Member IMC Group
Ingersoll
Cutting Tools

TECHNICAL GUIDE
MILLING AND DRILLING

SPEED UP
HIGH SPEED & FEED

Ingersoll's standard program comprises a broad and world-wide established range of cutting tools, suitable for the most various applications.

This range of cutting tools is constantly expanded: End mills, shell end mills, shoulder-type milling cutters, face mills, slotting cutters, form milling cutters, indexable drills, solid carbide, adaptions, set-up equipment and indexable inserts. With a complete line of turning and engraving tools we can now offer our customers a new group of products, thus expanding our capabilities as a broad-range supplier.

The development and production of special-purpose tools according to customer-specific requirements is another important factor for Ingersoll Werkzeuge GmbH. Our know-how and great potential of experience, combined with our own demand for quality, functionality and innovation, guarantees our customers the optimum cutting tool solution – for individual machining tasks, for all industries.

Content

Workpiece material comparison chart	Page	6-11
Surface finish comparison chart	Page	12
Hardness comparison chart	Page	13
Grades – Coatings	Page	14-15
Designation system for ISO indexable inserts	Page	16-17
General formulas	Page	18
Conventional and climb milling / Average chip thickness	Page	19
Correct instalation of the inserts	Page	20
Torque wrench range	Page	21
Chip formation model / Initial contact	Page	22
Consolidated heat value / Heat flow	Page	23
Chips	Page	24
Measures for milling problems	Page	25
Circular interpolation milling / Ramping Capability / High feed milling	Page	26-27
Characteristics of GoldSFeed 13 & 19 mm for 90° shoulder milling	Page	28
Characteristics of GoldQuad^{XXX} 13 & 19 mm for 90° shoulder milling	Page	29
GoldQuad^{XXX} 30° bevel high feed face mill	Page	30
Power output calculation / Drive Power	Page	31
Tips	Page	32
Operating instructions	Page	33
Circular thread milling / Finish milling	Page	34
Finish milling	Page	35-36
Operating instructions / Finish milling	Page	37
Finish milling	Page	38
Adjustment instructions finish mill	Page	39-43
Adjustment instructions rough-finishing cutter	Page	44
Adjustment instructions semi-finishing cutter	Page	45
Adjustment instructions shoulder-type milling cutter	Page	46
Adjustment instructions slotting cutter	Page	47-48
Adjustment instructions duo-bore finishing tool	Page	49
Technical features	Page	50-52
Solid carbide cutters	Page	53
ChipSurfer	Page	54
Installation of the ChipSurfer	Page	55
Technical features conical segment cutter	Page	56-60
Thread milling	Page	62-65
Drilling QuadDrillPlus	Page	66-67
Drilling QuadTwist	Page	68-71
Drilling GoldTwin	Page	72-73
Drilling SpadeTwist	Page	74
Drilling GoldTwist	Page	75-81
Drilling DeepTwist	Page	82
Reamer QwikReam	Page	83-85
Deep hole drill DeepTrio	Page	86
Deep hole drill	Page	87-90

Workpiece material comparison chart

	Material No.	Designation	Old designation	CN	IN
P	Unalloyed steel				
	1.0044	S275JR	St 44-2	Q255A; U12552	E250
	1.0050	E295	St 50-2	Q275A; U12752	E350
	1.0060	E335	St 60-2	HRB 335; L33350	Fe540
	1.0301	C10	–	DX 1; U59110	10C4
	1.0345	P235GH	St 35.8	Q245R; U50245	Fe360H
	1.0401	C15	–	15; U20152	14C6
	1.0402	C22	–	–	20C8
	1.0473	P355GH	19 Mn 6	Q345R; U50345	–
	1.0481	P295GH	17 Mn 4	–	–
	1.0501	C35	–	–	35C8
	1.0503	C45	C 45	–	45C8
	1.0562	P355N	StE 355	–	–
	1.0601	C60	–	–	60C6
	1.0715	11SMn30	9SMn28	–	11C10S25
	1.0726	35S20	35 S 20	Y35; U70352	–
	1.0727	46S20	45 S 20	Y45; U70452	48C8S20
	Tool steel				
	1.2067	102Cr6	100 Cr 6	Cr2; T30201	TAC6; T105Cr5
	1.2312	40CrMnMoS8-6	–	–	–
	1.2316	X38CrMo16	X 36 CrMo 17	–	–
	1.2343	X37CrMoV5-1	X 38 CrMoV 5 1	4Cr5MoSiV; T20501	TAH2; XT35Cr5Mo1V3
	1.2344	X40CrMoV5-1	X40CrMoV5-1	SM 4Cr5MoSiV1; T22052	TAH3; XT35Cr5MoV1
	1.2363	X100CrMoV5	X100CrMoV5-1	Cr5Mo1V; T20503	T100Cr20Mo10V4
	1.2379	X153CrMoV12	X153CrMoV12	Cr12Mo1V1; T21202	TAC21; XT160Cr12
	1.2419	105WCr6	105WCr6	CrWMn; T20111	–
	1.2714	55NiCrMoV7	56 NiCrMoV 7	–	TAH7; T55Ni7Cr4Mo5V1
	1.2767	45NiCrMo16	X 45 NiCrMo 4	–	–
	High speed steel				
	1.3343	HS6-5-2C	DMo 5	W6Mo5Cr4V2; T66541	THS4; XT87W6Mo5Cr4V2
	Construction, machine and container steel				
	1.5415	16Mo3	15 Mo 3	15MoG; A65158	16Mo3H
	1.5423	16Mo5	–	–	15Mo6H
	1.6311	20MnMoNi4-5	–	–	–
	Case hardened steel				
	1.7131	16MnCr5	–	20CrMnH; A22205	16Mn5Cr4
	1.7147	20MnCr5	–	20CrMnAH; A22207	20Mn5Cr5
	Tempering steel				
	1.7262	15CrMo5	–	–	15Cr4Mo3
	1.7218	25CrMo4	–	30CrMo; A30302	CDS-7
	1.7225	42CrMo4	–	42CrMo; A30422	CDS-8
	1.6582	34CrNiMo6	–	–	36Ni6Cr6Mo2
	Heat resistant steel				
	1.7321	20MoCr4	–	–	–
	1.7335	13CrMo4-5	13 CrMo 4 4	12CrMo; A30122	12Cr4Mo5H
	1.7362	X12CrMo5	–	1Cr5Mo; S45110	10Cr20Mo6
	1.7380	10CrMo9-10	10 CrMo 9 10	12Cr2Mo1R; A30127	12Cr9Mo10H
	1.7386	X11CrMo9-1	X12CrMo9-1	–	10Cr36Mo10H
	Spring steel				
	1.7176	55Cr3	–	55CrMnA; A22553	TAC23; T55Cr3
	1.8159	51CrV4	50 CrV 4	50CrVA; A23503	50Cr4V2
	Nitriding steel				
	1.8507	34CrAlMo5-10	34CrAlMo5	–	–
	1.8519	31CrMoV9	–	–	–
	1.8550	34CrAlNi7-10	–	–	–
	Fine grained steel				
	1.8902	S420N	StE 420	Q420C; L04203	–

JP	RU	USA	Strength / Hardness	Machinability
Unalloyed steel				
STKR 400	St4sp	1020	380–580 MPa	++
SS 490	St5sp	–	440–660 MPa	++
SM 570	St6sp	–	540–770 MPa	++
S 10 C	10	1010	490–780 MPa	++
SPV 450	–	K02801	360–500 MPa	++
S 15 C	15	1015	330–600 MPa	++
S 22 C	–	1022	380–580 MPa	++
SPV 355	–	K12437	510–650 MPa	++
SPV 315	14G2	K03501	460–580 MPa	++
S 35 C	–	1035	550–780 MPa	++
S 45 C	–	1045	630–850 MPa	++
SM 490 A	15GF	K12609	450–630 MPa	++
S 58 C	60	1060	750–1000 MPa	++
SUM 22	–	1215	360–570 MPa	++
–	–	1140	500–720 MPa	++
–	–	1139	580–760 MPa	++
Tool steel				
SUJ 2	Ch	52100; J19965	≤ 223 HB	++
–	–	–	≤ 230 HB	++
–	–	–	≤ 250 HB	++
SKD 6	4Ch5MFS	H 11	≤ 229 HB	+
SKD 61	4Ch5MF1S	H 13	≤ 229 HB	+
SKD 12	–	A 2	≤ 241 HB	+
SKD 10	–	D 2	≤ 255 HB	-
SKS 2	ChW1G	–	≤ 230 HB	+
SKT 4	5ChGNM	L 6	≤ 248 HB	++
SKT 6	–	–	≤ 285 HB	++
High speed steel				
SKH 51	R6M5	M 2	≤ 269 HB	-
Construction, machine and container steel				
STFA 12	–	–	440–590 MPa	++
SB 450 M	–	4419 H	450–590 MPa	++
SQV 2 A	–	K12554	570–750 MPa	++
Case hardened steel				
–	18ChG	5115	≤ 550 MPa	++
SMnC 420 (H)	–	5120	780–1370 MPa	++
Tempering steel				
SCM 415	–	–	≤ 207 MPa	++
SCM 420 TK	30ChM	4130	≤ 580 MPa	++
SCM 440 (H)	–	4140	≤ 630 MPa	++
SNCM447	38Ch2N2MA	4340	≤ 680 MPa	++
Heat resistant steel				
–	–	4118	≤ 550 MPa	++
STPA 22	12ChM	K12062	450–660 MPa	++
STPA 25	–	501	590–740 MPa	+
STPA 24	12Ch8	K21590	480–630 MPa	++
STPA 26	–	K90941	590–740 MPa	+
Spring steel				
SUP 9	50ChGA	5160	≤ 210 HB	++
SUP 10	50ChFA	6145	≤ 248 HB	+
Nitriding steel				
–	–	K23510	≤ 1000 MPa	++
–	–		1100–1230 MPa	+
–	–	K52440	1000–1470 MPa	++
Fine grained steel				
SM 490 C	–	K12437	500–680 MPa	-

Workpiece material comparison chart

	Material No.	Designation	Old designation	CN	IN
M	Stainless steel with < 2,5% Ni				
	1.4002	X6CrAl13	–	06Cr13Al; S11348	X04Cr12 (405)
	1.4006	X12Cr13	X10Cr13	12Cr12; S40310	X12Cr12
	1.4016	X6Cr17	–	10Cr17; S11710	X07Cr17
	1.4021	X20Cr13	X20Cr13	20Cr13; S42020	X20Cr13
	1.4028	X30Cr13	–	30Cr13; S42030	X30Cr13
	1.4031	X39Cr13	–	40Cr13; S42040	X40Cr13
	1.4034	X46Cr13	–	–	–
	1.4057	X17CrNi16-2	X 22 CrNi 17	17Cr16Ni2; S43120	X15Cr16Ni2
	Stainless steel with Mo and < 2,5% Ni				
	1.4104	X14CrMoS17	X 12 CrMoS 17	Y10Cr17; S11717	–
	1.4113	X6CrMo17-1	–	10Cr17Mo; S11790	–
	Stainless steel with ≥ 2,5% Ni				
	1.4301	X5CrNi18-10	X 5 CrNi 18-9	06Cr19Ni9; S30408	X04Cr19Ni9
	1.4303	X4CrNi18-12	X5CrNi18-12	06Cr18Ni12; S30508	X04Cr18Ni10
	1.4305	X8CrNiS18-9	X 10 CrNiS 18 9	Y12Cr18Ni9; S30317	–
	1.4306	X2CrNi19-11	–	022Cr19Ni10; S30403	X02Cr19Ni10
	1.4310	X10CrNi18-8	X12CrNi17-7	12Cr17Ni17; S30110	X07Cr18Ni9
	1.4311	X2CrNiN18-10	–	022Cr19Ni10N; S30453	TP304LN
	1.4317	GX4CrNi13-4	–	ZG06Cr12Ni4; C54860	–
	1.4362	X2CrNiN23-4	–	022Cr23Ni4MoCuN; S23043	–
	1.4371	X2CrMnNiN17-7-5	–	12Cr18Mn9Ni5N; S35450	TP202
	Stainless steel with Mo and ≥ 2,5% Ni				
	1.4401	X5CrNiMo17-12-2	X 5 CrNiMo 18 10	06Cr17Ni12Mo2; S31608	X04Cr17Ni12Mo2
	1.4404	X2CrNiMo17-12-2	–	022Cr17Ni12Mo2; S31603	X02Cr17Ni12Mo2
	1.4429	X2CrNiMoN17-13-3	–	022Cr17Ni12Mo2N; S31653	X04Cr17Ni12Mo2N
	1.4438	X2CrNiMo18-15-4	–	022Cr19Ni13Mo3; S31703	TP317L
	1.4462	X2CrNiMoN22-5-3	–	022Cr22Ni5Mo3N; S22253	–
	Stainless steel with special additives				
	1.4501	X2CrNiMoCuWN25-7-4	–	022Cr25Ni7Mo4WCuN; S27603	–
	1.4510	X3CrTi17	–		–
	1.4512	X2CrTi12	–	022Cr11NbTi; S11173	–
	1.4539	X1NiCrMoCu25-20-5	X 1 NiCrMoCuN 25 20 5	015Cr21Ni26Mo5Cu2; S31782	–
	1.4541	X6CrNiTi18-10	X 6 CrNiTi 18 10	06Cr18Ni11Ti; S32168	X04Cr18Ni10Ti
	1.4542	X5CrNiCuNb16-4	X 5 CrNiCuNb 17 4	05Cr17Ni4Cu4Nb; S51740	–
	1.4550	X6CrNiNb18-10	X 6 CrNiNb 18 10	1Cr19Ni11Nb; S34771	X04Cr18Ni10Nb
	1.4571	X6CrNiMoTi17-12-2	X 6 CrNiMoTi 17 12 2	06Cr17Ni12Mo2Ti; S31668	X04Cr17Ni12Mo2Ti

K	Cast iron with flake graphite				
	EN-JL1020	EN-GJL-150	GG-15	HT150; C00150	–
	EN-JL1030	EN-GJL-200	GG-20	HT200; C00200	–
	EN-JL1040	EN-GJL-250	GG-25	HT250; C00250	–
	EN-JL1050	EN-GJL-300	GG-30	HT300; C00300	–
	EN-JL1060	EN-GJL-350	GG-35	HT350; C00350	–
	Modular cast iron				
	EN-JS1072	EN-GJS-400-15U	GGG-40	QT400-15; C01401	–
	EN-JS1082	EN-GJS-500-7U	GGG-50	–	–
	EN-JS1092	EN-GJS-600-3U	GGG-60	–	–
	EN-JS1102	EN-GJS-700-2U	GGG-70	QT700-2; C01700	–
	EN-JS1080	EN-GJS-800-2	GGG-80	QT800-2; C01800	–
	Malleable cast iron				
	EN-JM1130	EN-GJMB-350-10	GTS-35-10	KTH350-10; C02354	–
	EN-JM1140	EN-GJMB-450-6	GTS-45-06	KTZ450-06; C02452	–
	EN-JM1160	EN-GJMB-550-4	GTS-55-04	KTZ550-04; C02551	–
	EN-JM1180	EN-GJMB-650-2	GTS-65-02	KTZ650-02; C02650	–
	EN-JM1190	EN-GJMB-700-2	GTS-70-02	KTZ700-02; C02700	–

JP	RU	USA	Strength / Hardness	Machinability
Stainless steel with < 2,5% Ni				
SUS 405TB	–	405	400–600 MPa	-
SUS 410-SD	15Ch13L	410	≤ 730 MPa	+
SUS 430TB	12Ch17	430	400–630 MPa	+
SUS 420J1	20Ch13	420	≤ 760 MPa	+
SUS 420J2	30Ch13	420 F	≤ 800 MPa	+
–	–	S42080	800–1000 MPa	+
–	40Ch13	S42000	≤ 950 MPa	+
SUS 431	20Ch17N2	431	≤ 950 MPa	+
Stainless steel with Mo and < 2,5% Ni				
SUS 430F	–	430 F	≤ 730 MPa	+
SU S434	–	434	440–660 MPa	-
Stainless steel with ≥ 2,5% Ni				
SUS 304-SD	08Ch18N10	304	500–700 MPa	-
SUS 305	06Ch18N11	305 L	500–700 MPa	-
SUS 303	–	303	500–750 MPa	-
SUS 304LTB	03Ch18N11	304 L	460–680 MPa	-
SUS 301	12Ch18N9	301	500–750 MPa	-
SUS 304LN	–	304 LN	550–760 MPa	-
SCS 6	–	J91540	760–960 MPa	-
–	–	S32304	600–830 MPa	-
–	–	–	650–850 MPa	-
Stainless steel with Mo and ≥ 2,5% Ni				
SUS 316-SD	08Ch16N11M3	316 H	500–700 MPa	-
SUS 316LTP	03Ch17N14M3	316 L	500–700 MPa	-
SUS 316LN	–	316 LN	580–800 MPa	-
SUS 317LTB	–	317 L	500–700 MPa	-
SUS 329J3LTB	–	S31803	650–880 MPa	-
Stainless steel with special additives				
–	–	S32760	630–710 MPa	-
SUS 430LXTB	08Ch17T	439	450–600 MPa	-
SUS 409TB	–	409	390–560 MPa	+
–	–	904 L	530–730 MPa	-
SUS 321TB	06Ch18N10T	321	500–700 MPa	-
SUS 630	–	S17400	≤ 1200 MPa	--
SUS 347TB	08Ch18N12B	347	510–940 MPa	-
SUS 316TiTB	08Ch16N11M3T	316 Ti	500–950 MPa	-

Cast iron with flake graphite				
FC 15	Sč 15	ASTM A 48 (25B)	80–155 HB	++
FC 20	Sč 20	ASTM A 48 (30B)	115–205 HB	++
FC 25	Sč 25	ASTM A 48 (40B)	155–250 HB	+
FC 30	Sč 30	ASTM A 48 (45B)	195–270 HB	+
FC 35	Sč 35	ASTM A 48 (50B)	275–285 HB	+
Modular cast iron				
FCD 400	VC 40	ASTM A 536 (60-40-18)	≥ 400 MPa	+
FCD 500	VC 50	ASTM A 536 (60-45-12)	≥ 500 MPa	+
FCD 600	**VC 60**	**ASTM A 536 (80-55-06)**	**≥ 600 MPa**	**+**
FCD 700	VC 70	ASTM A 536 (100-70-03)	≥ 700 MPa	-
FCD 800	VC 80	ASTM A 536 (120-90-02)	≥ 800 MPa	-
Malleable cast iron				
FCMB 340	KC 35-10	ASTM A 47 (Grade 22010)	≥ 350 MPa	+
–	KC 45-7	–	≥ 450 MPa	+
FCMP 540	KC 55-4	–	≥ 550 MPa	+
–	KC 63-3	–	≥ 650 MPa	+
FCMP 690	KC 70-2	ASTM A 220 (Grade 70003)	≥ 700 MPa	+

Workpiece material comparison chart

	Material No.	Designation	Old designation	CN	IN
N	Aluminum alloys for mechanical engineering and aerospace				
	EN AW-5005	AlMg1	33.315	–	–
	EN AW-5052	AlMg2,5	33.523	–	–
	EN AW-5083	AlMg4,5Mn0,7	33.547	–	–
	EN AW-6060	AlMgSi0,5	33.206	–	–
	EN AW-6061	AlMg1SiCu	33.214	–	–
	EN AW-6082	**AlMgSi1**	**32.315**	**–**	**–**
	EN AW-2017	AlCuMg1	31.324	–	–
	EN AW-2024	**AlCu4Mg1**	**31.354**	**–**	**–**
	EN AW-7010	AlZn6MgCu	34.394	–	–
	EN AW-7050	AlZn6CuMgZr	34.144	–	–
	EN AW-7075	**AlZn5,5MgCu**	**34.364**	**–**	**–**
	EN AC-47000	**EN AC-Al Si12(Cu)**	**32.583**	**G-AlSi12(Cu)**	**–**
	Cooper alloys				
	CW021A	Cu-HCP	20.070	–	–
	CW452K	CuSn6	21.020	–	–
	Brass				
	CW614N	CuZn39Pb3	20.401	–	–
	Bronze				
	CC493K	CuSn7Zn4Pb7-C	21.090	–	–
S	Heat resistant steel				
	1.4724	**X10CrAlSi13**	**X 10 CrAl 13**	**–**	**–**
	1.4742	X10CrAlSi18	X 10 CrAl 18	–	–
	1.4762	X10CrAlSi25	X 10 CrAl 24	–	–
	Heat resistant steel with ≥ 2,5% Ni				
	1.4828	X15CrNiSi20-12	–	16Cr20Ni14Si2; S38240	X15Cr24Ni13
	1.4845	X8CrNi25-21	X 12 CrNi 25 21	0Cr25Ni20; S31008	10Cr25Ni18
	1.4848	GX40CrNiSi25-20	–	ZG40Cr25Ni20Si2; C53901	–
	1.4849	GX40NiCrSiNb38-19	–	ZG40Ni38Cr19Si2Nb1; C53952	–
	1.4864	X12NiCrSi35-16	X 12 NiCrSi 36 16	12Cr16Ni35; S33010	–
	1.4876	X10NiCrAlTi32-21	X 10 NiCrAlTi 32 20	NS 112; H01120	–
	1.4878	X8CrNiTi18-10	X 12 CrNiTi 18 9	1Cr18Ni9Ti; S32160	–
	High heat resisting material				
	1.4923	X22CrMoV12-1	X21CrMoNiV12-1	21Cr12MoV; S46020	X21Cr12Mo1V3
	1.4948	X6CrNi18-10	X6CrNi18-11	07Cr19Ni10; S30409	7Cr18Ni10H
	Titanium and superalloys				
	2.4610	**NiMo16Cr16Ti**	**–**	**NS 335; H03350**	**–**
	2.4631	NiCr20TiAl	–	–	–
	2.4632	NiCr20Co18Ti	–	–	–
	2.4634	NiCo20Cr15MoAlTi	–	–	–
	2.4668	NiCr19Fe19Nb5Mo3	Inconel 718	GH 4169; H41690	–
	3.7025	**Ti 1**	**–**	**–**	**–**
	3.7035	Ti 2	–	–	–
	3.7055	Ti 3	–	–	–
	3.7065	Ti 4	–	–	–
	3.7164	Ti 6 Al 4 V	–	–	–
	3.7115	Ti 5 Al 2.5 Sn	–	–	–

Member IMC Group Ingersoll Cutting Tools

JP	RU	USA	Strength / Hardness	Machinability
Aluminum alloys for mechanical engineering and aerospace				
–	–	5005	100–155 MPa	+
–	–	5052	160–235 MPa	+
–	–	5083	255–275 MPa	+
–	–	6060	120–200 MPa	++
–	–	6061	195–315 MPa	++
–	–	6082	195–350 MPa	++
–	–	2017	375–410 MPa	++
–	–	2024	410–470 MPa	++
–	–	7010	430–500 MPa	++
–	–	7050	430–500 MPa	++
–	–	7075	420–530 MPa	++
–	–	–	150–170 MPa	+
Cooper alloys				
–	–	C10800	200–360 MPa	-
C5191	–	C51900	350–720 MPa	-
Brass				
C3603	–	C38500	360–550 MPa	++
Bronze				
–	–	C93200	230–260 MPa	++

Heat resistant steel				
–	–	–	450–650 MPa	-
SUH 21	15Ch18SJu	–	500–700 MPa	-
SUH 446	–	S44600	520–720 MPa	-
Heat resistant steel with ≥ 2,5% Ni				
SUS 309TB	20Ch20N14S2	309	500–750 MPa	-
SUS 310TB	10Ch23N18	310 H	500–700 MPa	-
SCH 22X	–	J94224	440–640 MPa	-
SCH 20XNb	–	N08005	400–600 MPa	-
SUH 330	–	N08330	550–750 MPa	-
NCF 800 TB	–	N08800	450–680 MPa	-
SUS 321HTB	–	321 H	500–720 MPa	-
High heat resisting material				
–	–	–	800–950 MPa	-
SUS 302	–	304 H	500–700 MPa	-
Titanium and superalloys				
–	–	Hastelloy C-4	≤ 690 MPa	-
NCF 80A	–	Nimonic 80A; N07080	≤ 980 MPa	--
–	–	Nimonic 90; N07090	≤ 1200 MPa	--
–	–	Nimonic 105; N13021	≤ 980 MPa	--
NCF 718	–	Inconel 718; N07718	≤ 1230 MPa	--
–	–	Grade 1		--
–	–	Grade 2	345 MPa	--
–	–	Grade 3	441 MPa	--
–	–	Grade 4	539 MPa	--
–	–	Grade 5	895 MPa	--
–	–	Grade 6	686 MPa	--

Note:
The tensile strength (Rm) and hardness (HB) specifications relate in the main to an annealed or non-tempered material state.
The machinability specifications are based on the combined heat value

Surface finish comparison chart

	Surface symbol (DIN 3141)	Roughness grade Nr.	Average surface roughness R_a in µm	Average roughness depth R_z in µm	Roughness (USA) CLA in µin	Roughness (France) R
Roughing	▽	N 12	50	180...220	2000	–
	▽	N 11	25	90...110	1000	–
	▽	N 10	13	46...57	500	R 100
Finishing	▽▽	N 9	6,3	23...32	250	R 40
	▽▽	N 8	3	12...16	125	R 25/R 16
	▽▽	N 7	2	5,90...8,00	63	R 10
Fine finishing	▽▽▽	N 6	0,8	3,00...4,80	32	R 6,3
	▽▽▽	N 5	0,4	1,60...2,80	16	R 3,2/R 2
	▽▽▽	N 4	0,2	1,00...1,80	8	R 1,25
Micro finishing	▽▽▽▽	N 3	0,1	0,80...1,10	4	R 0,8/R 0,5
	▽▽▽▽	N 2	0,05	0,45...0,60	2	–
	▽▽▽▽	N 1	0,025	0,22...0,30	1	–

Hardness comparison chart

tensile strength Rm MPa	Vickers hardness HV	Brinell hardness HB	Rockwell hardness HRC
255	80	76	–
270	85	81	–
285	90	86	–
305	95	90	–
320	100	95	–
335	105	100	–
350	110	105	–
370	115	109	–
385	120	114	–
400	125	119	–
415	130	124	–
430	135	128	–
450	140	133	–
465	145	138	–
480	150	143	–
495	155	147	–
510	160	152	–
530	165	156	–
545	170	162	–
560	175	166	–
575	180	171	–
595	185	176	–
610	190	181	–
625	195	185	–
640	200	190	–
660	205	195	–
675	210	199	–
690	215	204	–
705	220	209	–
720	225	214	–
740	230	219	–
755	235	223	–
770	240	228	20,3
785	245	233	21,3
800	250	238	22,2
820	255	242	23,1
835	260	247	24,0
850	265	252	24,8
865	270	257	25,6
880	275	261	26,4
900	280	266	27,1
915	285	271	27,8
930	290	276	28,5
950	295	280	29,2
965	300	285	29,8
995	310	295	31,0
1030	320	304	32,2
1060	330	314	33,3
1095	340	323	34,4
1125	350	333	35,5
1155	360	342	36,6
1190	370	352	37,7
1220	380	361	38,8
1255	390	371	39,8
1290	400	380	40,8
1320	410	390	41,8
1350	420	399	42,7
1385	430	409	43,6
1420	440	418	44,5
1455	450	428	45,3
1485	460	437	46,1
1520	470	447	46,9
1555	480	456	47,7
1595	490	466	48,4
1630	500	475	49,1
1665	510	485	49,8
1700	520	494	50,5
1740	530	504	51,1
1775	540	513	51,7
1810	550	523	52,3
1845	560	532	53
1880	570	542	53,6
1920	580	551	54,1
1955	590	561	54,7
1995	600	570	55,2
2030	610	580	55,7
2070	620	589	56,3
2105	630	599	56,8
2145	640	608	57,3
2180	650	618	57,8
–	660	–	58,3
–	670	–	58,8
–	680	–	59,2
–	690	–	59,7
–	700	–	60,1
–	720	–	61
–	740	–	61,8
–	760	–	62,5
–	780	–	63,3
–	800	–	64
–	820	–	64,7
–	840	–	65,3
–	860	–	65,9
–	880	–	66,4
–	900	–	67
–	920	–	67,5
–	940	–	68

Grades – Coatings

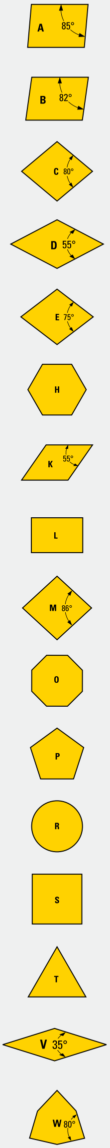
	Grade	Coating	ISO-Group	Milling	Drilling	Solid carbide	Application and Material
Carbide	IN05S	–	N10-N25	•		•	for machining of AL-alloys and non-ferrous materials
	IN10K	–	K10-K25	•			for finish machining of cast iron
			N10-N25	•	•		for finish machining of AL-alloys and non-ferrous materials
	IN15K	–	N15-N30	•			for machining of AL-alloys and non-ferrous materials
PVD coated	IN2004	TiAlN	P10-P20	•			for milling of alloyed steel
			K10-K25	•			for medium machining of gray cast iron-especially CGI
			H05-H15	•			for finish machining of hardened steel at medium up to high cutting speed
	IN2005	TiAlN	P15-P30	•	•	•	for general machining of steel at high cutting speed
			M15-M35	•	•	•	for general machining of stainless steel
			K20-K40	•	•	•	for general machining of cast iron
			S05-S20	•		•	for general milling of heat resistant alloys and titanium also for wet machining
	IN2006	TiAlN	P05-P20	•		•	for finish machining at high cutting speed and low cutting depth
			H05-H20	•		•	for finish machining of hardened steel up to 63 HRC
	IN2010	TiAlN	K10-K30	•	•		for finish machining and drilling of cast iron
	IN2035	TiAlN	P25-P50	•			for high feed machining of steel
			M20-M40	•			for machining of stainless and austenitic steel and heat resistant alloys
			S20-S30	•			mainly for milling of materials of machining group ‚S'
	IN2040	TiAlN	P15-P35	•			for finish machining of unalloyed steel and tempered steel
	IN2504	TiAlN / TiN	P05-P25	•		•	for milling of steel at medium up to high cutting speed
			H05-H25	•		•	for milling of hardened steel at medium up to high cutting speed
	IN2505	TiAlN / TiN	P15-P30	•	•		for semi-finish and rough machining of steel with high strength
			M15-M35	•	•		for general machining of stainless steel
			S05-S20	•	•		for general machining of heat resistant alloys
	IN2510	TiAlN / TiN	K10-K30	•			for general machining of gray cast and non-ferrous metal
	IN2515	TiAlN / TiN	P20-P35	•			for milling of steel with high strength at medium cutting speed
			K30-K50	•			for general machining of gray cast and nodular cast iron
	IN2530	TiAlN / TiN	P20-P40	•	•		tough grade for general machining of steel
			M15-M30	•	•		for general machining of stainless steel
			K20-K40	•			for general machining of cast iron
			S15-S30	•	•		for general machining of heat resistant alloys
	IN2535	TiAlN / TiN	P25-P50	•			for high feed machining of steels
			M20-M40	•			for machining stainless and austenitic steel and heat-resistant alloys
			S20-S30	•			preferably for the milling of materials of the cutting group „S"
	IN2540	TiAlN / TiN	P15-P35	•			for semi-finish and rough machining of unalloyed steel and tempered steel
	IN4005	TiAlN / Al_2O_3	P15-P30	•			for general machining of steel
			M15-M35	•			for general machining of stainless steel
			K20-K40	•			for general machining of cast iron
			S05-S20	•			for general machining of heat resistant alloys and titanium
	IN4005	TiAlN / Al_2O_3	K10-K30	•			for general machining of cast iron
	IN4015	TiAlN / Al_2O_3	P20-P35	•			for milling of steel with high strength at medium cutting speed
			K30-K50	•			for general milling of gray cast and nodular cast iron
	IN4030	TiAlN / Al_2O_3	P20-P40	•			tough grade for general machining of steel
			M15-M30	•			for general machining of stainless and austenitic steel
			S15-S25	•			for general machining of heat resistant alloys
	IN4035	TiAlN / Al_2O_3	P25-P50	•			for high feed machining of steel
			M20-M40	•			for machining of stainless steel, austenitic steel and heat resistant alloys
			S20-S30	•			mainly for milling of materials of machining group ‚S'
	IN4040	TiAlN / Al_2O_3	P15-P30	•			for medium machining of unalloyed and tempered steel

	Grade	Coating	ISO-Group	Milling	Drilling	Solid carbide	Application and Material
CVD coated	IN6505	TiCN / Al_2O_3 / TiN	P10-P25		•		for drilling of steel, used only at peripheral insert of QuadTwist drill
	IN6520	TiCN / Al_2O_3 / TiN	P10-P40		•		for drilling of steel, used only at peripheral insert of QuadDrill+ drill
	IN6535	TiCN / Al_2O_3 / TiN	M20-M45	•			for dry machining of stainless steel and heat resistant alloys at high Vc
			S15-S30	•			primarily for milling of materials of machining group ‚S'
	IN6537	TiCN / Al_2O_3 / TiN	P30-P45	•			for roughing carbon steels and alloyed steels at high Vc
			M30-M45	•			for milling stainless steels at medium cutting speed
			K30-K45	•			for rough milling of gray cast and nodular cast iron
	IN7035	TiCN / Al_2O_3 / TiN	P20-P40	•			for high feed machining of steel
			M20-M35	•			for machining of stainless and austenitic steel and heat resistant alloys
			S15-S30	•			primarily for milling of materials of machining group ‚S'
Cermet	IN0560	TiN	P05-P15	•			for finish machining of steel at medium up to high cutting speed
			M05-M15	•			for finish machining of stainless steel at medium up to high cutting speed
Ceramic	IN70N	Si_3N_4	K10-K20	•			for machining of gray cast iron at extreme high cutting speed
	IN75N	SiAlON	K10-K20	•		•	for machining of cast iron at extreme high cutting speed
	IN76N	SiAlON	S25-S35	•		•	for roughing of heat resistant alloys
CBN	IN80B	–	K05-K15	•			for machining of surface hardened cast materials and chill cast
		–	H05-H15	•			for machining of hardened steel
PCD	IN90D	–	N01-N10	•			for machining of aluminum, non-ferrous materials and graphite

Application	Grade	ISO-Group					
Milling	IN2504	P05-P25				H05-H25	harder
	IN2006	P05-P20				H05-H20	
	IN2004	P10-P20		K10-K20		H05-H15	
	IN4010			K10-K30			
	IN2510			K10-K30			
	IN2005	P15-P30	M15-M35	K20-K40	S05-S20		
	IN2505	P15-P30	M15-M35		S05-S20		
	IN4040	P15-P30					
	IN2540	P15-P35					
	IN4015	P20-P30		K30-K50			
	IN2515	P20-P30		K30-K50			
	IN4030	P20-P40	M15-M30		S15-S25		
	IN2530	P20-P40	M15-M30	K20-K40	S15-S25		
	IN6535		M20-M35		S15-S30		
	IN6537	P30-P45	M30-M45	K30-K45			
	IN7035	P20-P40	M20-M35		S15-S30		
	IN4035	P25-P50	M20-M40		S20-S30		
	IN2035	P25-P50	M20-M40		S20-S30		tougher
Drilling	IN2010			K10-K30			harder
	IN6505	P10-P25					
	IN6520	P10-P40					
	IN2505	P20-P40	M20-M40		S05-S20		
	IN2005	P15-P30	M15-M35	K20-K40	S05-S20		tougher
Solid carbide	IN2504	P05-P25				H05-H25	harder
	IN2006	P05-P20				H05-H20	
	IN2005	P15-P30	M15-M35	K20-K40	S05-S20		tougher

Designation system for ISO indexable inserts

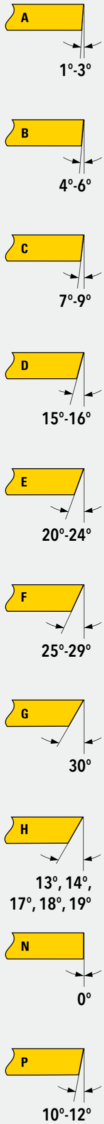
O – Insert shape



U = High feed geometry

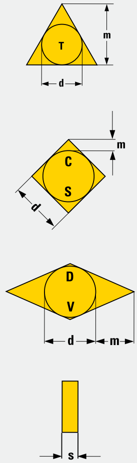
Z = Chip breaking geometry

N – Relief angle



0 = Special angle

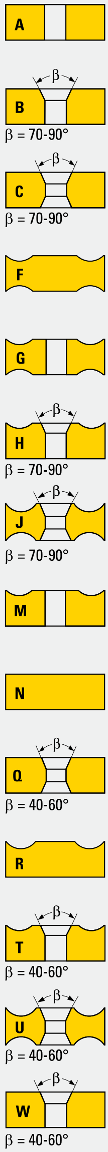
C – Tolerances



	d	m	s
A	±0,025	±0,005	±0,025
C	±0,025	±0,013	±0,025
E	±0,025	±0,025	±0,025
F	±0,013	±0,005	±0,025
G	±0,025	±0,025	±0,05-0,13
H	±0,013	±0,013	±0,025
J[1]	±0,05-0,152	±0,005	±0,025
K[1]	±0,05-0,152	±0,013	±0,025
L[1]	±0,05-0,152	±0,013	±0,025
M	±0,05-0,152	±0,08-0,202	±0,013
N	±0,05-0,152	±0,08-0,202	±0,025
U	±0,05-0,252	±0,13-0,382	±0,05-0,13

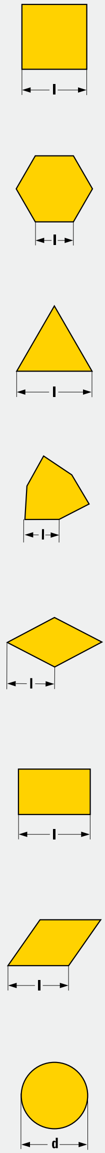
[1]Inserts with ground plane cutting edges
[2]Depending on insert size (see ISO 1832)

U – Insert type



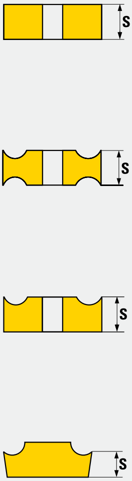
X Special design

05 – Cutting edge length



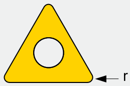
05	AN	T	N	–	HR
Insert thickness	Corner rounding	Cutting edge form	Cutting direction		Internal designation

Insert thickness



01	s = 1,59
T1	s = 1,98
02	s = 2,38
T2	s = 2,78
03	s = 3,18
T3	s = 3,97
04	s = 4,76
05	s = 5,56
06	s = 6,35
07	s = 7,94
09	s= 9,52

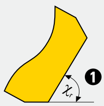
Corner rounding



2	r= 0,2
4	r= 0,4
8	r= 0,8
12	r= 1,2
16	r= 1,6
24	r= 2,4

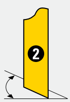
00 for diameters in inch measures converted into mm

M0 for metric diameters



1 Bevel angle χ_r

- A = 45°
- D = 60°
- E = 75°
- F = 85°
- P = 90°
- Z = others



2 Relief angle on wiper

- A = 3°
- B = 5°
- C = 7°
- D = 15°
- E = 20°
- F = 25°
- G = 30°
- N = 0°
- P = 11°
- Z = other relief angles

Cutting edge form

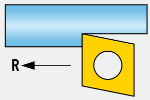
rounded

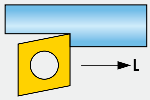
sharp

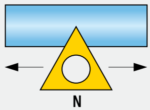
chamfered

chamfered and rounded

Cutting direction







Internal designation

for example

- **P** = polished
- **W** = with wiper finis-hing edge
- **HR** = heat flutes

General formulas

Cutting data calculation

Parameters	
n:	Speed of rotation (min^{-1})
Vc:	Cutting speed
D:	Tool diameter (mm)

$$n = \frac{Vc}{D \times \pi} \times 1000 \qquad Vc = \frac{D \times \pi \times n}{1000}$$

Parameters	
Vf:	Feed rate (mm/min)
fz:	Feed per tooth (mm)
$Z_{(eff)}$:	Number of teeth

$$Vf = n \times fz \times Z \qquad fz = \frac{Vf}{n \times Z}$$

Variable	Unit	Formula
Speed of rotation:	min^{-1}	$n = \frac{V_c \times 1000}{D \times \pi}$
Cutting speed:	m/min	$V_c = \frac{D \times \pi \times n}{1000}$
Feed rate:	mm/min	$v_f = f_z \times Z_{eff} \times n$
Feed per tooth:	mm	$f_z = \frac{v_f}{Z_{eff} \times n}$
Chip removal rate:	cm^3/min	$Q = \frac{a_e \times a_p \times v_f}{1000}$
Average chip thickness:	mm	$h_m = f_z \times \sqrt{a_e/D}$
Specific cutting force:	MPa	$k_c = h_m^{-mc} \times k_{c1.1}$
Spindle power:	kW	$P_c = \frac{a_p \times a_e \times v_f \times k_c}{60 \times 10^6}$
Motor power:	kW	$P_{mot} = \frac{P}{\eta}$

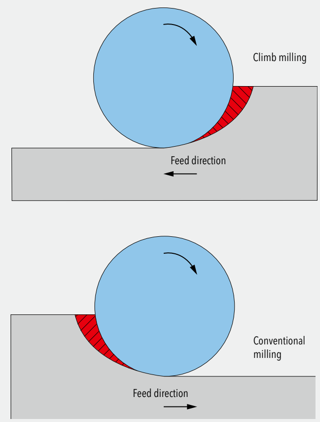
Calculation Example	
Material:	42CrMo4 (1.7225)
Cutter type:	2J1R080R00
Insert:	BOMT130404R
Tool diameter:	80 mm
Effective number of teeth:	9
Depth of cut a_p:	4 mm
Width of cut a_e:	50 mm
Cutting speed V_c:	200 m/min
Feed per tooth f_z:	0,12 mm
Efficiency η:	0,80 (assumed)
Calculation of speed of rotation:	$n = \frac{200 \times 1000}{80 \times \pi} = 796\ min^{-1}$
Calculation of feed rate:	$v_f = 0{,}12 \times 796 \times 9 = 859\ mm/min$
Calculation of chip removal rates:	$Q = \frac{4 \times 50 \times 859}{1000} = 172\ cm^3/min$
Calculation of specific cutting force:	$k_c = 0{,}15^{-0{,}24} \times 1615 = 2546\ MPa$
Calculation of the required spindle power:	$P_c = \frac{4 \times 50 \times 859 \times 2546}{60 \times 10^6} = 7{,}3\ kW$
Calculation of the motor power:	$P_{mot} = \frac{7{,}3}{0{,}8} = 9{,}1\ kW$

Conventional and climb milling / Average chip thickness

Conventional (upcut) and climb (downcut) milling

In milling processes, differentiation is made between conventional (upcut) and climb (downcut) milling. In climb milling, the cutting edges move in the same direction as the feed, in conventional milling on the other hand, the cutting edges move in the opposite direction to the feed. Generally speaking, a longer tool life and better surface finish are achieved with climb milling, as the cutting edge cuts into the material with the greatest chip thickness, without first pressing or scraping until the chip thickness, cutting forces and workpiece material allow the tool to cut, which is the case with conventional milling.

An exception to this rule are hard material crusts. Here, the cutting edge engagement in soft material that occurs in conventional milling has proven to be most beneficial, because an impact into the hard crust would quickly destroy the cutting edge.



Average chip thickness

As the width of cut decreases, the chip tapers in a comma shape. Therefore, from widths of cut of less than 1/3 of the cutter diameter, the feed per tooth must be compensated using the formula shown in Fig. 2. This is often the case for contouring (Fig. 2) or when using side and face mills. For full slot milling (Fig. 1) or for widths of cut greater than 1/3 of the cutter diameter, use of this formula is not necessary.

The ideal average chip thicknesses (hm) feed per tooth (fz) for the Ingersoll inserts are stated in the respective cutting parameter recommendations and each is different depending on the design of the cutting edge. (refer to the „Recommended Cutting Data“). To put it more simply, an insert with a large protective chamfer at the cutting edge can or must be loaded with a higher chip thickness than that for a sharp edge. The use of an insert with a chip thickness that is too low leads to poor chip formation and increased friction or heat build-up, resulting in decreased tool life. Overloading the insert with chip thicknesses that are too high, on the other hand, can cause the cutting edge to chip or break off completely. To achieve the best possible cutting results, the ideal chip thickness to match the insert that is in use is therefore imperative. As well as increased tool life, the use of the above-mentioned formula for contouring also leads to higher productivity.

Fif. 1: Full slot

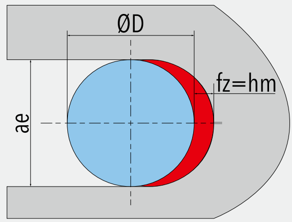
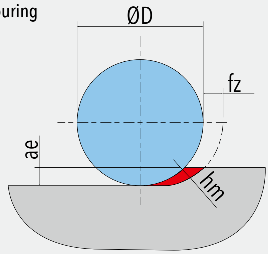


Fig. 2: Contouring

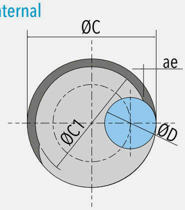
$$fz=hm\sqrt{\frac{D}{ae}}$$



Effective width of cut

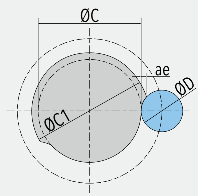
For circular internal and external machining operations, the stock allowance is not equal to the actual, effective width of cut. The following formulae are used for correct calcluation of this. The value aeeff determined in this way is then used to determine the feed per tooth in the formula given in Fig. 2.

Internal



$$ae_{eff}=\frac{C^2\text{-}C1^2}{4\,x\,(C\text{-}D)}$$

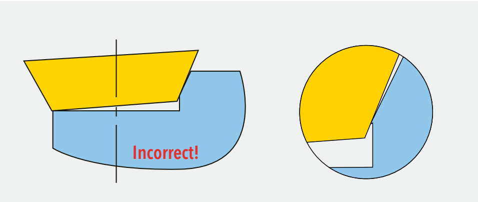
External



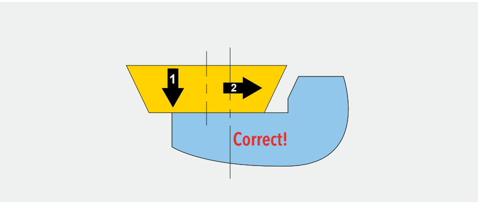
$$ae_{eff}=\frac{C1^2\text{-}C^2}{4\,x\,(C+D)}$$

Correct instalation of the inserts

Tool damage due to canting of the insert

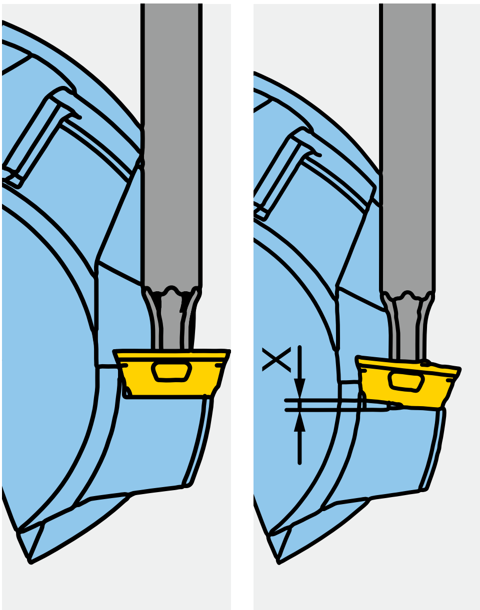


Correct installation of the inserts



Tips

- Occassionally apply copper paste to the thread
- Make sure that the insert pockets are clean
- Correctly insert the inserts into the pocket
- Use a torque wrench



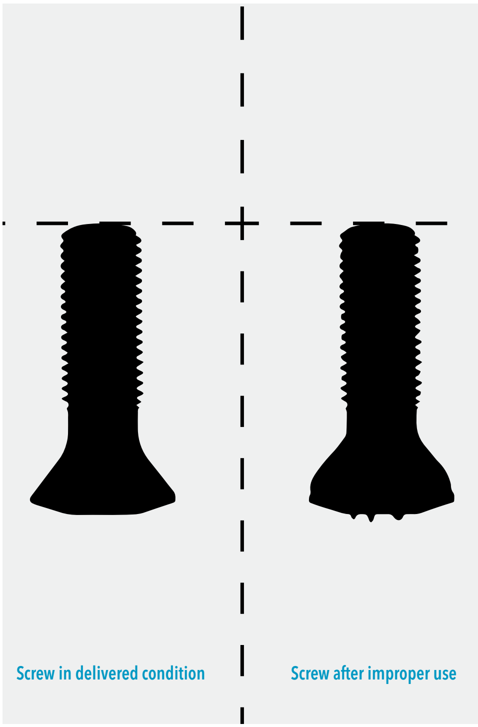
Important for initial installation or new screws:
Tighten – Loosen – Tighten

Correct use of the insert screw

The importance of correct use of the insert screws is highly underestimated. It is however, extremely important for a successful cutting process.

Even if carbide grade, chipbreaker and cutting parameters have been optimally selected, the failure of the insert screw inevitably leads to the loss of the installed insert and often to a tool defect. Moreover, damage to the machine tool and the part being machined cannot be ruled out.

The use of a suitable torque wrench is generally the solution here. While the tightening torque that is subjectively deemed to be correct varies greatly from employee to employee, a torque wrench always objectively provides the correct torque, no matter who is using it. The following diagram shows how serious improper use of the insert screw can be:



In addition to the use of torque wrenches, the use of ceramic or copper paste and changing the insert screws at regular intervals can provide additional process reliability. An overview of the Ingersoll torque wrench range is given below.

Torque wrench range

Designation	Torque	Matching bit	Picture
Preset torque wrench with straight handle, mechanical			
DTN005S	0,5 Nm	A	
DTN011S	1,1 Nm	A	
DTN020S	2,0 Nm	A	
DTN030S	3,0 Nm	A	
Preset torque wrench, mechanical			
DTN045F	4,5 Nm	B	
Adjustable torque wrench with straight handle, mechanical			
DTNV01S	0,4–1,0 Nm	A	
DTNV02S	0,8–5,0 Nm	A	
DTNV00S	2,0–7,0 Nm	A	
Magnetic bit holder without torque function			
V1X105-BH	–	C	
Adapter for torque wrench (bit type A to bit type C)			
DT-008-06	max. 8,0 Nm	A → C	

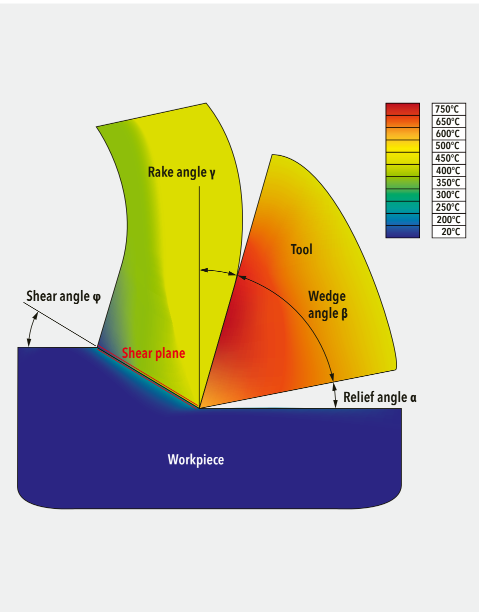
Designation	Wrench size	max. torque	Bit	Picture
DS-T05TB	Torx 5	0,4 Nm	A	
DS-T06TB	Torx 6	0,6 Nm	A	
DS-T07TB	Torx 7	0,9 Nm	A	
DS-T08TB	Torx 8	1,3 Nm	A	
DS-T09TB	Torx 9	2,5 Nm	A	
DS-T10TB	Torx 10	3,8 Nm	A	
DS-T15TB	Torx 15	5,5 Nm	A	
DS-T20TB	Torx 20	8,0 Nm	A	
DS-T25TB	Torx 25	8,0 Nm	A	
DS-TP06TB	Torx Plus 6	0,8 Nm	A	
DS-TP07TB	Torx Plus 7	1,3 Nm	A	
DS-TP08TB	Torx Plus 8	2,0 Nm	A	
DS-TP09TB	Torx Plus 9	3,0 Nm	A	
DS-TP10TB	Torx Plus 10	4,5 Nm	A	
DS-TP15TB	Torx Plus 15	6,6 Nm	A	
DS-H015TB	SW 1,5	0,9 Nm	A	
DS-H02TB	SW 2	1,8 Nm	A	
DS-H025TB	SW 2,5	3,8 Nm	A	
DS-H03TB	SW 3	5,5 Nm	A	
DS-T15B	Torx 15, short	4,5 Nm	B	
DS-T15B1	Torx 15, long	4,5 Nm	B	
TXPLUS06X90-B	Torx Plus 6	0,8 Nm	C	
TXPLUS07X90-B	Torx Plus 7	1,3 Nm	C	
TXPLUS08X90-B	Torx Plus 8	2,0 Nm	C	
TX07X90-B	Torx 7	0,9 Nm	C	
TX08X90-B	Torx 8	1,3 Nm	C	
TX09X90-B	Torx 9	2,5 Nm	C	
TX10X90-B	Torx 10	3,8 Nm	C	
TX15X90-B	Torx 15	5,5 Nm	C	
TX20X90-B	Torx 20	8,0 Nm	C	
TX25X90-B	Torx 25	8,0 Nm	C	

Chip formation model / Initial contact

Chip formation model

Many interrelationships and phenomena of the cutting process can be clearly explained by means of the chip formation model. First, when enough cutting force is present, the tool wedge penetrates the material that is to be machined, i.e. it deforms it elastically to begin with and - after the yield point is reached - plastically. The actual separation or cutting process thereby takes place in the shear plane. After the binding forces have been overcome, the workpiece molecules are pushed past each other under the release of heat.

The direct correlation between the insert resp. tool geometry and the cutting forces and temperatures that occur now also becomes clear. As can be seen in the diagram, a reduction of the rake angle leads to a smaller shear angle and thus to a longer shear plane. Consequently, the required cutting force and the resulting cutting temperature increase. In practice, the increased temperature can result in a shorter tool life and moreover a higher machine output is necessary. Conversely, a more positive rake angle leads to a larger shear angle and to a shorter shear plane. The cutting forces and cutting temperatures decrease.



However, the rake angle cannot be increased at will, as this always leads to a reduction of the wedge angle and thus the stability of the cutting edge. Therefore, the aim here is to reach the best possible compromise between smooth cutting and tool stability dependent on the material that is being cut.

It is also interesting to look at the temperature distribution. Based on the temperature jump it is clear to see that the actual cutting work and the greatest heat input take place in the shear plane. Due to friction on the rake face, the temperature rises even more. Cratering can thus occur at the point at which the cutting edge temperature is the highest. Measures to reduce the friction, such as minimum quantity lubrication, or smooth coatings can help to limit the temperature development and thus have a positive impact on the tool life.

Initial contact

The term initial contact refers to the type of contact that occurs between the tool cutting edge and the workpiece during the cutting process. As shown in Fig. 1, punctiform (e.g. S-contact), linear (e.g. S-T contact) or full surface (see light blue surface) contacts are possible. The type of initial contact can potentially have a great impact on the tool life.

The longer the time between the initial contact of the cutting edge and the engagement of point S, the better the tool life expectancy is, which studies have shown. This can be explained by the quite different stability of the cutting edge at the respective points. In practise, chipping/breakage of the cutting edge very often occurs in the area of the corner S, whereas this is virtually ruled out at point U. Therefore, a U-impact has proven to be beneficial for hard material crusts in particular.

The engagement of the tool (relative position between the workpiece and the tool) as well as the geometry of the tool (axial and radial installation positions) and indexable insert (neutral or positive chip breaker) can have an effect on the initial contact.

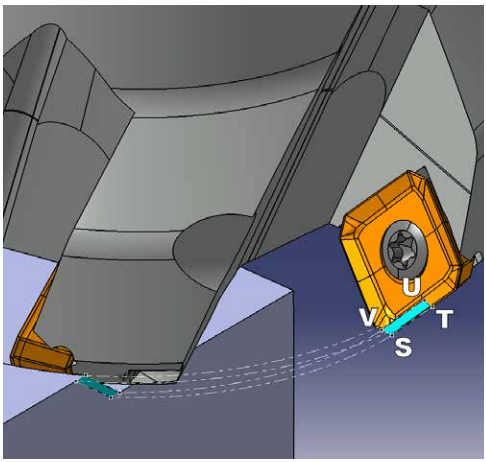


Fig. 1

Consolidated heat value / Heat flow

Consolidated heat value

In daily practice, the metal cutter has to deal with a lot of different materials. Their machinabilty, however, greatly differs depending on their alloy content, physical-chemical properties, homogeneity and heat treatment state.

Materials with good machinability can thereby be machined at high cutting speeds and materials with poor machinability at lower cutting speeds. The crucial factor in the machining process is the heat that is produced, or rather how well this can be dissipated from the cutting tool material. The hot hardness of the cutting tool material and any coating must therefore be taken into consideration.

Knowledge of the consolidated heat value, which is the product of the heat capacity, density and thermal conductivity of the material that is to be machined, can be helpful in assessing the machinability, whereby the thermal conductivity has the greatest influence on the consolidated heat value (and thus also the machinability). This also explains, e.g. the very different machinability of aluminum and titanium alloys, as titanium has a 20 times poorer thermal conductivity than aluminum. Therefore, for titanium, the heat input into the cutting tool material can only be limited by the consistent use of coolants and by selecting moderate cutting speeds. However, the individual heat input is also important for all other materials for the definition of carbide grade, coating, cutting parameters and coolant.

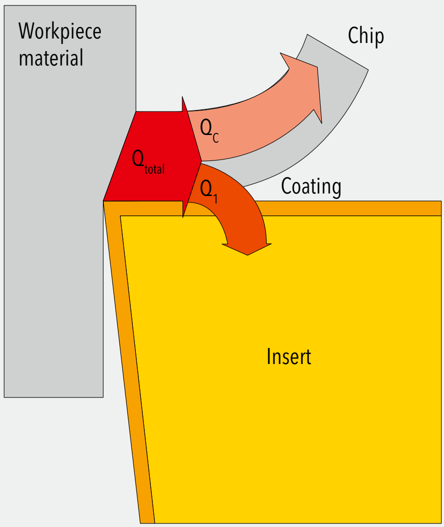
Heat flow

As already illustrated in the chip formation model, the actual cutting work takes place in the shear plane area. Heat is thus produced, which has to be dissipated from the cutting tool material as effectively as possible. Ultimately, this is the sole purpose of all of the usual methods.

Due to the oil content, the water-oil emulsion of a conventional cooling lubricant reduces the friction on the one hand (and thus additional heat input) and on the other hand, the water content also contributes to the cooling. A reduction of friction is the background of minimum quantity lubrication (MQL). In cryogenic cutting processes, the tool cutting edge is cooled down using -196°C cold nitrogen. This results in considerably longer tool life, especially for difficult to machine materials, such or titanium or nickel-based alloys. Cutting tool material coatings generally have a higher hot hardness that the cutting tool material substrate.

Apart from that, they also minimize friction and act to a certain degree as a thermal insulation layer. However, the selection of the cutting parameters can also be used to influence the heat flow.

Large volume chips can dissipate more heat from the cutting edge; lower cutting speeds reduce the machining temperatures that arise. The use of moderate cutting speeds in conjunction with a higher feed can thus often increase the tool's serviceable life.



Chips

Types of chips

The types of chips that are produced during the machining process mainly depend on the workpiece material that is being machined, the geometry of the tool and the machining parameters. There are three main types of chip, whereby the transitions may be continuous:

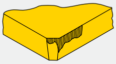
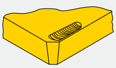
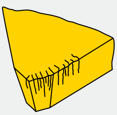
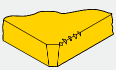
Chip type	Characteristics	Chip formation	Conditions
Continuous chip	• continuous chip • differing texture • bottom side always smooth	• continuous flow-off of the workpiece material • chip elements are not separated in the shear zone, but are continuously reshaped	• tough workpiece material under favorable cutting conditions (high cutting speed, positive rake angle)
Continuous chip with built-up edge	• individual, chip segments can be seen • very jagged surface	• chip lamellae are only minimally re-shaped in the shear plane, separated from each other, but are then re-welded to each other	• faulty form of the continuous chip • causes of the fault: flaws in the workpiece material, vibrations, rake angle too small, large depth of cut, low cutting speed
Discontinuous chip (segmental chip)	• single, non-connected chip elements • rough surface due to fracture microstructure (advancing crack)	• brittle materials fracture after just minimal deformation in the shear zone (e.g. cast iron, chilled casting, cast bronze, brass) • in extremely brittle materials, complete disintegration of the chip lamellae	• material with low plastic behavior (low cutting speed, negative rake angle)

Chip form

The chip form is influenced by many variables. In turning operations in particular, the chip form is of great importance and can, e.g. be considerably influenced by the selection of a suitable chip breaker. Chip forms that combine a smallest possible volume with a low risk of injury are considered to be favorable. The chip forms can be divided up into the following categories:

Category	Chip form and volume ratio R		Assessment
1	Ribbon chip R ≥ 90		Unfavorable
2	Snarled chip R ≥ 90		Unfavorable
3	Helical or screw chips R ≈ 60		Acceptable
4	Helical or screw chip segments R ≈ 25		Favorable
5	Spiral chip R ≈ 10		Favorable
6	Spiral chip segment R ≈ 5		Favorable
7	Discontinuous chip or chip fragments R ≈ 3; Partially welded together		Acceptable

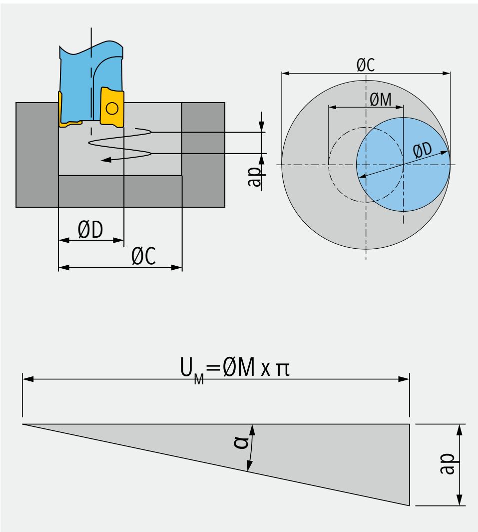
Measures for milling problems

Type of wear	Description	Cause	Solutions
Built-up cutting edge 	Build-up of material welded to the cutting edge occurs when the chip doesn't flow off properly as due to the cutting temperature being too low.	• Cutting speed is too low • Rake angle is too small • Cutter material has too little wear resistance • improper cooling	• Increase the cutting speed • Increase the rake angle • Use of coatings • Use of an efficient coolant
Flank wear 	Extreme flank wear reduces the relief angle and impairs the surface quality	• Cutting speed is too high • Cutter material has too little wear resistance • Incorrect feed rate (feed rate too low)	• Reduce the cutting speed • Choose a cutting tool material with a higher wear resistance, choose a coated quality grade • Adjust the feed rate to the proper ratio to the cutting speed and depth of cut (increase the feed rate)
Cratering 	Cratering weakens the cutting edge and changes the cutting edge geometry.	• Cutting speed is too high • Incorrect feed rate • Rake angle is too small • Wrong supply of coolant • Cutter material has too little wear resistance	• Reduce the cutting speed • Set the feed rate, cutting speed and depth of cut to the correct ratio to each other • Use a positive rake angle • Increase the coolant quantity and/or pressure • Use coatings
Cutting edge deformation 	High mechanical stresses and high cutting temperatures can lead to plastic deformation of the cutting edges.	• The work temperature is too high resulting in softening of the base material due to too high cutting speeds and feed rates and hard workpiece materials • Coating damage	• Reduce the cutting speed, use more wear-resistant cutting material grades, reduce the chip cross section (especially feed), use of suitable edge rounding, reduce the bevel angle, provide cooling • Change the insert early enough
Comb cracks 	Cracks increasingly develop at right angles to the cutting edge caused by variations in thermal stress in the interrupted cut. Risk of breakage.	• Change in chip thickness • Fluctuating cooling lubricant supply • Alternating cooling and heating with accompanying change in stress (for interrupted cut)	• Select uniform contact conditions • Consistent supply of an adequate quantity of coolant, or in the case of carbides and ceramic cutting tool materials in particular, avoid cooling • Select a tougher cutting tool material and better resistance to changes in temperature • Supply an adequate cooling lubricant quantity, or for carbides, avoid it completely
Chipping of the cutting edge 	Small material break-outs at the cutting edge, main in connection with flank wear.	• Oxidation • Abrasion	• Choose a suitable coating • reduce the cutting speed; exception: when machining heat-resistant material with ceramic cutting tool materials, increase the cutting speed

Circular interpolation milling

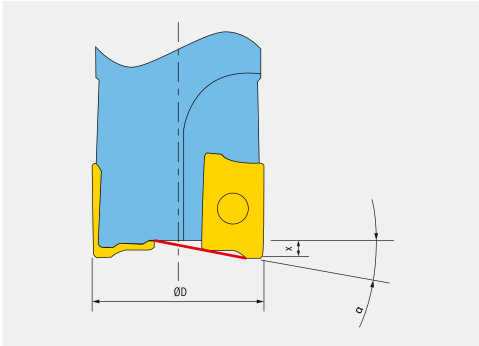
Milling tools with a known ramp angle α can also be used for circular interpolation milling. Provided that it can be produced with the respective tool, the hole diameter C is unimportant. Even an annular groove can be produced in this way. However, the hole diameter must not go below the minimum hole diameter Cmin, otherwise the tool cannot cut itself free and would get stuck. The maximum and minimum possible hole diameter, as well as the areas for even or uneven hole grounds are individually different depending on the indexable insert that is used. Please refer to the appropriate cutting parameter recommendations for details. The depth of cut ap per revolution can be calculated by means of the circumference of the tool center path using the following formula: $ap=(C-D) \times \pi \times \tan\alpha$

Please note that here both, the ramp angle α and the calculated depth of cut per revolution, are maximum values, which can only be achieved under optimum conditions and with materials that have very good machinability. In general, only roughly half of the depth of cut that can be achieved in full slot milling should be undertaken.



Ramping Capability

The ramping capability of a milling tool is mainly determined by its diameter and the projection of the insert beyond the tool body. Therefore as a rule, the smaller the diameter of a tool is, the better its ramping capability will be. In the same way, a large projection dimension X contributes to a good ramping capability. Exceptions may arise if the indexable insert does not have a relief angle or if the angle is too small.



High feed milling

Although already described in the relevant technical literature at the end of the 1930's, the high feed milling principle only became popular more recently, as suitable machinery and tools became widely available. The objective of high feed milling is to increase productivity, which can be achieved with tools that have very small lead angles of $\varkappa < 20°$. As shown in the following diagram, while maintaining the feed per tooth fz and the same depth of cut ap, a change in the slenderness ratio b/h occurs, which is directly influenced by the lead angle that is used.

The productivity advantage of high feed milling occurs when the usual chip thickness of h = fz during shoulder milling with $\varkappa=90°$ (shown at the far left in Fig. 1) is transferred to a tool with a very small lead angle (e.g. $\varkappa=15°$, shown at the far right in Fig. 1). To achieve the same chip thickness here, the feed per tooth fz must or can be considerably increased - in the above case by a factor of 3.9.

In this context it must be noted that due to the small lead angle, more axial cutting forces occur. As longer tools are mostly used for the deep cavities that are often encountered in die and mould making, increasing axial forces along with simultaneous minimization of the radial forces is, however, actually useful. The paring entry of the cutting edge is another benefit of high feed milling. While during shoulder milling, the cutting edge penetrates the workpiece material rather abruptly, complete engagement of the high feed cutting edge takes much longer. A greatly extended tool life often results.

As shown in Fig. 2, round inserts are also suitable for high feed milling, provided that the depth of cut ap does not exceed as certain value. It must however be noted, that in the case of round inserts, the lead angle is not constant. Rather it begins at 0° and theoretically increases up to 90°. Therefore, in practice it has proven best to assume an average lead angle. As per definition, the term high feed millling can be used from a lead angle of

$\varkappa \leq 20°$, the maximum suitable depth of cut for high feed milling can be calculated with the help of the following formula, whereby D stands for the diameter of the round insert.

Exact calculation: $ap = D \times \sin(20°)^2$
Approximation: $ap = D \times 0{,}1$

To now determine the feed per tooth that has to be programmed, the diameter of the round insert D and the desired chip thickness h are inserted into the following equation.

$$fz = -(D/2) \times \tan(20°) + \sqrt{[D/2 \times \tan(20°)]^2 + h \times D/\cos(20°)}$$

The use of round inserts for high feed milling does, however, have some disadvantages too. On the one hand due to the design, only a relatively low number of teeth can be accommodated by the tool body, which leads to a lower total feed rate. On the other hand, milling into the 90° shoulders cannot be done at a high feed rate; in the corners or shoulders of the parts being machined, stock equal to at least the radius of the round insert is left unmachined. This remaining stock then has to be removed later with a different tool.

The use of tool systems that have been specially developed for high feed milling is therefore recommended in many cases. An exemplary comparison of the high feed indexable insert UOMT0602TR with a 6 mm round insert illustrates this (see Fig. 3.). Thanks to the UOMT0602TR's low space requirement, a lot more teeth can be accomodated with the same tool diameter. Due to the lead angle of more than 90°, milling into the shoulder can be done and less stock is left for later removal.

Insert/Geometry	Programming radius	max. depth of cut
UOMT06…	R1	0,5
PEMT05…	R2,5	1
DPM324R004 / DPM324R126	R4,5	3
DPM436R10…	–	3,5
UNLU04…	R0,9	0,5
UNLU06.	R1,6–R1,9	1
UNLU09…	R2,5	1,5
UNLU11…	R3	2
UNHU04…	R1,2	0,5
UNKT05…	R0,8–R1	0,5
UNHU06…	R2	1
UNHU09…	R3,2	1,5
UNHU11…	R4	2

Insert/Geometry	Programming radius	max. depth of cut
UNHU14…	R5	3
SDXS04…	R1	0,5
SDXS09…	R2,5	1,5
SDES13… / SDMS13… / SDXS13…	R3	2,2
SDXS16…	R4,2	2,5
SDES19… / SDMS19… / SDXS19…	R4,5–R5,5	3,7
WCNW06… / WCNT06…	R2	0,9
LNXF09…	R3,4	1,5
TNXN12 …	R4,5	2,5
ChipSurfer 45A…	R2–R3	0,6–1,5
ChipSurfer 47A…	R1,6–R3,6	0,4–1,2
Plendur INCER…Z3	R0,42–R2,5	0,35–1,0
Plendur INCOO…Z3	R0,28–R1,9	0,2–1,3

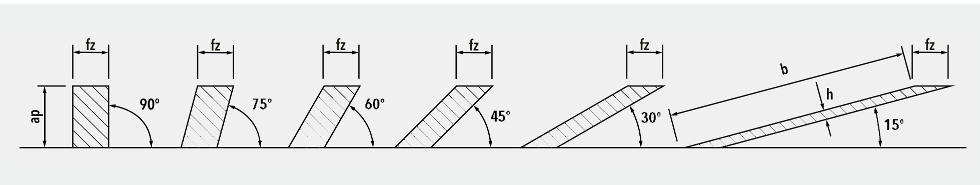


Fig. 1: Change in the chip thickness dependent on the lead angle $\varkappa$ at a constant tooth feed fz

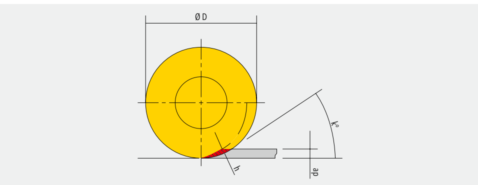


Fig. 2: Round indexable inserts are also generally suitable for high feed milling.

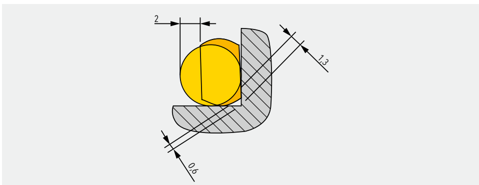


Fig. 3: Comparison of a 6 mm round insert with the high feed insert UOMT0602TR

Characteristics of GOLDSFEED 13 & 19 mm for 90° shoulder milling

Due to the special wide angle geometry of the tool and of the insert, maximum infeeds must be taken into consideration when machining 90° shoulders.

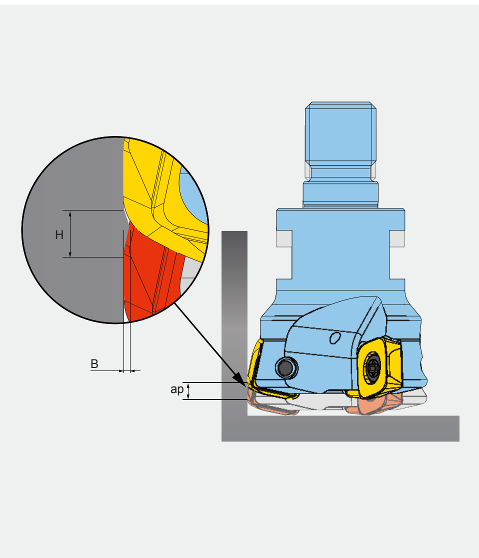
Contrary to conventional milling on free flat surfaces and contours, some stock remains when milling 90° shoulders. Due to this remaining stock, an undercut is produced at the unused cutting edge, which, depending on the material and insert geometry, in the worst case can cause the insert to break. The tables and figures below should be used to avoid this problem.

Consideration of the undercut in relation to D.O.C. for machining 90° shoulders with GoldSFeed SD_S13

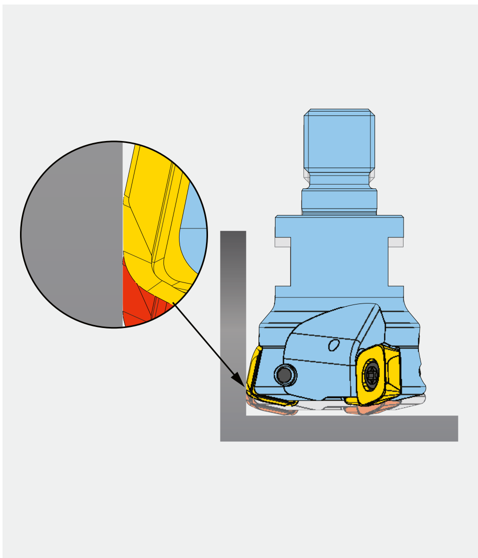
GoldSFeed insert	ap_{max} with undercut at 90° shoulder	Stock dimensions in the undercut (height x width)	ap_{max} without undercut at 90° shoulder
SDES130515N	2	1,0 x 0,076	0,5
SDES1305MDR	2	1,0 x 0,076	0,95
SDES1305MPR / SDXS1305MPR-MR	2,2	0,74 x 0,05	1,45
SDMS130515R-PH / SDXS130515R-PH	2	1,0 x 0,076	0,5
SDMS1305MDR-PH / SDXS1305MDR-PH	2	1,0 x 0,076	0,95
SDES130515N-001	2	1,0 x 0,076	0,5
SDES1305MDR-001	2	1,0 x 0,076	0,95
SDES1305MPR-001	2,2	0,74 x 0,05	1,45

Consideration of the undercut in relation to D.O.C. for machining 90° shoulders with GoldSFeed SD_S19

GoldSFeed insert	ap_{max} with undercut at 90° shoulder	Stock dimensions in the undercut (height x width)	ap_{max} without undercut at 90° shoulder
SDES190620N	3	1,88 x 0,28	1,1
SDES1906MDR	3	1,88 x 0,28	1,1
SDES1906MPR / SDXS1906MPR-MR	3,7	2,16 x 0,23	1,5
SDMS190620R-PH	3	1,88 x 0,28	1,1
SDMS1906MDR-PH	3	1,88 x 0,28	1,1
SDES190620N-001	3	1,88 x 0,28	1,1
SDES1906MPR-001	3,7	2,16 x 0,23	1,5



Infeed sequence with remaining undercut


Infeed sequence without remaining undercut

Characteristics of GOLDQUAD^{XXX} 13 & 19 mm for 90° shoulder milling

Due to the special wide angle geometry of the tool and of the insert, maximum infeeds must be taken into consideration when machining 90° shoulders.

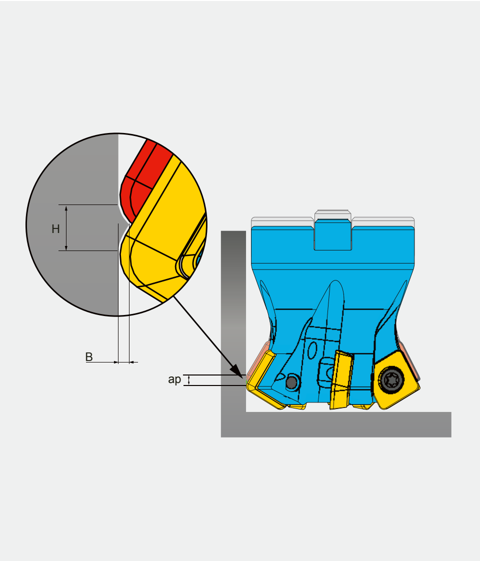
Contrary to conventional milling on free flat surfaces and contours, some stock remains when milling 90° shoulders. Due to this remaining stock, an undercut is produced at the unused cutting edge, which, depending on the material and insert geometry, in the worst case can cause the insert to break. The tables and figures below should be used to avoid this problem.

Consideration of the undercut in relation to D.O.C. for machining 90° shoulders with GoldQuad^{XXX} SD_S13

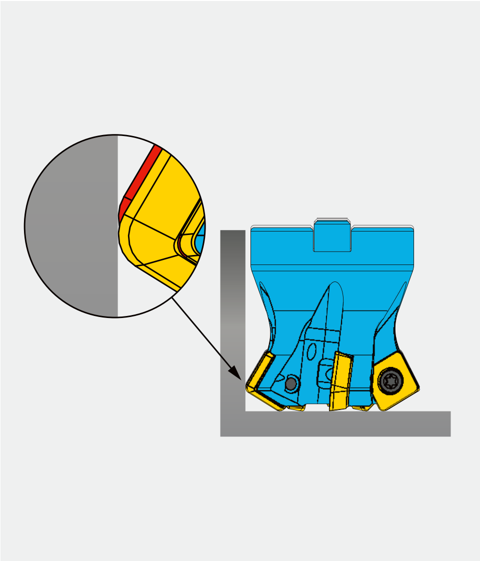
GoldQuad^{XXX} insert	ap_{max} with undercut at 90° shoulder	Stock dimensions in the undercut (height x width)	ap_{max} without undercut at 90° shoulder
SDES1305MPR / SDXS1305MPR-MR	4,9	4,85 x 1,27	0,5
SDES130516N-PF	4,9	4,9 x 1,52	0,5
SDMS130516R-PP	4,9	4,9 x 1,52	0,5
SDES130516N-PF1	4,9	4,9 x 1,52	0,5
SDES1305MPR-001	4,9	4,9 x 1,52	0,5

Consideration of the undercut in relation to D.O.C. for machining 90° shoulders with GoldQuad^{XXX} SD_S19

GoldQuad^{XXX} insert	ap_{max} with undercut at 90° shoulder	Stock dimensions in the undercut (height x width)	ap_{max} without undercut at 90° shoulder
SDES190620N	7,8	7,44 x 2,64	0,5
SDES1906MPR / SDXS1906MPR-MR	7,8	7,44 x 2,03	0,5
SDES1906ZPR-PF	7,8	7,44 x 2,64	0,5
SDMS190620R-PH	7,8	7,44 x 2,64	0,5
SDMS1906ZPR-PP	7,8	6,45 x 2,48	1,3
SDES190620N-001	7,8	7,44 x 2,64	0,5
SDES1906MPR-001	7,8	7,44 x 2,03	0,5



Infeed sequence with remaining undercut

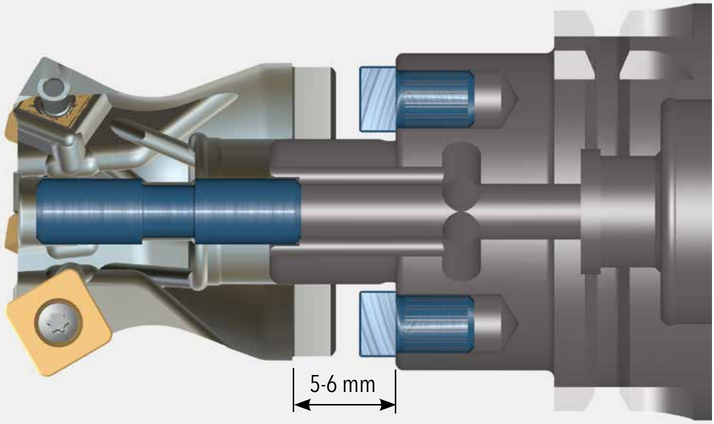


Infeed sequence without remaining undercut

Mounting instruction for shell mill Ø 50 mm

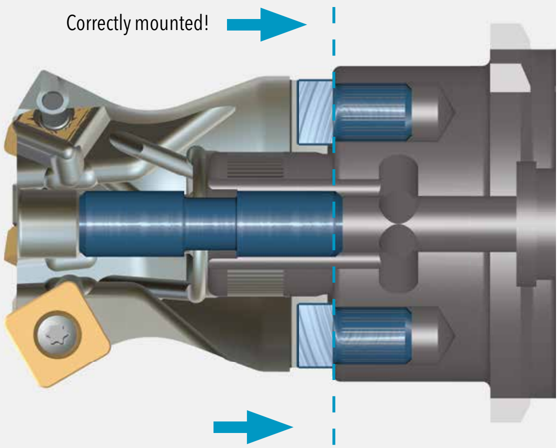
Torque at the mounting screw:
20 Nm

Correct position of the cutter mounting screw prior to mounting



Mounting instruction

1. Screw in the cutter mounting screw/differential screw backwards with the fine thread into cutter body until it reaches the limit stop.
2. Push the cutter onto the cutter arbor; a gap of approx. 5 - 6 mm results.
3. Tighten the cutter mounting screw/differential screw at the front side with a torque of 20 Nm.



Programming tips

- Program twice the factor for the feed rate to compensate for chip thinning
- As a general rule, select 60% of the tool diameter to determine the width of cut
- *If possible, choose deeper depths of cut to improve the chip removal rate and achieve the highest productivity*
- *Ensure the optimal direction, flow and pressure of coolant and compressed air*
- *Use the HSM (High Speed Machining) functions of the CAM system for programming*

Power output calculation / Drive Power

Power output calculation

Due to many influencing variables (e.g. insert and tool geometry, etc.) it is generally impractical to calculate the exact machine power output. It therefore makes more sense to just roughly calculate the required machine power output. The following table contains values for the specific stock removal rate Q_{sp}, which illustrates how many cubic centimeters of a certain material can usually be cut per kilowatt and minute.

A high Qsp value thus represents materials with good machinability, or which make a high stock removal at low machine power outputs possible. The lower the value, the greater the power output of the machine must be and the more difficult the material is to machine. Please note that the calculated vales of Ptheo are only a rough rule of thumb!

Specific stock removal rate Qsp [cm³/kw x min]		
P	Steel	15–30
M	Stainless steel	~20
K	Cast iron	25–37
N	Non ferrous metals	~52
S	Superalloys	11,5–15
H	Hard materials	~11,5
O	Plastic & graphite	100–250

For milling: $P_{theo} = \frac{a_p \times a_e \times v_f}{Q_{sp} \times 1000}$

For drilling: $P_{theo} = \frac{D^2 \times \pi/4 \times v_f}{Q_{sp} \times 1000}$

For turning: $P_{theo} = \frac{a_p \times f \times v_f}{Q_{sp} \times 1000}$

Drive Power

Approximate required drive power determined with the help of the specific stock removal rate Q_{sp}!

$$P_{mot} = \frac{Q\ (cm^3/min)}{Q_{sp}\ (cm^3/kW \times min)}$$

$$P_{mot} = \frac{a_e \times a_p \times v_f}{1000 \times Q_{sp}}$$

Table for the Qsp of different materials dependent on Q_{sp} (cm³ / kW x min)

Machined material	fz = 0,1 mm	fz = 0,25 mm	fz = 0,6 mm
35 Ni Cr Mo 16	15 - 17	18 - 20	22 - 24
38 CR Al Mo 7	16 - 18	19 - 21	23 - 25
42 Cr Mo 4	16,5 - 18,5	19,5 - 21,5	23,5 - 25,5
X 5 Cr Ni Mo 18 10	17,5 - 19,5	20,5 - 22,5	24,5 - 26,5
50 Cr V 4	17,5 - 19,5	20,5 - 22,5	24,5 - 27
16 Mn Cr 5	18 - 20	21,5 - 23,5	25,5 - 28
C 45 - C 60	19,5 - 21,5	23,5 - 25,5	28 - 31
Ti6Al4V	20 - 22	26 - 28	31 - 33
GGG	25 - 27,5	30 - 33	36 - 39
GG 26	28 - 31	33,5 - 37	39,5 - 43
GTW - GTS	32,5 - 36	38,5 - 42	45,5 - 49
MS 80	39 - 43	58 - 62	69 - 73
Al - Si	69 - 72	82 - 85	-
Al - Mg	83 - 85	100 - 105	-

The higher value applies for normal positive cutting edge geometry, without a dull cutting edge.

Example	
Conditions:	
Material:	C45
Tool:	ON5H080R00 (Ø80 mm; Z=10)
Vc:	150 m/min
fz:	0,25 mm
n =	600 U/min
Vf =	1500 mm/min
ae =	60 mm
ap =	3 mm
Machine power:	22 kW

Calculation P_{mot}

Q=270 cm³/min

Q_{sp}=24,5 cm³/kW x min

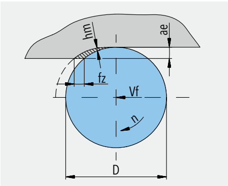
$$P_{mot} = \frac{Q\ (cm^3/min)}{Qsp\ (cm^3/kW \times min)}$$

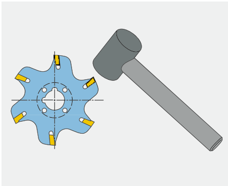
P_{mot}=11 kW

Tips

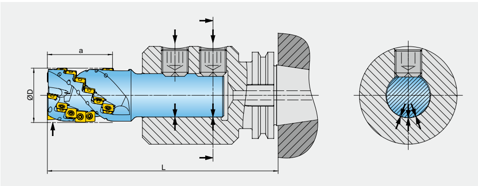
Mounting the cutting inserts

Insert the cutting inserts into the pockets and gently tap them into position with the plastic hammer until they reach the fixed stop. Before inserting the cutting inserts, the pockets must be cleaned and all chip residues removed.

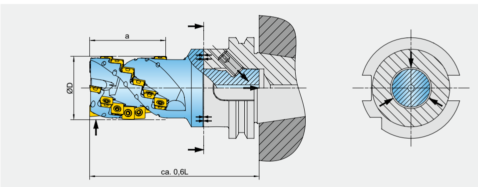




Weldon versus InnoFit

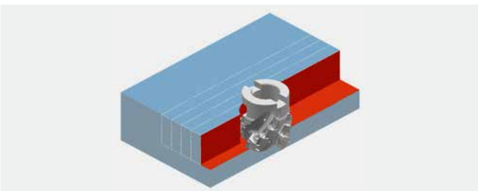


Weldon vibrations



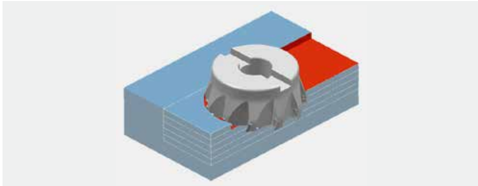
InnoFit 40% shorter

Shoulder mill versus extended flute end mill



Advantages:

- Low engagement
- Small cutting arc
- Higher RPM
- More power
- For small machines, highQ [cm³/min]



Advantages:

- Always the first choice for small depths

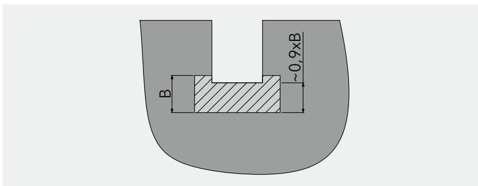
Application example T-Clamp slot mill

Depending on the depth of cut ae, the recommended feed rate must be corrected in accordance with the following table:

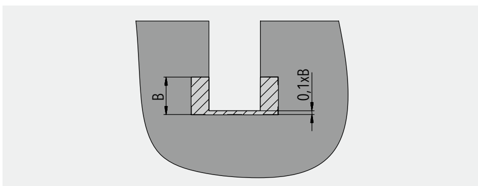
$\frac{a_e}{D}$	$\frac{1}{6}$	$\frac{1}{8}$	$\frac{1}{10}$	$\frac{1}{20}$
fz	15%	30%	45%	100%

Cutting data	
D:	Cutter diameter (mm)
n:	Speed of rotation (min^{-1})
V_f:	Feed rate (mm/min)
V_c	Cutting speed (m/min)
a_e	Radial depth of cut (mm)
h_m	Average chip thickness (mm)
f_z	Feed per tooth (mm)
Z_{eff}:	Number of effective cutting edges

Application example HiPosQuad T-slot mill



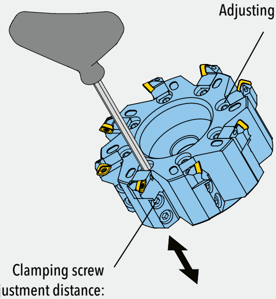
Under normal machining conditions: Supply plenty of coolant or compressed air to ensure good chip evacuation!



For low-power machines: Supply plenty of coolant or compressed air to ensure good chip evacuation!

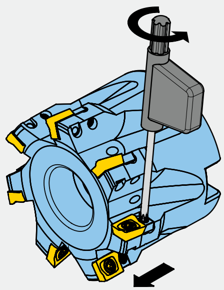
Operating instructions

Cartridge mill 4W3A...	Mounting the cartridge mill



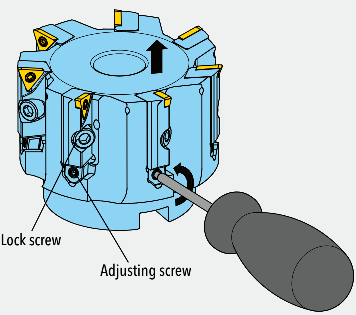
- Tighten the adjusting screws
- Insert the cartridges and press them against the axial stop
- Tighten the clamping screws
- Measure the axial run out of the cutter using the cutting edges and determine the highest point (cartridge)
- Loosen the clamping screws of the cartridge that is to be adjusted then re-tighten them slightly
- Based on the highest measured value, adjust all of the cartridges with the adjusting screw
- Fully tighten the clamping screws

Shoulder mill 2J5P...	Set the axial runout



- Screw in the differential screw clockwise to move down the adjusting wedge
- Mount the inserts and tighten the insert screws with a torque wrench (1.1 Nm)
- Turn the differential screw counter-clockwise to move the adjustment wedge into position
- Measure the axial runout of the tool and find the cutting edge with the highest measurement
- Turn the differential screw counter-clockwise to adjust the cutting edge to the nominal dimension
- Tighten the insert screws again with the torque wrench
- The adjustment range should not exceed 70 µm

Finish mill 4W5D...	Mounting and Adjustment



- Turn the adjusting screw clockwise to screw in the adjusting wedge until it reaches the stop
- Slightly loosen the adjusting wedge again by turning the adjusting screw approx. half a turn counter-clockwise
- Insert the cartridge and push it against the adjusting wedge
- Slightly tighten the insert screw
- Mount the inserts and tighten the insert clamping screw with 1.1 Nm
- Measure the axial runout of the tool and find the cutting edge with the highest measurement
- Turn the adjusting screw counter-clockwise to adjust the cutting edge to the nominal dimension
- Tighten the screws at the cartridge with 9 Nm
- Check the axial runout again, and if necessary, slightly adjust in the µ-range with the adjusting screws

Circular thread milling / Finish milling

Process reliable machining

Controlled chip evacuation:

- Whirling starts at the bottom of the bore
- Short comma-shaped chips
- Climb milling produces smooth cut

High repeatability:

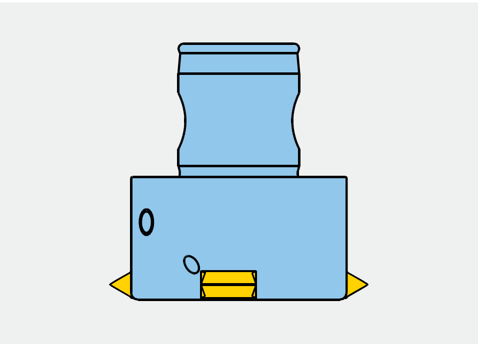
- Due the use of inserts, the tolerance qualities are always the same

Highest quality

- Surface quality of the flanks better than µm Rt 1
- Perfect geometric thread form

Low tool costs one tool for:

- Multiple diameters
- Multiple pitches
- Right and left hand threads
- Metric and imperial threads
- Custom threads, which lie within the specified diameter range
- Low storage costs
- No regrinding
- All thread-cutting inserts can be used continuously for a certain pitch range



All thread cutters have internal coolant supply

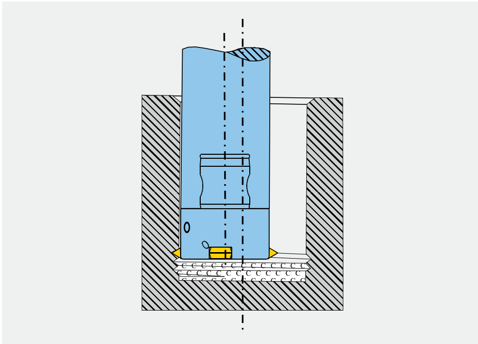
Approximate values for circular milling of internal threads

Material	Cutting speed (m/min)		
	Rm [MPa]	dry	wet
Steel	400–700	260	320
	500–800	230	300
	800–1100	200	280
	1100–1400	60	150
Cast steel	400-700	150	280

Thread pitch	fu (thread outer-Ø*)
1 mm	0,40 mm
2 mm	0,35 mm
3 mm	0,30 mm
4 mm	0,25 mm
5 mm	0,20 mm
6 mm	0,15 mm
7 mm	0,12 mm
8 mm	0,10 mm

* The feed rate of the cutter center point is calcuated using the following formula:

$$V_{fm} = \frac{(\text{Thread outer-Ø} - D) \times n \times f_z \times z}{\text{Thread outer-Ø}}$$



Attention!

These approximate values must be adjusted to meet the appropriate conditions! For very deep threads it is advisable to cut the thread in 2 passes.

In this chapter we would like to explain finish milling using Ingersoll MicroMill wiper insert tools.

Finish milling principles

Whenever possible, the wiper insert milling method should be used for finish milling. With this method, all cutting edges involved in the cut have a secondary cutting edge, a so-called wiper, that runs parallel to the surface that is to be milled, and has the length of the feed that takes place during one spindle revolution plus overlapping.

The effect of the wiper insert method is that the main peripheral cutting edges remove the stock and the wiper edges smooth the surface of the workpiece. In the course of one spindle revolution, the cutting edge that protrudes the most thereby removes the unevenness and roughness left by the preceding cutting edge. A level and smooth surface of a high quality is thus produced.

After a certain length of time, the highest cutting edge wears down more at the wiper than the other cutting edges. At that

Finish milling

point, other cutting edges come into play, until the milling tool has evened out to a certain axial runout plane.

In this state, the tool produces a very good surface quality over a long service life of the tool. In theory, just one single wiper edge would be enough to produce a finished surface.

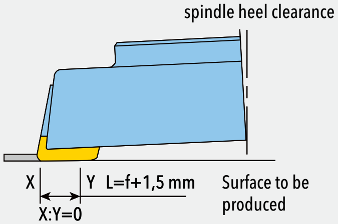
Experience, however, has shown that the tool life is too short if a single trailing edge is used. Therefore, if the stability of the workpiece permits it, all of the cutting edges involved in the cut should be designed as wiper edges.

For particularly unstable workpieces, however, the face of every 2nd cutting edge can e.g. be ground down, or for insert tools, can be set back. This reduces the axial pressure of the cutting tool. The smoothness of the milled surface can thus be improved and vibrations avoided to the greatest possible extent.

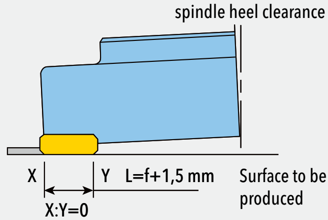
The following compares the features of a properly designed, regrindable finishing tool and of the insert tool for finish milling.

Optimal length and position of the wiper edge

Regrindable tool



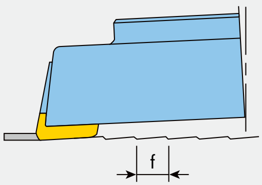
Indexable insert tool



The wiper edge is positioned parallel to the surface that is to be machined and is approx. 1.5 mm longer than the feed per revolution (f).

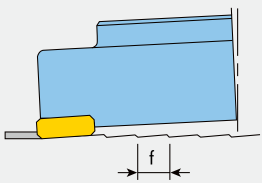
Sources of flaws

Regrindable tool



Flaw attributable to incorrect grinding of the cutter head.

Indexable insert tool

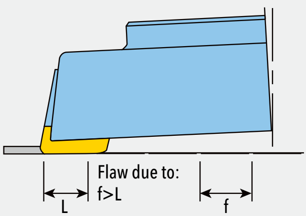


Flaw attributable to incorrect installation of the insert.

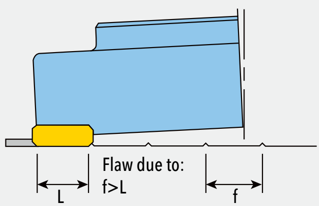
The wiper is not parallel to the surface that is to be milled.

Sources of flaws

Regrindable tool



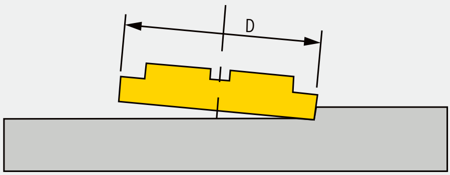
Indexable insert tool

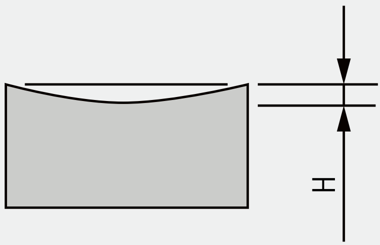


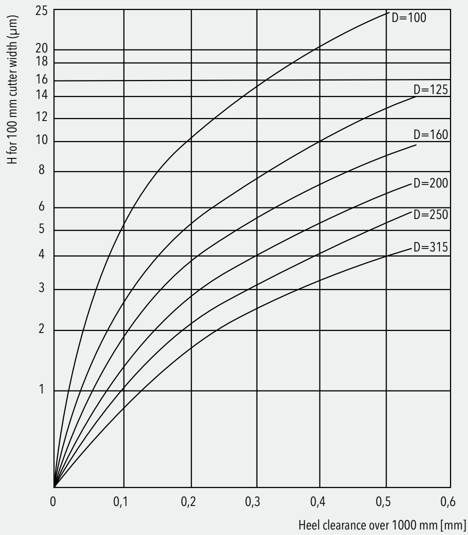
The length of the wiper insert is shorter than the feed per revolution (f).

Finish milling

Heel clearance at the tool spindle prevents the tool from recutting

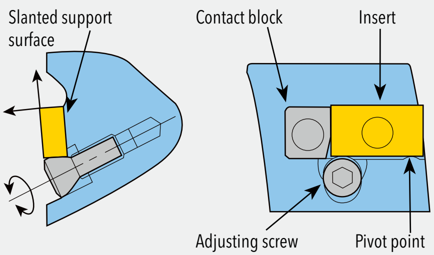






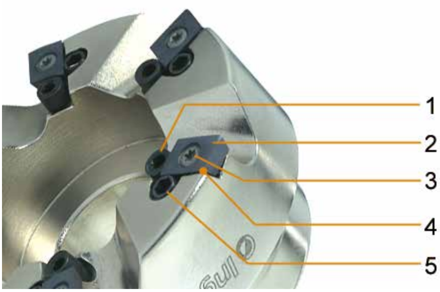
Occuring concave form deviation H

Function description



- The insert is moved on its insert seat tangentially along the slanted rest by screwing the adjusting screw out or in.
- At the same time, a height adjustment of the insert and thus a change in the position of the wiper takes place.
- The position of the wiper can be inclined or declined as necessary based on the application and the spindle heel clearance.
- The insert is normally set just once in the manufacturer's facility or when the tool is put into intial use.

Details of a finish mill 6F2B

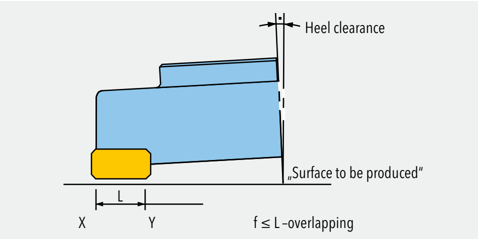


1. Eccentric pin
2. Insert
3. Insert screw
4. Pivot point
5. Adjusting screw

Operating instructions / Finish milling

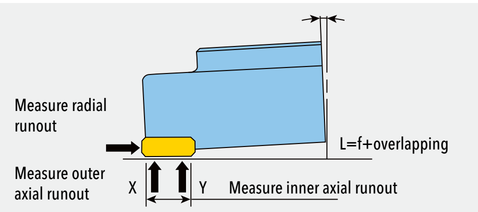
Maximum feed per revolution

The maximum feed per revolution (f) should be at least min. 0.4 mm smaller than the length (L) of the wiper edge. For tool series with different wiper lengths, refer to the tool description.

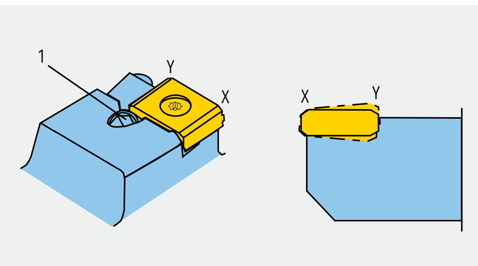


Adjustment of wiper edge

The wiper edge of the 6F2B/SF2B series has been set in the factory so that „X“ is 0.01 - 0.015 mm higher than „Y“. If the position of the wiper edge has to be corrected due to operating conditions, this is done by turning the adjusting screw 1. The insert screw 3 (see below) should be loosened before the adjusting screw 1 is turned. To increase the point „Y“ of the 6F2B/SF2B series relative to the point „X“, the adjusting screw must be turned counter-clockwise.



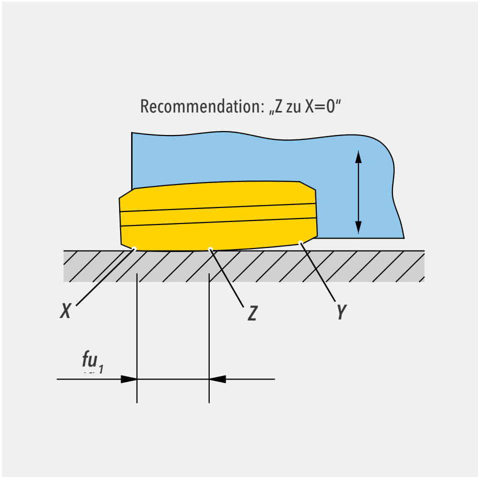
The inner and the outer axial runout are measured, i.e. at Y and X, whereby deviations of 0.01 - 0.015 mm are permitted. It should be noted, however, that the axial runout on the inside (at Y) is generally poorer than at the outside (at X), as the insert tolerances and devations of the wiper setting add up there. The radial runout should be within 40 µm. The required axial runout accuracy is achieved by interchanging the inserts.



Wiper adjustment recommendation

The Ingersoll MicroMill tools are delivered with a diameter dependent default wiper edge setting. In some cases, the wiper edge position has to be adjusted once to suit the intended application.

For this purpose, it is advisable to set the height of the wiper edge at point „Z“ to the same level as „X“. -> „Z to X = 0“ An optimal waviness of the surface can thus still be achieved with a minimum of axial pressure. „Z“ is located with respect to „X“ at the distance of the necessary feed per revolution „fu“.



Cutting arcs

Large cutting arcs:

- high thermal load
- low cutting speeds

Small cutting arcs:

- low thermal load
- high cutting speeds

ae-values of approx. 65 - 75% of the tool diameter should therefore be aimed for.

Effect of the stock allowance on the tool life

- Stock allowances that are too small, which are subject to edge rounding, mean increased cutting pressure, as the cutting edge pushes the chip ahead of itself.
- If the stock allowances are too big, the main cutting edge is subjected to too much load.
- The stock allowance has no effect on the wiper edge..

Finish milling

Change of feed rate for small and large widths

- For small widths of cut, the feed rate can be increased, because the hm-value decreases.
- Conversely, the feed rate should be reduced for large widths of cut.

Countermeasures for adverse conditions

Adverse conditions	Countermeasures
unstable clamping	• use suitable clamping devices • install additional clamping devices
unstable workpieces	• reduce the number of indexable inserts • for large stock allowances: adjust the cutting parameters
big adaption length	• for large diameters: use a plane position, where possible • for small diameters: provide suitable support

Defects and remedies

Defect	Remedy	
Waviness	Check the milling machine's guide for backlash Check the milling spindle backlash / tension	Note: The waviness is not produced by the tool!
Saw tooth profile	Heel clearance is not set correctly Re-adjust the tool with the adjusting screw	
R-value out of tolerance	Indexable inserts damaged, Stock allowance exceeded Indexable inserts have reached their wear limit	

Problem / Remedy	Chipping of the cutting edge	Extreme flank wear	Heavy cratering	Built up cutting edge	Cutting edge deformation	Comb cracks, insert breakage	Poor workpiece surface	Chatter, vibrations	Chip jamming	Edge break-outs at the workpiece	Machine overload
Cutting speed	G	H	H	G	H		G	P			H
Feed per tooth	H	G	H	G	H	H	H	P		H	H
Cutting material toughness	G					G					
Cutting material resistance to wear		G	G		G P						
Lead angle				P				H	P	H	H
Rake angle	H			G P	G P	P		G	P		
Cutting edge land	G			P			H			H	
Stability, tension	G					G	G	G			
Axial and radial runout							G	G		P	
Positioning of the milling tool						P	P	P		P	
Cooling, chip evacuation			G	G	P		P		P		
Depth of cut	P					P	P	P		H	H
Key	G: increase			H: reduce			P: optimize				

Adjustment instructions finish mill

OCTOPLUS OWHH

This tool is mainly used at high cutting speeds in cast iron range with SiN and CBN inserts. The OWHH-series is designated for short shipping materials. As an alternative carbide roughing and carbide finishing inserts can be assembled for usage at lower cutting speeds. In both cases the lower number of wipers will reduce the axial pressure into the workpiece.

Recommended cutting data

Cutter material	ISO	Workpiece material	Recom. cutting depth ap [mm] finishing	Cutting speed Vc [m/min]	Feed per tooth fz [mm]
Carbide	K	Grey cast iron GG	0,2–0,5	110–180	0,1–0,25
SiN/CBN			0,2–0,5	450–1000	0,08–0,2

SiN/CBN		HM	
			
ONCU0505ANN roughing SiN	ONCX050408FN-WCT finishing CBN	ONCU0505ANEN roughing carbide	ONCU050508FN-W finishing carbide

Attention! The OWHHxxxR00 fi nishing tool can be used either with the adjacent SiN/BN-insert grades or with the carbide inserts only.

Adjusting instructions

The fi nishing pocket can be adjusted as follows:

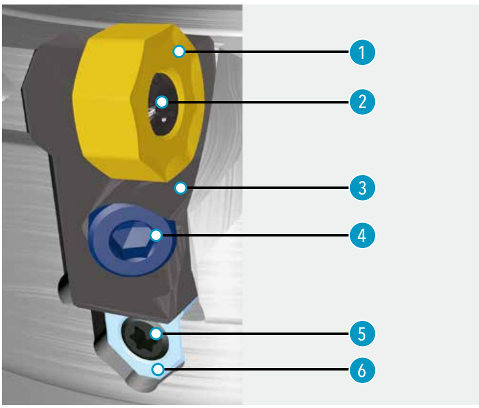
a) Finish over roughing:

- First locate the highest roughing insert.
- With the mounting screw (4) loose, use the adjusting screw (5) to set the finishing inserts 0.05 mm above the highest roughing insert.
- Tighten the mounting screw with 8 Nm.
- If necessary repeat this sequence.

b) Face run-out:

The more accurate the face run-out is adjusted (approx. 2 - 4 µm), the better the fi nal surface quality will be.

- For face run-out adjustment loosen the mounting screw (4) in order to avoid initial tension in the system.
- By turning the adjustment screw (5) clockwise, the height position of the insert increases. The highest point of the finishing insert is in the middle of the wiper.
- Der höchste Punkt der Schlichtschneide liegt in Schneidkantenmitte.
- Tighten the mounting screw with 8 Nm.
- If necessary repeat this sequence.



1	Finishing insert	ONCU / ONCX
2	Insert screw	SM40-100-10 (4,5 Nm)
3	Cartridge	6OH183R02
4	Mounting screw	DIN 912 M5X16-12.9 (8 Nm)
5	Adjusting screw	SB060-10
6	Wedge	PA-5189

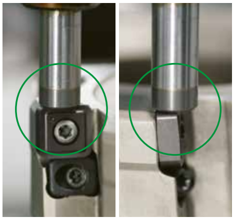
Adjustment instructions finish mill

QUADPLUS FINISH

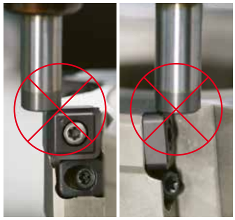
Description of components

1	Dial gauge	4	Adjusting wedge
2	Insert	5	Wedge screw
3	Insert screw	6	Screw driver

Gauge user guide

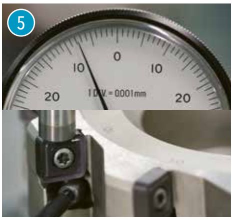


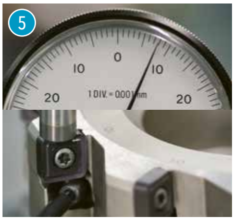
CORRECT

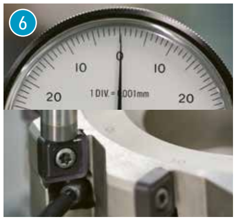


INCORRECT

Adjusting instructions







1	Move adjusting wedge [4] to lowest position by turning the wedge screw [5] clockwise. Please avoid using too much force.
2	Mount new cutting edge of insert. Make sure, that the insert pocket is thoroughly cleaned before mounting the insert. Please fix the insert screw [3] completely, as readjustment is not expected once it is done.
3	Measure the runout of the cutter when all inserts are mounted and select the highest insert as a reference. Please ensure that insert edge is not damaged during adjusting. Use optimum dial pressure only.
4	Set the height of the cutter raising the reference insert by turning the wedge screw [5] counter-clockwise. Increase height by 0.01 mm at least from the highest insert.
5	Adjust axial run-out of the remaining inserts with the same process as used with the reference insert. Max. adjustment height should not exceed 0.1 mm. Adjust the run-out in the range of 0.005 mm by turning the screw driver gradually. If it is beyond the acceptable range, please repeat steps 1; 2 and 5.
6	The insert screws do not have to be re-tightened after the axial runout has been set.

Special precautions

- While loading a new insert corner, ensure that the adjusting wedge is in the lowest position. Bottom out the adjusting wedge completely before unclamping the insert from cutter.
- Clean the insert and pocket thoroughly before mounting fresh insert/corner.
- While assembling adjusting wedge onto cutter body, please ensure that the adjusting wedge is tightened until it reaches the bottom. Also pay attention to the position of the wedge screw.

Adjustment instructions finish mill

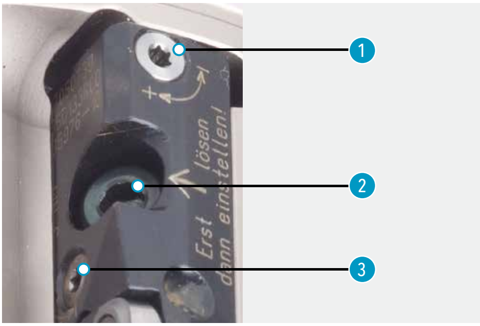
FORMMASTERV 5V2D...R00

Defined „incremental adjustment“! Adjustment range: +/- 0.5 mm (after presetting in the factory)

Tighten the clamping screw with 3 Nm!
Use a Torx6 or Torx6Plus for tightening.

CBN insert (2 cutting edges)
CNHU060310N-001 IN80B (R1) / CNHU060304N-001 IN80B (R0,4)

Adjustment instruction



1	Adjusting screw for axial adjustment 5 µm in height per increment
2	Clamping screw DIN912 M4 x 12
3	Adjusting screw for radial adjustment 5 µm in diameter per increment

Adjustment of diameter

1. Check the diameter on the tool presetter(e.g. Ø 66.02 mm).

2. Loosen clamping screw (2) of cartridge with 2 revolutions.Adjust setting screw (3) by 4 noticeable increments.Retighten clamping screw (2) with a torque wrench (adjustment at diameter of 4 x 0.005 mm = 0.02 mm).

Adjusting screw for radial adjustment

1. Check the linear dimension on the tool presetter (e.g.135.03 mm).

2. Loosen the clamping screw (2) at the cartridge by 2 turns. Turn the adjusting screw (1) with 6 noticeable increments. Re-tighten the clamping screw (2) with the torque wrench (axial adjustment = 6 x 0.005 mm = 0.03 mm).

Recommended cutting data for FormMasterV (adjustable)

ISO	Material	Cutting data	IN2005	IN05S	IN80B
P	unalloyed steel	cutting speed vc [m/min]	200–350	–	–
		feed per tooth fz [mm]	0,1–0,2	–	–
	alloyed steel < 800 N/mm²	cutting speed vc [m/min]	180–300	–	–
		feed per tooth fz [mm]	0,1–0,2	–	–
	alloyed steel < 1100 N/mm²	cutting speed vc [m/min)	150–250	–	–
		feed per tooth fz [mm]	0,1–0,2	–	–
K	gray cast iron	cutting speed vc [m/min]	200–300	–	800–1200
		feed per tooth fz [mm]	0,1–0,2	–	0,05–0,12
	cast materials	cutting speed vc [m/min]	180–300	–	600–1000
		feed per tooth fz [mm]	0,1–0,2	–	0,05–0,12
S	aluminum	cutting speed vc [m/min]	–	800–1200	–
		feed per tooth fz [mm]	–	0,1–0,2	–

- Recommended stock allowance for plunging: ae = 0.1 – 0.15 mm.
- At max. extension lengths the cutting speed vc must be reduced!
- For gray cast iron with a stock allowance of max. 0.1 mm plunging and back stroke machining is possible!

Adjustment instructions finish mill

MICROMILL 5FKX

This tool is designed for usage at high speed finishing in short chipping materials only. This cutter series creates low axial pressure into the workpiece due to a reduced number of wipers. In order to receive best surface results the wiper inserts should be adjusted in face run-out in between 3 - 5 µm.
The wiper insert with CBN layer can cover a max. feed per revolution of 6 mm. So the feed per revolution should be < 6 mm accordingly, however the max. allowed feed per tooth should not be exceeded (see below chart).

Recommended Cutting Data:
Fumax. = 6 mm

ISO	Material	Recommended cutting depth ap [mm] finishing	Cutting speed Vc [m/min]	Cutting speed fz [mm]
K	gray cast iron	0,2–0,5	110–180	0,1–0,25
		0,2–0,5	450–1000	0,08–0,2

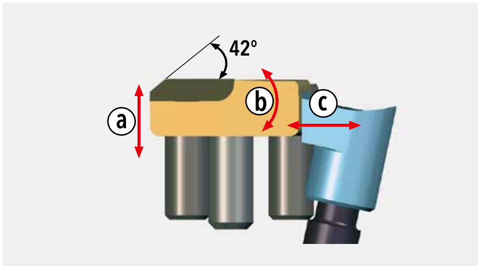
Adjustment instructions for 5FKX finish mills

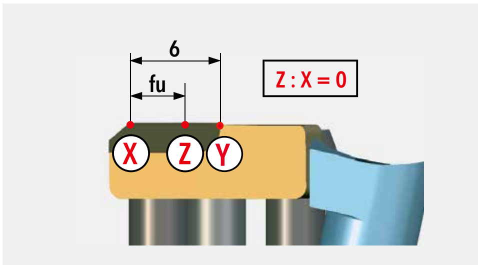
The finishing pocket can be adjusted in the following degrees of freedom:

a) Face run-out:
Uniformly turn both adjusting screws (1 + 2) in the same direction. (The wiper stations should thereby stand approx. 0.03 - 0.05 mm over the highest roughing station). To reduce the protrusion of the wiper stations, turn the adjusting screws (1+2) clockwise, to increase it, turn them counter-clockwise. Both adjusting screws must be adjusted by the same amount.

b) Wiper adjustment:
Turn the adjusting screw (2) counter-clockwise to set Y > X.

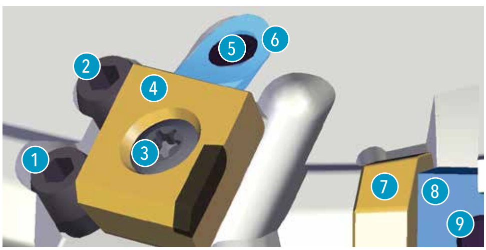
c) Diameter adjustment:
By adjusting with screw and wedge (5 + 6). The main cutting edge should overlap with the SiN roughing station in the 42°-area or stay max. 0.02 mm behind the roughers in radial direction.
These adjustments must be done with the insert unclamped! After each adjustment (a. b. c) re-tighten the insert screw with 6 Nm.





to b): adjustment of the wiper inclination

1	Adjusting screw	SC050-01
2	Adjusting screw	SC050-01
3	Insert screw	SM50-160-10 (6 Nm)
4	Insert (CBN)	YWH454L101
5	Differential screw	SB060-02
6	Wedge	2M0612-01
7	Insert (SiN)	OPDN53-100
8	Clamping wedge	2K0610-01
9	Clamping screw	SB060-01 (4,5 Nm)



Adjustment instructions finish mill

MICROMILL VFBN & VFEN

This cutter is mainly designed for finishing applications in short chipping materials. With these cutter series a lower number of wiper stations is in use, what creates less axial forces into the workpiece. For achieving a good surface finishing, these wiper stations should be adjusted in a face run out of approx. 3 - 4 µm. The feed per revolution should be choosen according to the required surface specifications, bit within the maximum allowed feed per tooth.

ISO	Material	Recommended cutting depth ap [mm] finishing	Cutting speed Vc [m/min]	Cutting speed fz [mm]
K	GG	0,3 – 0,5	110 – 180	0,10 – 0,35
	GGG	0,3 – 0,5	100 – 160	0,10 – 0,35
	GJV	0,3 – 0,5	80 – 120	0,10 – 0,35
	ADI	0,3 – 0,5	80 – 120	0,10 – 0,35

Axial adjustment

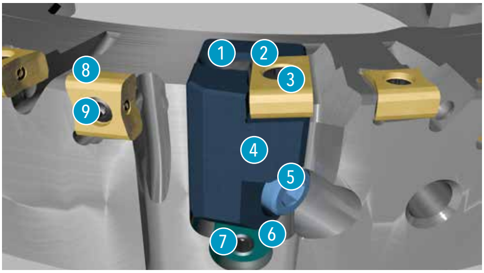
1. Clean the tool.

2. Assemble the ▽-insert NND313.111. The face run out should be max. 0.025 mm. The insert screw (9) has to be tightened with 2 Nm for clamping the insert. Then search for the highest ▽-insert.

3. The finish-step over the highest roughing insert should be checked and adjusted with one YXD-insert (2) at the finish seats. The standard adjustment is 0.05±0.02 mm higher than the ▽-insert. For adjusting the cartridge (4) please loosen the clamping screw (5) and adjust it in axial direction with the differential screw (7). Tighten the clamping screw again with 10 Nm and check the insert position. If necessary repeat and correct.

4. Assemble the other fi nishing inserts and and adjust those with the same sequence for a face run out of 3 - 4 µm for finishing or 4 - 8 µm for roughing application.

5. Check face and diameter run out again, check cutter height, if necessary correct.

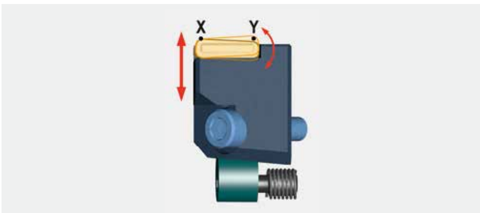


No.	Item	Designation
1	Barrel screw	SC050-14
2	Adjustable finishing insert	YXD323-101/YXD334-100
3	Insert screw	SM40-110-00
4	Cartridge	56B193R00/56D193R00
5	Clamping screw	DIN 912 M6X30-12.9
6	Wedge	KR012-002
7	Differential screw	SB060-01
8	Insert	NND313-111
9	Insert screw	SM30-090-10

Adjustment of the wiper flat Y to X

Usually the standard adjustment is sufficient for the most finishing applications, but with the adjustment developed by Ingersoll, the wiper fl at can be adjusted to the required feed per revolution.

For further information of 'Adjustment of wiper' see page S. 37-38



Adjustment instructions rough-finishing cutter

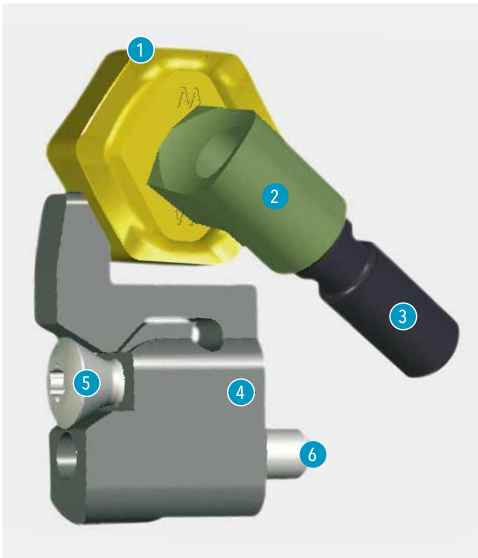
ECOMILL 5NKL

The cutter is mainly designed for semi-finsih operations of short chipping materials, but depending on the conditions and requirement is can be used also for roughing operations with improved surface qualities as well as for finishing operations.
The cutter has a certain number of axial adjustable finishing stations, depending on cutter diameter. These are cutting effectively as a main cutting edge. In order to get good surface results, the face run out over the finish inserts should be approx. 4 µm. Thereby the feed/rev. can be bigger than the length of the secondary cutting edge of 6.5 mm.

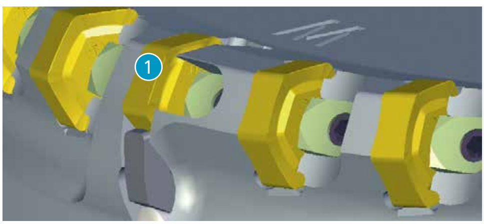
ISO	Material	Recommended cutting depth ap [mm]			Cutting speed Vc [m/min]	Feed per tooth fz [mm]
		maximum	semi-finish.	finishing		
K	GG	7	1 – 1,5	0,3 – 0,5	110 – 180	0,12 – 0,35
	GGG	7	1 – 1,5	0,3 – 0,5	100 – 160	0,12 – 0,35
	GJV	7	1 – 1,5	0,3 – 0,5	80 – 120	0,12 – 0,35
	ADI	7	1 – 1,5	0,3 – 0,5	80 – 120	0,12 – 0,35

Adjusting instruction

1. Clean the tool.
2. Assemble the roughing inserts HNCF090516TN-WE, their face run out should be max. 0,025 mm. The differential screw (3) should be tighten with 5 Nm for clamping the insert. Then search for the highest insert.
3. The finish-step over the highest roughing insert should be checked and adjusted with one finish insert HNCF0905DNN-W (1) at the finishing seats. The standard adjustment is 0,07±0,01 mm higher than the roughing insert (assemble the finishing insert with the pressed 'W' to the top). For adjusting the element (4) please loosen the differential screw (3) and adjust it in axial direction with the adjusting screw (5). Tighten the differential screw (3) again with 5 Nm and check the insert position. If necessary repeat and correct.
4. Assemble the other finishing inserts and adjust those with the same sequence in order to get the face run out within 3 - 4 µm for finishing, or 4 - 8 µm for roughing operations.
5. Check face and diameter run again, check cutter diameter and cutter height, if necessary correct.



No.	Item	Designation
1	Adjustable finishing insert	HNCF0905DNN-W
2	Wedge	2M0612-01
3	Differential screw	SB060-02
4	Adjusting element	AJHN 10N
5	Adjusting screw	SO 40140I
6	Mounting bolt	SM40-110-00



Adjustment instructions semi-finishing cutter

ECOMILL VF2V

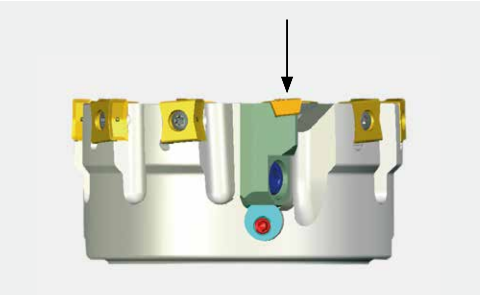
At finishing operations the following feed rates per revolution should not be exceeded:

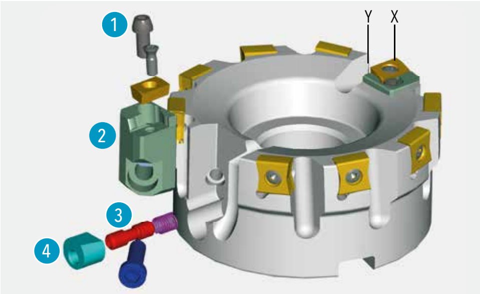
Insert	Feed rate per revolution
YDA323L101	8 mm/rev.
YDA323L114	4 mm/rev.
NNE324L109	4 mm/rev.

The feed rate per revolution must be selected accordingly to the required surface finishes, however, must not exceed the maximum feed per tooth of the roughing inserts!

Adjustment of finish insert above roughing insert

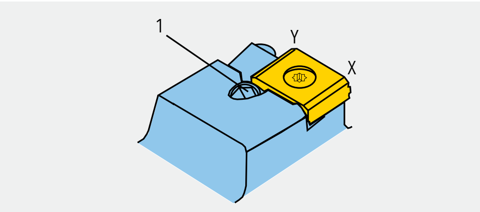
0,08 +/-0,02 (factory setting) can be adjusted by shifting insert holder (2) by means of wedge (4) via differential screw (3).

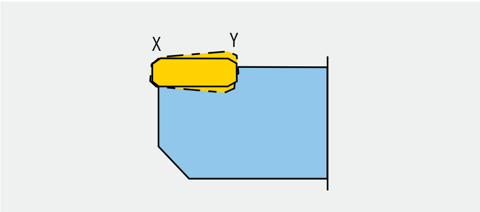


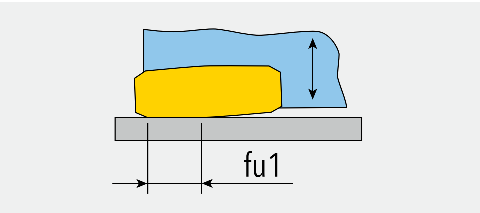


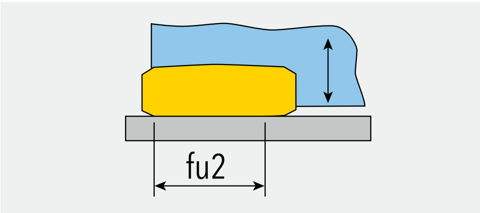
Adjustment of wiper flat

With the adjustment patented by Ingersoll the wiper flat can be adjusted to the required feed rate per revolution. Should the application conditions require a change of the position of the wiper flat, it is carried out by turning the adjusting screw (1). Before adjusting the wiper flat, the insert must be loosened. For the series VF2V adjustment is carried out in lefthand direction Y>X.





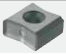
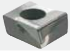




Adjustment instructions shoulder-type milling cutter

VMAX VK2V

NOTE: At finish operations the following feed rates per revolution should not be exceeded:

Insert	Feed rate per revolution
 YNE324-100	max. 3,8 mm/rev..
 NYE324R100 (PKD)	max. 3,0 mm/rev.

Adjustment instruction:

No.	Item	Designation
1	L-nest	PAR0615
2	Insert screw	SM40-120-20
3	Insert	NNE324... or NJE324... or YNE324... or NCE325...
4	Adjusting screw	SC05-14

1. Clean cutting tool.

2. At first loosen adjusting screw (4) to relaease the L-nest. Then turn in the adjusting screw only until contact with L-nest has been made (without making any adjustment yet).

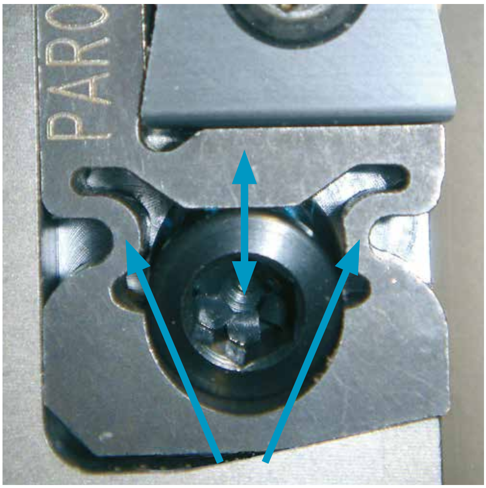
3. Equip cutting tool with inserts (3) and slightly fasten insert screws (2) by approx. 1 Nm.

4. Look for highest insert seat and adjust it in axial direction by 0.02 mm with adjusting screw (4).

5. Now tighten insert screw with 4.5 Nm by pressing the insert into the insert seat at the same time.

6. Adjust remaining inserts in axial direction by repeating steps 2 to 5 until an axial run-out of 3 to 4 µm is obtained.

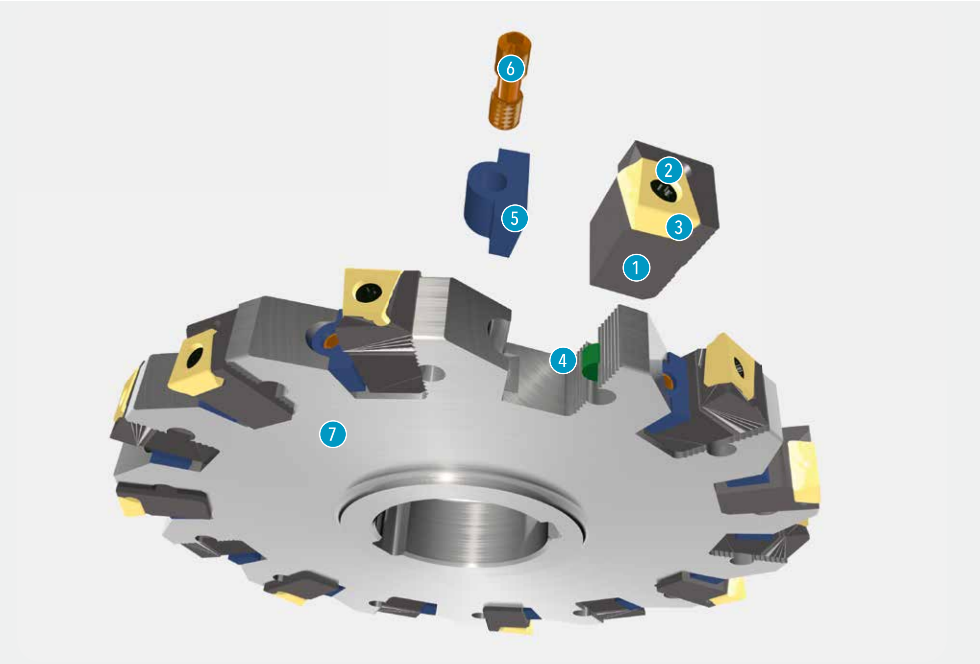
7. Finally check axial and radial run-out again as well as cutting tool dia. and overall height and correct them, if necessary.



Flexible area

Adjustment instructions slotting cutter

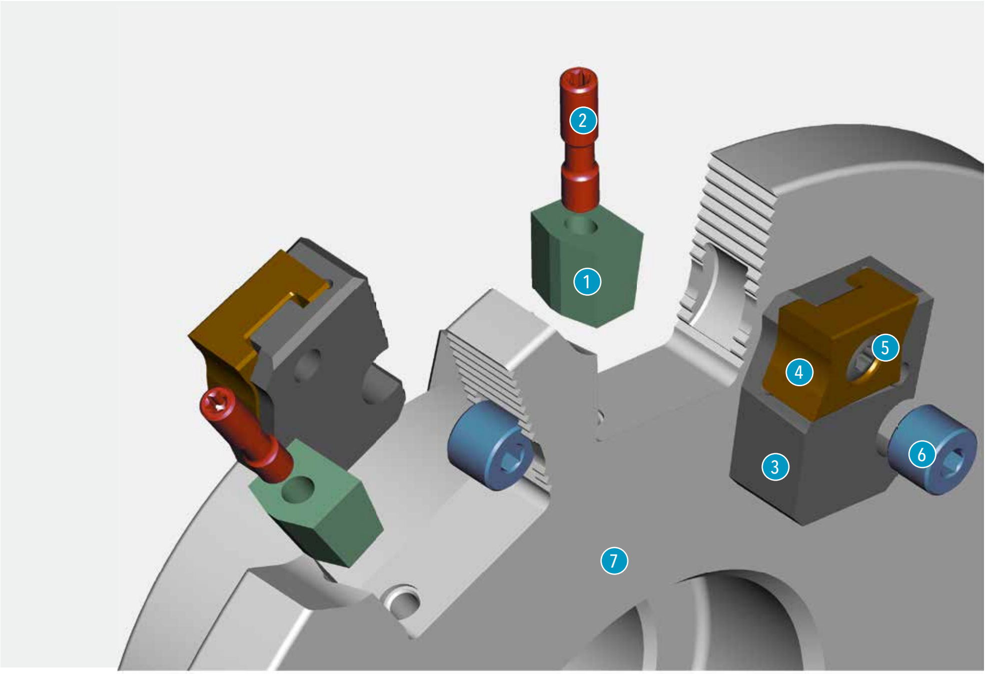
SMAX 4SJ6...



1	Slightly grease the adjusting screw [4] and mount it into the cutter body [7].
2	Slightly grease the differential screw [6] and screw it into the mounting wedge [5] until a projection of approx. 1 mm to the surface of the mounting wedge has been achieved.
3	Mount the insert cartridge [1] into the cutter body [7], so that the adjusting screw [4] is positioned in the circular groove of the insert seat.
4	Locate insert cartridge [1] with your fingers and position the mounting wedge [5] by means of the differential screw [6] into the respective slightway in the cutter body [7] and screw it slightly, however, backlash-free.
5	Repeat steps 1 - 4 with the opposite cartridge pocket and continue crosswise until all cartridge pockets are equipped with insert cartridges.
6	Then adjustment of mounting height and cutting width is carried out via the indexable insert [3] which is screwed into the insert cartridge [1] by means of the adjusting screw [4] and by using a measuring table with dial gauge according to the parameters and cutting width ranges indicated in the catalogue. A cutter inspection device or an equivalent profile projector make the adjustment work easier.
7	After the desired cutting position has been achieved, tighten the differential screw [6] by means of a torque wrench with approx. 4.5 Nm.
8	Repeat steps 6 - 7 for all other insert cartridges, until the cutter has the desired cutting width and an axial run-out of approx. 0.02 - 0.03 m.

Adjustment instructions slotting cutter

GOLDSLOT 4VJ5V



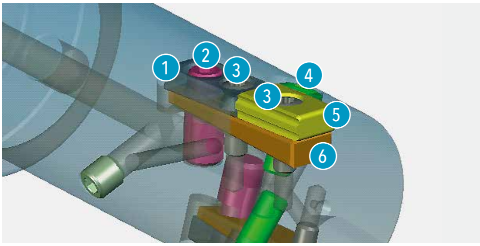
1	Mount adjusting screw [6] lightly greased into the cartridge [3].
2	Screw the lightly greased differential screw [2] into the mounting wedge [1] until it overlaps approx. 1 mm from the wedge surface.
3	Mount cartridge [3] into the cutter body [7], so that screw [6] is positioned in the circular groove at the cartridge contact face.
4	Fix the cartridge [3] with your fi ngers and use differential screw [2] to position the clamping wedge [1] in the V-shape of the cutter body [7] and mount it loosely free of clearance.
5	Repeat steps 1 - 4 for the opposite cartridge and proceed diagonally until all cutter slots are mounted with cartridges.
6	Tool height and cutting width are adjusted with the mounted insert [4] in the cartridge [3] with adjusting screw [6] on a measuring table and dialgauge according to the parameters and cutting width range in the catalogue. A runout inspection fixture or profile projector simplifies the adjustment.
7	After the exact cutting edge position has been reached, please tighten the differential screw [2] with a torque wrench adjusted 2.5 Nm.
8	Repeat steps 6 - 7 for all cartridges, until the exact cutting width is adjusted and an axial run-out of 0,02 - 0,03 mm has been reached.

Adjustment instructions duo-bore finishing tool

POWERMAX XJ-50004SJ6

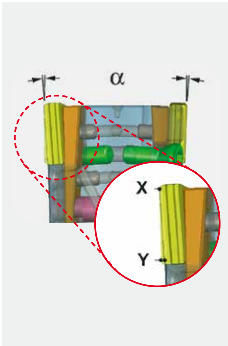
In addition to the diameter adjustment Ingersoll cutters with adjustable duo-finishing pockets also offer the possibility to adjust the inclination of the minor cutting edge towards the bore wall. That way the inclination of the minor cutting edge of the insert (5) can be adjusted according to the required feed per revolution fu.

NOTE: Adjustment of inclination is not affected by a following diameter adjustment!



No.	Item	Designation
1	Anvil	PA-5039
2	Eccentric pin	PN080-01
3	Screw	SM40-130-00
4	Barrel screw	SC050-14
5	Insert	YCE323-107/108
6	Wedge	GZ-5022

Adjustment of inclination



1. Clean the tool, mount it on an optical inspection device and lock the spindle.

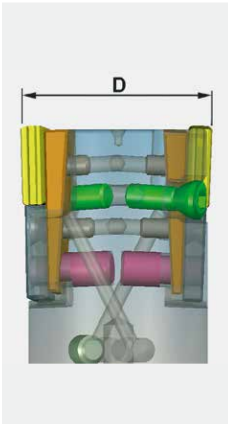
2. Loosen insert screw (3) by approx. 1/2 to 3/4 of a turn.

3. The diameter position of point Y can be adjusted in relation to point X by turning the barrel screw (4):
   - Clockwise turn of barrel screw (4) Y grows in diameter in relation to X.
   - Counter-clockwise turn of barrel screw (4) Y diameter is reduced in relation to X.

4. Press insert (5) into the pocket and tighten the insert screw (3) with a torque of 4,5 Nm.

5. Check position of X to Y, if necessary repeat adjustment.

Diameter adjustment



1. Clean the tool, mount it on an optical inspection device and lock the spindle.

2. Loosen insert screw (3) for insert (5) and anvil (1) by approx. 1/2 to 3/4 of a turn.

3. Adjust insert (5) by turning the eccentric pin (2) for a first pre-adjustment. The setting should be approx. 0.01 mm smaller than the nominal dimension.

4. Tighten insert screw (3) for insert (5) and anvil (1) with a torque of 4.5 Nm.

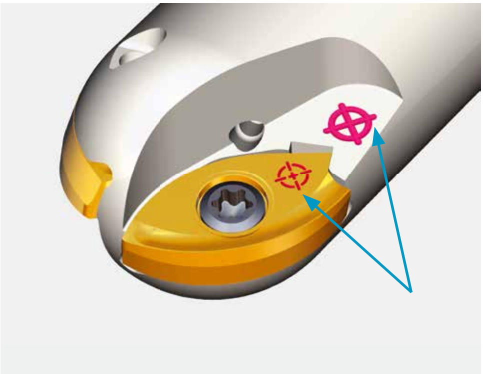
5. Repeat procedure 2 to 4 for final diameter adjustment. It is recommended to record the diameter position after loosening the screws (3). Then the remaining amount to the nominal diameter dimension can be adjusted easily.

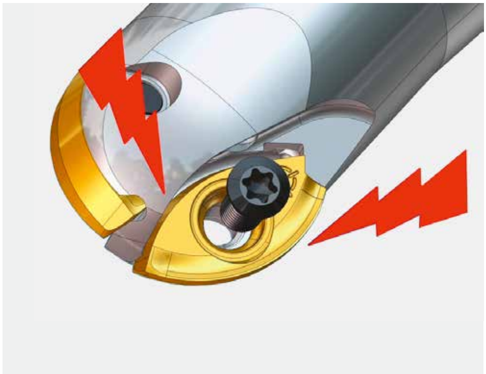
6. Check diameter and run-out again, and if necessary correct it.

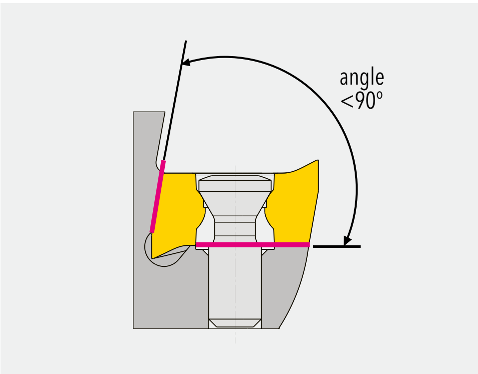
Technical features

PRODUO Ball nose end mill 1TW

- One insert type fitting into both insert seats (central cutting edge and peripheral cutting edge).
- Different insert types are not necessary (less insert variety)!
- Insert and pocket are marked with crossline-signs for correct assembly (protection against wrong positioning).







Dovetailed machanism

Helix-cutting edge for smooth milling

Axial supporting side for strong clamping.

Technical features

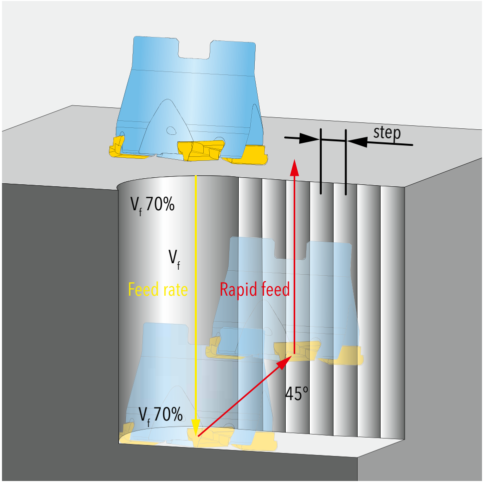
PUNCHIN Plunge mill

Why choose PunchIn:

- Considerably shorter roughing times
- Shortens the workpiece throughput time
- Very high stock removal rate
- Smooth cut
- Particularly suitable for deep contours
- Effective use from 2.5 x D
- Mainly axial forces, little radial deflection

TIPS plunge milling:

- For plunging and for contact with the bottom at a length of approx. 3 mm, the feed rate should be reduced to 70-80%.
- When machining VA steel, the feed rate must not be reduced when plunging.
- Movement away from the contour before retracting to the safety level is recommended, depending on the workpiece and diameter between 0.3 - 1 mm.



Examples of tools suitable for plunging

DHU…X_R00

PHU…X

15V1D…X

15V1G…X

Slot cutter with angular radius

SHU…R50/51

5V2D

5V6G

FHU…X

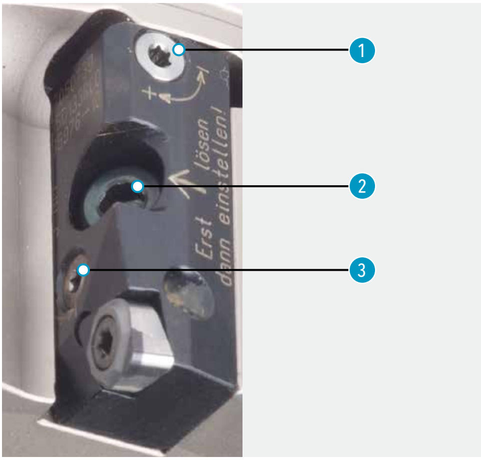
QHU…F/E

12U5… carbide shank

Technical features

FORMMASTER Adjustable finish mill 5V2D...R00

Adjustment instruction



Cartridge: 7050739 65D133R00

1	Setting screw: axial adjustment 5 µm in height per increment
2	Clamping screw DIN912 M4 x 12
3	Setting screw: radial adjustment 5 µm in diameter per increment

Defined „incremental adjustment"! Adjustment range: +/- 0.5 mm (after presetting in the factory)

Tighten the clamping screw with 3 Nm!
Use a Torx6 or Torx6Plus for tightening.

Adjustment in diameter

1. Check the diameter on the tool presetter (e.g. Ø 66.02 mm).

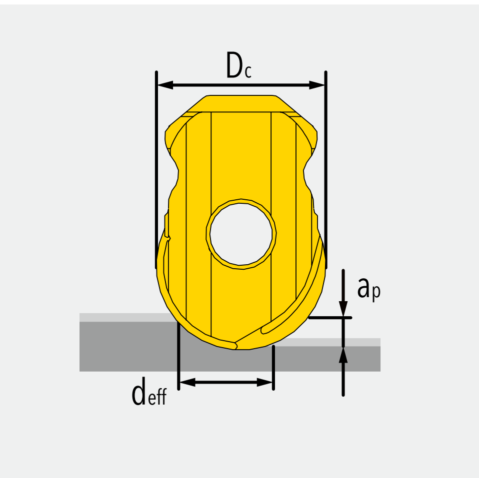
2. Loosen clamping screw (2) of cartridge with 2 revolutions. Adjust setting screw (3) by 4 noticeable increments. Retighten clamping screw (2) with a torque wrench (adjustment at diameter of 4 x 0.005 mm = 0.02 mm).

Axial adjustment

1. Check the linear dimension on the tool presetter (e.g.135.03 mm).

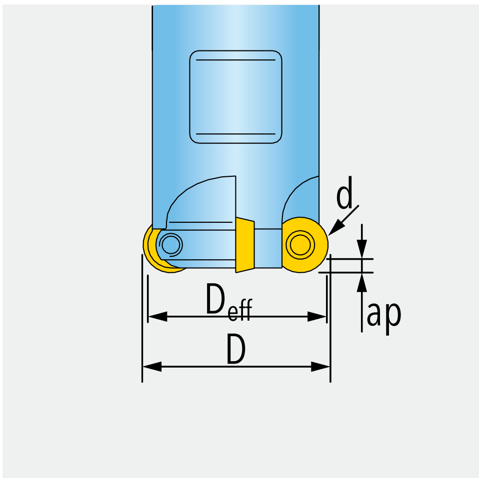
2. Loosen the clamping screw (2) at the cartridge by 2 turns. Turn the adjusting screw (1) with 6 noticeable increments. Re-tighten the clamping screw (2) with the torque wrench (axial adjustment = 6 x 0.005 mm = 0.03 mm).

Determination of the effective diameter



Effect of the tool geometry on the effective cutting speed

$d_{eff.} = 2 \times \sqrt{a_p \times (D_c \times a_p)}$



15B1E030043X8R00
RHHW0802MOTN IN2006

$d_{eff} = 2 \times \sqrt{a_p \times (d-a_p)} + (D-d)$

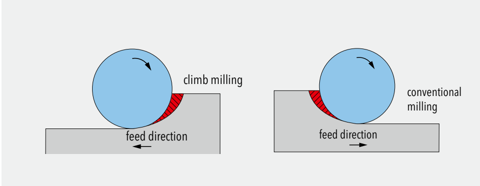
Tool Ø mm: 30
Insert Ø mm: 8
ap mm: 0,4
eff. tool Ø mm: approx. 25,5

Solid carbide cutters

General formulae

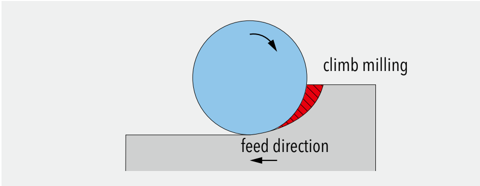
Variable	Unit	Formula
Speed of rotation	min^{-1}	$n = \frac{v_c \times 1000}{D \times \pi}$
Cutting speed	m/min	$v_c = \frac{D \times \pi \times n}{1000}$
Feed rate	mm/min	$v_f = f_z \times Z_{eff} \times n$
Feed per tooth	mm	$f_z = \frac{v_f}{Z_{eff} \times n}$
Average chip thickness	mm	$h_m = f_z \times \sqrt{a_e/D}$
Main utilization time	min	$t_h = \frac{L \times i}{V_f}$

Machining of soft materials



Preferably use climb milling method!

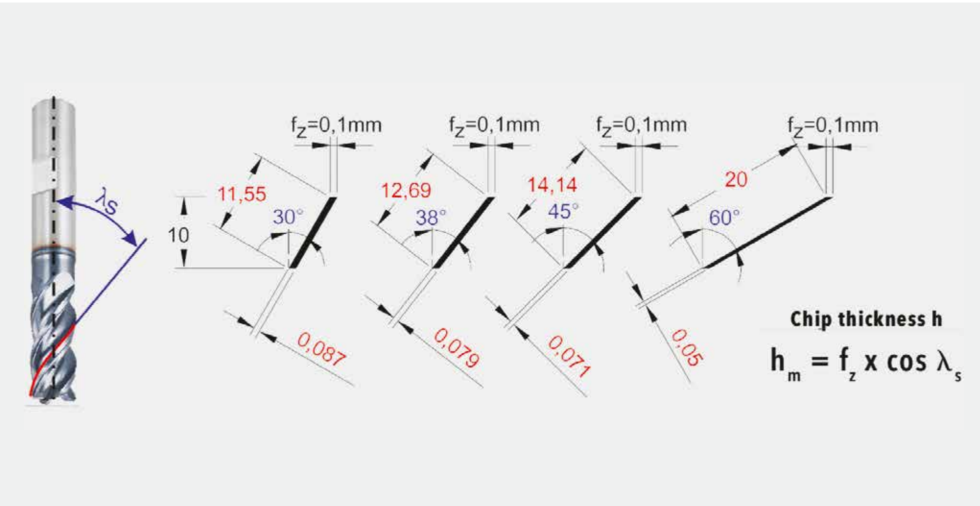
Machining of hard materials > 50 HRC



Preferably use climb milling method!

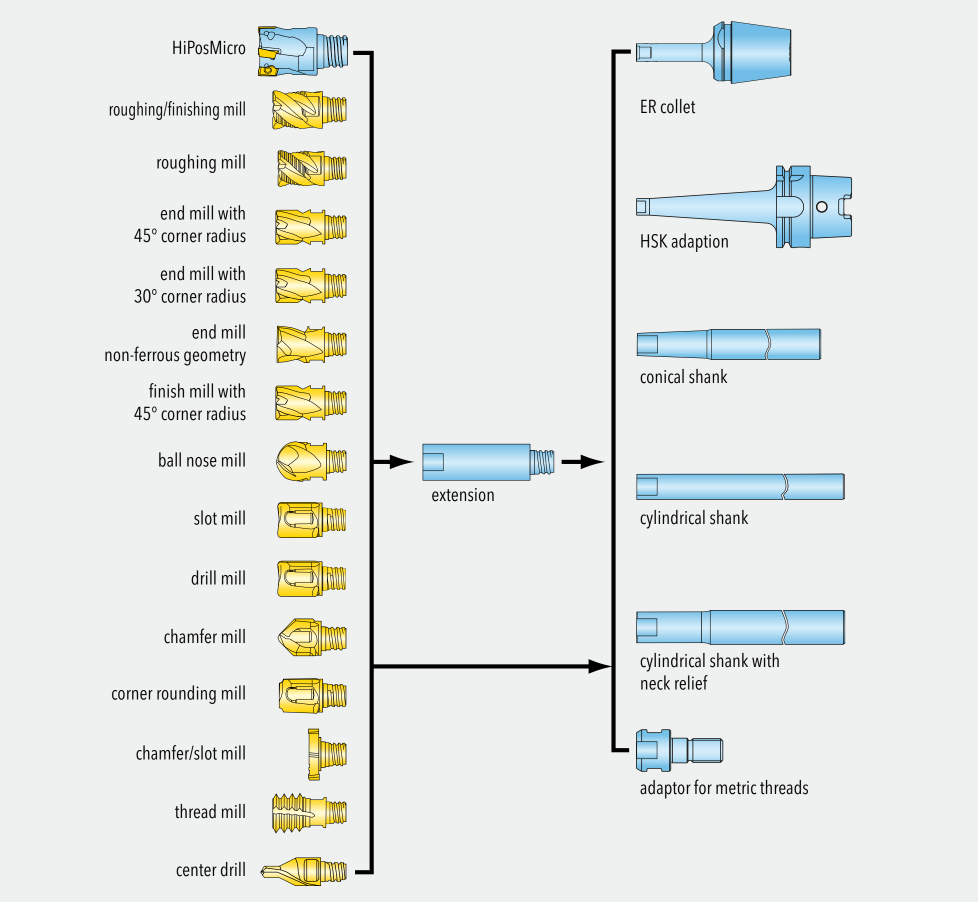
Chip thickness h

As the helix angle increases, the chip thickness decreases!



Key			
vc = Cutting speed	fz = Feed per tooth	Q = Stock removal rate	L = Machining length
D = Tool diameter	vf = Feed rate	ae = Lateral width of cut	i = Number of cuts
n = Speed of rotation	z = No. of effective teeth	ap = Depth of cut	hm = Average chip thickness
π = Pi (3,1415…)	h = Chip thickness	th = Main usage time	E% = Engagement ratio
λs = Lambda (helix angle)			

ChipSurfer



The ChipSurfer belongs to a modular tool series, consisting of a shank with multiple interchangeable tips for a multitude of different applications.
Roughing and finishing of 2D and 3D contours.
Shoulder milling with close pitch, 90° milling tips.
Special slotting tips for steel and aluminum.
Custom geometries possible thanks to new sintering technology.
Rigid screw connections for vibration-free milling and high feed rates.
Different tool shanks available for optimal contour adaptation.
When changing the tips, attention must be paid to proper tightening.
The contact surface of the tip and the holder must be level.
Before tightening, ensure tools are clean.

Benefits
Lightning-fast exchange of the tips heads at the machine.
High repeatability and true running accuracy.

Cutter-Ø	Thread-in thread	Nm
6,4 / 8	T05	7
9,5	T06	10
12,7	T08	15
15,9	T10	28
19	T12	28
25,4	T15	28

Installation of the ChipSurfer

Step 1
Screw in the ChipSurfer hand-tight (Fig. 1) until you see a small air gap at the connection (Fig. 2).

Step 2
Tighten the ChipSurfer 1/4-turn (Fig. 3).

Step 3
Check that the ChipSurfer is properly seated with a feeler gauge.
The feeler gauge must not fit between the holder and the ChipSurfer (Fig. 4).
If it does still fit, tighten the ChipSurfer again until the feeler gauge no longer fits in the gap (Fig. 5).

NOTE: You can purchase a preset torque wrench (DT- ... series)

Fig. 1: tighten by hand

Fig. 2: air gap at the connection

Fig. 3: tighten tip in the holder

Fig. 4: the feeler gauge must NOT fit in the gap

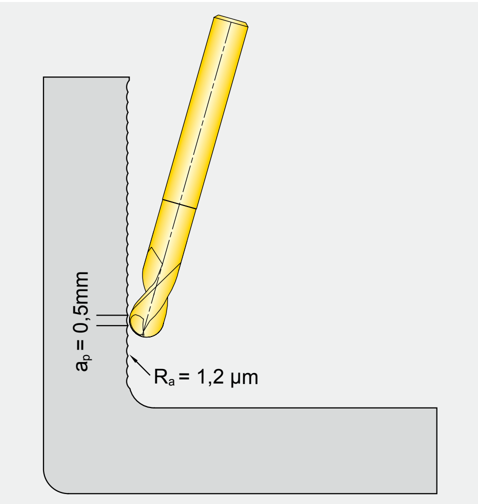
Fig. 5: correct fit

Technical features conical segment cutter

CHIPSURFER Tapered end mill 46W…

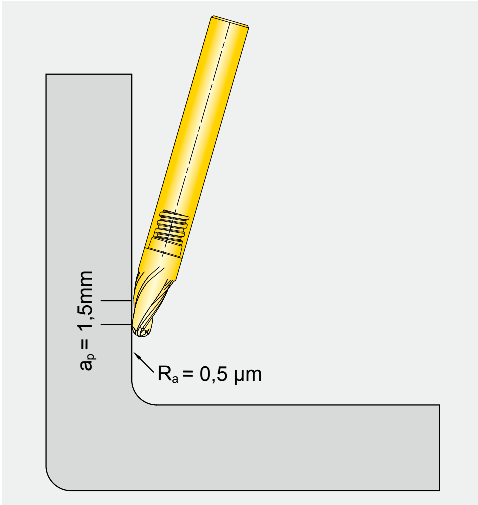
The uncoated version in the grade IN05S with polished chip gullets covers machining operations in the aluminum industry. The version in coated grade IN2005 with the special cutting edge geometry ensures the best results in mold and die industry and aerospace industry. The grades IN05S and IN2005 are ideal for machining aluminum materials, steels of material group „P", stainless steels of material group „M" and difficult to machine materials of group „S".

Compared to conventional ball nose cutters the cutting edge form in 3-fold radius design has the great advantage that up to 3 times higher ap can be reached, which drastically reduces the machining time. Furthermore, the very large flank radii create a soft overlapping and thus a visible and palpable improvement of the generated surface quality. Due to the tapered basic geometry side walls <90° can also be machined very efficiently, economically and reliably.



The high-precision cutting edge profile with a profile tolerance of +/- 10 µm and the indexing tolerance of the ChipSurfer system of +/- 20 µm allow the tools to be changed directly at the machine's spindle, thus simplifying the practical handling of the tools. Thanks to the long-proven ChipSurfer system, every version of shank extension is available. The short-design steel extensions and the overlong vibration-damped carbide and heavy metal shanks make the ChipSurfer bevel cutters very flexible to meet the different requirements with respect to machining cavities, machines, and above all, workpiece conditions.

Suitable CAD/CAM systems are required, which can calculate the multi-axis machining operations with tapered end mills and convert to machining strategies accordingly, machining strategies, as e.g. strategy „Advanced Toolform" of Work NC.



Advantages

- Highly economic finishing tools for multiple-axis machining and for side walls <90.
- Improved surface quality.
- Up to 3 times faster machining times compared to ball nose cutters.
- 3-fold radius at cutting edge contour.
- Unique interchangeable head solution for mold and die industry, aluminum machining and aerospace. industry, especially for turbine blades, impellers and blisks.
- Ø8/Ø10/Ø12/Ø16.
- Uncoated aluminum geometry and coated steel geometry.
- Shank extensions in steel / carbide / heavy metal.
- Profile accuracy +/- 10 µm / tool change accuracy +/- 20 µm.

Produktübersicht

Aluminum geometry – IN05S

Steel geometry – IN2005

CHIPSURFER Finsh mill lens-shape 46D… with I.C.

The finishing end mill in lens-shape is designed to handle expecially long-lasting finishing operations with standard ball nose end mills in less time while improving the surface quality. The high-precision ground front geometry is suitable for larger path offsets in semifinish and finish operations. The large radius creates smoother transitions of the machined paths and thus an improved tactile, visible and measurable surface quality. The new finishing end mill will cover the diameter range Ø8/Ø10/Ø12/Ø16 and Ø20 mm.

A special feature is the internal coolant supply, which is exactly directed into the machining area to improve the tool life significantly.

Application Range

Suitable for multi-axis semi-finishing and finishing of blisks, impellers and turbine blades in aircraft industry as well as for applications in die & mold industry and general mechanical engineering.

Grade IN2005 in combination with the special insert geometry provide best results in die & mold industry as well as in aircraft industry. Steels of machining group "P", stainless steels of group "M", difficult to machine materials of group "S" and cast materials of group "K" can be machined excellently.

Technical Features

Due to the high-precise insert's profile tolerance of +/- 10 µm and the exchange accuracy of the ChipSurfer system of +/- 20 µm tools can be exchanged directly at the machine spindle, which allows much easier handling of tools in practice. The longproven ChipSurfer system provides any type of shaft extensions. The short-design steel extensions and the overlong vibrationdamped carbide and heavy metal shanks make the lense-shape ChipSurfer multi purpose to meet the different requirements with respect to machining cavities, machines, and above all, workpiece conditions.

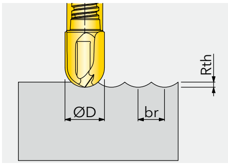
If necessary, particular attention must be paid to suitable CAD/CAM systems, which are able to program multi-axis machining with lens shape end mills and to develop appropriate machining strategies.

Advantages

- High perfomance finish mills
- Improved surface quality and machining time compared to ball nose tools
- Precise lens-shape geometry
- Unique exchangeable head system for mold & die industry
- Ø8/Ø10/Ø12/Ø16
- With internal coolant supply and 3 coolant outlet bores
- Shaft extensions in steel / carbide / heavy metal
- Profile accuracy +/- 10 µm / exchange accuracy +/- 20µm

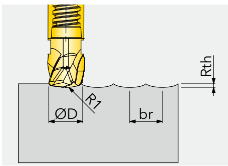
Calculation of Path Distance b_r

Ball nose end mill



Calculation of path distance br at desired roughness depth R_{th} 2,5 µ	
R_{th}:	0,0025 mm
Ø / R :	Ø 8 / R= 4
Formula:	$b_r = 2x\sqrt{R_{th}(D - R_{th})}$
b_r:	0,28 mm

Lens-shape end mill



Calculation of path distance br at desired roughness depth R_{th} 2,5 µ	
R_{th}:	0,0025 mm
Ø / R :	Ø 8 / R= 4
Formula:	$b_r = 2x\sqrt{R_{th}((DR1 \times 2) - R_{th})}$
b_r:	0,55mm

Comparison of Path Distance Ball Nose End Mill vs. Lens-shape End Mill

Path distance b_r at roughness depth R_{th} 2,5µ:

ball nose end mill			lens-shape end mill		
Ø [mm]	r [mm]	br [mm]	Ø [mm]	r [mm]	br [mm]
8	4	0,28	8	15	0,55
10	5	0,32	10	20	0,63
12	6	0,35	12	25	0,71
16	8	0,40	16	35	0,84
20	10	0,45	20	45	0,95

Path distance b_r at roughness depth R_{th} 5,0µ:

ball nose end mill			lens-shape end mill		
Ø [mm]	r [mm]	br [mm]	Ø [mm]	r [mm]	br [mm]
8	4	0,40	8	15	0,77
10	5	0,45	10	20	0,89
12	6	0,49	12	25	1,00
16	8	0,57	16	35	1,18
20	10	0,63	20	45	1,34

CHIPSURFER BARREL-FORM END MILL 48E...

The series of barrel-form end mills is designed to handle expecially long-lasting finishing operations with standard ball nose end mills or toric mills in less time while improving the surface quality. The high-precision ground barrel shape geometry is suitable for larger path offsets in semi-finishing and finishing operations. The large radius creates a softer transition of the machined paths and thus a sensible, visible and measurable improved surface quality.

The barrel-form end mills cover the diameter range Ø12 and Ø16 mm.

Application Range

Thanks to the unique cutting geometry of the tool - the 6-flute design paired with the advantages of the ChipSurfer interchangeable head system - the barrel cutter covers finishing applications on 90° shoulders as well as steep free-form surfaces where we do not expect collision due to clamping and/or part profile. The special design of the barrel-form end mill also enables it to be used on 3-axis machines and components, for which machining on deep cavities represents an enormous challenge.

Grade IN2005 together with the special cutting edge geometry ensure best results in mould & die industry as well as in aerospace industry. Steels from material group "P", stainless steels from material group "M", difficult-to-machine materials from group "S" and cast materials from group "K" can be machined excellently.

Technical Features

Due to the high-precise insert's profile tolerance of +/- 10 µm and the exchange accuracy of the ChipSurfer system of +/- 20 µm the tools can be exchanged directly at the machine spindle, which allows much easier tool handling in practice. The long-proven ChipSurfer system provides any type of shaft extensions. The short-designed steel extensions and the overlong vibration-damped carbide and heavy metal shanks make the ChipSurfer barrel-form end mills multi-purpose to meet the different requirements with respect to machining cavities, machines, and above all, workpiece conditions.

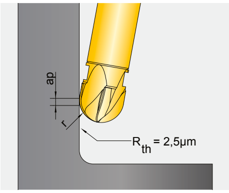
If necessary, particular attention must be paid to suitable CAD/CAM-systems, which are able to program multi-axis machining with circle segment cutters and to develop appropriate machining strategies.

Advantages

- Highly economical finishing cutters
- Improved surface quality and machining times several times faster than ball nose cutters
- High precision barrel geometry
- Unique interchangeable head system
- Ø12/Ø16
- Shaft extensions in steel / carbide / heavy metal
- Profile accuracy: +/- 10 µm, exchange accuracy: +/- 20µm

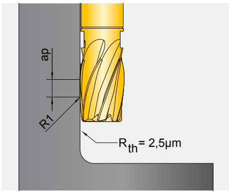
Calculation of Path Distance ap

Ball nose end mill



Calculation of path distance ap at desired roughness depth R_{th} 2,5 µ	
R_{th}:	0,0025 mm
Ø / R :	Ø 12 / r= 6
Formel:	$b_r = 2x\sqrt{R_{th}\,(D - R_{th})}$
ap:	0,35 mm

Barrel-form end mill



Calculation of path distance ap at desired roughness depth R_{th} 2,5 µ	
R_{th}:	0,0025 mm
Ø / R :	Ø 12 / R1= 70
Formel:	$b_r = 2x\sqrt{R_{th}\,((DR1 \text{ x } 2) - R_{th}}$
ap:	1,18mm

Comparison of Path Distance Ball Nose End Mill vs. Barrel-form End Mill

Path distance ap at roughness depth R_{th} 2,5µ:

ball nose end mill		
Ø [mm]	r [mm]	ap [mm]
12	6	0,35
16	8	0,40

barrel-form end mill		
Ø [mm]	r [mm]	ap [mm]
12	25	1,18
16	35	1,41

Path distance ap at roughness depth R_{th} 5,0µ:

ball nose end mill		
Ø [mm]	r [mm]	ap [mm]
12	6	0,49
16	8	0,57

barrel-form end mill		
Ø [mm]	r [mm]	ap [mm]
12	25	1,67
16	35	2,00

Path distance ap at roughness depth R_{th} 10,0µ:

ball nose end mill		
Ø [mm]	r [mm]	ap [mm]
12	6	0,69
16	8	0,80

barrel-form end mill		
Ø [mm]	r [mm]	ap [mm]
12	25	2,37
16	35	2,83

HIPOSPRO^B COPY MILL 12L8J

With the finishing cutters of the HiPosProB series, semi-finishing and finishing applications on 90 ° sides or bevels, where traditional round insert tools were previously used, are now carried out more effectively and productively. Thanks to the special barrel shape of the insert cutting edge, higher path distances and feed rates can be achieved, with at least the same surface quality.

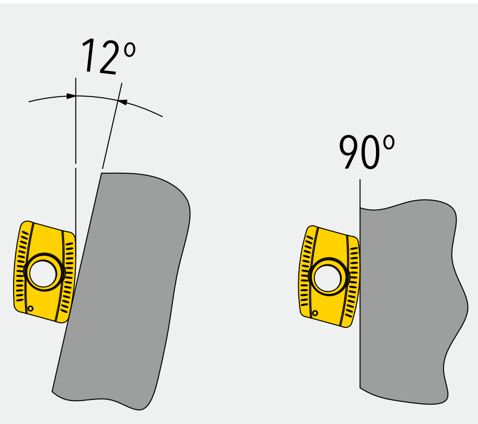
The new series is offered as a screw-in type with metric thread connection in diameters Ø16 /20/25/35/42 and as well proven high-precision type with a Ts-adaption in diameters Ø16 /20/25. On customer's request, diameters and tool holder types that are not listed can also be manufactured as semi-standard tools.

Application Range

Especially suited for die and mold industry, but also across industries for general mechanical engineering and aerospace industry.

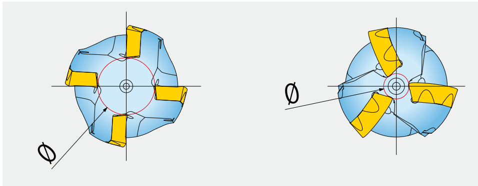
Thanks to the inclined insert seat in the tool body, a wide range of different mold inclines of up to 12 ° can be covered on workpieces without additionally adjusting the spindle axis.

Due to the availability of different cutting material grades, the common material groups P/M/K can be machined as well as hardened material group link H up to 54HRC and difficult-to-machine materials of material group S.

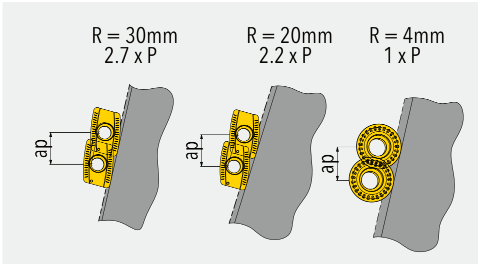


Technical Features

- Stable tool core compared to classic round insert tools
- Finer pitch



- Up to 2.7 times of infeed/path distance compared to round insert tools with the same theoretical roughness depth Rth



Advantages

- Barrel-shaped finishing insert
- 2 radius sizes and 3 grades each
- Large path distances = short machining times
- Finishing on 90 ° shoulders and mold inclines up to 12 ° without additional spindle adjustment
- Up to 2.7 times higher productivity compared to round insert tools
- Stable tool core compared to round insert tools
- Tool extensions in steel, hard metal and heavy metal

Note

Thread milling

Comparison of thread mill and tap drill geometry

- Thread mills do not have a continuous pitch (spiral).
- The profile teeth are arranged behind each other.
- The pitch is established by the tool-machine combination.

Advantages of thread mills

- Exact thread depth
- No thread run-out
- Low cutting pressure, difficult to machine workpiece materials, process stability.
- Reversing tool spindles not needed
- Thin-walled parts can be produced

Calculation of the correct thread mill diameter

1. Check the max. permissible profile distortion:

$$\Delta L = \frac{P^2 \times r}{8 \times \pi^2 \times C \times x(R-r) \times R}$$

2. Determine the max. possible tool diameter D_C:

$$D_C = 2 \times \left[\frac{\Delta L \times 8 \times \pi^2 \times C \times R^2}{\Delta L \times 8 \times \pi^2 \times C \times R \times P^2} \right]$$

3. Determine the minimum core hole diameter D2 based on the specified tool diameter:

$$D_2 = 2 \times \left[\frac{r}{2} + \frac{r^2}{4} + \frac{P^2 \times r}{8 \times \pi^2 \times C \times \Delta L} \right]$$

Process reliability

- Thread straightness is guaranteed
- No chip problems
- No eroding out of broken tools necessary

Reduction of down times

One tool for:

- Different thread diameters with the same pitch
- Blind hole and through holes
- Right- and left-hand threads
- Various thread tolerances (6G, 6H, …)

Max. permissible profile distortion	ΔL	=max. 0,02mm
tool diameter	D_C	
flank angle	w	
$\tan \varphi = C$	C	
pitch	P	
core hole diameter	D_2	
$0{,}2 \times D_2$	R	
profile height	h	
$0{,}2 \times D_2 \times h$	r	

Appraisal

For coarse threads the reference value for the thread mill diameter is max. 2/3 of the hole diameter.

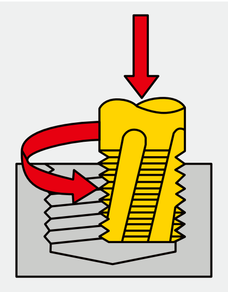
Example: M20 internal thread
20 mm x 0.66 = 13.2 mm (Ø14 mm thread mill)

For fine pitch threads the factor 0.75 must be taken as a reference value.

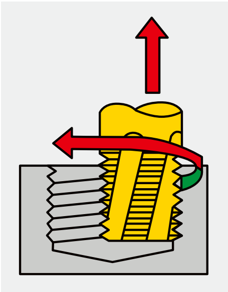
Example: M20x1,5 internal thread
20 mm x 0.75 = 15 mm (Ø15 mm thread mill)

Thread production with one tool

Left-hand thread

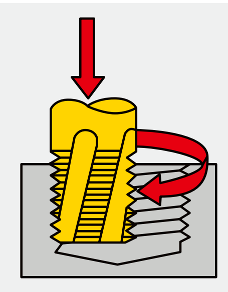


from top to bottom, climb milling

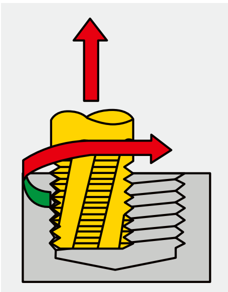


from bottom to top, conventional milling

Right-hand thread



from top to bottom, conventional milling



to bottom to top, climb milling

Axiale and radial infeeds during the thread milling process

In general, the thread milling process should be conducted in a single infeed (productivity). Certain parameters, however, require that the thread is milled in several passes.

More radial infeeds for:

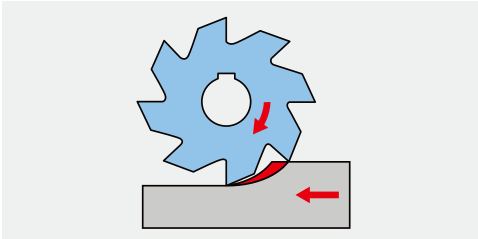
- Large overhand of the tool (radial deflection)
- Tapered (conical) thread
- Thin-walled parts
- Hardened materials
- Difficult to machine materials (austenite, inconel, titanium)

More axial infeeds for:

- Long threads (thread depth greater than the cutting edge length)
- Too much cutting pressure

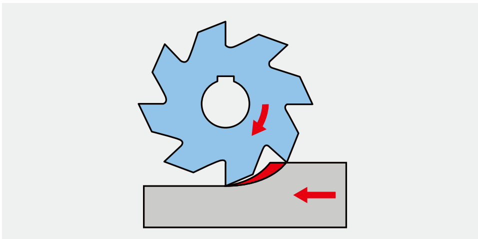
Advantage of climb milling:

- Thread mill enters the cut immediately
- Protects the cutting edge against edge break-outs and flank wear



Advantages of conventional milling

- Better for filigree parts
- If the thread becomes conical with climb milling
- For smaller thread mill diameters



Important formulae

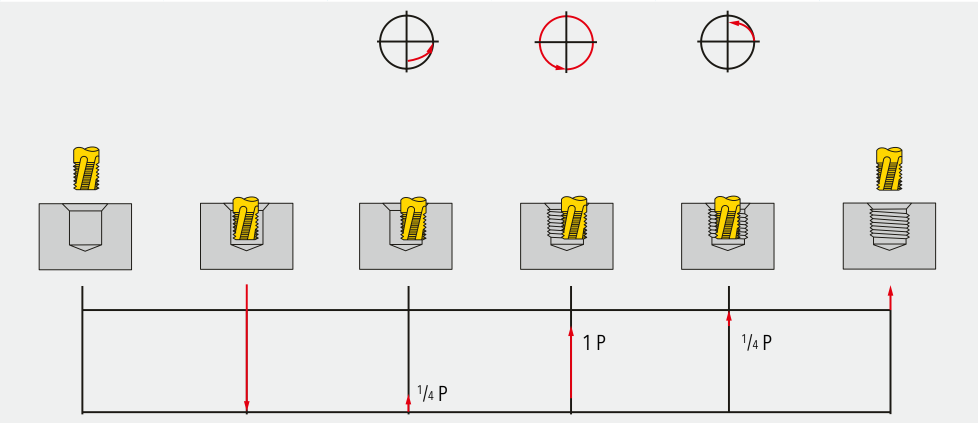
Variable	Unit	Formula
Speed of rotation	min^{-1}	$n = \frac{v_c \times 1000}{D \times \pi}$
Cutting speed	m/min	$v_c = \frac{D \times \pi \times n}{1000}$
Feed rate	mm/min	$v_f = f_z \times Z_{eff} \times n$
Feed per tooth	mm	$f_z = \frac{v_f}{Z_{eff} \times n}$
Average chip thickness	mm	$h_m = f_z \times \sqrt{a_e/D}$
Main utilization time	min	$t_h = \frac{L \times i}{V_f}$

Thread milling

Thread milling processes

Internal thread

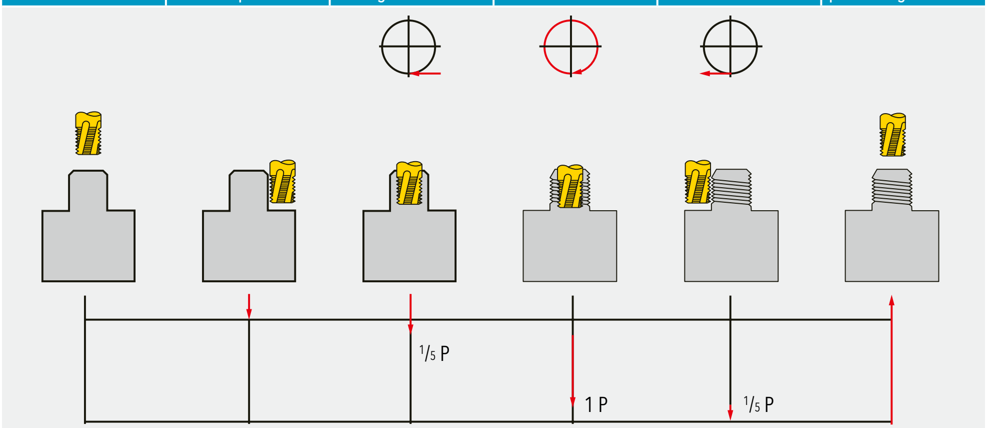
Positioning	Plunge to thread depth	Approach radius for thread milling	Thread milling	Departing radius	Retraction to positioning level



NOTE: A tapered (conical) internal thread is produced as shown above, on the hole is pre-machined conically.

External thread

Positioning	Plunge to thread depth	Approach for milling the thread	Thread milling	Move away	Retraction to positioning level



Correct tool clamping

The teeth of a thread mill are very filigree and delicate, therefore good concentricity (true running) of the thread mill is a must for producing threads.

Suitable clamping means are:

- Shrinking
- Hydro-expansion chuck
- Collet chuck (use collets with better true running 5µm)
  To achieve a good tool life, the true running accuracy should be within a range of 3 - 8 µm.

When milling threads attention must be paid to which path is being used for feed rate calculation purposes. Just like with circular interpolation milling, differentiation is made between countour path or midpoint path feedrate. This depends on the machine or its controller.

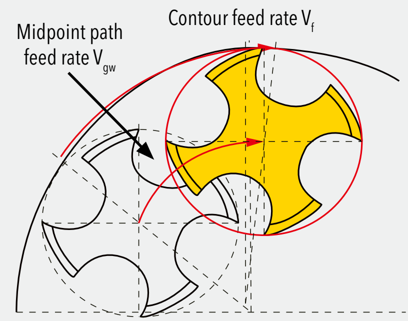
D	Thread diameter [mm]
d	Milling cutter diameter [mm]
V_f	Feed rate [mm]
V_{an}	Midpoint path feed rate for approach [mm/min]
V_{gw}	Midpoint path feed rate [mm/min]

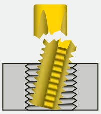
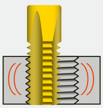
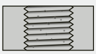
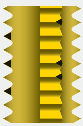
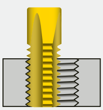
Midpoint path feed rate calculation V_{gw}:

$$V_{gw} = \frac{Vf(D-d)}{D}$$

Midpoint path calculation for moving into the workpiece V_{an}:

$$V_{gw} = \frac{Vf(D-d)}{D+d}$$



Fault		Cause	Remedy
Tool breakage		• Feed rate too high • Stability of the machine • Chip jam • Inclusions in the part • High tool wear	• Reduce the feed rate • Reduce the cutting speed • Change the strategy • Adjust the feed rate • Change the tool sooner
Vibrations		• Cutting speed too high • Feed rate too low • Stability of the machine • Overhang too big (thread depth)	• Adjust the cutting speed • Increase the feed rate • Adjust the cutting parameters • Reduce the cutting speed
Poor surface quality		• Cutting speed too high/low • Feed rate to high/low • Wrong lubrication selected or no lubrication	• Adjust the cutting speed • Adjust / Recalculate the feed rate
Flank wear		• Cutting speed too high • Feed rate too low • Runout error • Wrong lubrication selected or no lubrication	• Reduce the cutting speed • Increase the feed rate • Check the runout • Use a suitable cooling lubricant
Cutting edge breakage		• Feed rate too high • Stability of the machine • Chip jam	• Reduce the feed rate • Adjust the cutting parameters • Check the coolant or tool type
Conical thread		• Feed rate too high • Wrong strategy • Tool clamping stability • Tool not strong enough	• Reduce the feed rate • Check the strategy • Use suitable clamping means • Choose the most stable tool

QUADDRILL+

Application:

- Medium to large bore diameter
- For short hole depths up to 5xD
- For average tolerance requirements
- Relatively flat bore bottom
- High stock removal rate
- Easy adaptation of the cutting material
- Boring application

Diameter range:

QuadDrillPlus indexable drills:

- Diameter range 2xD: Ø 13–50 mm
- Diameter range 2,5xD: Ø 51–80 mm
- Diameter range 3xD: Ø 12,5–60 mm
- Diameter range 3,5xD: Ø 51–80 mm
- Diameter range 4xD: Ø 13–50 mm
- Diameter range 5xD: Ø 13–41 mm

QuadDrillPlus plunge mill:

- Diameter range Weldon version: Ø 10–48 mm
- Diameter range screw-in version: Ø 15–48 mm

QuadDrillPlus drill/plunge mill:

- Diameter range Weldon version: Ø 16–40 mm

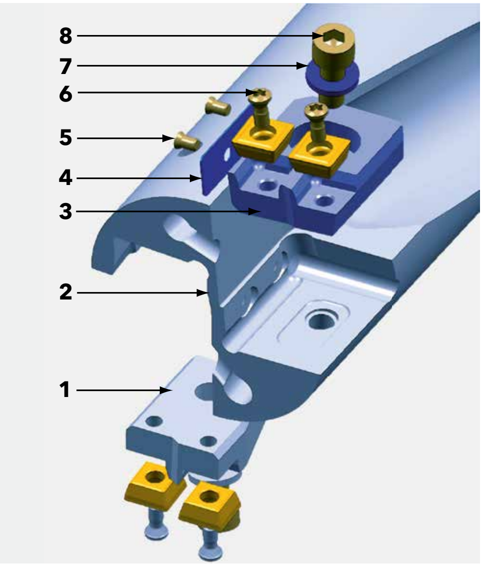
Diameters achievable by radial adjustment of the drill		
2xD / 3xD / 4xD		
drill Ø	max. rad. adjustment	max. drill Ø
13	0,5	~14,0
14	0,5	~15,0
15	0,5	~16,0
16	0,5	~17,0
17	0,5	~18,0
18	0,5	~19,0
19	0,5	~20,0
20	0,5	~21,0
21	0,25	~21,5
22	0,5	~23,0
23	0,5	~24,0
24	0,5	~25,0
25	0,5	~26,0
26	0,25	~26,5
27	0,25	~27,5
28	0,5	~29,0
29	0,5	~30,0
30	0,5	~31,0
31	0,25	~31,5
32	0,25	~32,5
33	0,25	~33,5
34	0,5	~35,0
35	0,5	~36,0
36	0,5	~37,0
37	0,5	~38,0
38	0,5	~39,0
39	0,5	~40,0
40	0,25	~40,5
41	0,25	~41,5
42	0,5	~43,0
43	0,5	~44,0
44	0,5	~45,0
45	0,5	~46,0
46	0,5	~47,0
47	0,5	~48,0
48	0,25	~48,5
49	0,25	~49,5
50	0,25	~50,5

To achieve the best performance and highest productivity, select the shortest possible drill

Cartridge drill:

Each indexable cartridge drill covers a certain diameter range. This can be changed by adding and removing setting plates [4].

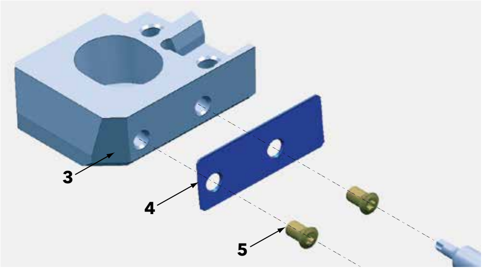
This option makes it possible to drill multiple diameters with one main body [2]. The diameter accuracy can be improved even more by changing the setting plate thickness.



No.	Item	No.	Item
1	Center cartridge	5	Clamping screw setting plate
2	Drill body	6	Insert screw
3	Peripheral cartridge	7	Washer
4	Setting plate	8	Clamping screw cartridge

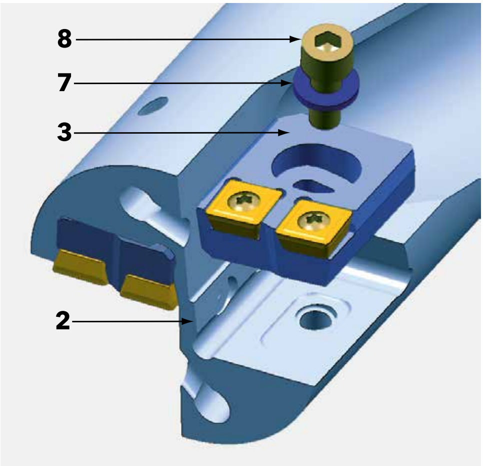
Assembly of setting plate:

- Measure the correct setting plate [4]: 0,5 mm thickness = 1 mm bigger diameter
  Note: The smallest possible drilling diameter is achieved without a setting plate!
- Screw the setting plate [4] onto the peripheral cartridge [3] with the clamping screws [5]. Pay attention to the hole center distance!
  Attention: The setting plate must not protrude after it has been installed!



Assembly of cartridge:

- Clean cartridge seats and cartridge [3].
- Place the cartridge [3] in the cartridge seat and press it against the contact areas.
- Slide the cartridge clamping screw [5] through the washer [7], then through the through hole in the cartridge [3] into the thread of the main body [2].
- Tighten the cartridge clamping screw [8].



QUADTWIST

The QuadTwist drill uses a 4-edged insert that can be used in both center and peripheral pockets. This provides for effi cient stock keeping, thus reducing costs.

Furthermore, the unique insert design ensures ideal chip control and reduction of the cutting force. The effi cient, twisted fl utes and twisted, internal coolant supply ensure not only reliable chip evacuation during the drilling process, but also very good drilled hole surfaces.

Optimally arranged inserts provide for very good drilling results in carbon and alloyed steels, as well as in diffi cult to machine materials, including steel with low carbon content and structural steel.

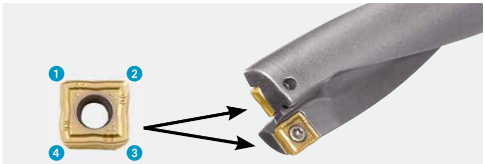
The QuadTwist product line covers a diameter range of **Ø 14.0 to Ø 50.0 mm** and is available in the lengths **2xD, 3xD, 4xD** and **5xD**.

Available tools:

- in 1 mm increments in the lengths 2xD, 4xD and 5xD
- in 0.5 mm increments from Ø14 to Ø30 in the length 3xD.

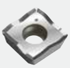
Inserts:

- Economical, 4-edged insert
- Same insert can be used for center and peripheral pockets



Application features:

The Ingersoll QuadTwist drill series uses an economical, 4-edged insert which, in grades IN2505 or IN2530, can be used both as central insert and peripheral insert.

Coating	Picture	Desription
IN2530 (PVD)		• Tough carbide • For interrupted cuts and for break-outs in the center • For medium cutting speeds
IN2505 (PVD)		• First choice for general applications • Multi-layer coating • For medium to high cutting speeds
IN2510 (PVD)		• Greater wear resistance • Optimal drilling results in cast iron
IN6505 (CVD) (black surface)		• Exceptionally wear-resistant sub-micron carbide • For drilling in steel • Can only be used in peripheral pockets • For high cutting speeds
IN10K (Carbide)		• Excellent for drilling in aluminum • Sharp cutting edge • Reduced cutting forces at high cutting speeds

Chipping space:

- Twisted coolant supply with large chip flutes
- Ideal chip evacuation thanks to optimized chip flutes

Notes:

- The data given here are reference values only and may differ in your particular application.
- The optimal chip shape is produced by varying the cutting speed and feed rate.
- Refer to the catalog for information about the number of cutting edges (Z_{eff}).
- When the drill exits the workpiece, a disc is produced, which can be thrown out by rotating workpieces. **Necessary precautions must therefore be taken!**
- It is advisable to pre-calculate the required machine output and to synchronize it with the machine's actually available drive power.
- For drills with L/D = 5 use the lower feed rate; for drilling starter holes decrease to up to 50% of the lower feed rate.
- Ensure that adequate internal coolant is supplied to ensure reliable chip evacuation.

Eccentric Sleeve

Eccentric Sleeve for reducing or enlarging nominal drilling diameters.

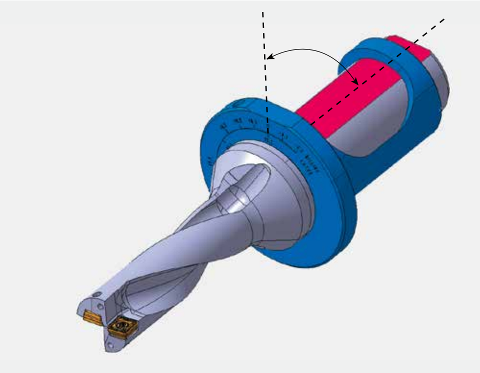
The sleeve can be used on machining centres, lathes and even misaligned turning centres. The eccentric sleeves come in four different standards – 20, 25, 32, and 40. They can be used for enlarging the nominal drilling diameter to max. +0.4mm and reducing the diameter to -0.2mm on milling machines. When used on lathes a maximum enlargement of +0.2mm can be achieved.

When using on a milling machine, refer to the "MILLING" scale on the frontal flange of the Eccentric sleeve. When using on a lathe, refer to the "LATHE" scale.

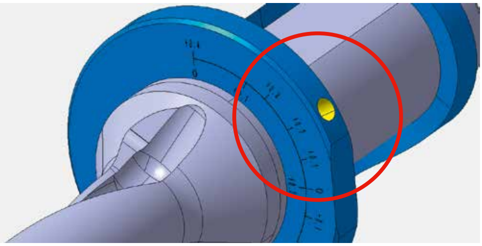
Please read the instructions thoroughly before using Ingersoll's Eccentric sleeves.

Adjustment

During initial setting ensure the flat on the eccentric sleeve coincides with the flat on the drill shank. (Both planes must be in parallel condition to each other)

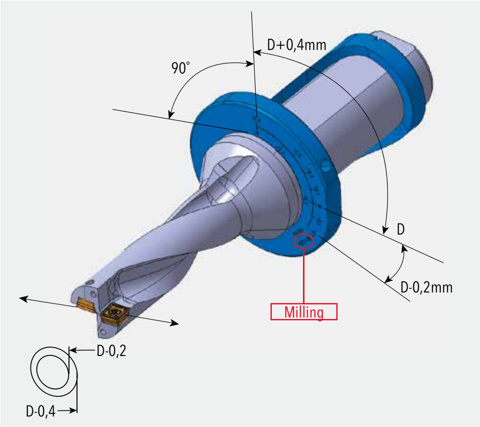


To facilitate the rotation of the sleeve, a metal rod or a screw key may be inserted into the hole on the Eccentric sleeve flange. Unlock adapter screw before adjusting sleeve.



Milling application

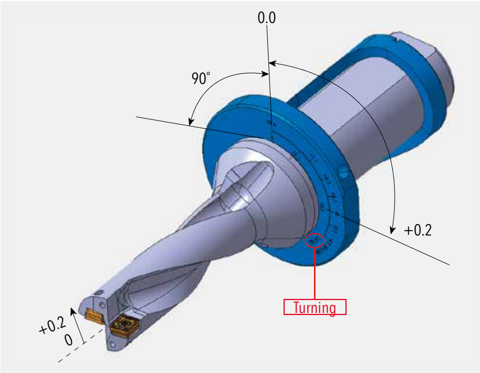
On a milling machine the sleeve can change the drill's nominal diameter by shifting the drill's axis out of the tool spindle.



To enlarge the diameter, turn the sleeve clockwise & to reduce the diameter, turn sleeve counterclockwise as shown.

Lathe application

On a lathe, the Eccentric sleeve can shift the drill's axis to coincide with the spindle axis.



The eccentric sleeve enables the user to align the drill's axis with the spindle axis to within 0.2mm range (turn the sleeve counter clock wise to raise it).

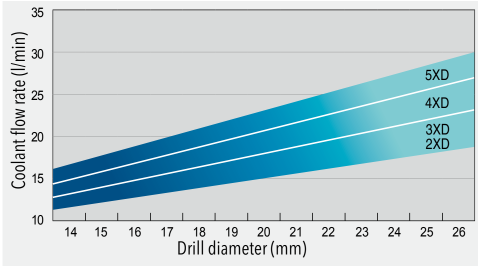
Drilling

Possible diameters achievable by radial adjustment of the drill		
2xD / 3xD / 4xD		
Drill Ø	max. rad. adjustment	max. drill Ø
12	+0,4	12,8
13	+0,3	13,6
14	+0,5	15,0
15	+0,4	15,8
16	+0,3	16,6
17	+0,5	18,0
18	+0,4	18,8
19	+0,3	19,6
20	+0,5	21,0
21	+0,4	21,8
22	+0,3	22,6
23	+0,5	24,0
24	+0,5	25,0
25	+0,4	25,8
26	+0,3	26,6
27	+0,5	28,0
28	+0,5	29,0
29	+0,5	30,0
30	+0,5	31,0
32	+0,5	33,0
31	+0,3	31,6
33	+0,5	34,0
34	+0,5	35,0
35	+0,5	36,0
36	+0,4	36,8
37	+0,5	38,0
38	+0,5	39,0
39	+0,5	40,0
40	+0,5	41,0
41	+0,5	42,0
42	+0,5	43,0
43	+0,5	44,0
44	+0,5	45,0
45	+0,5	46,0
46	+0,5	47,0
47	+0,5	48,0
48	+0,5	49,0
49	+0,5	50,0
50	+0,5	51,0

SAFETY WARNING:

When the drill exits the workpiece, a disc drops off. Therefore there is a risk of accident in assocation with rotating workpieces! Necessary precautions must therefore be taken!

Coolant:



Important:

Always provide adequate coolant for the internal coolant supply. Minimum pressure 8 - 10 bar.

Formulae

Variable	Unit	Formula
Machine rating	kW	$Pa = \frac{v_c \times f \times D \times kc}{1000 \times 60 \times 4 \times \pi}$
Feed force:	N	$Ff = 0{,}7 \times \frac{D}{2} \times f \times kc$

Pa = Machine output in kW

kc = Specific cutting force in N/mm²

D = Diameter in mm

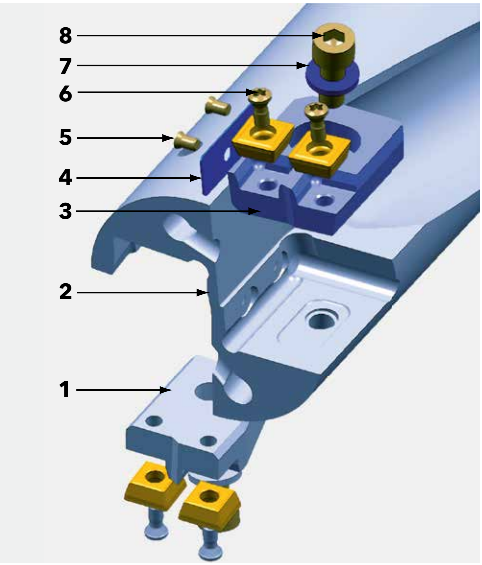
Vc = Cutting speed in m/min

η = Machine efficiency 0,7 – 0,8

Cartridge drill:

Each indexable cartridge drill covers a certain diameter range. This can be changed by adding and removing setting plates [4].

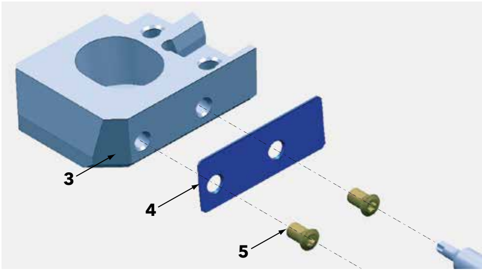
This option makes it possible to drill multiple diameters with one main body [2]. The diameter accuracy can be improved even more by changing the setting plate thickness.



No.	Item	No.	Item
1	Center cartridge	5	Clamping screw setting plate
2	Drill body	6	Insert screw
3	Peripheral cartridge	7	Washer
4	Setting plate	8	Clamping screw cartridge

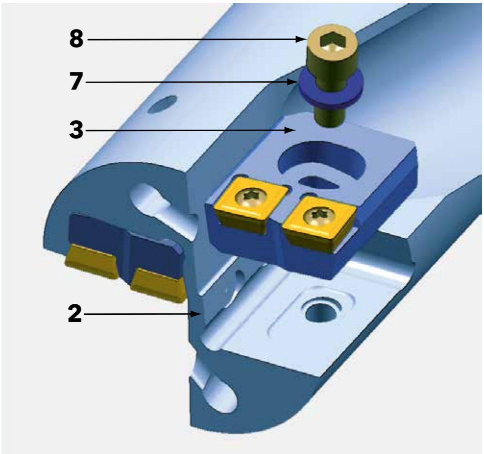
Assembly of setting plate:

- Measure the correct setting plate [4]: 0,5 mm thickness = 1 mm bigger diameter
  Note: The smallest possible drilling diameter is achieved without a setting plate!
- Screw the setting plate [4] onto the peripheral cartridge [3] with the clamping screws [5]. Pay attention to the hole center distance!
  Attention: The setting plate must not protrude after it has been installed!



Assembly of cartridge:

- Clean cartridge seats and cartridge [3].
- Place the cartridge [3] in the cartridge seat and press it against the contact areas.
- Slide the cartridge clamping screw [5] through the washer [7], then through the through hole in the cartridge [3] into the thread of the main body [2].
- Tighten the cartridge clamping screw [8].



GOLDTWIN

Product overview

Ingersoll expands its BoreLine with the new **GoldTwin drill**, a combined interchangeable head and insert drilling system for holes with larger diameters.

The **GoldTwin drilling system** combines 2 types of drills, for one thing the interchangeable head system with solid carbide head and for another thing the indexable insert drilling system. The solid carbide head garantees an excellent self centering of the drill while the 4-edged insert with wiper-geometry improves the surface quality.

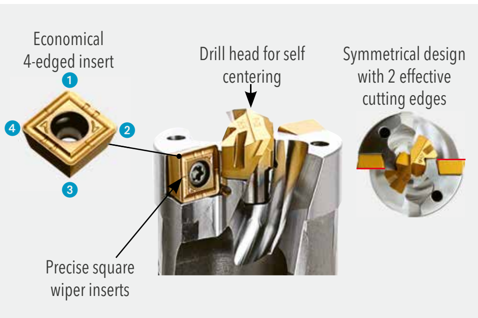
This combination gives an economic drilling system with 2 effective cutting edges for highest cutting performance. The drill body is designed with internal coolant supply, a hardened body for improved rigidity and induces high wear resistance due to the special surface post-treatment process.

The standard **GoldTwin drilling system** is available in diameter ranges from Ø 26 mm to Ø 45 mm in 1 mm steps and 5xD length. The solid carbide heads and inserts are provided in the established, high wear-resistant grade IN2505.

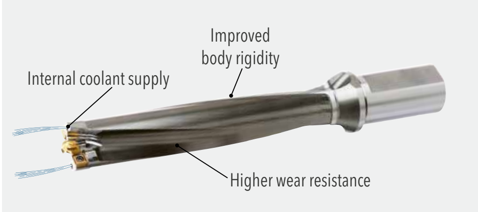
Advantages:

- 2 effective inserts for high cutting performance
- Precise 4-edge insert with wiper geometry
- High hole accuracy and excellent surface quality
- Excellent chip control
- Improved body rigidity
- Very economical solution
- Drill diameter: Ø 26 mm up to Ø 45 mm / drilling depth 5xD

Construction of drill bit:



Technical Informations:



GOLDTWIN Modular drill system

Product overview

The modular type GoldTwin line provides superior machining and excellent hole quality for hole machining of 26 mm diameter and above, as well as improved productivity due to the symmetric design which reduces tool cost, yet expands the range of GoldTwin applications.

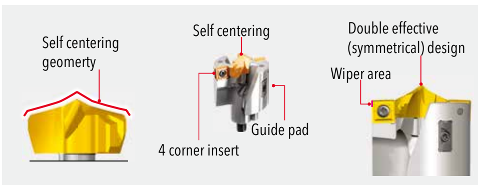
Advantages:

- Unique cutting edge design improves self-centering and hole straightness
- TPAxxxxR01-C will be phased out when stock is sold-out
- Excellent surface roughness due to an insert with a wiper area
- Improved machining stability due to its dedicated guide pads
- Modular body changes enable 3xD and 5xD machining

Technical features

By assembling a modular holder that fits the depth of the hole into a single modular head, the new offering is able to maintain high-feed machining and quality. It also reduces tool cost by only requiring the modular head be removed when changing the tool.

In the center of the modular head, a new uniquely designed head (TPCxxxxR01-C) has been applied with a self-centering capability to improve the hole straightness. In addition, the exterior inserts with wiper (SPGX-WG) and dedicated guide pads provide excellent surface roughness.



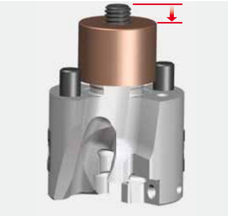
Modular head replacement instructions

1. Remove both outer inserts, then remove the center drill head. (When clamping, go in the reverse order)

2. Use a wrench to turn the screw counter-clock-wise to remove the modular head.

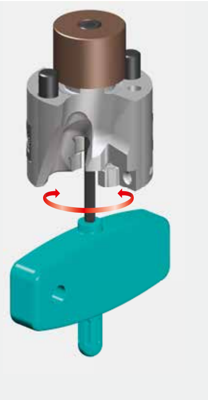
3. Insert the setting gauge into the bottom of the disconnected modular head.

4. Rotate the screw to adjust to the same height as the setting gauge.

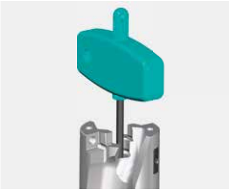


Setting gauge

Drill dia.	Designation
D26-D29	SG CD26-29-TP
D30-D35	SG CD30-35-TP
D36-D39	SG CD36-39-TP
D40-D43	SG CD40-43-TP
D44-D50	SG CD44-50-TP



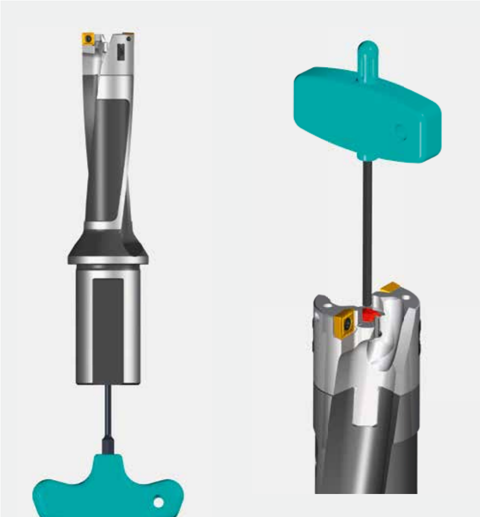
5. Remove the height adjusted modular head from the setting gauge and attach it to the holder.



Modular head disassembly in the event of center drill damage

If the modular head cannot be unclamped due to center drill damage, insert the wrench into the rear section of the shank. Then, turn it clock-wise to disassemble the modular head.

Disassembling wrench and handle are included with the modular drill body.

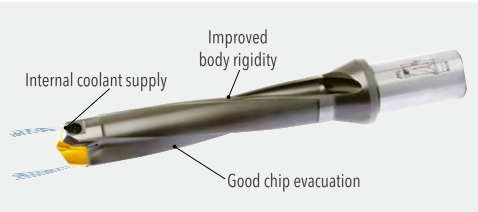


SpadeTwist

Product overview

The SpadeTwist drill with optimized cutting edge geometry and unique, very robust clamping system provides the greatest productivity and outstanding performance. The unique clamping technology allows the drill head to be changed quickly, without having to fi rst remove the clamping screw. This reduces the tool setup time and also the machine down times.

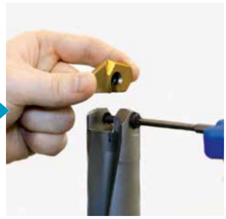
The SpadeTwist drill line is available as a standard tool in the length-to-diameter ratios of 3xD and 5xD in a diameter range of 26.0 mm to 41.0 mm in 0.5 mm increments.



Product features:

- Unique, quick-change clamping system
- 2 Effective cutting edges for the greatest productivity
- Self-centering geometry
- Easy change of the drill head at the machine
- Asymmetrical insert pocket for error-free mounting, high precision and excellent surface fi nish
- High rigidity of the unique clamping system means greater productivity
- Internal coolant supply
- Diameter range:
  Ø 26.9 mm to 41.0 mm in 0.5 mm increments
- Length-to-diameter ratio of the tool body:
  3xD and 5xD
- Semi-standard tools available on request

To unclamp, turn the clamping screw 3 to 5 turns counterclockwise



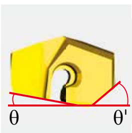
No need to remove the screw from the shank body

Drill head

The asymmetrical pocket design guarantees error-free mounting of the drill head, thus increasing the tool precision and tool change accuracy.

Wide contact area

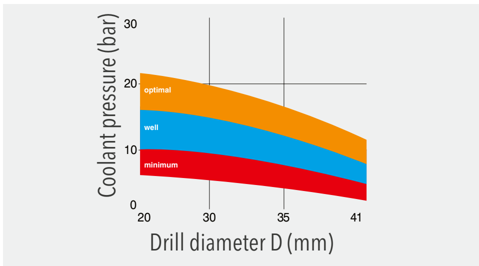
Standard helix angle

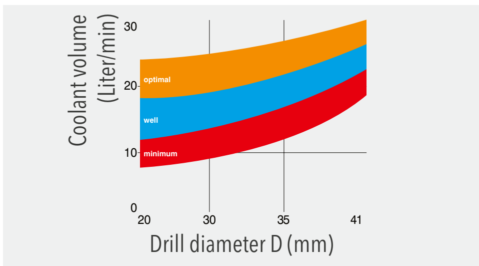


asymmetrical insert pocket

error-free mounting

Recommended coolant pressure and coolant volumes





Drilling

General information

The Ingersoll Gold-Twist product line is the next generation of interchangeable head drilling systems. The new design is a standard product line and covers a diameter range of Ø 7.0 mm to Ø 25.9 mm in increments of 0.1 mm and is available in length/diameter (L/D) ratios of 1.5xD, 3xD, 5xD and 8xD. The L/D ratio of 12xD is available in the diameter range Ø12 mm to Ø 22.9 mm.

Product Description

The precision drill bodies have an improved interface with an innovative clamping system, which guarantees reliable clamping even after many changes of the tips. These drill bodies also offer twisted coolant holes, polished flutes and a PVD coating, which ensures easy chip evacuation and extends the serviceable life of the body. Depending on its diameter, each body can cover a diameter range of between 0.5 mm - 1 mm.

The IN2505 grade, solid carbide, interchangeable heads provide an excellent PVD coating, improved wear resistance and longer serviceable life for many different applications. Cutting edge geometries are available for machining general purpose steel (TPA), for machining cast iron (TKA) and for machining stainless steels (TMA).

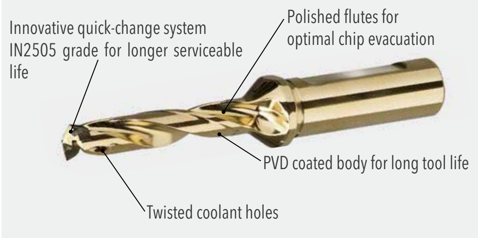
Application features

The Ingersoll Gold-Twist drill line provides excellent results in applications with higher cutting speeds and offers a unique, robust, quick-change clamping system. The result is a product for your drilling application, which combines cost efficiency with a high productivity.

Product features

- The precision drill bodies have an innovative clamping system with excellent stability to guarantee reliable clamping.
- The system is characterized by its ease of use.
- The drill bodies have twisted coolant holes, polished flutes and a PVD coating, which guarantees good chip evacuation and a long serviceable life of the body.
- Depending on its diameter, each body can cover a diameter range of between 0.5 mm - 1 mm.
- The IN2505 grade, solid carbide, interchangeable drill heads are offered in 0.1 mm increments in P, M, and K geometries with an excellent PVD coating. This ensures a high wear resistance and longer serviceable life for many applications.

Technical information



Depending on its diameter, each body covers a diameter range of between 0.5 mm - 1 mm.

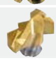
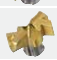
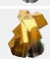
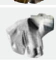
Advantages

- High productivity
- Exceptionally economical
- Innovative drill head clamping system
- Twisted and polished flutes
- Twisted coolant holes
- Excellent chip evacuation

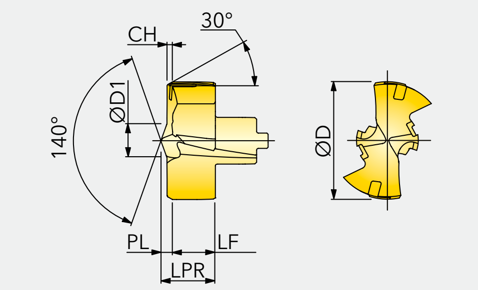
Notes & tips

- When the drill exits the workpiece, a disc is produced which can be thrown out by rotating workpieces.
  Please take suitable precautions!
- When drilling through holes, make sure that the drill tip remains guided when the drill exits the workpiece.
- For drills with L/D = 8 and 12, use the lower feed rate; for drilling starter holes, decrease to up to 50% of the lower feed rate.
- For drills with L/D = 8 and 12, use a pilot hole, if necessary

Application of drill heads

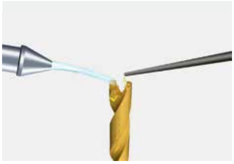
Drill head	Picture	Geometry for
TPA_R01		general purpose steel
TMA_R01		stainless steel
TKA_R01		cast iron
TPF_R01		flat hole bottom
TPA_R01-M2		double guide chamfer
TNA_R01		aluminum

Drill head dimensions for a level hole bottom

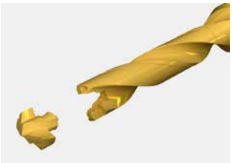


Designation	D	D1	LPR	t	PL	LF	CH	Bs
TPF0800R01	8	1,2	4,4	4	1,09	3,3	0,7	8
TPF0850R01	8,5	1,2	4,4	4	1,09	3,3	0,7	8
TPF0900R01	9	1,21	4,6	4,2	1,11	3,5	0,7	9
TPF0950R01	9,5	1,21	4,6	4,2	1,11	3,5	0,7	9
TPF1000R01	10	1,27	4,9	4,4	1,17	3,7	0,7	10
TPF1050R01	10,5	1,27	4,9	4,4	1,17	3,7	0,7	10
TPF1100R01	11	1,49	5,1	4,5	1,25	3,8	0,7	11
TPF1150R01	11,5	1,49	5,1	4,5	1,25	3,8	0,7	11
TPF1200R01	12	1,5	5,4	4,8	1,26	4,1	0,7	12
TPF1250R01	12,5	1,5	5,4	4,8	1,26	4,1	0,7	12
TPF1300R01	13	1,64	5,7	5,1	1,3	4,4	0,7	13
TPF1350R01	13,5	1,64	5,7	5,1	1,3	4,4	0,7	13
TPF1400R01	14	1,68	6,1	5,5	1,31	4,8	0,7	14
TPF1450R01	14,5	1,68	6,1	5,5	1,31	4,8	0,7	14
TPF1500R01	15	1,78	6,6	5,9	1,35	5,23	0,7	15
TPF1550R01	15,5	1,78	6,6	5,9	1,35	5,23	0,7	15
TPF1600R01	16	1,89	7	6,3	1,39	5,6	0,7	16
TPF1650R01	16,5	1,89	7	6,3	1,39	5,6	0,7	16
TPF1700R01	17	1,91	7,3	6,6	1,4	5,9	0,7	17
TPF1750R01	17,5	1,91	7,3	6,6	1,4	5,9	0,7	17
TPF1800R01	18	1,97	7,6	6,9	1,42	6,18	0,7	18
TPF1850R01	18,5	1,97	7,6	6,9	1,42	6,18	0,7	18
TPF1900R01	19	1,96	7,9	7,2	1,44	6,5	0,7	19
TPF1950R01	19,5	1,96	7,9	7,2	1,44	6,5	0,7	19
TPF2000R01	20	3,42	9,3	8,2	1,77	7,5	0,7	20
TPF2050R01	20,5	3,42	9,3	8,2	1,77	7,5	0,7	20
TPF2100R01	21	3,6	9,7	8,6	1,79	7,9	0,7	21
TPF2150R01	21,5	3,6	9,7	8,6	1,79	7,9	0,7	21
TPF2200R01	22	3,8	10	8,9	1,81	8,2	0,7	22
TPF2250R01	22,5	3,8	10	8,9	1,81	8,2	0,7	22
TPF2300R01	23	3,9	10,4	9,3	1,83	8,6	0,7	23
TPF2350R01	23,5	3,9	10,4	9,3	1,83	8,6	0,7	23
TPF2400R01	24	4,1	10,9	9,7	1,86	9	0,7	24
TPF2450R01	24,5	4,1	10,9	9,7	1,86	9	0,7	24
TPF2500R01	25	4,3	11,3	10,1	1,89	9,4	0,7	25
TPF2550R01	25,5	4,3	11,3	10,1	1,89	9,4	0,7	25

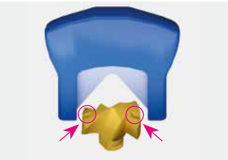
Mounting of drill heads



1. Clean the pocket and apply oil.



2. Mount the drill head on the pocket

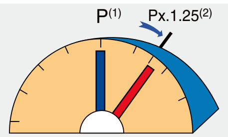


3. Insert the keys into the slots at the drill head

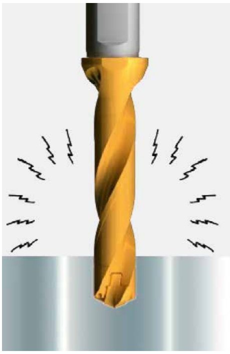
3. Secure the drill head into place by turning clockwise

Types of wear

Power limit

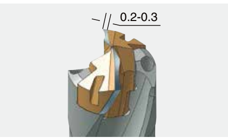


(1) new drill head
(2) worn drill head

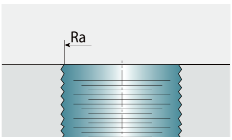


Vibrations or noise increase dramatically

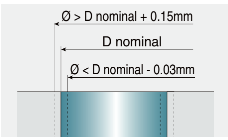
Wear limit



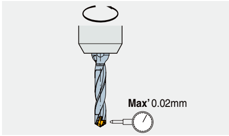
Surface deterioration



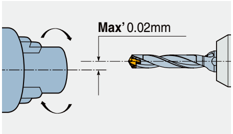
Change in diameter



Max. runout error



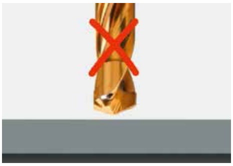
Rotational runout



Stationary runout

Drilling limitation

Inclined surface

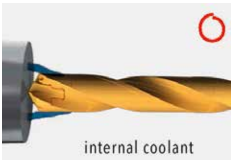


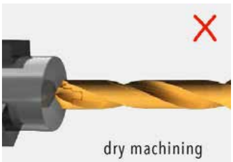
Cross hole

Interrupted cut

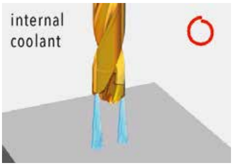
Coolant recommendations

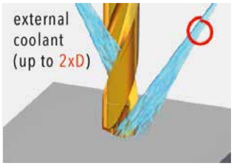
Lathes

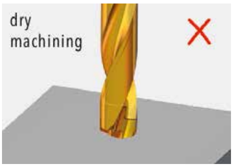




Machining center



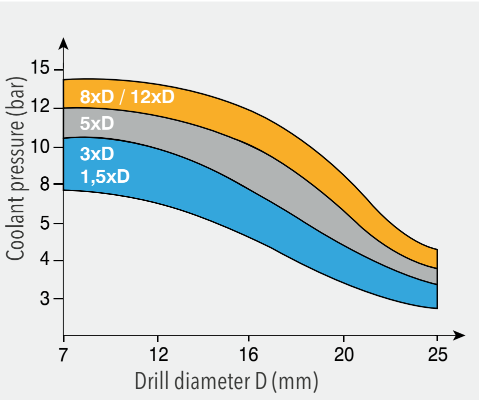




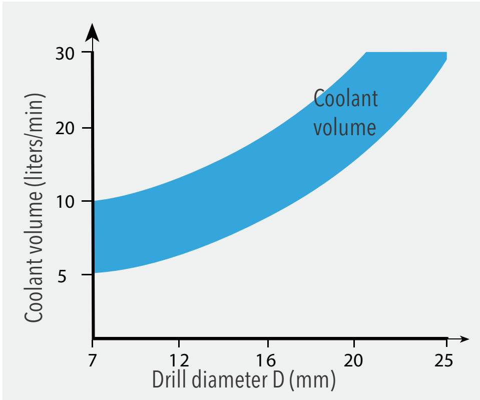
GOLDTWIST

Coolant supply

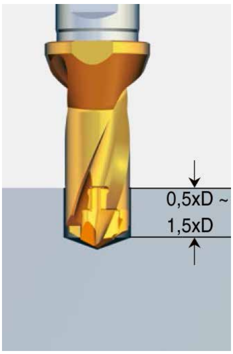
Recommended coolant pressure (bar)



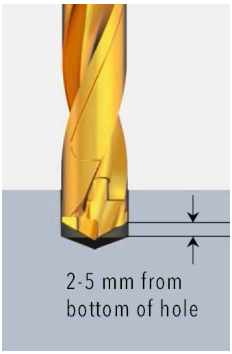
Recommended coolant volume (liters/min)



Recommended procedure for long drills, such as 8xD and 12xD

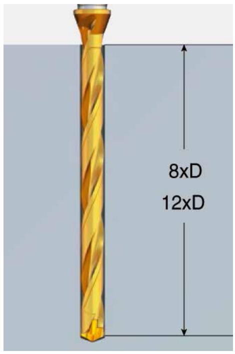


1. When using longer drills, such as 8xD and 12xD, it is advisable to drill a pilot hole of 0.5xD to 1.5xD.



2. Feed the drill into the pilot hole until 2mm above the bottom of the hole at reduced cutting speed.

3. Switch on the internal coolant supply 2 to 3 sec. before starting the drilling operation, set the required rotation speed and feed rate.



4. Drill continuously using the recommended cutting data

GOLDTWIST Tap drills

Product overview

The new tap drills with integrated chamfer step are an addition to our Ingersoll GoldTwist product line. The tools can be used to produce 45° chamfered through-holes and blind holes and for producing holes with a 45° countersink.

The tools feature the tried and tested GoldTwist interchangeable tip system and the interchangeable chamfering inserts.

The tap drills can be used for metric thread sizes M10 to M24.

Advantages

Thanks to the combination of the tap drill and the chamfer step, 2 machining steps can be consolidated into one for more economic machining.

Furthermore, the re-grinding and high stock-keeping costs associated with the use of solid carbide drills are eliminated

Apllications

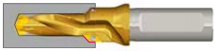
Drilling with 45° chamfer:

Blind hole

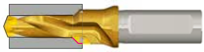
Through hole

Drilling with 45° countersink

Blind hole



Through hole

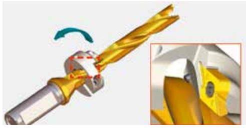


Drilling

GOLDTWIST Chamfer ring

1. Slide the chamfer ring onto the drill body; the limit stop must be within the flute.

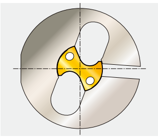
2. Slide the chamfer ring to the required position.



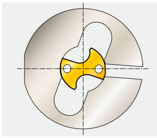
3. Turn the chamfer ring clockwise until the limit stop touches the edge of the flute

4. Fasten the chamfer ring into place and mount the drill tip

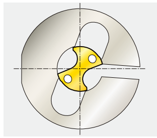
The flutes of drill and chamfer ring must be properly aligned



Correct

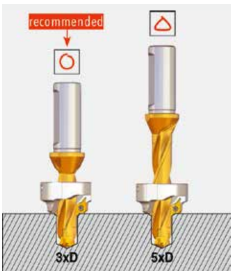


Incorrect



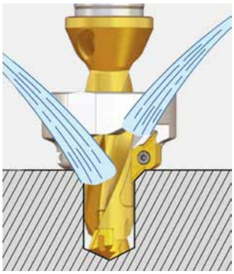
Incorrect

Stable machining

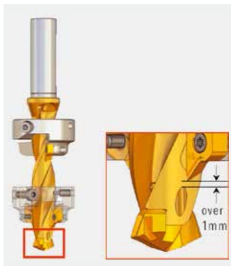


Always use a short tool, if possible. If not possible, reduce the cutting speed to minimize vibrations.

Mount the chamfer ring as close to the drill shank as possible



For a better tool life, use internal and external coolant.

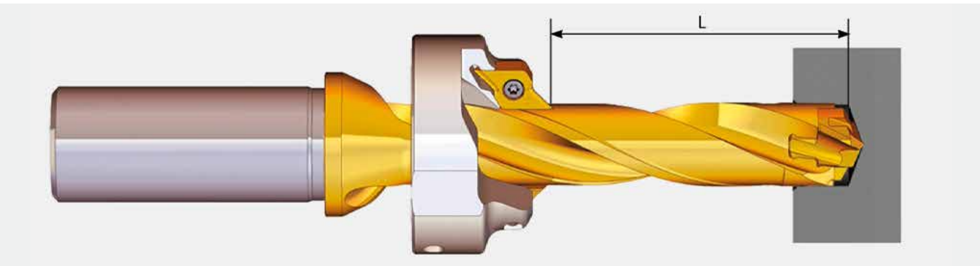


When mounting the chamfer ring, make sure that the coolant flow is not impeded.

Min. and max. length GoldTwist with chamfer ring

	Designation drill	Designation chamfer ring	Length	
			min.	max.
3D	TD1300039JDR00	CB130R02	19	19
	TD1350040JDR00	CB135R02	19	20
	TD1400042JDR00	CB140R02	21	22
	TD1450043JDR00	CB145R02	22	23
	TD1500045JER00	CB150R02	23	23
	TD1600048JER00	CB160R02	24	25
	TD1700051JER00	CB170R02	26	28
	TD1800054JFR00	CB180R02	27	30
	TD1900057JFR00	CB190R02	29	33
	TD2000060JFR00	CB200R02	30	36
	TD2100063JFR00	CB210R02	32	39
	TD2200066JFR00	CB220R02	33	42
	TD2300069JGR00	CB230R02	35	45
	TD2400072JGR00	CB240R02	36	48
	TD2500075JGR00	CB250R02	38	51
5D	TD1000050JDR00	CB100R02	28	28
	TD1050052JDR00	CB105R02	29	30
	TD1100055JDR00	CB110R02	31	33
	TD1150057JDR00	CB115R02	32	35
	TD1200060JDR00	CB120R02	33	45
	TD1250062JDR00	CB125R02	34	40
	TD1300065JDR00	CB130R02	36	43
	TD1350067JDR00	CB135R02	37	43
	TD1400070JDR00	CB140R02	38	48
	TD1450072JDR00	CB145R02	39	48
	TD1500075JER00	CB150R02	41	53
	TD1600080JER00	CB160R02	43	58
	TD1700085JER00	CB170R02	46	63
	TD1800090JFR00	CB180R02	48	68
	TD1900095JFR00	CB190R02	51	73
	TD2000100JFR00	CB200R02	53	78
	TD2100105JFR00	CB210R02	56	79
	TD2200110JFR00	CB220R02	58	84
	TD2300115JGR00	CB230R02	61	89
	TD2400120JGR00	CB240R02	63	94
	TD2500125JGR00	CB250R02	66	99

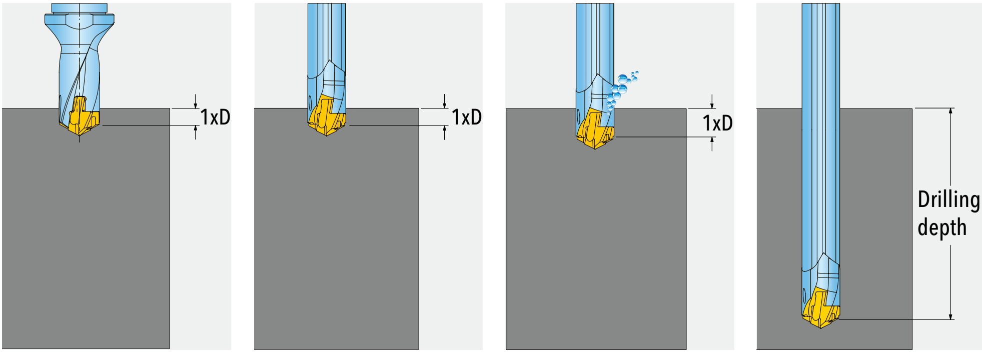
	Designation drill	Designation chamfer ring	Length	
			min.	max.
8D	TD1000080JDR00	CB100R02	45	58
	TD1050084JDR00	CB105R02	49	62
	TD1100088JDR00	CB110R02	49	66
	TD1150092JDR00	CB115R02	53	70
	TD1200096JDR00	CB120R02	53	74
	TD1250100JDR00	CB125R02	57	78
	TD1300104JDR00	CB130R02	57	82
	TD1350108JDR00	CB135R02	61	84
	TD1400112JDR00	CB140R02	61	88
	TD1450116JDR00	CB145R02	65	92
	TD1500120JER00	CB150R02	65	96
	TD1600128JER00	CB160R02	69	103
	TD1700136JER00	CB170R02	73	111
	TD1800144JFR00	CB180R02	77	118
	TD1900152JFR00	CB190R02	81	126
	TD2000160JFR00	CB200R02	85	134
	TD2100168JFR00	CB210R02	89	142
	TD2200176JFR00	CB220R02	93	150
	TD2300184JGR00	CB230R02	97	158
	TD2400192JGR00	CB240R02	101	166
	TD2500200JGR00	CB250R02	105	174
12D	TD1200144T3R00	CB120R02	87	121
	TD1250150T3R00	CB125R02	90	127
	TD1300156T3R00	CB130R02	93	133
	TD1350162T3R00	CB135R02	96	137
	TD1400168T3R00	CB140R02	99	142
	TD1450174T3R00	CB145R02	102	149
	TD1500180T4R00	CB150R02	105	155
	TD1600192T4R00	CB160R02	111	166
	TD1700204T4R00	CB170R02	117	178
	TD1800216T5R00	CB180R02	123	189
	TD1900228T5R00	CB190R02	129	201
	TD2000240T5R00	CB200R02	135	213
	TD2100252T5R00	CB210R02	141	225
	TD2200264T5R00	CB220R02	147	237



Drilling

Drill Penetration Instructions on Milling or Turning Machines

The following procedure (1-4) is recommended for up to 400 mm hole depths using 400mm drills.
For hole depths between 400 up to 800 mm, use 800mm drill only after reaching 400 mm depth with an 400mm drill.



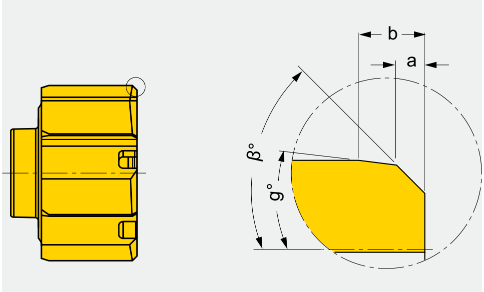
1. Drill a pilot hole 0.5xD deep with a short drill in the same diameter as for the GoldTwist drill
2. Enter the pre-hole at slow speed, feed, and 50 RPM until 1-2 mm before reaching the bottom.
3. Activate the cooling system and increase rotation speed to recommended drilling speed, maintain for 2-3 seconds, then continue at recommended drilling feed. No pecking is required. Apply maximum possible coolant flow rate.
4. After having reached the required depth, reduce speed to 50-100 RPM while exiting from the hole.

Reamer

QWIKREAM

Front end parameters

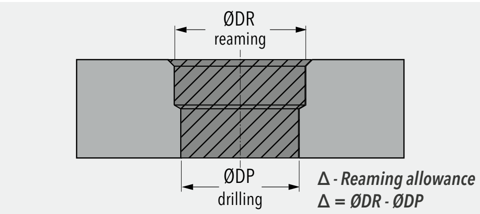
front end configuration code	ß°	a [mm]	g°	b [mm]
A	45°	0,5	-	-
B	25°	1,07	-	-
C	45°	0,5	8°	0,75
D	30°	0,5	4°	1,85
E	45°	0,2	-	-
F	90°	-	-	-
G	75°	0,15	-	-
X	special design (no definition)			



When selecting a reamer, it is important to choose a chamfer geometry which takes account of the reaming allowance.

Additional grades (on request):

- **IN60C:** Cermet-tipped, recommended for reaming in the following materials:
  Unalloyed steel, low-alloy steel, free-cutting steel, as well as spheroidal graphite cast iron.
- **IN91D (PCD):** Recommended for high-speed reaming of aluminum (special applications).
- **IN3305 (DLC-coated):** Recommended for reaming in the following materials:
  Cast aluminum and wrought alloys, brass, bronze and non-ferrous materials.



Hole Ø	11,5–13,5	13,5–16	16–32	
Material				
Steel and cast iron	0,10 – 0,20	0,10 – 0,30	0,10 – 0,30	mm / Ø
Aluminum and brass	0,15 – 0,25	0,20 – 0,30	0,20 – 0,40	mm / Ø

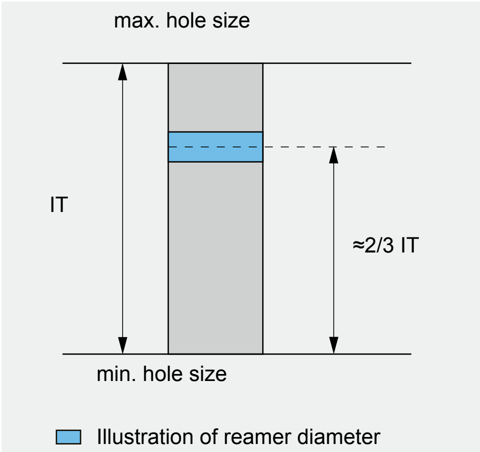
Specification

Basic design

Diameter range:	Ø11,5 bis Ø32,0 mm
Available diameters:	Full millimeter sizes
Accuracy::	Borehole tolerance H7
Grades:	IN05S und IN2005
Available shanks:	3xD, 5xD und 8xD
Shank material:	Steel

Special

Diameter range:	Ø11,5 bis Ø32,0 mm
diameter sizes:	On request
Accuracy::	ISO IT6 und höher
Grades:	IN05S, IN2005, IN91D IN3305 and IN60D
Available shanks:	On request
Shank material:	Steel, carbide, heavy metal

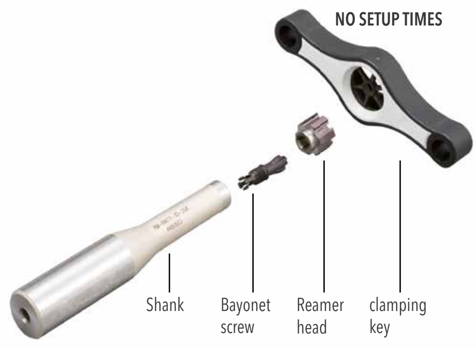


Reamer head diameter & machining concept

Reamer

Technical advantages

- High cutting speeds and feed rates (for increased productivity)
- High true-running accuracy (runout max. 3 µm)
- No setup times
- One shank can be used for a range of diameters and with different cutting edge types and grades
- High repeatability (max 3 µm)
- Durable due to the combination of a solid carbide head and a flexible steel shank
- No risk of losing any clamping parts which may fall during indexing
- Suitable for almost dry machining (minimum quantity lubrication)
- The internal coolant supply is directed optimally to the cutting edges, thus ensuring extremely efficient lubrication of the head and a very long tool life



Initial setup

1. Clean the locating hole in the shank (Fig. 1).
2. Clean the taper of the new reaming head.
3. Insert the clamping screw into the shank and turn it 2-3 turns clockwise (Fig. 2).
4. Put the reaming head onto the screw. Attention! BN8 and BN9 can only be mounted onto the screw in one position (turn the head until the correct position is reached) (Fig. 3).
5. Manually turn the head until is sits securely in the locating hole.
6. Hold the clamping key with one hand and tighten the head until the limit stop is reached. To apply the necessary torque, each BN size has its own clamping key (Fig. 4) (the shank should be clamped in an adapter).
7. Make sure that the contact faces of the shank and the reaming head are in full contact with each other! (Fig. 5).

Attention:

Cutting tools can break off during mounting. To prevent injury, personal protection gear, such as protective gloves, safety mask and safety goggles must be worn.

Exchanging the head

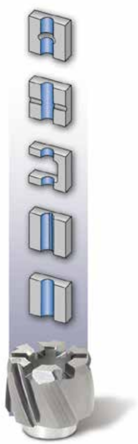
1. Loosen the reaming head by turning it counterclockwise with the clamping key until the head turns freely. Remove the reaming head from the tool. The clamping screw should remain inside! Clean the locating hole of the shank (Fig. 1).
2. Clean the taper of the new reaming head.
3. Put the reaming head onto the screw. Attention! BN8 and BN9 can only be mounted onto the screw in one position (turn the head until the correct position is reached) (Fig. 3).
4. Turn the reaming head clockwise by hand. At the beginning the head turns without the screw, but after 1/6th of a turn, it should mesh into the thread.
5. Keep turning the head until it sits securely in the locating hole. If the screw turns together with the reaming head from the very start, remove the reaming head and unscrew the screw one more turn.
6. Hold the clamping key with one hand and tighten the head until the limit stop is reached. To apply the necessary torque, each BN size has its own clamping key (Fig. 4) (the shank should be clamped in an adapter).
7. Make sure that the contact faces of the shank and the reaming head are in full contact with each other! (Fig. 5).

Applications

Through hole

Left-handed flute

The left-handed flute has been specially designed for through-hole reaming. With this design, the chips are immediately pushed forward as soon as they are produced.

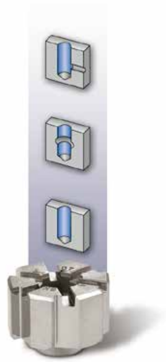


Blind Hole

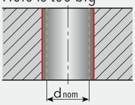
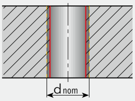
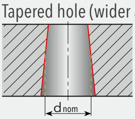
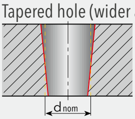
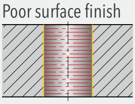
Straight flute

The coolant flow helps evacuate the chips. It carries the produced chips backwards. The chips run through the straight flutes and are discharged from the hole without damaging the reamer and the surface of the hole.

Heads with straight flutes are also used for drilling through-holes in short-chipping materials (cast iron).



Troubleshooting

Problem	Cause	Remedy
Hole is too big 	• Reamer or pilot hole not centered • Reamer too big • Cooling/lubrication error	• Insert a floating chuck or correct the pilot hole • Check the size of the reamer; correct it if necessary • Change the lubricant and increase the coolant pressure
Hole is too small 	• Reamer is worn • Insufficient reaming allowance • Cooling/lubrication error	• Change the reamer • Increase the reaming allowance • Change the lubricant and increase the coolant pressure
Tapered hole (wider at the bottom) 	• Center of the pilot hole and the reaming head incorrectly aligned	• Re-align or use a floating chuck
Tapered hole (wider at the top) 	• Center of the pilot hole and the reaming head incorrectly aligned • Material jammed between the reamer and the hole in the top section of the hole	• Re-align or use a floating chuck • Axially fasten the tool
Poor surface finish 	• Reamer is worn • Center of the pilot hole and the reaming head incorrectly aligned • Chip evacuation problem • Incorrect machining parameters • Burr formation	• Change the tool • Re-align or use a floating chuck • Increase the coolant pressure • Change the machining parameters • Change the machining parameters or increase the coolant supply

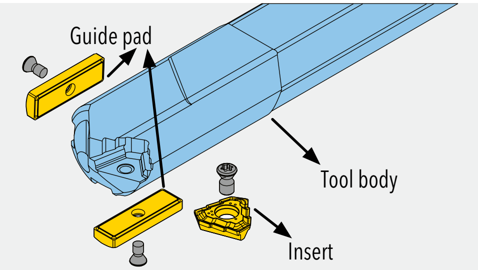
Deep hole drill

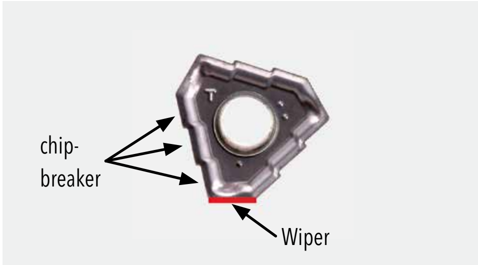
DEEPTRIO

Product overview

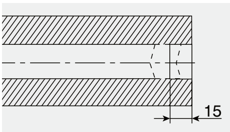
The DeepTrio deep hole drill is available in a diameter range of Ø 16 mm to Ø 28 mm and in the lengths 10xD, 15xD and 25xD.

The insert has 3 cutting edges, with chipbreaker and wiper geometry. They come in the tried and tested, wear-resistant IN2005 grade. The tools are ideally guided by interchangeable guide pads and have internal coolant supply.

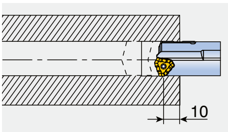




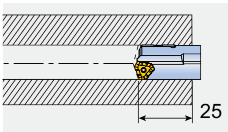
Drilling process



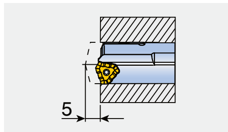
Drill a pilot hole D (+0.03 / +0.10) 15 mm deep.



Feed the drill 10 mm deep into the pilot hole at a low speed of rotation (approx. 50 rpm) and at a low feed rate.
Start the coolant and set the required cutting speed.

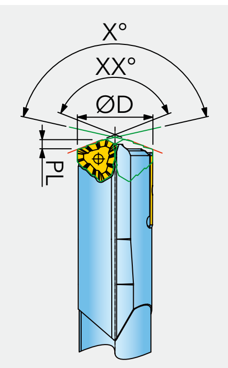


Use approx. 80% of the required feed rate up to a depth of 25 mm, then increase the feed rate to 100%.



For through holes, drill approx. 5 mm deeper, reduce the speed of rotation to approx. 50 rpm and withdraw from the hole.

Point angle

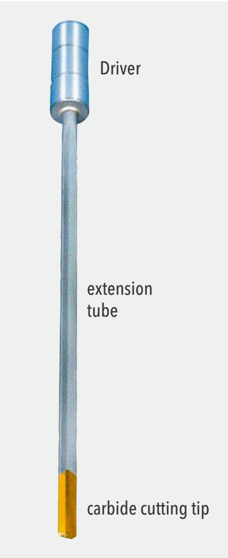


ØD	X°	XX°	PL	Insert
14	154°	137°	2	TPHT070304R
14,5	154°	137°	2	TPHT070304R
15	154°	137°	2	TPHT070304R
16	155°	140°	2,2	TPHT080305R
16,5	155°	140°	2,2	TPHT080305R
17	154°	138°	2,2	TPHT080305R
17,5	154°	138°	2,2	TPHT080305R
18	154°	138°	2,2	TPHT080305R
18,5	154°	137°	3	TPHT090305R
19	154°	137°	3	TPHT090305R

ØD	X°	XX°	PL	Insert
19,5	154°	137°	3	TPHT090305R
20	155°	137°	3	TPHT090305R
21	154°	139°	3,2	TPHT100305R
22	154°	140°	3,4	TPHT110405R
23	154°	140°	3,4	TPHT110405R
24	154°	140°	3,4	TPHT110405R
25	154°	140°	3,4	TPHT110405R
26	154°	140°	3,4	TPHT110405R
27	154°	141°	3,6	TPHT120405R
28	154°	141°	3,6	TPHT120405R

Deep hole drill

Single flute brazed gun drills



Diameter range: Ø2,5 – Ø40 mm
Length range: 150 – 3000 mm

Application

- For drilling deep holes up to 50xD.
- Hole tolerances of IT7 to IT9 attainable.
- Excellent straightness and concentricity.
- Reduced true position deviation.
- Surface finish Ra 0.4 to 1.6 easily achieved.
- Can eliminate the necessity of secondary reaming operation.

Driver

Steel driver for clamping the deep hole drill in the machine.

Extension tube

Designed with a crimped „V" flute and large internal coolant ports. The hardened steel extension tube provides the optimal conditions for twist resistance, coolant flow and ship evacuation.

Carbide cutting tip

Available in 4 different standard cutting geometries, 8 different standard pad forms and 4 coating options for many different machining applications.

Solid carbide deep hole drill

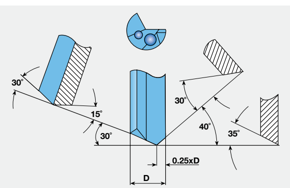
Diameter range: Ø 1.4 mm to Ø 16 mm
Lengths: 40 x drill diameter up to 200 mm flute length

Carbide driver and carbide cutting tip

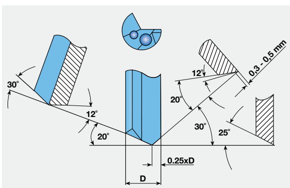
The drill head and the extension tube are produced from one piece of solid carbide. The driver may be made of steel or carbide.
The solid carbide gun drill is designed for use in conventional machining centers and lathes. These drills provide superior rigidity with optimum coolant flow. As a result, speeds and feeds up to 100% faster may be obtained.

Standard gun drill sharpening angles

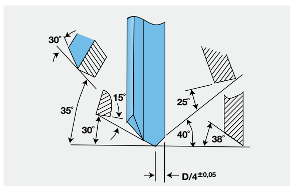
Depending on the required tolerances, cutting performance and the desired chip shape, the following standard sharpening angles are recommended.



Standard sharpening for diameters less than 4.0 mm



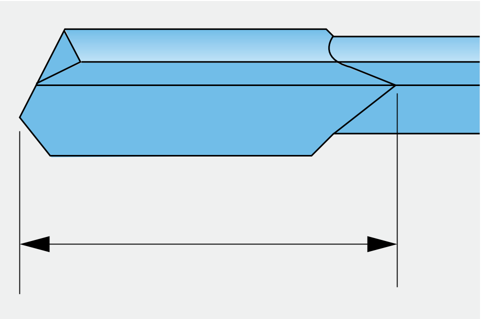
Standard sharpening for diameters greater than 4.0 mm



Optional sharpening for materials where it is difficult to break a chip.

Deep hole drill

Standard gun drill carbide length



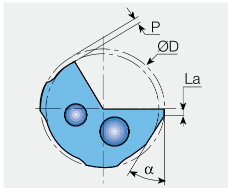
Diameter range	Head length
2,50–3,80	20
3,80–4,05	23
4,05–5,05	25
5,05–6,55	30
6,55–11,05	35
11,05–18,35	40
18,35–21,35	45
21,35–23,35	50
23,35–26,35	55
26,35–32,00	65

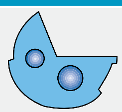
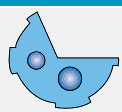
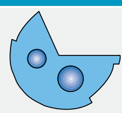
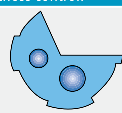
Re-grindable length = length-dia.

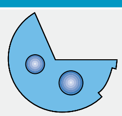
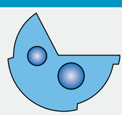
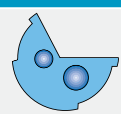
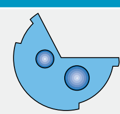
Standard gun drill pad forms

Drilling capacity and hole finish are dependent on the geometrical shape of the drill head. Both the pad form and the sharpening must be matched to the workpiece material. The pad form is determined when the tool is manufactured. Regrinding may change the cutting geometry, but the pad form will remain the same.

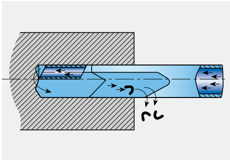
All of the cross section profile parameters, such as P, La and α must be precisely matched to the workpiece material properties



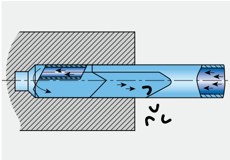
Universal G-form	Standard A-form	Standard B-form	Standard C-form
For all material groups. Works well in materials that tend to shrink. Maintains precision bore tolerance and straightness.	For cast iron & aluminum alloys (coated). Drilling through cross holes and angled entry and exits. Large gap between pads ensures good lubrication.	For cast iron & aluminum alloys. Maintains high precision hole tolerances. Excellent surface finish.	With larger back taper for use in materials that tend to shrink (some alloys and stainless). Drilling through cross holes and angled entry and exit. Not recommended for precise straightness control.
			

Standard D-form	Standard E-form	Standard E-form	Standard I-form
For cast iron only (with coating). Works very well in gray cast iron.	General purpose for all materials. Commonly used in crankshaft and other forged materials.	For all non-ferrous materials. For cast iron for hole diameters greater than 5 mm. Can be used for wood and plastic, with a larger back taper.	For aluminum and brass when best surface finish is required. Can be used in cross hole and interrupted cut applications
			

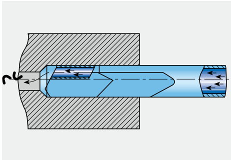
Applications



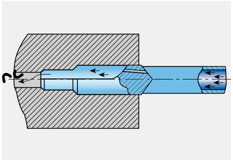
Gun drilling operation in a blind hole when chips and coolant are evacuated back through the flute.



Stepped gun drill boring operation in blind hole when chips and coolant are evacuated back through the flute.



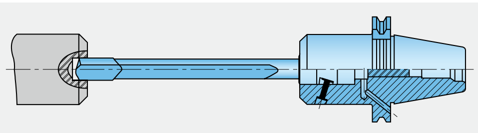
Gun drill boring operation in through hole when chips and coolant are evacuated ahead of the drill tip.



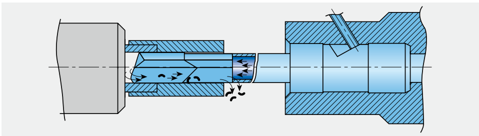
Stepped gun drill boring operation in through hole when chips and coolant are evacuated ahead of the drill tip.

Gun drills in machining centers

- Insert the deep hole drill into the pilot hole either with the drill at a standstill or rotating at a low speed (n < 50 rpm)
- Switch on the coolant
- Set the nominal speed of rotation
- Drill until the drilling depth is reached, for blind holes, withdraw slightly (1 - 2 mm)
- Stop the rotation or set a low speed of rotation (n < 50 rpm)
- Withdraw from the hole



Centering/guiding hole for gun drill operation in machining center



Bushing guide barrel for gun drill stabilizing in machining center

Coolant pressure and volume

The correct coolant

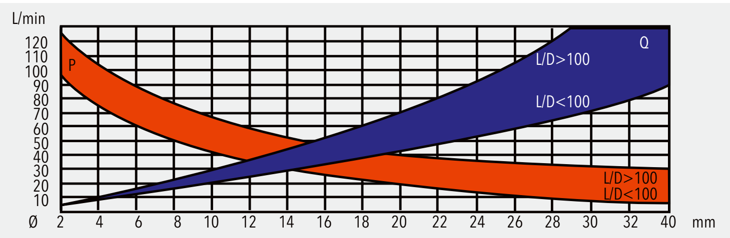
Use oil rather than drilling emulsion when possible

Pure oil

- Oil is used on most conventional deep hole drilling machines
- Provides the best possible lubrication, better tool life and surface finish
- No problem getting the concentration right and no evaporation

Drilling emulsion

- Used on most machining centers



Poor < 8%

Acceptable 10–12%

Good 12–15%

Deep hole drill

Troubleshooting

	Drill problems											Hole problems					
Possible causes	Cratering	Built-up edge	Damaged wear pad	Flute bending	Drill heat	Excessive flank wear	Excessive corner wear	Excessive margin wear	Poor drill life	Chipping	Drill breakage	Curved hole axis	Conical entrance	Runout	Rough surface finish	Undersized hole	Oversized hole
Poor clamping								+			+	+		+			+
Insufficient coolant flow					+	+			+		+				+		+
Low coolant pressure									+		+				+	+	
Incorrect coolant type	+	+	+			+	+	+	+						+		
Feed fluctuations		+		+					+	+	+	+		+	+		
Too high feed	+	+		+	+	+			+		+	+	+	+	+		+
Too low feed		+							+	+		+					
Spindle speed too high			+	+	+	+	+	+	+		+						
Spindle speed too low	+	+													+		
Material structure	+	+	+				+		+	+	+	+		+			
Material shrinking due to heat			+		+		+		+		+			+	+	+	
Workpiece thin wall section									+		+	+		+			
Misalignment			+	+		+		+	+		+	+		+			+
Undersized hole			+		+	+		+	+		+				+		
Rough cutting edge finish	+	+					+		+	+	+	+		+			
Built-up edge							+		+		+				+		+
Worn out cutting edge	+	+					+	+	+	+	+	+		+	+		
Interrupted chip flow			+	+		+		+	+		+	+		+	+		+
Too small flute clearance			+		+	+		+	+	+		+		+			
Incorrect drill profile	+	+	+	+			+	+	+		+	+		+	+	+	+
Incorrect head angles	+	+		+		+	+		+	+	+	+	+		+	+	+
Vibrations	+	+	+	+			+		+	+	+	+	+	+	+		+
Oversized bushing									+		+	+	+	+			+
Gap between bushing and workpiece						+			+		+	+	+	+	+	+	+
Bushing undersized			+	+		+		+	+		+			+		+	
Loss of coolant pressure		+	+	+	+				+		+		+		+	+	
High coolant pressure												+					+
Overheating coolant	+		+		+	+	+	+	+						+		
Insufficient coolant	+	+	+	+	+				+		+	+		+	+		+
Head inside angle excessive wear			+	+					+					+	+		+
Head outside angle excessive wear		+		+			+		+					+	+	+	
Too short carbide head	+	+	+	+	+	+	+	+	+	+	+	+		+	+	+	+
Tool heel drag			+	+	+	+		+	+		+	+		+	+		+
Worn supporting pads	+		+	+	+		+		+		+			+			+
Tool whip - reverse tool rotation	+	+	+	+			+		+	+	+			+	+		+

Note

Member IMC Group
Ingersoll Cutting Tools

only online /pdf edition 06/20
Changes and printing errors reseved

Ingersoll Cutting Tools
Marketing & Technology

Germany
Ingersoll Werkzeuge GmbH

Kalteiche-Ring 21-25
35708 Haiger, Germany
Phone: +49 2773 742-0
Fax: +49 2773 742-812
Email: info@ingersoll-imc.de
Internet: www.ingersoll-imc.de

USA
Ingersoll Cutting Tools

845 S. Lyford Road
Rockford, Illinois 61108-2749, USA
Phone: +1-815-387-6600
Fax: +1-815-387-6968
Email: info@ingersoll-imc.com
Internet: www.ingersoll-imc.com

France
Ingersoll France

22, rue Albert Einstein
F-77420 CHAMPS-sur-MARNE
Phone: +33 164684536
Fax: +33 164684524
Email: info@ingersoll-imc.fr
Internet: www.ingersoll-imc.fr

www.ingersoll-imc.de