Member IMC Group

Ingersoll Cutting Tools

CUTTING DATA & PARAMETERS

MILLING AND DRILLING

SFEEDUP HIGH SPEED & FEED

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 customer-specific requirements is another important factor for Ingersoll Werkzeuge GmbH.

Our know-how and great potential of experience, combined with our own demand for quality, functionality and innovation, guarantees our customers the optimum cutting tool solution – for individual machining tasks, for all industries.

Content

BORING

General informations

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

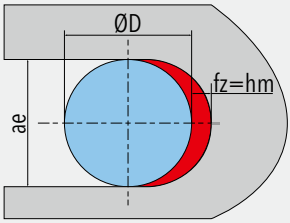
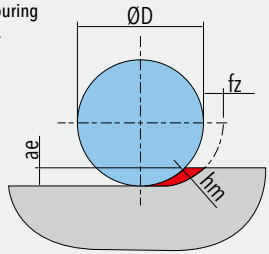


Fig. 2: Contouring

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



General formulas

Cutting data calculation

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

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

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

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

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

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

Grades – Coatings

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

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

Application	Grade	ISO-Group					
Milling	IN2504	P05-P25				H05-H25	harder
	IN2006	P05-P20				H05-H20	
	IN2004	P10-P20		K10-K20		H05-H15	
	IN4010			K10-K30			
	IN2510			K10-K30			
	IN2005	P15-P30	M15-M35	K20-K40	S05-S20		
	IN2505	P15-P30	M15-M35		S05-S20		
	IN4040	P15-P30					
	IN2540	P15-P35					
	IN4015	P20-P30		K30-K50			
	IN2515	P20-P30		K30-K50			
	IN4030	P20-P40	M15-M30		S15-S25		
	IN2530	P20-P40	M15-M30	K20-K40	S15-S25		
	IN6535		M20-M35		S15-S30		
	IN6537	P30-P45	M30-M45	K30-K45			
	IN7035	P20-P40	M20-M35		S15-S30		
	IN4035	P25-P50	M20-M40		S20-S30		
	IN2035	P25-P50	M20-M40		S20-S30		tougher
Drilling	IN2010			K10-K30			harder
	IN6505	P10-P25					
	IN6520	P10-P40					
	IN2505	P20-P40	M20-M40		S05-S20		
	IN2005	P15-P30	M15-M35	K20-K40	S05-S20		tougher
Solid carbide	IN2504	P05-P25				H05-H25	harder
	IN2006	P05-P20				H05-H20	
	IN2205	P20-P40	M20-M40		S20-S40		
	IN2005	P15-P30	M15-M35	K20-K40	S05-S20		tougher



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

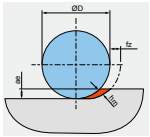
Recommended cutting data:

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
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,06 – 0,10
P	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
K	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
S	titanium alloys	IN2505	40 – 50	IN2035	30 – 40	0,06
H	hard machining < 54 HRC	IN2504	30 – 40	–	–	0,06
H	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	AOCT1705_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

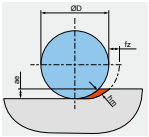
Recommended cutting data:

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	IN4030	250 – 290	IN2530	200 – 240	0,18 – 0,30
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,18 – 0,25
P	alloyed steel 1100 N/mm²	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	IN4030	150 – 200	0,18 – 0,30
K	nodular cast iron	IN2510	140 – 210	IN4030	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
S	titanium alloys	IN2505	40 – 50	IN2035	30 – 40	0,12
H	hard machining < 54 HRC	IN2504	30 – 40	–	–	0,18
H	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**

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

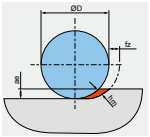
Recommended cutting data:

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/mm²	IN2530	210 – 250	IN2530	120 – 160	0,08 – 0,15
	alloyed steel 1100 N/mm²	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**

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:	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

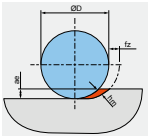
Recommended cutting data:

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/mm²	IN2540	210 – 250	IN2540	120 – 160	0,08 – 0,15
	alloyed steel 1100 N/mm²	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**

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:	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:

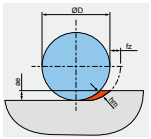
ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN4030	250 – 290	IN2530	200 – 240	0,10 – 0,15
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,10 – 0,12
P	alloyed steel 1100 N/mm²	IN2505	160 – 180	IN2530	110 – 130	0,10
M	stainless steel	IN2035	120 – 180	IN2035	80 – 130	0,10 – 0,12
K	gray cast iron	IN2504	180 – 250	IN4030	150 – 200	0,10 – 0,15
K	nodular cast iron	IN2504	140 – 210	IN4030	110 – 160	0,10 – 0,12
N	aluminum	IN90D	800 – 1500	IN10K	500 – 800	0,05 – 0,20
S	high temperature alloys	IN2035	110 – 125	IN2035	60 – 80	0,10
S	titanium alloys	IN2505	40 – 50	IN2035	30 – 40	0,10
H	hard machining < 54 HRC	IN2504	70 – 220*	–	–	0,05 – 0,10
H	hard machining < 63 HRC	IN2504	50 – 200*	–	–	0,05 – 0,10

*depending on the cutting width æ

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** insert screw from Ø 20: **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:	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:

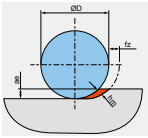
ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN4030	250 – 290	IN2530	200 – 240	0,12 – 0,20
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,12 – 0,15
P	alloyed steel 1100 N/mm²	IN2505	160 – 180	IN2530	110 – 130	0,12
M	stainless steel	IN2035	120 – 180	IN2035	80 – 130	0,12 – 0,20
K	gray cast iron	IN2504	180 – 250	IN4030	150 – 200	0,12 – 0,20
K	nodular cast iron	IN2504	140 – 210	IN4030	110 – 160	0,12 – 0,15
N	aluminum	IN90D	800 – 1500	IN10K	500 – 800	0,12 – 0,20
S	high temperature alloys	IN2035	110 – 125	IN2035	60 – 80	0,12
S	titanium alloys	IN2505	40 – 50	IN2035	30 – 40	0,12
H	hard machining < 54 HRC	IN2504	70 – 220*	–	–	0,05 – 0,10
H	hard machining < 63 HRC	IN2504	50 – 200*	–	–	0,05 – 0,10

* depending on the cutting width æ

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:	DFM213R_	DPM314_	DPM324_	DPM424_	DPM434_
Average chip thickness:	hm = 0,12 mm	hm = 0,15 mm	hm = 0,18 mm	hm = 0,20 mm	hm = 0,22 mm
max. cutting depth:	ap = 7,5 mm	ap = 9,5 mm	ap = 11 mm	ap = 14 mm	ap = 17 mm

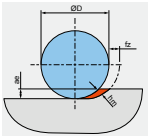
Recommended cutting data:

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	IN2040	150 – 250	IN2030	120 – 200	hm x 1,2
P	alloyed steel 800 N/mm²	IN2005	120 – 180	IN2030	100 – 160	hm x 1,0
P	alloyed steel 1100 N/mm²	IN2005	100 – 180	IN2030	80 – 160	hm x 0,9
M	stainless steel	IN2030	80 – 160	IN2030	80 – 160	hm x 1,2
K	gray cast iron	IN6515	160 – 250	IN2015	140 – 200	hm x 1,2
K	nodular cast iron	IN6515	120 – 200	IN2015	100 – 180	hm x 1,0
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN2030	50 – 80	IN2030	40 – 70	hm x 0,9
S	titanium alloys	–	–	IN2030	30 – 40	hm x 1,0
H	hard machining < 54 HRC	–	–	–	–	–
H	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 213: **SM30-090-10** torque: **2 Nm** torque wrench: **DTN020S with bit DS-T09TB**

insert screw size 314/324: **SM40-120-20** torque: **4,5 Nm** torque wrench: **DTN045F with bit DS-T15B**

insert screw size 424/434/436: **SM50-160-10** torque: **6 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:	DGM212R_	DGM212-1_	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

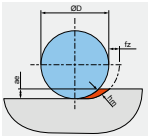
Recommended cutting data:

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	IN4040	150 – 250	IN4030	120 – 200	hm x 1,2
P	alloyed steel 800 N/mm²	IN2505	120 – 180	IN4030	100 – 160	hm x 1,0
P	alloyed steel 1100 N/mm²	IN2505	100 – 180	IN4030	80 – 160	hm x 0,9
M	stainless steel	IN4035	80 – 160	IN4035	80 – 160	hm x 1,2
K	gray cast iron	IN4015	160 – 250	IN4030	140 – 200	hm x 1,2
K	nodular cast iron	IN4015	120 – 200	IN4030	100 – 180	hm x 1,0
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN4035	50 – 80	IN4035	40 – 70	hm x 0,9
S	titanium alloys	–	–	IN4030	30 – 40	hm x 1
H	hard machining < 54 HRC	IN2004	60 – 200	–	–	hm x 0,8
H	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 212: **SM30-082-20** torque: **2 Nm** torque wrench: **DS-TP08S 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:	DGM313R3
Average chip thickness:	hm = 0,15 mm
max. cutting depth:	ap = 9,7 mm

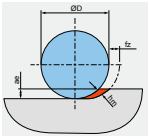
Recommended cutting data:

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	IN4040	150 – 250	IN4030	120 – 200	hm x 1,2
P	alloyed steel 800 N/mm²	IN2505	120 – 180	IN4030	100 – 160	hm x 1,0
P	alloyed steel 1100 N/mm²	IN2505	100 – 180	IN4030	80 – 160	hm x 0,9
M	stainless steel	IN4035	80 – 160	IN4035	80 – 160	hm x 1,2
K	gray cast iron	IN2505	160 – 250	IN4015	140 – 200	hm x 1,2
K	nodular cast iron	IN2505	120 – 200	IN4015	100 – 180	hm x 1,0
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN4035	50 – 80	IN4035	40 – 70	hm x 0,9
S	titanium alloys	–	–	IN4035	30 – 40	hm x 1
H	hard machining < 54 HRC	–	–	–	–	–
H	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 313: **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:	DGM314R0_	DGM324R0_	ZGM324R2_
Average chip thickness:	hm = 0,17 mm	hm = 0,20 mm	hm = 0,20 mm
max. cutting depth:	ap = 10,7 mm	ap = 13 mm	ap = 13 mm

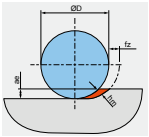
Recommended cutting data:

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	IN2040	150 – 250	IN2030	120 – 200	hm x 1,2
P	alloyed steel 800 N/mm²	IN2005	120 – 180	IN2030	100 – 160	hm x 1,0
P	alloyed steel 1100 N/mm²	IN2005	100 – 180	IN2030	80 – 160	hm x 0,9
M	stainless steel	IN2035	80 – 160	IN2035	80 – 160	hm x 1,2
K	gray cast iron	IN6515	160 – 250	IN2015	140 – 200	hm x 1,2
K	nodular cast iron	IN6515	120 – 200	IN2015	100 – 180	hm x 1,0
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN2035	50 – 80	IN2035	40 – 70	hm x 0,9
S	titanium alloys	–	–	IN2030	30 – 40	hm x 1
H	hard machining < 54 HRC	–	–	–	–	–
H	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 314: **SM35-114-H0** torque: **3 Nm** torque wrench: **DTN030S with bit DS-T15TB**

insert screw size 324: **SM40-143-H0** 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:	DGM324R3_	ZGM324R3_
Average chip thickness:	hm = 0,20 mm	hm = 0,20 mm
max. cutting depth:	ap = 13 mm	ap = 13 mm

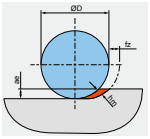
Recommended cutting data:

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	IN4040	150 – 250	IN4030	120 – 200	hm x 1,2
P	alloyed steel 800 N/mm²	IN2505	120 – 180	IN4030	100 – 160	hm x 1,0
P	alloyed steel 1100 N/mm²	IN2505	100 – 180	IN4030	80 – 160	hm x 0,9
M	stainless steel	IN4035	80 – 160	IN4035	80 – 160	hm x 1,2
K	gray cast iron	IN2505	160 – 250	IN4015	140 – 200	hm x 1,2
K	nodular cast iron	IN2505	120 – 200	IN4015	100 – 180	hm x 1,0
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN4035	50 – 80	IN4035	40 – 70	hm x 0,9
S	titanium alloys	–	–	IN4035	30 – 40	hm x 1
H	hard machining < 54 HRC	–	–	–	–	–
H	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 324: **SM40-143-H0** 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:	DGM315-_	DGM325-_	DGM426-_
Average chip thickness:	hm = 0,15 mm	hm = 0,17 mm	hm = 0,20 mm
max. cutting depth:	ap = 10,5 mm	ap = 12,7 mm	ap = 15 mm

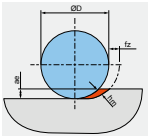
Recommended cutting data:

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	IN2040	150 – 250	IN2030	120 – 200	hm x 1,2
P	alloyed steel 800 N/mm²	IN2005	120 – 180	IN2030	100 – 160	hm x 1,0
P	alloyed steel 1100 N/mm²	IN2005	100 – 180	IN2030	80 – 160	hm x 0,9
M	stainless steel	IN2035	80 – 160	IN2035	80 – 160	hm x 1,2
K	gray cast iron	IN6515	160 – 250	IN2015	140 – 200	hm x 1,2
K	nodular cast iron	IN6515	120 – 200	IN2015	100 – 180	hm x 1,0
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN2035	50 – 80	IN2035	40 – 70	hm x 0,9
S	titanium alloys	–	–	IN2030	30 – 40	hm x 1,0
H	hard machining < 54 HRC	–	–	–	–	–
H	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 315/325: **SM40-143-H0** torque: **4,5 Nm** torque wrench: **DTN045F with bit DS-T15B**

insert screw size 426: **SM50-160-10** 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:	DNM434R201	DNM434R202	DNM434R204	DNM434R245
Average chip thickness:	hm = 0,35 mm	hm = 0,35 mm	hm = 0,35 mm	hm = 0,35 mm
max. cutting depth:	ap = 17 mm	ap = 16 mm	ap = 14,5 mm	ap = 8,8 mm

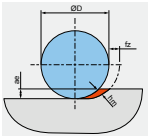
Recommended cutting data:

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	150 – 250	IN2530	120 – 200	hm x 1,2
P	alloyed steel 800 N/mm²	IN2505	120 – 180	IN2530	100 – 160	hm x 1,0
P	alloyed steel 1100 N/mm²	IN2505	100 – 180	IN2540	80 – 160	hm x 0,9
M	stainless steel	IN2505	80 – 160	IN2540	80 – 160	hm x 1,0
K	gray cast iron	IN2505	160 – 250	IN2515	140 – 200	hm x 1,2
K	nodular cast iron	IN2505	120 – 200	IN2515	100 – 180	hm x 1,0
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN2530	50 – 80	IN2530	40 – 70	hm x 0,9
S	titanium alloys	–	–	IN2530	30 – 40	hm x 1
H	hard machining < 54 HRC	–	–	–	–	–
H	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: **SM50-160-10** 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:	DPD_-S	DNE_-S
Average chip thickness:	hm = 0,20 mm	hm = 0,25 mm

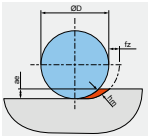
Recommended cutting data:

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	IN4005	140 – 200	IN4030	120 – 180	hm x 1,2
P	alloyed steel 800 N/mm²	IN4005	120 – 160	IN4030	100 – 140	hm x 1,0
P	alloyed steel 1100 N/mm²	IN4005	100 – 160	IN4030	100 – 140	hm x 0,9
M	stainless steel	IN4005	80 – 160	IN4030	80 – 140	hm x 1,0
K	gray cast iron	IN4005	150 – 200	IN4030	130 – 180	hm x 1,2
K	nodular cast iron	IN4015	120 – 160	IN4030	100 – 140	hm x 1,0
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN4005	60 – 100	IN4030	40 – 80	hm x 0,8
S	titanium alloys	–	–	IN4003	30 – 60	hm x 0,8
H	hard machining < 54 HRC	–	–	–	–	–
H	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: **SM40-140-R0** torque: **4,5 Nm** torque wrench: **DT40-01 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:	DPM436R101(T12)	DPM436R045	DPM436R046	DPM436R105(T11)	DPM436R060
Average chip thickness:	hm = 0,50 (0,70) mm	hm = 0,50 mm	hm = 0,50 mm	hm = 0,50 (0,70) mm	hm = 0,50 mm
max. cutting depth:	ap up to 20 mm	ap up to 13 mm	ap up to 13 mm	ap up to 3 mm	ap up to 23 mm

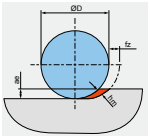
Recommended cutting data:

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	IN2540	150 – 250	IN2530	120 – 200	hm x 1,2
	alloyed steel 800 N/mm²	IN2505	120 – 180	IN2530	100 – 160	hm x 1,0
	alloyed steel 1100 N/mm²	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	IN2515	140 – 200	hm x 1,2
	nodular cast iron	IN2515	120 – 200	IN2515	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 size 436: **SM50-160-10** torque: **6 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:	FNKT0502_R-M	FNHT0502_R-L
Average chip thickness:	hm = 0,04 mm	hm = 0,03 mm
max. cutting depth:	ap = 5,0 mm	ap = 5,0 mm

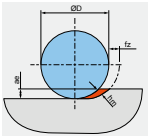
Recommended cutting data:

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	0,04 – 0,08
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,04 – 0,06
P	alloyed steel 1100 N/mm²	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
K	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
S	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,04
H	hard machining < 54 HRC	–	–	–	–	–
H	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 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:	HNGU0605ANTR-M	HNGU0605ANTR-MM
Average chip thickness:	hm = 0,15 mm	hm = 0,20 mm
max. cutting depth:	ap = 3,0 mm	ap = 3,0 mm

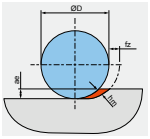
Recommended cutting data:

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
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN6537	160 – 200	0,15 – 0,20
P	alloyed steel 1100 N/mm²	IN2505	160 – 180	IN6537	110 – 130	0,15
M	stainless steel	IN2505	120 – 180	IN6537	80 – 130	0,15 – 0,20
K	gray cast iron	IN2510	180 – 250	IN6537	150 – 200	0,15 – 0,35
K	nodular cast iron	IN2510	140 – 210	IN6537	110 – 160	0,15 – 0,20
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN2505	110 – 125	IN6537	60 – 80	0,15
S	titanium alloys	IN2505	40 – 50	IN6537	30 – 40	0,15
H	hard machining < 54 HRC	–	–	–	–	–
H	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**

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

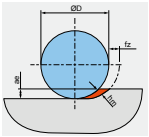
Recommended cutting data:

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	IN6537	200 – 240	0,20 – 0,30
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN6537	160 – 200	0,20 – 0,25
P	alloyed steel 1100 N/mm²	IN2505	160 – 180	IN6537	110 – 130	0,20
M	stainless steel	IN2505	120 – 180	IN6537	80 – 130	0,20 – 0,25
K	gray cast iron	IN6510	180 – 250	IN2010	150 – 200	0,20 – 0,70
K	nodular cast iron	IN2510	140 – 210	IN2010	110 – 160	0,20 – 0,45
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN2505	110 – 125	IN6537	60 – 80	0,20
S	titanium alloys	IN2505	40 – 50	IN6537	30 – 40	0,20
H	hard machining < 54 HRC	–	–	–	–	–
H	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**

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:	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

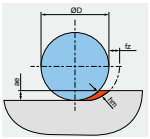
Recommended cutting data:

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/mm²	IN2505	120 – 180	IN2530	100 – 160	hm x 1,0
	alloyed steel 1100 N/mm²	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	IN2515	160 – 250	IN2515	140 – 200	hm x 1,2
	nodular cast iron	IN2515	120 – 200	IN2515	100 – 180	hm x 1,0
N	aluminum	IN05S	500 – 1200	IN05S	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**	torque: **0,7 Nm**	torque wrench: **DTNV01S with bit DS-T06TB**
insert screw size 311: **SM35-034-50**	torque: **2,0 Nm**	torque wrench: **DTN020S with bit DS-T09TB**
insert screw size 312: **SM35-042-50**	torque: **2,0 Nm**	torque wrench: **DTN020S with bit DS-T09TB**
insert screw size 412: **SM40-040-50**	torque: **4,5 Nm**	torque wrench: **DTN045F with bit DS-T15B**
insert screw size 413: **SM40-070-50**	torque: **4,5 Nm**	torque wrench: **DTN045F with bit DS-T15B**
insert screw size 414: **SM40-080-50**	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.

GOLDSLOT IXH415 / IXH416


Insert:	IXH415	IXH416-MM
Average chip thickness:	hm = 0,15 mm	hm = 0,15 mm

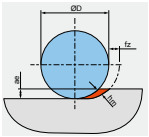
Recommended cutting data:

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	IN4005	150 – 250	IN4030	120 – 200	hm x 1,2
	alloyed steel 800 N/mm²	IN4005	120 – 180	IN4030	100 – 160	hm x 1,0
	alloyed steel 1100 N/mm²	IN4005	100 – 180	IN4030	80 – 160	hm x 0,9
M	stainless steel	IN4035	80 – 160	IN4035	80 – 160	hm x 1,2
K	gray cast iron	IN4030	160 – 250	IN4030	140 – 200	hm x 1,2
	nodular cast iron	IN4030	120 – 200	IN4030	100 – 180	hm x 1,0
N	aluminum	IN05S	500 – 1200	IN05S	400 – 800	hm x 1,3
S	high temperature alloys	IN4005	50 – 80	IN4030	40 – 70	hm x 0,9
	titanium alloys	–	–	IN4005	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 415: **SM40-090-00** torque: **4,5 Nm** torque wrench: **DTN045F with bit DS-T15B**

insert screw size 416: **SM40-110-00** 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:	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
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,06 – 0,08
P	alloyed steel 1100 N/mm²	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
K	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
S	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,06
H	hard machining < 54 HRC	–	–	–	–	–
H	hard machining < 63 HRC	–	–	–	–	–

Ramping data and circular interpolation:

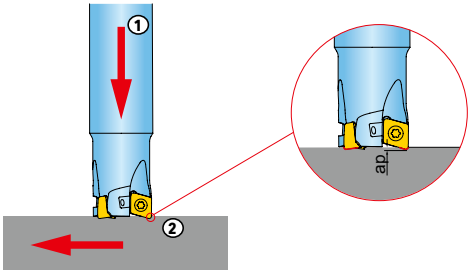
tool diameter [mm]	max. ramping angle [°]	min. bore dia. uneven ground [mm]	max. ap/rev. [mm]	max. bore dia. even ground [mm]	max. ap/rev. [mm]
10	4,2	15,4	1,2	19,2	2,1
11	4,1	17,1	1,3	21,2	2,2
12	4,0	18,8	1,4	23,2	2,4
13	3,9	20,5	1,6	25,2	2,6
16	4,6	25,1	2,3	31,3	3,5
20	3,1	33,0	2,2	39,2	3,2
25	2,4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

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

Step-down-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: **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:	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	IN2530	200 – 240	0,07 – 0,15
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,07 – 0,10
P	alloyed steel 1100 N/mm²	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	IN2505	180 – 250	IN2530	150 – 200	0,07 – 0,15
K	nodular cast iron	IN2505	140 – 210	IN2530	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
S	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,05
H	hard machining < 54 HRC	–	–	–	–	–
H	hard machining < 63 HRC	–	–	–	–	–

Ramping data and circular interpolation:

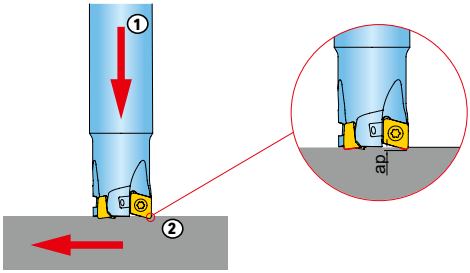
tool diameter [mm]	max. ramping angle [°]	min. bore dia. uneven ground [mm]	max. ap/rev. [mm]	max. bore dia. even ground [mm]	max. ap/rev. [mm]
16	3,5	21,5	0,8	32,0	2,5
20	4,2	29,5	1,8	40,0	3,7
25	4,9	39,5	3,1	50,0	5,4
32	3,5	53,5	3,3	64,0	4,9
40	2,6	69,5	3,4	80,0	4,6
50	2,0	89,5	3,5	100,0	4,4
63	1,5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

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

Step-down-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
Average chip thickness:	hm = 0,10 mm
max. cutting depth:	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	IN2530	200 – 240	0,10 – 0,15
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,10 – 0,12
P	alloyed steel 1100 N/mm²	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 – 0,15
K	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
S	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,10
H	hard machining < 54 HRC	–	–	–	–	–
H	hard machining < 63 HRC	–	–	–	–	–

Ramping data and circular interpolation:

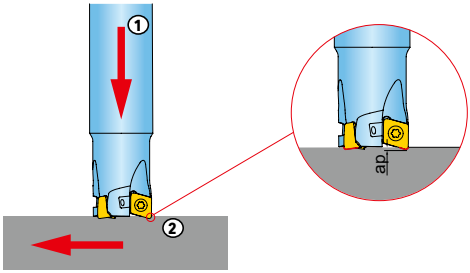
tool diameter [mm]	max. ramping angle [°]	min. bore dia. uneven ground [mm]	max. ap/rev. [mm]	max. bore dia. even ground [mm]	max. ap/rev. [mm]
20	3,5	32	2,3	38,5	3,5
25	4,7	38,1	3,3	48,4	6,0
32	4,9	48,4	4,4	62,4	8,0
40	3,7	64,1	4,8	78,4	7,8
50	2,8	83,9	5,2	98,4	7,4
63	2,1	109,7	5,3	124,4	7,0
80	1,6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

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

Step-down-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
Average chip thickness:	hm = 0,10 mm	hm = 0,10 mm
max. cutting depth:	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	IN2530	200 – 240	0,10 – 0,15
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,10 – 0,12
P	alloyed steel 1100 N/mm²	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 – 0,15
K	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
S	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,10
H	hard machining < 54 HRC	–	–	–	–	–
H	hard machining < 63 HRC	–	–	–	–	–

Ramping data and circular interpolation:

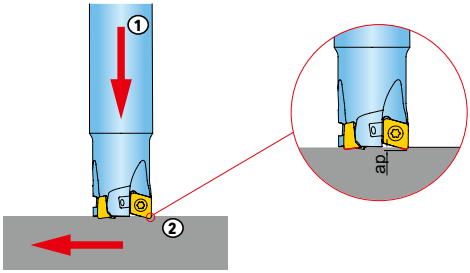
tool diameter [mm]	max. ramping angle [°]	min. bore dia. uneven ground [mm]	max. ap/rev. [mm]	max. bore dia. even ground [mm]	max. ap/rev. [mm]
25	5,0	37,7	3,4	48,5	6,4
32	5,3	47,8	4,6	62,5	8,8
40	5,0	60,3	5,5	78,5	10,5
50	3,8	80,2	6,3	98,5	10,1
63	2,9	105,8	6,8	124,5	9,7
80	2,2	139,6	7,1	158,4	9,4
100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

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

Step-down-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
Average chip thickness:	hm = 0,13 mm	hm = 0,13 mm
max. cutting depth:	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	IN2530	200 – 240	0,13 – 0,18
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,13 – 0,15
P	alloyed steel 1100 N/mm²	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	IN2530	150 – 200	0,13 – 0,18
K	nodular cast iron	IN2510	140 – 210	IN2530	110 – 160	0,13 – 0,15
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN2505	110 – 125	IN2530	60 – 80	0,13
S	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,13
H	hard machining < 54 HRC	–	–	–	–	–
H	hard machining < 63 HRC	–	–	–	–	–

Ramping data and circular interpolation:

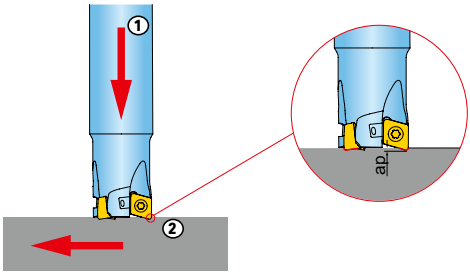
tool diameter [mm]	max. ramping angle [°]	min. bore dia. uneven ground [mm]	max. ap/rev. [mm]	max. bore dia. even ground [mm]	max. ap/rev. [mm]
50	5,4	74,5	7,2	98,5	13,5
63	3,9	100,3	7,9	124,5	13,1
80	2,9	134,3	8,6	158,5	12,4
100	2,2	174,2	8,9	198,4	11,8
125	1,7	224,2	9,2	248,4	11,5

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

Step-down-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.

VMAX NNE_ / NJE_ / YNE_ / NYE_





Insert:	NNE313	NNE324	NCE324	NJE	YNE324
Average chip thickness:	hm = 0,12 mm	hm = 0,20 mm	hm = 0,15 mm	hm = 0,10 mm	hm = 0,15 mm

Insert:	NYE324
Average chip thickness:	hm = 0,10 mm

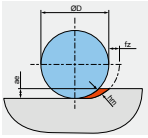
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				mittlere Spanstärke hm [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2040	150 – 250	IN2030	120 – 200	hm x 1,2
	alloyed steel 800 N/mm²	IN2005	120 – 180	IN2030	100 – 160	hm x 1,0
	alloyed steel 1100 N/mm²	IN2005	100 – 180	IN2030	80 – 160	hm x 0,9
M	stainless steel	IN2030	80 – 160	IN2030	80 – 160	hm x 1,2
K	gray cast iron	IN6515	160 – 250	IN2015	140 – 200	hm x 1,2
	nodular cast iron	IN2015	120 – 200	IN2015	100 – 180	hm x 1,0
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN2030	50 – 80	IN2030	40 – 70	hm x 0,9
	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 size 313: **SM30-082-00** torque: **2,0 Nm** torque wrench: **DTN020S with bit DS-T09TB**

insert screw size 324: **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:	OFMT05T3AFN-HR	OFCT05T3AFTN-HR	OFCT05T3TN-HS	OFMW05T3AFTN	OFCT05T3AFFN-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

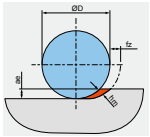
Recommended cutting data:

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
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,12 – 0,20
P	alloyed steel 1100 N/mm²	IN2505	160 – 180	IN2530	110 – 130	0,12
M	stainless steel	IN6535	120 – 180	IN2035	80 – 130	0,12 – 0,25
K	gray cast iron	IN2510	180 – 250	IN4030	150 – 200	0,20 – 0,40
K	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	IN6535	110 – 125	IN2035	60 – 80	0,12
S	titanium alloys	IN2505	40 – 50	IN2035	30 – 40	0,12
H	hard machining < 54 HRC	IN2505	30 – 40	–	–	0,20
H	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. even 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**

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.

ROTOMILL OFMT07_ / OFCT07_ / OFMW07_





Insert:	OFMT0705AFR-HR	OFCT0705AFTN-HR	OFMT0705AFTN	OFMW0705AFTN	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

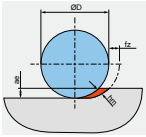
Recommended cutting data:

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
P	alloyed steel 800 N/mm²	IN4005	210 – 250	IN4030	160 – 200	0,15 – 0,25
P	alloyed steel 1100 N/mm²	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
K	nodular cast iron	IN4005	140 – 210	IN4030	110 – 160	0,25 – 0,40
N	aluminum	IN05S	800 – 1500	IN05S	500 – 800	0,05 – 0,30
S	high temperature alloys	IN6535	110 – 125	IN2035	60 – 80	0,18
S	titanium alloys	IN4005	40 – 50	IN2035	30 – 40	0,18
H	hard machining < 54 HRC	IN4005	30 – 40	–	–	0,25
H	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. even 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: **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:	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

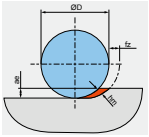
Recommended cutting data:

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	IN4030	250 – 290	IN4030	200 – 240	0,22 – 0,40
	alloyed steel 800 N/mm²	IN2505	210 – 250	IN4030	160 – 200	0,22 – 0,30
	alloyed steel 1100 N/mm²	IN2505	160 – 180	IN4030	110 – 130	0,22
M	stainless steel	IN6535	120 – 180	IN2035	80 – 130	0,08 – 0,30
K	gray cast iron	IN70N	600 – 900	IN4030	150 – 200	0,08 – 0,40
	nodular cast iron	IN4010	140 – 120	IN4030	110 – 160	0,22 – 0,30
N	aluminum	IN10K	800 – 1500	IN10K	500 – 800	0,05 – 0,30
S	high temperature alloys	IN6535	110 – 125	IN2035	60 – 80	0,08
	titanium alloys	IN2505	40 – 50	IN2035	30 – 40	0,22
H	hard machining < 54 HRC	IN2505	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**

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:	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

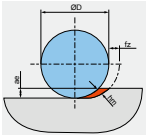
Recommended cutting data:

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	IN4030	200 – 240	0,30 – 0,40
P	alloyed steel 800 N/mm²	IN4005	210 – 250	IN4030	160 – 200	0,30 – 0,35
P	alloyed steel 1100 N/mm²	IN4005	160 – 180	IN4030	110 – 130	0,30
M	stainless steel	IN2035	120 – 180	IN2035	80 – 130	0,08 – 0,30
K	gray cast iron	IN70N	600 – 900	IN4030	150 – 200	0,08 – 0,40
K	nodular cast iron	IN4010	140 – 210	IN4030	110 – 160	0,30 – 0,35
N	aluminum	IN10K	800 – 1500	IN10K	500 – 800	0,05 – 0,40
S	high temperature alloys	IN2035	110 – 125	IN2035	60 – 80	0,08
S	titanium alloys	IN4005	40 – 50	IN2035	30 – 40	0,30
H	hard machining < 54 HRC	IN4005	30 – 40	–	–	0,30
H	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 clamping by screw:

insert screw: **SM50-130-R0** torque: **6,0 Nm** torque wrench: **DTNV00S with bit DS-T20TB**

for clamping by wedge:

insert screw: **SB080-03** torque: **6,0 Nm** torque wrench: **DTNV00S with bit DS-H04TB**

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.

DIPOSDEKA® PNCU0503GNFR-P / PNMU0503GNTR


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

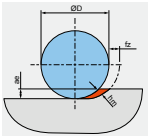
Recommended cutting data:

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
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,10 – 0,20
P	alloyed steel 1100 N/mm²	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
K	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
S	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,10
H	hard machining < 54 HRC	–	–	–	–	–
H	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**

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:	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

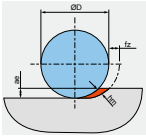
Recommended cutting data:

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	IN4030	250 – 290	IN4030	200 – 240	0,20 – 0,40
P	alloyed steel 800 N/mm²	IN4005	210 – 250	IN4030	160 – 200	0,20 – 0,30
P	alloyed steel 1100 N/mm²	IN4005	160 – 180	IN4030	110 – 130	0,20
M	stainless steel	IN2035	120 – 180	IN2035	80 – 130	0,08 – 0,30
K	gray cast iron	IN70N	600 – 900	IN4030	150 – 200	0,10 – 0,40
K	nodular cast iron	IN4015	140 – 210	IN4030	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
S	titanium alloys	IN2505	40 – 50	IN2035	30 – 40	0,20
H	hard machining < 54 HRC	IN2505	30 – 40	–	–	0,20
H	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**

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:	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

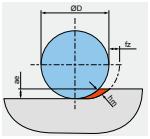
Recommended cutting data:

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
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2505	160 – 200	0,06 – 0,10
P	alloyed steel 1100 N/mm²	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
K	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
S	titanium alloys	IN2505	40 – 50	IN2505	30 – 40	0,06
H	hard machining < 54 HRC	IN2505	30 – 40	–	–	0,06
H	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**

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:	SDE-_	SEE-_
Average chip thickness:	hm = 0,13 mm	hm = 0,05 mm
max. Cutting depth:	4 – 15	4 – 15

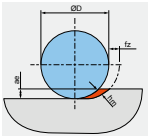
Recommended cutting data:

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
P	alloyed steel 800 N/mm²	IN4030	210 – 250	IN4030	160 – 200	0,13 – 0,20
P	alloyed steel 1100 N/mm²	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
K	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
S	titanium alloys	IN4030	40 – 50	IN4030	30 – 40	0,13
H	hard machining < 54 HRC	IN4030	30 – 40	–	–	0,13
H	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 depth 4 mm: **SM35-034-50**	torque: **1,4 Nm**	torque wrench: **DTNV02S with bit DS-T09B**
insert screw for cutting depth 5 mm: **SM35-042-50**	torque: **1,4 Nm**	torque wrench: **DTNV02S with bit DS-T09B**
insert screw for cutting depth 6 mm: **SM40-050-50**	torque: **4,5 Nm**	torque wrench: **DTN045F with bit DS-T15B**
insert screw for cutting depth 7 – 8 mm: **SM40-060-50**	torque: **4,5 Nm**	torque wrench: **DTN045F with bit DS-T15B**
insert screw for cutting depth 9 – 10 mm: **SM40-080-50**	torque: **4,5 Nm**	torque wrench: **DTN045F with bit DS-T15B**
insert screw for cutting depth 12 – 15 mm: **SM40-106-50**	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:	SDES0904_TN	SDES0904_TN-002	SDXS0904_FN-HR
Average chip thickness:	hm = 0,11 mm	hm = 0,11 mm	hm = 0,08 mm
max. cutting depth:	ap = 9 mm	ap = 9 mm	ap = 8,5 mm

Recommended cutting data:

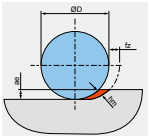
ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN4030	250 – 290	IN4030	200 – 240	0,11 – 0,20
P	alloyed steel 800 N/mm²	IN4030	210 – 250	IN4030	160 – 200	0,11 – 0,15
P	alloyed steel 1100 N/mm²	IN4030	160 – 180	IN4030	110 – 130	0,11
M	stainless steel	IN2035	120 - 180	IN2035	80 – 130	0,08 – 0,20
K	gray cast iron	IN4030	180 – 250	IN4030	150 – 200	0,11 – 0,20
K	nodular cast iron	IN4030	140 – 210	IN4030	110 – 160	0,11 – 0,20
N	aluminum	IN2035	800 – 1500	IN2505	500 – 800	0,08 – 0,20
S	high temperature alloys	IN2035	110 – 125	IN2035	60 – 80	0,08 – 0,20
S	titanium alloys	IN2035	40 – 50	IN2035	30 – 40	0,08 – 0,20
H	hard machining < 54 HRC	IN4030	30 – 40	–	–	0,11
H	hard machining < 63 HRC	–	–	–	–	–

When using T-slot cutters, a start value of Vc=80 m/min is usually best for material groups P, M and K. Ensure optimum chip evacuation.

Tips:

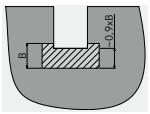
- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Application example T-slot mill:

Under normal conditions, leave the head of the T-slot largely unmachined initially. With a powerful supply of coolant or compressed air, chip removal is usually better then.



General information:

insert screw: **SM30-075-R0** torque: **2,0 Nm** torque wrench: **DTN020S 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.

GOLDQUAD SDES13_ / SDMS13_ / SDXS13_




Insert:	SDES1305_N	SDES1305_N-001	SDMS1305_R-PH	SDXS130515N-HR
Average chip thickness:	hm = 0,20 mm	hm = 0,08 mm	hm = 0,10 mm	hm = 0,18 mm
max. cutting depth:	ap = 11,3 mm	ap = 11,3 mm	ap = 11,3 mm	ap = 11,3 mm

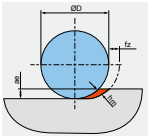
Recommended cutting data:

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
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,10 – 0,30
P	alloyed steel 1100 N/mm²	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
K	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	IN2035	60 – 80	0,08 – 0,18
S	titanium alloys	IN4035	40 – 50	IN2035	30 – 40	0,08 – 0,18
H	hard machining < 54 HRC	IN2505	30 – 40	–	–	0,08
H	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.

HIPOSQUAD SDMT08_ / SDMW08_ / SDCT08_




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:

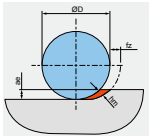
ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN4030	250 – 290	IN4030	200 – 240	0,13 – 0,20
P	alloyed steel 800 N/mm²	IN4030	210 – 250	IN4030	160 – 200	0,13 – 0,15
P	alloyed steel 1100 N/mm²	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
K	nodular cast iron	IN4030	140 – 210	IN4030	110 – 160	0,13 – 0,15
N	aluminum	IN05S	800 – 1500	IN05S	500 – 800	0,05 – 0,20
S	high temperature alloys	IN2505	110 – 125	IN2505	60 – 80	0,13
S	titanium alloys	IN2505	40 – 50	IN2505	30 – 40	0,13
H	hard machining < 54 HRC	IN2505	30 – 40	–	–	0,13
H	hard machining < 63 HRC	–	–	–	–	–

**When using T-slot cutters, a start value of Vc=80 m/min is usually best for material groups P, M and K. Ensure optimum chip evacuation.*

Tips:

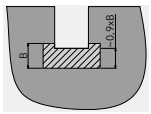
- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Application example T-slot mill:

Under normal conditions, leave the head of the T-slot largely unmachined initially. With a powerful supply of coolant or compressed air, chip removal is usually better then.



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: **DTN020S 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:	SGM-44R001	SGM-44R100
Average chip thickness:	hm = 0,18 mm	hm = 0,18 mm
max. cutting depth:	ap = 8,7 mm	ap = 7 mm

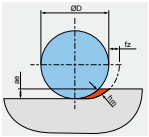
Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				mittlere Spanstärke 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
P	alloyed steel 800 N/mm²	IN4005	120 – 180	IN4030	100 – 160	hm x 1,0
P	alloyed steel 1100 N/mm²	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
K	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
S	titanium alloys	–	–	IN2530	30 – 40	hm x 1,0
H	hard machining < 54 HRC	–	–	–	–	–
H	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**

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:	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
	> 1% Pb copper alloys	well machinable	110	IN10K	200 – 800	0,05 – 0,10
		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. revolutions per minute:

tool diameter [mm]	max. no. of revolutions n [rpm]	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 inserts.
- 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

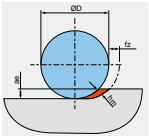
Recommended cutting data:

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
P	alloyed steel 800 N/mm²	IN4005	210 – 250	IN4030	160 – 200	0,20 – 0,30
P	alloyed steel 1100 N/mm²	IN4005	160 – 180	IN4030	110 – 130	0,20
M	stainless steel	IN2035	120 – 180	IN2035	80 – 130	0,20 – 0,35
K	gray cast iron	IN70N	600 – 900	IN4030	150 – 200	0,20 – 0,35
K	nodular cast iron	IN2504	140 – 210	IN4030	110 – 160	0,20 – 0,30
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN2035	110 – 125	IN2035	60 – 80	0,20
S	titanium alloys	IN2035	40 – 50	IN2035	30 – 40	0,20
H	hard machining < 54 HRC	IN2504	30 – 40	–	–	0,20
H	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**

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:	SNES1204ANN	SNED120420	SNEV1204ANN-PH	SNED1204ANR-DT	SNEX120404R-W
Average chip thickness:	hm = 0,10 mm	hm = 0,10 mm	hm = 0,08 mm	hm = 0,05 mm	hm = 0,08 mm
max. cutting depth:	ap = 1,0 mm	ap = 0,5 mm	ap = 1,0 mm	ap = 0,5 mm	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	150 – 350	IN2505	130 – 300	0,08 – 0,20
P	alloyed steel 800 N/mm²	IN2505	140 – 250	IN2505	120 – 200	0,08 – 0,15
P	alloyed steel 1100 N/mm²	IN2505	130 – 200	IN2505	110 – 150	0,08
M	stainless steel	–	–	–	–	–
K	gray cast iron	IN80B	400 – 800	IN2510	150 – 280	0,08 – 0,20
K	nodular cast iron	IN2510	130 – 250	IN2510	120 – 200	0,08 – 0,20
N	aluminum	–	–	–	–	–
S	high temperature alloys	–	–	–	–	–
S	titanium alloys	–	–	–	–	–
H	hard machining < 54 HRC	–	–	–	–	–
H	hard machining < 63 HRC	–	–	–	–	–

Tips:

- Best results are realized with cutting depth < 0.5 mm.
- The indicated cutting data require optimum machining conditions. If machine stability, workpiece clamping or extension length are not ideal, please adjust the cutting data accordingly.
- In many cases an optical better result of the machined surfaces is achieved by using coolant.

General information:

insert screw: **SM35-110-R0** torque: **3 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:	SNGU1205ENN	SNGU1205EFN-P	SNGS1205ANN-W
Average chip thickness:	hm = 0,20 mm	hm = 0,05 mm	fu max. = 6,9 mm
max. cutting depth:	ap = 6,0 mm	ap = 6,0 mm	ap = 6,0 mm

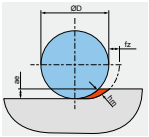
Recommended cutting data:

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	IN2530	200 – 240	0,20 – 0,35
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,20 – 0,30
P	alloyed steel 1100 N/mm²	IN2505	160 – 180	IN2530	110 – 130	0,20
M	stainless steel	IN6535	120 – 180	IN2035	80 – 130	0,20 – 0,35
K	gray cast iron	IN2510	180 – 250	IN4030	150 – 200	0,20 – 0,35
K	nodular cast iron	IN2510	140 – 210	IN4030	110 – 160	0,20 – 0,30
N	aluminum	IN10K	800 – 1500	IN10K	500 – 800	0,05 – 0,40
S	high temperature alloys	IN6535	110 – 125	IN2035	60 – 80	0,20
S	titanium alloys	IN2505	40 – 50	IN2035	30 – 40	0,20
H	hard machining < 54 HRC	IN2505	30 – 40	–	–	0,20
H	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**

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

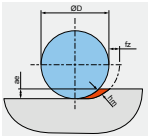
Recommended cutting data:

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
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,10 – 0,12
P	alloyed steel 1100 N/mm²	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
K	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
S	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,10
H	hard machining < 54 HRC	–	–	–	–	–
H	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**

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.

MICROMILL TCHW110204R-W / TCHH110204FR-PW



Insert:	TCHW110204R-W	TCHH110204FR-PW	TCHW110202R-WCT
Average chip thickness:	hm = 0,08 mm	hm = 0,05 mm	hm = 0,08 mm
max. cutting depth:	ap = 1,5 mm	ap = 1,5 mm	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	IN2504	250 – 290	IN4004	200 – 240	0,08 – 0,15
P	alloyed steel 800 N/mm²	IN2504	210 – 250	IN4004	160 – 200	0,08 – 0,10
P	alloyed steel 1100 N/mm²	IN2504	160 – 180	IN4004	110 – 130	0,08
M	stainless steel	IN2504	120 – 180	IN4004	80 – 130	0,08 – 0,15
K	gray cast iron	IN2504	180 – 250	IN4004	150 – 200	0,08 – 0,15
K	nodular cast iron	IN2504	140 – 210	IN4004	110 – 160	0,08 – 0,10
N	aluminum	IN2504	800 – 1500	IN4004	500 – 800	0,08 – 0,15
S	high temperature alloys	IN2504	110 – 125	IN4004	60 – 80	0,08
S	titanium alloys	IN2504	40 – 50	IN4004	30 – 40	0,08
H	hard machining < 54 HRC	IN2504	70 – 100	–	–	0,08
H	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**

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

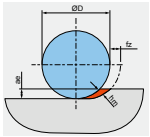
Recommended cutting data:

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
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,04 – 0,06
P	alloyed steel 1100 N/mm²	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
K	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
S	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,04
H	hard machining < 54 HRC	–	–	–	–	–
H	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**

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:	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

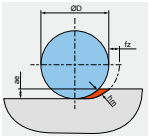
Recommended cutting data:

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	IN4030	250 – 290	IN2530	200 – 240	0,08 – 0,12
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,08 – 0,10
P	alloyed steel 1100 N/mm²	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	IN4030	150 – 200	0,08 – 0,12
K	nodular cast iron	IN2510	140 – 210	IN4030	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
S	titanium alloys	IN2505	40 – 50	IN2035	30 – 40	0,08
H	hard machining < 54 HRC	IN2504	30 – 40	–	–	0,08
H	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**

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:	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

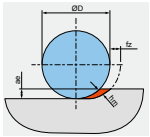
Recommended cutting data:

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	IN4030	250 – 290	IN2530	200 – 240	0,10 – 0,20
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,10 – 0,15
P	alloyed steel 1100 N/mm²	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	IN4030	150 – 200	0,10 – 0,20
K	nodular cast iron	IN2510	140 – 210	IN4030	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
S	titanium alloys	IN2505	40 – 50	IN2035	30 – 40	0,08
H	hard machining < 54 HRC	IN2504	30 – 40	–	–	0,10
H	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:

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



Tauchwerte:

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**

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:	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

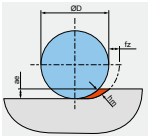
Recommended cutting data:

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	IN4030	250 – 290	IN2530	200 – 240	0,10 – 0,25
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,10 – 0,20
P	alloyed steel 1100 N/mm²	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	IN2510	180 – 250	IN4030	150 – 200	0,10 – 0,25
K	nodular cast iron	IN2510	140 – 210	IN4030	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
S	titanium alloys	IN2505	40 – 50	IN2035	30 – 40	0,08
H	hard machining < 54 HRC	IN2504	30 – 40	–	–	0,10
H	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:

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



Tauchwerte:

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**

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:	TIMC				
Cutting depth:	1,6	2	3	4	5
Average chip thickness:	hm = 0,050 mm	hm = 0,055 mm	hm = 0,065 mm	hm = 0,075 mm	hm = 0,075 mm

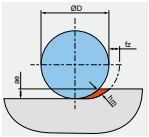
Recommended cutting data:

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	IN2005	90 – 230	IN2005	80 – 210	–
	alloyed steel 800 N/mm²	IN2005	100 – 180	IN2005	90 – 160	–
	alloyed steel 1100 N/mm²	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 ensure to use these tools only up to a max. cutting speed 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.

DIPOS DUO UNHU040212R-HF

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
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,2 – 0,3
P	alloyed steel 1100 N/mm²	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	IN2510	180 – 250	IN2530	150 – 200	0,2 – 0,4
K	nodular cast iron	IN2510	140 – 210	IN2530	110 – 160	0,2 – 0,3
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN2505	110 – 125	IN2530	60 – 80	0,20
S	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,20
H	hard machining < 54 HRC	–	–	–	–	–
H	hard machining < 63 HRC	–	–	–	–	–

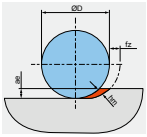
Ramping data and circular interpolation:

tool diameter [mm]	max. ramp. angle [°]	min. bore dia. uneven ground [mm]	max. ap/rev. [mm]	max. bore dia. even ground [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:

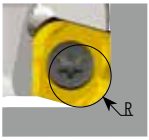
- The worse the material machinability, the smaller the tool engagement should be choosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



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: **DTN005S with bit DS-TP06TB**

DIPOS DUO UNHU060320R-HF

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	IN2505	250 – 290	IN2530	200 – 240	0,2 – 0,8
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,2 – 0,6
P	alloyed steel 1100 N/mm²	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	IN2510	180 – 250	IN2530	150 – 200	0,2 – 0,8
K	nodular cast iron	IN2510	140 – 210	IN2530	110 – 160	0,2 – 0,6
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN2505	110 – 125	IN2530	60 – 80	0,20
S	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,20
H	hard machining < 54 HRC	–	–	–	–	–
H	hard machining < 63 HRC	–	–	–	–	–

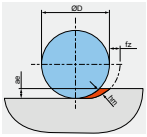
Ramping data and circular interpolation:

tool diameter [mm]	max. ramp. angle [°]	min. bore dia. uneven ground [mm]	max. ap/rev. [mm]	max. bore dia. even ground [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:

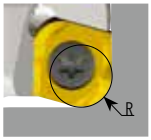
- The worse the material machinability, the smaller the tool engagement should be choosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



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.

DIPOS DUO UNHU090432R-HF

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

Recommended cutting data:

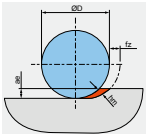
ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 – 290	IN2530	200 – 240	0,2 – 1,0
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,2 – 0,8
P	alloyed steel 1100 N/mm²	IN2505	160 – 180	IN2530	110 – 130	0,2 – 0,5
M	stainless steel	IN2505	120 – 180	IN2530	80 – 130	0,2 – 0,8
K	gray cast iron	IN2510	180 – 250	IN2530	150 – 200	0,2 – 1,0
K	nodular cast iron	IN2510	140 – 210	IN2530	110 – 160	0,2 – 0,8
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN2505	110 – 125	IN2530	60 – 80	0,2 – 0,5
S	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,2 – 0,5
H	hard machining < 54 HRC	–	–	–	–	–
H	hard machining < 63 HRC	–	–	–	–	–

Ramping data and circular interpolation:

tool diameter [mm]	max. ramp. angle [°]	min. bore dia. uneven ground [mm]	max. ap/rev. [mm]	max. bore dia. even ground [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

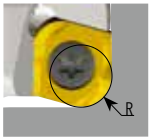
Tips:

- The worse the material machinability, the smaller the tool engagement should be choosen.
- 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: **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.

DIPOS DUO UNHU110640R-HF

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

Recommended cutting data:

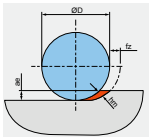
ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 – 290	IN2530	200 – 240	0,3 – 1,2
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,3 – 1,0
P	alloyed steel 1100 N/mm²	IN2505	160 – 180	IN2530	110 – 130	0,3 – 0,6
M	stainless steel	IN2505	120 – 180	IN2530	80 – 130	0,3 – 1,0
K	gray cast iron	IN2510	180 – 250	IN2530	150 – 200	0,3 – 1,2
K	nodular cast iron	IN2510	140 – 210	IN2530	110 – 160	0,3 – 1,0
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN2505	110 – 125	IN2530	60 – 80	0,3 – 0,6
S	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,3 – 0,6
H	hard machining < 54 HRC	–	–	–	–	–
H	hard machining < 63 HRC	–	–	–	–	–

Ramping data and circular interpolation:

tool diameter [mm]	max. ramp. angle [°]	min. bore dia. uneven ground [mm]	max. ap/rev. [mm]	max. bore dia. even ground [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

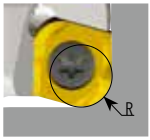
Tips:

- The worse the material machinability, the smaller the tool engagement should be choosen.
- 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:

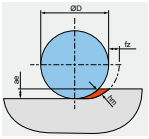
ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 – 290	IN2530	200 – 240	0,3 – 1,5
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,3 – 1,0
P	alloyed steel 1100 N/mm²	IN2505	160 – 180	IN2530	110 – 130	0,3 – 0,8
M	stainless steel	IN2505	120 – 180	IN2530	80 – 130	0,3 – 1,0
K	gray cast iron	IN2510	180 – 250	IN2530	150 – 200	0,3 – 1,5
K	nodular cast iron	IN2510	140 – 210	IN2530	110 – 160	0,3 – 1,0
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN2505	110 – 125	IN2530	60 – 80	0,3 – 0,8
S	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,3 – 0,8
H	hard machining < 54 HRC	–	–	–	–	–
H	hard machining < 63 HRC	–	–	–	–	–

Ramping data and circular interpolation:

tool diameter [mm]	max. ramp. angle [°]	min. bore dia. uneven ground [mm]	max. ap/rev. [mm]	max. bore dia. even ground [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

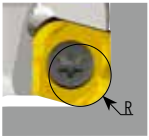
Tips:

- The worse the material machinability, the smaller the tool engagement should be choosen.
- 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**

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

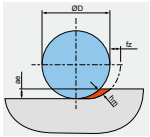
Recommended cutting data:

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
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,2 – 0,3
P	alloyed steel 1100 N/mm²	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
K	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
S	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,20
H	hard machining < 54 HRC	–	–	–	–	–
H	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.
- 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. ramp. angle [°]	min. bore dia. uneven ground [mm]	max. ap/rev. [mm]	max. bore dia. even ground [mm]	max. ap/rev. [mm]
8	0,45	10,5	0,1	16	0,2
9	0,55	12,5	0,1	18	0,2
10	0,30	14,5	0,1	20	0,1
11	0,35	16,5	0,1	22	0,2
12	0,70	18,5	0,2	24	0,4
13	0,75	20,5	0,3	26	0,5
14	0,85	22,5	0,3	28	0,5

Programming Tips:

Please use a corner radius of 0.8 up to 1 mm in your NC-program when machining 3D-contours.
Refer to the following table for max. allowance resp. over cut:

	R program	A over cut	B un-machined
UNKT0502TR-HF	0,80	0,00	0,21
	0,90	0,00	0,18
	1,00	0,02	0,14

General information:

Insert screw: **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:	WNGU1306_N	WNGU1306_FR-P
Average chip thickness:	hm = 0,10 mm	hm = 0,05 mm
max. cutting depth:	ap = 9,2 mm	ap = 9,2 mm

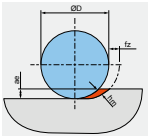
Recommended cutting data:

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/mm²	IN2505	210 – 250	IN2530	160 – 200	0,10 – 0,20
	alloyed steel 1100 N/mm²	IN2505	160 – 180	IN2530	110 – 130	0,10
M	stainless steel	IN2505	120 – 180	IN2530	80 – 130	0,10 – 0,25
K	gray cast iron	IN2504	180 – 250	IN2510	150 – 200	0,10 – 0,25
	nodular cast iron	IN2504	140 – 210	IN2510	110 – 160	0,10 – 0,20
N	aluminum	IN10K	800 – 1500	IN10K	500 – 800	0,05 – 0,35
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	–	–	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:

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



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.


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:

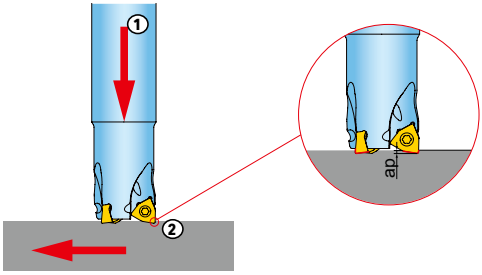
ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN4030	250 – 290	IN2530	200 – 240	0,07 – 0,18
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,07 – 0,13
P	alloyed steel 1100 N/mm²	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	IN4030	150 – 200	0,07 – 0,18
K	nodular cast iron	IN2504	140 – 210	IN4030	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
S	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,07
H	hard machining < 54 HRC	IN2504	30 – 40	–	–	0,07
H	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

Step-down-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 material

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.

DIPOSTRIO WNMU06_ / WNCU06_


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:

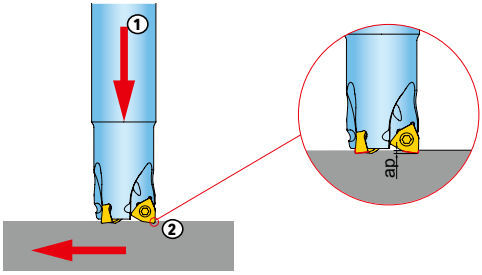
ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN4030	250 – 290	IN2530	200 – 240	0,13 – 0,25
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,13 – 0,20
P	alloyed steel 1100 N/mm²	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	IN4030	150 – 200	0,13 – 0,25
K	nodular cast iron	IN2504	140 – 210	IN4030	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
S	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,13
H	hard machining < 54 HRC	IN2504	30 – 40	–	–	0,13
H	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

Step-down-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 material

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
	> 12% Si	high temperature application	130	IN10K	200 – 1000	0,05 – 0,15
	> 1% Pb copper alloys	well machinable	110	IN10K	200 – 800	0,05 – 0,15
		brass	90	IN10K	300 – 1000	0,05 – 0,15
		electrolytic copper	100	IN10K	300 – 800	0,05 – 0,15
	nonmetal	duroplasts, fibre reinforced plastics	–	IN10K	100 – 500	0,05 – 0,15
		ebonite	–	IN10K	100 – 300	0,05 – 0,15

max revolutions per minute:

Tool diameter [mm]	max. no. of revolutions 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 inserts.
- Please avoid to extend the cutters.
- Please use high speed cutters only on encapsulated machines.

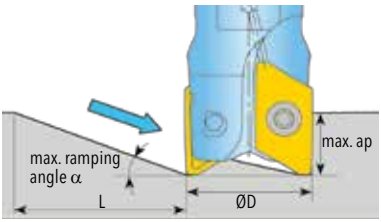
General information:

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

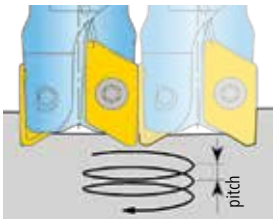
insert screw from Ø32: **SM40-093-21**
torque wrench: **DTN045F with bit DS-T15B1**

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.

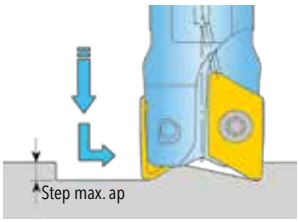
Ramping data and circular interpolation:



Strtaight ramping



Helical ramping



Step milling

XEVT16: R0,4 - R1,6

Tool dia. (D)	Strtaight ramping			Helical ramping			Step milling
	max. ramping angle (α°)	max. ap (mm)	min. length(L)	min. Ø	max. Ø	max. pitch /rev.	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)	Strtaight ramping			Helical ramping			Step milling
	max. ramping angle (α°)	max. ap (mm)	min. length(L)	min. Ø	max. Ø	max. pitch /rev.	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)	Strtaight ramping			Helical ramping			Step milling
	max. ramping angle (α°)	max. ap (mm)	min. length(L)	min. Ø	max. Ø	max. pitch /rev.	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)	Strtaight ramping			Helical ramping			Step milling
	max. ramping angle (α°)	max. ap (mm)	min. length(L)	min. Ø	max. Ø	max. pitch /rev.	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

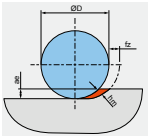
Recommended cutting data:

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	IN2530	150 – 200	IN2040	120 – 200
	alloyed steel 800 N/mm²	IN2530	140 – 200	IN2040	100 – 180
	alloyed steel 1100 N/mm²	IN2530	120 – 180	IN2040	100 – 160
M	stainless steel	IN2530	80 – 160	IN2040	80 – 140
K	gray cast iron	IN2530	150 – 200	IN2010	130 – 180
	nodular cast iron	IN2530	120 – 160	IN2010	100 – 140
N	aluminum	IN2530	500 – 1200	IN04S	400 – 800
S	high temperature alloys	IN2530	110 – 130	IN2004	60 – 80
	titanium alloys	IN2530	–	IN2004	30 – 40
H	hard machining < 54 HRC	IN2530	30 – 50	–	–
	hard machining < 63 HRC	IN2530	30 – 40	–	–

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**

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.

MICROMILL YDA323L_ / YDA334L_ / YCE434_




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

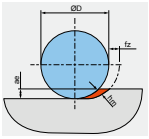
Recommended cutting data:

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/mm²	IN2040	120 – 180	IN2040	100 – 160
	alloyed steel 1100 N/mm²	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 eveness of the machined surface will be.
- **When using peripheral inserts in finish mills the feed per revolution depends on the max. feed rate of the peripheral insert!**

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



General information:

insert screw for YDA323: **SM40-110-00** torque: **4,5 Nm** torque wrench: **DTN045F with bit DS-T15B**

insert screw for YDA334: **SM40-110-00** torque: **4,5 Nm** torque wrench: **DTN045F with bit DS-T15B**

insert screw for YCE434: **SM50-160-10** torque: **6,0 Nm** torque wrench: **DTNV00S with bit DS-T06TB**

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.

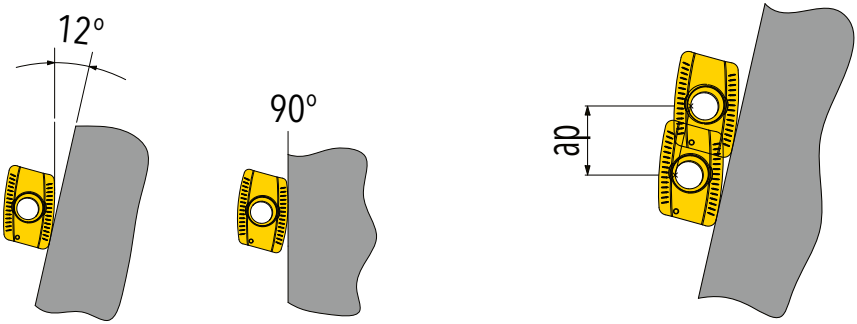

Insert:	CDHT090320R-ML	CDHT090330R-ML
Radius barrel shape	R20	R30
recom. path distance:	ap = 1,0-1,2 mm	ap = 1,2-1,5 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]				radial 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/mm²	IN2505	160 – 220	IN2530	130 – 180	0,2 – 0,4	0,1 – 0,25
	alloyed steel 1100 N/mm²	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 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:

Insert screw: **SM25-044-00** 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:	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
P	alloyed steel 800 N/mm²	IN2505	180 – 250	IN2005	300 – 450	0,1 – 0,3
P	alloyed steel 1100 N/mm²	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
K	nodular cast iron	IN2505	180 – 250	IN2005	250 – 450	0,1 – 0,3
N	aluminum	IN05S	500 – 1000	IN05S	800 – 1200	0,1 – 0,15
S	high temperature alloys	IN2005	40 – 100	IN2005	50 – 150	0,1 – 0,2
S	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
H	hard machining < 63 HRC	IN2006	80 – 130	IN2006	100 – 150	0,07 – 0,12

Tips:

- For extreme cavities plunge milling is recommended, for finishing ae = 0.008 – 0.01 x D.
- With CBN-inserts cast materials can be machined with cutting speeds vc = 1000 to 1500 m/min and feed rate per tooth fz = 0.05 to 0.1 mm.
- The lower fz-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 Nm** torque wrench: **DTN030S with bit DS-T10TB**





Insert:	NBEU1603M0R-ML	NBEU2004M0R-ML	NBEU2505M0R-ML	NBEU1603M0R	NBEU2004M0R
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 mm	8 mm	10 mm


Insert:	NBEU2505M0R	NBEU3207M0R
Feed per tooth fz:	0,05 – 0,50 mm	0,05 – 0,50 mm
Axial cutting edge length:	L = 17,7 mm	L = 21,4 mm
radius:	12,5 mm	16,0 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 – 350	IN2530	160 – 280	0,08 – 0,5
	alloyed steel 800 N/mm²	IN2505	140 – 200	IN2530	110 – 160	0,08 – 0,5
	alloyed steel 1100 N/mm²	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**	torque: **1,1 Nm**	torque wrench: **DS-T08S**
insert screw for Ø20: **SM30-085-10**	torque: **2,0 Nm**	torque wrench: **DS-0022**
insert screw for Ø25: **TS 35085I/HG**	torque: **3,0 Nm**	torque wrench: **TD15**
insert screw for Ø32: **TS 40A115I**	torque: **4,5 Nm**	torque wrench: **TD15**
insert screw for Ø16: **TS 25055I/HG**	torque: **1,1 Nm**	torque wrench: **DS-T085**
insert screw for Ø20: **TS 25055I/HG**	torque: **1,1 Nm**	torque wrench: **DS-T085**
insert screw for Ø25: **TS 25055I/HG**	torque: **1,1 Nm**	torque wrench: **DS-T085**
insert screw for Ø32: **TS 35085I/HG**	torque: **3,0 Nm**	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_
Application:	strong and precise geometry for high-strength materials	positive helical geometry for soft and ductile materials	torical geometry
Recommended cutting depth:	ap = 0,02 – 0,04 mm x Ø	ap = 0,02 – 0,04 mm x Ø	ap = 0,1 – 0,2 mm x radius
max. cutting depth:	ap = 1/2 mm x Ø	ap = 1/2 mm x Ø	ap = R + 1 mm

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min] Semi finishing		Feed per tooth fz [mm]	Cutting speed Vc [m/min] Finishing		Feed per tooth fz [mm]
P	unalloyed steel	IN2005	150 – 250	0,15 – 0,25	IN2005	180 – 350	0,10 – 0,20
P	alloyed steel 800 N/mm²	IN2005	130 – 220	0,15 – 0,25	IN2005	150 – 280	0,10 – 0,20
P	alloyed steel 1100 N/mm²	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
K	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
S	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
H	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**	torque: **1,1 Nm**	torque wrench: **DTN011S with bit DS-TP08TB**
insert screw for Ø 10: **SM30-083-B1**	torque: **2,0 Nm**	torque wrench: **DTN020S with bit DS-TP10TB**
insert screw for Ø 12: **SM40-106-B1**	torque: **4,0 Nm**	torque wrench: **DTNV00S with bit DS-TP15TB**
insert screw for Ø 16: **SM50-139-B1**	torque: **5,0 Nm**	torque wrench: **DTNV00S with bit DS-T20TB**
insert screw for Ø 20: **SM60-167-B1**	torque: **6,0 Nm**	torque wrench: **DTNV00S with bit DS-T25TB**
insert screw for Ø 25: **SM70-210-B1**	torque: **6,0 Nm**	torque wrench: **DTNV00S with bit DS-T25TB**
insert screw for Ø 30/32: **SM80-250-B1**	torque: **8,0 Nm**	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_/SHGT07_	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/mm²	IN2005	130 – 180	IN4015	130 – 180	IN4040	120 – 180
	alloyed steel 1100 N/mm²	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	IN05S	300 – 800	IN05S	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	–	–	–	–	–	–

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 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**

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.

HFDMINI PEMT0502ZCTR-HR

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

Recommended cutting data:

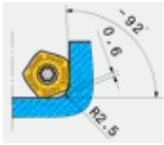
ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 – 290	IN2530	200 – 240	0,40 – 1,50
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,40 – 1,00
P	alloyed steel 1100 N/mm²	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	IN2504	180 – 250	IN2530	150 – 200	0,40 – 1,50
K	nodular cast iron	IN2504	140 – 210	IN2530	110 – 160	0,40 – 1,00
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN2035	110 – 125	IN2035	60 – 80	0,40
S	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,40
H	hard machining < 54 HRC	IN2504	130 – 150	–	–	0,40
H	hard machining < 63 HRC	IN2504	110 – 130	–	–	0,40

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. even 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**





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
P	alloyed steel 800 N/mm²	IN4040	140 – 200	IN4030	110 – 160	0,12 – 0,22
P	alloyed steel 1100 N/mm²	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
K	nodular cast iron	IN4015	140 – 200	IN4015	120 – 170	0,12 – 0,22
N	aluminum	IN05S	500 – 1200	IN05S	500 – 1200	0,15 – 0,25
S	high temperature alloys	IN2035 / IN4030	50 – 80	IN2035 / IN4030	50 – 70	0,1 – 0,18
S	titanium alloys	IN2005	40 – 50	IN2035 / IN4030	30 – 40	0,1 – 0,15
H	hard machining < 54 HRC	–	–	–	–	–
H	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 inserts RCLT...CP / CC / CC1 / C22
- 8-edged inserts RCLT...PH / PH2

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.

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**

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:	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 Ø12:	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 und 12		1st choice for inserts Ø 16 und 20		
P	unalloyed steel	IN2005 / IN2505	180 – 250	IN4040	170 – 220	RHHW / RHHT / RHKW / RHKT
	alloyed steel 800 N/mm²	IN2005 / IN2505	170 – 220	IN4040	150 – 200	RHHW / RHKW / RHKT
	alloyed steel 1100 N/mm²	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	IN05S	500 – 1200	IN05S	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.

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.

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**	torque: **1,1 Nm**	torque wrench: **DTN011S with bit DS-T08TB**
insert screw for RH_08_: **SM30-053-00**	torque: **2,0 Nm**	torque wrench: **DTN020S with bit DS-T09TB**
insert screw for RH_10_: **SM40-080-10**	torque: **4,5 Nm**	torque wrench: **DTN045F with bit DS-T15B**
insert screw for RH_12_: **SM40-080-10**	torque: 4**,5 Nm**	torque wrench: **DTN045F with bit DS-T15B**
insert screw for RH_16_: **SM50-100-10**	torque: **6,0 Nm**	torque wrench: **DTNV00S with bit DS-T20TB**
insert screw for RH_20_: **SM50-100-10**	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:	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
P	alloyed steel 800 N/mm²	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
P	alloyed steel 1100 N/mm²	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	IN2530 / 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	IN2504 / 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
K	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
S	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	IN2530	130 – 180	0,13 – 0,6	0,13 – 0,55	0,13 – 0,5	0,13 – 0,4	0,13 – 0,25
P	alloyed steel 800 N/mm²	IN2505	140 – 200	IN2530	110 – 160	0,13 – 0,6	0,13 – 0,55	0,13 – 0,5	0,13 – 0,4	0,13 – 0,25
P	alloyed steel 1100 N/mm²	IN2505	120 – 180	IN2530	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	IN2530 / IN2035	90 – 150	IN2530 / IN2035	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	IN2504 / IN2505	160 – 250	IN2530	140 – 200	0,13 – 0,6	0,13 – 0,55	0,13 – 0,5	0,13 – 0,4	0,13 – 0,25
K	nodular cast iron	IN2505 / IN2530	140 – 200	IN2530	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	IN2530/IN2035/ IN7035	50 – 80	IN2530/IN2035/ IN7035	50 – 70	0,13 – 0,5	0,13 – 0,4	0,13 – 0,4	0,13 – 0,3	0,13 – 0,2
S	titanium alloys	IN2530/IN2035/ IN7035	-	IN2530/IN2035/ IN7035	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	IN2530	130 – 180	0,15 – 0,8	0,15 – 0,6	0,15 – 0,4	0,15 – 0,3
P	alloyed steel 800 N/mm²	IN2505	140 – 200	IN2530	110 – 160	0,15 – 0,8	0,15 – 0,6	0,15 – 0,4	0,15 – 0,3
P	alloyed steel 1100 N/mm²	IN2505	120 – 180	IN2530	100 – 150	0,15 – 0,8	0,15 – 0,6	0,15 – 0,4	0,15 – 0,3
M	stainless steel	IN2530 / IN7035	90 – 150	IN2530 / IN7035	80 – 130	0,15 – 0,8	0,15 – 0,4	0,1 – 0,3	0,1 – 0,2
K	gray cast iron	IN2504 / IN2505	160 – 250	IN2530	140 – 200	0,15 – 0,8	0,15 – 0,6	0,15 – 0,4	0,15 – 0,3
K	nodular cast iron	IN2505 / IN2530	140 – 200	IN2530	120 – 170	0,15 – 0,8	0,15 – 0,6	0,15 – 0,4	0,15 – 0,3
S	high temperature alloys	IN2530 / IN7035	50 – 80	IN2530 / IN7035	50 – 70	0,15 – 0,8	0,15 – 0,4	0,1 – 0,3	0,1 – 0,2
S	titanium alloys	IN2530 / IN7035	-	IN2530 / IN7035	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**	torque: **3,0 Nm**	torque wrench: **DTNV030S with bit DS-T15TB**
insert screw for RNLU12_: **SM40-110-00**	torque: **4,5 Nm**	torque wrench: **DT – 40 – 01 with bit DS-T15TB**
insert screw for RNLU16_: **TS50A121I/HG**	torque: **6,0 Nm**	torque wrench: **DTNV00S with bit DS-T20T3**

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:	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	PRLX10	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/mm²			IN2505	130 – 280	110 – 220	0,15 – 0,35	0,25 – 0,40	0,80 – 2,50	1,00 – 3,00
M	stainless steel	1.4021	X20Cr13	IN2535/IN7035	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	IN2535/IN7035	130 – 200	110 – 180	0,15 – 0,40	0,25 – 0,50	0,80 – 2,50	1,00 – 3,00
		1.4301	X5CrNi18-10	IN2535/IN7035	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	IN2535/IN7035	80 – 150	60 – 120	0,15 – 0,40	0,25 – 0,50	0,80 – 2,50	1,00 – 3,00
		1.4541	X6CrNiTi18-10	IN2535/IN7035	80 – 150	60 – 120	0,15 – 0,40	0,25 – 0,50	0,80 – 2,50	1,00 – 3,00
		1.4542	X5CrNiCuNb16-4	IN2535/IN7035	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	IN2535/IN7035	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	IN2535/IN7035	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	IN2535/IN7035	50 – 150	40 – 80	0,10 – 0,30	0,15 – 0,40	0,80 – 2,50	1,00 – 3,00
		1.4531	GX2NiCrMoCuN20-18	IN2535/IN7035	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	high temperature alloys	1.4826	GX40CrNiSi22-10	IN2535/IN7035	70 – 140	60 – 120	0,08 – 0,25	0,10 – 0,35	0,80 – 2,50	1,00 – 3,00
		1.4837	GX40CrNiSi25-12	IN2535/IN7035	70 – 140	60 – 120	0,08 – 0,25	0,10 – 0,35	0,80 – 2,50	1,00 – 3,00
		1.4848	GX40CrNiSi22-20	IN2535/IN7035	70 – 140	60 – 120	0,08 – 0,25	0,10 – 0,35	0,80 – 2,50	1,00 – 3,00
		1.4849	GX40NiCrSiNb38-19	IN2535/IN7035	70 – 140	60 – 120	0,08 – 0,25	0,10 – 0,35	0,80 – 2,50	1,00 – 3,00
		1.4923	X22CrMoV12-1	IN2535/IN7035	150 – 280	120 – 250	0,15 – 0,40	0,25 – 0,50	0,80 – 2,50	1,00 – 3,00
		1.4938	X12CrNiMoV12-3	IN2535/IN7035	150 – 280	120 – 250	0,15 – 0,40	0,25 – 0,50	0,80 – 2,50	1,00 – 3,00
		1.4913	X19CrMoVbN11-1	IN2535/IN7035	120 – 210	90 – 160	0,10 – 0,30	0,15 – 0,40	0,80 – 2,50	1,00 – 3,00
		1.4939	X12CrNiMo12	IN2535/IN7035	120 – 210	90 – 160	0,10 – 0,30	0,15 – 0,40	0,80 – 2,50	1,00 – 3,00
		1.4962	X12CrNiWTiB16-13	IN2535/IN7035	80 – 180	60 – 150	0,10 – 0,30	0,12 – 0,40	0,80 – 2,50	1,00 – 3,00
		1.4980	X5NiCrTi26-15	IN2535/IN7035	50 – 110	40 – 80	0,10 – 0,30	0,12 – 0,40	0,80 – 2,50	1,00 – 3,00
	titanium alloys			IN2535/IN7035	–	40 – 60	0,08 – 0,25	0,10 – 0,35	0,80 – 2,50	1,00 – 3,00
	nickel alloys			IN2535/IN7035	–	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 $\mathbf{V}_c$ has to be reduced!

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.

Ramping data and circular interpolation::

tool diameter [mm]	max. ramping angle [°]	min. boring dia. [mm]	max. boring dia. [mm]	boring 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** torque: **3,0 Nm** torque wrench: **DTNV02S mit Klinge DS-T10TB**

insert screw for RPLX12_: **SO 35080I** torque: **4,5 Nm** torque wrench: **DS-T15S**

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:	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
P	alloyed steel 800 N/mm²	IN2505	140 - 200	IN2530	110 - 160	0,4 - 0,8
P	alloyed steel 1100 N/mm²	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
K	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
S	titanium alloys	-	-	IN2530	30 - 40	0,3 - 0,5
H	hard machining < 54 HRC	IN2505	60 - 100	-	-	0,2 - 0,5
H	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.
- Approchaing feed rate should be reduced by 30%.
- 4-edged insert
- Programming radius R0,9

Ramping data and circular interpolation:

tool diameter [mm]	max. ramp. 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** torque: **0,5 Nm** torque wrench: **DTN005S with bit DS-T06TB**

Diameter Ø 16–25: **SM18-041-00** torque: **0,5 Nm** torque wrench: **DTNV00S with bit DS-T06TB**

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:	SDXS0904MPR-MR	SDXS0904MPR-MM	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 / IN4005	160 – 220	IN4030	130 – 180	0,8 – 1,5	0,5 – 1,6
	alloyed steel 800 N/mm²	IN2505 / IN4005	140 – 200	IN4030	110 – 160	0,8 – 1,5	0,5 – 1,6
	alloyed steel 1100 N/mm²	IN2505 / IN4005 / IN2504	120 – 180	IN4030	100 – 150	0,8 – 1,5	0,5 – 1,6
M	stainless steel	IN4035 / IN7035	90 – 150	IN4035 / IN7035	80 – 130	0,8 – 1,5	0,5 – 1,4
K	gray cast iron	IN2505 / IN4005	160 – 250	IN4030	140 – 200	0,8 – 1,5	0,5 – 1,6
	nodular cast iron	IN2505 / IN4005	140 – 200	IN4030	120 – 170	0,8 – 1,5	0,5 – 1,6
N	aluminum	–	–	–	–	–	–
S	high temperature alloys	IN4035 / IN7035	50 – 80	IN4035 / IN7035	50 – 70	0,8 – 1,3	0,5 – 1,4
	titanium alloys	–	–	IN4035	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

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.
- Approchaing 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**

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:	SDMS130516R – PP	SDES130516N – PF1	SDES130516N – PF	SDES1305MPR SDXS1305MPR – MR SDES1305MPR – 001	SDXS1305MPR-MM
max. cutting depth:	ap = 4,9 mm	ap = 4,9 mm	ap = 4,9 mm	ap = 4,9 mm	ap = 4,9 mm
ap_{max} with undercut at 90° shoulder:	ap_{max} = 4,9 mm	ap_{max} = 4,9 mm	ap_{max} = 4,9 mm	ap_{max} = 4,9 mm	no undercut
ap_{max} without undercut at 90° shoulder:	ap_{max} = 0,5 mm	ap_{max} = 0,5 mm	ap_{max} = 0,5 mm	ap_{max} = 0,5 mm	ap_{max} = 2,0 mm
programming radius:	6,4	6,4	6,4	6,14	6,14

Empfohlene Schnittwerte:

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	IN2530 / IN4030	130 – 180	0,2 – 0,5
P	alloyed steel 800 N/mm²	IN4005	140 – 200	IN2530 / IN4030	110 – 160	0,2 – 0,5
P	alloyed steel 1100 N/mm²	IN4005	120 – 180	IN2530 / IN4030	100 – 150	0,2 – 0,5
M	stainless steel	IN4035	90 – 150	IN4035	80 – 130	0,2 – 0,45
K	gray cast iron	IN2505	160 – 250	IN2530 / IN4030	140 – 200	0,2 – 0,6
K	nodular cast iron	IN2505	140 – 200	IN2530 / IN4030	120 – 170	0,2 – 0,6
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN4035	50 – 80	IN4035	50 – 70	0,2 – 0,45
S	titanium alloys	–	–	IN4035	30 – 40	0,2 – 0,45
H	hard machining < 54 HRC	IN2504	60 – 100	–	–	0,2 – 0,4
H	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.
- Approchaing feed rate should be reduced by 30%.
- 4-edged insert

Ramping Data and Circular Interpolation:

tool diameter [mm]	PF / PF1 / PP-Geometry					MPR / MPR-001-Geometry				
	max. ramping angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. [mm]	max. ap/rev. [mm]	max. ramping angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. even ground [mm]	max. ap/rev. [mm]
40	4,6	57	0,5	80	0,5	3,3	56,5	0,5	78	0,5
42	4,2	61	0,5	84	0,5	2,9	60,5	0,5	82	0,5
50	3,4	76,5	0,5	100	0,5	2,4	76	0,5	98	0,5
63	2,4	102,5	0,5	126	0,5	1,7	102	0,5	124	0,5
80	1,8	136,5	0,5	160	0,5	1,3	136	0,5	158	0,5
100	1,4	176,5	0,5	200	0,5	1	176	0,5	198	0,5

General information:

insert screw: **SM40-100-R0** torque: **4 Nm** torque wrench: **DTNV00S mit Klinge 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-MR	SDXS1605MPR-MM	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

Empfohlene Schnittwerte:

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	IN2530	130 – 200	1,0 – 2,0	1,2 – 2,5
P	alloyed steel 800 N/mm²	IN2505	140 – 220	IN2530	110 – 180	1,0 – 2,0	1,2 – 2,5
P	alloyed steel 1100 N/mm²	IN2505	120 – 200	IN2530	100 – 160	1,0 – 2,0	1,0 – 2,2
M	stainless steel	IN2530 / IN4035	90 – 160	IN4035 / IN7035	80 – 140	1,0 – 2,0	1,0 – 2,0
K	gray cast iron	IN2505	160 – 250	IN2530	140 – 200	1,0 – 2,0	1,2 – 2,5
K	nodular cast iron	IN2505	140 – 200	IN2530	120 – 180	1,0 – 2,0	1,0 – 2,2
N	aluminum	–	–	–	–	–	–
S	high temperature alloys	–	–	IN4035 / IN7035	50 – 70	1,0 – 2,0	1,0 – 1,8
S	titanium alloys	–	–	IN4035 / IN7035	30 – 50	1,0 – 2,0	1,0 – 1,8
H	hard machining < 54 HRC	–	–	–	–	–	0,5 – 1,4
H	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.
- Approchaing feed rate should be reduced by 30%.
- 4-edged insert

Ramping Data and Circular Interpolation:

tool diameter [mm]	SDXS1605MPR-MR, -MM, -MR1 and -MRH-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

General information:

insert screw: **SM50-130-R0** torque: **6,0 Nm** torque wrench: **DTNV00S mit Klinge 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:	SDMS13_-PH SDXS13_-PH	SDXS1305MPR-MM	SDES13_MDR_	SDES13_MPR_ SDXS13_-MPR-MR	SDMS19_ – PH
Recom. Depth of cut:	ap = 1,5 mm	ap = 1,5 mm	ap = 1,8 mm	ap = 1,8 mm	ap = 2,0 mm
max. cutting depth:	ap = 2,0 mm	ap = 2,0 mm	ap = 2,0 mm	ap = 2,2 mm	ap = 3,0 mm
ap_{max} with undercut at 90° shoulder:	ap = 2,0 mm	no undercut	ap = 2,0 mm	ap = 2,2 mm	ap = 3,0 mm
ap_{max} without undercut at 90° shoulder:	ap = 0,50 mm	ap = 2,0 mm	ap = 0,95 mm	ap = 1,45 mm	ap = 1,10 mm
programming radius:	R 3,2	R 3,5	R 3,2	R 3,5	R 4,5

			
Insert:	SDES19_MDR_	SDES19_MPR_ SDXS19_-MPR-MR	
Recom. Depth of cut:	ap = 2,5 mm	ap = 2,5 mm	
max. cutting depth:	ap = 3,0 mm	ap = 3,7 mm	
ap_{max} with undercut at 90° shoulder:	ap = 3,0 mm	ap = 3,7 mm	
ap_{max} without undercut at 90° shoulder:	ap = 1,10 mm	ap = 1,50 mm	
programming radius:	R 4,5	R 5,5	

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	IN2530 / IN4030	130 – 180	1,5 – 3
P	alloyed steel 800 N/mm²	IN4005	140 – 200	IN2530 / IN4030	110 – 160	1,2 – 3
P	alloyed steel 1100 N/mm²	IN4005 / IN2504	120 – 180	IN2530 / IN4030	100 – 150	1 – 2,5
M	stainless steel	IN4035	90 – 150	IN4035	80 – 130	1 – 2
K	gray cast iron	IN2505	160 – 250	IN2530 / IN4030	140 – 200	1,5 – 3
K	nodular cast iron	IN2505	140 – 200	IN2530 / IN4030	120 – 170	1,2 – 3
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN4035	50 – 80	IN4035	50 – 70	0,8 – 1,8
S	titanium alloys	–	–	IN4035	30 – 40	0,5 – 1,5
H	hard machining < 54 HRC	IN2504	60 – 100	–	–	0,8 – 2
H	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

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.

Ramping angle and circular interpolation: MDR / N und R Geometry:

Insert size	tool diameter [mm]	max. ramping angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. [mm]	max. ap/rev. [mm]
13	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
19	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 angle and circular interpolation: MPR / MPR-MR / MPR-MM Geometry:

Insert size	tool diameter [mm]	max. ramping angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. [mm]	max. ap/rev. [mm]
13	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
19	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 insert size 13: **SM40-100-R0** torque: **4 Nm** torque wrench: **DTNV00S with bit DS-T15TB**

insert screw insert size 19: **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	SDES130516N-PF SDES130516N-PF1	SDMS130512R-PP	SDMS130516R-PP
Feed per tooth:	fz = 0,1 – 0,3 mm	fz = 0,1 – 0,3 mm	fz = 0,1 – 0,3 mm	fz = 0,1 – 0,3 mm
Recom. width of cut :	ae = 9 mm	ae = 8 mm	ae = 8 mm	ae = 8 mm
max. width of cut:	ae = 11,9 mm	ae = 11,1 mm	ae = 11,6 mm	ae = 11,1 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/mm²	IN4005	130 – 180	–	–
	alloyed steel 1100 N/mm²	IN4005	110 – 170	–	–
M	stainless steel	–	–	IN4035 / IN4030	80 – 150
K	gray cast iron	IN4015 / IN4005	160 – 220	–	–
	nodular cast iron	IN4015 / IN4030	140 – 200	–	–
N	aluminum	–	–	–	–
S	high temperature alloys	–	–	IN4035	40 – 70
	titanium alloys	IN4035	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 (ca. 0,2 – 0,5 mm).

General information:

insert screw: **SM40-100-R0** torque: **4 Nm** torque wrench: **DTNV00S 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.

Notes





Insert:	SDMS1906ZPR-PP	SDMS190620R-PH	SDES1906ZPR-PF	SDES190620N SDES190620N-001	SDES1906MPR SDXS1906MPR-MR SDES1906MPR-001
max. cutting depth:	ap = 7,8 mm	ap = 7,8 mm	ap = 7,8 mm	ap = 7,8 mm	ap = 7,8 mm
ap_{max} with undercut at 90° shoulder:	ap_{max} = 7,8 mm	ap_{max} = 7,8 mm	ap_{max} = 7,8 mm	ap_{max} = 7,8 mm	ap_{max} = 7,8 mm
ap_{max} without undercut at 90° shoulder:	ap_{max} = 1,3 mm	ap_{max} = 0,5 mm	ap_{max} = 0,5 mm	ap_{max} = 0,5 mm	ap_{max} = 0,5 mm
programming radius:	8,4	9,5	8,4	9,5	9,2

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	IN4030	130 – 180	0,25 – 0,6
P	alloyed steel 800 N/mm²	IN4005	140 – 200	IN4030	110 – 160	0,25 – 0,6
P	alloyed steel 1100 N/mm²	IN4005	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	IN2505	160 – 250	IN4030	140 – 200	0,25 – 0,8
K	nodular cast iron	IN2505	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
S	titanium alloys	–	–	IN4035	30 – 40	0,2 – 0,55
H	hard machining < 54 HRC	IN4005	60 – 100	–	–	0,2 – 0,5
H	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

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.

Ramping angle and circular interpolation: PF- / PP Geometry:

tool diameter [mm]	max. ramping angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. [mm]	max. ap/rev. [mm]
80	2,4	126,0	0,5 (PF)/1,3 (PP)	160,0	0,5 (PF)/1,3 (PP)
100	1,8	166,0	0,5 (PF)/1,3 (PP)	200,0	0,5 (PF)/1,3 (PP)
125	1,3	216,0	0,5 (PF)/1,3 (PP)	250,0	0,5 (PF)/1,3 (PP)
160	0,9	286,0	0,5 (PF)/1,3 (PP)	320,0	0,5 (PF)/1,3 (PP)

Ramping angle and circular interpolation: PH- / N- / N-001 Geometry:

tool diameter [mm]	max. ramping angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. [mm]	max. ap/rev. [mm]
80	2,6	126,0	0,5	160,0	0,5
100	1,9	166,0	0,5	200,0	0,5
125	1,4	216,0	0,5	250,0	0,5
160	1,0	286,0	0,5	320,0	0,5

Ramping angle and circular interpolation: MPR- / MPR-001 Geometry:

tool diameter [mm]	max. ramping angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. [mm]	max. ap/rev. [mm]
80	1,6	126,0	0,5	157,4	0,5
100	1,1	166,0	0,5	197,4	0,5
125	0,8	216,0	0,5	247,4	0,5
160	0,7	286,0	0,5	317,4	0,5

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:	LNXF0905R01
Recomm. cutting depth:	ap = 1,0 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	–	–	–	–	–
	alloyed steel 800 N/mm²	–	–	–	–	–
	alloyed steel 1100 N/mm²	–	–	–	–	–
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 choosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- Approach feed should be reduced by 30%.

- 4-edged insert
- Programming radius R3,4

Ramping data and circular interpolation:

tool diameter [mm]	max. ramp. 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

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.

CERASFEED TNXN1207N0104

Insert:	TNXN1207N0104
Recomm. cutting depth:	ap = 1,5 mm
max. cutting depth:	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	–	–	–	–	–
	alloyed steel 800 N/mm²	–	–	–	–	–
	alloyed steel 1100 N/mm²	–	–	–	–	–
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 choosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- Approach feed should be reduced by 30%.

- 6-edged insert
- Programming radius R3,4

Ramping data and circular interpolation:

tool diameter [mm]	max. ramp. 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

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:	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	IN2530	130 – 180	0,5 – 1,0
P	alloyed steel 800 N/mm²	IN2505 / IN2504	140 – 200	IN2530	110 – 160	0,5 – 0,8
P	alloyed steel 1100 N/mm²	IN2505 / IN2504	120 – 180	IN2530	100 – 150	0,5 – 0,7
M	stainless steel	IN2530 / IN2035	90 – 150	IN2530 / IN2035	80 – 130	0,5 – 0,8
K	gray cast iron	IN2505 / IN2530	160 – 250	IN2530	140 – 200	0,5 – 1,0
K	nodular cast iron	IN2505 / IN2530	140 – 200	IN2530	120 – 170	0,5 – 0,8
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN2530	50 – 80	IN2530 / IN2035	50 – 70	0,3 – 0,6
S	titanium alloys	–	–	IN2530 / IN2035	30 – 40	0,3 – 0,6
H	hard machining < 54 HRC	IN2504	60 – 100	–	–	0,2 – 0,5
H	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.
- Approach feed should be reduced by 30%.

- 4-edged insert
- Programming radius R0,9

Ramping data and circular interpolation:

tool diameter [mm]	max. ramp. 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**

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.

Notes



Insert:	UNLU0603M0TR	UNLU0603M0TR – ML	UNLU0603M0TR – MM
Recomm. Depth of cut:	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	IN2530	130 – 180	0,7 – 1,3
	alloyed steel 800 N/mm²	IN2505 / IN2504	140 – 200	IN2530	110 – 160	0,6 – 1,2
	alloyed steel 1100 N/mm²	IN2505 / IN2504	120 – 180	IN2530	100 – 150	0,5 – 1
M	stainless steel	IN2030/IN2035/IN7035	90 – 150	IN2030 / IN2035 / IN7035 / IN6537	80 – 130	0,6 – 0,9
K	gray cast iron	IN2505 / IN2504	160 – 250	IN2530	140 – 200	0,7 – 1,3
	nodular cast iron	IN2505 / IN2504	140 – 200	IN2530	120 – 170	0,5 – 1
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN2035 / IN7035	50 – 80	IN2035 / IN7035	50 – 70	0,4 – 0,7
	titanium alloys	–	–	IN2035/IN7035/IN6537	30 – 40	0,3 – 0,6
H	hard machining < 54 HRC	IN2504	60 – 100	–	–	0,2 – 0,5
	hard machining < 63 HRC	–	–	–	–	–

Tips:

- 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 %.
- For 16 mm tool diameter the max. cutting depth ap = 0.7 mm.

- 4-edged insert
- Programming radius R2

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.

Ramping data and circular interpolation::

tool diameter [mm]	max. ramping angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	even bore diameter [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

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:	UNLU0904M0TR	UNLU0904M0TR-ML	UNLU0904M0TR-MM
Recomm. Depth of cut:	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	IN2530	130-180	0,8-1,5
P	alloyed steel 800 N/mm²	IN2505 / IN2504	140 – 200	IN2530	110-160	0,8-1,3
P	alloyed steel 1100 N/mm²	IN2505 / IN2504	120 – 180	IN2530	100-150	0,8-1,2
M	stainless steel	IN2530 / IN2035	90 – 150	IN2530 / IN2035	80-130	0,7-1,0
K	gray cast iron	IN2505 / IN2530	160 – 250	IN2530	140-200	0,8-1,5
K	nodular cast iron	IN2505 / IN2530	140 – 200	IN2530	120-170	0,8-1,2
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN2035 / IN7035	50 – 80	IN2035 / IN7035	50-70	0,7-1,0
S	titanium alloys	–	–	IN2035 / IN7035	30-40	0,7-0,9
H	hard machining < 54 HRC	IN2504	60 – 100	–	–	0,4 – 0,7
H	hard machining < 63 HRC	–	–	–	–	–

Tips:

- 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 %.
- For 16 mm tool diameter the max. cutting depth ap = 0.7 mm.

- 4-edged insert
- Programming radius R2

Ramping data and circular interpolation:

tool diameter [mm]	max. ramp. 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**

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:	UNLU1105MOTR	UNLU1105MOTR-ML
Recomm. Depth of cut:	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	IN2530	130-180	0,8-2,0
P	alloyed steel 800 N/mm²	IN2505 / IN2504	140 – 200	IN2530	110-160	0,8-1,8
P	alloyed steel 1100 N/mm²	IN2505 / IN2504	120 – 180	IN2530	100-150	0,8-1,6
M	stainless steel	IN2530	90 – 150	IN2530 / IN6537	80-130	0,8-1,2
K	gray cast iron	IN2505 / IN2530	160 – 250	IN2530	140-200	0,8-2,0
K	nodular cast iron	IN2505 / IN2530	140 – 200	IN2530	120-170	0,8-1,6
N	aluminum	–	–	–	–	–
S	high temperature alloys	IN2530	50 – 80	IN2530	50-70	0,8-1,4
S	titanium alloys	–	–	IN2530 / IN6537	30-40	0,7-1,2
H	hard machining < 54 HRC	IN2504	60 – 100	–	–	0,5 –1,0
H	hard machining < 63 HRC	–	–	–	–	–

Tips:

- 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 %.
- For 16 mm tool diameter the max. cutting depth ap = 0.7 mm.

- 4-edged insert
- Programming radius R2

Ramping data and circular interpolation:

tool diameter [mm]	max. ramp. 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 50A121I/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.

HIPOS® UOMT0602_

Insert:	UOMT0602TR
max. cutting depth:	ap = 0,5 mm

Recommended cutting data:

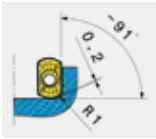
ISO	Material	Cutting speed Vc [m/min]				Feed per tooth fz [mm]
		1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
P	unalloyed steel	IN2505	250 – 290	IN2530	200 – 240	0,30 – 0,80
P	alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0,30 – 0,50
P	alloyed steel 1100 N/mm²	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
K	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
S	titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0,30
H	hard machining < 54 HRC	IN2504	130 – 150	–	–	0,30
H	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/rev. [mm]	min. bore dia. even ground [mm]	max. bore diameter 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]				empfohlene 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	IN2530	130 – 180	0,5 – 0,8	0,5 – 1,0
P	alloyed steel 800 N/mm²	IN2505	140 – 200	IN2530	110 – 160	0,5 – 0,8	0,5 – 1,0
P	alloyed steel 1100 N/mm²	IN2505	120 – 180	IN2530	100 – 150	0,5 – 0,8	0,5 – 1,0
M	stainless steel	IN2530/IN4035	90 – 150	IN2530/IN4035	80 – 130	0,5 – 0,8	0,5 – 0,9
K	gray cast iron	IN2505	160 – 250	IN2530	140 – 200	0,5 – 0,8	0,5 – 1,1
K	nodular cast iron	IN2505	140 – 200	IN2530	120 – 170	0,5 – 0,8	0,5 – 1,1
N	aluminum	–	–	–	–	–	–
S	high temperature alloys	–	–	IN4035	50 – 70	0,5 – 0,7	0,5 – 0,9
S	titanium alloys	–	–	IN4035	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
H	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 %.

- 3-edged insert

Ramping data and circular interpolation:

tool diameter [mm]	Ramping angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	Bohrungs-Ø [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/mm²	IN2505	210 – 250	IN2530 / IN4030	160 – 200	0,07 – 0,13
	alloyed steel 1100 N/mm²	IN2505	160 – 180	IN2530 / IN4030	110 – 130	0,07
M	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 indexable insert selection.

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.

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 v_c [m/min]	Feed per tooth f_z [mm]	Cutting speed v_c [m/min]	Feed per tooth f_z [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	–	–
		16T/17T/18T	80 – 120 [80]	0,03 – 0,15	80 – 140 [110]	0,05 – 0,15
K	cast alloys gray cast iron	16N/17N	70 – 100 [80]	0,03 – 0,12	–	–
		16T/17T/18T	120 – 200 [120]	0,03 – 0,12	160 – 250 [200]	0,05 – 0,15
N	aluminum	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 titanium	16N/17N	30 – 70 [40]	0,02 – 0,12	–	–
		16T/17T/18T	30 – 60 [40]	0,02 – 0,12	30 – 70 [40]	0,02 – 0,12
H	hard machining	–	–	–	–	–
		–	–	–	–	–

Recommended values in brackets [ ].

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]	Maximum 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/mm²	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/mm²	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	–	–	–	–	–
		–	–	–	–	–

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.

MULTISURFER 18F_/19F_ MILLING CUTTER / 18T_/19T_ SLOT MILL / 18Y_/19Y_ THREAD MILL

Empfohlene Schnittwerte:

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 temp. alloys	Inconel 718	HRC 36-40	20 – 30	0,015 – 0,10	0,02 – 0,12	0,02 – 0,12
		AMS R56400	HRC40-45	30 – 40	0,015 – 0,06	0,02 – 0,12	0,02 – 0,12
H	Hard machining	–	–	–	–	–	–
		–	–	–	–	–	–

Parameters:

MultiSurfer Thread Mill 18Y / 19Y										
tool	tool-Ø	Z	∧C	P	TPI	D-min (internal thread)	thread-Ø ISO (internal 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

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.

CHIPSURFER 45A_ SPEED MILL

Rough mills for following applications:

Slot milling, 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	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/mm²	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/mm²	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
		20	R3	200 – 300	0,6 – 1	1,5
	cast alloys	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!

45B_ / 46B_ / 47B_ / 45X_ BALL NOSE MILL

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/mm²	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/mm²	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%
H	hardened steel 48 – 54 HRC	2 – 6	40 – 70	80 – 200	0,025 – 0,08	0,024 – 0,05	0,06 x D	25%
		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%
	hardened steel 54 – 63 HRC	2 – 6	30 – 50	80 – 200	0,02 – 0,06	0,02 – 0,04	0,05 x D	20%
		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%
	hardened steel > 63 HRC	2 – 6	20 – 50	30 – 110	0,015 – 0,04	0,02 – 0,03	0,05 x D	10%
		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]	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
	alloyed steel 800 N/mm²	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/mm²	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	IN05S / IN3005
		10	250 – 1000	0,06 – 0,15	7	IN05S / IN3005
		12	250 – 1000	0,06 – 0,16	9	IN05S / IN3005
		16	250 – 1000	0,08 – 0,20	12	IN05S / IN3005
		20	250 – 1000	0,08 – 0,20	15	IN05S / 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
		14 – 16	160 – 240	0,05 – 0,08	0,08 – 0,2	IN2005
	cast alloys	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 < 50 HRC	8	80 – 140	–	0,05 – 0,08	IN2005 / IN2006
		10	80 – 140	–	0,06 – 0,14	IN2005 / IN2006
		12	80 – 140	–	0,06 – 0,14	IN2005 / IN2006
		14 – 16	80 – 140	–	0,08 – 0,18	IN2005 / IN2006
	hardened steel < 58 HRC	8	60 – 120	–	0,03 – 0,06	IN2005
		10	60 – 120	–	0,04 – 0,08	IN2005
		12	60 – 120	–	0,04 – 0,08	IN2005
		14 – 16	60 – 120	–	0,05 – 0,1	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.

CHIPSURFER 45N_ / 46N_ / 45M_ / 45P_ / 47N_ / 48N_ / 45R_ / 18S_

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
	tool steel	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	cast 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/mm²	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/mm²	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
		20	160 – 220	0,1 – 0,2 x R*	0,2 x R*	0,15 – 0,23	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

* Corner radius

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.

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/mm²	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	super alloys	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/mm²	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	super alloys	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/mm²	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	super alloys	30 – 60	0,02 – 0,06	IN2005
H	hard machining	–	–	–

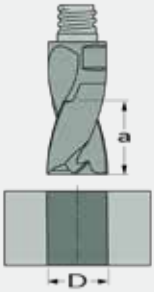
Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.

Recommended cutting data up to 1.2xØ for drilling on fl at surfaces:

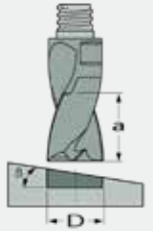
ISO	Material	Cutting speed Vc [m/min]	Revolution speed fu [mm]	Grade
P	unalloyed steel	120 – 220	0,10 – 0,25	IN2005
	alloyed steel 800 N/mm²	120 – 220	0,10 – 0,25	IN2005
	alloyed steel 1000 N/mm²	120 – 220	0,10 – 0,25	IN2005
M	stainless steel	60 – 120	0,06 – 0,15	IN2005
K	gray cast iron	160 – 220	0,10 – 0,25	IN2005
	nodular cast iron	160 – 220	0,10 – 0,25	IN2005
N	aluminum	–	–	–
S	heat resistant alloys	30 – 60	0,05 – 0,12	IN2005
	titanium alloys	30 – 60	0,05 – 0,12	IN2005
H	hard machining < 54 HRC	–	–	–
	hard machining < 63 HRC	–	–	–

Tips:

- For through bores it is recommended to reduce feed rate by 50% before exiting workpiece material.



- For drilling on inclined surfaces up to 30° it is recommended to reduce parameters by 30% and on surfaces up to 45° by 50%.



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.

CHIPSURFER 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/mm²	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/mm²	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	IN05S / IN3005
		10	250 – 1000	0,06 – 0,15	7	IN05S / IN3005
		12	250 – 1000	0,06 – 0,16	9	IN05S / IN3005
		16	250 – 1000	0,08 – 0,20	12	IN05S / IN3005
		20	250 – 1000	0,08 – 0,20	15	IN05S / 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.

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
		20	140 – 200	0,05 – 0,18	1,5 x Ø	IN2005
	alloyed steel < 800 N/mm²	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
		20	140 – 200	0,05 – 0,18	1,5 x Ø	IN2005
	alloyed steel < 1100 N/mm²	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
		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
M	stainless steel	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
		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
K	gray cast iron cast alloys	2 – 4	160 – 220	0,03 – 0,10	2 x Ø	IN2005
		5 – 6	160 – 220	0,03 – 0,11	2 x Ø	IN2005
		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
N	non-ferrous metals duroplastics	4 – 6	250 – 1000	0,03 – 0,08	2 x Ø	IN05S / IN3005
		8	250 – 1000	0,05 – 0,10	2 x Ø	IN05S / IN3005
		10	250 – 1000	0,06 – 0,15	2 x Ø	IN05S / IN3005
		12	250 – 1000	0,06 – 0,16	2 x Ø	IN05S / IN3005
		16	250 – 1000	0,08 – 0,20	2 x Ø	IN05S / IN3005
		20	250 – 1000	0,08 – 0,20	1,5 x Ø	IN05S / IN3005
S	super alloys	2 – 4	25 – 80	0,02 – 0,05	2 x Ø	IN2005
		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
H	hard machining	–	–	–	–	–
		–	–	–	–	–

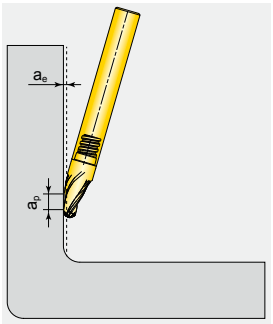
Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.

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 stock ae [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	–	–	–	–	–	–
		–	–	–	–	–	–

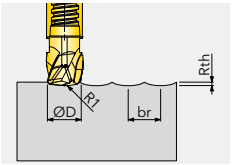


Successful machining results depend on many factors, so cutting data recommendations can only be a rough guideline. Therefore in any case of doubt do not hesitate to contact your Ingersoll partner.

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/mm²	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/mm²	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



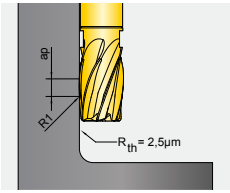
Calculation of path distance br at desired roughness depth Rth 2,5 µ	
R_{th}:	0,0025 mm
Ø / R:	Ø 8 / R= 4
Formula:	$b_r = 2x\sqrt{R_{th}\,((DR1 \times 2) - R_{th}}$
br:	0,55mm

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]	Grade
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/mm²	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/mm²	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 ap at desired roughness depth Rth 2,5 µ	
Rth:	0,0025 mm
Ø / R:	Ø 12 / R1= 70
Formula:	$b_r = 2x\sqrt{R_{th}((DR1 \times 2) - R_{th}}$
ap:	1,18mm

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.

Notes

47A / 48A / 47B SPEED MILL

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/mm²	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/mm²	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.

CHIPSURFER 47C_ / 47D_ HPC 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	0,02 – 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/mm²	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/mm²	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.

CHIPSURFER 47D_ ROUGH/FINISH MILLS 1,5XD

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]	Width of cut 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
		25	140 – 200	0,08 – 0,15	37	1,0 – 5,0	0,04 – 0,10	25
	alloyed steel < 800 N/mm²	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
		25	140 – 200	0,08 – 0,15	37	1,0 – 5,0	0,04 – 0,10	25
	alloyed steel < 1100 N/mm²	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
		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
M	stainless steel	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
		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
K	gray cast iron cast alloys	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
		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
N	non-ferrous metals	–	–	–	–	–	–	–
		–	–	–	–	–	–	–
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.

CHIPSURFER 48D_ FINISH MILL 1,5XD

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]	Width of cut 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
		25	160 – 220	0,08 – 0,15	37	2,5
	alloyed steel < 800 N/mm²	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
		25	160 – 220	0,08 – 0,15	37	2,5
	alloyed steel < 1100 N/mm²	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
		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
M	stainless steel	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
		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
K	gray cast iron cast alloys	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
		25	160 – 220	0,08 – 0,15	37	2,5
N	non-ferrous metals	–	–	–	–	–
		–	–	–	–	–
S	super alloys	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
		20	40 – 80	0,05 – 0,10	30	2,0
		25	40 – 80	0,06 – 0,12	37	2,5
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.

CHIPSURFER 47C_ ROUGH MILL 1,5XD

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]	Width of cut 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
		25	140 – 200	0,08 – 0,18	37	1,0 – 7,5	0,04 – 0,10	25
	alloyed steel < 800 N/mm²	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
		25	140 – 200	0,08 – 0,18	37	1,0 – 7,5	0,04 – 0,10	25
	alloyed steel < 1100 N/mm²	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
		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
M	stainless steel	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
		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
K	gray cast iron cast alloys	8	160 – 220	0,03 – 0,06	12	0,4 – 2,4	0,02 – 0,04	8
		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
N	non-ferrous metals	–	–	–	–	–	–	–
		–	–	–	–	–	–	–
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
		25	25 – 80	0,05 – 0,15	37	1,0 – 7,5	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.

SOLIDCARBIDE 47C / 48C / 46D / 47D / 47J ROUGHING END MILL INNOV

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/mm²	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/mm²	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	IN2005
		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
		25	160 – 220	0,05 – 0,22	2 x Ø	IN2005
N	non-ferrous metals duroplastics	6	250 – 1000	0,05 – 0,08	1,5 x Ø	IN05S / IN3005
		8	250 – 1000	0,05 – 0,10	1,5 x Ø	IN05S / IN3005
		10	250 – 1000	0,06 – 0,15	1,2 x Ø	IN05S / IN3005
		12	250 – 1000	0,06 – 0,16	1,0 x Ø	IN05S / IN3005
		16	250 – 1000	0,08 – 0,20	1,0 x Ø	IN05S / IN3005
		20	250 – 1000	0,08 – 0,20	1,0 x Ø	IN05S / IN3005
S	super alloys	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
		25	25 – 80	0,05 – 0,15	1,5 x Ø	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 47C HPC CUTTER

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]
			at 5% ae	at 10% ae	at 20% ae	at 5% ae	at 10% ae	at 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/mm²	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/mm²	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%.

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.

Notes

SOLID CARBIDE 47C HPC MILLING 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/mm²	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 < 1100 N/mm²	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
		25	120 – 180	0,07 – 0,20	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	
Width of cut	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/mm²	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/mm²	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.

Notes

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/mm²	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/mm²	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
		25	160 – 220	0,05 – 0,20	0,04 – 0,09 x Ø	92	IN2005
N	non-ferrous metals	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
		12	250 – 1000	0,06 – 0,16	0,03 – 0,15 x Ø	75	IN05S / IN3005
S	super alloys	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
		25	40 – 80	0,05 – 0,13	0,025 – 0,075 x Ø	92	IN2005
H	hard machining	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/mm²	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/mm²	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
		16	160 – 220	0,15 – 0,23	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	Werkstückstoff	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

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.

INCERAMIC SOLID CERAMIC SPEED END MILL Z6

Recommended cutting data:

ISO	Material	Dc [mm]	Cutting speed Vc [m/min]	Feed per tooth fz [mm]	Shouldering, profiling		Slotting
					Depth of cut ap [mm]	Width of cut ae [mm]	Depth of cut ap [mm]
P	unalloyed steel	–	–	–	–	–	–
		–	–	–	–	–	–
M	stainless steel	–	–	–	–	–	–
		–	–	–	–	–	–
K	gray cast iron cast alloys	6	300 – 100	0,02 – 0,03	– 0,6xD	– 0,1xD	– 0,05xD
		8	300 – 100	0,02 – 0,03	– 0,6xD	– 0,1xD	– 0,05xD
		10	300 – 100	0,02 – 0,03	– 0,6xD	– 0,1xD	– 0,05xD
		12	300 – 100	0,02 – 0,03	– 0,6xD	– 0,1xD	– 0,05xD
		16	300 – 100	0,02 – 0,03	– 0,6xD	– 0,1xD	– 0,05xD
N	non-ferrous metals	–	–		–		
		–	–		–		
S	super alloys	6	300 – 100	0,02 – 0,03	– 0,6xD	– 0,1xD	– 0,05xD
		8	300 – 100	0,02 – 0,03	– 0,6xD	– 0,1xD	– 0,05xD
		10	300 – 100	0,02 – 0,03	– 0,6xD	– 0,1xD	– 0,05xD
		12	300 – 100	0,02 – 0,03	– 0,6xD	– 0,1xD	– 0,05xD
		16	300 – 100	0,02 – 0,03	– 0,6xD	– 0,1xD	– 0,05xD
H	hardened steel < 54 HRC	–	–	–	–	–	–
		–	–	–	–	–	–

ae must not exceed a maximum 1 mm

Apply a 30% reduction in feed during slotting, ramping (less 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.

TWISTSFEED FPC_R01 DRILL HEADS

Drill heads

Drill head:	FPC_R01
Geometry for:	self-centering

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]	Feed rate fu [mm/U]			1st choice grade / geometry
			15,0 – 15,9	16,0 – 19,9	20,0 – 20,9	
P	unalloyed steel	70–140	0,35 – 0,55	0,35 – 0,55	0,40 – 0,60	IN2205 / FPC
P	alloyed steel 800 N/mm²	50–110	0,30 – 0,45	0,30 – 0,50	0,35 – 0,55	IN2205 / FPC
P	alloyed steel 1100 N/mm²	40–70	0,25 – 0,40	0,30 – 0,45	0,35 – 0,50	IN2205 / FPC
M	stainless steel	–	–	–	–	–
K	gray cast iron	80 – 160	0,40 – 0,60	0,45 – 0,65	0,50 – 0,70	IN2205 / FPC
K	nodular cast iron	90 – 180	0,35 – 0,55	0,40 – 0,60	0,45 – 0,65	IN2205 / FPC
N	aluminum	–	–	–	–	–
S	high temperature alloys	–	–	–	–	–
S	titanium alloys	–	–	–	–	–
H	hard machining < 54 HRC	–	–	–	–	–
H	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.
- The data given are guidelines that may differ from your application.
- The optimum chip shape is created by varying the cutting speed and feed.

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 LPA_R01 DRILL HEADS

Drill heads


Drill head:	LPA_R01	LPF_R01
Geometry for:	machining of steel	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/mm²	50 – 110	0,22 – 0,40	0,25 – 0,45	0,25 – 0,45	0,30 – 0,50	IN2505 / LPA
	alloyed steel 1100 N/mm²	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:

LPA2000R01 bis LPA2190R01:	insert screw: **TS 40178D25**	torque: **4,5 Nm**	torque wrench: **DTNV00S**	bit: **DS.T20TB**
LPA2200R01 bis LPA2390R01:	insert screw: **TS 40198D28**	torque: **4,5 Nm**	torque wrench: **DTNV00S**	bit: **DS-T20TB**
LPA2400R01 bis LPA2590R01:	insert screw: **TS 40210D3**	torque: **4,5 Nm**	torque wrench: **DTNV00S**	bit: **DS-T20TB**
LPA2600R01 bis LPA2790R01:	insert screw: **TS 50230D3**	torque: **5 Nm**	torque wrench: **DTNV00S**	bit: **DS-T20TB**
LPA2800R01 bis LPA2990R01:	insert screw: **TS 50250D35**	torque: **5,5 Nm**	torque wrench: **DTNV00S**	bit: **DS-T25TB**
LPA3000R01 bis LPA3190R01:	insert screw: **TS 60265D4**	torque: **6 Nm**	torque wrench: **DTNV00S**	bit: **DS-T25TB**
LPA3200R01 bis LPA3490R01:	insert screw: **TS 60285D42**	torque: **6 Nm**	torque wrench: **DTNV00S**	bit: **DS-T25TB**
LPA3500R01 bis LPA3790R01:	insert screw: **TS 60320D5**	torque: **6 Nm**	torque wrench: **DTNV00S**	bit: **DS-T25TB**
LPA3800R01 bis LPA4100R01:	insert screw: **TS 80340D6**	torque: **7 Nm**	torque wrench: **DTNV00S**	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.

Notes

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/mm²	120 – 200	0,05 – 0,12	0,08 – 0,15	0,10 – 0,18	IN2005
	alloyed steel 1100 N/mm²	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-PH SDGT140512-HP	
P	unalloyed steel	200 – 300	0,07 – 0,13	0,08 – 0,15	0,08 – 0,16	IN2005
	alloyed steel 800 N/mm²	120 – 200	0,12 – 0,22	0,12 – 0,24	0,13 – 0,25	IN2005
	alloyed steel 1100 N/mm²	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**

Indexable Drills




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 SOMT07T306HP 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/mm²	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/mm²	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/mm²	100 – 240	0,08 – 0,20	0,10 – 0,22	0,10 – 0,22	0,10 – 0,24	IN2505
	alloyed steel 1100 N/mm²	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

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:

SOMT040204_	insert screw: **TS 18041I/HG**	torque: **0,5Nm**	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 22052I/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 / 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/mm²	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/mm²	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/mm²	100 – 190	IN2505 / TPC – C	IN2505 / -WG
	alloyed steel 1100 N/mm²	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**


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
P	alloyed steel 800 N/mm²	100 – 160	0,05 – 0,10	0,05 – 0,10	IN2530
P	alloyed steel 1100 N/mm²	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
K	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
S	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
H	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 c_{utt}ing edges (Z_{eff}) please see catalogue.

General information:

insert screw: **SM30-065-00** torque: **2 Nm** torque wrench: **DTN020S** 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.

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
			6,0 – 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	0,09 – 0,22	0,15 – 0,28	0,18 – 0,30	0,20 – 0,35	0,25 – 0,45	0,25 – 0,45	IN2505 / TPA
P	alloyed steel 800 N/mm²	50 – 110	0,09 – 0,25	0,14 – 0,28	0,16 – 0,32	0,18 – 0,35	0,23 – 0,40	0,25 – 0,45	IN2505 / TPA
P	alloyed steel 1100 N/mm²	40 – 80	0,09 – 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	0,10 – 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	0,12 – 0,30	0,20 – 0,35	0,25 – 0,40	0,30 – 0,45	0,35 – 0,55	0,35 – 0,60	IN2505 / TKA
K	nodular cast iron	90 – 180	0,12 – 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	0,20 – 0,35	0,25 – 0,35	0,30 – 0,45	0,35 – 0,50	0,40 – 0,60	0,45 – 0,70	IN10K / 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
S	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
H	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 TPFxxxxR01 head please use the lower feed rate values (fu).

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/mm²	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/mm²	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 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!
- 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.

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:	LPHT060204R	TPHT070304R	TPHT080305R	TPHT090305R	TPHT100305R
					
Insert:	TPHT110405R	TPHT120405R	TPHT130408R		

Recommended cutting data:

ISO	Material	Cutting speed Vc [m/min]	Feed rate fu [mm]							1st choice grade
			LPHT060204R	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
P	alloyed steel 800 N/mm²	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
P	alloyed steel 1100 N/mm²	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
K	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
S	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	–	–	–	–	–	–	–	–	–
H	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**	torque: **1,1 Nm**	torque wrench: **DTN011S**	bit: **DS-T07TB**
THPT070304R:	insert screw: **SR14-560/S**	torque: **1,1 Nm**	torque wrench: **DTN011S**	bit: **DS-T08TB**
THPT080305R:	insert screw: **SR14-560/S**	torque: **1,1 Nm**	torque wrench: **DTN011S**	bit: **DS-T08TB**
THPT090305R:	insert screw: **SR14-560/S**	torque: **1,1 Nm**	torque wrench: **DTN011S**	bit: **DS-T08TB**
THPT100305R:	insert screw: **SR34-506**	torque: **2 Nm**	torque wrench: **DTN020S**	bit: **DS-T09TB**
THPT110405R:	insert screw: **SR14-571/S**	torque: **3 Nm**	torque wrench: **DTN030S**	bit: **DS-T09TB**
THPT120405R:	insert screw: **SR14-506**	torque: **4,5 Nm**	torque wrench: **DTN045F**	bit: **DS-T15B**
TPHT130408R:	insert screw: **SR16-212/L10**	torque: **7,5 Nm**	torque wrench: **DTNV00S**	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.

SINGLE FLUTE BRAZED GUN DRILLS

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/mm²	50 - 90	0,01 - 0,03	0,03 - 0,05	0,035 - 0,06	0,04 - 0,07	0,05 - 0,10	IN05S / G
	alloyed steel 1100 N/mm²	40-70	0,01 - 0,03	0,025 - 0,04	0,03 - 0,045	0,035 - 0,05	0,04 - 0,10	IN05S / G
M	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 umdr./min), 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

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 rate fu [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
P	alloyed steel 800 N/mm²	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
P	alloyed steel 1100 N/mm²	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
K	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	–	–	–	–	–	–	–	–
S	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
H	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.
- Recomended 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 rate fu [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
P	alloyed steel 800 N/mm²	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
P	alloyed steel 1100 N/mm²	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
K	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	–	–	–	–	–	–	–	–
S	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
H	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.

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

Note

Member IMC Group
Ingersoll Cutting Tools

Online edition 10/22

Ingersoll Cutting Tools
Marketing & Technology

Germany
Ingersoll Werkzeuge GmbH

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

USA
Ingersoll Cutting Tools

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

France
Ingersoll France

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

www.ingersoll-imc.de