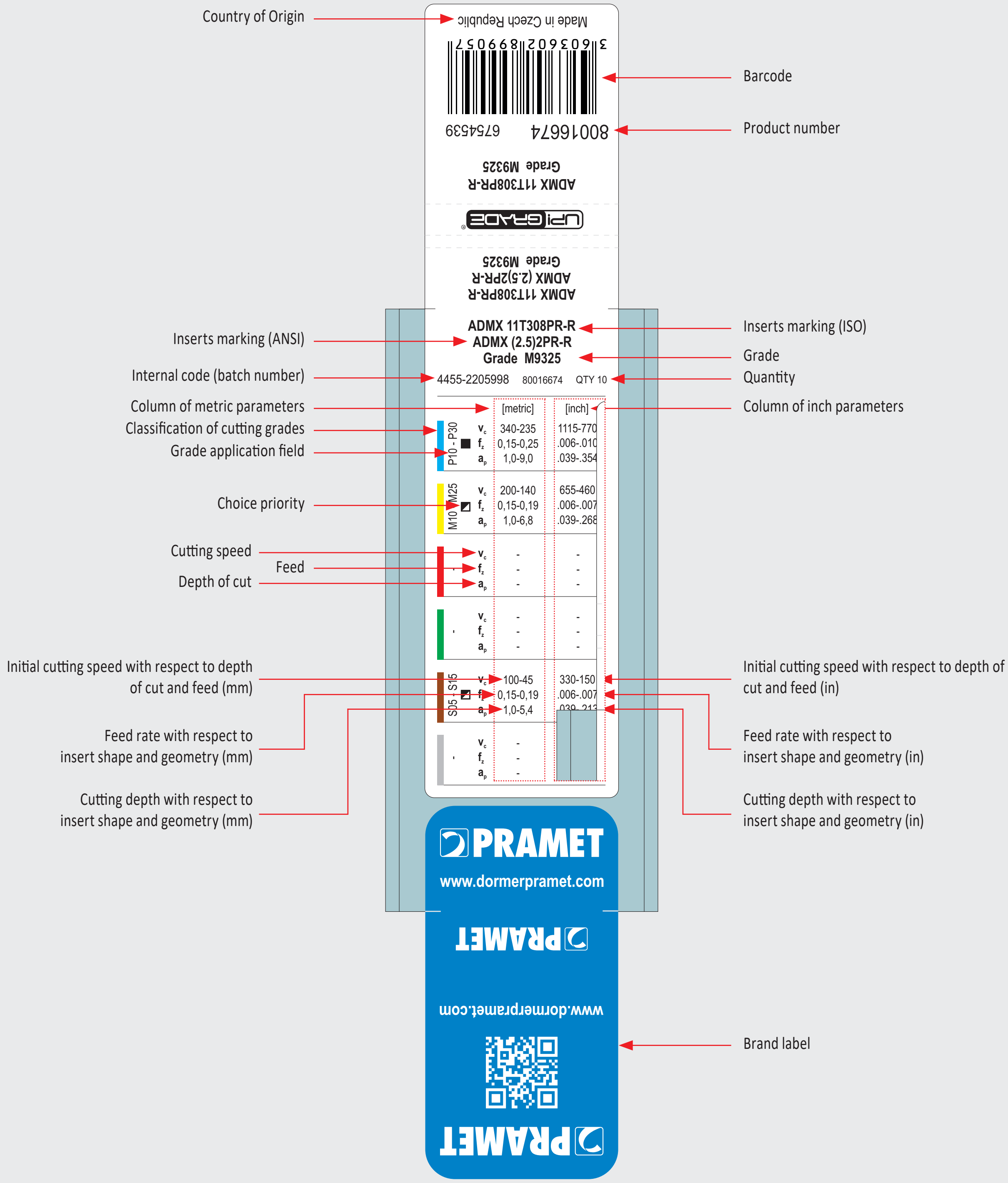
RECOMMENDED DRILL SIZES FOR TAPPING

Metric ISO threads		Recommended drill diameter for	
Thread	Pitch	Cutting tap	Fluteless tap
M16 × 1.0	1.00	15.0	15.5
M16 × 0.75	0.75	15.3	–
M17 × 1.0	1.00	16.0	–
M18	2.50	15.5	16.8
M18 × 2.0	2.00	16.0	–
M18 × 1.5	1.50	16.5	17.3
M18 × 1.0	1.00	17.0	–
M20	2.50	17.5	18.8
M20 × 2.0	2.00	18.0	–
M20 × 1.5	1.50	18.5	19.3
M20 × 1.0	1.00	19.0	–
M22	2.50	19.5	20.8
M22 × 2.0	2.00	20.0	–
M22 × 1.5	1.50	20.5	21.3
M22 × 1.0	1.00	21.0	–
M24	3.00	21.0	22.5
M24 × 2.0	2.00	22.0	–
M24 × 1.5	1.50	22.5	23.3
M27	3.00	24.0	–
M27 × 2.0	2.00	25.0	–
M30	3.50	26.5	–
M30 × 2.0	2.00	28.0	–
M33	3.50	29.5	–
M36	4.00	32.0	–
M36 × 3.0	3.00	33.0	–
M39	4.00	35.0	–
M42	4.50	37.5	–
M42 × 3.0	3.00	39.0	–
M45	4.50	40.5	–
M48	5.00	43.0	–
M48 × 3.0	3.00	45.0	–
M52	5.00	47.0	–
M52 × 3.0	3.00	48.0	–

Inch threads UNC		Recommended drill diameter for	
Thread	Pitch	Cutting tap	Fluteless tap
3/4"	10	16.7	17.8
7/8"	9	19.5	20.8
1"	8	22.2	23.8
1 1/8"	7	25.0	–
1 1/4"	7	28.2	–
1 3/8"	6	31.0	–
1 1/2"	6	34.0	–
1 3/4"	5	39.5	–
2"	4 1/2	45.2	–
2 1/4"	4 1/2	51.6	–
2 1/2"	4	57.2	–

Whitworth pipe threads		Recommended drill diameter for	
Thread	Pitch	Cutting tap	Fluteless tap
G 3/8"	19	15.3	16.0
G 1/2"	14	19.0	20.0
G 5/8"	14	21.0	22.0
G 3/4"	14	24.5	25.5
G 7/8"	14	28.3	29.3
G 1"	11	30.8	32.0
G 1 1/8"	11	35.5	–
G 1 1/4"	11	39.5	–
G 1 3/8"	11	41.8	–
G 1 1/2"	11	45.3	–
G 1 3/4"	11	51.0	–
G 2"	11	57.0	–

Inch threads UNF		Recommended drill diameter for	
Thread	Pitch	Cutting tap	Fluteless tap
3/4"	16	17.5	18.3
7/8"	14	20.5	21.3
1"	12	23.4	24.3
1 1/8"	12	26.5	–
1 1/4"	12	29.8	–
1 3/8"	12	33.0	–
1 1/2"	12	36.0	–


Country of Origin
Made in Czech Republic
Barcode
Product number
80016674
ADMX 11T308PR-R
Grade M9325
ADMX (2.5)2PR-R
Inserts marking (ANSI)
Inserts marking (ISO)
Grade
Internal code (batch number)
4455-2205998
Quantity
QTY 10
Column of metric parameters
[metric]
Column of inch parameters
[inch]
Classification of cutting grades
Grade application field
P10 - P30
340-235
0,15-0,25
1,0-9,0
1115-770
.006-.010
.039-.354
Choice priority
M10 - M25
200-140
0,15-0,19
1,0-6,8
655-460
.006-.007
.039-.268
Cutting speed
Feed
Depth of cut
Initial cutting speed with respect to depth of cut and feed (mm)
Initial cutting speed with respect to depth of cut and feed (in)
S05 - S15
100-45
0,15-0,19
1,0-5,4
330-150
.006-.007
Feed rate with respect to insert shape and geometry (mm)
Feed rate with respect to insert shape and geometry (in)
Cutting depth with respect to insert shape and geometry (mm)
Cutting depth with respect to insert shape and geometry (in)
PRAMET
www.dormerpramet.com
Brand label

SIMPLY RELIABLE

As a professional you can judge the quality of work by just looking at the chip. Our chip is a clean and uncomplicated shape that in itself tells a story. It is a clear and consistent signal and that's why we use it as a symbol for being **Simply Reliable**.

DORMER PRAMET www.dormerpramet.com

CONTACTS TO LOCAL SALES SUPPORT ALWAYS UP TO DATE!

FOLLOW US...

ONLINE

SEGMENTS

LIBRARY APP.

CALCULATOR APP.