

CUTTING DATA & PARAMETERS
MILLING AND DRILLING

WINSFEED



Standard plus – Expanding the range



Ingersoll's standard program comprises a broad and worldwide 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 customerspecific 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.



Contend

General informations / Average chip thickness	PAGE 6
General formulas	PAGE 7
Grades - Coatings	PAGE 8

MILLING CUTTERS

AOMT06_ / AOCT06_ HiPosMicro	PAGE 12
AOMT17_ / AOCT17_ / BOCT17_ / BOMT17_ HiPos+	PAGE 13
BEEW1203_ / FEEW2503_ FastBREAK	PAGE 14
BEHW2503_ RoughAIR	PAGE 15
BOMT09_ / ZOMT09_ / BOCT09_ / BODT09_ HiPos+	PAGE 16
BOMT13_ / ZOMT13_ / BOCT13_ / BODT13_ HiPos+	PAGE 17
CGM(X)101R_ / CGM(X)101_ MaxSFEED	PAGE 18
DGM212R_ / DGD212R_-W / ZGM212_ EvoTecMAX	PAGE 19
DGM313R3_ / EvoTecMAX	PAGE 20
DGM324R3_ / ZGM324R3_ EvoTecMAX	PAGE 21
FNKT05_ / FNHT05_ / FNCT050202R-AL HiPosSFEEDV	PAGE 23
HNGU06_ DiPosHEXa	PAGE 24
HNGU10_ DiPosHEXa	PAGE 25
IEE_ / IXE_ GoldSlot	PAGE 26
MNHU04_ DiPosDuo	PAGE 28
MNHU06_ / MNCU06_ DiPosTetra	PAGE 29
MNHU09_ DiPosDuo	PAGE 30
MNHU11_ DiPosDuo	PAGE 31
MNHU14_ DiPosDuo	PAGE 32
OFMT05_ / OFCT05_ / OFMW05_ RotoMill	PAGE 34
OFMT07_ / OFCT07_ / OFMW07_ RotoMill	PAGE 35
ONCU05_ OctoPlus	PAGE 36
ONCU09_ / ONCO09_ OctoPlus	PAGE 37
PNCU0503GNFR-P / PNMU0503GNTR DiPosDekaMicro	PAGE 38
PNCU08_ / PNCQ08_ DiPosDeka	PAGE 39
SDE_ SlotMax	PAGE 40
SGM-44R_ GoldMax8	PAGE 41
SD_T05_ HiPosQuad	PAGE 42
SDMT08_ / SDMW08_ / SDCT08_ HiPosQuad	PAGE 43
SDES13_ / SDMS13_ GoldQuad	PAGE 44
SHE11_ ALUMINATOR	PAGE 46
SNC_11T3_ IsoPlus	PAGE 47
SQGU07_ DiPosQuad	PAGE 48
SQGU10_ DiPosQuad	PAGE 49
SQGU11_ DiPosQuadF	PAGE 50
SQGU14_ DiPosQuadF	PAGE 51
TCHW110204R-W / TCHH110204FR-PW MicroMill	PAGE 52
THLS04_ HiPosTrio	PAGE 54
THLS06_ / THES06_ HiPosTrio	PAGE 55
THLS10_ / THES10_ HiPosTrio	PAGE 56
THLS13_ / THES13_ HiPosTrio	PAGE 57
TIMC TCLAMP	PAGE 58
SF_ WinCut	PAGE 59

UNHU040212R-HF DiPosDuo	Page 60
UNHU060320R-HF DiPosDuo	Page 61
UNHU090432R-HF DiPosDuo	Page 62
UNHU110640R-HF DiPosDuo	Page 63
UNHU140750R-HF DiPosDuo	Page 64
UNKT0502TR-HF HiPosSFEEDV	Page 65
WNMU04_ / WNCU04_ DiPosTrio	Page 66
WNMU06_ / WNCU06_ DiPosTrio	Page 67
XEVT16_ HiPosAlu	Page 68
YDA223L_ MicroMill	Page 70
YDA323L_ / YDA334L_ / YCE434_ MicroMill	Page 71

MOULD & DIE

CDHT09_ HiPosProB	Page 72
CDHJ_ DiPosProB	Page 73
CNHU_ FormMasterV	Page 74
NBEU_M0R_ ProDuo	Page 76
NQH_ / GQHG_ FineMold+	Page 77
SHLT_ / SHGT_ / SPLT_ / SDGT_ / CDE_ / DPM_ PunchIN	Page 78
PEMT05022CTR-HR HFDMini	Page 79
RCLT_ FormMasterPro	Page 80
RH_MO.N_ FormMaster/FormMaster+	Page 82
RNLU 10/12/16 FormMasterR	Page 84
RPLX10_ / RPLX12_ BladeMaster+	Page 86
SDXS04_ GoldSFEED	Page 88
SDXS06_ GoldSFEED	Page 89
SDXS09_ GoldSFEED	Page 90
SDXS13_ GoldSFEED	Page 91
SDXS16_ GoldSFEED	Page 92
SD_S19_ GoldSFEED	Page 93
SDMS13_ GoldQuad^{XXX}	Page 94
SDMS19_ GoldQuad^{XXX}	Page 95
SDES13_ / SDMS13_ GoldQuad^{PLUNGE}	Page 96
LNFX0905R01 CERASFEED	Page 98
TXNX1207N0104 CERASFEED	Page 99
UNLU04_ DiPosFEED	Page 100
UNLU06_ DiPosFEED	Page 101
UNLU09_ DiPosFEED	Page 102
UNLU11_ DiPosFEED	Page 103
UNLV06_ DiFEED^{WINV}	Page 104
UOMT0602_ HiPosMicro	Page 105
WCNT06_ / WCNW06_ PowerFEEDMini	Page 106
WNMU04_ / WNCU04_ DiPosTrio	Page 107

SOLIDCARBIDE & CHIPSURFER

16T_ / 17T_ / 18T_ & 16N_ / 17N_ CHIPSURFER	Page 108
18F_ SHOULDER/FACE MILL CHIPSURFER	Page 110
18F_ / 19F_ / 18T_ / 19T_ / 18Y_ / 19Y_ MULTISURFER	Page 111
45A_ SPEED MILL CHIPSURFER	Page 112
45B_ / 46B_ / 47B_ / 45X_ BALL NOSE MILL CHIPSURFER/SOLIDCARBIDE	Page 113
45D_ / 46D_ / 47D_ / 46J_ / 47J_ END MILL CHIPSURFER	Page 114
45D_ SLOT MILL CHIPSURFER	Page 115
45N_ / 46N_ / 45M_ / 45P_ / 47N_ / 48N_ / 45R_ / 18S_ CHIPSURFER	Page 116
45U_ TORIC CUTTER CHIPSURFER	Page 117
45Z_ NC CENTER DRILL CHIPSURFER	Page 118
45Z_ CENTERING DRILL CHIPSURFER	Page 118
45Z_ / 45Q_ ENGRAVING CUTTER CHIPSURFER	Page 118
CND_ 90° FLAT CONTERSINK CHIPSURFER	Page 119
46D_ / 47C_ / 48C_ ROUGHING CUTTER CHIPSURFER	Page 120
46J_ / 46D_ / 45J_ END MILL CHIPSURFER/SOLIDCARBIDE	Page 121
46W_ TAPERED END MILL CHIPSURFER	Page 122
46D_ FINISH MILL LENS-SHAPE CHIPSURFER	Page 123
48E_ BARREL FORM END MILL CHIPSURFER	Page 124
47A_ / 48A_ / 47B_ SPEED MILL CHIPSURFER/SOLIDCARBIDE	Page 126
47C_ / 47D_ HPC CUTTER CHIPSURFER	Page 128
47D_ ROUGH/FINISH MILLS 1,5xD CHIPSURFER	Page 129
48D_ FINISH MILL 1,5xD CHIPSURFER	Page 130
47C_ ROUGH MILL 1,5xD CHIPSURFER	Page 131
47C_ / 48C_ / 46D_ / 47D_ ROUGHING END MILL SOLIDCARBIDE	Page 132
47C_ HPC CUTTER SOLIDCARBIDE	Page 134
47C_ HPC CUTTER Z = 4/5 SOLIDCARBIDE	Page 136
47D_ / 48D_ / 49D_ / 48J_ FINISHING CUTTER CHIPSURFER	Page 138
47J_ / 48J_ FINISHING END MILL SOLIDCARBIDE	Page 140
48U_ TORIC CUTTER CHIPSURFER	Page 142
SOLID CERAMIC SPEED END MILL Z3 INCERAMIC	Page 143

BORING

FR_T_R01 SOLIDDRILL³	Page 144
FP_R01 DRILL HEADS TWISTSFEEED	Page 145
TNA_ / TPA_ / TMA_ / TKA_ / TPF_ / TPC_ GOLDTWIST	Page 146
LP_R01 DRILL HEADS SPADETWIST	Page 147
SPGX_ GOLDTWIN	Page 148
SCLT_ / SHGT_ / SHLT_ / SPLT_ / SDGT_ QUADDRILL+	Page 150
SOMT_ QUADTWIST	Page 152
TFLT_ SPOTIN	Page 154
PHT DEEPTRIO	Page 155
REAMERS WITH STRAIGHT FLUTE HEAD QWIKREAMS	Page 156
REAMERS WITH TWISTED FLUTE HEAD QWIKREAMS	Page 157
REAMERS WITH STRAIGHT FLUTE HEAD QWIKREAM	Page 158
REAMERS WITH TWISTED FLUTE HEAD QWIKREAM	Page 159
SINGLE FLUTE GUN DRILLS	Page 160

General information / Average chip thickness

General information

Cutting speed and feed rate are important parameters in machining, as they have a decisive influence on the production time and workpiece quality.

The choice of the right cutting speed depends essentially on the composition and strength of the material to be machined, the toughness and hardness of the used grade, as well as the desired dimensional accuracy and surface quality. Due to the parabolic rise of the insert temperature, it influences significantly the wear and thus the tool life as the speed increases.

All cutting parameters given here are to be considered as recommended values. They should be optimized depending on the respective machine performance and stability.

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore, in any case doubt do not hesitate to contact your Ingersoll partner.



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 formula for contouring also leads to higher productivity.

Fig. 1: Full slot

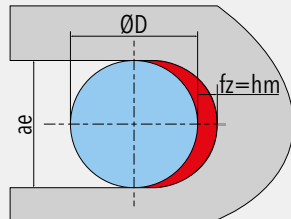
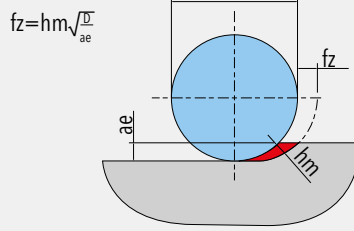


Fig. 2: Contouring



General formulas

Cutting data calculation

Parameters	
n:	Speed of rotation (min ⁻¹)
V _c :	Cutting speed
D:	Tool diameter (mm)
$n = \frac{V_c}{D \times \pi} \times 1000 \qquad V_c = \frac{D \times \pi \times n}{1000}$	

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

Parameters	
V _f :	Feed rate (mm/min)
f _z :	Feed per tooth (mm)
Z _(eff) :	Number of teeth
$V_f = n \times f_z \times Z \qquad f_z = \frac{V_f}{n \times Z}$	

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 :	$n = \frac{200 \times 1000}{80 \times \pi} = 796 \text{ min}^{-1}$
Calculation of feed rate:	$v_f = 0,12 \times 796 \times 9 = 859 \text{ mm/min}$
Calculation of chip removal rates:	$Q = \frac{4 \times 50 \times 859}{1000} = 172 \text{ cm}^3/\text{min}$
Calculation of specific cutting force:	$k_c = 0,15^{-0,24} \times 1615 = 2546 \text{ MPa}$
Calculation of the required spindle power:	$P_c = \frac{4 \times 50 \times 859 \times 2546}{60 \times 10^6} = 7,3 \text{ kW}$
Calculation of the motor power:	$P_{\text{mot}} = \frac{7,3}{0,8} = 9,1 \text{ kW}$

Grades – Coatings

Grade	Coating	ISO-Group	Milling	Drilling	Solid carbide	Application and Material
PVD coated	IN0545	TiN	P25-P45	•	•	special grade for circular milling of threads
		M25-M45	•	•	drilling of carbon and alloy steels	
	IN1205	TiB ₂	N20-N40	•	•	for machining aluminum and aluminum alloys
			S20-S40	•	•	as well as non-ferrous metal and non-metallic material
	IN1530	TiCN / TiN	P25-P40	•	•	very tough quality for machining steel under unstable conditions
			M30-M40	•	•	very tough quality for machining stainless steel under unstable conditions
			S20-S30	•	•	very tough quality for machining heat resistant steels under unstable conditions
	IN2004	TiAlN	P05-P20	•	•	for milling of alloyed steel
			K05-K25	•	•	for medium machining of gray cast iron especially CGI
			H05-H25	•	•	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
	IN2006	TiAlN	S05-S20	•	•	for general milling of heat resistant alloys and titanium also for wet machining
			P10-P30	•	•	for finish machining at high cutting speed and low cutting depth
			M10-M30	•	•	for finish machining of stainless steel
	IN2010	TiAlN	S10-S30	•	•	for finish machining of heat resistant alloys and titanium
			H10-H30	•	•	for finish machining of hardened steel up to 55 HRC
			K10-K30	•	•	for finish machining and drilling of cast iron
	IN2030	TiAlN	P30-P50	•	•	for machining steel with high cutting speed
			M30-M50	•	•	for machining stainless steel with high cutting speed
			K30-K50	•	•	for machining cast iron
	IN2035	TiAlN	S30-S50	•	•	for machining of heat resistant alloys and titanium
			M30-M50	•	•	for machining of stainless and austenitic steel and heat resistant alloys
			S30-S50	•	•	mainly for milling of materials of machining group 'S'
	IN2036	AlTiN	M30-M50	•	•	AlTiN coated grade with highest toughness for processing stainless steels and for machining difficult-to-machine materials of machining group "S"
	IN2040	TiAlN	S30-S50	•	•	for machining difficult-to-machine materials of machining group "S"
			P10-P30	•	•	for finish machining of unalloyed steel and tempered steel
			P10-P30	•	•	for machining steel at low to medium cutting speeds
	IN2205	AlTiN / TiAlSiN	M30-M50	•	•	for machining stainless steels (<40 HRC) at low to medium cutting speeds
			S30-S50	•	•	for machining heat-resistant alloys at low to medium cutting speeds
			P05-P25	•	•	for milling of steel at medium up to high cutting speed
IN2504	AlTiN	K05-K25	•	•	for milling hardened steels at medium to high cutting speeds	
		H05-H25	•	•	for milling of hardened steel at medium up to high cutting speed	
		P20-P40	•	•	for semi-finish and rough machining of steel with high strength	
IN2505	TiAlN / TiN	M20-M40	•	•	for general machining of stainless steel	
		S20-S40	•	•	for general machining of heat resistant alloys	
		K10-K30	•	•	for general machining of gray cast and non-ferrous metal	
IN2510	TiAlN / TiN	H10-H30	•	•	for general milling of hardened steel	
		P30-P50	•	•	tough grade for general machining of steel	
		M30-M50	•	•	for general machining of stainless steel	
IN2530	TiAlN / TiN	K30-K50	•	•	for general machining of cast iron	
		S30-S50	•	•	for general machining of heat resistant alloys	
		P10-P30	•	•	for semi-finish and rough machining of unalloyed steel and tempered steel	
IN2540	TiAlN / TiN	K10-K30	•	•	for leveling and roughing cast iron materials	
		P20-P40	•	•	DLC coated fine-grain metal for processing aluminum and non-ferrous materials	
		M20-M40	•	•	for general machining of steel	
IN4005	TiAlN / AlCrO	M20-M40	•	•	for general machining of stainless steel	
IN4010	TiAlN / AlCrO	S20-S40	•	•	for general machining of heat resistant alloys and titanium	
		K10-K30	•	•	for general machining of cast iron	
		P15-P35	•	•	for milling of steel with high strength at medium cutting speed	
IN4015	TiAlN / AlCrO	K15-K35	•	•	for general milling of gray cast and nodular cast iron	
		P30-P50	•	•	tough grade for general machining of steel	
		M30-M50	•	•	for general machining of stainless and austenitic steel	
IN4030	TiAlN / AlCrO	K30-K50	•	•	for general machining of cast iron	
		S30-S50	•	•	for general machining of heat resistant alloys	
		P30-P50	•	•		

	Grade	Coating	ISO-Group	Milling	Drilling	Solid carbide	Application and Material	
PVD coated	IN4035	TiAlN / AlCrO	M30-M50	•			for machining of stainless steel, austenitic steel and heat resistant alloys mainly for milling of materials of machining group, S'	
			S30-S50					
	IN4036	TiAlN / AlCrO	M30-M50	•				for machining of stainless steel, austenitic steel and heat resistant alloys mainly for milling of materials of machining group, S'
IN4040	TiAlN / AlCrO	P10-P30	•			for medium machining of unalloyed and tempered steel		
		K10-K30					for medium machining of cast iron	
Carbide	IN055	-	N10-N25	•	•		for machining of AL-alloys and non-ferrous materials	
	IN10K	-	N10-N25	•	•		for finish machining of AL-alloys and non-ferrous materials	
	IN15K	-	K15-K30	•			for machining of cast iron	
N15-N30			for machining of AL-alloys and non-ferrous materials					
CVD coated	IN6505	TiCN / Al ₂ O ₃ / TiN	P10-P25	•			for drilling of steel, used only at peripheral insert of QuadTwist drill	
	IN6535	TiCN / Al ₂ O 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'
			P30-P45				•	for roughing carbon steels and alloyed steels at high Vc
	IN6537	TiCN / Al ₂ O ₃ / TiN	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 ₂ O ₃ / TiN	M20-M35	•			for machining of stainless and austenitic steel and heat resistant alloys primarily for milling of materials of machining group, S'		
Cermets	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 ₃ N ₄	K10-K20	•			for machining of 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	



Grades – Coatings

Application	Grade	ISO-Group					
Milling	IN2504	P05-P25				H05-H25	↑ harder ↓ tougher
	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		
Drilling	IN2010		K10-K30				↑ harder
	IN6505	P10-P25					↓ tougher
	IN6520	P10-P40					↑ harder
	IN2505	P20-P40	M20-M40		S05-S20		↓ tougher
	IN2005	P15-P30	M15-M35	K20-K40	S05-S20		↑ harder
Solid carbide	IN2504	P05-P25				H05-H25	↑ harder
	IN2006	P05-P20				H05-H20	↓ tougher
	IN2205	P20-P40	M20-M40		S20-S40		↑ harder
	IN2005	P15-P30	M15-M35	K20-K40	S05-S20		↓ tougher

Note

A large grid of graph paper for taking notes, consisting of 20 columns and 30 rows of small squares. The grid is bounded by a thin grey line on the top, left, and bottom, and a thick blue vertical bar on the right side.



Insert:	AOMT0602_R	AOCT0602_FR-P	AOMT0602_R-DT1
Average chip thickness:	hm = 0,06 mm	hm = 0,05 mm	hm = 0,05 mm
max. cutting depth:	ap = 5,7 mm	ap = 5,7 mm	ap = 2,0 mm

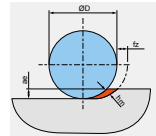
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 - 290	IN2530	200 - 240	0,06 - 0,12
	alloyed steel 800 N/mm²	IN2505	210 - 250	IN2530	160 - 200	0,06 - 0,10
	alloyed steel 1100 N/mm²	IN2505	160 - 180	IN2530	110 - 130	0,06
M	stainless steel	IN2035	120 - 180	IN2035	80 - 130	0,06 - 0,12
K	gray cast iron	IN2504	180 - 250	IN2530	150 - 200	0,06 - 0,12
	nodular cast iron	IN2504	140 - 210	IN2530	110 - 160	0,06 - 0,10
N	aluminum	IN90D	800 - 1500	IN05S	500 - 800	0,05 - 0,12
S	high temperature alloys	IN2035	110 - 125	IN2035	60 - 80	0,06
	titanium alloys	IN2505	40 - 50	IN2035	30 - 40	0,06
H	hard machining < 54 HRC	IN2504	30 - 40	-	-	0,06
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- For tightening the insert screws please always use a torque driver (0,5 Nm).
- For difficult to machine materials please use a max. depth of cut ap = 2 mm and a feed per tooth fz = 0,06 mm.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. uneven ground [mm]	max. ap/rev. [mm]	min. Bore dia. even ground [mm]	max. ap/rev. [mm]	max. Bore dia. even ground [mm]	max. ap/rev. [mm]
9,5	10,5	11	0,9	17	4,4	18	4,5
10,0	10,0	12	1,1	18	4,4	19	4,5
11,5	7,0	15	1,4	21	3,7	22	4,1
12,0	6,5	16	1,4	22	3,6	23	3,9
13,5	5,5	19	1,5	25	3,5	26	3,8
14,0	5,2	20	1,5	26	3,4	27	3,7
15,0	4,4	22	1,6	28	3,1	29	3,4
16,0	4,0	24	1,6	30	3,1	31	3,3
19,0	2,6	30	1,6	36	2,4	37	2,6
20,0	2,5	32	1,7	38	2,5	39	2,6
22,0	2,3	36	1,7	42	2,5	43	2,6
25,0	2,0	42	1,7	48	2,5	49	2,6
30,0	1,7	52	1,7	58	2,6	59	2,7
32,0	1,6	56	1,7	62	2,6	63	2,7
35,0	1,4	62	1,7	68	2,5	69	2,6
40,0	1,2	72	1,7	78	2,5	79	2,6

General information:

insert screw: **SM18-041-00**

torque: **0,5 Nm**

torque wrench: **DTN005S with bit DS-TP06TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	AOMT1705_R	AOMT1705_PER	AOMT1705_R-HS	BOCT1705_FR-P	BOMT1706_R
Average chip thickness:	hm = 0,18 mm	hm = 0,18 mm	hm = 0,12 mm	hm = 0,05 mm	hm = 0,15 mm
max. cutting depth:	ap = 16 mm	ap = 16 mm	ap = 16 mm	ap = 16 mm	ap = 16 mm

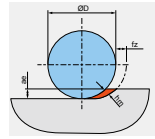
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]		Feed per tooth fz [mm]		
		1st choice dry machining resp. wear resistant carbide	1st choice wet machining resp. tough carbide			
P	unalloyed steel	IN6537	250 - 290	IN2530	200 - 240	0,18 - 0,30
	alloyed steel 800 N/mm ²	IN2505	210 - 250	IN2530	160 - 200	0,18 - 0,25
	alloyed steel 1100 N/mm ²	IN2505	160 - 180	IN2530	110 - 130	0,18
M	stainless steel	IN2035	120 - 180	IN2035	80 - 130	0,12 - 0,25
K	gray cast iron	IN2510	180 - 250	IN6537	150 - 200	0,18 - 0,30
	nodular cast iron	IN2510	140 - 210	IN6537	110 - 160	0,18 - 0,25
N	aluminum	IN10K	800 - 1500	IN10K	500 - 800	0,05 - 0,30
S	high temperature alloys	IN2035	110 - 125	IN2035	60 - 80	0,12
	titanium alloys	IN2505	40 - 50	IN2035	30 - 40	0,12
H	hard machining < 54 HRC	IN2504	30 - 40	-	-	0,18
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. uneven ground [mm]	max. ap/rev. [mm]	min. Bore dia. even ground [mm]	max. ap/rev. [mm]	max. Bore dia. even ground [mm]	max. ap/rev. [mm]
25	12,0	31	4,0	43,9	12,6	49	16,0
32	8,0	45	5,7	57,9	11,4	63	13,6
40	5,0	61	5,7	73,9	9,3	79	10,7
50	4,4	81	7,4	93,9	10,6	99	11,8
63	3,2	107	7,7	119,9	9,9	125	10,8
80	2,3	141	7,6	153,9	9,3	159	9,9
100	1,8	181	7,9	193,9	9,2	199	9,7
125	1,4	231	8,1	234,9	9,1	249	9,5
160	0,7	301	5,4	313,9	5,9	319	6,1

General information:

insert screw: **SM40-093-20**

torque: **4,5 Nm**

torque wrench: **DTN045F with bit DS-T15B1**



Insert:	BEEW1203_R-CR	FEEW2503_R-CR
Average chip thickness:	hm = 0,08 mm	hm = 0,08 mm
max. cutting depth:	ap = Radius	ap = Radius

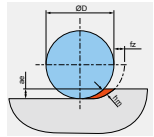
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2530	250 – 290	IN2530	160 – 200	0,08 – 0,20
	alloyed steel 800 N/mm ²	IN2530	210 – 250	IN2530	120 – 160	0,08 – 0,15
	alloyed steel 1100 N/mm ²	IN2530	160 – 180	IN2530	70 – 90	0,08
M	stainless steel	IN2530	120 – 180	IN2035	70 – 90	0,08 – 0,20
K	gray cast iron	IN2530	180 – 250	IN2530	150 – 200	0,08 – 0,20
	nodular cast iron	IN2530	140 – 210	IN2530	110 – 160	0,08 – 0,15
N	aluminum	IN2530	800 – 1500	IN2530	500 – 800	0,08 – 0,20
S	high temperature alloys	IN2530	110 – 125	IN2530	60 – 80	0,08
	titanium alloys	IN2530	40 – 50	IN2530	30 – 40	0,08
H	hard machining < 54 HRC	IN2530	30 – 40	-	-	0,08
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



General information:

insert screw BEEW12: **SM35-070-00**

torque: **3,0 Nm**

torque wrench: **DTN030S with bit DS-T15B**

insert screw FEEW25: **SM40-093-20**

torque: **4,5 Nm**

torque wrench: **DTN045F with bit DS-T15B1**



Insert:	BEHW2503_R	BEHW2503_R-P
Average chip thickness:	hm = 0,08 mm	hm = 0,05 mm
max. cutting depth:	ap = see catalog	ap = see catalog

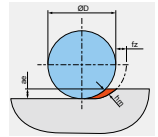
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2540	250 - 290	IN2540	160 - 200	0,08 - 0,20
	alloyed steel 800 N/mm ²	IN2540	210 - 250	IN2540	120 - 160	0,08 - 0,15
	alloyed steel 1100 N/mm ²	IN2540	160 - 180	IN2540	70 - 90	0,08
M	stainless steel	IN2540	120 - 180	IN2540	70 - 90	0,08 - 0,20
K	gray cast iron	IN2540	180 - 250	IN2540	150 - 200	0,08 - 0,20
	nodular cast iron	IN2540	140 - 210	IN2540	110 - 160	0,08 - 0,15
N	aluminum	IN15K	800 - 1500	IN15K	500 - 800	0,08 - 0,20
S	high temperature alloys	IN2540	110 - 125	IN2540	60 - 80	0,08
	titanium alloys	IN2540	40 - 50	IN2540	30 - 40	0,08
H	hard machining < 54 HRC	IN2540	30 - 40	IN2540	-	0,08
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



General information:

insert screw BEEW12: **SM35-089-00**

torque: **3,0 Nm**

torque wrench: **DTN030S with bit DS-T15B**



Insert:	BOMT09T3_R	ZOMT09T3_R	BOCT09T3_FR-P	BOMT09T3_R-DT1	BOMT09T3_R-DT2
Average chip thickness:	hm = 0,10 mm	hm = 0,10 mm	hm = 0,05 mm	hm = 0,05 mm	hm = 0,05 mm
max. cutting depth:	ap = 9 mm	ap = 9 mm	ap = 9 mm	ap = 3 mm	ap = 9 mm



Insert:	BODT09T3_R	BODT09T3_R-001
Average chip thickness:	hm = 0,05 mm	hm = 0,05 mm
max. cutting depth:	ap = 8,9 mm	ap = 3 mm

Recommended cutting data:

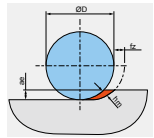
ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN6537	250 – 290	IN2530	200 – 240	0,10 – 0,15
	alloyed steel 800 N/mm ²	IN2505	210 – 250	IN2530	160 – 200	0,10 – 0,12
	alloyed steel 1100 N/mm ²	IN2505	160 – 180	IN2530	110 – 130	0,10
M	stainless steel	IN2036	120 – 180	IN2036	80 – 130	0,10 – 0,12
K	gray cast iron	IN2504	180 – 250	IN6537	150 – 200	0,10 – 0,15
	nodular cast iron	IN2504	140 – 210	IN6537	110 – 160	0,10 – 0,12
N	aluminum	IN90D	800 – 1500	IN10K	500 – 900	0,05 – 0,20
S	high temperature alloys	IN2036	110 – 125	IN2036	60 – 80	0,10
	titanium alloys	IN2505	40 – 50	IN2036	30 – 40	0,10
H	hard machining < 54 HRC	IN2504	70 – 220*	-	-	0,05 – 0,10
	hard machining < 63 HRC	IN2504	50 – 200*	-	-	0,05 – 0,10

*depending on the cutting width ae

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. uneven ground [mm]	max. ap/rev. [mm]	min. Bore dia. even ground [mm]	max. ap/rev. [mm]	max. Bore dia. even ground [mm]	max. ap/rev. [mm]
12	1,5	16	0,3	21,2	0,7	23	0,9
15	11,0	18	1,8	26,8	7,2	29	8,5
16	10,0	20	2,2	28,8	7,0	31	8,5
20	7,0	28	3,0	36,7	6,4	39	7,3
25	4,4	38	3,1	46,7	5,2	49	5,8
28	3,7	44	3,2	52,7	5,0	55	5,4
32	2,8	52	3,0	60,7	4,4	63	4,7
40	2,4	68	3,6	76,7	4,8	79	5,1
50	1,3	88	2,7	96,7	3,3	99	3,4
63	1,0	114	2,7	122,7	3,2	125	3,3

General information:

insert screw up to Ø 16: **SM25-054-00**
torque: **1,1 Nm**

insert screw from Ø 20: **SM25-064-00**
torque wrench: **DTN011S with bit DS-T08TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	BOMT1304_R	ZOMT1304_R	BOMT1304_R-HS	BOCT1304_FR-P	BOMT1304_R-DT1
Average chip thickness:	hm = 0,12 mm	hm = 0,12 mm	hm = 0,08 mm	hm = 0,05 mm	hm = 0,05 mm
max. cutting depth:	ap = 12 mm	ap = 12 mm	ap = 12 mm	ap = 12 mm	ap = 1 mm



Insert:	BOMT1304_R-DT2	BODT1304_R	BODT1304_R-001
Average chip thickness:	hm = 0,05 mm	hm = 0,05 mm	hm = 0,05 mm
max. cutting depth:	ap = 12 mm	ap = 12 mm	ap = 4 mm

Recommended cutting data:

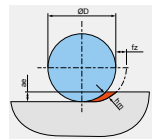
ISO	Material	Cutting speed Vc [m/min]		Feed per tooth fz [mm]		
		1st choice dry machining resp. wear resistant carbide	1st choice wet machining resp. tough carbide			
P	unalloyed steel	IN6537	250 - 290	IN2530	200 - 240	0,12 - 0,20
	alloyed steel 800 N/mm ²	IN2505	210 - 250	IN2530	160 - 200	0,12 - 0,15
	alloyed steel 1100 N/mm ²	IN2505	160 - 180	IN2530	110 - 130	0,12
M	stainless steel	IN2036	120 - 180	IN2036	80 - 130	0,12 - 0,20
K	gray cast iron	IN2504	180 - 250	IN6537	150 - 200	0,12 - 0,20
	nodular cast iron	IN2504	140 - 210	IN6537	110 - 160	0,12 - 0,15
N	aluminum	IN90D	800 - 1500	IN10K	500 - 800	0,12 - 0,20
S	high temperature alloys	IN2036	110 - 125	IN2036	60 - 80	0,12
	titanium alloys	IN2505	40 - 50	IN2036	30 - 40	0,12
H	hard machining < 54 HRC	IN2504	70 - 220*	-	-	0,05 - 0,10
	hard machining < 63 HRC	IN2504	50 - 200*	-	-	0,05 - 0,10

*depending on the cutting width a

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. uneven ground [mm]	max. ap/rev. [mm]	min. Bore dia. even ground [mm]	max. ap/rev. [mm]	max. Bore dia. even ground [mm]	max. ap/rev. [mm]
20	7,0	26	2,3	36	6,1	39	7,3
25	7,9	37	5,2	46	9,0	49	10,4
32	5,0	49	4,6	60	7,6	63	8,5
35	4,2	55	4,6	66	7,2	69	7,9
40	3,2	65	4,3	76	6,2	79	6,8
50	2,1	85	4,1	96	5,4	99	5,7
52	2,0	89	4,0	100	5,2	103	5,5
63	1,4	111	3,6	122	4,5	125	4,7
66	1,2	117	3,3	128	4,0	131	4,2
80	1,0	145	3,5	156	4,1	159	4,3
85	0,9	155	3,4	166	3,9	169	4,1
100	0,8	185	3,7	196	4,2	199	4,3
125	0,6	235	3,6	246	3,9	249	4,0

General information:

insert screw: **SM35-088-10**

torque: **3 Nm**

torque wrench: **DTN030S with bit DS-T10TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	CGX101R_	CGM101_
Feed per tooth:	fz = 0,04 - 0,08 mm	fz = 0,04 - 0,08 mm
max. cutting depth:	ap = 3,5 mm	ap = 3,5 mm

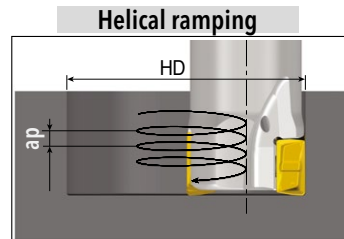
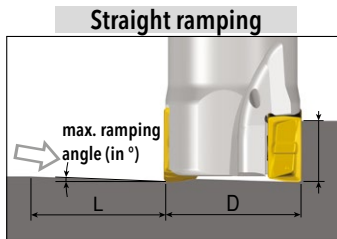
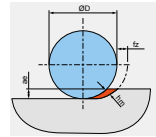
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]			
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide	
P	unalloyed steel	IN2505	150 - 250	IN2530	120 - 200
	alloyed steel 800 N/mm ²	IN2505	120 - 180	IN2530	100 - 160
	alloyed steel 1100 N/mm ²	IN2505	100 - 180	IN2530	80 - 160
M	stainless steel	IN2505	80 - 160	IN2530	80 - 160
K	gray cast iron	IN2505	160 - 250	IN2530	140 - 200
	nodular cast iron	IN2505	120 - 200	IN2530	100 - 180
N	aluminum	-	-	-	-
S	high temperature alloys	IN2505	50 - 80	IN2530	40 - 70
	titanium alloys	-	-	IN2530	30 - 40
H	hard machining < 54 HRC	-	-	-	-
	hard machining < 63 HRC	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 40 % of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Ramping data and circular interpolation:

Tool diameter [mm]	Straight ramping			Helical ramping			
	max. Ramping angle (°)	max. ap	min. Length (L)	min. Ø (HD)	max. ap/rev.	max. Ø (HD)	max. ap/rev.
10	1,8	4,6	142	16,8	0,6	20	0,9
12	1,5	4,6	176	20,8	0,6	24	0,8
16	1,0	4,6	251	28,8	0,6	32	0,8
20	0,8	4,6	330	36,8	0,6	40	0,7
25	0,6	4,6	439	46,8	0,6	50	0,7
32	0,4	4,6	586	60,8	0,6	64	0,7

General information:

insert screw: **SM18-041-00**

torque: **0,5 Nm**

torque wrench: **TXPLUS06x90-B**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	DGM212R_	DGD212R_-W	ZGM212_
Average chip thickness:	hm = 0,12 mm	hm = 0,12 mm	hm = 0,12 mm
max. cutting depth:	ap = 7,5 mm	ap = 7,5 mm	ap = 7,5 mm

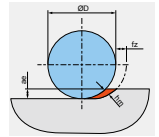
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	150 – 250	IN4030	120 – 200	hm x 1,2
	alloyed steel 800 N/mm ²	IN2505	120 – 180	IN4030	100 – 160	hm x 1,0
	alloyed steel 1100 N/mm ²	IN2505	100 – 180	IN4030	80 – 160	hm x 0,9
M	stainless steel	IN2505	80 – 160	IN2505	80 – 160	hm x 1,2
K	gray cast iron	IN4015	160 – 250	IN4030	140 – 200	hm x 1,2
	nodular cast iron	IN4015	120 – 200	IN4030	100 – 180	hm x 1,0
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2505	50 – 80	IN2505	40 – 70	hm x 0,9
	titanium alloys	-	-	IN4030	30 – 40	hm x 1
H	hard machining < 54 HRC	IN2004	60 – 200	-	-	hm x 0,8
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



General information:

insert screw: **SM30-082-20**

torque: **2,0 Nm**

torque wrench: **DS-TP08S with bit DS-T08TB**



Insert:	DGM313R3
Average chip thickness:	hm = 0,15 mm
max. cutting depth:	ap = 9,7 mm

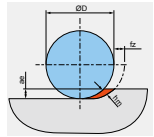
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	150 – 250	IN4030	120 – 200	hm x 1,2
	alloyed steel 800 N/mm ²	IN2505	120 – 180	IN4030	100 – 160	hm x 1,0
	alloyed steel 1100 N/mm ²	IN2505	100 – 180	IN4030	80 – 160	hm x 0,9
M	stainless steel	IN2505	80 – 160	IN2505	80 – 160	hm x 1,2
K	gray cast iron	IN2505	160 – 250	IN4015	140 – 200	hm x 1,2
	nodular cast iron	IN2505	120 – 200	IN4015	100 – 180	hm x 1,0
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2505	50 – 80	IN2505	40 – 70	hm x 0,9
	titanium alloys	-	-	IN2505	30 – 40	hm x 1
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



General information:

insert screw: **SM35-107-H0**

torque: **3,0 Nm**

torque wrench: **DTN030S with bit DS-T15TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	DGM324R3_	ZGM324R3_
Average chip thickness:	hm = 0,20 mm	hm = 0,20 mm
max. cutting depth:	ap = 13 mm	ap = 13 mm

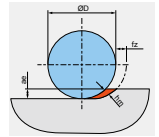
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	150 – 250	IN4030	120 – 200	hm x 1,2
	alloyed steel 800 N/mm ²	IN2505	120 – 180	IN4030	100 – 160	hm x 1,0
	alloyed steel 1100 N/mm ²	IN2505	100 – 180	IN4030	80 – 160	hm x 0,9
M	stainless steel	IN2505	80 – 160	IN2505	80 – 160	hm x 1,2
K	gray cast iron	IN2505	160 – 250	IN4015	140 – 200	hm x 1,2
	nodular cast iron	IN2505	120 – 200	IN4015	100 – 180	hm x 1,0
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2505	50 – 80	IN2505	40 – 70	hm x 0,9
	titanium alloys	-	-	IN2505	30 – 40	hm x 1
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



General information:

insert screw: **SM40-143-H0**

torque: **4,5 Nm**

torque wrench: **DTN045F with bit DS-T15B**

Note

A large grid of graph paper for taking notes, consisting of 20 columns and 30 rows of small squares. The grid is positioned to the right of a vertical blue bar that runs down the left side of the page.



Insert:	FNKT0502_R-M	FNHT0502_R-L	FNCT050202R-AL
Average chip thickness:	hm = 0,04 mm	hm = 0,03 mm	hm = 0,03 mm
max. cutting depth:	ap = 5,0 mm	ap = 5,0 mm	ap = 5,0 mm

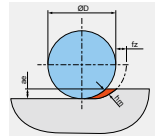
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 – 290	IN2530	200 – 240	0,04 – 0,08
	alloyed steel 800 N/mm ²	IN2505	210 – 250	IN2530	160 – 200	0,04 – 0,06
	alloyed steel 1100 N/mm ²	IN2505	160 – 180	IN2530	110 – 130	0,03
M	stainless steel	IN2505	120 – 180	IN2530	80 – 130	0,04 – 0,06
K	gray cast iron	IN2505	180 – 250	IN2530	150 – 200	0,04 – 0,08
	nodular cast iron	IN2505	140 – 210	IN2530	110 – 160	0,04 – 0,06
N	aluminum	IN05S	800 – 1500	IN05S	500 – 800	0,03 – 0,07
S	high temperature alloys	IN2505	110 – 125	IN2530	60 – 80	0,04
	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,04
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. uneven ground [mm]	max. ap/rev. [mm]	min. Bore dia. even ground [mm]	max. ap/rev. [mm]	max. Bore dia. even ground [mm]	max. ap/rev. [mm]
6	2,50	7,9	0,4	14,9	0,7	15,5	0,7
8	1,90	12,0	0,4	14,9	0,7	15,5	0,7
9	1,70	13,8	0,4	16,9	0,7	17,5	0,7
10	1,50	15,7	0,4	18,9	0,7	19,5	0,7
11	1,20	17,7	0,4	20,9	0,6	21,5	0,6
12	1,10	19,6	0,4	22,9	0,6	23,5	0,6
13	1,00	21,6	0,4	24,9	0,6	25,5	0,6
14	1,00	23,5	0,5	26,9	0,7	27,5	0,7
15	0,80	25,4	0,4	28,9	0,6	29,5	0,6
16	0,75	27,4	0,4	30,9	0,6	31,5	0,6
20	0,60	35,4	0,5	38,9	0,6	39,5	0,6

General information:

insert screw: **SM18-033-00**

torque: **0,5 Nm**

torque wrench: **DTN005S with bit DS-TP06TB**



Insert:	HNGU0605ANTR-M	HNGU0605ANTR-MM	HNGU0605ANTR-ML
Average chip thickness:	hm = 0,15 mm	hm = 0,20 mm	hm = 0,08 mm
max. cutting depth:	ap = 3,0 mm	ap = 3,0 mm	ap = 3,0 mm

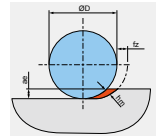
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 – 290	IN6537	200 – 240	0,15 – 0,35
	alloyed steel 800 N/mm ²	IN2505	210 – 250	IN6537	160 – 200	0,15 – 0,20
	alloyed steel 1100 N/mm ²	IN2505	160 – 180	IN6537	110 – 130	0,15
M	stainless steel	IN2505	120 – 180	IN2530	80 – 130	0,15 – 0,20
K	gray cast iron	IN2510	180 – 250	IN6537	150 – 200	0,15 – 0,35
	nodular cast iron	IN2510	140 – 210	IN6537	110 – 160	0,15 – 0,20
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2505	110 – 125	IN2530	60 – 80	0,15
	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,15
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



General information:

insert screw: **SM40-100-R0**

torque: **4,5 Nm**

torque wrench: **DTN045F with bit DS-T15B1**



Insert:	HNGU1007ANTR-M	HNGU1007ANTR-MM	HNGU1007ANTR-HR	HNGU1007ANTR-ML
Average chip thickness:	hm = 0,20 mm	hm = 0,30 mm	hm = 0,45 mm	hm = 0,08 mm
max. cutting depth:	ap = 5,0 mm	ap = 5,0 mm	ap = 5,0 mm	ap = 5,0 mm

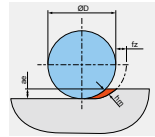
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 - 290	IN6537	200 - 240	0,20 - 0,30
	alloyed steel 800 N/mm ²	IN2505	210 - 250	IN6537	160 - 200	0,20 - 0,25
	alloyed steel 1100 N/mm ²	IN2505	160 - 180	IN6537	110 - 130	0,20
M	stainless steel	IN2505	120 - 180	IN2530	80 - 130	0,20 - 0,25
K	gray cast iron	IN2510	180 - 250	IN6537	150 - 200	0,20 - 0,70
	nodular cast iron	IN2510	140 - 210	IN6537	110 - 160	0,20 - 0,45
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2505	110 - 125	IN2530	60 - 80	0,20
	titanium alloys	IN2505	40 - 50	IN2530	30 - 40	0,20
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



General information:

insert screw: **SM50-130-R0**

torque: **6,0 Nm**

torque wrench: **DTNV00S with bit DS-T20TB**



Insert:	IEE211	IEE311	IEE312	IXE412	IXE413
Average chip thickness:	hm = 0,05 mm	hm = 0,05 mm	hm = 0,07 mm	hm = 0,10 mm	hm = 0,15 mm



Insert:	IXE414
Average chip thickness:	hm = 0,15 mm

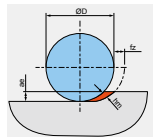
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	150 - 250	IN2530	120 - 200	hm x 1,2
	alloyed steel 800 N/mm ²	IN2505	120 - 180	IN2530	100 - 160	hm x 1,0
	alloyed steel 1100 N/mm ²	IN2505	100 - 180	IN2530	80 - 160	hm x 0,9
M	stainless steel	IN2530	80 - 160	IN2530	80 - 160	hm x 1,2
K	gray cast iron	IN2505	160 - 250	IN2530	140 - 200	hm x 1,2
	nodular cast iron	IN2505	120 - 200	IN2530	100 - 180	hm x 1,0
N	aluminum	IN055	500 - 1200	IN055	400 - 800	hm x 1,3
S	high temperature alloys	IN4035	50 - 80	IN4035	40 - 70	hm x 0,9
	titanium alloys	-	-	IN4035	30 - 40	hm x 1,0
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



General information:

insert screw size 211: **SM25-024-80**
 insert screw size 311: **SM35-034-50**
 insert screw size 312: **SM35-042-50**
 insert screw size 412: **SM40-040-50**
 insert screw size 413: **SM40-070-50**
 insert screw size 414: **SM40-080-50**

torque: **0,7 Nm**
 torque: **2,0 Nm**
 torque: **2,0 Nm**
 torque: **4,5 Nm**
 torque: **4,5 Nm**
 torque: **4,5 Nm**

torque wrench: **DTN01S with bit DS-T06TB**
 torque wrench: **DTN020S with bit DS-T09TB**
 torque wrench: **DTN020S with bit DS-T09TB**
 torque wrench: **DTN045F with bit DS-T15B**
 torque wrench: **DTN045F with bit DS-T15B**
 torque wrench: **DTN045F with bit DS-T15B**

Note

A large grid of graph paper for taking notes, consisting of 20 columns and 30 rows of small squares. The grid is bounded by a thin grey line on the top, left, and bottom, and a thick blue vertical bar on the right side.



Insert:	MNHU0402_R-M
Average chip thickness:	hm = 0,06 mm
max. cutting depth:	ap = 3,5 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 – 290	IN2530	200 – 240	0,06 – 0,10
	alloyed steel 800 N/mm ²	IN2505	210 – 250	IN2530	160 – 200	0,06 – 0,08
	alloyed steel 1100 N/mm ²	IN2505	160 – 180	IN2530	110 – 130	0,06
M	stainless steel	IN2505	120 – 180	IN2530	80 – 130	0,06 – 0,08
K	gray cast iron	IN2510	180 – 250	IN2530	150 – 200	0,06 – 0,10
	nodular cast iron	IN2510	140 – 210	IN2530	110 – 160	0,06 – 0,08
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2505	110 – 125	IN2530	60 – 80	0,06
	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,06
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

Ramping data and circular interpolation:

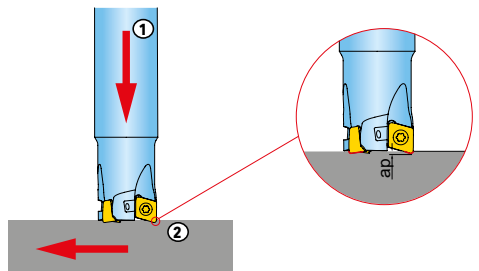
Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. uneven ground [mm]	max. ap/rev. [mm]	max. Bore dia. even ground [mm]	max. ap/rev. [mm]
10	4,2	15,4	1,2	19,2	2,1
11	4,1	17,1	1,3	21,2	2,2
12	4,0	18,8	1,4	23,2	2,4
13	3,9	20,5	1,6	25,2	2,6
16	4,6	25,1	2,3	31,3	3,5
20	3,1	33,0	2,2	39,2	3,2
25	2,4	42,9	2,3	49,2	3,1

The specified ramping data are maximum data that can only be achieved under optimal conditions - e.g. with smooth, short-chipping materials and the best possible chip removal. The values were determined for inserts with a corner radius R0.8. Deviations occur with other corner radii. Therefore, please always start with significantly lower ramping data (ramping angle or ap/U).

Slot Milling:

Tool diameter [mm]	max. ap* [mm]
10	0,6
11	0,7
12	0,7
13	0,7
16	0,7
20	0,6
25	0,6

*based on insert with R0,2



General information:

insert screw: **SM18-041-00**

torque: **0,5 Nm**

torque wrench: **DTN05S with bit DS-TP06TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	MNHU0603_R	MNHU0603_R-PH	MNCU0603_FR-P
Average chip thickness:	hm = 0,07 mm	hm = 0,05 mm	hm = 0,05 mm
max. cutting depth:	ap = 6 mm	ap = 6 mm	ap = 6 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]			Feed per tooth fz [mm]	
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 – 290	IN6537	200 – 240	0,07 – 0,15
	alloyed steel 800 N/mm ²	IN2505	210 – 250	IN2530	160 – 200	0,07 – 0,10
	alloyed steel 1100 N/mm ²	IN2505	160 – 180	IN2530	110 – 130	0,07
M	stainless steel	IN2035	120 – 180	IN2035	80 – 130	0,05 – 0,10
K	gray cast iron	IN2510	180 – 250	IN6537	150 – 200	0,07 – 0,15
	nodular cast iron	IN2510	140 – 210	IN6537	110 – 160	0,07 – 0,10
N	aluminum	IN10K	800 – 1500	IN10K	500 – 800	0,05 – 0,15
S	high temperature alloys	IN2035	110 – 125	IN2530	60 – 80	0,05
	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,05
H	hard machining < 54 HRC	IN2504	30 – 40	-	-	0,07
	hard machining < 63 HRC	-	-	-	-	-

Ramping data and circular interpolation:

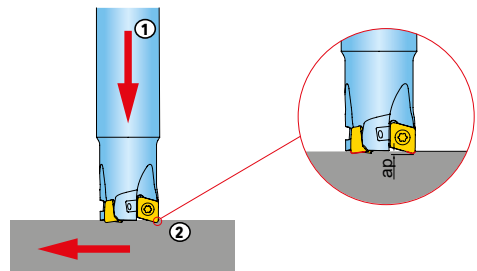
Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. uneven ground [mm]	max. ap/rev. [mm]	max. Bore dia. even ground [mm]	max. ap/rev. [mm]
16	3,5	21,5	0,8	32,0	2,5
20	4,2	29,5	1,8	40,0	3,7
25	4,9	39,5	3,1	50,0	5,4
32	3,5	53,5	3,3	64,0	4,9
40	2,6	69,5	3,4	80,0	4,6
50	2,0	89,5	3,5	100,0	4,4
63	1,5	115,5	3,5	126,0	4,1

The specified ramping data are maximum data that can only be achieved under optimal conditions - e.g. with smooth, short-chipping materials and the best possible chip removal. The values were determined for inserts with a corner radius R0.8. Deviations occur with other corner radii. Therefore, please always start with significantly lower ramping data (ramping angle or ap/U).

Slot Milling:

Tool diameter [mm]	max. ap* [mm]
16	0,7
20	1,0
25	1,5
32	1,5
40	1,5
50	1,5
63	1,5

*based on insert with R0,8



General information:

insert screw: **SM30-068-30**

torque: **1,1 Nm**

torque wrench: **DTN020S with bit DS-T08TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	MNHU0904_R-M	MNHU0904_R-ML	MNCU0904_FR-AL
Average chip thickness:	hm = 0,10 mm	hm = 0,08 mm	hm = 0,05 mm
max. cutting depth:	ap = 8,0 mm	ap = 8,0 mm	ap = 8,0 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]		Feed per tooth fz [mm]		
		1st choice dry machining resp. wear resistant carbide	1st choice wet machining resp. tough carbide			
P	unalloyed steel	IN2505	250 – 290	IN6537	200 – 240	0,10 – 0,15
	alloyed steel 800 N/mm ²	IN2505	210 – 250	IN2530	160 – 200	0,10 – 0,12
	alloyed steel 1100 N/mm ²	IN2505	160 – 180	IN2530	110 – 130	0,10
M	stainless steel	IN2505	120 – 180	IN2530	80 – 130	0,08 – 0,12
K	gray cast iron	IN2510	180 – 250	IN6537	150 – 200	0,10 – 0,15
	nodular cast iron	IN2510	140 – 210	IN6537	110 – 160	0,10 – 0,12
N	aluminum	IN10K	800 – 1500	IN10K	500 – 800	0,05 – 0,15
S	high temperature alloys	IN2505	110 – 125	IN2530	60 – 80	0,08
	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,08
H	hard machining < 54 HRC	IN2504	30 – 40	-	-	0,10
	hard machining < 63 HRC	-	-	-	-	-

Ramping data and circular interpolation:

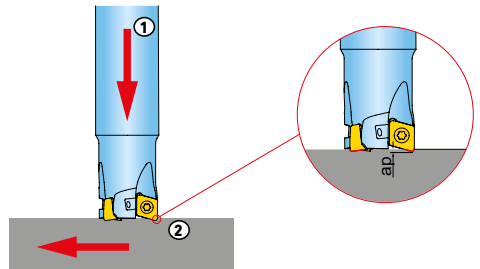
Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. uneven ground [mm]	max. ap/rev. [mm]	max. Bore dia. even ground [mm]	max. ap/rev. [mm]
20	3,5	32	2,3	38,5	3,5
25	4,7	38,1	3,3	48,4	6,0
32	4,9	48,4	4,4	62,4	8,0
40	3,7	64,1	4,8	78,4	7,8
50	2,8	83,9	5,2	98,4	7,4
63	2,1	109,7	5,3	124,4	7,0
80	1,6	143,5	5,5	158,4	6,8

The specified ramping data are maximum data that can only be achieved under optimal conditions - e.g. with smooth, short-chipping materials and the best possible chip removal. The values were determined for inserts with a corner radius R0.8. Deviations occur with other corner radii. Therefore, please always start with significantly lower ramping data (ramping angle or ap/U).

Slot Milling:

Tool diameter [mm]	max. ap* [mm]
20	0,9
25	0,9
32	0,9
40	0,9
50	0,9
63	0,9
80	0,9

* based on insert with R0.8



General information:

insert screw: **SM35-088-10**

torque: **3,0 Nm**

torque wrench: **DTN020S with bit DS-T10TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	MNHU1106_R-M	MNHU1106_RPNR-M	MNCU1106_FR-AL
Average chip thickness:	hm = 0,10 mm	hm = 0,10 mm	hm = 0,05 mm
max. cutting depth:	ap = 10,5 mm	ap = 10,5 mm	ap = 10,5 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 - 290	IN6537	200 - 240	0,10 - 0,15
	alloyed steel 800 N/mm ²	IN2505	210 - 250	IN2530	160 - 200	0,10 - 0,12
	alloyed steel 1100 N/mm ²	IN2505	160 - 180	IN2530	110 - 130	0,10
M	stainless steel	IN2505	120 - 180	IN2530	80 - 130	0,10 - 0,12
K	gray cast iron	IN2510	180 - 250	IN6537	150 - 200	0,10 - 0,15
	nodular cast iron	IN2510	140 - 210	IN6537	110 - 160	0,10 - 0,12
N	aluminum	IN10K	800 - 1500	IN10K	500 - 800	0,05 - 0,15
S	high temperature alloys	IN2505	110 - 125	IN2530	60 - 80	0,10
	titanium alloys	IN2505	40 - 50	IN2530	30 - 40	0,10
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

Ramping data and circular interpolation:

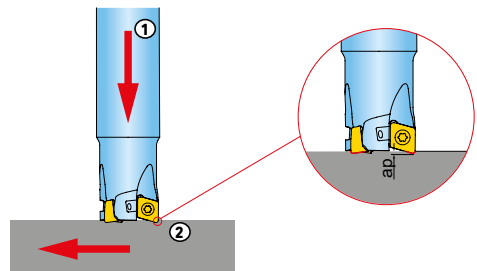
Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. uneven ground [mm]	max. ap/rev. [mm]	max. Bore dia. even ground [mm]	max. ap/rev. [mm]
25	5,0	37,7	3,4	48,5	6,4
32	5,3	47,8	4,6	62,5	8,8
40	5,0	60,3	5,5	78,5	10,5
50	3,8	80,2	6,3	98,5	10,1
63	2,9	105,8	6,8	124,5	9,7
80	2,2	139,6	7,1	158,4	9,4
100	1,7	179,4	7,4	198,4	9,1

The specified ramping data are maximum data that can only be achieved under optimal conditions - e.g. with smooth, short-chipping materials and the best possible chip removal. The values were determined for inserts with a corner radius R0.8. Deviations occur with other corner radii. Therefore, please always start with significantly lower ramping data (ramping angle or ap/U).

Slot Milling:

Tool diameter [mm]	max. ap* [mm]
25	1,4
32	1,3
40	1,2
50	1,2
63	1,2
80	1,2
100	1,2

* based on insert with R0,8



General information:

insert screw: **SM40-100-10**

torque: **4,5 Nm**

torque wrench: **DTN045F with bit DS-T15B1**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	MNHU1407_R-M	MNHU1407_RPNR-M	MNCU1407_FR-AL
Average chip thickness:	hm = 0,13 mm	hm = 0,13 mm	hm = 0,05 mm
max. cutting depth:	ap = 13,5 mm	ap = 13,5 mm	ap = 13,5 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 - 290	IN6537	200 - 240	0,13 - 0,18
	alloyed steel 800 N/mm ²	IN2505	210 - 250	IN2530	160 - 200	0,13 - 0,15
	alloyed steel 1100 N/mm ²	IN2505	160 - 180	IN2530	110 - 130	0,13
M	stainless steel	IN2505	120 - 180	IN2530	80 - 130	0,13 - 0,15
K	gray cast iron	IN2510	180 - 250	IN6537	150 - 200	0,13 - 0,18
	nodular cast iron	IN2510	140 - 210	IN6537	110 - 160	0,13 - 0,15
N	aluminum	IN10K	800 - 1500	IN10K	500 - 800	0,05 - 0,18
S	high temperature alloys	IN2505	110 - 125	IN2530	60 - 80	0,13
	titanium alloys	IN2505	40 - 50	IN2530	30 - 40	0,13
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

Ramping data and circular interpolation:

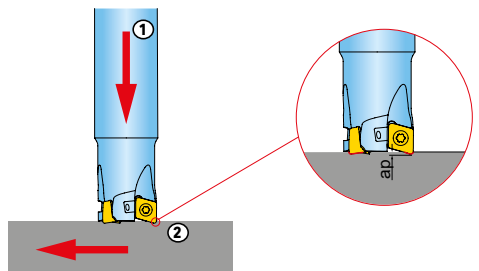
Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. uneven ground [mm]	max. ap/rev. [mm]	max. Bore dia. even ground [mm]	max. ap/rev. [mm]
50	5,4	74,5	7,2	98,5	13,5
63	3,9	100,3	7,9	124,5	13,1
80	2,9	134,3	8,6	158,5	12,4
100	2,2	174,2	8,9	198,4	11,8
125	1,7	224,2	9,2	248,4	11,5

The specified ramping data are maximum data that can only be achieved under optimal conditions - e.g. with smooth, short-chipping materials and the best possible chip removal. The values were determined for inserts with a corner radius R0.8. Deviations occur with other corner radii. Therefore, please always start with significantly lower ramping data (ramping angle or ap/U).

Slot Milling:

Tool diameter [mm]	max. ap* [mm]
50	1,5
63	1,3
80	1,3
100	1,3
125	1,3

* based on insert with R0,8



General information:

insert screw: **SM50-127-10**

torque: **6,0 Nm**

torque wrench: **DTNV00S with bit DS-T20TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	OFMT053AFN-HR	OFCT053AFN-HR	OFCT053TN-HS	OFMW053AFTN	OFCT053AFFN-P
Average chip thickness:	hm = 0,12 mm	hm = 0,10 mm	hm = 0,12 mm	hm = 0,20 mm	hm = 0,05 mm
max. cutting depth:	ap = 3,4 mm	ap = 3,4 mm	ap = 3,4 mm	ap = 3,4 mm	ap = 3,4 mm

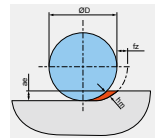
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN4030	250 - 290	IN2530	200 - 240	0,12 - 0,25
	alloyed steel 800 N/mm ²	IN2505	210 - 250	IN2530	160 - 200	0,12 - 0,20
	alloyed steel 1100 N/mm ²	IN2505	160 - 180	IN2530	110 - 130	0,12
M	stainless steel	IN2035	120 - 180	IN2035	80 - 130	0,12 - 0,25
K	gray cast iron	IN2510	180 - 250	IN4030	150 - 200	0,20 - 0,40
	nodular cast iron	IN2510	140 - 210	IN4030	110 - 160	0,20 - 0,30
N	aluminum	IN10K	800 - 1500	IN10K	500 - 800	0,05 - 0,30
S	high temperature alloys	IN2035	110 - 125	IN2035	60 - 80	0,12
	titanium alloys	IN2505	40 - 50	IN2035	30 - 40	0,12
H	hard machining < 54 HRC	IN2505	30 - 40	-	-	0,20
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. uneven ground [mm]	max. ap/rev. [mm]	min. Bore dia. even ground [mm]	max. ap/rev. [mm]	max. Bore dia. even ground [mm]	max. ap/rev. [mm]	max. Bore dia. uneven ground [mm]	max. ap/rev. [mm]
24,3	15,6	39,8	3,4	53,2	3,4	55,8	3,4	64,1	3,4
32	9,1	55,7	3,4	69,1	3,4	71,7	3,4	80,1	3,4
33	8,7	57,3	3,4	70,7	3,4	73,4	3,4	81,6	3,4
40	6,4	71,8	3,4	85,0	3,4	87,7	3,4	96,1	3,4
50	4,7	91,7	3,4	105,0	3,4	107,6	3,4	116,1	3,4
63	3,4	117,7	3,4	131,0	3,4	133,6	3,4	142,0	3,4
80	2,6	151,6	3,4	165,0	3,4	167,6	3,4	176,0	3,4
100	2,0	191,6	3,4	205,0	3,4	207,6	3,4	216,0	3,4
125	1,5	241,0	3,4	255,0	3,4	257,5	3,4	265,9	3,4

General information:

insert screw 5N6H wide pitch: **SM40-093-20**
torque: **4,5 Nm**

insert screw 5N5H fine pitch: **SM40-100-R0**
torque wrench: **DTN045F with bit DS-T15B1**



Insert:	OFMT0705AFR-HR	OFCT0705AFN-HR	OFMT0705AFIN	OFMW0705AFIN	OFCT0705AFFN-P
Average chip thickness:	hm = 0,15 mm	hm = 0,18 mm	hm = 0,15 mm	hm = 0,25 mm	hm = 0,05 mm
max. cutting depth:	ap = 4,8 mm	ap = 4,8 mm	ap = 4,8 mm	ap = 4,8 mm	ap = 4,8 mm



Insert:	OFCT0705AFFR-W
Average chip thickness:	fu = 3,8 mm
max. cutting depth:	ap = 4,8 mm

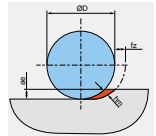
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN4030	250 - 290	IN4030	200 - 240	0,15 - 0,30
	alloyed steel 800 N/mm ²	IN4005	210 - 250	IN4030	160 - 200	0,15 - 0,25
	alloyed steel 1100 N/mm ²	IN4005	160 - 180	IN4030	110 - 130	0,15
M	stainless steel	IN6535	120 - 180	IN2035	80 - 130	0,18 - 0,30
K	gray cast iron	IN4005	180 - 250	IN4030	150 - 200	0,25 - 0,50
	nodular cast iron	IN4005	140 - 210	IN4030	110 - 160	0,25 - 0,40
N	aluminum	IN055	800 - 1500	IN055	500 - 800	0,05 - 0,30
S	high temperature alloys	IN6535	110 - 125	IN2035	60 - 80	0,18
	titanium alloys	IN4005	40 - 50	IN2035	30 - 40	0,18
H	hard machining < 54 HRC	IN4005	30 - 40	-	-	0,25
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. uneven ground [mm]	max. ap/rev. [mm]	min. Bore dia. even ground [mm]	max. ap/rev. [mm]	max. Bore dia. even ground [mm]	max. ap/rev. [mm]	max. Bore dia. uneven ground [mm]	max. ap/rev. [mm]
50	7,1	90,8	4,8	108,9	4,8	112	4,8	124,6	4,8
63	5,1	116,6	4,8	134,8	4,8	137,9	4,8	150,5	4,8
80	3,7	150,4	4,8	168,8	4,8	171,9	4,8	184,4	4,8
100	2,8	190,2	4,8	208,6	4,8	211,7	4,8	224,2	4,8
125	2,2	240,3	4,8	258,7	4,8	261,8	4,8	274,3	4,8
160	1,6	310,1	4,8	328,6	4,8	331,7	4,8	344,1	4,8

General information:

insert screw 5N6L wide pitch: **SM50-120-30**
torque: **7,5 Nm**

insert screw 5N5L fine pitch: **SM50-130-R0**
torque wrench: **DTN005 with bit DS-T20TB**



Insert:	ONCU0505ANTN-HR	ONCU050520TN	ONCU0505ANEN	ONCU0505ANFN-P	ONCU0505ANN
Average chip thickness:	hm = 0,22 mm	hm = 0,25 mm	hm = 0,08 mm	hm = 0,05 mm	hm = 0,15 mm
max. cutting depth:	ap = 3 mm	ap = 1,5 - 2,5 mm	ap = 3 mm	ap = 3 mm	ap = 3 mm



Insert:	ONCU0505ANTN-W
Average chip thickness:	fu = 2,4 mm
max. cutting depth:	ap = 3 mm

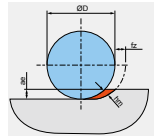
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]		Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide	1st choice wet machining resp. tough carbide	
P	unalloyed steel	IN6537	250 - 290	0,22 - 0,40
	alloyed steel 800 N/mm ²	IN2505	210 - 250	0,22 - 0,30
	alloyed steel 1100 N/mm ²	IN2505	160 - 180	0,22
M	stainless steel	IN2035	120 - 180	0,08 - 0,30
K	gray cast iron	IN70N	600 - 900	0,08 - 0,40
	nodular cast iron	IN4010	140 - 120	0,22 - 0,30
N	aluminum	IN10K	800 - 1500	0,05 - 0,30
S	high temperature alloys	IN2035	110 - 125	0,08
	titanium alloys	IN2505	40 - 50	0,22
H	hard machining < 54 HRC	IN2504	30 - 40	0,22
	hard machining < 63 HRC	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



General information:

insert screw: **SM40-100-10**

torque: **4,5 Nm**

torque wrench: **DTN045F with bit DS-T15B1**



Insert:	ONCU090612TN-HR	ONCU090638TN-HR	ONCU0906ANTN-HR	ONCU090630TN	ONCQ090612TN
Average chip thickness:	hm = 0,30 mm	hm = 0,30 mm	hm = 0,30 mm	hm = 0,30 mm	hm = 0,22 mm
max. cutting depth:	ap = 5 mm	ap = 5 mm	ap = 5 mm	ap = 2,3 - 4,0 mm	ap = 5 mm



Insert:	ONCU0906ANFN-WE	ONCQ0906ANN	ONCU090612FN-P	ONCU090612TN-W	ONCU0906ANTN-W
Average chip thickness:	hm = 0,8 mm	hm = 0,15 mm	hm = 0,05 mm	fu max = 2,4 mm	fu max = 2,4 mm
max. cutting depth:	ap = 5 mm	ap = 5 mm	ap = 5 mm	ap = 3,9 mm	ap = 3,8 mm

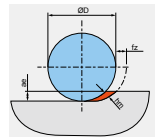
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]		Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide	1st choice wet machining resp. tough carbide	
P	unalloyed steel	IN4030	250 - 290	0,30 - 0,40
	alloyed steel 800 N/mm ²	IN4005	210 - 250	0,30 - 0,35
	alloyed steel 1100 N/mm ²	IN4005	160 - 180	0,30
M	stainless steel	IN2035	120 - 180	0,08 - 0,30
K	gray cast iron	IN70N	600 - 900	0,08 - 0,40
	nodular cast iron	IN4010	140 - 210	0,30 - 0,35
N	aluminum	IN10K	800 - 1500	0,05 - 0,40
S	high temperature alloys	IN2035	110 - 125	0,08
	titanium alloys	IN4005	40 - 50	0,30
H	hard machining < 54 HRC	IN4005	30 - 40	0,30
	hard machining < 63 HRC	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



General information:

for screw clamping:

insert screw: **SM50-130-R0**

torque: **6,0 Nm**

torque wrench: **DTNV005 with bit DS-T20TB**

for wedge clamping:

insert screw: **SB080-03**

torque: **6,0 Nm**

torque wrench: **DTNV005 with bit DS-H04TB**



Insert:	PNMU0503GNTR	PNCU0503GNFR-P
Average chip thickness:	hm = 0,10 mm	hm = 0,05 mm
max. cutting depth:	ap = 3,8 mm	ap = 3,8 mm

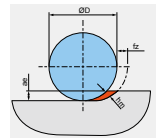
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 – 290	IN2530	200 – 240	0,10 – 0,25
	alloyed steel 800 N/mm ²	IN2505	210 – 250	IN2530	160 – 200	0,10 – 0,20
	alloyed steel 1100 N/mm ²	IN2505	160 – 180	IN2530	110 – 130	0,10
M	stainless steel	IN2035	120 – 180	IN2035	80 – 130	0,10 – 0,25
K	gray cast iron	IN2505	180 – 250	IN2530	150 – 200	0,10 – 0,30
	nodular cast iron	IN2505	140 – 210	IN2530	110 – 160	0,10 – 0,20
N	aluminum	IN10K	800 – 1500	IN10K	500 – 800	0,05 – 0,35
S	high temperature alloys	IN2035	110 – 125	IN2530	60 – 80	0,10
	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,10
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



General information:

insert screw: **SM25-064-00**

torque: **1,1 Nm**

torque wrench: **DTN011S with bit DS-T08TB**



Insert:	PNCU0805GNTR	PNCU0805GNFR-HS	PNCQ0804GNTN	PNCU0805GNFR-P	PNCU0805GNR
Average chip thickness:	hm = 0,20 mm	hm = 0,08 mm	hm = 0,20 mm	hm = 0,05 mm	hm = 0,10 mm
max. cutting depth:	ap = 6 mm	ap = 6 mm	ap = 6 mm	ap = 6 mm	ap = 6 mm



Insert:	PNCU0805GNTR-W
Average chip thickness:	fu max = 3,6 mm
max. cutting depth:	ap = 6 mm

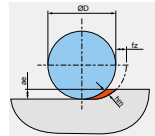
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN6537	250 - 290	IN6537	200 - 240	0,20 - 0,40
	alloyed steel 800 N/mm ²	IN4005	210 - 250	IN6537	160 - 200	0,20 - 0,30
	alloyed steel 1100 N/mm ²	IN4005	160 - 180	IN6537	110 - 130	0,20
M	stainless steel	IN2035	120 - 180	IN2035	80 - 130	0,08 - 0,30
K	gray cast iron	IN70N	600 - 900	IN6537	150 - 200	0,10 - 0,40
	nodular cast iron	IN4015	140 - 210	IN6537	110 - 160	0,20 - 0,30
N	aluminum	IN05S	800 - 1500	IN05S	500 - 800	0,05 - 0,40
S	high temperature alloys	IN2035	110 - 125	IN2035	60 - 80	0,08
	titanium alloys	IN2505	40 - 50	IN2035	30 - 40	0,20
H	hard machining < 54 HRC	IN2505	30 - 40	-	-	0,20
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



General information:

insert screw: **SM40-100-10**

torque: **4,5 Nm**

torque wrench: **DTN045F with bit DS-T15B1**



Insert:	SDE_
Average chip thickness:	hm = 0,13 mm
max. cutting width:	4 - 15

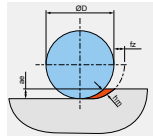
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN4030	250 - 290	IN4030	200 - 240	0,13 - 0,25
	alloyed steel 800 N/mm ²	IN4030	210 - 250	IN4030	160 - 200	0,13 - 0,20
	alloyed steel 1100 N/mm ²	IN4030	160 - 180	IN4030	110 - 130	0,13
M	stainless steel	IN4030	120 - 180	IN4030	80 - 130	0,13 - 0,20
K	gray cast iron	IN4030	180 - 250	IN4030	150 - 200	0,13 - 0,25
	nodular cast iron	IN4030	140 - 210	IN4030	110 - 160	0,13 - 0,20
N	aluminum	IN30M	500 - 800	IN30M	500 - 800	0,13 - 0,25
S	high temperature alloys	IN4030	110 - 125	IN4030	60 - 80	0,13
	titanium alloys	IN4030	40 - 50	IN4030	30 - 40	0,13
H	hard machining < 54 HRC	IN4030	30 - 40	-	-	0,13
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



General information:

insert screw for cutting width 4 mm: **SM35-034-50**
 insert screw for cutting width 5 mm: **SM35-042-50**
 insert screw for cutting width 6 mm: **SM40-050-50**
 insert screw for cutting width 7 - 8 mm: **SM40-060-50**
 insert screw for cutting width 9 - 10 mm: **SM40-080-50**
 insert screw for cutting width 12 - 15 mm: **SM40-106-50**

torque: **1,4 Nm**
 torque: **1,4 Nm**
 torque: **4,5 Nm**
 torque: **4,5 Nm**
 torque: **4,5 Nm**
 torque: **4,5 Nm**

torque wrench: **DTNV02S with bit DS-T09B**
 torque wrench: **DTNV02S with bit DS-T09B**
 torque wrench: **DTN045F with bit DS-T15B**
 torque wrench: **DTN045F with bit DS-T15B**
 torque wrench: **DTN045F with bit DS-T15B**
 torque wrench: **DTN045F with bit DS-T15B**



Insert:	SGM-44R001
Average chip thickness:	hm = 0,18 mm
max. cutting depth:	ap = 8,7 mm

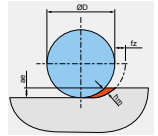
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				average chip thickness hm [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	150 – 250	IN2530	120 – 200	hm x 1,2
	alloyed steel 800 N/mm ²	IN4005	120 – 180	IN4030	100 – 160	hm x 1,0
	alloyed steel 1100 N/mm ²	IN4005	100 – 180	IN4030	80 – 160	hm x 0,9
M	stainless steel	IN4030	80 – 160	IN2530	80 – 160	hm x 1,2
K	gray cast iron	IN4015	160 – 250	IN4030	140 – 200	hm x 1,2
	nodular cast iron	IN4015	120 – 200	IN4015	100 – 180	hm x 1,0
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2530	50 – 80	IN2530	40 – 70	hm x 0,9
	titanium alloys	-	-	IN2530	30 – 40	hm x 1,0
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



General information:

insert screw: **SM40-120-20**

torque: **4,5 Nm**

torque wrench: **DTN045F with bit DS-T15B**



Insert:	SDMT050204N	SDCT050204FN-P
Average chip thickness:	hm = 0,06 mm	hm = 0,05 mm
max. cutting depth:	ap = 4,6 mm	ap = 4,6 mm

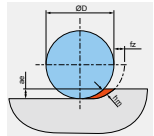
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 - 290	IN2505	200 - 240	0,06 - 0,12
	alloyed steel 800 N/mm ²	IN2505	210 - 250	IN2505	160 - 200	0,06 - 0,10
	alloyed steel 1100 N/mm ²	IN2505	160 - 180	IN2505	110 - 130	0,06
M	stainless steel	IN2505	120 - 180	IN2505	80 - 130	0,06 - 0,12
K	gray cast iron	IN2505	180 - 250	IN2505	150 - 200	0,06 - 0,15
	nodular cast iron	IN2505	140 - 210	IN2505	110 - 160	0,06 - 0,12
N	aluminum	IN10K	800 - 1500	IN10K	500 - 800	0,05 - 0,15
S	high temperature alloys	IN2505	110 - 125	IN2505	60 - 80	0,06
	titanium alloys	IN2505	40 - 50	IN2505	30 - 40	0,06
H	hard machining < 54 HRC	IN2505	30 - 40	-	-	0,06
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



General information:

insert screw: **SM20-043-00**

torque: **0,7 Nm**

torque wrench: **DTNV01S with bit DS-TP06TB**



Insert:	SDMT080305N	SDMW080305TN	SDCT080305FN-P	SDMW080305TN-W
Average chip thickness:	hm = 0,13 mm	hm = 0,13 mm	hm = 0,05 mm	hm = 0,13 mm
max. cutting depth:	ap = 7,5 mm	ap = 7,5 mm	ap = 7,5 mm	ap = 3,0 mm

Recommended cutting data:

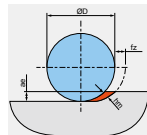
ISO	Material	Cutting speed Vc [m/min]*		Feed per tooth fz [mm]		
		1st choice dry machining resp. wear resistant carbide	1st choice wet machining resp. tough carbide			
P	unalloyed steel	IN4030	250 – 290	IN4030	200 – 240	0,13 – 0,20
	alloyed steel 800 N/mm ²	IN4030	210 – 250	IN4030	160 – 200	0,13 – 0,15
	alloyed steel 1100 N/mm ²	IN4030	160 – 180	IN4030	110 – 130	0,13
M	stainless steel	IN4030	120 – 180	IN4030	80 – 130	0,13 – 0,20
K	gray cast iron	IN4030	180 – 250	IN4030	150 – 200	0,13 – 0,20
	nodular cast iron	IN4030	140 – 210	IN4030	110 – 160	0,13 – 0,15
N	aluminum	IN055	800 – 1500	IN055	500 – 800	0,05 – 0,20
S	high temperature alloys	IN2505	110 – 125	IN2505	60 – 80	0,13
	titanium alloys	IN2505	40 – 50	IN2505	30 – 40	0,13
H	hard machining < 54 HRC	IN2505	30 – 40	-	-	0,13
	hard machining < 63 HRC	-	-	-	-	-

*When using T-slot cutters, a starting value of Vc=80 m/min has proven to be effective for material groups P, M and K. Pay attention to the best possible chip removal.

Tips:

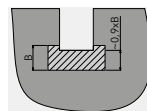
- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Application example T-slot cutter:

Under normal conditions, initially leave the T-slot bottom largely unmachined. Strong coolant or compressed air supply makes chip removal easier.



Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. uneven ground [mm]	max. ap/rev. [mm]	max. Bore dia. even ground [mm]	max. ap/rev. [mm]
16	5,2	20,0	1,1	31	4,2
18	4,7	22,7	1,2	35	4,3
20	3,8	26,1	1,2	39	3,9
25	2,3	35,5	1,3	49	3,0
32	1,3	49,4	1,2	63	2,2
40	1,3	65,2	1,7	79	2,7
50	0,9	85,2	1,7	99	2,4
63	0,6	111,2	1,5	125	2,0
80	0,4	145,2	1,4	159	1,7

General information:

insert screw: **SM30-065-00**

torque: **2,0 Nm**

torque wrench: **DTN0205 with bit DS-T09TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	SDES1305_N	SDMS1305_R-PH
Average chip thickness:	hm = 0,20 mm	hm = 0,10 mm
max. cutting depth:	ap = 11,3 mm	ap = 11,3 mm

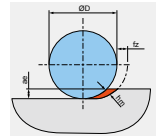
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 – 290	IN2530	200 – 240	0,10 – 0,35
	alloyed steel 800 N/mm ²	IN2505	210 – 250	IN2530	160 – 200	0,10 – 0,30
	alloyed steel 1100 N/mm ²	IN2505	160 – 180	IN2530	110 – 130	0,10 – 0,25
M	stainless steel	IN4035	120 – 180	IN2530	80 – 130	0,08 – 0,25
K	gray cast iron	IN2505	180 – 250	IN4030	150 – 200	0,10 – 0,35
	nodular cast iron	IN2505	140 – 210	IN4030	110 – 160	0,10 – 0,35
N	aluminum	IN2505	800 – 1500	IN2505	500 – 800	0,08 – 0,25
S	high temperature alloys	IN4035	110 – 125	IN4035	60 – 80	0,08 – 0,18
	titanium alloys	IN4035	40 – 50	IN4035	30 – 40	0,08 – 0,18
H	hard machining < 54 HRC	IN2505	30 – 40	-	-	0,08
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. uneven ground [mm]	max. ap/rev. [mm]	max. Bore dia. even ground [mm]	max. ap/rev. [mm]
50	3,7	77,2	5,5	97	9,5
63	2	103,2	4,4	123	6,5
80	1,3	137,2	4,0	157	5,4
100	1	177,1	4,2	197	5,3
125	0,7	227,1	3,9	247	4,6

General information:

insert screw: **SM40-100-R0**

torque: **4,5 Nm**

torque wrench: **DTN045F with bit DS-T15B1**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.

Note

A large grid of graph paper for taking notes, consisting of 20 columns and 30 rows of small squares. The grid is empty and occupies most of the page.



Insert:	SHET1105_FR-P
Average chip thickness:	hm = 0,05 mm
max. cutting depth:	ap = 8,4 mm

Recommended cutting data:

ISO	Material	Remark	Hardness [HB]	Carbide grade	Cutting speed Vc [m/min]	Feed per tooth fz [mm]	
N	wrought aluminum alloys	not hardened	60	IN10K	300 - 5000	0,05 - 0,30	
		hardened	100	IN10K	200 - 2000	0,05 - 0,20	
	<= 12% Si aluminum cast alloys	not hardened	75	IN10K	200 - 2000	0,05 - 0,25	
		hardened	90	IN10K	200 - 1500	0,05 - 0,20	
	> 12% Si	high temperature application	130	IN10K	200 - 1000	0,05 - 0,10	
		well machinable	110	IN10K	200 - 800	0,05 - 0,10	
	> 1% Pb copper alloys	brass	90	IN10K	300 - 1000	0,05 - 0,10	
		electrolytic copper	100	IN10K	300 - 800	0,05 - 0,10	
	nonmetal	duroplasts, fibre reinforced plastics	-	-	IN10K	100 - 500	0,05 - 0,10
		ebonite	-	-	IN10K	100 - 300	0,05 - 0,10

max. permissible speed:

Tool diameter [mm]	max. no. of revolutions n [min ⁻¹]	max. cutting speed Vc [m/min]
25	30800	2400
32	25200	2500
40	25800	3200
50	23100	3600
63	20500	4000
80	18200	4500
100	16300	5100

Important remarks:

The specified maximum speeds are valid only under optimal conditions.

These include in particular:

- Please use for mounting of the inserts only a torque wrench with 4.5 Nm.
- The tool has to be balanced only when completely mounted and joined with adaption.
- Please use only correct and as good as new insert screws.
- Please avoid to extend the cutters.
- Please use high speed cutters only on encapsulated machines.

Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. uneven ground [mm]	max. ap/rev. [mm]	min. Bore dia. even ground [mm]	max. ap/rev. [mm]	max. Bore dia. even ground [mm]	max. ap/rev. [mm]
25	7,0	30	1,9	45	7,7	49	8,4
32	6,0	44	4,0	59	8,4	63	8,4
40	4,0	60	4,4	75	7,7	79	8,4
50	2,0	80	3,3	95	4,9	99	5,4
63	1,0	106	2,4	121	3,2	125	3,4
80	0,5	140	1,6	155	2,1	159	2,2
100	0,5	180	2,2	195	2,6	199	2,7

General information:

insert screw: **SM40-120-20**

torque: **4,5 Nm**

torque wrench: **DTN045F with bit DS-T15B**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	SNCG11T3_TN-HR	SNCN11T3_TN
Average chip thickness:	hm = 0,18 mm	hm = 0,18 mm
max. cutting depth:	ap = 8,7 mm	ap = 7 mm

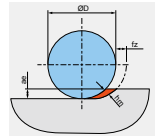
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN4030	250 - 290	IN4030	200 - 240	0,20 - 0,35
	alloyed steel 800 N/mm ²	IN4005	210 - 250	IN4030	160 - 200	0,20 - 0,30
	alloyed steel 1100 N/mm ²	IN4005	160 - 180	IN4030	110 - 130	0,20
M	stainless steel	IN4005	120 - 180	IN4030	80 - 130	0,20 - 0,35
K	gray cast iron	IN70N	600 - 900	IN4030	150 - 200	0,20 - 0,35
	nodular cast iron	IN2504	140 - 210	IN4030	110 - 160	0,20 - 0,30
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN4005	110 - 125	IN4030	60 - 80	0,20
	titanium alloys	IN4005	40 - 50	IN4030	30 - 40	0,20
H	hard machining < 54 HRC	IN2504	30 - 40	-	-	0,20
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



General information:

insert screw: **SB40-03**

torque: **2,0 Nm**

torque wrench: **DTN020S with bit DS-T09TB**



Insert:	SQGU0704_TR-M	SQGU0703_R-ML
Average chip thickness:	hm = 0,10 mm	hm = 0,10 mm
max. cutting depth:	ap = 5,0 mm	ap = 5,0 mm

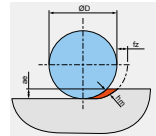
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 – 290	IN2530	200 – 240	0,10 – 0,15
	alloyed steel 800 N/mm ²	IN2505	210 – 250	IN2530	160 – 200	0,10 – 0,12
	alloyed steel 1100 N/mm ²	IN2505	160 – 180	IN2530	110 – 130	0,10
M	stainless steel	IN2505	120 – 180	IN2530	80 – 130	0,10 – 0,12
K	gray cast iron	IN2510	180 – 250	IN2530	150 – 200	0,10 – 0,20
	nodular cast iron	IN2510	140 – 210	IN2530	110 – 160	0,10 – 0,12
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2505	110 – 125	IN2530	60 – 80	0,10
	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,10
H	hard machining < 54 HRC	IN2504	30 – 40	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



General information:

insert screw: **SM25-060-90**

torque: **1,1 Nm**

torque wrench: **DTN011S with bit DS-TP07TB**



Insert:	SQGU100408TR-M	SQGU100408PNR-MP	SQGU100408PNR-MMP
Average chip thickness:	hm = 0,10 mm	hm = 0,10 mm	hm = 0,10 mm
max. cutting depth:	ap = 7,0 mm	ap = 7,0 mm	ap = 7,0 mm

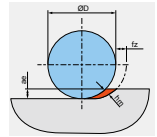
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 - 290	IN2530	200 - 240	0,10 - 0,15
	alloyed steel 800 N/mm ²	IN2505	210 - 250	IN2530	160 - 200	0,10 - 0,12
	alloyed steel 1100 N/mm ²	IN2505	160 - 180	IN2530	110 - 130	0,10
M	stainless steel	IN2505	120 - 180	IN2530	80 - 130	0,10 - 0,12
K	gray cast iron	IN2510	180 - 250	IN2530	150 - 200	0,10 - 0,20
	nodular cast iron	IN2510	140 - 210	IN2530	110 - 160	0,10 - 0,12
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2505	110 - 125	IN2530	60 - 80	0,10
	titanium alloys	IN2505	40 - 50	IN2530	30 - 40	0,10
H	hard machining < 54 HRC	IN2504	30 - 40	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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

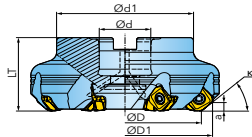


General information:

insert screw: **TS 35A0881/HG**

torque: **3,0 Nm**

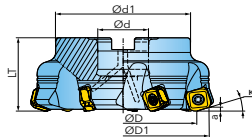
torque wrench: **DTN030S with bit DS-TP10TB**



Insert:	SQGU1105ANR-M
Average chip thickness:	hm = 0,20 mm
max. cutting depth:	ap = 4,2 mm

Recommended cutting data face mill (DN_G) lead angle K= 45°:

ISO	Material	Cutting speed Vc [m/min]		Feed per tooth fz [mm]		
		1st choice dry machining resp. wear resistant carbide	1st choice wet machining resp. tough carbide			
P	unalloyed steel	IN2505	250 - 290	IN6537	200 - 240	0,20 - 0,35
	alloyed steel 800 N/mm ²	IN2505	210 - 250	IN6537	160 - 200	0,20 - 0,30
	alloyed steel 1100 N/mm ²	IN2505	160 - 180	IN6537	110 - 130	0,20
M	stainless steel	IN2505	120 - 180	IN6537	80 - 130	0,20 - 0,30
K	gray cast iron	IN2510	180 - 250	IN2510	150 - 200	0,20 - 0,35
	nodular cast iron	IN2510	140 - 210	IN2510	110 - 160	0,20 - 0,30
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2505	110 - 125	IN2505	60 - 80	0,20
	titanium alloys	IN2505	40 - 50	IN2505	30 - 40	0,20
H	hard machining < 54 HRC	IN2505	30 - 40	-	-	0,20
	hard machining < 63 HRC	-	-	-	-	-



Insert:	SQGU1105ANR-M
Average chip thickness:	hm = 0,30 mm
max. cutting depth:	ap = 1,8 mm

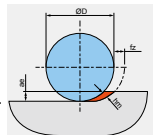
Recommended cutting data high feed cutting (DG6G) lead angle K= 20°:

ISO	Material	Cutting speed Vc [m/min]		Feed per tooth fz [mm]		
		1st choice dry machining resp. wear resistant carbide	1st choice wet machining resp. tough carbide			
P	unalloyed steel	IN2505	250 - 290	IN6537	200 - 240	0,30 - 1,50
	alloyed steel 800 N/mm ²	IN2505	210 - 250	IN6537	160 - 200	0,30 - 1,00
	alloyed steel 1100 N/mm ²	IN2505	160 - 180	IN6537	110 - 130	0,30 - 1,00
M	stainless steel	IN2505	120 - 180	IN6537	80 - 130	0,30 - 1,00
K	gray cast iron	IN2510	180 - 250	IN2510	150 - 200	0,30 - 1,50
	nodular cast iron	IN2510	140 - 210	IN2510	110 - 160	0,30 - 1,50
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2505	110 - 125	IN2505	60 - 80	0,30 - 1,00
	titanium alloys	IN2505	40 - 50	IN2505	30 - 40	0,30 - 1,00
H	hard machining < 54 HRC	IN2505	30 - 40	-	-	0,30
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



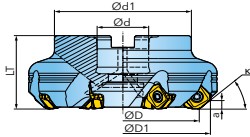
General information:

insert screw: **SM40-115-00**

torque: **4,5 Nm**

torque wrench: **DTN045S with bit DS-T15B1**

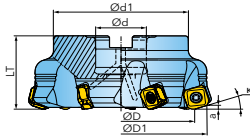
Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	SQGU1406ANR-M
Average chip thickness:	hm = 0,25 mm
max. cutting depth:	ap = 6,0 mm

Recommended cutting data face mill (DN_K) lead angle K= 45°:

ISO	Material	Cutting speed Vc [m/min]		Feed per tooth fz [mm]		
		1st choice dry machining resp. wear resistant carbide	1st choice wet machining resp. tough carbide			
P	unalloyed steel	IN2505	250 - 290	IN6537	200 - 240	0,25 - 0,40
	alloyed steel 800 N/mm ²	IN2505	210 - 250	IN6537	160 - 200	0,25 - 0,35
	alloyed steel 1100 N/mm ²	IN2505	160 - 180	IN6537	110 - 130	0,25
M	stainless steel	IN2505	120 - 180	IN6537	80 - 130	0,25 - 0,30
K	gray cast iron	IN2510	180 - 250	IN2510	150 - 200	0,25 - 0,40
	nodular cast iron	IN2510	140 - 210	IN2510	110 - 160	0,25 - 0,35
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2505	110 - 125	IN2505	60 - 80	0,25
	titanium alloys	IN2505	40 - 50	IN2505	30 - 40	0,25
H	hard machining < 54 HRC	IN2505	30 - 40	-	-	0,25
	hard machining < 63 HRC	-	-	-	-	-



Insert:	SQGU1406ANR-M
Average chip thickness:	hm = 0,40 mm
max. cutting depth:	ap = 2,5 mm

Recommended cutting data high feed cutting (DG6K) lead angle K= 20°:

ISO	Material	Cutting speed Vc [m/min]		Feed per tooth fz [mm]		
		1st choice dry machining resp. wear resistant carbide	1st choice wet machining resp. tough carbide			
P	unalloyed steel	IN2505	250 - 290	IN6537	200 - 240	0,40 - 2,30
	alloyed steel 800 N/mm ²	IN2505	210 - 250	IN6537	160 - 200	0,40 - 1,80
	alloyed steel 1100 N/mm ²	IN2505	160 - 180	IN6537	110 - 130	0,40 - 1,80
M	stainless steel	IN2505	120 - 180	IN6537	80 - 130	0,40 - 1,80
K	gray cast iron	IN2510	180 - 250	IN2510	150 - 200	0,40 - 2,30
	nodular cast iron	IN2510	140 - 210	IN2510	110 - 160	0,40 - 2,30
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2505	110 - 125	IN2505	60 - 80	0,40 - 1,80
	titanium alloys	IN2505	40 - 50	IN2505	30 - 40	0,40 - 1,80
H	hard machining < 54 HRC	IN2505	30 - 40	-	-	0,40
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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

General information:

insert screw: **TS 50C130I/HG**

torque: **6,0 Nm**

torque wrench: **DTNV00S with bit DS-T20TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	TCHW110204R-W	TCHH110204FR-PW
Average chip thickness:	hm = 0,08 mm	hm = 0,05 mm
max. cutting depth:	ap = 1,5 mm	ap = 1,5 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN0560	250 - 500	IN2504	200 - 240	0,08 - 0,15
	alloyed steel 800 N/mm ²	IN2504	210 - 250	IN2504	160 - 200	0,08 - 0,10
	alloyed steel 1100 N/mm ²	IN2504	160 - 180	IN2504	110 - 130	0,08
M	stainless steel	IN2030	120 - 180	IN2035	80 - 130	0,08 - 0,15
K	gray cast iron	IN2504	180 - 250	IN2504	150 - 200	0,08 - 0,15
	nodular cast iron	IN2504	140 - 210	IN2504	110 - 160	0,08 - 0,10
N	aluminum	IN045	800 - 1500	IN045	500 - 800	0,05 - 0,15
S	high temperature alloys	IN2035	110 - 125	IN2035	60 - 80	0,08
	titanium alloys	IN2035	40 - 50	IN2035	30 - 40	0,08
H	hard machining < 54 HRC	IN2504	70 - 100	-	-	0,08
	hard machining < 63 HRC	IN2504	50 - 80	-	-	0,08

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.

General information:

insert screw: **SM25-064-00**

torque: **1,1 Nm**

torque wrench: **DTN011S with bit DS-T08TB**

Note

A large grid of graph paper for taking notes, consisting of 20 columns and 30 rows of small squares. The grid is white with thin grey lines. To the right of the grid is a vertical blue bar.



Insert:	THLS0402_R-M
Average chip thickness:	hm = 0,04 mm
max. cutting depth:	ap = 3,5 mm

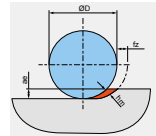
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 – 290	IN2530	200 – 240	0,04 – 0,08
	alloyed steel 800 N/mm ²	IN2505	210 – 250	IN2530	160 – 200	0,04 – 0,06
	alloyed steel 1100 N/mm ²	IN2505	160 – 180	IN2530	110 – 130	0,04
M	stainless steel	IN2505	120 – 180	IN2530	80 – 130	0,04 – 0,06
K	gray cast iron	IN2505	180 – 250	IN2530	150 – 200	0,04 – 0,08
	nodular cast iron	IN2505	140 – 210	IN2530	110 – 160	0,04 – 0,06
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2505	110 – 125	IN2530	60 – 80	0,04
	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,04
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. uneven ground [mm]	max. ap/rev. [mm]	min. Bore dia. even ground [mm]	max. ap/rev. [mm]	max. Bore dia. even ground [mm]	max. ap/rev. [mm]
10	5,5	15,2	1,5	18,2	2,4	19	2,7
12	2,5	19,8	1,0	22,1	1,3	23	1,5
16	1,7	27,7	1,0	30,1	1,3	31	1,3
20	1,3	35,7	1,1	38,1	1,2	39	1,3
25	0,7	46,0	0,8	48,1	0,8	49	0,9

General information:

insert screw: **SM18-041-00**

torque: **0,5 Nm**

torque wrench: **DTN005S with bit DS-TP06TB**



Insert:	THLS0604_R	THES0604_R	THLS0604_R-HR	THES0604_FR-P
Average chip thickness:	hm = 0,08 mm	hm = 0,08 mm	hm = 0,08 mm	hm = 0,05 mm
max. cutting depth:	ap = 7,0 mm	ap = 7,0 mm	ap = 7,0 mm	ap = 7,0 mm

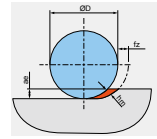
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 - 290	IN2530	200 - 240	0,08 - 0,12
	alloyed steel 800 N/mm ²	IN2505	210 - 250	IN2530	160 - 200	0,08 - 0,10
	alloyed steel 1100 N/mm ²	IN2505	160 - 180	IN2530	110 - 130	0,08
M	stainless steel	IN2035	120 - 180	IN2035	80 - 130	0,08 - 0,12
K	gray cast iron	IN2510	180 - 250	IN2530	150 - 200	0,08 - 0,12
	nodular cast iron	IN2510	140 - 210	IN2530	110 - 160	0,08 - 0,10
N	aluminum	IN10K	800 - 1500	IN10K	500 - 800	0,05 - 0,30
S	high temperature alloys	IN2035	110 - 125	IN2035	60 - 80	0,08
	titanium alloys	IN2505	40 - 50	IN2035	30 - 40	0,08
H	hard machining < 54 HRC	IN2504	30 - 40	-	-	0,08
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. uneven ground [mm]	max. ap/rev. [mm]	min. Bore dia. even ground [mm]	max. ap/rev. [mm]	max. Bore dia. even ground [mm]	max. ap/rev. [mm]
20	3,3	32,2	2,2	36,6	3,0	39	3,4
25	2,8	41,9	2,6	46,6	3,3	49	3,7
32	1,8	56,1	2,4	60,5	2,8	63	3,1
35	1,6	62,1	2,3	66,5	2,8	69	3,0
40	1,4	72,0	2,5	76,5	2,8	79	3,0
50	1,2	91,8	2,8	96,0	3,0	99	3,2
63	0,9	117,8	2,7	122,5	2,9	125	3,1

General information:

insert screw: **SM25-065-R0**

torque: **1,1 Nm**

torque wrench: **DTN011S with bit DS-T08TB**



Insert:	THLS1005_R	THES1005_R	THLS1005_R-HR	THES1005_FR-P
Average chip thickness:	hm = 0,10 mm	hm = 0,10 mm	hm = 0,08 mm	hm = 0,05 mm
max. cutting depth:	ap = 11,0 mm	ap = 11,0 mm	ap = 11,0 mm	ap = 11,0 mm

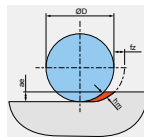
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 - 290	IN2530	200 - 240	0,10 - 0,20
	alloyed steel 800 N/mm ²	IN2505	210 - 250	IN2530	160 - 200	0,10 - 0,15
	alloyed steel 1100 N/mm ²	IN2505	160 - 180	IN2530	110 - 130	0,10
M	stainless steel	IN2035	120 - 180	IN2035	80 - 130	0,08 - 0,20
K	gray cast iron	IN2510	180 - 250	IN2530	150 - 200	0,10 - 0,20
	nodular cast iron	IN2510	140 - 210	IN2530	110 - 160	0,10 - 0,15
N	aluminum	IN10K	800 - 1500	IN10K	500 - 800	0,05 - 0,35
S	high temperature alloys	IN2035	110 - 125	IN2035	60 - 80	0,08
	titanium alloys	IN2505	40 - 50	IN2035	30 - 40	0,08
H	hard machining < 54 HRC	IN2504	30 - 40	-	-	0,10
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. uneven ground [mm]	max. ap/rev. [mm]	min. Bore dia. even ground [mm]	max. ap/rev. [mm]	max. Bore dia. even ground [mm]	max. ap/rev. [mm]
32	2,80	52,2	3,1	59,2	4,2	62	4,6
35	2,40	58,4	3,0	65,2	3,9	68	4,3
40	1,70	68,9	2,7	75,2	3,3	78	3,5
50	1,20	89,1	2,6	95,2	3,0	98	3,2
63	1,00	114,9	2,8	121,2	3,2	124	3,3
80	0,75	148,9	2,8	155,2	3,1	158	3,2
100	0,60	188,9	2,9	195,2	3,1	198	3,2
125	0,45	239,0	2,8	245,2	3,0	248	3,0

General information:

insert screw: **SM40-100-R0**

torque: **4,5 Nm**

torque wrench: **DT40-01 with bit DS-T15B1**



Insert:	THLS1306_R	THES1306_R	THLS1306_R-HR	THES1306_FR-P
Average chip thickness:	hm = 0,10 mm	hm = 0,10 mm	hm = 0,08 mm	hm = 0,05 mm
max. cutting depth:	ap = 14,9 mm	ap = 14,9 mm	ap = 14,9 mm	ap = 14,9 mm

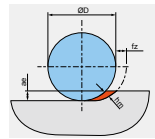
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 - 290	IN2530	200 - 240	0,10 - 0,25
	alloyed steel 800 N/mm ²	IN2505	210 - 250	IN2530	160 - 200	0,10 - 0,20
	alloyed steel 1100 N/mm ²	IN2505	160 - 180	IN2530	110 - 130	0,10
M	stainless steel	IN2035	120 - 180	IN2035	80 - 130	0,08 - 0,25
K	gray cast iron	IN2505	180 - 250	IN2530	150 - 200	0,10 - 0,25
	nodular cast iron	IN2505	140 - 210	IN2530	110 - 160	0,10 - 0,20
N	aluminum	IN10K	800 - 1500	IN10K	500 - 800	0,05 - 0,40
S	high temperature alloys	IN2035	110 - 125	IN2035	60 - 80	0,08
	titanium alloys	IN2505	40 - 50	IN2035	30 - 40	0,08
H	hard machining < 54 HRC	IN2504	30 - 40	-	-	0,10
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. uneven ground [mm]	max. ap/rev. [mm]	min. Bore dia. even ground [mm]	max. ap/rev. [mm]	max. Bore dia. even ground [mm]	max. ap/rev. [mm]
63	1,40	112,9	3,8	120,5	4,4	124	4,7
80	0,86	154,5	3,5	158,5	3,7	158	3,7
100	0,67	187,9	3,2	198,5	3,6	198	3,6
125	0,52	238,0	3,2	244,5	3,4	248	3,5
160	0,40	308,0	3,2	314,7	3,4	318	3,5

General information:

insert screw: **SM45-120-R0**

torque: **5,0 Nm**

torque wrench: **DTNV02S with bit DS-T20TB**



Insert:	TIMC			
Cutting width:	1,6	2	3	4
Average chip thickness:	hm = 0,050 mm	hm = 0,055 mm	hm = 0,065 mm	hm = 0,075 mm

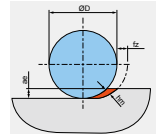
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]			
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide	
P	unalloyed steel	IN2005	90 - 230	IN2005	80 - 210
	alloyed steel 800 N/mm ²	IN2005	100 - 180	IN2005	90 - 160
	alloyed steel 1100 N/mm ²	IN2005	70 - 110	IN2005	60 - 90
M	stainless steel	IN2005	70 - 150	IN2005	60 - 130
K	gray cast iron	IN2005	110 - 140	IN2005	90 - 120
	nodular cast iron	IN2005	55 - 110	IN2005	45 - 90
N	aluminum	IN2005	250	IN2005	250
S	high temperature alloys	IN2005	20 - 40	IN2005	20 - 40
	titanium alloys	IN2005	30 - 60	IN2005	30 - 60
H	hard machining < 54 HRC	-	-	-	-
	hard machining < 63 HRC	-	-	-	-

Tips:

- Please note that these tools may only be used up to a maximum cutting speed of Vc = 250 m/min.
- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	SFC 2	SFJ 2	SFC 3	SFJ 3
Cutting width:	2	2	3	3
Average chip thickness:	hm = 0,07 - 0,18 mm	hm = 0,04 - 0,12 mm	hm = 0,08 - 0,25 mm	hm = 0,04 - 0,18 mm

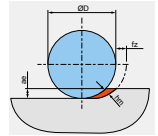
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]	
		1st choice dry machining resp. wear resistant carbide	1st choice wet machining resp. tough carbide
P	unalloyed steel	250 - 290	200 - 240
	alloyed steel 800 N/mm ²	210 - 250	160 - 200
	alloyed steel 1100 N/mm ²	160 - 180	110 - 130
M	stainless steel	120 - 180	80 - 130
K	gray cast iron	180 - 250	150 - 200
	nodular cast iron	140 - 210	110 - 160
N	aluminum	800 - 1500	500 - 800
S	high temperature alloys	110 - 125	60 - 80
	titanium alloys	40 - 50	30 - 40
H	hard machining < 54 HRC	30 - 40	-
	hard machining < 63 HRC	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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





Insert:	UNHU040212R-HF
Average chip thickness:	hm = 0,2 mm
max. cutting depth:	ap = 0,5 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 – 290	IN2530	200 – 240	0,2 – 0,4
	alloyed steel 800 N/mm ²	IN2505	210 – 250	IN2530	160 – 200	0,2 – 0,3
	alloyed steel 1100 N/mm ²	IN2505	160 – 180	IN2530	110 – 130	0,20
M	stainless steel	IN2505	120 – 180	IN2530	80 – 130	0,2 – 0,3
K	gray cast iron	IN2505	180 – 250	IN2530	150 – 200	0,2 – 0,4
	nodular cast iron	IN2505	140 – 210	IN2530	110 – 160	0,2 – 0,3
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2505	110 – 125	IN2530	60 – 80	0,20
	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,20
H	hard machining < 54 HRC	IN2504	130 – 150	-	-	0,20
	hard machining < 63 HRC	IN2504	110 – 130	-	-	0,20

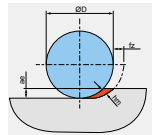
Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. [mm]	max. ap/rev. [mm]	max. Bore dia. [mm]	max. ap/rev. [mm]
10	0,8	13,9	0,2	20	0,4
11	1,6	15,9	0,4	22	0,5
12	1,6	17,9	0,4	24	0,5
13	2,0	19,9	0,5	26	0,5
16	1,9	25,9	0,5	32,0	0,5
20	2,3	33,9	0,5	40,0	0,5
25	1,7	43,9	0,5	50,0	0,5

Tips:

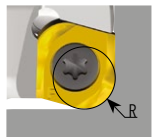
- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Programming Tip:

Please use a corner radius of 1.2 mm for machining 3D contours in the NC program. Due to the full radius of the cutting edge, no unprocessed material allowance remains.



General information:

insert screw: **SM18-041-00**

torque: **0,5 Nm**

torque wrench: **DTN055 with bit DS-TP06TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	UNHU060320R-HF
Average chip thickness:	hm = 0,2 mm
max. cutting depth:	ap = 1,0 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN6537	250 – 290	IN6537	200 – 240	0,2 – 0,8
	alloyed steel 800 N/mm ²	IN2505	210 – 250	IN2530	160 – 200	0,2 – 0,6
	alloyed steel 1100 N/mm ²	IN2505	160 – 180	IN2530	110 – 130	0,20
M	stainless steel	IN2505	120 – 180	IN2530	80 – 130	0,2 – 0,6
K	gray cast iron	IN2505	180 – 250	IN6537	150 – 200	0,2 – 0,8
	nodular cast iron	IN2505	140 – 210	IN6537	110 – 160	0,2 – 0,6
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2505	110 – 125	IN2530	60 – 80	0,20
	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,20
H	hard machining < 54 HRC	IN2504	130 – 150	-	-	0,20
	hard machining < 63 HRC	IN2504	110 – 130	-	-	0,20

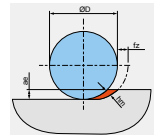
Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. [mm]	max. ap/rev. [mm]	max. Bore dia. [mm]	max. ap/rev. [mm]
16	0,6	21,4	0,2	32	0,5
20	1,7	29,4	0,7	40	1,0
25	2,3	39,4	0,9	50	1,0
32	2,4	53,4	1,0	64	1,0
40	1,8	69,4	1,0	80,0	1,0
50	1,4	89,4	1,0	100,0	1,0
63	1,1	115,4	1,0	126,0	1,0

Tips:

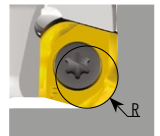
- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Programming Tip:

Please use a corner radius of 2.0 mm for machining 3D contours in the NC program. Due to the full radius of the cutting edge, no unprocessed material allowance remains.



General information:

insert screw: **SM30-068-30**

torque: **1,1 Nm**

torque wrench: **DTN020S with bit DS-T08TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	UNHU090432R-HF
Average chip thickness:	hm = 0,2 mm
max. cutting depth:	ap = 1,5 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 – 290	IN6537	200 – 240	0,2 – 1,0
	alloyed steel 800 N/mm²	IN2505	210 – 250	IN6537	160 – 200	0,2 – 0,8
	alloyed steel 1100 N/mm²	IN2505	160 – 180	IN6537	110 – 130	0,2 – 0,5
M	stainless steel	IN2505	120 – 180	IN6537	80 – 130	0,2 – 0,8
K	gray cast iron	IN2505	180 – 250	IN6537	150 – 200	0,2 – 1,0
	nodular cast iron	IN2505	140 – 210	IN6537	110 – 160	0,2 – 0,8
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2505	110 – 125	IN6537	60 – 80	0,2 – 0,5
	titanium alloys	IN2505	40 – 50	IN6537	30 – 40	0,2 – 0,5
H	hard machining < 54 HRC	IN2504	130 – 150	-	-	0,2
	hard machining < 63 HRC	IN2504	110 – 130	-	-	0,2

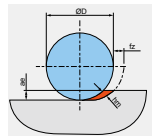
Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. [mm]	max. ap/rev. [mm]	max. Bore dia. [mm]	max. ap/rev. [mm]
20	0,3	25	0,1	40	0,3
25	1,2	35	0,6	50	1,5
32	2,4	49	1,5	64	1,5
40	2,3	65	1,5	80	1,5
50	1,7	85	1,5	100	1,5
63	1,3	111	1,5	126	1,5
80	1,0	145,0	1,5	160	1,5

Tips:

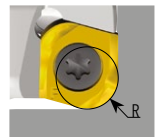
- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Programming Tip:

Please use a corner radius of 3.2 mm for machining 3D contours in the NC program. Due to the full radius of the cutting edge, no unprocessed material allowance remains.



General information:

insert screw: **SM35-088-10**

torque: **3,0 Nm**

torque wrench: **DTN20S with bit DS-T10TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	UNHU110640R-HF
Average chip thickness:	hm = 0,3 mm
max. cutting depth:	ap = 2,0 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 – 290	IN6537	200 – 240	0,3 – 1,2
	alloyed steel 800 N/mm ²	IN2505	210 – 250	IN6537	160 – 200	0,3 – 1,0
	alloyed steel 1100 N/mm ²	IN2505	160 – 180	IN6537	110 – 130	0,3 – 0,6
M	stainless steel	IN2505	120 – 180	IN6537	80 – 130	0,3 – 1,0
K	gray cast iron	IN2505	180 – 250	IN6537	150 – 200	0,3 – 1,2
	nodular cast iron	IN2505	140 – 210	IN6537	110 – 160	0,3 – 1,0
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2505	110 – 125	IN6537	60 – 80	0,3 – 0,6
	titanium alloys	IN2505	40 – 50	IN6537	30 – 40	0,3 – 0,6
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

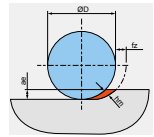
Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. [mm]	max. ap/rev. [mm]	max. Bore dia. [mm]	max. ap/rev. [mm]
25	0,3	31	0,2	50	0,9
32	1,2	45	1,2	64	2,0
40	2,4	61	2,0	80	2,0
50	2,3	81	2,0	100	2,0
63	1,7	107	2,0	126	2,0
80	1,3	141	2,0	160	2,0
100	1,0	181,0	2,0	200	2,0

Tips:

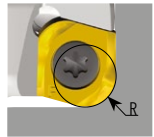
- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Programming Tip:

Please use a corner radius of 4.0 mm for machining 3D contours in the NC program. Due to the full radius of the cutting edge, no unprocessed material allowance remains.



General information:

insert screw: **SM40-100-10**

torque: **4,5 Nm**

torque wrench: **DTN045F with bit DS-T15B1**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	UNHU140750R-HF
Average chip thickness:	hm = 0,3 mm
max. cutting depth:	ap = 3,0 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 – 290	IN6537	200 – 240	0,3 – 1,5
	alloyed steel 800 N/mm ²	IN2505	210 – 250	IN6537	160 – 200	0,3 – 1,0
	alloyed steel 1100 N/mm ²	IN2505	160 – 180	IN6537	110 – 130	0,3 – 0,8
M	stainless steel	IN2505	120 – 180	IN6537	80 – 130	0,3 – 1,0
K	gray cast iron	IN2505	180 – 250	IN6537	150 – 200	0,3 – 1,5
	nodular cast iron	IN2505	140 – 210	IN6537	110 – 160	0,3 – 1,0
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2505	110 – 125	IN6537	60 – 80	0,3 – 0,8
	titanium alloys	IN2505	40 – 50	IN6537	30 – 40	0,3 – 0,8
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

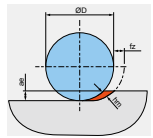
Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. [mm]	max. ap/rev. [mm]	max. Bore dia. [mm]	max. ap/rev. [mm]
50	3,9	31	3,0	100	3,0
63	2,7	45	3,0	126	3,0
80	1,9	61	3,0	160	3,0
100	1,0	81	3,0	200	3,0
125	0,5	107	3,0	250	3,0

Tips:

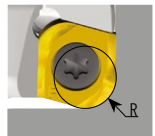
- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Programming Tip:

Please use a corner radius of 5.0 mm for machining 3D contours in the NC program. Due to the full radius of the cutting edge, no unprocessed material allowance remains.



General information:

insert screw: **SM50-127-10**

torque: **6,0 Nm**

torque wrench: **DTNV00S with bit DS-T20TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	UNKT0502TR-HF
Average chip thickness:	hm = 0,2 mm
max. cutting depth:	ap = 0,5 mm

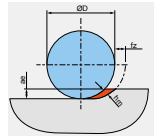
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 - 290	IN2530	200 - 240	0,2 - 0,4
	alloyed steel 800 N/mm ²	IN2505	210 - 250	IN2530	160 - 200	0,2 - 0,3
	alloyed steel 1100 N/mm ²	IN2505	160 - 180	IN2530	110 - 130	0,20
M	stainless steel	IN2505	120 - 180	IN2530	80 - 130	0,2 - 0,3
K	gray cast iron	IN2505	180 - 250	IN2530	150 - 200	0,2 - 0,4
	nodular cast iron	IN2505	140 - 210	IN2530	110 - 160	0,2 - 0,3
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2505	110 - 125	IN2530	60 - 80	0,20
	titanium alloys	IN2505	40 - 50	IN2530	30 - 40	0,20
H	hard machining < 54 HRC	IN2504	130 - 150	-	-	0,20
	hard machining < 63 HRC	IN2504	110 - 130	-	-	0,20

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. [mm]	max. ap/rev. [mm]	max. Bore dia. [mm]	max. ap/rev. [mm]
6	0,20	8	0,0	12	0,1
8	0,45	12	0,1	16	0,2
9	0,55	14	0,1	18	0,2
10	0,30	16	0,1	20	0,1
11	0,35	18	0,1	22	0,2
12	0,70	20	0,3	24	0,4
13	0,75	22	0,3	26	0,5
14	0,85	24	0,4	28	0,5
15	0,75	26	0,4	30	0,5
16	0,55	28	0,4	32	0,5
20	0,50	36	0,4	40	0,5

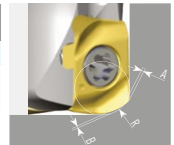
ProgrammierTips:

Please use a corner radius of 0.8 to 1.0 mm for machining 3D contours in the NC program.

Please refer to the following table for the maximum unprocessed material allowance or overcut.

	Programming radius (R)	Overcut (A)	Unprocessed (B)
UNKT0502TR-HF	0,80	0,00	0,21
	0,90	0,00	0,18
	1,00	0,02	0,14

Recommended programming radius R



General information:

insert screw size 436: **SM18-033-00**

torque: **0,5 Nm**

torque wrench: **DTN005S with bit DS-TP06TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	WNMU04T3_N	WNCU04T3_FN-P
Average chip thickness:	hm = 0,07 mm	hm = 0,05 mm
max. cutting depth:	ap = 3,8 mm	ap = 3,8 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN6537	250 – 290	IN2530	200 – 240	0,07 – 0,18
	alloyed steel 800 N/mm ²	IN2505	210 – 250	IN2530	160 – 200	0,07 – 0,13
	alloyed steel 1100 N/mm ²	IN2505	160 – 180	IN2530	110 – 130	0,07
M	stainless steel	IN2035	120 – 180	IN2035	80 – 130	0,07 – 0,18
K	gray cast iron	IN2504	180 – 250	IN6537	150 – 200	0,07 – 0,18
	nodular cast iron	IN2504	140 – 210	IN6537	110 – 160	0,07 – 0,13
N	aluminum	IN10K	800 – 1500	IN10K	500 – 800	0,05 – 0,20
S	high temperature alloys	IN2035	110 – 125	IN2530	60 – 80	0,07
	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,07
H	hard machining < 54 HRC	IN2504	30 – 40	-	-	0,07
	hard machining < 63 HRC	-	-	-	-	-

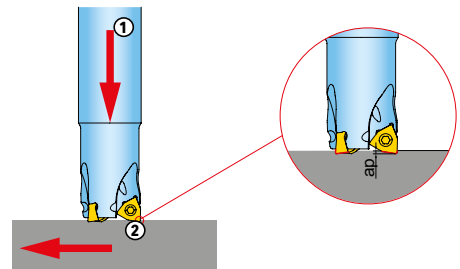
Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. uneven ground [mm]	max. ap/rev. [mm]	max. Bore dia. even ground [mm]	max. ap/rev. [mm]
16	3,0	24,1	1,3	31,2	2,5
20	2,4	31,7	1,5	39,2	2,5
25	1,9	41,4	1,7	49,2	2,5
32	1,5	55,2	1,9	63,2	2,6
35	1,7	60,5	2,4	69,2	3,2
40	2,1	69,3	3,4	79,2	3,8
50	2,5	87,2	3,8	99,2	3,8
63	2,4	111,5	3,8	125,2	3,8

Slot Milling:

Tool diameter [mm]	max. ap* [mm]
16	0,5
20	0,5
25	0,6
32	0,6
35	0,8
40	1,2*
50	1,9*
63	2,3*

*only for short-chipping materials



General information:

insert screw: **SM25-064-00**

torque: **1,1 Nm**

torque wrench: **DTN011S with bit DS-T08TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	WNMU0606_N	WNCU0606_FN-P
Average chip thickness:	hm = 0,13 mm	hm = 0,05 mm
max. cutting depth:	ap = 5,8 mm	ap = 5,8 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN6537	250 – 290	IN2530	200 – 240	0,13 – 0,25
	alloyed steel 800 N/mm ²	IN2505	210 – 250	IN2530	160 – 200	0,13 – 0,20
	alloyed steel 1100 N/mm ²	IN2505	160 – 180	IN2530	110 – 130	0,13
M	stainless steel	IN2035	120 – 180	IN2035	80 – 130	0,13 – 0,25
K	gray cast iron	IN2504	180 – 250	IN6537	150 – 200	0,13 – 0,25
	nodular cast iron	IN2504	140 – 210	IN6537	110 – 160	0,13 – 0,20
N	aluminum	IN10K	800 – 1500	IN10K	500 – 800	0,13 – 0,35
S	high temperature alloys	IN2035	110 – 125	IN2530	60 – 80	0,13
	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,13
H	hard machining < 54 HRC	IN2504	30 – 40	-	-	0,13
	hard machining < 63 HRC	-	-	-	-	-

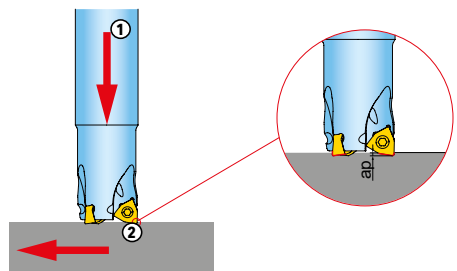
Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. uneven ground [mm]	max. ap/rev. [mm]	max. Bore dia. even ground [mm]	max. ap/rev. [mm]
25	2,9	38,2	2,1	48	3,7
32	2,2	51,4	2,3	62	3,7
40	1,8	67,0	2,7	78	3,8
50	1,7	86,0	3,4	98	4,5
63	2,6	108,0	5,8	124	5,8
80	2,9	138,5	5,8	158	5,8
100	2,2	178,5	5,8	198	5,8
125	1,3	231,1	5,8	248	5,8

Slot Milling:

Tool diameter [mm]	max. ap* [mm]
25	0,9
32	0,9
40	1,0
50	1,3
63	2,5*
80	3,4*
100	3,4*
125	2,8*

*only for short-chipping materials



General information:

insert screw: **SM35-088-60**

torque: **3,0 Nm**

torque wrench: **DTN030S with bit DS-T10TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	XEVT1605_FR - P
Average chip thickness:	hm = 0,05 mm
max. cutting depth:	ap = 15,5 mm

Recommended cutting data:

ISO	Material	Remark	Hardness [HB]	Carbide grade	Cutting speed Vc [m/min]	Feed per tooth fz [mm]	
N	wrought aluminum alloys	not hardened	60	IN10K	300 - 5000	0,05 - 0,35	
		hardened	100	IN10K	200 - 2000	0,05 - 0,25	
	<= 12% Si aluminum cast alloys>	not hardened	75	IN10K	200 - 2000	0,05 - 0,30	
		hardened	90	IN10K	200 - 1500	0,05 - 0,25	
	> 1% Pb copper alloys	high temperature application	130	IN10K	200 - 1000	0,05 - 0,15	
		well machinable	110	IN10K	200 - 800	0,05 - 0,15	
		brass	90	IN10K	300 - 1000	0,05 - 0,15	
	ninmetal	electrolytic copper	100	IN10K	300 - 800	0,05 - 0,15	
		duroplasts, fibre reinforced plastics	-	-	IN10K	100 - 500	0,05 - 0,15
		ebonite	-	-	IN10K	100 - 300	0,05 - 0,15

Ramping data and circular interpolation:

Tool diameter [mm]	max. permissible speed n [rpm]	max. cutting speed Vc [m/min]
25	52.000	4.000
32	46.000	4.600
40	41.200	5.100
50	36.800	5.700
63	32.700	6.400
80	29.000	7.200
100	26.000	8.100
125	23.200	9.100

Important remarks:

The specified maximum speeds are valid only under optimal conditions.

These include in particular:

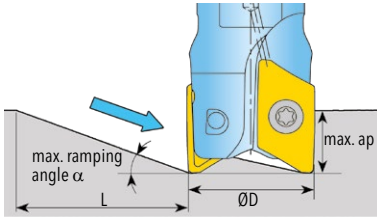
- Please use for mounting of the inserts only a torque wrench with 4.5 Nm.
- The tool has to be balanced only when completely mounted and joined with adaption.
- Please use only correct and as good as new insert screws.
- Please avoid to extend the cutters.
- Please use high speed cutters only on encapsulated machines.
- If in doubt, do not hesitate to contact your Ingersoll partner.

General information:

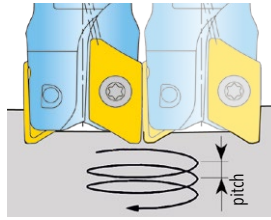
insert screw up to Ø 25: **SM40-085-20**
torque: **4,5 Nm**

insert screw from Ø 32: **SM40-093-21**
torque wrench: **DTN045F with bit DS-T15B1**

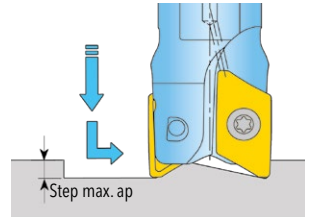
Ramping data and circular interpolation:



Straight ramping



Helical ramping



Step milling

XEVT16: R0,4 - R1,6

Tool dia. (D)	Straight ramping			Helical ramping			Step milling
	max. ramping angle (α)	max. ap (mm)	min. length (L)	min. Ø	max. Ø	max. ramping (J)	max. ap
Ø25	23,5	16	37	29,1	-	4,8	4
				-	50	13,6	4
Ø32	14,5	16	62	43,1	-	7,7	4
				-	64	13,6	4
Ø40	10,0	16	91	59,1	-	9,0	4
				-	80	13,6	4
Ø50	7,5	16	122	79,1	-	10,2	4
				-	100	13,6	4
Ø63	5,5	16	166	105,1	-	10,8	4
				-	126	13,6	4
Ø80	4,5	16	203	139,1	-	12,4	4
				-	160	13,6	4
Ø100	3,3	16	278	179,1	-	12,2	4
				-	200	13,6	4
Ø125	2,5	16	367	229,1	-	12,1	4
				-	250	13,6	4

XEVT16: R2,0

Tool dia. (D)	Straight ramping			Helical ramping			Step milling
	max. ramping angle (α)	max. ap (mm)	min. length (L)	min. Ø	max. Ø	max. ramping (J)	max. ap
Ø25	23,5	15,5	36	29,1	-	4,8	3,5
				-	50	13,2	3,5
Ø32	14,5	15,5	60	43,1	-	7,7	3,5
				-	64	13,2	3,5
Ø40	10,0	15,5	88	59,1	-	9,0	3,5
				-	80	13,2	3,5
Ø50	7,5	15,5	118	79,1	-	10,2	3,5
				-	100	13,2	3,5
Ø63	5,5	15,5	161	105,1	-	10,8	3,5
				-	126	13,2	3,5
Ø80	4,5	15,5	197	139,1	-	12,4	3,5
				-	160	13,2	3,5
Ø100	3,3	15,5	269	179,1	-	12,2	3,5
				-	200	13,2	3,5
Ø125	2,5	15,5	355	229,1	-	12,1	3,5
				-	250	13,2	3,5

XEVT16: R3,0 - R3,2

Tool dia. (D)	Straight ramping			Helical ramping			Step milling
	max. ramping angle (α)	max. ap (mm)	min. length (L)	min. Ø	max. Ø	max. ramping (J)	max. ap
Ø25	22,5	14,5	35	29,1	-	4,5	2,8
				-	50	12,3	2,8
Ø32	13,5	14,5	60	43,1	-	7,1	2,8
				-	64	12,3	2,8
Ø40	9,0	14,5	92	59,1	-	8,1	2,8
				-	80	12,3	2,8
Ø50	6,5	14,5	127	79,1	-	8,8	2,8
				-	100	12,3	2,8
Ø63	5,0	14,5	166	105,1	-	9,8	2,8
				-	126	12,3	2,8
Ø80	4,0	14,5	207	139,1	-	11,0	2,8
				-	160	12,3	2,8
Ø100	3,0	14,5	277	179,1	-	11,1	2,8
				-	200	12,3	2,8
Ø125	2,0	14,5	415	229,1	-	9,7	2,8
				-	250	11,6	2,8

XEVT16: R4,0 - R5,0

Tool dia. (D)	Straight ramping			Helical ramping			Step milling
	max. ramping angle (α)	max. ap (mm)	min. length (L)	min. Ø	max. Ø	max. ramping (J)	max. ap
Ø25	20,0	14,5	40	29,1	-	4,0	2,4
				-	50	12,3	2,4
Ø32	12,0	14,5	68	43,1	-	6,3	2,4
				-	64	12,3	2,4
Ø40	7,5	14,5	110	59,1	-	6,7	2,4
				-	80	12,3	2,4
Ø50	5,5	14,5	151	79,1	-	7,5	2,4
				-	100	12,3	2,4
Ø63	4,5	14,5	184	105,1	-	8,8	2,4
				-	126	12,3	2,4
Ø80	3,5	14,5	237	139,1	-	9,6	2,4
				-	160	12,3	2,4
Ø100	3,0	14,5	277	179,1	-	11,1	2,4
				-	200	12,3	2,4
Ø125	2,0	14,5	415	229,1	-	9,7	2,4
				-	250	11,6	2,4

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	YDA223L114	YDA223L101-P
max. turn feed:	fu = 5 mm	fu = 11 mm
max. cutting depth:	ap = 0,5 mm	ap = 0,5 mm

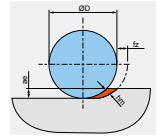
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]			
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide	
P	unalloyed steel	IN2040	150 - 200	IN2040	120 - 200
	alloyed steel 800 N/mm ²	IN2040	140 - 200	IN2040	100 - 180
	alloyed steel 1100 N/mm ²	IN2040	120 - 180	IN2040	100 - 160
M	stainless steel	IN2040	80 - 160	IN2040	80 - 140
K	gray cast iron	IN2010	150 - 200	IN2010	130 - 180
	nodular cast iron	IN2010	120 - 160	IN2010	100 - 140
N	aluminum	-	-	-	-
S	high temperature alloys	IN2040	110 - 130	IN2040	60 - 80
	titanium alloys	-	-	-	-
H	hard machining < 54 HRC	-	-	-	-
	hard machining < 63 HRC	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



General information:

insert screw: **SM40-110-00**

torque: **4,5 Nm**

torque wrench: **DT40-01 with bit DS-T15B**



Insert:	YDA323L_	YDA323L114	YDA334L102	YCE434-001
max. turn feed:	fu = 11 mm	fu = 4,5 mm	fu = 15 mm	fu = 15 mm
max. cutting depth:	ap = 0,25 mm	ap = 0,25 mm	ap = 2,0 mm	ap = 0,1 mm

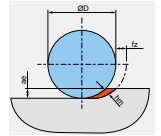
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]			
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide	
P	unalloyed steel	IN2040	150 - 250	IN2040	120 - 200
	alloyed steel 800 N/mm ²	IN2040	120 - 180	IN2040	100 - 160
	alloyed steel 1100 N/mm ²	IN0560	180 - 250	IN2040	80 - 160
M	stainless steel	IN2040	80 - 160	IN2040	80 - 160
K	gray cast iron	IN2010	120 - 180	IN2010	140 - 200
	nodular cast iron	IN2010	100 - 160	IN2010	100 - 180
N	aluminum	IN10K	400 - 600	IN10K	400 - 600
S	high temperature alloys	IN2040	60 - 120	IN2040	80 - 140
	titanium alloys	-	-	-	-
H	hard machining < 54 HRC	-	-	-	-
	hard machining < 63 HRC	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- The smaller the chosen material allowance, the higher the quality of surface and evenness of the machined surface will be.
- You can achieve optimal surface results with an ap = 0.03 - 0.07 mm
- **When using peripheral inserts in finish mills the feed per revolution depends on the max. feed rate of the peripheral insert!**

$$f_z = h_m \times \sqrt{\frac{D}{ae}}$$



General information:

insert screw for YDA323: **SM40-110-00**
 insert screw for YDA334: **SM40-110-00**
 insert screw for YCE434: **SM50-160-10**

torque: **4,5 Nm**
 torque: **4,5 Nm**
 torque: **6,0 Nm**

torque wrench: **DTN045F with bit DS-T15B**
 torque wrench: **DTN045F with bit DS-T15B**
 torque wrench: **DTN005 with bit DS-T06TB**



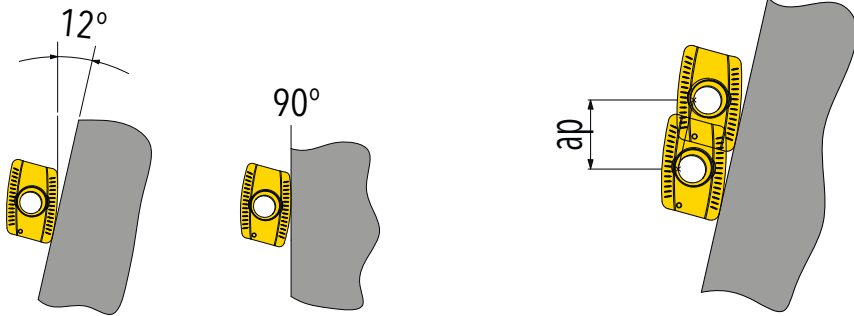
Insert:	CDHT090320R-ML	CDHT090330R-ML
Radius barrel shape:	R20	R30
recom. path distance:	ap = 1,0 - 1,2 mm	ap = 1,2 - 1,5 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				radiale Infeed ae [mm]	Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide			
P	unalloyed steel	IN2505	180 - 240	IN2530	150 - 200	0,3 - 0,5	0,1 - 0,25
	alloyed steel 800 N/mm ²	IN2505	160 - 220	IN2530	130 - 180	0,2 - 0,4	0,1 - 0,25
	alloyed steel 1100 N/mm ²	IN2505 / IN2504	140 - 200	IN2530	120 - 170	0,2 - 0,4	0,1 - 0,2
M	stainless steel	IN2530	120 - 180	IN2530	100 - 150	0,2 - 0,4	0,1 - 0,15
K	gray cast iron	IN2505 / IN2504	180 - 270	IN2530	140 - 220	0,3 - 0,5	0,1 - 0,25
	nodular cast iron	IN2505 / IN2504	160 - 220	IN2530	120 - 190	0,3 - 0,5	0,1 - 0,25
N	aluminum	-	-	-	-	-	-
S	high temperature alloys	IN2530	50 - 80	IN2530	40 - 70	0,1 - 0,3	0,07 - 0,15
	titanium alloys	-	-	IN2530	30 - 40	0,1 - 0,3	0,07 - 0,15
H	hard machining < 54 HRC	IN2504	60 - 100	-	-	0,1 - 0,2	0,1 - 0,15
	hard machining < 63 HRC	-	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- The starting feed rate should be reduced by 30%.



General information:

insert screw: **SM25-044-00**

torque: **1,1 Nm**

torque wrench: **DTN011S with bit DS-T08TB**



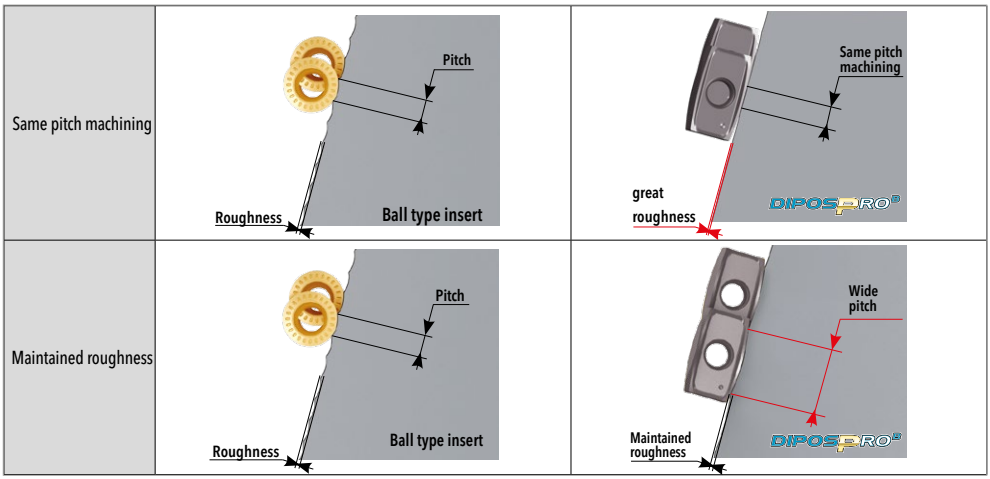
Insert:	CNHJ060220	CNHJ120430
Radius barrel shape:	R20	R30
recom. path distance:	ap = 0,5 - 0,8 mm	ap = 0,8 - 1,2 mm
Feed per tooth fz [mm]	0,07 - 0,15	0,10 - 0,25

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				radial Infeed ae [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2006	180 - 240	IN2006	150 - 200	0,3 - 0,5
	alloyed steel 800 N/mm ²	IN2006	160 - 220	IN2006	130 - 180	0,2 - 0,4
	alloyed steel 1100 N/mm ²	IN2006	140 - 200	IN2006	120 - 170	0,2 - 0,4
M	stainless steel	IN2006	120 - 180	IN2006	100 - 150	0,2 - 0,4
K	gray cast iron	IN2006	180 - 270	IN2006	140 - 220	0,3 - 0,5
	nodular cast iron	IN2006	160 - 220	IN2006	120 - 190	0,3 - 0,5
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2006	50 - 80	IN2006	40 - 70	0,1 - 0,3
	titanium alloys	-	-	IN2006	30 - 40	0,1 - 0,3
H	hard machining < 54 HRC	IN2006	60 - 100	-	-	0,1 - 0,2
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be choosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- The starting feed rate should be reduced by 30%.



General information CNHJ06_:

insert screw: **SM18-041-00**

torque: **0,5 Nm**

torque wrench: **DTN011S with bit DS-T06TB**

General information CNHJ12_:

insert screw: **SM30-068-30**

torque: **1,1 Nm**

torque wrench: **DTN011S with bit DS-T08TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	CNHU060310N	CNHU060315N	CNHU060304N-001	CNHU060310N-001	CNHU110420N
max. cutting depth = corner radius:	ap = 1,0 mm	ap = 1,5 mm	ap = 0,4 mm	ap = 1,0 mm	ap = 2,0 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	200 - 250	IN2005	300 - 500	0,1 - 0,3
	alloyed steel 800 N/mm ²	IN2505	180 - 250	IN2005	300 - 450	0,1 - 0,3
	alloyed steel 1100 N/mm ²	IN2505	180 - 220	IN2005	250 - 400	0,1 - 0,3
M	stainless steel	IN2505	100 - 180	IN2005	200 - 250	0,1 - 0,25
K	gray cast iron	IN2505	220 - 280	IN2005	300 - 600	0,15 - 0,3
	nodular cast iron	IN2505	180 - 250	IN2005	250 - 450	0,1 - 0,3
N	aluminum	IN055	500 - 1000	IN055	800 - 1200	0,1 - 0,15
S	high temperature alloys	IN2005	40 - 100	IN2005	50 - 150	0,1 - 0,2
	titanium alloys	IN2005	40 - 50	IN2005	50 - 80	0,1 - 0,2
H	hard machining < 54 HRC	IN2006	100 - 150	IN2006	120 - 180	0,08 - 0,15
	hard machining < 63 HRC	IN2006	80 - 130	IN2006	100 - 150	0,07 - 0,12

Tips:

- For extreme cavities plunge milling is recommended, for finishing $a_e = 0.008 - 0.01 \times D$.
- With CBN-inserts cast materials can be machined with cutting speeds $v_c = 1000$ to 1500 m/min and feed rate per tooth $f_z = 0.05$ to 0.1 mm.
- The lower f_z -value of the table refers to CNHU06..., the higher value to CNHU11...
- Recommended cutting depth for finishing:

CNHU06	ap = 0,1 - 0,2 mm
CNHU11	ap = 0,15 - 0,3 mm
- 4-edged carbide insert
- 2-edged CBN insert

General information CNHU06_:

insert screw: **SM25-075-20**

torque: **1,1 Nm**

torque wrench: **DTN011S with bit DS-T08TB**

General information CNHU11_:

insert screw: **SM35-088-10**

torque: **3,0 Nm**

torque wrench: **DTN030S with bit DS-T10TB**

Note

A large grid of graph paper for taking notes, consisting of 20 columns and 30 rows of small squares. The grid is empty and occupies most of the page.



Insert:	NBEU1603MOR-ML	NBEU2004MOR-ML	NBEU2505MOR-ML	NBEU1603MOR	NBEU2004MOR
Feed per tooth fz:	0,05 - 0,30 mm	0,05 - 0,40 mm	0,05 - 0,50 mm	0,05 - 0,30 mm	0,05 - 0,40 mm
Axial cutting edge length:	L = 11,8 mm	L = 13,6 mm	L = 17,7 mm	L = 11,8 mm	L = 13,6 mm
Radius:	8 mm	10 mm	12,5	8 mm	10 mm



Insert:	NBEU2505MOR	NBEU3207MOR
Feed per tooth fz [mm]:	0,05 - 0,50 mm	0,05 - 0,50 mm
Axial cutting edge length:	L = 17,7 mm	L = 21,4 mm
Radius [mm]:	12,5	16,0

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	200 - 350	IN2530	160 - 280	0,08 - 0,5
	alloyed steel 800 N/mm ²	IN2505	140 - 200	IN2530	110 - 160	0,08 - 0,5
	alloyed steel 1100 N/mm ²	IN2505	120 - 180	IN2530	100 - 150	0,08 - 0,5
M	stainless steel	IN2505	90 - 150	IN2530 / IN2035	80 - 130	0,05 - 0,35
K	gray cast iron	IN2504 / IN2505	240 - 380	IN2530	220 - 300	0,1 - 0,45
	nodular cast iron	IN2504 / IN2505	180 - 280	IN2530	160 - 240	0,1 - 0,45
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2035	40 - 80	IN2530 / IN2035	50 - 70	0,05 - 0,35
	titanium alloys	IN2505	30 - 60	IN2035	30 - 50	0,05 - 0,35
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- Starting feed rate should be reduced by 30%.
- 2-edged insert.

General information:

insert screw for Ø16: **SM25-064-00**
 insert screw for Ø20: **SM30-085-10**
 insert screw for Ø25: **TS 35085I/HG**
 insert screw for Ø32: **TS 40A115I**
 insert screw for Ø16: **TS 25055I/HG**
 insert screw for Ø20: **TS 25055I/HG**
 insert screw for Ø25: **TS 25055I/HG**
 insert screw for Ø32: **TS 35085I/HG**

torque: **1,1 Nm**
 torque: **2,0 Nm**
 torque: **3,0 Nm**
 torque: **4,5 Nm**
 torque: **1,1 Nm**
 torque: **1,1 Nm**
 torque: **1,1 Nm**
 torque: **3,0 Nm**

torque wrench: **DS-T085**
 torque wrench: **DS-0022**
 torque wrench: **TD15**
 torque wrench: **TD15**
 torque wrench: **DS-T085**
 torque wrench: **DS-T085**
 torque wrench: **DS-T085**
 torque wrench: **TD15**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	NQHG_FM	NQHG_SM	GQHG_	NQHB_	NQHL_
Application:	strong and precise geometry for high-strength materials	positive helical geometry for soft and ductile materials	torical geometry	conical cutting edge for inclined profiles with undercut	lower and side lenticular cutting edge
Recom. depth of cut	ap = 0,02 - 0,04 mm x Ø	ap = 0,02 - 0,04 mm x Ø	ap = 0,1 - 0,2 mm x radius	ap = 1,0 mm	ap = 0,1 mm
max. cutting depth:	ap = 1/2 mm x Ø	ap = 1/2 mm x Ø	ap = R + 1 mm	ap = 1,5 mm	ap = 0,5 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]		Feed per tooth fz [mm]	Cutting speed Vc [m/min]		Feed per tooth fz [mm]
		Semi finishing			Finishing		
P	unalloyed steel	IN2005	150 - 250	0,15 - 0,25	IN2005	180 - 350	0,10 - 0,20
	alloyed steel 800 N/mm ²	IN2005	130 - 220	0,15 - 0,25	IN2005	150 - 280	0,10 - 0,20
	alloyed steel 1100 N/mm ²	IN2005	120 - 200	0,15 - 0,25	IN2005	140 - 250	0,08 - 0,18
M	stainless steel	IN2005	100 - 180	0,12 - 0,20	IN2005	120 - 200	0,08 - 0,15
K	gray cast iron	IN2005	150 - 250	0,15 - 0,25	IN2005	180 - 350	0,10 - 0,20
	nodular cast iron	IN2005	130 - 220	0,15 - 0,25	IN2005	150 - 250	0,10 - 0,20
N	aluminum	-	-	-	-	-	-
S	high temperature alloys	IN2005	60 - 90	0,10 - 0,20	IN2005	60 - 120	0,05 - 0,12
	titanium alloys	IN2005	40 - 60	0,10 - 0,15	IN2005	40 - 80	0,05 - 0,10
H	hard machining < 54 HRC	IN2006	100 - 180	0,12 - 0,18	IN2006	120 - 200	0,05 - 0,13
	hard machining < 63 HRC	IN2006	80 - 140	0,10 - 0,15	IN2006	100 - 170	0,05 - 0,12

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen, but max. 30 % of tool diameter.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- The longer the extension length, the lower the cutting speed.
- For finishing the contact width should be approx. 0.02 x Ø.

General information:

insert screw for Ø 8: **SM25-080-B1**
 insert screw for Ø 10: **SM30-083-B1**
 insert screw for Ø 12: **SM40-106-B1**
 insert screw for Ø 16: **SM50-139-B1**
 insert screw for Ø 20: **SM60-167-B1**
 insert screw for Ø 25: **SM70-210-B1**
 insert screw for Ø 30/32: **SM80-250-B1**

torque: **1,1 Nm**
 torque: **2,0 Nm**
 torque: **4,0 Nm**
 torque: **5,0 Nm**
 torque: **6,0 Nm**
 torque: **6,0 Nm**
 torque: **8,0 Nm**

torque wrench: **DTN011S with bit DS-TP08TB**
 torque wrench: **DTN020S with bit DS-TP10TB**
 torque wrench: **DTNV00S with bit DS-TP15TB**
 torque wrench: **DTNV00S with bit DS-T20TB**
 torque wrench: **DTNV00S with bit DS-T25TB**
 torque wrench: **DTNV00S with bit DS-T25TB**
 torque wrench: **DTNV00S with bit DS-T30T**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	SHLT06_/SHGT06_	SPLT07_/SDGT07_	SHLT09_/SHGT09_	CDE313L_	DPM324L_
Feed per tooth:	fz = 0,10 - 0,20 mm	fz = 0,12 - 0,22 mm	fz = 0,12 - 0,25 mm	fz = 0,08 - 0,18 mm	fz = 0,12 - 0,20 mm
Recom. depth of cut:	step 3	step 4	step 6	step 5	step 8
max. cutting depth:	step 5	step 7	step 8	step 7	step 10

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]					
		SHLT / SHGT / SPLT / SDGT		CDEL13L...		DPM324L...	
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		1st choice wet machining resp. tough carbide	
P	unalloyed steel	IN2005	150 - 200	IN4015	150 - 200	IN4040	140 - 200
	alloyed steel 800 N/mm ²	IN2005	130 - 180	IN4015	130 - 180	IN4040	120 - 180
	alloyed steel 1100 N/mm ²	IN2005	110 - 170	IN4015	110 - 170	IN4040	100 - 170
M	stainless steel	IN2005	90 - 150	IN2005	90 - 150	IN2035 / IN4030	80 - 150
K	gray cast iron	IN2010	140 - 200	IN4015	140 - 200	IN4030	120 - 180
	nodular cast iron	IN2010	120 - 180	IN4015	120 - 180	IN4030	100 - 160
N	aluminum	IN10K	300 - 800	IN055	300 - 800	IN055	300 - 800
S	high temperature alloys	IN2005	40 - 70	IN2005	40 - 70	IN2035 / IN4030	40 - 70
	titanium alloys	IN2530	30 - 40	IN2005	30 - 40	IN2035 / IN4030	30 - 40
H	hard machining < 54 HRC	-	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- The longer the extension length, the lower the cutting speed should be.
- At plunging and base contact the feed rate should be reduced by 30 % for a way of 3 mm.
- Please retract from contour, before retracting on safety plane (approx. 0.2 - 0.5 mm).

General information SHLT06 / SHGT06:

insert screw: **SM22-052-00**

torque: **0,8 Nm**

torque wrench: **DTNV01S with bit DS-T07TB**

General information SPLT07 / SDGT07:

insert screw: **SM25-064-00**

torque: **1,1 Nm**

torque wrench: **DTN011S with bit DS-T08TB**

General information SHLT09 / SHGT09:

insert screw: **SM35-088-60**

torque: **3 Nm**

torque wrench: **DTN030S with bit DS-T10TB**

General information CDE313L:

insert screw: **SM40-090-00**

torque: **4,5 Nm**

torque wrench: **DTN045F with bit DS-T15B1**

General information DPM324L:

insert screw: **SM40-120-20**

torque: **4,5 Nm**

torque wrench: **DTN045F with bit DS-T15B1**



Insert:	PEMT0502ZCTR-HR
max. cutting depth:	ap = 1,0 mm

Recommended cutting data:

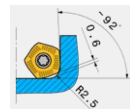
ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 – 290	IN2530	200 – 240	0,40 – 1,50
	alloyed steel 800 N/mm ²	IN2505	210 – 250	IN2530	160 – 200	0,40 – 1,00
	alloyed steel 1100 N/mm ²	IN2505	160 – 180	IN2530	110 – 130	0,40
M	stainless steel	IN2035	120 – 180	IN2035	80 – 130	0,40 – 1,50
K	gray cast iron	IN2505	180 – 250	IN2530	150 – 200	0,40 – 1,50
	nodular cast iron	IN2505	140 – 210	IN2530	110 – 160	0,40 – 1,00
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2035	110 – 125	IN2035	60 – 80	0,40
	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,40
H	hard machining < 54 HRC	IN2505	30 – 40	-	-	0,40
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- Please only use a torque wrench (1.1 Nm) to install the insert screws.
- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.

Programming Tip:

Please use a corner radius of 2.5 mm in your NC-program when machining 3D-contours. The maximum allowance will then be up to 0.6 mm.



Ramping data and circular interpolation:

Tool diameter [mm]	max. ramping angle [°]	max. feed per rev. [mm]	min. bore dia. uneven ground [mm]	min. bore dia. even ground [mm]	max. bore dia. uneven ground [mm]	max. bore dia. even ground [mm]
20 R2,5	9,6°	1	27,3	30	30,5	39
25 R2,5	5,2°	1	37,3	40	40,5	49
32 R2,5	3,4°	1	51,3	54	54,5	63
35 R2,5	3,0°	1	57,3	60	60,5	69
40 R2,5	2,5°	1	67,3	70	70,5	79
42 R2,5	2,3°	1	71,3	74	74,5	83
50 R2,5	1,8°	1	87,3	90	90,5	99
52 R2,5	1,7°	1	91,3	94	94,5	103
63 R2,5	1,4°	1	113,3	116	116,5	125
66 R2,5	1,4°	1	119,3	122	122,5	131

General information:

insert screw: **SM25-064-00**

torque: **1,1 Nm**

torque wrench: **DTN011S with bit DS-T08TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	RCLT_CP	RCLT_CC	RCLT_CC1	RCLT_CC2	RCLT_PH
Factor for Feed per tooth fz:	1,0	0,8	1,0	1,2	2,0
Recom. Depth of cut Ø12:	ap = 4,0 mm	-	ap = 3,0 mm	ap = 3,0 mm	ap = 1,5 mm
Recom. Depth of cut Ø16:	ap = 6,0 mm	ap = 4,0 mm	ap = 4,0 mm	-	ap = 2,0 mm



Insert:	RCLT_PH2
Factor for Feed per tooth fz:	4,0
Recom. Depth of cut Ø12:	ap = 2,0 mm
Recom. Depth of cut Ø16:	ap = 2,5 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN4040	160 - 220	IN4030	130 - 180	0,15 - 0,25
	alloyed steel 800 N/mm ²	IN4040	140 - 200	IN4030	110 - 160	0,12 - 0,22
	alloyed steel 1100 N/mm ²	IN2005	120 - 180	IN4030	100 - 150	0,1 - 0,2
M	stainless steel	IN2035 / IN4030	90 - 150	IN2035 / IN4030	80 - 130	0,1 - 0,2
K	gray cast iron	IN4015	160 - 250	IN4015	140 - 200	0,15 - 0,25
	nodular cast iron	IN4015	140 - 200	IN4015	120 - 170	0,12 - 0,22
N	aluminum	IN055	500 - 1200	IN055	500 - 1200	0,15 - 0,25
S	high temperature alloys	IN2035 / IN4030	50 - 80	IN2035 / IN4030	50 - 70	0,1 - 0,18
	titanium alloys	IN2005	40 - 50	IN2035 / IN4030	30 - 40	0,1 - 0,15
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- 4-edged insert RCLT...CP / CC / CC1 / C22
- 8-edged insert RCLT...PH / PH2

Ramping data and circular interpolation:

Tool diameter [mm]	max. ramping angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	bore dia. even ground [mm]	max. ap/rev. [mm]	max. bore dia. [mm]	max. ap/rev. [mm]
24 R6	1,4	32,9	0,6	36,2	0,9	48	1,8
32 R6	2,6	46,3	2,0	52,1	2,8	64	4,5
32 R8	1,5	43,6	0,9	48,2	1,3	64	2,6
35 R6	2,9	51,6	2,6	58,1	3,6	70	5,5
40 R6	5,1	59,0	5,3	68,1	6,0	80	6,0
40 R8	2,8	56,5	2,5	64,1	3,7	80	6,1
42 R6	4,7	63,0	5,4	72,1	6,0	84	6,0
42 R8	3,6	59,3	3,4	68,1	5,1	84	8,0
50 R6	3,6	78,9	5,7	88,0	6,0	100	6,0
50 R8	8,0	70,5	8,0	84,0	8,0	100	8,0
52 R6	4,0	82,4	6,0	92,0	6,0	104	6,0
52 R8	7,5	74,5	8,0	88,0	8,0	104	8,0
63 R6	2,6	104,9	5,9	114,0	6,0	126	6,0
63 R8	5,4	96,4	8,0	110,0	8,0	126	8,0
66 R6	2,4	110,9	5,9	120,0	6,0	132	6,0
66 R8	5,0	102,4	8,0	116,0	8,0	132	8,0
80 R6	1,9	138,8	6,0	147,9	6,0	160	6,0
80 R8	3,8	130,4	8,0	144,0	8,0	160	8,0
100 R8	2,8	170,4	8,0	184,0	8,0	200	8,0
125 R8	2,3	220,0	8,0	234,0	8,0	250	8,0
160 R8	1,8	289,7	8,0	304,0	8,0	320	8,0

General information RCLT12_:

insert screw: **SM40-090-00**

torque: **4,5 Nm**

torque wrench: **DTN045F with bit DS-T15B**

General information RCLT16_:

insert screw: **SM50-105-10**

torque: **6 Nm**

torque wrench: **DTNV00S with bit DS-T20TB**



Insert:	RH_06_	RH_08_	RH_10_	RH_12_	RH_16_
Feed per tooth:	fz = 0,1 - 0,3 mm	fz = 0,2 - 0,5 mm	fz = 0,3 - 0,7 mm	fz = 0,4 - 0,8 mm	fz = 0,5 - 1,0
Recom. Depth of cut:	ap = 0,1 - 0,4 mm	ap = 0,3 - 0,6 mm	ap = 0,5 - 1,0 mm	ap = 0,5 - 1,5 mm	ap = 1,5 - 3,0 mm



Insert:	RH_20_
Feed per tooth:	fz = 0,6 - 1,5 mm
Recom. Depth of cut:	ap = 2,0 - 5,0 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Insert type
		1st choice for inserts Ø 6 / 8 10 and 12		1st choice for inserts Ø 16 and 20		
P	unalloyed steel	IN2005 / IN2505	180 - 250	IN4040	170 - 220	RHHW / RHHT / RHKW / RHKT
	alloyed steel 800 N/mm ²	IN2005 / IN2505	170 - 220	IN4040	150 - 200	RHHW / RHKW / RHKT
	alloyed steel 1100 N/mm ²	IN2005 / IN2505	150 - 200	IN4040	100 - 150	RHHW / RHKW / RHKT
M	stainless steel	IN2035 / IN4035 / IN7035	90 - 150	IN2035 / IN4035 / IN7035	80 - 130	RHHT / RHKT
K	gray cast iron	IN2005 / IN2505	160 - 250	IN4015	140 - 200	RHHW / RHKT / RHKW
	nodular cast iron	IN2005 / IN2505	140 - 200	IN4015	120 - 170	RHHW / RHKT / RHKW
N	aluminum	IN055	500 - 1200	IN055	500 - 1000	RHHT...P
S	high temperature alloys	IN2035 / IN4035 / IN7035	50 - 80	IN2035 / IN4035 / IN7035	50 - 70	RHHT
	titanium alloys	IN2005 / IN2505	40 - 50	IN2035 / IN4035 / IN7035	30 - 40	RHHT
H	hard machining < 54 HRC	IN2004	80 - 120	IN2004	60 - 100	RHHW
	hard machining < 63 HRC	IN2006	50 - 80	-	-	RHHW

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- The longer the extension length, the lower the cutting speed.
- FormMaster (neutral mounting position) suitable for hard machining > 35HRC.
- FormMasterPlus (positive mounting position) suitable for rough milling, instable machining conditions and weak machines.

Ramping data and circular interpolation:

Tool diameter [mm]	max. ramping angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	bore dia. even ground [mm]	max. ap/rev. [mm]	max. bore dia. [mm]	max. ap/rev. [mm]
12 R3	10,6	14,2	1,2	18,0	3,0	24	3,0
16 R3	8,0	21,5	2,4	26,0	3,0	32	3,0
16 R4	2,0	21,7	0,6	24,3	0,9	32	1,7
20 R3	8,2	28,7	3,0	34,0	3,0	40	3,0
20 R5	3,6	26,0	1,1	30,2	2,0	40	3,9
24 R6	3,6	31,1	1,4	36,2	2,4	48	4,7
25 R3	5,6	38,7	3,0	44,0	3,0	50	3,0
25 R5	2,2	35,9	1,3	40,2	1,8	50	3,0
30 R4	7,6	44,7	4,0	52,0	4,0	60	4,0
30 R5	10,2	41,3	5,0	50,2	5,0	60	5,0
32 R6	2,3	46,7	1,8	52,3	2,5	64	4,0
32 R8	1,8	43,2	1,1	48,3	1,6	64	3,1
35 R5	7,7	51,3	5,0	60,2	5,0	70	5,0
35 R6	2,5	52,1	2,3	58,2	3,1	70	4,8
40 R6	2,8	61,1	3,2	68,3	4,3	80	6,0
42 R5	5,7	65,3	5,0	74,2	5,0	84	5,0
42 R6	2,9	64,7	3,6	72,2	4,8	84	6,0
42 R8	2,3	61,0	2,3	68,3	3,3	84	5,2
52 R5	4,1	85,3	5,0	94,2	5,0	104	5,0
52 R6	4,2	81,4	6,0	82,3	6,0	104	6,0
52 R8	2,8	79,0	4,1	88,3	5,5	104	7,9
66 R5	3,3	112,9	5,0	122,0	5,0	132	5,0
66 R6	3,0	110,3	6,0	120,2	6,0	132	6,0
66 R8	3,8	104,0	7,9	116,3	8,0	132	8,0
66 R10	2,8	100,6	5,3	112,5	7,1	132	10,0
80 R6	2,3	138,3	6,0	148,2	6,0	160	6,0
80 R8	2,9	131,9	8,0	144,3	8,0	160	8,0
80 R10	5,4	123,4	10,0	140,5	10,0	160	10,0
100 R8	2,1	171,9	8,0	184,3	8,0	200	8,0
100 R10	3,8	163,4	10,0	180,5	10,0	200	10,0
125 R8	1,6	221,9	8,0	234,3	8,0	250	8,0
125 R10	2,8	106,7	2,8	230,5	10,0	250	10,0
160 R8	1,2	291,8	8,0	304,3	8,0	320	8,0
160 R10	2,1	283,3	10,0	300,5	10,0	320	10,0

General information:

insert screw for RH_06_: **SM25-049-00**

insert screw for RH_08_: **SM30-053-00**

insert screw for RH_10_: **SM40-080-10**

insert screw for RH_12_: **SM40-080-10**

insert screw for RH_16_: **SM50-100-10**

insert screw for RH_20_: **SM50-100-10**

torque: **1,1 Nm**

torque: **2,0 Nm**

torque: **4,5 Nm**

torque: **4,5 Nm**

torque: **6,0 Nm**

torque: **6,0 Nm**

torque wrench: **DTN011S with bit DS-T08TB**

torque wrench: **DTN020S with bit DS-T09TB**

torque wrench: **DTN045F with bit DS-T15B**

torque wrench: **DTN045F with bit DS-T15B**

torque wrench: **DTNV00S with bit DS-T20TB**

torque wrench: **DTNV00S with bit DS-T20TB**



Insert:	RNLU1004MON-M	RNLU1004MON-S	RNLU1205MON-M	RNLU1205MON-S	RNLU1606MON-M
Usable cutting edges:	8	8	16	8	16
recom. Depth of cut:	ap = 1,5 - 3,0 mm	ap = 3,0 mm	ap = 1,7 mm	ap = 4,0 mm	ap = 2,3 mm
max. cutting depth:	ap = 5,0 mm	ap = 5,0 mm	ap = 6,0 mm	ap = 6,0 mm	ap = 8,0 mm



Insert:	RNLU1606MON-S
Usable cutting edges:	8
recom. Depth of cut:	ap = 6,0 mm
max. cutting depth:	ap = 8,0 mm

Recommended cutting data RNLU10_:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]				
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		ap [1mm]	ap [2mm]	ap [3mm]	ap [4mm]	ap [5mm]
P	unalloyed steel	IN2505	160 - 220	IN2530	130 - 180	0,1 - 0,5	0,1 - 0,4	0,1 - 0,3	0,1 - 0,25	0,1 - 0,2
	alloyed steel 800 N/mm ²	IN2505	140 - 200	IN2530	110 - 160	0,1 - 0,5	0,1 - 0,4	0,1 - 0,3	0,1 - 0,25	0,1 - 0,2
	alloyed steel 1100 N/mm ²	IN2505	120 - 180	IN2530	100 - 150	0,1 - 0,5	0,1 - 0,4	0,1 - 0,3	0,1 - 0,25	0,1 - 0,2
M	stainless steel	IN2530 / IN2035	90 - 150	IN2035	80 - 130	0,1 - 0,4	0,1 - 0,3	0,1 - 0,25	0,1 - 0,2	0,1 - 0,2
K	gray cast iron	IN2505	160 - 250	IN2530	140 - 200	0,1 - 0,5	0,1 - 0,4	0,1 - 0,3	0,1 - 0,25	0,1 - 0,2
	nodular cast iron	IN2505 / IN2530	140 - 200	IN2530	120 - 170	0,1 - 0,5	0,1 - 0,4	0,1 - 0,3	0,1 - 0,25	0,1 - 0,2
S	high temperature alloys	IN2530 / IN2035	50 - 80	IN2530 / IN2035	50 - 70	0,1 - 0,4	0,1 - 0,3	0,1 - 0,25	0,1 - 0,2	0,1 - 0,2
	titanium alloys	IN2530 / IN2035	-	IN2530 / IN2035	30 - 40	0,1 - 0,4	0,1 - 0,3	0,1 - 0,25	0,1 - 0,2	0,1 - 0,2

Recommended cutting data RNLU12_:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]				
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		ap [2mm]	ap [3mm]	ap [4mm]	ap [5mm]	ap [6mm]
P	unalloyed steel	IN2505	160 - 220	IN6537	130 - 180	0,13 - 0,6	0,13 - 0,55	0,13 - 0,5	0,13 - 0,4	0,13 - 0,25
	alloyed steel 800 N/mm ²	IN2505	140 - 200	IN6537	110 - 160	0,13 - 0,6	0,13 - 0,55	0,13 - 0,5	0,13 - 0,4	0,13 - 0,25
	alloyed steel 1100 N/mm ²	IN2505	120 - 180	IN6537	100 - 150	0,13 - 0,6	0,13 - 0,55	0,13 - 0,5	0,13 - 0,4	0,13 - 0,25
M	stainless steel	IN6537	90 - 150	IN7036	80 - 130	0,13 - 0,5	0,13 - 0,40	0,13 - 0,4	0,13 - 0,4	0,13 - 0,2
K	gray cast iron	IN2505	160 - 250	IN6537	140 - 200	0,13 - 0,6	0,13 - 0,55	0,13 - 0,5	0,13 - 0,4	0,13 - 0,25
	nodular cast iron	IN2505 / IN6537	140 - 200	IN6537	120 - 170	0,13 - 0,6	0,13 - 0,55	0,13 - 0,5	0,13 - 0,4	0,13 - 0,25
S	high temperature alloys	IN6537 / IN7036	50 - 80	IN7036	50 - 70	0,13 - 0,5	0,13 - 0,4	0,13 - 0,4	0,13 - 0,3	0,13 - 0,2
	titanium alloys	IN6537 / IN7036	-	IN7036	30 - 40	0,13 - 0,5	0,13 - 0,4	0,13 - 0,4	0,13 - 0,3	0,13 - 0,2

Recommended cutting data RNLU16_:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]			
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		ap [2mm]	ap [4mm]	ap [6mm]	ap [8mm]
P	unalloyed steel	IN2505	160 - 220	IN6537	130 - 180	0,15 - 0,8	0,15 - 0,6	0,15 - 0,4	0,15 - 0,3
	alloyed steel 800 N/mm ²	IN2505	140 - 200	IN6537	110 - 160	0,15 - 0,8	0,15 - 0,6	0,15 - 0,4	0,15 - 0,3
	alloyed steel 1100 N/mm ²	IN2505	120 - 180	IN6537	100 - 150	0,15 - 0,8	0,15 - 0,6	0,15 - 0,4	0,15 - 0,3
M	stainless steel	IN6537 / IN7036	90 - 150	IN7036	80 - 130	0,15 - 0,8	0,15 - 0,4	0,1 - 0,3	0,1 - 0,2
K	gray cast iron	IN2505	160 - 250	IN6537	140 - 200	0,15 - 0,8	0,15 - 0,6	0,15 - 0,4	0,15 - 0,3
	nodular cast iron	IN2505 / IN6537	140 - 200	IN6537	120 - 170	0,15 - 0,8	0,15 - 0,6	0,15 - 0,4	0,15 - 0,3
S	high temperature alloys	IN6537 / IN7036	50 - 80	IN7036	50 - 70	0,15 - 0,8	0,15 - 0,4	0,1 - 0,3	0,1 - 0,2
	titanium alloys	IN6537 / IN7036	-	IN7036	30 - 40	0,15 - 0,8	0,15 - 0,4	0,1 - 0,3	0,1 - 0,2

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- Starting feed rate should be reduced by 30%

Ramping data and circular interpolation:

Tool diameter [mm]	RNLU 10/12/16			
	max. Ramping angle [°]	min. Bore dia. [mm]	max. ap/rev. [mm]	Bore dia. even ground [mm]
25/R5	1,1	33,0	0,4	50,0
32/R5	0,9	35,0	0,6	64,0
32/R6	1,4	42,0	0,7	64,0
40/R5	0,9	63,0	1,0	80,0
40/R6	1,3	58,0	1,1	80,0
42/R5	0,9	67,0	1,0	84,0
50/R5	0,7	83,0	1,1	100,0
50/R6	1,0	78,0	1,3	100,0
50/R8	1,3	72,0	1,3	100,0
52/R5	0,8	87,0	1,3	104,0
52/R6	1,0	82,0	1,4	104,0
52/R8	1,0	76,0	1,1	104,0
63/R6	1,0	104,0	1,9	126,0
63/R8	1,0	104,0	1,6	126,0
66/R6	1,0	110,0	2,0	132,0
66/R8	1,0	104,0	1,8	132,0
80/R6	0,9	138,0	2,4	160,0
80/R8	1,0	132,0	2,4	160,0
100/R6	0,7	178,0	2,5	200,0
100/R8	0,9	172,0	3,0	200,0
125/R8	0,9	222,0	4,1	250,0
160/R8	0,9	292,0	4,1	320,0

General information:

insert screw for RNLU10_: **TS35085I/HG**

insert screw for RNLU12_: **SM40-110-00**

insert screw for RNLU16_: **TS50A121I/HG**

torque: **3,0 Nm**

torque: **4,5 Nm**

torque: **6,0 Nm**

torque wrench: **DTNV030S with bit DS-T15TB**

torque wrench: **DT - 40 - 01 with bit DS-T15TB**

torque wrench: **DTNV00S with bit DS-T20T3**



Insert:	RPLX10T3MON-HR	RPLX10T3MOTN-HR	RPLX10T3MOTN-FL	RPLX1204MON-HR1	RPLX1204MOTN-HR
Geometry:	positive geometry	pos. geometry, neg. chamfered	pos. geometry, chamfered	positive geometry	pos. geometry, neg. chamfered
recom. Depth of cut:	ap = 0,8 - 1,5 mm	ap = 0,8 - 1,5 mm	ap = 0,8 - 1,5 mm	ap = 1,5 - 2,3 mm	ap = 1,5 - 2,3 mm
max. cutting depth:	ap = 2,5 mm	ap = 2,5 mm	ap = 2,5 mm	ap = 3,0 mm	ap = 3,0 mm



Insert:	RPLX1204MOTN-FL
Geometry:	pos. geometry, chamfered
recom. Depth of cut:	ap = 1,5 - 2,3 mm
max. cutting depth:	ap = 3,0 mm

Recommended cutting data RNLU10_:

ISO	Material	Grade	Cutting speed Vc [m/min]		Feed per tooth fz [mm]		Depth of cut ap [mm]	
			dry machining	wet machining	RPLX10	RPLX12	RPLX10	RPLX12
P	unalloyed steel	IN2505	150 - 350	120 - 250	0,15 - 0,40	0,25 - 0,50	0,80 - 2,50	1,00 - 3,00
	alloyed steel 800 N/mm ²	IN2505	130 - 280	110 - 220	0,15 - 0,35	0,25 - 0,40	0,80 - 2,50	1,00 - 3,00
M	1.4021 X20Cr13	IN7036	150 - 280	120 - 250	0,15 - 0,40	0,25 - 0,50	0,80 - 2,50	1,00 - 3,00
	1.4418 X4CrNiMo16-5-1	IN7036	130 - 200	110 - 180	0,15 - 0,40	0,25 - 0,50	0,80 - 2,50	1,00 - 3,00
	1.4301 X5CrNi18-10	IN7036	120 - 180	100 - 160	0,15 - 0,40	0,25 - 0,50	0,80 - 2,50	1,00 - 3,00
	1.4534 X3CrNiMoAl13-8-2	IN7036	80 - 150	60 - 120	0,15 - 0,40	0,25 - 0,50	0,80 - 2,50	1,00 - 3,00
	1.4541 X6CrNiTi18-10	IN7036	80 - 150	60 - 120	0,15 - 0,40	0,25 - 0,50	0,80 - 2,50	1,00 - 3,00
	1.4542 X5CrNiCuNb16-4	IN7036	80 - 150	60 - 120	0,15 - 0,40	0,25 - 0,50	0,80 - 2,50	1,00 - 3,00
	1.4571 X6CrNiMoTi17-12-2	IN7036	80 - 150	60 - 120	0,15 - 0,40	0,25 - 0,50	0,80 - 2,50	1,00 - 3,00
	1.4507 X2CrNiMoCuN25-6-3	IN7036	50 - 150	40 - 80	0,10 - 0,30	0,15 - 0,40	0,80 - 2,50	1,00 - 3,00
	1.4529 X1NiCrMoCuN25-20-7	IN7036	50 - 150	40 - 80	0,10 - 0,30	0,15 - 0,40	0,80 - 2,50	1,00 - 3,00
	1.4531 GX2NiCrMoCuN20-18	IN7036	50 - 150	40 - 80	0,10 - 0,30	0,15 - 0,40	0,80 - 2,50	1,00 - 3,00
K	gray cast iron	IN2505	200 - 250	140 - 180	0,15 - 0,40	0,25 - 0,50	0,80 - 2,50	1,00 - 3,00
	nodular cast iron	IN2505	200 - 250	140 - 180	0,15 - 0,40	0,25 - 0,50	0,80 - 2,50	1,00 - 3,00
N	aluminum	-	-	-	-	-	-	-
S	1.4826 GX40CrNiSi22-10	IN7036	70 - 140	60 - 120	0,08 - 0,25	0,10 - 0,35	0,80 - 2,50	1,00 - 3,00
	1.4837 GX40CrNiSi25-12	IN7036	70 - 140	60 - 120	0,08 - 0,25	0,10 - 0,35	0,80 - 2,50	1,00 - 3,00
	1.4848 GX40CrNiSi22-20	IN7036	70 - 140	60 - 120	0,08 - 0,25	0,10 - 0,35	0,80 - 2,50	1,00 - 3,00
	1.4849 GX40NiCrSiNb38-19	IN7036	70 - 140	60 - 120	0,08 - 0,25	0,10 - 0,35	0,80 - 2,50	1,00 - 3,00
	1.4923 X22CrMoV12-1	IN7036	150 - 280	120 - 250	0,15 - 0,40	0,25 - 0,50	0,80 - 2,50	1,00 - 3,00
	1.4938 X12CrNiMoV12-3	IN7036	150 - 280	120 - 250	0,15 - 0,40	0,25 - 0,50	0,80 - 2,50	1,00 - 3,00
	1.4913 X19CrMoVbN11-1	IN7036	120 - 210	90 - 160	0,10 - 0,30	0,15 - 0,40	0,80 - 2,50	1,00 - 3,00
	1.4939 X12CrNiMo12	IN7036	120 - 210	90 - 160	0,10 - 0,30	0,15 - 0,40	0,80 - 2,50	1,00 - 3,00
	1.4962 X12CrNiWTiB16-13	IN7036	80 - 180	60 - 150	0,10 - 0,30	0,12 - 0,40	0,80 - 2,50	1,00 - 3,00
	1.4980 X5NiCrTi26-15	IN7036	50 - 110	40 - 80	0,10 - 0,30	0,12 - 0,40	0,80 - 2,50	1,00 - 3,00
H	titanium alloys	IN7036	-	40 - 60	0,08 - 0,25	0,10 - 0,35	0,80 - 2,50	1,00 - 3,00
	nickel alloys	IN7036	-	20 - 50	0,08 - 0,25	0,10 - 0,35	0,80 - 2,50	1,00 - 3,00
H	hard machining < 54 HRC	-	-	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-	-	-

At max extension lengths the cutting speed Vc has to be reduced!

Ramping data and circular interpolation:

Tool diameter [mm]	max. ramping angle [°]	min. bore dia. [mm]	max. bore dia. [mm]	bore dia. even ground [mm]	max. ap/rev. [mm]	recom. ap/rev. [mm]
20R5	6,0	25,5	40,0	30,2	5,0	1,0
25R5	4,4	35,5	50,0	40,2	5,0	1,0
25R6	6,0	29,0	50,0	38,3	6,0	1,5
30R5	4,0	44,0	60,0	50,2	5,0	1,0
30R6	4,4	35,0	60,0	48,3	6,0	1,5
32R5	2,2	49,0	64,0	60,2	5,0	1,0
32R6	3,9	41,0	64,0	52,3	6,0	1,5
35R5	2,0	55,0	70,0	60,2	5,0	1,0
35R6	2,6	47,0	70,0	58,3	6,0	1,5
40R5	5,6	62,0	80,0	70,2	5,0	1,0
40R6	2,4	57,0	80,0	68,3	6,0	1,5
42R5	5,8	65,5	84,0	74,2	5,0	1,0
42R6	4,0	61,0	84,0	72,3	6,0	1,5
50R5	5,0	81,5	100,0	90,2	5,0	1,0
50R6	5,6	77,0	100,0	88,3	6,0	1,5
52R5	4,7	85,5	104,0	94,2	5,0	1,0
52R6	5,3	81,0	104,0	92,3	6,0	1,5
63R5	3,6	107,5	126,0	106,2	5,0	1,0
63R6	4,0	103,0	126,0	114,3	6,0	1,5
66R6	3,7	109,0	132,0	120,3	6,0	1,5
80R5	2,6	141,5	160,0	150,2	5,0	1,0
80R6	2,9	137,0	160,0	148,3	6,0	1,5

Recommended ramping angle for all diameters: 2°

General information:

insert screw for RPLX10_: **SM35-076-10**

insert screw for RPLX12_: **SO 350801**

torque: **3,0 Nm**

torque: **4,5 Nm**

torque wrench: **DTNV02S with bit DS-T10TB**

torque wrench: **DS-T15S**



Insert:	SDXS0402MPR-MR1	SDXS0402MPR-MM
max. cutting depth:	ap = 0,5 mm	ap = 0,5 mm
Machining group:	P / K / H	P / M / S

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	160 - 220	IN2530	130 - 180	0,4 - 0,9
	alloyed steel 800 N/mm ²	IN2505	140 - 200	IN2530	110 - 160	0,4 - 0,8
	alloyed steel 1100 N/mm ²	IN2505	120 - 180	IN2530	100 - 150	0,4 - 0,7
M	stainless steel	IN2530	90 - 150	IN2530	80 - 130	0,4 - 0,7
K	gray cast iron	IN2505	160 - 250	IN2530	140 - 200	0,4 - 0,9
	nodular cast iron	IN2505	140 - 200	IN2530	120 - 170	0,4 - 0,8
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2530	50 - 80	IN2530	50 - 70	0,4 - 0,6
	titanium alloys	-	-	IN2530	30 - 40	0,3 - 0,5
H	hard machining < 54 HRC	IN2505	60 - 100	-	-	0,2 - 0,5
	hard machining < 63 HRC	-	-	-	-	-

Tipp:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- Approaching feed rate should be reduced by 30%.
- 4-edged insert
- Programming radius R0,9

Ramping data and circular interpolation:

Tool diameter [mm]	max. ramping angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. [mm]
10	9°	13	0,5	20
12	6°	17	0,5	24
16	3,2°	25	0,5	32
20	2,1°	33	0,5	40
25	1,5°	43	0,5	50

General information:

insert screw:

Diameter Ø 10-12: **SM18-033-00**

Diameter Ø 16-25: **SM18-041-00**

torque: **0,5 Nm**

torque: **0,5 Nm**

torque wrench: **DTN005S with bit DS-T06TB**

torque wrench: **DTNV00S with bit DS-T06TB**



Insert:	SDXS0603MPR-MM	SDXS0603MPR-MR	SDXS0603MPR-MR1
max. cutting depth:	ap = 1,0 mm	ap = 1,0 mm	ap = 1,0 mm
Programming radius:	1,7	1,7	1,7

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				recommended depth of cut ap [mm]	Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide			
P	unalloyed steel	IN2505	160 – 240	IN6537	130 – 200	0,5 – 1,0	0,5 – 1,2
	alloyed steel 800 N/mm ²	IN2505	140 – 220	IN6537	110 – 180	0,5 – 1,0	0,5 – 1,2
	alloyed steel 1100 N/mm ²	IN2505	120 – 200	IN6537	100 – 160	0,5 – 1,0	0,5 – 1,0
M	stainless steel	IN4036	90 – 160	IN4036 / IN6537	80 – 140	0,5 – 1,0	0,5 – 1,0
K	gray cast iron	IN2505	160 – 250	IN6537	140 – 200	0,5 – 1,0	0,5 – 1,2
	nodular cast iron	IN2505	140 – 200	IN6537	120 – 180	0,5 – 1,0	0,5 – 1,2
N	aluminum	-	-	-	-	-	-
S	high temperature alloys	-	-	IN4036 / IN6537	50 – 70	0,5 – 1,0	0,5 – 1,0
	titanium alloys	-	-	IN4036 / IN6537	30 – 50	0,5 – 1,0	0,5 – 1,0
H	hard machining < 54 HRC	-	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-	-

Tipp:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- Approaching feed rate should be reduced by 30%.
- 4-edged insert

Ramping data and circular interpolation:

Tool diameter [mm]	SDXS0603MPR-MM, -MR and -MR1-Geometry			
	max. ramping angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. [mm]
16	3,5°	22,0	1,0	32,0
20	3,5°	30,0	1,0	40,0
25	2,5°	40,0	1,0	50,0
32	2,0°	54,0	1,0	64,0
35	2,0°	60,0	1,0	70,0
40	1,3°	70,0	1,0	80,0
42	1,2°	74,0	1,0	84,0
50	1,0°	90,0	1,0	100,0

General information:

insert screw: **SM25-054-00**

torque: **1,1 Nm**

torque wrench: **DTN011S with bit DS-T08TB**



Insert:	SDXS0904MPR-MM	SDXS0904MPR-MR	SDXS0904MPR-MR1	SDXS0904MPR-MRH
max. cutting depth:	ap = 1,5 mm	ap = 1,5 mm	ap = 1,5 mm	ap = 1,5 mm
programming radius:	2,5	2,5	2,5	2,5

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				recommended depth of cut ap [mm]	Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide			
P	unalloyed steel	IN2505	160 – 220	IN6537	130 – 180	0,8 – 1,5	0,5 – 1,6
	alloyed steel 800 N/mm ²	IN2505	140 – 200	IN6537	110 – 160	0,8 – 1,5	0,5 – 1,6
	alloyed steel 1100 N/mm ²	IN2505 / IN2504	120 – 180	IN6537	100 – 150	0,8 – 1,5	0,5 – 1,6
M	stainless steel	IN4036 / IN7035	90 – 150	IN4036 / IN7035	80 – 130	0,8 – 1,5	0,5 – 1,4
K	gray cast iron	IN2505	160 – 250	IN6537	140 – 200	0,8 – 1,5	0,5 – 1,6
	nodular cast iron	IN2505	140 – 200	IN6537	120 – 170	0,8 – 1,5	0,5 – 1,6
N	aluminum	-	-	-	-	-	-
S	high temperature alloys	IN4036 / IN7035	50 – 80	IN4036 / IN7035	50 – 70	0,8 – 1,3	0,5 – 1,4
	titanium alloys	-	-	IN4036	30 – 40	0,8 – 1,3	0,5 – 1,4
H	hard machining < 54 HRC	IN2504	60 – 100	IN2504	60 – 100	0,2 – 0,8	0,5 – 1,4
	hard machining < 63 HRC	IN2504	40 – 80	IN2504	40 – 80	0,2 – 0,8	0,5 – 1,2

Tipp:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- Approaching feed rate should be reduced by 30%.
- 4-edged insert

Ramping data and circular interpolation:

Tool diameter [mm]	SDXS0904MPR-MR, -MM, -MR1 and -MRH-Geometry			
	max. ramping angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. [mm]
25	5,5	32,5	1,5	50,0
30	3,5	42,5	1,5	60,0
32	3,3	46,5	1,5	64,0
35	2,6	52,5	1,5	70,0
40	2,2	62,5	1,5	80,0
42	2,0	66,5	1,5	84,0
50	1,5	82,5	1,5	100,0
52	1,3	86,5	1,5	104,0
63	1,1	109,0	1,5	126,0
66	1,0	115,0	1,5	162,0
80	0,6	143,0	1,5	160,0
85	0,4	153,0	1,5	170,0

General information:

insert screw: **SM30-075-R0**

torque: **2,4 Nm**

torque wrench: **DTNV00S with bit DS-T09TB**



Insert:	SDXS1305MPR-MM	SDXS13_MPR-MR	SDXS13_MPR-MR1	SDXS13_PH	SDXS13_HR
recom. cutting depth:	ap = 1,5 mm	ap = 1,8 mm	ap = 1,8 mm	ap = 1,5 mm	ap = 1,5 mm
max. cutting depth:	ap = 2,0 mm	ap = 2,2 mm	ap = 2,2 mm	ap = 2,0 mm	ap = 2,0 mm
ap... with undercut at 90° shoulder:	no undercut	ap = 2,2 mm	ap = 2,2 mm	ap = 2,0 mm	ap = 2,0 mm
ap... without undercut at 90° shoulder:	ap = 2,0 mm	ap = 1,45 mm	ap = 1,45 mm	ap = 0,50 mm	ap = 0,50 mm
Programming radius:	R 3,5	R 3,5	R 3,5	R 3,2	R 3,2

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN4005	160 - 220	IN6537	130 - 180	0,5 - 1,8
	alloyed steel 800 N/mm ²	IN4005	140 - 200	IN6537	110 - 160	0,5 - 1,8
	alloyed steel 1100 N/mm ²	IN4005 / IN2504	120 - 180	IN6537	100 - 150	0,5 - 1,5
M	stainless steel	IN4036	90 - 150	IN4036	80 - 130	0,5 - 1,5
K	gray cast iron	IN2505	160 - 250	IN6537	140 - 200	0,5 - 1,8
	nodular cast iron	IN2505	140 - 200	IN6537	120 - 170	0,5 - 1,8
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN4036	50 - 80	IN4036	50 - 70	0,5 - 1,5
	titanium alloys	-	-	IN4036	30 - 40	0,5 - 1,5
H	hard machining < 54 HRC	IN2504	60 - 100	-	-	0,5 - 1,7
	hard machining < 63 HRC	-	-	-	-	-

Ramping data and circular interpolation PH and HR Geometry:

ToolØ [mm]	max. ramping angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. [mm]	max. ap/rev. [mm]
32	15,0	38,1	2,0	64	2,0
35	10,8	43,3	2,0	70	2,0
40	7,4	54,0	2,0	80	2,0
42	6,6	58,0	2,0	84	2,0
50	4,7	74,0	2,0	100	2,0
52	4,3	78,0	2,0	104	2,0
63	3,1	100,0	2,0	126	2,0
66	2,9	106,1	2,0	132	2,0
80	1,8	134,1	2,0	160	2,0
100	1,3	174,1	2,0	200	2,0

Ramping data and circular interpolation MM , MR and MR1 Geometry:

ToolØ [mm]	max. ramping angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. [mm]	max. ap/rev. [mm]
32	10,8	38,1	2,2	64	2,2
35	8,1	44,1	2,2	70	2,2
40	5,7	54,0	2,2	80	2,2
42	5,1	58,0	2,2	84	2,2
50	3,8	73,9	2,2	100	2,2
52	3,4	77,9	2,2	104	2,2
63	2,5	99,9	2,2	126	2,2
66	2,3	105,9	2,2	132	2,2
80	1,3	134,3	2,2	160	2,2
100	1,0	174,3	2,2	200	2,2

General information:

insert screw for insert size 19: **SM40-100-R0**

torque: **4 Nm**

torque wrench: **DTNV005 with bit DS-T15TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	SDXS1605MPR-MM	SDXS1605MPR-MR	SDXS1605MPR-MR1
max. cutting depth:	ap = 2,7 mm	ap = 2,7 mm	ap = 2,7 mm
Programming radius:	R4,2	R4,2	R4,2

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				recommended depth of cut ap [mm]	Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide			
P	unalloyed steel	IN2505	160 - 240	IN6537	130 - 200	1,0 - 2,0	1,2 - 2,5
	alloyed steel 800 N/mm ²	IN2505	140 - 220	IN6537	110 - 180	1,0 - 2,0	1,2 - 2,5
	alloyed steel 1100 N/mm ²	IN2505	120 - 200	IN6537	100 - 160	1,0 - 2,0	1,0 - 2,2
M	stainless steel	IN6537 / IN4036	90 - 160	IN4036 / IN7035	80 - 140	1,0 - 2,0	1,0 - 2,0
K	gray cast iron	IN2505	160 - 250	IN6537	140 - 200	1,0 - 2,0	1,2 - 2,5
	nodular cast iron	IN2505	140 - 200	IN6537	120 - 180	1,0 - 2,0	1,0 - 2,2
N	aluminum	-	-	-	-	-	-
S	high temperature alloys	-	-	IN4036 / IN7035	50 - 70	1,0 - 2,0	1,0 - 1,8
	titanium alloys	-	-	IN4036 / IN7035	30 - 50	1,0 - 2,0	1,0 - 1,8
H	hard machining < 54 HRC	-	-	-	-	-	0,5 - 1,4
	hard machining < 63 HRC	-	-	-	-	-	-

Ramping data and circular interpolation:

Tool diameter [mm]	SDXS1605MPR-MM, -MR, and -MR1-Geometry			
	max. Ramping angle [°]	min. Bore dia. [mm]	max. ap/rev. [mm]	max. Bore dia. [mm]
50	5,2	74,4	2,7	100,0
52	4,9	78,4	2,7	104,0
63	3,4	101,0	2,7	126,0
66	3,1	107,0	2,7	132,0
80	2,3	135,0	2,7	160,0
85	2,1	145,0	2,7	170,0
100	1,65	175,0	2,7	200,0
125	1,2	224,0	2,7	250,0
160	0,7	295,0	2,7	320,0

Tipp:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- Approaching feed rate should be reduced by 30%.
- 4-edged insert

General information:

insert screw: **SM50-130-R0**

torque: **6,0 Nm**

torque wrench: **DTNV00S with bit DS-T20TB**



Insert:	SDES19_MPR-MR1	SDXS19_MPR-MR	SDMS19_PH
recom. Depth of cut:	ap = 2,5 mm	ap = 2,5 mm	ap = 2,0 mm
max. cutting depth:	ap = 3,7 mm	ap = 3,7 mm	ap = 3,0 mm
ap... with undercut at 90° shoulder:	ap = 3,7 mm	ap = 3,7 mm	ap = 3,0 mm
ap... without undercut at 90° shoulder:	ap = 1,50 mm	ap = 1,50 mm	ap = 1,10 mm
Programming radius:	R 5,5	R 5,5	R 4,5

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]			Feed per tooth fz [mm]	
		1st choice dry machining resp. wear resistant carbide	1st choice wet machining resp. tough carbide			
P	unalloyed steel	IN4005	160 - 220	IN6537 / IN4030	130 - 180	1,5 - 3
	alloyed steel 800 N/mm ²	IN4005	140 - 200	IN6537 / IN4030	110 - 160	1,2 - 3
	alloyed steel 1100 N/mm ²	IN4005	120 - 180	IN6537 / IN4030	100 - 150	1 - 2,5
M	stainless steel	IN4036	90 - 150	IN4036	80 - 130	1 - 2
K	gray cast iron	IN2505	160 - 250	IN6537 / IN4030	140 - 200	1,5 - 3
	nodular cast iron	IN2505	140 - 200	IN6537 / IN4030	120 - 170	1,2 - 3
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN4036	50 - 80	IN4036	50 - 70	0,8 - 1,8
	titanium alloys	-	-	IN4036	30 - 40	0,5 - 1,5
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

Tipp:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- Approaching feed rate should be reduced by 30%.
- 4-edged insert

Ramping data and circular interpolation PH - Geometry:

Tool Ø [mm]	max. ramping angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. [mm]	max. ap/rev. [mm]
80	4,3	121,8	3,0	160	3,0
100	3,0	161,8	3,0	200	3,0
125	2,1	211,8	3,0	250	3,0
160	1,5	281,8	3,0	320	3,0

Ramping data and circular interpolation MPR-MR- / MPR-MR1-Geometry:

Tool Ø [mm]	max. ramping angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. [mm]	max. ap/rev. [mm]
80	3,6	121,9	3,7	160	3,7
100	2,5	161,9	3,7	200	3,7
125	1,8	211,9	3,7	250	3,7
160	1,3	281,9	3,7	320	3,7

General information:

insert screw for insert size 19: **SM60-135-R0**

torque: **8 Nm**

torque wrench: **DTNV005 with bit DS-T25TB**



Insert:	SDMS130516R - PP
max. cutting depth:	ap = 4,9 mm
ap ... with undercut at 90° shoulder:	ap _{max} = 4,9 mm
Programming radius:	6,4

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN4005	160 - 220	IN4030	130 - 180	0,2 - 0,5
	alloyed steel 800 N/mm ²	IN4005	140 - 200	IN4030	110 - 160	0,2 - 0,5
	alloyed steel 1100 N/mm ²	IN4005	120 - 180	IN4030	100 - 150	0,2 - 0,5
M	stainless steel	IN4036	90 - 150	IN4036	80 - 130	0,2 - 0,45
K	gray cast iron	IN4005	160 - 250	IN4030	140 - 200	0,2 - 0,6
	nodular cast iron	IN4005	140 - 200	IN4030	120 - 170	0,2 - 0,6
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN4036	50 - 80	IN4036	50 - 70	0,2 - 0,45
	titanium alloys	-	-	IN4036	30 - 40	0,2 - 0,45
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

Tipp:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- Approaching feed rate should be reduced by 30%.
- 4-edged insert

General information:

insert screw: **SM40-100-R0**

torque: **4 Nm**

torque wrench: **DTNV00S with bit DS-T15TB**



Insert:	SDMS1906ZPR-PP
max. cutting depth:	ap = 7,8 mm
ap _{max} with undercut at 90° shoulder:	ap _{max} = 7,8 mm
Programming radius:	8,4

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN4030	160 – 220	IN4030	130 – 180	0,25 – 0,6
	alloyed steel 800 N/mm ²	IN4030	140 – 200	IN4030	110 – 160	0,25 – 0,6
	alloyed steel 1100 N/mm ²	IN4030	120 – 180	IN4030	100 – 150	0,25 – 0,6
M	stainless steel	IN4035	90 – 150	IN4035	80 – 130	0,2 – 0,5
K	gray cast iron	IN4030	160 – 250	IN4030	140 – 200	0,25 – 0,8
	nodular cast iron	IN4030	140 – 200	IN4030	120 – 170	0,25 – 0,8
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN4035	50 – 80	IN4035	50 – 70	0,2 – 0,55
	titanium alloys	-	-	IN4035	30 – 40	0,2 – 0,55
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

Tipp:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- Approaching feed rate should be reduced by 30%.
- 4-edged insert

General information:

insert screw: **SM60-135-R0**

torque: **8 Nm**

torque wrench: **DTNV00S with bit DS-T25TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	SDES130508N-PF SDES130508N-PF1	SDMS130512R-PP
Feed per tooth:	fz = 0,1 - 0,3 mm	fz = 0,1 - 0,3 mm
Recom. width of cut:	ae = 9 mm	ae = 8 mm
max. width of cut:	ae = 11,9 mm	ae = 11,6 mm

Recommended cutting data:

ISO	Material	SDES 1305...		SDMS 1305...	
		Grade	Cutting speed Vc [m/min]	Grade	Cutting speed Vc [m/min]
P	unalloyed steel	IN4005	150 - 200	-	-
	alloyed steel 800 N/mm ²	IN4005	130 - 180	-	-
	alloyed steel 1100 N/mm ²	IN4005	110 - 170	-	-
M	stainless steel	-	-	IN4036 / IN4030	80 - 150
K	gray cast iron	IN4005	160 - 220	-	-
	nodular cast iron	IN4030	140 - 200	-	-
N	aluminum	-	-	-	-
S	high temperature alloys	-	-	IN4036	40 - 70
	titanium alloys	IN4036	30 - 50	-	-
H	hard machining < 54 HRC	-	-	-	-
	hard machining < 63 HRC	-	-	-	-

Tipp:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- The longer the extension length, the lower the cutting speed.
- If plunging or grounding feed rate should be reduced by 30% on a length of 3 mm.
- It is recommended to retract from the contour, before travelling back to safety area (approx. 0,2 - 0,5 mm).

General information:

insert screw: **SM40-100-R0**

torque: **4,0 Nm**

torque wrench: **DTNV00S with bit DS-T15TB**

Note

A large grid of graph paper for taking notes, consisting of 20 columns and 30 rows of small squares. The grid is bounded by a thin grey line on the top, left, and bottom, and a thick blue vertical bar on the right side.



Insert:	LNXF0905R01
Recomm. cutting depth:	ap = 0,5 mm
max. cutting depth:	ap = 1,0 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	-	-	-	-	-
	alloyed steel 800 N/mm ²	-	-	-	-	-
	alloyed steel 1100 N/mm ²	-	-	-	-	-
M	stainless steel	-	-	-	-	-
K	gray cast iron	IN76N	400 - 800	-	-	0,1 - 0,4
	nodular cast iron	IN76N	400 - 800	-	-	0,1 - 0,4
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN76N	400 - 800	-	-	0,1 - 0,2
	titanium alloys	-	-	-	-	-
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- Approaching feed rate should be reduced by 30%.
- 4-edged insert
- Programming radius R3,4

Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. [mm]	max. ap/rev. [mm]	max. Bore dia. [mm]
25	1	39	1	50
32	0,6	53	0,7	64
40	0,5	69	0,7	80



Insert:	TNXN1207N0104
Recomm. cutting depth:	ap = 1,0 mm
max. cutting depth:	ap = 2,0 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	-	-	-	-	-
	alloyed steel 800 N/mm ²	-	-	-	-	-
	alloyed steel 1100 N/mm ²	-	-	-	-	-
M	stainless steel	-	-	-	-	-
K	gray cast iron	IN76N	400 - 800	-	-	0,15 - 0,5
	nodular cast iron	IN76N	400 - 800	-	-	0,15 - 0,5
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN76N	400 - 800	-	-	0,15 - 0,3
	titanium alloys	-	-	-	-	-
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 63 HRC	-	-	-	-	-

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- Approaching feed rate should be reduced by 30%.
- 6-edged insert
- Programming radius R3,4

Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. [mm]	max. ap/rev. [mm]	max. Bore dia. [mm]
50	0,5	84	1,1	100
63	0,4	110	1,1	126
80	0,3	144	1,1	160



Insert:	UNLU0402M0TR	UNLU0402M0TR-ML
Recomm. cutting data:	ap = 0,5 mm	ap = 0,5 mm
Machining group:	P / M / K / H	M / S

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505 / IN2504	160 – 220	IN6537	130 – 180	0,5 – 1,0
	alloyed steel 800 N/mm ²	IN2505 / IN2504	140 – 200	IN6537	110 – 160	0,5 – 0,8
	alloyed steel 1100 N/mm ²	IN2505 / IN2504	120 – 180	IN6537	100 – 150	0,5 – 0,7
M	stainless steel	IN6537 / IN2535	90 – 150	IN6537 / IN2535	80 – 130	0,5 – 0,8
K	gray cast iron	IN2505 / IN6537	160 – 250	IN6537	140 – 200	0,5 – 1,0
	nodular cast iron	IN2505 / IN6537	140 – 200	IN6537	120 – 170	0,5 – 0,8
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN6537	50 – 80	IN6537 / IN2535	50 – 70	0,3 – 0,6
	titanium alloys	-	-	IN6537 / IN2535	30 – 40	0,3 – 0,6
H	hard machining < 54 HRC	IN2504	60 – 100	-	-	0,2 – 0,5
	hard machining < 63 HRC	-	-	-	-	-

Tipp:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- Approaching feed rate should be reduced by 30%.
- 4-edged insert
- Programming radius R0,9

Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. [mm]	max. ap/rev. [mm]	max. Bore dia. [mm]
10	0,6	16,6	0,2	10
12	1	20,6	0,4	24
16	1	28,6	0,5	32
20	1	36,6	0,5	40
25	0,7	46,6	0,5	50
32	0,6	60,6	0,5	64

General information:

insert screw: **SM18-041-00**

torque: **0,5 Nm**

torque wrench: **DTN005S with bit DS-TP06TB**



Insert:	UNLU0603M0TR	UNLU0603M0TR - ML	UNLU0603M0TR - MM
Recomm. cutting data:	ap = 0,8 mm	ap = 0,5 mm	ap = 0,6 mm
max. cutting depth:	ap = 1,0 mm	ap = 1,0 mm	ap = 1,0 mm
Machining group:	P / M / K / H	S	M / S

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]			Feed per tooth fz [mm]	
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505 / IN2504	160 - 220	IN6537	130 - 180	0,7 - 1,3
	alloyed steel 800 N/mm ²	IN2505 / IN2504	140 - 200	IN6537	110 - 160	0,6 - 1,2
	alloyed steel 1100 N/mm ²	IN2505 / IN2504	120 - 180	IN6537	100 - 150	0,5 - 1
M	stainless steel	IN2036	90 - 150	IN2036 / IN6537	80 - 130	0,6 - 0,9
K	gray cast iron	IN2505 / IN2504	160 - 250	IN6537	140 - 200	0,7 - 1,3
	nodular cast iron	IN2505 / IN2504	140 - 200	IN6537	120 - 170	0,5 - 1
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2036	50 - 80	IN2036	50 - 70	0,4 - 0,7
	titanium alloys	-	-	IN2036 / IN6537	30 - 40	0,3 - 0,6
H	hard machining < 54 HRC	IN2504	60 - 100	-	-	0,2 - 0,5
	hard machining < 63 HRC	-	-	-	-	-

Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. [mm]	max. ap/rev. [mm]	Bore dia. even ground [mm]	max. ap/rev. [mm]	max. Bore dia. [mm]	max. ap/rev. [mm]
16 R2	2,0	22,1	0,6	25,4	1,0	32	1,0
20 R2	1,8	28,6	0,8	32,3	1,0	40	1,0
25 R2	1,3	38,4	0,9	42,3	1,0	50	1,0
30 R2	1,0	48,4	1,0	52,3	1,0	60	1,0
32 R2	1,0	52,4	1,0	56,3	1,0	64	1,0
35 R2	0,8	58,4	1,0	62,3	1,0	70	1,0
40 R2	0,8	68,2	1,0	72,3	1,0	80	1,0
42 R2	0,7	72,3	1,0	76,3	1,0	84	1,0
50 R2	0,6	88,1	1,0	92,3	1,0	100	1,0
52 R2	0,6	92,1	1,0	96,2	1,0	104	1,0
63 R2	0,5	113,9	1,0	118,2	1,0	126	1,0
66 R2	0,5	120,0	1,0	124,2	1,0	132	1,0
80 R2	0,4	147,9	1,0	152,2	1,0	160	1,0
85 R2	0,4	157,9	1,0	162,2	1,0	170	1,0

Tipp:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- Approaching feed rate should be reduced by 30%.
- For 16 mm tool diameter the max. cutting depth ap = 0.7 mm.
- 4-edged insert
- Programming radius R2

General information:

insert screw: **SM25-064-00**

torque: **1,1 Nm**

torque wrench: **DTN011S with bit DS-T08TB**



Insert:	UNLU0904MOTR	UNLU0904MOTR-ML	UNLU0904MOTR-MM
Recomm. cutting data:	ap = 1,0 mm	ap = 0,8 mm	ap = 0,8 mm
max. cutting depth:	ap = 1,5 mm	ap = 1,5 mm	ap = 1,5 mm
Machining group:	P / M / K / H	S	M / S

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505 / IN2504	160 - 220	IN6537	130-180	0,8-1,5
	alloyed steel 800 N/mm ²	IN2505 / IN2504	140 - 200	IN6537	110-160	0,8-1,3
	alloyed steel 1100 N/mm ²	IN2505 / IN2504	120 - 180	IN6537	100-150	0,8-1,2
M	stainless steel	IN6537 / IN2036	90 - 150	IN6537 / IN2036	80-130	0,7-1,0
K	gray cast iron	IN2505 / IN6537	160 - 250	IN6537	140-200	0,8-1,5
	nodular cast iron	IN2505 / IN6537	140 - 200	IN6537	120-170	0,8-1,2
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2036	50 - 80	IN2036	50-70	0,7-1,0
	titanium alloys	-	-	IN2036	30-40	0,7-0,9
H	hard machining < 54 HRC	IN2504	60 - 100	-	-	0,4 - 0,7
	hard machining < 63 HRC	-	-	-	-	-

Tipp:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- Approaching feed rate should be reduced by 30%.
- 4-edged insert
- Programming radius R2,5

Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. [mm]	max. ap/rev. [mm]	max. Bore dia. [mm]
25	3,0	42	1,5	50
32	2,0	56	1,5	64
35	1,5	59	1,5	70
40	1,3	72	1,5	80
42	1,1	76	1,5	84
50	1,0	92	1,5	100
52	1,0	96	1,5	104
63	0,8	118	1,5	126
66	0,8	124	1,5	132
80	0,8	152	1,5	160

General information:

insert screw: **SM35-088-10**

torque: **3,0 Nm**

torque wrench: **DTN030S with bit DS-T10TB**



Insert:	UNLU1105MOTR	UNLU1105MOTR-ML
Recomm. cutting data:	ap = 1,5 mm	ap = 1,2 mm
max. cutting depth:	ap = 2,0 mm	ap = 2,0 mm
Machining group:	P / M / K / H	M / S

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505 / IN2504	160 - 220	IN6537	130-180	0,8-2,0
	alloyed steel 800 N/mm ²	IN2505 / IN2504	140 - 200	IN6537	110-160	0,8-1,8
	alloyed steel 1100 N/mm ²	IN2505 / IN2504	120 - 180	IN6537	100-150	0,8-1,6
M	stainless steel	IN6537	90 - 150	IN6537	80-130	0,8-1,2
K	gray cast iron	IN2505 / IN6537	160 - 250	IN6537	140-200	0,8-2,0
	nodular cast iron	IN2505 / IN6537	140 - 200	IN6537	120-170	0,8-1,6
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN6537	50 - 80	IN6537	50-70	0,8-1,4
	titanium alloys	-	-	IN6537	30-40	0,7-1,2
H	hard machining < 54 HRC	IN2504	60 - 100	-	-	0,5 - 1,0
	hard machining < 63 HRC	-	-	-	-	-

Tipp:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- Approaching feed rate should be reduced by 30%.
- 4-edged insert
- Programming radius R3,0

Ramping data and circular interpolation:

Tool diameter [mm]	max. Ramping angle [°]	min. Bore dia. [mm]	max. ap/rev. [mm]	max. Bore dia. [mm]
30	0,5	41	2	60
32	0,5	45	2	64
35	0,5	51	2	70
40	0,55	61	2	80
42	0,5	65	2	84
50	0,5	81	2	100
52	0,45	85	2	104
63	0,45	107	2	126
66	0,4	113	2	132
80	0,35	141	2	160
100	0,3	181	2	200
125	0,25	231	2	250
160	0,2	301	2	320
200	0,15	381	2	400

General information:

insert screw: **TS 50A1211/HG**

torque: **6,0 Nm**

torque wrench: **DTNV00S with bit DS-T20TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	UNLV0603MOTR
Recomm. cutting data:	ap = 0,8 mm
max. cutting depth:	ap = 1,0 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505 / IN2504	160 - 220	IN2530	130 - 180	0,7 - 1,6
	alloyed steel 800 N/mm ²	IN2505 / IN2504	140 - 200	IN2530	110 - 160	0,6 - 1,5
	alloyed steel 1100 N/mm ²	IN2505 / IN2504	120 - 180	IN2530	100 - 150	0,5 - 1,3
M	stainless steel	IN2530 / IN6537	90 - 150	IN2530 / IN6537	80 - 130	0,6 - 1,2
K	gray cast iron	IN2505 / IN2504	160 - 250	IN2530	140 - 200	0,7 - 1,6
	nodular cast iron	IN2505 / IN2504	140 - 200	IN2530	120 - 170	0,5 - 1,3
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2530	50 - 80	IN2530	50 - 70	0,4 - 1,0
	titanium alloys	-	-	IN2530 / IN6537	30 - 40	0,3 - 0,9
H	hard machining < 54 HRC	IN2504	60 - 100	-	-	0,2 - 0,7
	hard machining < 63 HRC	-	-	-	-	-

Tipp:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- Approaching feed rate should be reduced by 30%.
- For 16 mm tool diameter the max. cutting depth ap = 0.7 mm.
- 4-edged insert
- Programming radius R2,0

Ramping data and circular interpolation:

ToolØ [mm]	straight ramp down			helical ramp down			
	max. ramping angle [°]	max. ap [mm]	min. Length L [mm]	min. Bore dia. [mm]	max. ap/rev. [mm]	max. Bore dia. [mm]	max. ap/rev. [mm]
16	5,1	0,7	7,9	26	0,7	32	0,7
20	2,5	1,0	23,0	33	1,0	40	1,0
25	2,5	1,0	23,0	43	1,0	50	1,0
32	1,4	1,0	41,0	57	1,0	64	1,0
40	1,2	1,0	47,8	73	1,0	80	1,0
50	1,1	1,0	52,2	93	1,0	100	1,0
52	0,7	1,0	81,9	97	1,0	104	1,0
63	0,6	1,0	95,6	119	1,0	126	1,0

General information:

insert screw: **SM25-064-01**

torque: **1,1 Nm**

torque wrench: **DTN011S with bit DS-T08TB**



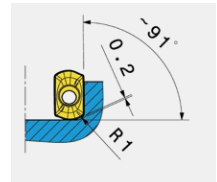
Insert:	UOMT0602R
max. cutting depth:	ap = 0,5 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 – 290	IN2530	200 – 240	0,30 – 0,80
	alloyed steel 800 N/mm ²	IN2505	210 – 250	IN2530	160 – 200	0,30 – 0,50
	alloyed steel 1100 N/mm ²	IN2505	160 – 180	IN2530	110 – 130	0,30
M	stainless steel	IN2035	120 – 180	IN2035	80 – 130	0,30 – 0,80
K	gray cast iron	IN2504	180 – 250	IN2530	150 – 200	0,30 – 0,80
	nodular cast iron	IN2504	140 – 210	IN2530	110 – 160	0,30 – 0,50
N	aluminum	-	-	-	-	-
S	high temperature alloys	IN2035	110 – 125	IN2035	60 – 80	0,30
	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,30
H	hard machining < 54 HRC	IN2504	130 – 150	-	-	0,30
	hard machining < 63 HRC	IN2504	110 – 130	-	-	0,30

Tipp:

- For tightening the insert screws please always use a torque driver (0,5 Nm).
- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.



Programming Tip:

Please use a corner radius of 1 mm in your NC-program when machining 3D-contours.
The maximum unmachined allowance will then be up to 0.2 mm.

Ramping data and circular interpolation:

Tool diameter [mm]	Ramping angle [°]	max. ap/U [mm]	min. Bore dia. even ground [mm]	max. Bore dia. even ground [mm]	max. Bore dia. uneven ground [mm]
9,5 R1	10,5	0,5	11	14,25	18
10 R1	10,0	0,5	12	15,25	19
11,5 R1	7,0	0,5	15	18,25	22
12 R1	6,5	0,5	16	19,25	23
13,5 R1	5,5	0,5	19	22,25	26
14 R1	5,2	0,5	20	23,25	27
15 R1	4,4	0,5	22	25,25	29
16 R1	4,0	0,5	24	27,25	31
20 R1	2,5	0,5	32	35,25	39
25 R1	2,0	0,5	42	45,25	49
30 R1	1,7	0,5	52	56,25	59
32 R1	1,6	0,5	56	59,25	63
35 R1	1,4	0,5	62	65,25	69
40 R1	1,2	0,5	72	75,25	79

General information:

insert screw: **SM18-041-00**

torque: **0,5 Nm**

torque wrench: **DTN005S with bit DS-TP06TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	WCNT060205FR - FL	WCNW060205TR
max. cutting depth:	ap = 0,8 mm	ap = 0,8 mm
Programming radius:	2 mm	2 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				recommended depth of cut ap [mm]	Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide			
P	unalloyed steel	IN2505	160 - 220	IN6537	130 - 180	0,5 - 0,8	0,5 - 1,0
	alloyed steel 800 N/mm ²	IN2505	140 - 200	IN6537	110 - 160	0,5 - 0,8	0,5 - 1,0
	alloyed steel 1100 N/mm ²	IN2505	120 - 180	IN6537	100 - 150	0,5 - 0,8	0,5 - 1,0
M	stainless steel	IN6537/IN4036	90 - 150	IN6537/IN4036	80 - 130	0,5 - 0,8	0,5 - 0,9
K	gray cast iron	IN2505	160 - 250	IN6537	140 - 200	0,5 - 0,8	0,5 - 1,1
	nodular cast iron	IN2505	140 - 200	IN6537	120 - 170	0,5 - 0,8	0,5 - 1,1
N	aluminum	-	-	-	-	-	-
S	high temperature alloys	-	-	IN4036	50 - 70	0,5 - 0,7	0,5 - 0,9
	titanium alloys	-	-	IN4036	30 - 40	0,5 - 0,7	0,5 - 0,9
H	hard machining < 54 HRC	IN2504	60 - 100	-	-	0,3 - 0,6	0,5 - 0,7
	hard machining < 63 HRC	-	-	-	-	-	-

Tipp:

- The worse the machinability of the material, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- Approaching feed rate should be reduced by 30%.
- Programming radius R2,0
- 3-edged insert

Ramping data and circular interpolation:

Tool diameter [mm]	Ramping angle [°]	min. Bore dia. [mm]	max. ap/rev. [mm]	Bore dia. [mm]
16	14,4	19,8	0,7	32,0
20	5,9	27,6	0,7	40,0
25	5,3	37,6	0,7	50,0
30	3,5	47,6	0,7	60,0
32	3,1	51,6	0,7	64,0
35	2,2	57,6	0,7	70,0
40	2,1	67,6	0,7	80,0
42	1,6	71,6	0,7	84,0
50	1,3	87,6	0,7	100,0
52	1,2	91,6	0,7	104,0

General information:

insert screw: **M25-054-00**

torque: **1,1 Nm**

torque wrench: **DTN011S with bit DS-T08TB**

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert:	WNMU04T302N	WNMU04T304N	WNCU04T302FN-P	WNCU04T304FN-P	WNCU04T308FN-P
Feed per tooth:	fz = 0,07 - 0,20 mm	fz = 0,07 - 0,20 mm	fz = 0,07 - 0,20 mm	fz = 0,07 - 0,20 mm	fz = 0,07 - 0,20 mm
Recom. side infeed [mm]:	step 2,5	step 2,5	step 2,5	step 2,5	step 2,5
max. side infeed [mm]:	step 3,8	step 3,8	step 3,8	step 3,8	step 3,8
eff. tool diameter [mm]:	nom.Ø + 0,1	nom.Ø	nom.Ø + 0,1	nom.Ø	nom.Ø + 0,1

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 - 290	IN2530 / IN4030	200 - 240	0,07 - 0,18
	alloyed steel 800 N/mm ²	IN2505	210 - 250	IN2530 / IN4030	160 - 200	0,07 - 0,13
M	alloyed steel 1100 N/mm ²	IN2505	160 - 180	IN2530 / IN4030	110 - 130	0,07
	stainless steel	IN2505	120 - 180	IN4030 / IN6535	80 - 130	0,07 - 0,18
K	gray cast iron	IN2504	180 - 250	IN2530 / IN4030	150 - 200	0,07 - 0,18
	nodular cast iron	IN2505	140 - 210	IN2530 / IN4030	110 - 160	0,07 - 0,13
N	aluminum	IN10K	800 - 1500	IN10K	500 - 800	0,07 - 0,20
S	high temperature alloys	IN2505	110 - 125	IN4030 / IN6535	60 - 80	0,07
	titanium alloys	IN2505	40 - 50	IN4030 / IN6535	30 - 40	0,07
H	hard machining < 54 HRC	IN2504	30 - 40	-	-	0,07
	hard machining < 63 HRC	-	-	-	-	-

Tipp:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- The longer the programming length, the lower the cutting speed is.
- When plunging and for bottom contact the feed rate should be reduced by 30% at a distance of 3 mm.
- Retraction of the contour in 2 axes before retracting to the safety plane (approx. 0.2 - 0.5 mm) is recommended.
- Pay attention to the change in diameter depending on the insert selection.

General information:

insert screw: **SM25-064-00**

torque: **1,1 Nm**





torque wrench: **DTN011S with bit DS-T08TB**

CHIPSURFER 16T_ / 17T_ / 18T_ & 16N_ / 17N_

Slot mill: 16T_ / 17T_ / 18T
 Slot mill & chamfer mill: 16N_ / 17N_



Recommended cutting data:

ISO	Material	Cutter type				
			Cutting speed Vc [m/min]	Feed per tooth fz [mm]	Cutting speed Vc [m/min]	Feed per tooth fz [mm]
P	steel	16N/17N	80 - 180 [120]	0,05 - 0,15	-	-
		16T/17T/18T	100 - 120 [100]	0,05 - 0,15	160 - 220 [200]	0,03 - 0,15
	tool steel	16N/17N	60 - 120 [80]	0,03 - 0,12	-	-
		16T/17T/18T	60 - 110 [90]	0,03 - 0,12	150 - 200 [180]	0,03 - 0,15
M	stainless steel	16N/17N	50 - 130 [60]	0,03 - 0,15	-	-
K	cast alloys gray cast iron	16T/17T/18T	80 - 120 [80]	0,03 - 0,15	80 - 140 [110]	0,05 - 0,15
		16N/17N	70 - 100 [80]	0,03 - 0,12	-	-
N	aluminum	16T/17T/18T	120 - 200 [120]	0,03 - 0,12	160 - 250 [200]	0,05 - 0,15
		16N/17N	150 - 600 [300]	0,05 - 0,15	-	-
		16T/17T/18T	400 - 800 [500]	0,10 - 0,20	400 - 1200 [1000]	0,1 - 0,25
	copper	16N/17N	70 - 100 [80]	0,03 - 0,15	-	-
		16T/17T/18T	80 - 100 [80]	0,10 - 0,15	150 - 250 [200]	0,1 - 0,2
S	nickel alloys	16N/17N	30 - 70 [40]	0,02 - 0,12	-	-
H	titanium	16T/17T/18T	30 - 60 [40]	0,02 - 0,12	30 - 70 [40]	0,02 - 0,12
		hard machining	-	-	-	-

Recommended values in brackets [],

Note

A large grid of graph paper for taking notes. The grid consists of 30 columns and 30 rows of small squares, providing ample space for handwritten or typed notes. The grid is located in the central and lower portions of the page.

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Feed per tooth fz[mm]	Recommended depth of cut ap [mm]	Grade
P	unalloyed steel	12	140 - 200	0,04 - 0,10	3,6	IN2005
		16	140 - 200	0,05 - 0,10	4,8	IN2005
		20	140 - 200	0,05 - 0,10	6	IN2005
		25	140 - 200	0,05 - 0,10	7,5	IN2005
	alloyed steel 800 N/mm ²	12	140 - 200	0,04 - 0,10	3,6	IN2005
		16	140 - 200	0,05 - 0,10	4,8	IN2005
		20	140 - 200	0,05 - 0,10	6	IN2005
		25	140 - 200	0,05 - 0,10	7,5	IN2005
	alloyed steel 1100 N/mm ²	12	120 - 180	0,03 - 0,08	3,6	IN2005
		16	120 - 180	0,04 - 0,10	4,8	IN2005
		20	120 - 180	0,04 - 0,10	6	IN2005
		25	120 - 180	0,04 - 0,10	7,5	IN2005
M	stainless steel	12	60 - 100	0,03 - 0,08	3,6	IN2005
		16	60 - 100	0,04 - 0,10	4,8	IN2005
		20	60 - 100	0,04 - 0,10	6	IN2005
		25	60 - 100	0,04 - 0,10	7,5	IN2005
K	gray cast iron cast alloys	12	160 - 220	0,04 - 0,10	3,6	IN2005
		16	160 - 220	0,05 - 0,10	4,8	IN2005
		20	160 - 220	0,05 - 0,10	6	IN2005
		25	160 - 220	0,05 - 0,10	7,5	IN2005
N	non-ferrous metals	-	-	-	-	-
		-	-	-	-	-
S	super alloys	12	25 - 80	0,03 - 0,08	3,6	IN2005
		16	25 - 80	0,04 - 0,10	4,8	IN2005
		20	25 - 80	0,04 - 0,10	6	IN2005
		25	25 - 80	0,04 - 0,10	7,5	IN2005
H	hard machining	-	-	-	-	-
		-	-	-	-	-


Recommended cutting data:

Material				Cutting speed Vc [m/min]	Feed per tooth f [mm]		
ISO		AISI/SAE/ASTM	HB		TR13	TR15	TR17
P	non-alloy steel	1020	130-180	120 - 200	0,04 - 0,12	0,05 - 0,15	0,06 - 0,15
	low alloy steel	4030	260-300	200 - 300	0,04 - 0,12	0,05 - 0,15	0,06 - 0,15
	low alloy steel	3135	HRC 35-40	80 - 120	0,02 - 0,06	0,03 - 0,12	0,04 - 0,12
	high alloy steel	H13	200-220	100 - 150	0,03 - 0,07	0,04 - 0,12	0,04 - 0,12
M	martensitic stainless steel	420	200	100 - 150	0,02 - 0,06	0,04 - 0,12	0,04 - 0,12
	austenitic stainless steel	304L	200	80 - 120	0,02 - 0,06	0,03 - 0,10	0,03 - 0,12
K	gray cast iron	Class 40	250	150 - 200	0,04 - 0,12	0,05 - 0,20	0,05 - 0,20
	malleable cast iron	Class 65 45 12	200	130 - 180	0,04 - 0,10	0,05 - 0,18	0,05 - 0,18
N	non-ferrous metals	-	-	-	-	-	-
		-	-	-	-	-	-
S	high temperature resistant alloys	Inconel 718	HRC 36-40	20 - 30	0,015 - 0,10	0,02 - 0,12	0,02 - 0,12
		AMS R56400	HRC 40-45	30 - 40	0,015 - 0,06	0,02 - 0,12	0,02 - 0,12
H	hard machining	-	-	-	-	-	-
		-	-	-	-	-	-

Parameter:

MultiSurfer Thread milling cutter 18Y / 19Y										
Tool	Tool Ø	Z	κ	P	TPI	D-min (inner thread)	thread-Ø ISO (inner thread)	thread-Ø ISO (External thread)	UNC	BSP
18Y24730LPRP60	24,7	6	60°	3-5	5-3	36	≥M36	M24-M48	G ≥ 1 3/4	-
19Y31740LQRP60	31,7	8	60°	4-6	6-4	46	≥M48	M36-M64	G ≥ 2 1/4	-
19Y39760LRRP60	39,7	10	60°	6-8	4-3	57	≥M64	≥M64	G ≥ 2 1/2	-
18Y24750LPRP55	24,7	6	55°	-	5-3	36	-	-	-	G ≥ 1 3/4
19Y31760LQRP55	31,7	8	55°	-	6-4	46	-	-	-	G ≥ 2 1/4
19Y39740LRRP55	39,7	10	55°	-	4-3	57	-	-	-	G ≥ 2 1/2

CHIP SURFER 45A_ SPEED MILL

Rough mills for following applications:
Slot milling, roughing from the solid,
machining of residual material and face
milling on HSC



Recommended cutting data:

ISO	Material	Dc [mm]	Programming radius	Cutting speed Vc [m/min]	Feed per tooth* fz[mm]	recommended depth of cut ap [mm]
P	unalloyed steel	10	R2	200 - 300	0,4 - 0,8	0,6
		12	R2,5	200 - 300	0,5 - 1	1
		16	R3	200 - 300	0,6 - 1	1,1
		20	R3	200 - 300	0,6 - 1	1,5
	alloyed steel 800 N/mm ²	10	R2	180 - 250	0,3 - 0,7	0,5
		12	R2,5	180 - 250	0,4 - 0,8	0,7
		16	R3	180 - 250	0,4 - 0,8	0,8
		20	R3	180 - 250	0,5 - 0,8	1
	alloyed steel 1100 N/mm ²	10	R2	160 - 220	0,3 - 0,7	0,3
		12	R2,5	160 - 220	0,4 - 0,8	0,4
		16	R3	160 - 220	0,4 - 0,8	0,5
		20	R3	160 - 220	0,5 - 0,8	0,8
M	stainless steel	10	R2	140 - 200	0,3 - 0,6	0,3
		12	R2,5	140 - 220	0,3 - 0,8	0,4
		16	R3	140 - 200	0,5 - 0,8	0,5
		20	R3	140 - 200	0,5 - 0,8	0,8
K	gray cast iron	10	R2	200 - 300	0,4 - 0,8	0,6
		12	R2,5	200 - 300	0,5 - 1	1
		16	R3	200 - 300	0,6 - 1	1,1
	cast alloys	20	R3	200 - 300	0,6 - 1	1,5
		10	R2	160 - 220	0,3 - 0,7	0,5
		12	R2,5	160 - 220	0,4 - 0,8	0,7
		16	R3	160 - 220	0,4 - 0,8	0,8
		20	R3	160 - 200	0,5 - 0,8	1
N	non-ferrous metals	-	-	-	-	-
		-	-	-	-	-
S	super alloys	10	R2	40 - 80	0,2 - 0,5	0,2
		12	R2,5	40 - 80	0,2 - 0,5	0,3
		16	R3	40 - 80	0,3 - 0,6	0,4
		20	R3	40 - 80	0,3 - 0,6	0,6
H	hardened steel < 50 HRC	10	R2	100 - 150	0,2 - 0,4	0,2
		12	R2,5	100 - 150	0,2 - 0,4	0,3
		16	R3	100 - 150	0,3 - 0,5	0,4
		20	R3	100 - 150	0,3 - 0,5	0,6
	hardened steel < 58 HRC	10	R2	50 - 80	0,2 - 0,4	0,2
		12	R2,5	50 - 80	0,2 - 0,4	0,3
		16	R3	50 - 80	0,3 - 0,5	0,4
		20	R3	50 - 80	0,3 - 0,5	0,5

* lower fz-value refers to 3D-machining, higher value to face milling!



Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]		Feed per tooth fz [mm]		recommended depth of cut ap [mm]	recommended cutting width ae [%]
			Roughing copying	Finishing copying	Roughing copying	Finishing copying		
P	unalloyed steel	2 - 6	160 - 220	220 - 300	0,045 - 0,08	0,028 - 0,055	0,1 x D	40%
		8 - 12	140 - 200	220 - 300	0,15 - 0,2	0,1 - 0,15	0,1 x D	40%
		16 - 25	140 - 200	220 - 300	0,2 - 0,3	0,15 - 0,22	0,1 x D	40%
	alloyed steel < 1100 N/mm ²	2 - 6	120 - 180	100 - 180	0,035 - 0,08	0,026 - 0,05	0,08 x D	30%
		8 - 12	120 - 180	180 - 260	0,10 - 0,18	0,08 - 0,13	0,08 x D	30%
		16 - 25	120 - 180	180 - 260	0,17 - 0,25	0,12 - 0,2	0,08 x D	30%
	alloyed steel < 1400 N/mm ²	2 - 6	100 - 160	160 - 240	0,025 - 0,08	0,024 - 0,05	0,06 x D	25%
		8 - 12	100 - 160	160 - 240	0,08 - 0,14	0,07 - 0,11	0,06 x D	25%
		16 - 25	100 - 160	160 - 240	0,14 - 0,20	0,1 - 0,18	0,06 x D	25%
M	stainless steel	2 - 6	80 - 140	100 - 180	0,02 - 0,04	0,02 - 0,04	0,08 x D	30%
		8 - 12	80 - 140	100 - 180	0,075 - 0,1	0,08 - 0,12	0,08 x D	30%
		16 - 25	80 - 140	100 - 180	0,1 - 0,2	0,12 - 0,2	0,08 x D	30%
K	gray cast iron	2 - 6	160 - 230	220 - 300	0,045 - 0,08	0,028 - 0,055	0,08 x D	40%
		8 - 12	160 - 230	220 - 300	0,15 - 0,2	0,1 - 0,15	0,08 x D	40%
		16 - 25	160 - 230	220 - 300	0,2 - 0,3	0,15 - 0,22	0,08 x D	40%
	cast alloys	2 - 6	120 - 200	180 - 260	0,035 - 0,08	0,026 - 0,05	0,08 x D	30%
		8 - 12	120 - 180	180 - 260	0,10 - 0,18	0,08 - 0,13	0,08 x D	30%
		16 - 25	120 - 180	180 - 260	0,17 - 0,25	0,12 - 0,2	0,08 x D	30%
N	aluminum	2 - 6	250 - 800	250 - 800	0,04 - 0,08	0,03 - 0,06	0,4 x D	30%
		8 - 12	800 - 1000	1000 - 1500	0,12 - 0,18	0,1 - 0,18	0,4 x D	30%
		16 - 25	800 - 1000	1500 - 2000	0,2 - 0,3	0,15 - 0,25	0,4 x D	30%
	duroplastics	2 - 6	150 - 300	200 - 400	0,02 - 0,04	0,02 - 0,04	0,1 x D	10%
		8 - 12	150 - 300	200 - 400	0,08 - 0,1	0,08 - 0,1	0,1 x D	10%
		16 - 25	150 - 300	200 - 400	0,1 - 0,2	0,1 - 0,2	0,1 x D	10%
S	super alloys	2 - 6	20 - 50	30 - 70	0,015 - 0,03	0,02 - 0,04	0,1 x D	10%
		8 - 12	20 - 50	30 - 70	0,05 - 0,08	0,08 - 0,12	0,1 x D	10%
		16 - 25	20 - 50	30 - 70	0,08 - 0,15	0,12 - 0,2	0,1 x D	10%
		2 - 6	40 - 70	80 - 200	0,025 - 0,08	0,024 - 0,05	0,06 x D	25%
H	hardened steel 48 - 54 HRC	8 - 12	40 - 70	80 - 200	0,08 - 0,14	0,07 - 0,11	0,06 x D	25%
		16 - 25	40 - 70	80 - 200	0,14 - 0,20	0,1 - 0,18	0,06 x D	25%
		2 - 6	30 - 50	80 - 200	0,02 - 0,06	0,02 - 0,04	0,05 x D	20%
	hardened steel 54 - 63 HRC	8 - 12	30 - 50	80 - 200	0,06 - 0,12	0,06 - 0,08	0,05 x D	20%
		16 - 25	30 - 50	80 - 200	0,12 - 0,18	0,08 - 0,16	0,05 x D	20%
		2 - 6	20 - 50	30 - 110	0,015 - 0,04	0,02 - 0,03	0,05 x D	10%
	hardened steel > 63 HRC	8 - 12	20 - 50	30 - 110	0,04 - 0,1	0,04 - 0,07	0,05 x D	10%
		16 - 25	20 - 50	30 - 110	0,1 - 0,15	0,07 - 0,12	0,05 x D	10%

General information:

Machining of aluminum and duroplastics with grade IN05S, any other materials with IN2005 / IN2006.

Max. cutting depth is determined by radius.

Please consider the limitation of max. speed rate of the machine! Cutting data refer to nmax = 40000 rpm.



Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Feed per tooth fz [mm]	max. Depth of cut ap [mm]	Grade
P	unalloyed steel	8	140 - 200	0,03 - 0,10	5	IN2005
		10	140 - 200	0,04 - 0,11	7	IN2005
		12	140 - 200	0,04 - 0,12	9	IN2005
		16	140 - 200	0,05 - 0,15	12	IN2005
		20	140 - 200	0,05 - 0,18	15	IN2005
	alloyed steel 800 N/mm ²	8	140 - 200	0,03 - 0,10	5	IN2005
		10	140 - 200	0,04 - 0,11	7	IN2005
		12	140 - 200	0,04 - 0,12	9	IN2005
		16	140 - 200	0,05 - 0,15	12	IN2005
		20	140 - 200	0,05 - 0,18	15	IN2005
	alloyed steel 1100 N/mm ²	8	120 - 180	0,03 - 0,08	5	IN2005
		10	120 - 180	0,03 - 0,09	7	IN2005
12		120 - 180	0,04 - 0,10	9	IN2005	
16		120 - 180	0,04 - 0,12	12	IN2005	
20		120 - 180	0,05 - 0,14	15	IN2005	
M	stainless steel	8	60 - 100	0,03 - 0,07	5	IN2005
		10	60 - 100	0,03 - 0,08	7	IN2005
		12	60 - 100	0,04 - 0,10	9	IN2005
		16	60 - 100	0,04 - 0,12	12	IN2005
		20	60 - 100	0,05 - 0,15	15	IN2005
K	gray cast iron cast alloys	8	160 - 220	0,03 - 0,10	5	IN2005
		10	160 - 220	0,03 - 0,11	7	IN2005
		12	160 - 220	0,04 - 0,12	9	IN2005
		16	160 - 220	0,05 - 0,15	12	IN2005
		20	160 - 220	0,05 - 0,18	15	IN2005
N	non-ferrous metals duroplastics	8	250 - 1000	0,05 - 0,10	5	IN055 / IN3005
		10	250 - 1000	0,06 - 0,15	7	IN055 / IN3005
		12	250 - 1000	0,06 - 0,16	9	IN055 / IN3005
		16	250 - 1000	0,08 - 0,20	12	IN055 / IN3005
		20	250 - 1000	0,08 - 0,20	15	IN055 / IN3005
S	super alloys	8	25 - 80	0,03 - 0,08	5	IN2005
		10	25 - 80	0,03 - 0,09	7	IN2005
		12	25 - 80	0,04 - 0,10	9	IN2005
		16	25 - 80	0,04 - 0,12	12	IN2005
		20	25 - 80	0,05 - 0,13	15	IN2005
H	hard machining	-	-	-	-	-
		-	-	-	-	-

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Feed per tooth fz [mm]		Grade		
P	unalloyed steel	8	160 - 220	0,04 - 0,06	0,05 - 0,1	IN2005		
		10	160 - 220	0,05 - 0,07	0,06 - 0,15	IN2005		
		12	160 - 220	0,05 - 0,07	0,06 - 0,15	IN2005		
		14 - 16	160 - 220	0,05 - 0,08	0,08 - 0,2	IN2005		
	alloyed steel < 800 N/mm²	8	140 - 200	0,04 - 0,06	0,05 - 0,1	IN2005		
		10	140 - 200	0,05 - 0,07	0,06 - 0,15	IN2005		
		12	140 - 200	0,05 - 0,07	0,06 - 0,15	IN2005		
		14 - 16	140 - 200	0,05 - 0,08	0,08 - 0,2	IN2005		
	alloyed steel < 1100 N/mm²	8	120 - 180	0,04 - 0,06	0,05 - 0,08	IN2005		
		10	120 - 180	0,05 - 0,07	0,06 - 0,14	IN2005		
		12	120 - 180	0,05 - 0,07	0,06 - 0,14	IN2005		
		14 - 16	120 - 180	0,05 - 0,08	0,08 - 0,18	IN2005		
M	stainless steel	8	80 - 140	0,02 - 0,03	0,05 - 0,08	IN2005		
		10	80 - 140	0,02 - 0,03	0,06 - 0,14	IN2005		
		12	80 - 140	0,03 - 0,04	0,06 - 0,14	IN2005		
		14 - 16	80 - 140	0,04 - 0,06	0,08 - 0,18	IN2005		
K	gray cast iron	8	160 - 240	0,04 - 0,06	0,05 - 0,1	IN2005		
		10	160 - 240	0,05 - 0,07	0,06 - 0,15	IN2005		
		12	160 - 240	0,05 - 0,07	0,06 - 0,15	IN2005		
	cast alloys	14 - 16	160 - 240	0,05 - 0,08	0,08 - 0,2	IN2005		
		8	160 - 220	0,04 - 0,06	0,05 - 0,1	IN2005		
		10	160 - 220	0,05 - 0,07	0,06 - 0,15	IN2005		
		12	160 - 220	0,05 - 0,07	0,06 - 0,15	IN2005		
		14 - 16	160 - 220	0,05 - 0,08	0,08 - 0,2	IN2005		
		N	non-ferrous metals	-	-	-	-	-
				-	-	-	-	-
S	super alloys	8	40 - 80	0,02 - 0,03	0,05 - 0,08	IN2005		
		10	40 - 80	0,02 - 0,03	0,06 - 0,14	IN2005		
		12	40 - 80	0,03 - 0,04	0,06 - 0,14	IN2005		
		14 - 16	40 - 80	0,04 - 0,06	0,08 - 0,18	IN2005		
H	hardened steel < 54 HRC	8	80 - 140	-	0,05 - 0,08	IN2005		
		10	80 - 140	-	0,06 - 0,14	IN2005		
		12	80 - 140	-	0,06 - 0,14	IN2005		
		14 - 16	80 - 140	-	0,08 - 0,18	IN2005		
	hardened steel < 58 HRC	8	60 - 120	-	0,03 - 0,06	IN2005 / IN2006		
		10	60 - 120	-	0,04 - 0,08	IN2005 / IN2006		
		12	60 - 120	-	0,04 - 0,08	IN2005 / IN2006		
		14 - 16	60 - 120	-	0,05 - 0,1	IN2005 / IN2006		





Chamfering cutter:

45N/46N/45M/45P/47N/48N

Concave radius milling cutter: 45R & 18S



Recommended cutting data:

ISO	Material				
		Cutting speed Vc [m/min]	Feed per tooth fz [mm]	Cutting speed Vc [m/min]	Feed per tooth fz [mm]
P	steel	160 - 220	0,05 - 0,08	160 - 220	0,03 - 0,06
	toolsteel	120 - 160	0,05 - 0,08	120 - 160	0,03 - 0,06
M	stainless steel	80 - 120	0,04 - 0,08	80 - 120	0,02 - 0,05
K	casting alloys / gray cast iron	160 - 240	0,08 - 0,15	160 - 240	0,05 - 0,10
N	aluminum	400 - 1200	0,08 - 0,15	400 - 1200	0,06 - 0,12
	copper	200 - 400	0,06 - 0,12	200 - 400	0,04 - 0,08
S	nickel alloys / titanium	30 - 70	0,03 - 0,06	30 - 70	0,02 - 0,05
H	hard machining	-	-	-	-

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Roughing		Finishing		Grade
				Feed per tooth fz [mm]	Depth of cut ap [mm]	Feed per tooth fz [mm]	Depth of cut ap [mm]	
P	unalloyed steel	10	160 - 220	0,1 - 0,2 x R*	0,2 x R*	0,10 - 0,13	0,1 x R*	IN2005
		12	160 - 220	0,1 - 0,2 x R*	0,2 x R*	0,13 - 0,17	0,1 x R*	IN2005
		16	160 - 220	0,1 - 0,2 x R*	0,2 x R*	0,15 - 0,20	0,1 x R*	IN2005
		20	160 - 220	0,1 - 0,2 x R*	0,2 x R*	0,15 - 0,23	0,1 x R*	IN2005
	alloyed steel < 1100 N/mm ²	10	160 - 220	0,1 - 0,2 x R*	0,2 x R*	0,10 - 0,13	0,1 x R*	IN2005
		12	160 - 220	0,1 - 0,2 x R*	0,2 x R*	0,13 - 0,17	0,1 x R*	IN2005
		16	160 - 220	0,1 - 0,2 x R*	0,2 x R*	0,15 - 0,20	0,1 x R*	IN2005
		20	160 - 220	0,1 - 0,2 x R*	0,2 x R*	0,15 - 0,23	0,1 x R*	IN2005
	alloyed steel < 1400 N/mm ²	10	140 - 200	0,1 - 0,2 x R*	0,2 x R*	0,10 - 0,13	0,1 x R*	IN2005 / IN2006
		12	140 - 200	0,1 - 0,2 x R*	0,2 x R*	0,13 - 0,17	0,1 x R*	IN2005 / IN2006
		16	140 - 200	0,1 - 0,2 x R*	0,2 x R*	0,15 - 0,20	0,1 x R*	IN2005 / IN2006
		20	140 - 200	0,1 - 0,2 x R*	0,2 x R*	0,15 - 0,23	0,1 x R*	IN2005 / IN2006
M	stainless steel	10	60 - 120	0,1 - 0,15 x R*	0,2 x R*	0,10 - 0,13	0,1 x R*	IN2005
		12	60 - 120	0,1 - 0,15 x R*	0,2 x R*	0,13 - 0,17	0,1 x R*	IN2005
		16	60 - 120	0,1 - 0,15 x R*	0,2 x R*	0,15 - 0,20	0,1 x R*	IN2005
		20	60 - 120	0,1 - 0,15 x R*	0,2 x R*	0,15 - 0,23	0,1 x R*	IN2005
K	gray cast iron cast alloys	10	160 - 220	0,1 - 0,2 x R*	0,2 x R*	0,10 - 0,13	0,1 x R*	IN2005
		12	160 - 220	0,1 - 0,2 x R*	0,2 x R*	0,13 - 0,17	0,1 x R*	IN2005
		16	160 - 220	0,1 - 0,2 x R*	0,2 x R*	0,15 - 0,20	0,1 x R*	IN2005
N	non-ferrous metals	-	-	-	-	-	-	-
		-	-	-	-	-	-	-
		-	-	-	-	-	-	-
S	super alloys	10	40 - 80	0,1 - 0,15 x R*	0,2 x R*	0,10 - 0,13	0,1 x R*	IN2005
		12	40 - 80	0,1 - 0,15 x R*	0,2 x R*	0,13 - 0,17	0,1 x R*	IN2005
		16	40 - 80	0,1 - 0,15 x R*	0,2 x R*	0,15 - 0,20	0,1 x R*	IN2005
		20	40 - 80	0,1 - 0,15 x R*	0,2 x R*	0,15 - 0,23	0,1 x R*	IN2005
H	hardened steel < 54 HRC	10	80 - 140	0,1 - 0,2 x R*	0,2 x R*	0,10 - 0,13	0,1 x R*	IN2005 / IN2006
		12	80 - 140	0,1 - 0,2 x R*	0,2 x R*	0,13 - 0,17	0,1 x R*	IN2005 / IN2006
		16	80 - 140	0,1 - 0,2 x R*	0,2 x R*	0,15 - 0,20	0,1 x R*	IN2005 / IN2006
		20	80 - 140	0,1 - 0,2 x R*	0,2 x R*	0,15 - 0,23	0,1 x R*	IN2005 / IN2006
	hardened steel < 63 HRC	10	80 - 140	0,1 - 0,15 x R*	0,2 x R*	0,08 - 0,12	0,1 x R*	IN2006
		12	80 - 140	0,1 - 0,15 x R*	0,2 x R*	0,11 - 0,15	0,1 x R*	IN2006
		16	80 - 140	0,1 - 0,15 x R*	0,2 x R*	0,13 - 0,18	0,1 x R*	IN2006
		16	80 - 140	0,1 - 0,15 x R*	0,2 x R*	0,13 - 0,18	0,1 x R*	IN2006

* Corner radius

CHIPSURFER 45Z_ NC CENTER DRILL



Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]	Feed rate fu [mm]	Grade
P	alloyed steel > 1100 N/mm ²	160 - 220	0,08 - 0,2	IN2005
M	stainless steel	60 - 120	0,08 - 0,15	IN2005
K	cast alloys / gray cast iron	160 - 220	0,08 - 0,25	IN2005
N	non-ferrous metals / duroplastics	250 - 1000	0,08 - 0,25	IN2005
S	superlegierung	30 - 60	0,06 - 0,12	IN2005
H	hard machining	-	-	-

CHIPSURFER 45Z_ CENTERING DRILL



Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]	Feed rate fu [mm]	Grade
P	alloyed steel > 1100 N/mm ²	160 - 220	0,06 - 0,12	IN2005
M	stainless steel	60 - 120	0,05 - 0,1	IN2005
K	cast alloys / gray cast iron	160 - 220	0,06 - 0,15	IN2005
N	non-ferrous metals / duroplastics	250 - 1000	0,08 - 0,15	IN2005
S	superlegierung	30 - 60	0,04 - 0,08	IN2005
H	hard machining	-	-	-

CHIPSURFER 45Z_ / 45Q_ ENGRAVING CUTTER



Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]	Feed rate fu [mm]	Grade
P	alloyed steel > 1100 N/mm ²	160 - 220	0,02 - 0,08	IN2005
M	stainless steel	60 - 120	0,02 - 0,06	IN2005
K	cast alloys / gray cast iron	160 - 220	0,02 - 0,1	IN2005
N	non-ferrous metals / duroplastics	250 - 1000	0,02 - 0,1	IN2005
S	superlegierung	30 - 60	0,02 - 0,06	IN2005
H	hard machining	-	-	-

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.

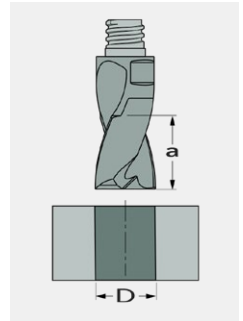


Recommended cutting data when solid drilling up to 1.2xØ on a flat material surface:

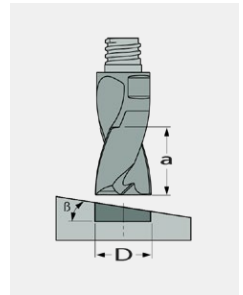
ISO	Material	Cutting speed V_c [m/min]	Feed rate f_u [mm]	Grade
P	unalloyed steel	120 - 220	0,10 - 0,25	IN2005
	alloyed steel 800 N/mm ²	120 - 220	0,10 - 0,25	IN2005
M	alloyed steel 1100 N/mm ²	120 - 220	0,10 - 0,25	IN2005
	stainless steel	60 - 120	0,06 - 0,15	IN2005
K	gray cast iron	160 - 220	0,10 - 0,25	IN2005
	nodular cast iron	160 - 220	0,10 - 0,25	IN2005
N	aluminum	-	-	-
S	high temperature alloys	30 - 60	0,05 - 0,12	IN2005
	titanium alloys	30 - 60	0,05 - 0,12	IN2005
H	hard machining < 54 HRC	-	-	-
	harbearbeitung < 63 HRC	-	-	-

Tips:

- For through holes, reducing the feed rate by 50% before exiting the material is recommended



- For inclined hole entry up to 30° of the workpiece surface, a reduction of the parameters by 30% is recommended and for surfaces up to 45°, a reduction of 50% is recommended.



CHIP SURFER 46D_ / 47C_ / 48C_ ROUGHING CUTTER



Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Feed per tooth fz [mm]	Depth of cut ap max [mm]	Grade	
P	unalloyed steel	8	140 - 200	0,03 - 0,10	5	IN2005	
		10	140 - 200	0,04 - 0,11	7	IN2005	
		12	140 - 200	0,04 - 0,12	9	IN2005	
		16	140 - 200	0,05 - 0,15	12	IN2005	
		20	140 - 200	0,05 - 0,18	15	IN2005	
	25	140 - 200	0,05 - 0,22	22	IN2005		
	alloyed steel < 800 N/mm ²	8	140 - 200	0,03 - 0,10	5	IN2005	
		10	140 - 200	0,04 - 0,11	7	IN2005	
		12	140 - 200	0,04 - 0,12	9	IN2005	
		16	140 - 200	0,05 - 0,15	12	IN2005	
		20	140 - 200	0,05 - 0,18	15	IN2005	
	25	140 - 200	0,05 - 0,22	22	IN2005		
	alloyed steel < 1100 N/mm ²	8	120 - 180	0,02 - 0,08	5	IN2005	
		10	120 - 180	0,02 - 0,09	7	IN2005	
		12	120 - 180	0,03 - 0,10	9	IN2005	
		16	120 - 180	0,04 - 0,12	12	IN2005	
		20	120 - 180	0,05 - 0,14	15	IN2005	
	25	120 - 180	0,05 - 0,16	22	IN2005		
	M	stainless steel	8	60 - 100	0,02 - 0,07	5	IN2005
			10	60 - 100	0,02 - 0,08	7	IN2005
12			60 - 100	0,03 - 0,10	9	IN2005	
16			60 - 100	0,04 - 0,12	12	IN2005	
20			60 - 100	0,05 - 0,15	15	IN2005	
25	60 - 100	0,05 - 0,15	22	IN2005			
K	gray cast iron cast alloys	8	160 - 220	0,02 - 0,09	5	IN2005	
		10	160 - 220	0,02 - 0,10	7	IN2005	
		12	160 - 220	0,03 - 0,11	9	IN2005	
		16	160 - 220	0,04 - 0,13	12	IN2005	
		20	160 - 220	0,05 - 0,17	15	IN2005	
25	160 - 220	0,05 - 0,20	22	IN2005			
N	non-ferrous metals duroplastics	8	250 - 1000	0,05 - 0,10	5	IN055 / IN3005	
		10	250 - 1000	0,06 - 0,15	7	IN055 / IN3005	
		12	250 - 1000	0,06 - 0,16	9	IN055 / IN3005	
		16	250 - 1000	0,08 - 0,20	12	IN055 / IN3005	
		20	250 - 1000	0,08 - 0,20	15	IN055 / IN3005	
S	super alloys	8	25 - 80	0,02 - 0,08	5	IN2005	
		10	25 - 80	0,02 - 0,09	7	IN2005	
		12	25 - 80	0,03 - 0,10	9	IN2005	
		16	25 - 80	0,04 - 0,12	12	IN2005	
		20	25 - 80	0,05 - 0,13	15	IN2005	
25	25 - 80	0,05 - 0,13	22	IN2005			
H	hard machining	-	-	-	-	-	
		-	-	-	-	-	

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.

SOLID CARBIDE 46J_ / 46D_ / 45J_ END MILL

DRILL SLOT DRILL 46J/46D Z = 3
END MILL 45J Z = 2



Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Feed per tooth fz [mm]	Depth of cut ap max [mm]	Grade
P	unalloyed steel	2 - 4	140 - 200	0,03 - 0,10	2 x Ø	IN2005
		5 - 6	140 - 200	0,03 - 0,11	2 x Ø	IN2005
		8 - 9	140 - 200	0,04 - 0,12	2 x Ø	IN2005
		10	140 - 200	0,05 - 0,15	2 x Ø	IN2005
		12 - 14	140 - 200	0,05 - 0,15	1,5 x Ø	IN2005
		16	140 - 200	0,05 - 0,16	1,5 x Ø	IN2005
	alloyed steel < 800 N/mm ²	20	140 - 200	0,05 - 0,18	1,5 x Ø	IN2005
		2 - 4	140 - 200	0,03 - 0,10	2 x Ø	IN2005
		5 - 6	140 - 200	0,03 - 0,11	2 x Ø	IN2005
		8 - 9	140 - 200	0,04 - 0,12	2 x Ø	IN2005
		10	140 - 200	0,05 - 0,15	2 x Ø	IN2005
		12 - 14	140 - 200	0,05 - 0,15	1,5 x Ø	IN2005
	alloyed steel < 1100 N/mm ²	16	140 - 200	0,05 - 0,16	1,5 x Ø	IN2005
		20	140 - 200	0,05 - 0,18	1,5 x Ø	IN2005
		2 - 4	120 - 180	0,03 - 0,08	2 x Ø	IN2005
		5 - 6	120 - 180	0,03 - 0,09	2 x Ø	IN2005
		8 - 9	120 - 180	0,04 - 0,10	2 x Ø	IN2005
		10	120 - 180	0,04 - 0,12	2 x Ø	IN2005
M	stainless steel	12 - 14	120 - 180	0,05 - 0,13	1,5 x Ø	IN2005
		16	120 - 180	0,05 - 0,14	1,5 x Ø	IN2005
		20	120 - 180	0,05 - 0,14	1,5 x Ø	IN2005
		2 - 4	60 - 100	0,02 - 0,05	2 x Ø	IN2005
		5 - 6	60 - 100	0,03 - 0,08	2 x Ø	IN2005
		8 - 9	60 - 100	0,04 - 0,10	2 x Ø	IN2005
K	gray cast iron cast alloys	10	60 - 100	0,04 - 0,12	2 x Ø	IN2005
		12 - 14	60 - 100	0,05 - 0,13	1,5 x Ø	IN2005
		16	60 - 100	0,05 - 0,15	1,5 x Ø	IN2005
		20	60 - 100	0,05 - 0,15	1,5 x Ø	IN2005
		2 - 4	160 - 220	0,03 - 0,10	2 x Ø	IN2005
		5 - 6	160 - 220	0,03 - 0,11	2 x Ø	IN2005
N	non-ferrous metals duroplastics	8 - 9	160 - 220	0,04 - 0,12	2 x Ø	IN2005
		10	160 - 220	0,05 - 0,15	2 x Ø	IN2005
		12 - 14	160 - 220	0,05 - 0,15	1,5 x Ø	IN2005
		16	160 - 220	0,05 - 0,16	1,5 x Ø	IN2005
		20	160 - 220	0,05 - 0,18	1,5 x Ø	IN2005
		4 - 6	250 - 1000	0,03 - 0,08	2 x Ø	IN055 / IN3005
S	super alloys	8	250 - 1000	0,05 - 0,10	2 x Ø	IN055 / IN3005
		10	250 - 1000	0,06 - 0,15	2 x Ø	IN055 / IN3005
		12	250 - 1000	0,06 - 0,16	2 x Ø	IN055 / IN3005
		16	250 - 1000	0,08 - 0,20	2 x Ø	IN055 / IN3005
		20	250 - 1000	0,08 - 0,20	1,5 x Ø	IN055 / IN3005
		2 - 4	25 - 80	0,02 - 0,05	2 x Ø	IN2005
H	hard machining	5 - 6	25 - 80	0,03 - 0,08	2 x Ø	IN2005
		8 - 9	25 - 80	0,04 - 0,10	2 x Ø	IN2005
		10	25 - 80	0,04 - 0,12	2 x Ø	IN2005
		12 - 14	25 - 80	0,05 - 0,13	1,5 x Ø	IN2005
		16	25 - 80	0,05 - 0,15	1,5 x Ø	IN2005
		20	25 - 80	0,05 - 0,15	1,5 x Ø	IN2005
		-	-	-	-	-
		-	-	-	-	-

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



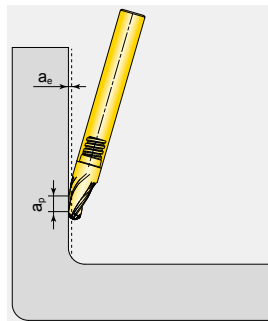
Aluminum geometry



Steel geometry

Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Feed per tooth fz [mm]	Depth of cut / Infeed ap [mm]	finish stockae [mm]	Grade
P	unalloyed steel	8	140 - 200	0,03 - 0,05	0,75 - 1,5	0,035 - 0,15	IN2005
		10	140 - 200	0,03 - 0,07	0,75 - 1,5	0,035 - 0,15	IN2005
		12	140 - 200	0,03 - 0,08	0,75 - 1,5	0,035 - 0,15	IN2005
		16	140 - 200	0,03 - 0,11	0,75 - 1,5	0,05 - 0,2	IN2005
	alloyed steel < 800 N/mm²	8	140 - 200	0,03 - 0,05	0,75 - 1,5	0,035 - 0,15	IN2005
		10	140 - 200	0,03 - 0,07	0,75 - 1,5	0,035 - 0,15	IN2005
		12	140 - 200	0,03 - 0,08	0,75 - 1,5	0,035 - 0,15	IN2005
		16	140 - 200	0,03 - 0,11	0,75 - 1,5	0,05 - 0,2	IN2005
	alloyed steel < 1100 N/mm²	8	120 - 180	0,02 - 0,04	0,75 - 1,5	0,035 - 0,15	IN2005
		10	120 - 180	0,02 - 0,06	0,75 - 1,5	0,035 - 0,15	IN2005
		12	120 - 180	0,03 - 0,07	0,75 - 1,5	0,035 - 0,15	IN2005
		16	120 - 180	0,04 - 0,08	0,75 - 1,5	0,05 - 0,2	IN2005
M	stainless steel	8	60 - 100	0,02 - 0,04	0,75 - 1,5	0,035 - 0,15	IN2005
		10	60 - 100	0,02 - 0,06	0,75 - 1,5	0,035 - 0,15	IN2005
		12	60 - 100	0,03 - 0,07	0,75 - 1,5	0,035 - 0,15	IN2005
		16	60 - 100	0,04 - 0,08	0,75 - 1,5	0,05 - 0,2	IN2005
K	gray cast iron cast alloys	8	160 - 220	0,03 - 0,05	0,75 - 1,5	0,035 - 0,15	IN2005
		10	160 - 220	0,03 - 0,07	0,75 - 1,5	0,035 - 0,15	IN2005
		12	160 - 220	0,03 - 0,08	0,75 - 1,5	0,035 - 0,15	IN2005
		16	160 - 220	0,03 - 0,11	0,75 - 1,5	0,05 - 0,2	IN2005
N	non-ferrous metals duroplastics	8	250 - 1000	0,04 - 0,06	0,75 - 1,5	0,035 - 0,15	IN05S
		10	250 - 1000	0,04 - 0,08	0,75 - 1,5	0,035 - 0,15	IN05S
		12	250 - 1000	0,04 - 0,09	0,75 - 1,5	0,035 - 0,15	IN05S
		16	250 - 1000	0,04 - 0,12	0,75 - 1,5	0,05 - 0,2	IN05S
S	super alloys	8	25 - 80	0,02 - 0,04	0,75 - 1,5	0,035 - 0,15	IN2005
		10	25 - 80	0,02 - 0,06	0,75 - 1,5	0,035 - 0,15	IN2005
		12	25 - 80	0,03 - 0,07	0,75 - 1,5	0,035 - 0,15	IN2005
		16	25 - 80	0,04 - 0,08	0,75 - 1,5	0,05 - 0,2	IN2005
H	hard machining	-	-	-	-	-	-
		-	-	-	-	-	-



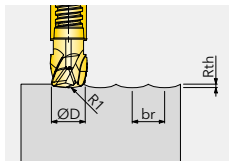
Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Feed per tooth fz [mm]	Depth of cut / Infeed ap [mm]	Grade
P	unalloyed steel	8	220 - 280	0,05 - 0,1	0,05 - 0,1	IN2005
		10	220 - 280	0,05 - 0,1	0,05 - 0,1	IN2005
		12	220 - 280	0,05 - 0,12	0,07 - 0,15	IN2005
		16	220 - 280	0,05 - 0,15	0,1 - 0,3	IN2005
		20	220 - 280	0,05 - 0,15	0,1 - 0,4	IN2005
	alloyed steel < 800 N/mm ²	8	180 - 220	0,04 - 0,08	0,05 - 0,1	IN2005
		10	180 - 220	0,04 - 0,08	0,05 - 0,1	IN2005
		12	180 - 220	0,04 - 0,1	0,07 - 0,15	IN2005
		16	180 - 220	0,04 - 0,12	0,1 - 0,3	IN2005
		20	180 - 220	0,04 - 0,12	0,1 - 0,4	IN2005
	alloyed steel < 1100 N/mm ²	8	160 - 200	0,03 - 0,06	0,05 - 0,1	IN2005
		10	160 - 200	0,03 - 0,06	0,05 - 0,1	IN2005
12		160 - 200	0,03 - 0,08	0,07 - 0,15	IN2005	
16		160 - 200	0,03 - 0,1	0,1 - 0,3	IN2005	
20		160 - 200	0,03 - 0,1	0,1 - 0,4	IN2005	
M	stainless steel	8	60 - 140	0,03 - 0,06	0,05 - 0,1	IN2005
		10	60 - 140	0,03 - 0,06	0,05 - 0,1	IN2005
		12	60 - 140	0,03 - 0,08	0,07 - 0,15	IN2005
		16	60 - 140	0,03 - 0,1	0,1 - 0,3	IN2005
		20	60 - 140	0,03 - 0,1	0,1 - 0,4	IN2005
K	gray cast iron cast alloys	8	180 - 260	0,04 - 0,08	0,05 - 0,1	IN2005
		10	180 - 260	0,04 - 0,08	0,05 - 0,1	IN2005
		12	180 - 260	0,04 - 0,1	0,07 - 0,15	IN2005
		16	180 - 260	0,04 - 0,12	0,1 - 0,3	IN2005
		20	180 - 260	0,04 - 0,12	0,1 - 0,4	IN2005
N	non-ferrous metals	-	-	-	-	-
		-	-	-	-	-
S	super alloys	8	40 - 90	0,03 - 0,06	0,05 - 0,1	IN2005
		10	40 - 90	0,03 - 0,06	0,05 - 0,1	IN2005
		12	40 - 90	0,03 - 0,08	0,07 - 0,15	IN2005
		16	40 - 90	0,03 - 0,1	0,1 - 0,3	IN2005
		20	40 - 90	0,03 - 0,1	0,1 - 0,4	IN2005
H	hard machining	-	-	-	-	-
		-	-	-	-	-

Calculation of path distance br



Calculation of path distance br at the desired roughness depth Rth 2,5 µ	
Rth:	0,0025 mm
Ø / R:	Ø 8 / R= 4
Calculation formula:	$br = 2x\sqrt{Rth ((DR1 x 2) \cdot Rth)}$
br:	0,55mm

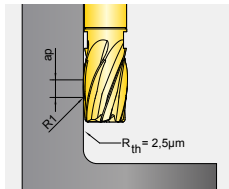
Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Feed per tooth fz [mm]	Depth of cut/Infeed ap [mm]	Cutting width/infeed ae [mm]
P	unalloyed steel	12	140 - 200	0,05 - 0,10	0,8 - 1,5	0,2 - 0,4
		16	140 - 200	0,05 - 0,12	1,0 - 2,0	0,2 - 0,5
	alloyed steel < 800 N/mm ²	12	140 - 200	0,05 - 0,10	0,8 - 1,5	0,2 - 0,4
		16	140 - 200	0,05 - 0,12	1,0 - 2,0	0,2 - 0,5
	alloyed steel < 1100 N/mm ²	12	120 - 180	0,04 - 0,08	0,8 - 1,5	0,2 - 0,4
		16	120 - 180	0,05 - 0,10	1,0 - 2,0	0,2 - 0,5
M	stainless steel	12	60 - 100	0,04 - 0,08	0,8 - 1,5	0,2 - 0,4
16		60 - 100	0,05 - 0,10	1,0 - 2,0	0,2 - 0,5	
K	gray cast iron cast alloys	12	160 - 220	0,05 - 0,10	0,8 - 1,5	0,2 - 0,4
		16	160 - 220	0,05 - 0,12	1,0 - 2,0	0,2 - 0,5
N	non-ferrous metals	-	-	-	-	-
S	super alloys	12	25 - 80	0,03 - 0,06	0,8 - 1,5	0,2 - 0,4
		16	25 - 80	0,03 - 0,08	1,0 - 2,0	0,2 - 0,5
H	hard machining	-	-	-	-	-
		-	-	-	-	-

Calculation of path distance ap



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

Note

A large grid of graph paper for taking notes, consisting of 20 columns and 30 rows of small squares. The grid is mostly empty, with a few faint lines visible. To the right of the grid is a vertical blue bar.

Rough mills for following applications:
Roughing from the solid, machining of residual material and face milling on HSC-machines



Recommended cutting data:

ISO	Material	Dc [mm]	Programming radius	Cutting speed Vc [m/min]	Feed per tooth fz[mm]	recommended depth of cut ap [mm]	
P	unalloyed steel	6	R1	200 – 300	0,3	0,3	
		8	R1,3 / R1,6	200 – 300	0,4	0,4	
		10	R1 / R1,6 / R2	200 – 300	0,5	0,5	
		12	R1,2 / R2 / R2,1	200 – 300	0,5	0,6	
		16	R2 / R2,6 / R3,2	200 – 300	0,6	0,8	
		20	R2,2 / R3,2 / R4	200 – 300	0,7	1	
		25	R3,6	200 – 300	0,7	1,2	
	alloyed steel 800 N/mm ²	6	R1	180 – 250	0,3	0,3	
		8	R1,3 / R1,6	180 – 250	0,4	0,4	
		10	R1 / R1,6 / R2	180 – 250	0,5	0,5	
		12	R1,2 / R2 / R2,1	180 – 250	0,5	0,6	
		16	R2 / R2,6 / R3,2	180 – 250	0,6	0,8	
		20	R2,2 / R3,2 / R4	180 – 250	0,7	1	
		25	R3,6	180 – 250	0,7	1,2	
	alloyed steel 1100 N/mm ²	6	R1	160 – 220	0,3	0,2	
		8	R1,3 / R1,6	160 – 220	0,4	0,3	
		10	R1 / R1,6 / R2	160 – 220	0,5	0,4	
		12	R1,2 / R2 / 2,1	160 – 220	0,5	0,5	
		16	R2 / R2,6 / R3,2	160 – 220	0,6	0,6	
		20	R2,2 / R3,2 / R4	160 – 220	0,7	0,8	
		25	R3,6	160 – 220	0,7	1,2	
	M	stainless steel	6	R1	140 – 200	0,3	0,2
			8	R1,3 / R1,6	140 – 200	0,4	0,3
			10	R1 / R1,6 / R2	140 – 200	0,5	0,4
			12	R1,2 / R2 / R2,1	140 – 200	0,5	0,5
16			R2 / R2,6 / R3,2	140 – 200	0,6	0,6	
20			R2,2 / R3,2 / R4	140 – 200	0,7	0,8	
25			R3,6	140 – 200	0,8	1	
K	gray cast iron	6	R1	200 – 300	0,3	0,3	
		8	R1,3 / R1,6	200 – 300	0,4	0,4	
		10	R1 / R1,6 / R2	200 – 300	0,5	0,5	
		12	R1,2 / R2 / R2,1	200 – 300	0,5	0,6	
		16	R2 / R2,6 / R3,2	200 – 300	0,6	0,8	
		20	R2,2 / R3,2 / R4	200 – 300	0,7	1	
		25	R3,6	200 – 300	0,7	1	
	cast alloys	6	R1	160 – 220	0,3	0,2	
		8	R1,3 / R1,6	160 – 220	0,4	0,3	
		10	R1 / R1,6 / R2	160 – 220	0,5	0,4	
		12	R1,2 / R2 / R2,1	160 – 220	0,5	0,5	
		16	R2 / R2,6 / R3,2	160 – 220	0,6	0,6	
		20	R2,2 / R3,2 / R4	160 – 220	0,7	0,8	
		25	R3,6	160 – 220	0,7	1	

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.

Rough mills for following applications:
Roughing from the solid, machining of residual material and face milling on HSC-machines



Recommended cutting data:

ISO	Material	Dc [mm]	Programming radius	Cutting speed Vc [m/min]	Feed per tooth fz [mm]	recommended depth of cut ap [mm]
N	non-ferrous metals	-	-	-	-	-
		-	-	-	-	-
S	super alloys	6	R1	40 - 80	0,2	0,1
		8	R1,3 / R1,6	40 - 80	0,2	0,2
		10	R1 / R1,6 / R2	40 - 80	0,3	0,3
		12	R1,2 / R2 / R2,1	40 - 80	0,3	0,3
		16	R2 / R2,6 / R3,2	40 - 80	0,4	0,5
		20	R2,2 / R3,2 / R4	40 - 80	0,4	0,5
		25	R3,6	40 - 80	0,4	0,5
H	hardened steel < 50 HRC	6	R1	100 - 150	0,3	0,1
		8	R1,3 / R1,6	100 - 150	0,3	0,2
		10	R1 / R1,6 / R2	100 - 150	0,4	0,3
		12	R1,2 / R2 / R2,1	100 - 150	0,4	0,3
		16	R2 / R2,6 / R3,2	100 - 150	0,5	0,5
		20	R2,2 / R3,2 / R4	100 - 150	0,5	0,5
		25	R3,6	100 - 150	0,5	0,5
	hardened steel < 58 HRC	6	R1	50 - 80	0,2	0,1
		8	R1,3 / R1,6	50 - 80	0,2	0,2
		10	R1 / R1,6 / R2	50 - 80	0,3	0,2
		12	R1,2 / R2 / R2,1	50 - 80	0,3	0,3
		16	R2 / R2,6 / R3,2	50 - 80	0,4	0,4
		20	R2,2 / R3,2 / R4	50 - 80	0,4	0,4
		25	R3,6	50 - 80	0,4	0,4

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Feed per tooth fz [mm]	Depth of cut ap max [mm]	Grade	
P	unalloyed steel	8	140 - 200	0,02 - 0,09	5	IN2005	
		10	140 - 200	0,02 - 0,10	7	IN2005	
		12	140 - 200	0,03 - 0,11	9	IN2005	
		16	140 - 200	0,04 - 0,13	12	IN2005	
		20	140 - 200	0,05 - 0,17	15	IN2005	
	25	140 - 200	0,05 - 0,20	22	IN2005		
	alloyed steel < 800 N/mm ²	8	140 - 200	0,02 - 0,09	5	IN2005	
		10	140 - 200	0,02 - 0,10	7	IN2005	
		12	140 - 200	0,03 - 0,11	9	IN2005	
		16	140 - 200	0,04 - 0,13	12	IN2005	
		20	140 - 200	0,05 - 0,17	15	IN2005	
	25	140 - 200	0,05 - 0,20	22	IN2005		
	alloyed steel < 1100 N/mm ²	8	120 - 180	0,02 - 0,07	5	IN2005	
		10	120 - 180	0,02 - 0,08	7	IN2005	
		12	120 - 180	0,03 - 0,09	9	IN2005	
		16	120 - 180	0,04 - 0,11	12	IN2005	
		20	120 - 180	0,05 - 0,13	15	IN2005	
	25	120 - 180	0,05 - 0,15	22	IN2005		
	M	stainless steel	8	60 - 100	0,02 - 0,07	5	IN2005
			10	60 - 100	0,02 - 0,08	7	IN2005
12			60 - 100	0,03 - 0,10	9	IN2005	
16			60 - 100	0,04 - 0,12	12	IN2005	
20			60 - 100	0,05 - 0,15	15	IN2005	
25	60 - 100	0,05 - 0,15	22	IN2005			
K	gray cast iron cast alloys	8	160 - 220	0,02 - 0,09	5	IN2005	
		10	160 - 220	0,02 - 0,10	7	IN2005	
		12	160 - 220	0,03 - 0,11	9	IN2005	
		16	160 - 220	0,04 - 0,13	12	IN2005	
		20	160 - 220	0,05 - 0,17	15	IN2005	
25	160 - 220	0,05 - 0,20	22	IN2005			
N	non-ferrous metals	-	-	-	-	-	
S	super alloys	8	25 - 80	0,02 - 0,08	5	IN1005 / IN2005	
		10	25 - 80	0,02 - 0,09	7	IN1005 / IN2005	
		12	25 - 80	0,03 - 0,10	9	IN1005 / IN2005	
		16	25 - 80	0,04 - 0,12	12	IN1005 / IN2005	
		20	25 - 80	0,05 - 0,13	15	IN1005 / IN2005	
25	25 - 80	0,05 - 0,13	22	IN1005 / IN2005			
H	hard machining	-	-	-	-	-	
		-	-	-	-	-	

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.

z = 4



Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Shoulder milling			Full slot milling	
				Feed per tooth fz [mm]	Depth of cut ap max [mm]	Cutting width ae max [mm]	Feed per tooth fz [mm]	Depth of cut ap max [mm]
P	unalloyed steel	8	140 - 200	0,03 - 0,05	12	0,4 - 1,6	0,02 - 0,04	8
		10	140 - 200	0,03 - 0,06	15	0,5 - 2,0	0,02 - 0,05	10
		12	140 - 200	0,05 - 0,08	18	0,6 - 2,4	0,03 - 0,06	12
		16	140 - 200	0,05 - 0,10	24	0,8 - 3,2	0,03 - 0,06	16
		20	140 - 200	0,06 - 0,12	30	1,0 - 4,0	0,04 - 0,08	20
	alloyed steel < 800 N/mm ²	25	140 - 200	0,08 - 0,15	37	1,0 - 5,0	0,04 - 0,10	25
		8	140 - 200	0,03 - 0,05	12	0,4 - 1,6	0,02 - 0,04	8
		10	140 - 200	0,03 - 0,06	15	0,5 - 2,0	0,02 - 0,05	10
		12	140 - 200	0,05 - 0,08	18	0,6 - 2,4	0,03 - 0,06	12
		16	140 - 200	0,05 - 0,10	24	0,8 - 3,2	0,03 - 0,06	16
	alloyed steel < 1100 N/mm ²	20	140 - 200	0,06 - 0,12	30	1,0 - 4,0	0,04 - 0,08	20
		25	140 - 200	0,08 - 0,15	37	1,0 - 5,0	0,04 - 0,10	25
		8	120 - 180	0,03 - 0,05	12	0,4 - 1,6	0,02 - 0,03	8
		10	120 - 180	0,03 - 0,06	15	0,5 - 2,0	0,02 - 0,04	10
		12	120 - 180	0,05 - 0,08	18	0,6 - 2,4	0,03 - 0,05	12
M	stainless steel	16	120 - 180	0,05 - 0,10	24	0,8 - 3,2	0,03 - 0,05	16
		20	120 - 180	0,06 - 0,12	30	1,0 - 4,0	0,04 - 0,06	20
		25	120 - 180	0,08 - 0,15	37	1,0 - 5,0	0,04 - 0,08	25
		8	60 - 100	0,02 - 0,08	12	0,4 - 1,6	0,02 - 0,03	8
		10	60 - 100	0,03 - 0,10	15	0,5 - 2,0	0,02 - 0,04	10
		12	60 - 100	0,04 - 0,12	18	0,6 - 2,4	0,03 - 0,05	12
K	gray cast iron cast alloys	16	60 - 100	0,05 - 0,15	24	0,8 - 3,2	0,03 - 0,05	16
		20	60 - 100	0,05 - 0,15	30	1,0 - 4,0	0,04 - 0,06	20
		25	60 - 100	0,05 - 0,15	37	1,0 - 5,0	0,04 - 0,08	25
		8	160 - 220	0,03 - 0,05	12	0,4 - 1,6	0,02 - 0,04	8
		10	160 - 220	0,03 - 0,05	15	0,5 - 2,0	0,02 - 0,05	10
		12	160 - 220	0,03 - 0,06	18	0,6 - 2,4	0,03 - 0,06	12
N	non-ferrous metals	16	160 - 200	0,05 - 0,08	24	0,8 - 3,2	0,03 - 0,06	16
		20	160 - 220	0,05 - 0,10	30	1,0 - 4,0	0,04 - 0,08	20
		25	160 - 220	0,06 - 0,12	37	1,0 - 5,0	0,04 - 0,10	25
		-	-	-	-	-	-	-
		-	-	-	-	-	-	-
		-	-	-	-	-	-	-
S	super alloys	8	25 - 80	0,02 - 0,08	12	0,4 - 1,6	0,02 - 0,03	8
		10	25 - 80	0,03 - 0,10	15	0,5 - 2,0	0,02 - 0,04	10
		12	25 - 80	0,04 - 0,12	18	0,6 - 2,4	0,03 - 0,05	12
		16	25 - 80	0,05 - 0,15	24	0,8 - 3,2	0,03 - 0,05	16
		20	25 - 80	0,05 - 0,15	30	1,0 - 4,0	0,04 - 0,06	20
		25	25 - 80	0,05 - 0,15	37	1,0 - 5,0	0,04 - 0,08	25
H	hard machining	-	-	-	-	-	-	-
		-	-	-	-	-	-	-

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.

z = 7/9



Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Shoulder milling		
				Feed per tooth fz [mm]	Depth of cut ap max [mm]	Cutting width ae max [mm]
P	unalloyed steel	8	160 - 220	0,03 - 0,05	12	0,8
		10	160 - 220	0,03 - 0,06	15	1,0
		12	160 - 220	0,05 - 0,08	18	1,2
		16	160 - 220	0,05 - 0,10	24	1,6
		20	160 - 220	0,06 - 0,12	30	2,0
	alloyed steel < 800 N/mm ²	25	160 - 220	0,08 - 0,15	37	2,5
		8	160 - 220	0,03 - 0,05	12	0,8
		10	160 - 220	0,03 - 0,06	15	1,0
		12	160 - 220	0,05 - 0,08	18	1,2
		16	160 - 220	0,05 - 0,10	24	1,6
	alloyed steel < 1100 N/mm ²	20	160 - 220	0,06 - 0,12	30	2,0
		25	160 - 220	0,08 - 0,15	37	2,5
		8	140 - 200	0,02 - 0,04	12	0,8
		10	140 - 200	0,03 - 0,05	15	1,0
		12	140 - 200	0,04 - 0,06	18	1,2
M	stainless steel	16	140 - 200	0,04 - 0,08	24	1,6
		20	140 - 200	0,05 - 0,10	30	2,0
		25	140 - 200	0,06 - 0,12	37	2,5
		8	60 - 120	0,02 - 0,04	12	0,8
		10	60 - 120	0,03 - 0,05	15	1,0
		12	60 - 120	0,04 - 0,06	18	1,2
K	gray cast iron cast alloys	16	60 - 120	0,04 - 0,08	24	1,6
		20	60 - 120	0,05 - 0,10	30	2,0
		25	60 - 120	0,06 - 0,12	37	2,5
		8	160 - 220	0,03 - 0,05	12	0,8
		10	160 - 220	0,03 - 0,06	15	1,0
N	non-ferrous metals	12	160 - 220	0,05 - 0,08	18	1,2
		16	160 - 220	0,05 - 0,10	24	1,6
S	super alloys	20	160 - 220	0,06 - 0,12	30	2,0
		25	160 - 220	0,08 - 0,15	37	2,5
		8	40 - 80	0,02 - 0,04	12	0,8
		10	40 - 80	0,03 - 0,05	15	1,0
		12	40 - 80	0,04 - 0,06	18	1,2
		16	40 - 80	0,04 - 0,08	24	1,6
H	hard machining	20	40 - 80	0,05 - 0,10	30	2,0
		25	40 - 80	0,06 - 0,12	37	2,5
		-	-	-	-	-

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.

z = 4/5/6



Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Shoulder milling			Full slot milling	
				Feed per tooth fz [mm]	Depth of cut ap max [mm]	Cutting width ae max [mm]	Feed per tooth fz [mm]	Depth of cut ap max [mm]
P	unalloyed steel	8	140 - 200	0,03 - 0,06	12	0,4 - 2,4	0,02 - 0,04	8
		10	140 - 200	0,03 - 0,06	15	0,5 - 3,0	0,02 - 0,05	10
		12	140 - 200	0,05 - 0,08	18	0,6 - 3,6	0,03 - 0,06	12
		16	140 - 200	0,05 - 0,12	24	0,8 - 4,8	0,03 - 0,06	16
		20	140 - 200	0,06 - 0,14	30	1,0 - 6,0	0,04 - 0,08	20
	alloyed steel < 800 N/mm²	25	140 - 200	0,08 - 0,18	37	1,0 - 7,5	0,04 - 0,10	25
		8	140 - 200	0,03 - 0,06	12	0,4 - 2,4	0,02 - 0,04	8
		10	140 - 200	0,03 - 0,06	15	0,5 - 3,0	0,02 - 0,05	10
		12	140 - 200	0,05 - 0,08	18	0,6 - 3,6	0,03 - 0,06	12
		16	140 - 200	0,05 - 0,12	24	0,8 - 4,8	0,03 - 0,06	16
	alloyed steel < 1100 N/mm²	20	140 - 200	0,06 - 0,14	30	1,0 - 6,0	0,04 - 0,08	20
		25	140 - 200	0,08 - 0,18	37	1,0 - 7,5	0,04 - 0,10	25
		8	120 - 180	0,03 - 0,05	12	0,4 - 2,4	0,02 - 0,03	8
		10	120 - 180	0,03 - 0,05	15	0,5 - 3,0	0,02 - 0,04	10
		12	120 - 180	0,05 - 0,08	18	0,6 - 3,6	0,03 - 0,05	12
M	stainless steel	16	120 - 180	0,05 - 0,10	24	0,8 - 4,8	0,03 - 0,05	16
		20	120 - 180	0,06 - 0,12	30	1,0 - 6,0	0,04 - 0,06	20
		25	120 - 180	0,08 - 0,15	37	1,0 - 7,5	0,04 - 0,06	25
		8	60 - 100	0,02 - 0,08	12	0,4 - 2,4	0,02 - 0,03	8
		10	60 - 100	0,03 - 0,10	15	0,5 - 3,0	0,02 - 0,04	10
K	gray cast iron cast alloys	12	60 - 100	0,04 - 0,12	18	0,6 - 3,6	0,03 - 0,05	12
		16	60 - 100	0,05 - 0,15	24	0,8 - 4,8	0,03 - 0,05	16
		20	60 - 100	0,05 - 0,15	30	1,0 - 6,0	0,04 - 0,06	20
		25	60 - 100	0,05 - 0,15	37	1,0 - 7,5	0,04 - 0,06	25
		8	160 - 220	0,03 - 0,06	12	0,4 - 2,4	0,02 - 0,04	8
N	non-ferrous metals	10	160 - 220	0,03 - 0,06	15	0,5 - 3,0	0,02 - 0,05	10
		12	160 - 220	0,05 - 0,08	18	0,6 - 3,6	0,03 - 0,06	12
		16	160 - 200	0,05 - 0,12	24	0,8 - 4,8	0,03 - 0,06	16
		20	160 - 220	0,06 - 0,14	30	1,0 - 6,0	0,04 - 0,08	20
		25	160 - 220	0,08 - 0,18	37	1,0 - 7,5	0,04 - 0,10	25
S	super alloys	-	-	-	-	-	-	-
		8	25 - 80	0,02 - 0,08	12	0,4 - 2,4	0,02 - 0,03	8
		10	25 - 80	0,03 - 0,10	15	0,5 - 3,0	0,02 - 0,04	10
		12	25 - 80	0,04 - 0,12	18	0,6 - 3,6	0,03 - 0,05	12
		16	25 - 80	0,05 - 0,15	24	0,8 - 4,8	0,03 - 0,05	16
		20	25 - 80	0,05 - 0,15	30	1,0 - 6,0	0,04 - 0,06	20
H	hard machining	25	25 - 80	0,05 - 0,15	37	1,0 - 7,5	0,04 - 0,08	25
		-	-	-	-	-	-	-

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.

Z = 3/4/5/7



Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Feed per tooth fz [mm]	Depth of cut ap max [mm]	Grade
P	unalloyed steel	5	140 - 200	0,03 - 0,10	2 x Ø	IN2005
		6	140 - 200	0,035 - 0,10	2 x Ø	IN2005
		8	140 - 200	0,04 - 0,11	2 x Ø	IN2005
		10	140 - 200	0,04 - 0,12	2 x Ø	IN2005
		12	140 - 200	0,05 - 0,15	2 x Ø	IN2005
		16	140 - 200	0,05 - 0,18	2 x Ø	IN2005
		20	140 - 200	0,05 - 0,22	2 x Ø	IN2005
	25	140 - 200	0,05 - 0,22	2 x Ø	IN2005	
	alloyed steel < 800 N/mm ²	5	140 - 200	0,03 - 0,10	2 x Ø	IN2005
		6	140 - 200	0,035 - 0,10	2 x Ø	IN2005
		8	140 - 200	0,04 - 0,11	2 x Ø	IN2005
		10	140 - 200	0,04 - 0,12	2 x Ø	IN2005
		12	140 - 200	0,05 - 0,15	2 x Ø	IN2005
		16	140 - 200	0,05 - 0,18	2 x Ø	IN2005
		20	140 - 200	0,05 - 0,22	2 x Ø	IN2005
	25	140 - 200	0,05 - 0,22	2 x Ø	IN2005	
	alloyed steel < 1100 N/mm ²	5	120 - 180	0,02 - 0,08	2 x Ø	IN2005
		6	120 - 180	0,02 - 0,08	2 x Ø	IN2005
		8	120 - 180	0,02 - 0,09	2 x Ø	IN2005
		10	120 - 180	0,03 - 0,10	2 x Ø	IN2005
		12	120 - 180	0,04 - 0,12	2 x Ø	IN2005
		16	120 - 180	0,05 - 0,14	2 x Ø	IN2005
		20	120 - 180	0,05 - 0,16	2 x Ø	IN2005
	25	120 - 180	0,05 - 0,16	2 x Ø	IN2005	
	M	stainless steel	5	60 - 100	0,02 - 0,07	1,5 x Ø
6			60 - 100	0,02 - 0,07	1,5 x Ø	IN2005
8			60 - 100	0,02 - 0,08	1,5 x Ø	IN2005
10			60 - 100	0,03 - 0,10	1,5 x Ø	IN2005
12			60 - 100	0,04 - 0,12	1,5 x Ø	IN2005
16			60 - 100	0,05 - 0,15	1,5 x Ø	IN2005
20			60 - 100	0,05 - 0,15	1,5 x Ø	IN2005
25			60 - 100	0,05 - 0,15	1,5 x Ø	IN2005

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.

Z = 3/4/5/7



ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Feed per tooth fz [mm]	Depth of cut ap max [mm]	Grade
K	gray cast iron cast alloys	5	160 - 220	0,03 - 0,10	2 x Ø	IN2005
		6	160 - 220	0,035 - 0,10	2 x Ø	IN2005
		8	160 - 220	0,04 - 0,11	2 x Ø	IN2005
		10	160 - 220	0,04 - 0,12	2 x Ø	IN2005
		12	160 - 220	0,05 - 0,15	2 x Ø	IN2005
		16	160 - 220	0,05 - 0,18	2 x Ø	IN2005
		20	160 - 220	0,05 - 0,22	2 x Ø	IN2005
N	non-ferrous metals duroplastics	25	160 - 220	0,05 - 0,22	2 x Ø	IN2005
		6	250 - 1000	0,05 - 0,08	1,5 x Ø	IN055 / IN3005
		8	250 - 1000	0,05 - 0,10	1,5 x Ø	IN055 / IN3005
		10	250 - 1000	0,06 - 0,15	1,2 x Ø	IN055 / IN3005
		12	250 - 1000	0,06 - 0,16	1,0 x Ø	IN055 / IN3005
		16	250 - 1000	0,08 - 0,20	1,0 x Ø	IN055 / IN3005
S	super alloys	20	250 - 1000	0,08 - 0,20	1,0 x Ø	IN055 / IN3005
		5	25 - 80	0,02 - 0,07	1,5 x Ø	IN2005
		6	25 - 80	0,02 - 0,07	1,5 x Ø	IN2005
		8	25 - 80	0,02 - 0,08	1,5 x Ø	IN2005
		10	25 - 80	0,03 - 0,10	1,5 x Ø	IN2005
		12	25 - 80	0,04 - 0,12	1,5 x Ø	IN2005
		16	25 - 80	0,05 - 0,15	1,5 x Ø	IN2005
		20	25 - 80	0,05 - 0,15	1,5 x Ø	IN2005
H	hard machining	25	25 - 80	0,05 - 0,15	1,5 x Ø	IN2005
		-	-	-	-	-
		-	-	-	-	-

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]			Feed per tooth fz [mm]			Depth of cut / Infeed ap max [mm]	hm [mm]
			bei 5% ae	bei 10% ae	bei 20% ae	bei 5% ae	bei 10% ae	bei 20% ae		
P	unalloyed steel	6	260-400	240-360	180-260	0,07 - 0,09	0,04 - 0,06	0,03 - 0,045	18	0,02
		8	260-400	240-360	180-260	0,09 - 0,11	0,06 - 0,08	0,045 - 0,055	24	0,022
		10	260-400	240-360	180-260	0,12 - 0,14	0,08 - 0,1	0,05 - 0,07	30	0,024
		12	260-400	240-360	180-260	0,2 - 0,2	0,14 - 0,16	0,09 - 0,11	36	0,035
		16	260-400	240-360	180-260	0,26 - 0,31	0,15 - 0,2	0,11 - 0,15	48	0,042
		20	260-400	240-360	180-260	0,36 - 0,41	0,22 - 0,28	0,15 - 0,2	60	0,05
	alloyed steel < 800 N/mm ²	6	260-400	220-340	180-240	0,07 - 0,09	0,04 - 0,06	0,03 - 0,045	18	0,02
		8	260-400	220-340	180-240	0,09 - 0,11	0,06 - 0,08	0,045 - 0,055	24	0,022
		10	260-400	220-340	180-240	0,12 - 0,14	0,08 - 0,1	0,05 - 0,07	30	0,024
		12	260-400	220-340	180-240	0,2 - 0,2	0,14 - 0,16	0,09 - 0,11	36	0,035
		16	260-400	220-340	180-240	0,26 - 0,31	0,15 - 0,2	0,11 - 0,15	48	0,045
		20	260-400	220-340	180-240	0,36 - 0,41	0,22 - 0,28	0,15 - 0,2	60	0,055
	alloyed steel < 1100 N/mm ²	6	220-360	200-320	140-200	0,07 - 0,09	0,04 - 0,06	0,03 - 0,045	18	0,02
		8	220-360	200-320	140-200	0,09 - 0,11	0,06 - 0,08	0,045 - 0,055	24	0,022
		10	220-360	200-320	140-200	0,12 - 0,14	0,08 - 0,1	0,05 - 0,07	30	0,024
		12	220-360	200-320	140-200	0,2 - 0,2	0,14 - 0,16	0,09 - 0,11	36	0,035
		16	220-360	200-320	140-200	0,26 - 0,31	0,15 - 0,2	0,11 - 0,15	48	0,05
		20	220-360	200-320	140-200	0,36 - 0,41	0,22 - 0,28	0,15 - 0,2	60	0,06
M	stainless steel	6	130 - 260	120 - 240	80 - 160	0,04 - 0,07	0,03 - 0,05	0,02 - 0,035	18	0,016
		8	130 - 260	120 - 240	80 - 160	0,07 - 0,09	0,04 - 0,07	0,03 - 0,045	24	0,018
		10	130 - 260	120 - 240	80 - 160	0,09 - 0,11	0,05 - 0,08	0,03 - 0,06	30	0,02
		12	130 - 260	120 - 240	80 - 160	0,13 - 0,16	0,08 - 0,11	0,05 - 0,08	36	0,025
		16	130 - 260	120 - 240	80 - 160	0,2 - 0,25	0,13 - 0,18	0,08 - 0,13	48	0,035
		20	130 - 260	120 - 240	80 - 160	0,3 - 0,36	0,2 - 0,25	0,13 - 0,18	60	0,045
K	gray cast iron cast alloys	6	320 - 460	280 - 420	220 - 300	0,07 - 0,09	0,04 - 0,06	0,03 - 0,045	18	0,02
		8	320 - 460	280 - 420	220 - 300	0,09 - 0,11	0,06 - 0,08	0,045 - 0,055	24	0,022
		10	320 - 460	280 - 420	220 - 300	0,12 - 0,14	0,08 - 0,1	0,05 - 0,07	30	0,024
		12	320 - 460	280 - 420	220 - 300	0,2 - 0,2	0,14 - 0,16	0,09 - 0,11	36	0,035
		16	320 - 460	280 - 420	220 - 300	0,26 - 0,31	0,15 - 0,2	0,11 - 0,15	48	0,045
		20	320 - 460	280 - 420	220 - 300	0,36 - 0,41	0,22 - 0,28	0,15 - 0,2	60	0,05
N	non-ferrous metals	-	-	-	-	-	-	-	-	
S	super alloys	-	-	-	-	-	-	-	-	
H	hard machining	-	-	-	-	-	-	-	-	

Reduce cutting values by 20% for materials with Cr/Ni> 8% or Mo> 0.5%.

Note

A large grid of graph paper for taking notes, consisting of 20 columns and 30 rows of small squares. The grid is white with light gray lines. To the right of the grid is a vertical blue bar.

SOLID CARBIDE 47C HPC CUTTER Z = 4/5

Z = 3/4/5/7



Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Feed per tooth fz [mm]	Depth of cut ap max [mm]	Grade
P	unalloyed steel	3	140 - 200	0,02 - 0,04	1,5 x Ø	IN2005 / IN2505
		4	140 - 200	0,02 - 0,05	1,5 x Ø	IN2005 / IN2505
		5	140 - 200	0,025 - 0,06	1,5 x Ø	IN2005 / IN2505
		6	140 - 200	0,025 - 0,07	1,5 x Ø	IN2005 / IN2505
		8	140 - 200	0,03 - 0,09	1,5 x Ø	IN2005 / IN2505
		10	140 - 200	0,03 - 0,10	1,5 x Ø	IN2005 / IN2505
		12	140 - 200	0,035 - 0,11	1,5 x Ø	IN2005 / IN2505
		16	140 - 200	0,05 - 0,13	1,5 x Ø	IN2005 / IN2505
		20	140 - 200	0,05 - 0,17	1,5 x Ø	IN2005 / IN2505
	25	140 - 200	0,07 - 0,20	1,5 x Ø	IN2005 / IN2505	
	alloyed steel < 800 N/mm ²	3	140 - 200	0,02 - 0,04	1,5 x Ø	IN2005 / IN2505
		4	140 - 200	0,02 - 0,05	1,5 x Ø	IN2005 / IN2505
		5	140 - 200	0,025 - 0,06	1,5 x Ø	IN2005 / IN2505
		6	140 - 200	0,025 - 0,07	1,5 x Ø	IN2005 / IN2505
		8	140 - 200	0,03 - 0,09	1,5 x Ø	IN2005 / IN2505
		10	140 - 200	0,03 - 0,10	1,5 x Ø	IN2005 / IN2505
		12	140 - 200	0,035 - 0,11	1,5 x Ø	IN2005 / IN2505
		16	140 - 200	0,05 - 0,13	1,5 x Ø	IN2005 / IN2505
		20	140 - 200	0,05 - 0,17	1,5 x Ø	IN2005 / IN2505
	alloyed steel < 1100 N/mm ²	3	120 - 180	0,02 - 0,04	1,5 x Ø	IN2005 / IN2505
		4	120 - 180	0,02 - 0,05	1,5 x Ø	IN2005 / IN2505
		5	120 - 180	0,025 - 0,06	1,5 x Ø	IN2005 / IN2505
		6	120 - 180	0,025 - 0,07	1,5 x Ø	IN2005 / IN2505
		8	120 - 180	0,03 - 0,09	1,5 x Ø	IN2005 / IN2505
		10	120 - 180	0,03 - 0,10	1,5 x Ø	IN2005 / IN2505
12		120 - 180	0,035 - 0,11	1,5 x Ø	IN2005 / IN2505	
16		120 - 180	0,05 - 0,13	1,5 x Ø	IN2005 / IN2505	
20		120 - 180	0,05 - 0,17	1,5 x Ø	IN2005 / IN2505	
M	stainless steel	3	60 - 120	0,02 - 0,03	0,5 - 0,8 x Ø	IN2005 / IN2505
		4	60 - 120	0,02 - 0,035	0,5 - 0,8 x Ø	IN2005 / IN2505
		5	60 - 120	0,02 - 0,04	0,5 - 0,8 x Ø	IN2005 / IN2505
		6	60 - 120	0,02 - 0,05	0,5 - 0,8 x Ø	IN2005 / IN2505
		8	60 - 120	0,02 - 0,07	0,5 - 0,8 x Ø	IN2005 / IN2505
		10	60 - 120	0,02 - 0,08	0,5 - 0,8 x Ø	IN2005 / IN2505
		12	60 - 120	0,03 - 0,09	0,5 - 0,8 x Ø	IN2005 / IN2505
		16	60 - 120	0,04 - 0,10	0,5 - 0,8 x Ø	IN2005 / IN2505
		20	60 - 120	0,04 - 0,13	0,5 - 0,8 x Ø	IN2005 / IN2505
		25	60 - 120	0,05 - 0,15	0,5 - 0,8 x Ø	IN2005 / IN2505

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.

Z = 3/4/5/7



ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Feed per tooth fz [mm]	Depth of cut ap max [mm]	Grade
K	gray cast iron cast alloys	3	160 - 220	0,02 - 0,04	1,5 x Ø	IN2005 / IN2505
		4	160 - 220	0,02 - 0,05	1,5 x Ø	IN2005 / IN2505
		5	160 - 220	0,025 - 0,06	1,5 x Ø	IN2005 / IN2505
		6	160 - 220	0,025 - 0,07	1,5 x Ø	IN2005 / IN2505
		8	160 - 220	0,03 - 0,09	1,5 x Ø	IN2005 / IN2505
		10	160 - 220	0,03 - 0,10	1,5 x Ø	IN2005 / IN2505
		12	160 - 220	0,035 - 0,11	1,5 x Ø	IN2005 / IN2505
		16	160 - 220	0,05 - 0,13	1,5 x Ø	IN2005 / IN2505
		20	160 - 220	0,05 - 0,17	1,5 x Ø	IN2005 / IN2505
25	160 - 220	0,07 - 0,20	1,5 x Ø	IN2005 / IN2505		
N	non-ferrous metals	-	-	-	-	-
S	super alloys	3	40 - 80	0,02 - 0,03	0,5 - 0,8 x Ø	IN2005 / IN2505
		4	40 - 80	0,02 - 0,035	0,5 - 0,8 x Ø	IN2005 / IN2505
		5	40 - 80	0,02 - 0,04	0,5 - 0,8 x Ø	IN2005 / IN2505
		6	40 - 80	0,02 - 0,05	0,5 - 0,8 x Ø	IN2005 / IN2505
		8	40 - 80	0,02 - 0,07	0,5 - 0,8 x Ø	IN2005 / IN2505
		10	40 - 80	0,02 - 0,08	0,5 - 0,8 x Ø	IN2005 / IN2505
		12	40 - 80	0,03 - 0,09	0,5 - 0,8 x Ø	IN2005 / IN2505
		16	40 - 80	0,04 - 0,10	0,5 - 0,8 x Ø	IN2005 / IN2505
		20	40 - 80	0,04 - 0,13	0,5 - 0,8 x Ø	IN2005 / IN2505
25	40 - 80	0,05 - 0,15	0,5 - 0,8 x Ø	IN2005 / IN2505		
H	hardened steel < 54 HRC	3	80 - 140	0,01 - 0,03	0,3 x Ø	IN2005 / IN2505
		4	80 - 140	0,01 - 0,035	0,3 x Ø	IN2005 / IN2505
		5	80 - 140	0,01 - 0,04	0,3 x Ø	IN2005 / IN2505
		6	80 - 140	0,01 - 0,04	0,3 x Ø	IN2005 / IN2505
		8	80 - 140	0,02 - 0,04	0,3 x Ø	IN2005 / IN2505
		10	80 - 140	0,05 - 0,05	0,3 x Ø	IN2005 / IN2505
		12	80 - 140	0,02 - 0,07	0,3 x Ø	IN2005 / IN2505
		16	80 - 140	0,03 - 0,08	0,3 x Ø	IN2005 / IN2505
		20	80 - 140	0,03 - 0,10	0,3 x Ø	IN2005 / IN2505
25	80 - 140	0,05 - 0,12	0,3 x Ø	IN2005 / IN2505		

Cutting data for finishing:

Cutting speed	Vc = 20 - 30 %	increase
Feed per tooth	lowest value in table	
Cutting width	ae = 0,1 - 0,3 mm	
Infeed	ap max	according to the tools table

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Feed per tooth fz [mm]	Depth of cut ap max [mm]	Cutting width ae [mm]	Grade
P	unalloyed steel	8	160 - 220	0,05 - 0,10	5	0,04 - 0,09 x Ø	IN2005
		10	160 - 220	0,05 - 0,11	7	0,04 - 0,09 x Ø	IN2005
		12	160 - 220	0,05 - 0,12	9	0,04 - 0,09 x Ø	IN2005
		16	160 - 220	0,05 - 0,15	12	0,04 - 0,09 x Ø	IN2005
		20	160 - 220	0,05 - 0,18	15	0,04 - 0,09 x Ø	IN2005
		25	160 - 220	0,05 - 0,22	22	0,04 - 0,09 x Ø	IN2005
	alloyed steel < 800 N/mm ²	8	160 - 220	0,05 - 0,10	5	0,04 - 0,09 x Ø	IN2005
		10	160 - 220	0,05 - 0,11	7	0,04 - 0,09 x Ø	IN2005
		12	160 - 220	0,05 - 0,12	9	0,04 - 0,09 x Ø	IN2005
		16	160 - 220	0,05 - 0,15	12	0,04 - 0,09 x Ø	IN2005
		20	160 - 220	0,05 - 0,18	15	0,04 - 0,09 x Ø	IN2005
		25	160 - 220	0,05 - 0,22	22	0,04 - 0,09 x Ø	IN2005
	alloyed steel < 1100 N/mm ²	8	140 - 200	0,05 - 0,08	5	0,025 - 0,075 x Ø	IN2005
		10	140 - 200	0,05 - 0,09	7	0,025 - 0,075 x Ø	IN2005
		12	140 - 200	0,05 - 0,10	9	0,025 - 0,075 x Ø	IN2005
		16	140 - 200	0,05 - 0,12	12	0,025 - 0,075 x Ø	IN2005
		20	140 - 200	0,05 - 0,14	15	0,025 - 0,075 x Ø	IN2005
		25	140 - 200	0,05 - 0,16	22	0,025 - 0,075 x Ø	IN2005
M	stainless steel	8	60 - 120	0,03 - 0,07	5	0,025 - 0,075 x Ø	IN2005
		10	60 - 120	0,03 - 0,08	7	0,025 - 0,075 x Ø	IN2005
		12	60 - 120	0,04 - 0,10	9	0,025 - 0,075 x Ø	IN2005
		16	60 - 120	0,05 - 0,12	12	0,025 - 0,075 x Ø	IN2005
		20	60 - 120	0,05 - 0,15	15	0,025 - 0,075 x Ø	IN2005
		25	60 - 120	0,05 - 0,15	22	0,025 - 0,075 x Ø	IN2005
K	gray cast iron cast alloys	8	160 - 220	0,05 - 0,09	5	0,04 - 0,09 x Ø	IN2005
		10	160 - 220	0,05 - 0,10	7	0,04 - 0,09 x Ø	IN2005
		12	160 - 220	0,05 - 0,11	9	0,04 - 0,09 x Ø	IN2005
		16	160 - 220	0,05 - 0,13	12	0,04 - 0,09 x Ø	IN2005
		20	160 - 220	0,05 - 0,17	15	0,04 - 0,09 x Ø	IN2005
		25	160 - 220	0,05 - 0,20	22	0,04 - 0,09 x Ø	IN2005
N	non-ferrous metals	-	-	-	-	-	-
		-	-	-	-	-	-
S	super alloys	8	40 - 80	0,03 - 0,08	5	0,025 - 0,075 x Ø	IN2005
		10	40 - 80	0,03 - 0,09	7	0,025 - 0,075 x Ø	IN2005
		12	40 - 80	0,04 - 0,10	9	0,025 - 0,075 x Ø	IN2005
		16	40 - 80	0,05 - 0,12	12	0,025 - 0,075 x Ø	IN2005
		20	40 - 80	0,05 - 0,13	15	0,025 - 0,075 x Ø	IN2005
		25	40 - 80	0,05 - 0,13	22	0,025 - 0,075 x Ø	IN2005
H	hardened steel < 54 HRC	8	80 - 140	0,02 - 0,06	5	0,025 - 0,075 x Ø	IN2005
		10	80 - 140	0,02 - 0,07	7	0,025 - 0,075 x Ø	IN2005
		12	80 - 140	0,03 - 0,08	9	0,025 - 0,075 x Ø	IN2005
		16	80 - 140	0,04 - 0,08	12	0,025 - 0,075 x Ø	IN2005
		20	80 - 140	0,04 - 0,09	15	0,025 - 0,075 x Ø	IN2005

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.

Note

A large grid of graph paper for taking notes, consisting of 20 columns and 30 rows of small squares. The grid is bounded by a thin grey line on the top, left, and bottom, and a thick blue vertical bar on the right side.

SOLID CARBIDE 47J_ / 48J_ FINISHING END MILL



Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Feed per tooth fz [mm]	Cutting width ae [mm]	Length of cut [mm]	Grade	
P	unalloyed steel	6	160 - 220	0,05 - 0,10	0,04 - 0,09 x Ø	26	IN2005	
		8	160 - 220	0,05 - 0,11	0,04 - 0,09 x Ø	32	IN2005	
		10	160 - 220	0,05 - 0,12	0,04 - 0,09 x Ø	46	IN2005	
		12 - 14	160 - 220	0,05 - 0,15	0,04 - 0,09 x Ø	56	IN2005	
		16	160 - 220	0,05 - 0,18	0,0 - 0,09 x Ø	80	IN2005	
		20	160 - 220	0,05 - 0,22	0,04 - 0,09 x Ø	80	IN2005	
		25	160 - 220	0,05 - 0,22	0,04 - 0,09 x Ø	92	IN2005	
	alloyed steel < 800 N/mm ²	6	160 - 220	0,05 - 0,10	0,04 - 0,09 x Ø	26	IN2005	
		8	160 - 220	0,05 - 0,11	0,04 - 0,09 x Ø	32	IN2005	
		10	160 - 220	0,05 - 0,12	0,04 - 0,09 x Ø	46	IN2005	
		12 - 14	160 - 220	0,05 - 0,15	0,04 - 0,09 x Ø	56	IN2005	
		16	160 - 220	0,05 - 0,18	0,04 - 0,09 x Ø	80	IN2005	
		20	160 - 220	0,05 - 0,22	0,04 - 0,09 x Ø	80	IN2005	
		25	160 - 220	0,05 - 0,22	0,04 - 0,09 x Ø	92	IN2005	
	alloyed steel < 1100 N/mm ²	6	140 - 200	0,05 - 0,08	0,025 - 0,075 x Ø	26	IN2005	
		8	140 - 200	0,05 - 0,09	0,025 - 0,075 x Ø	32	IN2005	
		10	140 - 200	0,05 - 0,10	0,025 - 0,075 x Ø	46	IN2005	
		12 - 14	140 - 200	0,05 - 0,12	0,025 - 0,075 x Ø	56	IN2005	
		16	140 - 200	0,05 - 0,14	0,025 - 0,075 x Ø	80	IN2005	
		20	140 - 200	0,05 - 0,16	0,025 - 0,075 x Ø	80	IN2005	
		25	140 - 200	0,05 - 0,16	0,025 - 0,075 x Ø	92	IN2005	
	M	stainless steel	6	60 - 120	0,03 - 0,07	0,025 - 0,075 x Ø	26	IN2005
			8	60 - 120	0,03 - 0,08	0,025 - 0,075 x Ø	32	IN2005
			10	60 - 120	0,04 - 0,10	0,025 - 0,075 x Ø	46	IN2005
			12 - 14	60 - 120	0,05 - 0,12	0,025 - 0,075 x Ø	56	IN2005
16			60 - 120	0,05 - 0,15	0,025 - 0,075 x Ø	80	IN2005	
20			60 - 120	0,05 - 0,15	0,025 - 0,075 x Ø	80	IN2005	
25			60 - 120	0,05 - 0,15	0,025 - 0,075 x Ø	92	IN2005	

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Feed per tooth fz [mm]	Cutting width ae [mm]	Length of cut [mm]	Grade
K	gray cast iron cast alloys	6	160 - 220	0,05 - 0,09	0,04 - 0,09 x Ø	26	IN2005
		8	160 - 220	0,05 - 0,10	0,04 - 0,09 x Ø	32	IN2005
		10	160 - 220	0,05 - 0,11	0,04 - 0,09 x Ø	46	IN2005
		12 - 14	160 - 220	0,05 - 0,13	0,04 - 0,09 x Ø	56	IN2005
		16	160 - 220	0,05 - 0,17	0,04 - 0,09 x Ø	80	IN2005
		20	160 - 220	0,05 - 0,20	0,04 - 0,09 x Ø	80	IN2005
N	non-ferrous metals	25	160 - 220	0,05 - 0,20	0,04 - 0,09 x Ø	92	IN2005
		3 - 4	250 - 1000	0,03 - 0,08	0,03 - 0,15 x Ø	30	IN05S / IN3005
		5 - 6	250 - 1000	0,05 - 0,10	0,03 - 0,15 x Ø	40 - 50	IN05S / IN3005
		8	250 - 1000	0,06 - 0,15	0,03 - 0,15 x Ø	50	IN05S / IN3005
		10	250 - 1000	0,06 - 0,16	0,03 - 0,15 x Ø	60	IN05S / IN3005
S	super alloys	12	250 - 1000	0,06 - 0,16	0,03 - 0,15 x Ø	75	IN05S / IN3005
		6	40 - 80	0,03 - 0,08	0,025 - 0,075 x Ø	26	IN2005
		8	40 - 80	0,03 - 0,09	0,025 - 0,075 x Ø	32	IN2005
		10	40 - 80	0,04 - 0,10	0,025 - 0,075 x Ø	46	IN2005
		12 - 14	40 - 80	0,05 - 0,12	0,025 - 0,075 x Ø	56	IN2005
		16	40 - 80	0,05 - 0,13	0,025 - 0,075 x Ø	80	IN2005
		20	40 - 80	0,05 - 0,13	0,025 - 0,075 x Ø	80	IN2005
H	hard machining	25	40 - 80	0,05 - 0,13	0,025 - 0,075 x Ø	92	IN2005
		6	80 - 140	0,02 - 0,06	0,025 - 0,075 x Ø	26	IN2005
		8	80 - 140	0,02 - 0,07	0,025 - 0,075 x Ø	32	IN2005
		10	80 - 140	0,03 - 0,08	0,025 - 0,075 x Ø	46	IN2005
		12 - 14	80 - 140	0,04 - 0,08	0,025 - 0,075 x Ø	56	IN2005
		16	80 - 140	0,04 - 0,08	0,025 - 0,075 x Ø	80	IN2005
		20	80 - 140	0,04 - 0,09	0,025 - 0,075 x Ø	80	IN2005
25	80 - 140	0,04 - 0,09	0,025 - 0,075 x Ø	92	IN2005		

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.


Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Finishing	
				Feed per tooth fz [mm]	Depth of cut ap [mm]
P	unalloyed steel	8	160 - 220	0,10 - 0,13	0,25 - 0,35
		10	160 - 220	0,13 - 0,17	0,25 - 0,35
		12	160 - 220	0,15 - 0,20	0,25 - 0,35
		16	160 - 220	0,15 - 0,23	0,25 - 0,35
	alloyed steel < 800 N/mm ²	8	160 - 220	0,10 - 0,13	0,25 - 0,35
		10	160 - 220	0,13 - 0,17	0,25 - 0,35
		12	160 - 220	0,15 - 0,20	0,25 - 0,35
		16	160 - 220	0,15 - 0,23	0,25 - 0,35
	alloyed steel < 1100 N/mm ²	8	140 - 200	0,10 - 0,13	0,15 - 0,25
		10	140 - 200	0,13 - 0,17	0,15 - 0,25
		12	140 - 200	0,15 - 0,20	0,15 - 0,25
		16	140 - 200	0,15 - 0,23	0,15 - 0,25
M	stainless steel	8	60 - 120	0,10 - 0,13	0,15 - 0,25
		10	60 - 120	0,13 - 0,17	0,15 - 0,25
		12	60 - 120	0,15 - 0,20	0,15 - 0,25
		16	60 - 120	0,15 - 0,23	0,15 - 0,25
K	gray cast iron cast alloys	8	160 - 220	0,10 - 0,13	0,25 - 0,35
		10	160 - 220	0,13 - 0,17	0,25 - 0,35
		12	160 - 220	0,15 - 0,20	0,25 - 0,35
N	non-ferrous metals	-	-	-	-
		-	-	-	-
		-	-	-	-
S	super alloys	8	40 - 80	0,10 - 0,13	0,15 - 0,25
		10	40 - 80	0,13 - 0,17	0,15 - 0,25
		12	40 - 80	0,15 - 0,20	0,15 - 0,25
		16	40 - 80	0,15 - 0,23	0,15 - 0,25
H	hardened steel < 54 HRC	10	80 - 140	0,10 - 0,13	0,15 - 0,25
		12	80 - 140	0,13 - 0,17	0,15 - 0,25
		12	80 - 140	0,15 - 0,20	0,15 - 0,25
		16	80 - 140	0,15 - 0,23	0,15 - 0,25

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.


Recommended cutting data:

ISO	Material	Material composition	Condition	Hardness [HB]	Material No.	Cutting speed (m/min)	Feed per tooth fz (mm)						Depth of cut ap [mm]
							Ø6	Ø8	Ø10	Ø12	Ø16	Ø20	
							R 0,42	R 0,56	R 0,70	R 1,10	R 1,90	R 2,50	
P	steel	-	-	-	-	-	-	-	-	-	-	-	-
		-	-	-	-	-	-	-	-	-	-	-	-
M	stainless steel	-	-	-	-	-	-	-	-	-	-	-	-
		-	-	-	-	-	-	-	-	-	-	-	-
K	gray cast iron	pearlitic/ferritic	-	180	15	250-1000	0,1	0,15	0,17	0,19	0,23	0,25	0,25-1,0
		pearlitic (martensitic)	-	260	16	250-1000	0,1	0,15	0,17	0,19	0,23	0,25	0,25-1,0
	cast iron nodular	ferritic	-	160	17	250-1000	0,1	0,15	0,17	0,19	0,23	0,25	0,25-1,0
		ferritic	-	250	18	250-1000	0,1	0,15	0,17	0,19	0,23	0,25	0,25-1,0
	malleable cast iron	ferritic	-	130	19	250-1000	0,1	0,15	0,17	0,19	0,23	0,25	0,25-1,0
		pearlitic	-	230	20	250-1000	0,1	0,15	0,17	0,19	0,23	0,25	0,25-1,0
N	non-ferrous metals	Graphite	-	-	29	500-1500	0,1	0,15	0,17	0,19	0,23	0,25	0,25-1,0
S	nickel-based alloy	Alpha+Beta-alloy	hardened	Rm1050 ¹⁾	37	250-1000	0,1	0,13	0,15	0,18	0,2	0,22	0,25-1,0
H	hard machining	-	-	-	-	-	-	-	-	-	-	-	-
		-	-	-	-	-	-	-	-	-	-	-	-

¹⁾Rm: Tensile strength in MPa

²⁾R: Programming radius



Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]	Feed rate fu [mm]					1st choice grade
			4 – 5	5,1 – 6	6,1 – 8	8,1 – 10	10,1 – 12	
P	unalloyed steel	70 – 140	0,15 – 0,25	0,20 – 0,35	0,25 – 0,45	0,30 – 0,55	0,35 – 0,60	IN2205
	alloyed steel 800 N/mm ²	50 – 110	0,15 – 0,25	0,20 – 0,35	0,25 – 0,40	0,30 – 0,50	0,35 – 0,55	IN2205
	alloyed steel 1100 N/mm ²	40 – 90	0,15 – 0,20	0,20 – 0,30	0,25 – 0,35	0,30 – 0,45	0,35 – 0,50	IN2205
M	stainless steel	-	-	-	-	-	-	-
K	gray cast iron	70 – 140	0,20 – 0,30	0,25 – 0,45	0,35 – 0,55	0,40 – 0,60	0,45 – 0,65	IN2205
	nodular cast iron	70 – 120	0,20 – 0,30	0,25 – 0,40	0,30 – 0,50	0,35 – 0,55	0,40 – 0,60	IN2205
N	aluminum	-	-	-	-	-	-	-
S	high temperature alloys	-	-	-	-	-	-	-
	titanium alloys	-	-	-	-	-	-	-
H	hard machining < 54 HRC	-	-	-	-	-	-	-
	hard machining < 60 HRC	-	-	-	-	-	-	-

Remarks & tips:

- The data given are guidelines that may differ from your application.
- The optimum chip shape is created by varying the cutting speed and feed.
- When drill retracts from borehole a disc drops down, so at rotating workpieces a risk of accidents exists!

Please take care for respective safety measures!

- It is recommended to calculate the required machine power in advance and compare it with the drive power actually available by the machine.
- Drill must always be supplied with enough coolant via internal coolant supply to ensure an optimum chip flow.

TWIST SPEED FP_R01 DRILL HEADS

Drill heads



Drill head:	FPC_R01	FPF_R01
Geometry for:	self-centering	flat bottom machining

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]	Feed rate fu [mm/U]				1st choice grade / geometry
			12,0 - 13,9	14,0 - 15,9	16,0 - 19,9	20,0 - 25,5	
P	unalloyed steel	80-140	0,25 - 0,45	0,30 - 0,50	0,35 - 0,55	0,40 - 0,60	IN2205 / FPC
	alloyed steel 800 N/mm ²	50-110	0,20 - 0,40	0,25 - 0,45	0,30 - 0,50	0,35 - 0,55	IN2205 / FPC
	alloyed steel 1100 N/mm ²	40-70	0,20 - 0,35	0,25 - 0,40	0,30 - 0,45	0,35 - 0,50	IN2205 / FPC
M	stainless steel	-	-	-	-	-	-
K	gray cast iron	80 - 160	0,30 - 0,50	0,40 - 0,60	0,45 - 0,65	0,50 - 0,70	IN2205 / FPC
	nodular cast iron	90 - 180	0,30 - 0,45	0,35 - 0,55	0,40 - 0,60	0,45 - 0,65	IN2205 / FPC
N	aluminum	-	-	-	-	-	-
S	high temperature alloys	-	-	-	-	-	-
	titanium alloys	-	-	-	-	-	-
H	hard machining < 54 HRC	-	-	-	-	-	-
	hard machining < 60 HRC	-	-	-	-	-	-

Hinweise & Tips:

- When drill retracts from borehole a disc drops down, so at rotating workpieces a risk of accidents exists!
Please take care for respective safety measures!
- At through holes take care, that the drill head is guided when the drill drops out.
- Drill must always be supplied with enough coolant via internal coolant supply to ensure an optimum chip flow.
- The data given are guidelines that may differ from your application.
- The optimum chip shape is created by varying the cutting speed and feed.

Drill heads



Drill head:	TNA_R01	TPA_R01	TMA_R01	TKA_R01	TPF -_R01
Geometry for:	aluminum machining	general steel machining	stainless steel machining	cast iron machining	flat bottom machining

Drill heads



Drill head:	TPC_R01
Geometry for:	self-centering

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]	Feed rate fu [mm]						1st choice grade / geometry
			4,0 - 9,9	10,0 - 11,9	12,0 - 13,9	14,0 - 15,9	16,0 - 19,9	20,0 - 25,9	
P	unalloyed steel	70 - 140	0,07 - 0,22	0,15 - 0,28	0,18 - 0,30	0,20 - 0,35	0,25 - 0,45	0,25 - 0,45	IN2505 / TPA
	alloyed steel 800 N/mm ²	50 - 110	0,07 - 0,25	0,14 - 0,28	0,16 - 0,32	0,18 - 0,35	0,23 - 0,40	0,25 - 0,45	IN2505 / TPA
	alloyed steel 1100 N/mm ²	40 - 80	0,07 - 0,20	0,12 - 0,22	0,14 - 0,25	0,16 - 0,28	0,18 - 0,30	0,22 - 0,33	IN2505 / TPA
M	stainless steel	30 - 70	0,08 - 0,15	0,12 - 0,18	0,14 - 0,20	0,16 - 0,24	0,18 - 0,26	0,20 - 0,28	IN2505 / TMA
K	gray cast iron	80 - 160	0,10 - 0,30	0,20 - 0,35	0,25 - 0,40	0,30 - 0,45	0,35 - 0,55	0,35 - 0,60	IN2505 / TKA
	nodular cast iron	90 - 180	0,10 - 0,30	0,20 - 0,35	0,25 - 0,40	0,30 - 0,45	0,35 - 0,55	0,35 - 0,60	IN2505 / TKA
N	aluminum	90 - 220	0,18 - 0,35	0,25 - 0,35	0,30 - 0,45	0,35 - 0,50	0,40 - 0,60	0,45 - 0,70	IN055 / TNA
S	high temperature alloys	20 - 60	0,06 - 0,11	0,08 - 0,13	0,10 - 0,15	0,12 - 0,18	0,12 - 0,20	0,14 - 0,22	IN2505 / TMA
	titanium alloys	20 - 50	0,06 - 0,12	0,08 - 0,15	0,10 - 0,18	0,12 - 0,20	0,14 - 0,22	0,16 - 0,24	IN2505 / TMA
H	hard machining < 54 HRC	20 - 50	0,06 - 0,12	0,08 - 0,15	0,10 - 0,18	0,12 - 0,20	0,14 - 0,22	0,16 - 0,24	IN2505 / TPA
	hard machining < 60 HRC	20 - 50	0,06 - 0,12	0,08 - 0,15	0,10 - 0,18	0,12 - 0,20	0,14 - 0,22	0,16 - 0,24	IN2505 / TPA

The indicated data are guide values, which can deviate according to your application.
The optimum chip shape is generated by variation of cutting speed and feed rate.

Tips:

- When drill retracts from borehole a disc drops down, so at rotating workpieces a risk of accidents exists!
Please take care for respective safety measures!
- At through holes take care, that the drill head is guided when the drill drops out.
- For drills with L/D = 8 and 12xD use lower feed rates, if necessary reduce lower feed rate by 50 % for spot-drilling.
- If necessary use pilot boring for drills with L/D = 8 and 12xD.
- Drill must always be supplied with enough coolant via internal coolant supply to ensure an optimum chip flow.
- For drilling with TPFxxxR01 head please use the lower feed rate values (fu).

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.

SPADETWIST LP_R01 DRILL HEADS

Drill heads



Drill head:	LPA_R01	LPC_R01	LPF_R01
Geometry for:	machining of steel	Steel-C-machining	flat bottom machining

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]	Feed rate fu [mm]				1st choice grade / geometry
			20,0 - 25,9	26,0 - 29,9	30,0 - 34,9	35,0 - 41,0	
P	unalloyed steel	70 - 140	0,25 - 0,45	0,30 - 0,50	0,30 - 0,50	0,35 - 0,55	IN2505 / LPA
	alloyed steel 800 N/mm ²	50 - 110	0,22 - 0,40	0,25 - 0,45	0,25 - 0,45	0,30 - 0,50	IN2505 / LPA
	alloyed steel 1100 N/mm ²	50 - 90	0,22 - 0,32	0,25 - 0,35	0,25 - 0,35	0,30 - 0,40	IN2505 / LPA
M	stainless steel	30 - 70	0,14 - 0,28	0,15 - 0,30	0,15 - 0,30	0,20 - 0,35	IN2505 / LPA
K	gray cast iron	80 - 180	0,25 - 0,50	0,30 - 0,55	0,35 - 0,55	0,45 - 0,60	IN2505 / LPA
	nodular cast iron	90 - 165	0,25 - 0,50	0,30 - 0,55	0,35 - 0,55	0,40 - 0,60	IN2505 / LPA
N	aluminum	80 - 220	0,25 - 0,45	0,30 - 0,50	0,40 - 0,60	0,50 - 0,70	IN2505 / LPA
S	high temperature alloys	20 - 60	0,10 - 0,16	0,10 - 0,20	0,15 - 0,25	0,15 - 0,25	IN2505 / LPA
	titanium alloys	20 - 50	0,10 - 0,16	0,10 - 0,20	0,15 - 0,25	0,15 - 0,25	IN2505 / LPA
H	hard machining < 54 HRC	20 - 50	0,10 - 0,16	0,10 - 0,20	0,15 - 0,25	0,15 - 0,25	IN2505 / LPA
	hard machining < 60 HRC	20 - 50	0,10 - 0,16	0,10 - 0,20	0,15 - 0,25	0,15 - 0,25	IN2505 / LPA

Remarks & tips:

- The indicated data are guide values, which can deviate according to your application.
- The optimum chip shape is generated by variation of cutting speed and feed rate.
- When drill retracts from borehole a disc drops down, so at rotating workpieces a risk of accidents exists.

Please take care for respective safety measures!

- It is recommended to calculate the required machine power and to match it with the actually available power of the machine.
- Drill must always be supplied with enough coolant via internal coolant supply to ensure an optimum chip flow.
- If necessary reduce feed rate until drill is guided and in perfect cut direction.
- For drills with L/D=8 use lower feed rates, if necessary reduce lower feed rate by 50 % for spot-drilling.
- If necessary use pilot boring for drills with L/D = 8.

General information:

LP_2000R01 up to LP_2190R01:	insert screw: TS 40178D25	torque: 4,5 Nm	torque wrench: DTNVO05	bit: DS-T20TB
LP_2200R01 up to LP_2390R01:	insert screw: TS 40198D28	torque: 4,5 Nm	torque wrench: DTNVO05	bit: DS-T20TB
LP_2400R01 up to LP_2590R01:	insert screw: TS 40210D3	torque: 4,5 Nm	torque wrench: DTNVO05	bit: DS-T20TB
LP_2600R01 up to LP_2790R01:	insert screw: TS 50230D3	torque: 5,0 Nm	torque wrench: DTNVO05	bit: DS-T20TB
LP_2800R01 up to LP_2990R01:	insert screw: TS 50250D35	torque: 5,5 Nm	torque wrench: DTNVO05	bit: DS-T25TB
LP_3000R01 up to LP_3190R01:	insert screw: TS 60265D4	torque: 6,0 Nm	torque wrench: DTNVO05	bit: DS-T25TB
LP_3200R01 up to LP_3490R01:	insert screw: TS 60285D42	torque: 6,0 Nm	torque wrench: DTNVO05	bit: DS-T25TB
LP_3500R01 up to LP_3790R01:	insert screw: TS 60320D5	torque: 6,0 Nm	torque wrench: DTNVO05	bit: DS-T25TB
LP_3800R01 up to LP_4100R01:	insert screw: TS 80340D6	torque: 7,0 Nm	torque wrench: DTNVO05	bit: DS-T25TB

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Insert / Drill head: SPGX_WG TPC_R01-C

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]	Feed rate fu [mm]				
			SPGX06 Ø26 - Ø28	SPGX07 Ø29 - Ø32	SPGX09 Ø33 - Ø36	SPGX11 Ø37 - Ø43	SPGX11 Ø44 - Ø50
P	unalloyed steel	120 - 200	0,16 - 0,33	0,18 - 0,35	0,20 - 0,38	0,22 - 0,40	0,25 - 0,45
	alloyed steel 800 N/mm ²	100 - 190	0,16 - 0,33	0,18 - 0,35	0,20 - 0,38	0,22 - 0,40	0,25 - 0,45
	alloyed steel 1100 N/mm ²	100 - 160	0,16 - 0,30	0,18 - 0,33	0,20 - 0,36	0,22 - 0,40	0,25 - 0,40
M	stainless steel	80 - 140	0,10 - 0,15	0,12 - 0,18	0,14 - 0,20	0,16 - 0,24	0,18 - 0,26
K	gray cast iron	100 - 250	0,15 - 0,30	0,20 - 0,35	0,25 - 0,40	0,30 - 0,45	0,35 - 0,55
	nodular cast iron	100 - 250	0,15 - 0,30	0,20 - 0,35	0,25 - 0,40	0,30 - 0,45	0,35 - 0,55
N	aluminum	160 - 250	0,20 - 0,30	0,25 - 0,35	0,30 - 0,40	0,35 - 0,45	0,40 - 0,55
S	high temperature alloys	30 - 80	0,06 - 0,11	0,08 - 0,13	0,10 - 0,15	0,12 - 0,18	0,12 - 0,20
	titanium alloys	30 - 80	0,06 - 0,12	0,08 - 0,15	0,10 - 0,18	0,12 - 0,20	0,14 - 0,22
H	hard machining < 54 HRC	20 - 50	0,06 - 0,12	0,08 - 0,15	0,10 - 0,18	0,12 - 0,20	0,14 - 0,22
	hard machining < 60 HRC	-	-	-	-	-	-

ISO	Material	Cutting speed Vc [m/min]	1st choice solid carbide drill head grade / geometry	1st choice insert grade / geometry
P	unalloyed steel	120 - 200	IN2505 / TPC - C	IN2505 / -WG
	alloyed steel 800 N/mm ²	100 - 190	IN2505 / TPC - C	IN2505 / -WG
	alloyed steel 1100 N/mm ²	100 - 160	IN2505 / TPC - C	IN2505 / -WG
M	stainless steel	80 - 140	IN2505 / TPC - C	IN2505 / -WG
K	gray cast iron	100 - 250	IN2505 / TPC - C	IN2505 / -WG
	nodular cast iron	100 - 250	IN2505 / TPC - C	IN2505 / -WG
N	aluminum	160 - 250	IN2505 / TPC - C	IN2505 / -WG
S	high temperature alloys	30 - 80	IN2505 / TPC - C	IN2505 / -WG
	titanium alloys	30 - 80	IN2505 / TPC - C	IN2505 / -WG
H	hard machining < 54 HRC	20 - 50	IN2505 / TPC - C	IN2505 / -WG
	hard machining < 60 HRC	-	-	-

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.

Remarks & tips:

- The indicated data are guide values, which can deviate according to your application.
- The optimum chip shape is generated by variation of cutting speed and feed rate.
- When drill retracts from borehole a disc drops down, so at rotating workpieces a risk of accidents exists!

Please take care for respective safety measures!

- It is recommended to calculate the required machine power and to match it with the actually available power of the machine.
- Drill must always be supplied with enough coolant via internal coolant supply to ensure an optimum chip flow.
- If necessary reduce feed rate until drill is guided and in perfect cut direction.

General information:

SPGX060204WG	insert screw: SM22-052-00	torque: 0,8 Nm	torque wrench: DTNV01S	bit: DS-T07TB
SPGX07T308WG	insert screw: SM25-064-00	torque: 1,1 Nm	torque wrench: DTN011S	bit: DS-T08TB
SPGX090408WG	insert screw: SM35-088-60	torque: 3,0 Nm	torque wrench: DTN030S	bit: DS-T10TB
SPGX110408WG	insert screw: SM40-093-20	torque: 4,5 Nm	torque wrench: DTN045F	bit: DS-T15TB
SPGX140512WG	insert screw: SO 50090I	torque: 7,5 Nm	torque wrench: DTNV00S	bit: DS-T20TB

Indexable Drills



Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]	Feed per tooth fz [mm]			1st choice grade / geometry
			SCLT050204N-PH SHGT050204-HP SCLT050204N	SHLT060204N-PH SHGT060204-HP SHLT060204N	SPLT07T308N-PH SDGT07T308-HP SPLT07T308N	
P	unalloyed steel	200 - 300	0,05 - 0,10	0,06 - 0,11	0,06 - 0,12	IN2005
	alloyed steel 800 N/mm ²	120 - 200	0,05 - 0,12	0,08 - 0,15	0,10 - 0,18	IN2005
	alloyed steel 1100 N/mm ²	120 - 170	0,05 - 0,11	0,08 - 0,15	0,10 - 0,18	IN2005
M	stainless steel	150 - 220	0,05 - 0,11	0,06 - 0,12	0,08 - 0,15	IN2005
K	gray cast iron	180 - 250	0,05 - 0,11	0,08 - 0,16	0,12 - 0,20	IN2010
	nodular cast iron	160 - 230	0,05 - 0,11	0,08 - 0,16	0,12 - 0,20	IN2010
N	aluminum	300 - 600	0,05 - 0,12	0,08 - 0,15	0,10 - 0,20	IN10K
S	high temperature alloys	30 - 70	0,04 - 0,11	0,06 - 0,14	0,08 - 0,18	IN2005 / IN2530
	titanium alloys	30 - 60	0,04 - 0,11	0,06 - 0,14	0,08 - 0,18	IN2530
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 60 HRC	-	-	-	-	-

ISO	Material	Cutting speed Vc [m/min]	Feed per tooth fz [mm]			1st choice grade / geometry
			SHLT090408N-PH1 SHGT090408-HP SHLT090408N	SHLT110408N-PH1 SHGT110408-HP SHLT110408N	SPLT140512N SDGT140512-HP	
P	unalloyed steel	200 - 300	0,07 - 0,13	0,08 - 0,15	0,08 - 0,16	IN2005
	alloyed steel 800 N/mm ²	120 - 200	0,12 - 0,22	0,12 - 0,24	0,13 - 0,25	IN2005
	alloyed steel 1100 N/mm ²	120 - 170	0,12 - 0,22	0,12 - 0,24	0,13 - 0,25	IN2005
M	stainless steel	150 - 220	0,09 - 0,16	0,10 - 0,17	0,11 - 0,19	IN2005
K	gray cast iron	180 - 250	0,15 - 0,25	0,16 - 0,28	0,18 - 0,30	IN2010
	nodular cast iron	160 - 230	0,15 - 0,25	0,16 - 0,28	0,18 - 0,30	IN2010
N	aluminum	300 - 600	0,12 - 0,22	0,14 - 0,23	0,15 - 0,26	IN10K
S	high temperature alloys	30 - 70	0,10 - 0,22	0,14 - 0,23	0,15 - 0,24	IN2005 / IN2530
	titanium alloys	30 - 60	0,10 - 0,22	0,14 - 0,23	0,15 - 0,24	IN2530
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 60 HRC	-	-	-	-	-

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.

Remarks & tips:

- The indicated data are guide values, which can deviate according to your application.
- The optimum chip shape is generated by variation of cutting speed and feed rate.
- For information regarding the number of cutting edges (Z_{eff}) please see catalogue.
- When drill retracts from borehole a disc drops down, so at rotating workpieces a risk of accidents exists!

Please take care for respective safety measures!

- At first it is recommended to calculate the required machine capacity and to match it with the actual drive capacity of the machine.
- For drills with $L/D=5$ use lower feed rates, if necessary reduce lower feed rate by 50 % for spot-drilling.
- Drill must always be supplied with enough coolant via internal coolant supply to ensure an optimum chip flow.

General information:

SCLT050204_ _ _	insert screw: SM20-043-00	torque: 0,7 Nm	torque wrench: DTNV01S	bit: DS-TP06TB
SHLT060204_ _ _	insert screw: SM22-052-00	torque: 0,8 Nm	torque wrench: DTNV01S	bit: DS-T07TB
SPLT07T308_ _ _	insert screw: SM25-064-00	torque: 1,1 Nm	torque wrench: DTNV01S	bit: DS-T08TB
SHLT090408_ _ _	insert screw: SM35-088-60	torque: 3,0 Nm	torque wrench: DTNV00S	bit: DS-T10TB
SHLT110408_ _ _	insert screw: SM40-093-20	torque: 4,5 Nm	torque wrench: DTNV00S	bit: DS-T15TB
SPLT120408_ _ _	insert screw: SM40-093-20	torque: 4,5 Nm	torque wrench: DTNV00S	bit: DS-T15TB
SPLT140512_ _ _	insert screw: SM50-122-50	torque: 7,5 Nm	torque wrench: DTNV00S	bit: DS-T20TB



Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]	Feed per tooth fz [mm]					1st choice grade peripheral/center
			SOMT040204SK	SOMT050204SK SOMT050204NG SOMT050204HP SOMT050204PS	SOMT060204SK SOMT060204NG SOMT060204HP SOMT060204PS	SOMT070306SK SOMT070306NG SOMT071306HP SOMT070306PS	SOMT08T306SK SOMT08T306NG SOMT08T306HP SOMT08T306PS	
P	unalloyed steel	180 - 350	0,04 - 0,10	0,04 - 0,10	0,04 - 0,10	0,04 - 0,12	0,04 - 0,12	IN2505
	alloyed steel 800 N/mm ²	100 - 240	0,06 - 0,16	0,06 - 0,16	0,06 - 0,16	0,08 - 0,20	0,08 - 0,20	IN2505
	alloyed steel 1100 N/mm ²	100 - 180	0,06 - 0,16	0,06 - 0,16	0,06 - 0,16	0,08 - 0,20	0,08 - 0,20	IN2505
M	stainless steel	150 - 250	0,04 - 0,10	0,04 - 0,10	0,04 - 0,10	0,05 - 0,12	0,05 - 0,12	IN2505
K	gray cast iron	160 - 260	0,08 - 0,18	0,08 - 0,18	0,08 - 0,18	0,10 - 0,20	0,10 - 0,20	IN2510
	nodular cast iron	160 - 260	0,08 - 0,18	0,08 - 0,18	0,08 - 0,18	0,10 - 0,20	0,10 - 0,20	IN2510
N	aluminum	150 - 350	0,06 - 0,15	0,06 - 0,15	0,06 - 0,15	0,08 - 0,16	0,08 - 0,16	IN10K
S	high temperature alloys	30 - 60	0,05 - 0,08	0,05 - 0,08	0,05 - 0,08	0,05 - 0,09	0,05 - 0,09	IN2505 / IN2530
	titanium alloys	50 - 80	0,06 - 0,09	0,06 - 0,09	0,06 - 0,09	0,06 - 0,10	0,06 - 0,10	IN2530
H	hard machining < 54 HRC	30 - 60	0,05 - 0,09	0,05 - 0,09	0,05 - 0,09	0,05 - 0,10	0,05 - 0,10	IN2505
	hard machining < 60 HRC	30 - 50	0,05 - 0,09	0,05 - 0,09	0,05 - 0,09	0,05 - 0,10	0,05 - 0,10	IN2505

ISO	Material	Cutting speed Vc [m/min]	Feed per tooth fz [mm]				1st choice grade peripheral/center
			SOMT09T308SK SOMT09T308NG SOMT09T308HP SOMT09T308PS	SOMT11T308SK SOMT11T308NG SOMT11T308HP SOMT11T308PS	SOMT130408SK SOMT130408NG SOMT130408HP SOMT130408PS	SOMT150510SK SOMT150510NG SOMT150510HP SOMT150510PS	
P	unalloyed steel	180 - 350	0,06 - 0,14	0,06 - 0,14	0,08 - 0,16	0,08 - 0,16	IN2505
	alloyed steel 800 N/mm ²	100 - 240	0,08 - 0,20	0,10 - 0,22	0,10 - 0,22	0,10 - 0,24	IN2505
	alloyed steel 1100 N/mm ²	100 - 180	0,08 - 0,20	0,08 - 0,22	0,10 - 0,22	0,10 - 0,22	IN2505
M	stainless steel	150 - 250	0,06 - 0,14	0,06 - 0,14	0,08 - 0,16	0,08 - 0,16	IN2505
K	gray cast iron	160 - 260	0,10 - 0,20	0,10 - 0,20	0,10 - 0,22	0,10 - 0,22	IN2510
	nodular cast iron	160 - 260	0,10 - 0,20	0,10 - 0,20	0,10 - 0,22	0,10 - 0,22	IN2510
N	aluminum	150 - 350	0,08 - 0,18	0,08 - 0,18	0,10 - 0,20	0,10 - 0,20	IN10K
S	high temperature alloys	30 - 60	0,06 - 0,10	0,06 - 0,10	0,06 - 0,12	0,06 - 0,12	IN2505/IN2530
	titanium alloys	50 - 80	0,06 - 0,11	0,06 - 0,11	0,06 - 0,12	0,06 - 0,12	IN2530
H	hard machining < 54 HRC	30 - 60	0,05 - 0,11	0,05 - 0,11	0,05 - 0,12	0,05 - 0,12	IN2505
	hard machining < 60 HRC	30 - 50	0,05 - 0,11	0,05 - 0,11	0,05 - 0,12	0,05 - 0,12	IN2505

Remarks & tips:

- The indicated data are guide values, which can deviate according to your application.
- The optimum chip shape is generated by variation of cutting speed and feed rate.
- For information regarding the number of cutting edges (Z_{eff}) please see catalogue.
- When drill retracts from borehole a disc drops down, so at rotating workpieces a risk of accidents exists.

Please take care for respective safety measures!

- At first it is recommended to calculate the required machine capacity and to match it with the actual drive capacity of the machine.
- For drills with $L/D=5$ use lower feed rates, if necessary reduce lower feed rate by 50 % for spot-drilling.
- Drill must always be supplied with enough coolant via internal coolant supply to ensure an optimum chip flow.

General information:

SOMT040204_	insert screw: TS 180411/HG	torque: 0,5 Nm	torque wrench: DTN005S	bit: DS-TP06TB
SOMT050204_	insert screw: SM20-043-00	torque: 0,7 Nm	torque wrench: DTNV01S	bit: DS-TP06TB
SOMT060204_ / SOMT070306_	insert screw: TS 220521/HG-P	torque: 0,8 Nm	torque wrench: DTNV01S	bit: DS-TP07TB
SOMT08T306_	insert screw: SO 25065I	torque: 1,1 Nm	torque wrench: DTN011S	bit: DS-T07TB
SOMT09T308_ / SOMT11T308_	insert screw: SM35-088-60	torque: 3,0 Nm	torque wrench: DTNV00S	bit: DS-T10TB
SOMT130408_	insert screw: SM40-093-20	torque: 4,5 Nm	torque wrench: DTNV00S	bit: DS-T15TB
SOMT150510_	insert screw: SM50-113-20	torque: 8,0 Nm	torque wrench: DTNV00S	bit: DS-T20TB



Insert:	TFLT15T303N	TFLT15T308N
---------	-------------	-------------

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]	Feed per tooth fz [mm]		1st choice grade peripheral / center
			TFLT15T303N	TFLT15T308N	
P	unalloyed steel	120 - 180	0,03 - 0,08	0,03 - 0,08	IN2530
	alloyed steel 800 N/mm ²	100 - 160	0,05 - 0,10	0,05 - 0,10	IN2530
	alloyed steel 1100 N/mm ²	80 - 140	0,05 - 0,10	0,05 - 0,10	IN2530
M	stainless steel	100 - 170	0,03 - 0,08	0,03 - 0,08	IN2530
K	gray cast iron	100 - 130	0,05 - 0,10	0,05 - 0,10	IN2530
	nodular cast iron	120 - 150	0,05 - 0,10	0,05 - 0,10	IN2530
N	aluminum	200 - 300	0,05 - 0,10	0,05 - 0,10	IN2530
S	high temperature alloys	30 - 60	0,03 - 0,08	0,03 - 0,08	IN2530
	titanium alloys	50 - 80	0,03 - 0,08	0,03 - 0,08	IN2530
H	hard machining < 54 HRC	13 - 30	0,05 - 0,10	0,05 - 0,10	IN2530
	hard machining < 60 HRC	15 - 30	0,05 - 0,10	0,05 - 0,10	IN2530

Remarks & tips:

- The indicated data are guide values, which can deviate according to your application.
- The optimum chip shape is generated by variation of cutting speed and feed rate.
- For information regarding the number of cutting edges (Z_{eff}) please see catalogue.

General information:

 insert screw: **SM30-065-00**

 torque: **2 Nm**

 torque wrench: **DTN020S**

 bit: **DS-T09TB**



Insert: LPHT060204R TPHT070304R TPHT080305R TPHT090305R TPHT100305R



Insert: TPHT110405R TPHT120405R TPHT130408R ZPHT060204R-DT

Recommended cutting data:

ISO	Material	Cutting speed Vc (m/min)	Feed rate fu [mm]							1st choice grade
			LPHT060204R ZPHT060204R	TPHT070304R	TPHT080305R TPHT090305R	TPHT100305R	TPHT110405R	TPHT120405R	TPHT130408R	
P	unalloyed steel	80 - 140	0,05 - 0,10	0,06 - 0,12	0,08 - 0,14	0,10 - 0,16	0,10 - 0,18	0,10 - 0,20	0,10 - 0,20	IN2005
	alloyed steel 800 N/mm ²	80 - 120	0,05 - 0,10	0,06 - 0,12	0,08 - 0,14	0,10 - 0,16	0,10 - 0,18	0,10 - 0,20	0,10 - 0,20	IN2005
	alloyed steel 1100 N/mm ²	80 - 120	0,05 - 0,10	0,06 - 0,12	0,08 - 0,14	0,10 - 0,16	0,10 - 0,18	0,10 - 0,20	0,10 - 0,20	IN2005
M	stainless steel	80 - 140	0,05 - 0,08	0,06 - 0,10	0,08 - 0,10	0,08 - 0,12	0,08 - 0,12	0,08 - 0,16	0,08 - 0,16	IN2005
K	gray cast iron	80 - 140	0,06 - 0,14	0,08 - 0,18	0,10 - 0,20	0,12 - 0,25	0,12 - 0,25	0,14 - 0,30	0,14 - 0,30	IN2005
	nodular cast iron	80 - 140	0,06 - 0,14	0,08 - 0,18	0,10 - 0,20	0,12 - 0,25	0,12 - 0,25	0,14 - 0,30	0,14 - 0,30	IN2005
N	aluminum	65 - 130	0,05 - 0,12	0,06 - 0,14	0,08 - 0,16	0,08 - 0,18	0,08 - 0,18	0,10 - 0,20	0,10 - 0,20	IN2005
S	high temperature alloys	20 - 50	0,05 - 0,12	0,06 - 0,14	0,08 - 0,16	0,08 - 0,18	0,08 - 0,18	0,10 - 0,20	0,10 - 0,20	IN2005
	titanium alloys	20 - 50	0,05 - 0,12	0,06 - 0,14	0,08 - 0,16	0,08 - 0,18	0,08 - 0,18	0,10 - 0,20	0,10 - 0,20	IN2005
H	hard machining < 54 HRC	-	-	-	-	-	-	-	-	-
	hard machining < 60 HRC	-	-	-	-	-	-	-	-	-

Tips:

- The indicated data are guide values, which can deviate according to your application.
- The optimum chip shape is generated by variation of cutting speed and feed rate.
- When drill retracts from borehole a disc drops down, so at rotating workpieces a risk of accidents exists!

Please take care for respective safety measures!

- It is recommended to calculate the required machine power and to match it with the actually available power of the machine.
- Drill must always be supplied with enough coolant via internal coolant supply to ensure an optimum chip flow.
- When using boring emulsion use appropriate amount of oil, to ensure sufficient lubrication of the guide pads.
- Pilot boring is necessarily required. Unguided tools must rotate only with low speeds of max. 50 rpm!

General information:

LPHT060204R: insert screw: SR10503833L040
 THPT070304R: insert screw: SR14-560/S
 THPT080305R: insert screw: SR14-560/S
 THPT090305R: insert screw: SR14-560/S
 THPT100305R: insert screw: SR34-506
 THPT110405R: insert screw: SR14-571/S
 THPT120405R: insert screw: SR14-506
 TPHT130408R: insert screw: SR16-212/L10
 ZPHT060204R: insert screw: SR M2.5X0.35L3.8

torque: 1,1 Nm
 torque: 1,1 Nm
 torque: 1,1 Nm
 torque: 1,1 Nm
 torque: 2,0 Nm
 torque: 3,0 Nm
 torque: 4,5 Nm
 torque: 7,5 Nm
 torque: 0,9 Nm

torque wrench: DTN011S bit: DS-T07TB
 torque wrench: DTN011S bit: DS-T08TB
 torque wrench: DTN011S bit: DS-T08TB
 torque wrench: DTN011S bit: DS-T08TB
 torque wrench: DTN020S bit: DS-T09TB
 torque wrench: DTN030S bit: DS-T09TB
 torque wrench: DTN045F bit: DS-T15B
 torque wrench: DTNV00S bit: DS-T20TB
 torque wrench: DTN011S bit: DS-T07TB

Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.



Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]	Feed per tooth fz [mm]			1st choice grade
			8,00 – 9,99	10,00 – 11,99	12,00 – 12,99	
P	unalloyed steel	50 - 150	0,04 - 0,08	0,05 - 0,10	0,06 - 0,13	IN2005
	alloyed steel 800 N/mm ²	50 - 150	0,04 - 0,08	0,05 - 0,10	0,06 - 0,13	IN2005
	alloyed steel 1100 N/mm ²	20 - 60	0,04 - 0,07	0,05 - 0,09	0,06 - 0,10	IN2005
M	stainless steel	20 - 40	0,04 - 0,06	0,04 - 0,07	0,06 - 0,08	IN2005
K	gray cast iron	50 - 200	0,05 - 0,10	0,06 - 0,13	0,07 - 0,16	IN2005
	nodular cast iron	50 - 200	0,05 - 0,10	0,06 - 0,13	0,07 - 0,16	IN2005
N	aluminum	100 - 200	0,05 - 0,10	0,06 - 0,13	0,07 - 0,16	IN05S
S	high temperature alloys	-	-	-	-	-
	titanium alloys	-	-	-	-	-
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 60 HRC	-	-	-	-	-

Remarks & Tips:

- The indicated data are guide values, which can deviate according to your application.
- Recommended cutting data refer to short holders (3xD effective reaming depth).
- All recommended cutting data refer to machines with sufficient internal coolant supply.
- The coolant must be suitable for reaming. Boring emulsion must consist of minimum 10% – 20% oil.
- Straight fluted heads are mainly used only for blind holes.
- The reaming allowance should be in diameter between 0.07 to 0.15 mm for steel and cast iron, and between 0.07 to 0.18 mm for aluminum and brass, always depending on the diameter.



Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]	Feed per tooth fz [mm]			1st choice rate
			8,00 – 9,99	10,00 – 11,99	12,00 – 12,99	
P	unalloyed steel	50 - 150	0,04 - 0,08	0,05 - 0,10	0,06 - 0,13	IN2005
	alloyed steel 800 N/mm ²	50 - 150	0,04 - 0,08	0,05 - 0,10	0,06 - 0,13	IN2005
	alloyed steel 1100 N/mm ²	20 - 60	0,04 - 0,07	0,05 - 0,09	0,06 - 0,10	IN2005
M	stainless steel	20 - 40	0,04 - 0,06	0,04 - 0,07	0,06 - 0,08	IN2005
K	gray cast iron	50 - 200	0,05 - 0,10	0,06 - 0,13	0,07 - 0,16	IN2005
	nodular cast iron	50 - 200	0,05 - 0,10	0,06 - 0,13	0,07 - 0,16	IN2005
N	aluminum	100 - 200	0,05 - 0,10	0,06 - 0,13	0,07 - 0,16	IN05S
S	high temperature alloys	-	-	-	-	-
	titanium alloys	-	-	-	-	-
H	hard machining < 54 HRC	-	-	-	-	-
	hard machining < 60 HRC	-	-	-	-	-

Remarks & Tips:

- The indicated data are guide values, which can deviate according to your application.
- Recommended cutting data refer to short holders (3xD effective reaming depth).
- All recommended cutting data refer to machines with sufficient internal coolant supply.
- The coolant must be suitable for reaming. Boring emulsion must consist of minimum 10% – 20% oil.
- Left-hand fluted heads are used only for through holes. The left-hand flute pushes the chips forward and avoids damages on the surface quality.
- The reaming allowance should be in diameter between 0.07 to 0.15 mm for steel and cast iron, and between 0.07 to 0.18 mm for aluminum and brass, always depending on the diameter.


Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]		Feed per tooth fz [mm]					1st choice grade
		Conventional	High speed	11,501 – 13,500	13,501 – 16,000	16,001 – 20,000	20,001 – 25,400	25,401 – 32,000	
P	unalloyed steel	8 – 30	60 – 160	0,05 – 0,14	0,05 – 0,16	0,05 – 0,18	0,07 – 0,19	0,07 – 0,20	IN2005
	alloyed steel 800 N/mm ²	5 – 8	20 – 60	0,03 – 0,06	0,03 – 0,08	0,03 – 0,10	0,04 – 0,11	0,04 – 0,13	IN2005
	alloyed steel 1100 N/mm ²	5 – 6	20 – 40	0,03 – 0,06	0,03 – 0,08	0,03 – 0,10	0,04 – 0,11	0,04 – 0,13	IN2005
M	stainless steel	5 – 6	20 – 40	0,03 – 0,06	0,03 – 0,08	0,03 – 0,10	0,04 – 0,11	0,04 – 0,13	IN2005
K	gray cast iron	15 – 25	40 – 140	0,05 – 0,12	0,05 – 0,14	0,05 – 0,17	0,06 – 0,19	0,06 – 0,23	IN2005
	nodular cast iron	10 – 15	40 – 140	0,05 – 0,12	0,05 – 0,14	0,05 – 0,17	0,06 – 0,19	0,06 – 0,23	IN2005
N	aluminum	10 – 30	50 – 300	0,07 – 0,12	0,07 – 0,14	0,07 – 0,16	0,08 – 0,18	0,09 – 0,19	IN05S
S	high temperature alloys	-	-	-	-	-	-	-	-
	titanium alloys	10 – 14	15 – 50	0,03 – 0,06	0,03 – 0,07	0,03 – 0,08	0,04 – 0,10	0,04 – 0,11	IN2005
H	hard machining < 54 HRC	10 – 14	15 – 50	0,06 – 0,11	0,06 – 0,13	0,06 – 0,15	0,10 – 0,17	0,10 – 0,20	IN2005
	hard machining < 60 HRC	10 – 14	15 – 50	0,06 – 0,11	0,06 – 0,13	0,06 – 0,15	0,10 – 0,17	0,10 – 0,20	IN2005

Remarks & Tips:

- The indicated data are guide values, which can deviate according to your application.
- Recommended cutting data refer to short holders (3xD effective reaming depth).
- All recommended cutting data refer to machines with sufficient internal coolant supply.
- The coolant must be suitable for reaming. Boring emulsion must consist of minimum 10% – 20% oil.
- Straight fluted heads are mainly used only for blind holes.
- The reaming allowance should be in diameter between 0.07 to 0.30 mm for steel and cast iron, and between 0.10 to 0.40 mm for aluminum and brass, always depending on the diameter.


Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]		Feed per tooth fz [mm]					1st choice grade
		Conventional	High speed	11,501 – 13,500	13,501 – 16,000	16,001 – 20,000	20,001 – 25,400	25,401 – 32,000	
P	unalloyed steel	8 – 30	60 – 200	0,08 – 0,17	0,08 – 0,19	0,08 – 0,21	0,10 – 0,24	0,10 – 0,27	IN2005
	alloyed steel 800 N/mm ²	5 – 8	20 – 60	0,04 – 0,09	0,04 – 0,11	0,04 – 0,13	0,05 – 0,12	0,07 – 0,17	IN2005
	alloyed steel 1100 N/mm ²	5 – 6	20 – 40	0,04 – 0,09	0,04 – 0,11	0,04 – 0,13	0,05 – 0,12	0,07 – 0,17	IN2005
M	stainless steel	5 – 6	20 – 40	0,04 – 0,09	0,04 – 0,11	0,04 – 0,13	0,05 – 0,12	0,07 – 0,17	IN2005
K	gray cast iron	15 – 25	40 – 140	0,06 – 0,13	0,06 – 0,15	0,06 – 0,18	0,07 – 0,20	0,07 – 0,24	IN2005
	nodular cast iron	10 – 15	40 – 140	0,06 – 0,13	0,06 – 0,15	0,06 – 0,18	0,07 – 0,20	0,07 – 0,24	IN2005
N	aluminum	10 – 30	50 – 300	0,08 – 0,13	0,08 – 0,15	0,08 – 0,17	0,09 – 0,18	0,09 – 0,20	IN05S
S	high temperature alloys	-	-	-	-	-	-	-	-
	titanium alloys	10 – 14	15 – 50	0,04 – 0,08	0,04 – 0,09	0,04 – 0,10	0,05 – 0,12	0,05 – 0,13	IN2005
H	hard machining < 54 HRC	10 – 14	15 – 50	0,05 – 0,09	0,05 – 0,11	0,05 – 0,13	0,09 – 0,16	0,09 – 0,19	IN2005
	hard machining < 60 HRC	10 – 14	15 – 50	0,05 – 0,09	0,05 – 0,11	0,05 – 0,13	0,09 – 0,16	0,09 – 0,19	IN2005

Remarks & Tips:

- The indicated data are guide values, which can deviate according to your application.
- Recommended cutting data refer to short holders (3xD effective reaming depth).
- All recommended cutting data refer to machines with sufficient internal coolant supply.
- The coolant must be suitable for reaming. Boring emulsion must consist of minimum 10% – 20% oil.
- Left-hand fluted heads are used only for through holes. The left-hand flute pushes the chips forward and avoids damages on the surface quality.
- The reaming allowance should be in diameter between 0.07 to 0.30 mm for steel and cast iron, and between 0.10 to 0.40 mm for aluminum and brass, always depending on the diameter.

SINGLE FLUTE GUN DRILL



Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]	Feed rate fu [mm]					1st choice grade
			4 - 5	5,1 - 6	6,1 - 8	8,1 - 10	10,1 - 12	
P	unalloyed steel	70 - 110	0,01 - 0,03	0,03 - 0,05	0,035 - 0,06	0,04 - 0,07	0,05 - 0,10	IN05S / G
	alloyed steel 800 N/mm ²	50 - 90	0,01 - 0,03	0,03 - 0,05	0,035 - 0,06	0,04 - 0,07	0,05 - 0,10	IN05S / G
M	alloyed steel 1100 N/mm ²	40-70	0,01 - 0,03	0,025 - 0,04	0,03 - 0,045	0,035 - 0,05	0,04 - 0,10	IN05S / G
	stainless steel	40 - 80	0,01 - 0,03	0,025 - 0,04	0,03 - 0,045	0,035 - 0,05	0,04 - 0,10	IN05S / G
K	gray cast iron	70 - 110	0,01 - 0,04	0,04 - 0,10	0,05 - 0,12	0,06 - 0,14	0,07 - 0,20	IN05S / G
	nodular cast iron	90 - 115	0,01 - 0,04	0,04 - 0,10	0,05 - 0,12	0,06 - 0,14	0,07 - 0,20	IN05S / G
N	aluminum	80 - 160	0,02 - 0,04	0,02 - 0,17	0,03 - 0,18	0,035 - 0,19	0,04 - 0,25	IN05S / G
S	high temperature alloys	25 - 60	0,01 - 0,03	0,025 - 0,03	0,03 - 0,035	0,03 - 0,04	0,04 - 0,10	IN05S / G
	titanium alloys	25 - 60	0,01 - 0,03	0,025 - 0,03	0,03 - 0,035	0,03 - 0,04	0,04 - 0,10	IN05S / G
H	hard machining < 54 HRC	20 - 50	0,01 - 0,03	0,025 - 0,03	0,03 - 0,035	0,03 - 0,04	0,04 - 0,10	IN05S / G
	hard machining < 60 HRC	-	-	-	-	-	-	-

Remarks & Tips:

- When drill retracts from borehole a disc drops down, so at rotating workpieces a risk of accidents exists!

Please take care for respective safety measures!

- At through holes take care, that the drill head is guided when the drill drops out.
- Drill must always be supplied with enough coolant via internal coolant supply to ensure an optimum chip flow.

Attention risk of accident:

- Allow unguided drills to rotate only at low rotation speed (max. 30-50 rpm), otherwise there is a risk of tool breakage.
- Long tools may need to be supported by bezels, otherwise tool there is a risk of breakage



Ingersoll Werkzeuge GmbH is specialized in the production of cutting tools with an excellent vibration-free performance in both standard and special-purpose design.

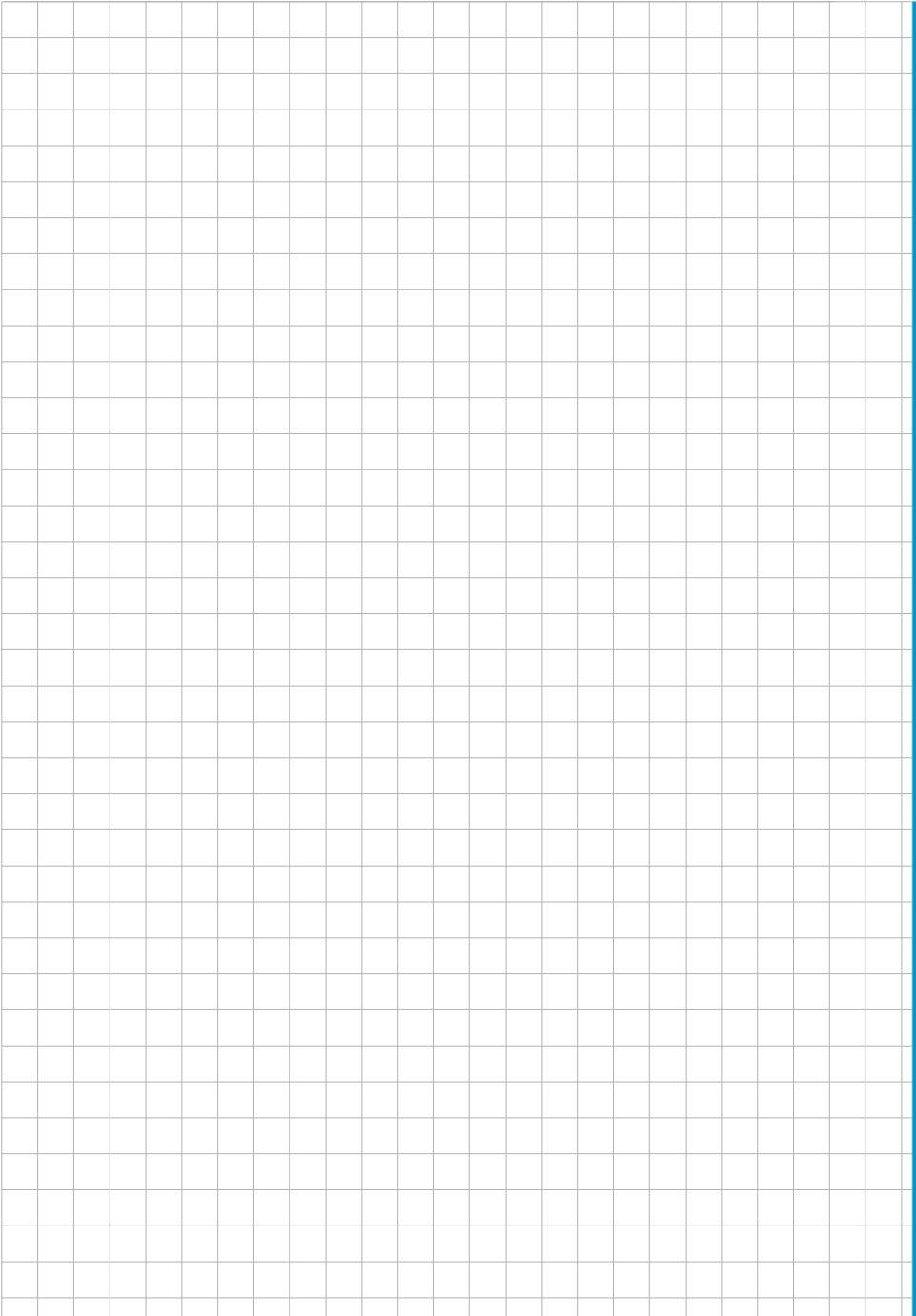
In addition to very successful solutions for heavy-duty milling and the project-oriented development of special cutting tool solutions, we offer a whole range of technology potential which is applied by the most various industries. The close cooperation with our customers for the development of technically demanding solutions for machining problems is the basis of long-term and durable partnerships-worldwide.

Small as well as medium-sized companies but also international enterprises trust our professional qualification and profit from the reliability and process security of our cutting tools.



Note

A large grid of graph paper for taking notes, consisting of 20 columns and 30 rows of small squares. The grid is positioned to the right of a vertical blue bar that runs down the left side of the page.



Ingersoll Cutting Tools
 Marketing & Technology

Germany

Ingersoll Werkzeuge GmbH

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



www.ingersoll-imc.de

