INNOTOOL

LOOK FORWARD

CUTTING DATA

>>>2019

INNOTOOL

INNOTOOL, which stands for „Innovative Tooling", is a market leader in indexable milling products.

The high shear geometry design of cutter body and inserts ensures that Innotool performs very well on low powered machines and often the cutting data can be increased considerably due to the soft cutting action.

The range of standard tooling has increased to now also contain a full range of tools for die & mould machining, as well as a range of indexable insert short hole drills.

In addition to the complete range of standard end mills, square shoulder mills, helical end mills, side and face mills and die and mould tooling, INNOTOOL can offer an excellent and fast service for special solutions.

We look forward to being of service.

INNOTOOL

CONTENT

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 Innotool partner.

GRADES

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		
	IN7035	P20-P40	M20-M35			S15-S30		tougher
	IN4035	P25-P50	M20-M40			S20-S30		
	IN2035	P25-P50	M20-M40			S20-S30		
Drilling	IN2010			K10-K30				harder
	IN6505	P10-P25						
	IN6520	P10-P40						
	IN2505	P20-P40	M20-M40			S05-S20		
	IN2005	P15-P30	M15-M35	K20-K40		S05-S20		tougher
Solid Carbide	IN2504	P05-P25					H05-H25	harder
	IN2006	P05-P20					H05-H20	
	IN2005	P15-P30	M15-M35	K20-K40		S05-S20		tougher

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 Innotool partner.

TABLE OF GRADES

	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
	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
	IN4010	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

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 Innotool partner.

TABLE OF GRADES

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
	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–M35	•			for dry machining of stainless steel and heat resistant alloys at high Vc
			S15–S30	•			primarily for milling of materials of machining group ‚S'
	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
Keramik	IN75N	–	K10–K20	•		•	for machining of cast iron at extreme high cutting speed
			S05–S20			•	for high speed machining of nickel-based alloy
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
PKD	IN90D	–	N01–N10	•			for machining of aluminum, non-ferrous materials and graphite

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 Innotool partner.

Parameters:
n: RPM (min^{-1})
Vc: cutting speed
D: tool diameter (mm)

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

Parameters:
Vf: feed rate (mm/min)
fz: feed per tooth (mm)
$Z_{(eff)}$: no. of tooth

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

General formulas

Parameter	Unit	Formula		
RPM	min^{-1}	n	=	$\frac{vc \bullet 1000}{D \times \pi}$
cutting speed	m/min	v_c	=	$\frac{D \bullet \pi \bullet n}{1000}$
feed rate	mm/min	v_f	=	$f_z \bullet Z_{eff} \bullet n$
feed per tooth	mm	f_z	=	$\frac{vf}{Z_{eff} \bullet n}$
chip removal rate	cm^3/min	Q	=	$\frac{a_e \bullet a_p \bullet v_f}{1000}$
average chip thickness	mm	h_m	=	$f_z \bullet \sqrt{a_e / D}$
specific cutting force	MPa	k_c	=	$h_m^{-mc} \bullet k_{c1.1}$
spindle power	kW	P_c	=	$\frac{a_p \bullet a_e \bullet v_f \bullet k_c}{60 \bullet 10^6}$
motor power	kW	P_{mot}	=	$\frac{P}{h}$

Calculation Example	
material:	42CrMo4 (1.7225)
milling cutter:	EB.080.002
insert:	BOMT130404R
cutter dia.:	80 mm
effective no. 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 RPM:	$n = \frac{200 \bullet 1000}{80 \bullet \pi} = 796\ min^{-1}$
calculation of feed rate:	$v_f = 0.12 \bullet 796 \bullet 9 = 859$ mm/min
calculation of chip removal rate:	$Q = \frac{4 \bullet 50 \bullet 859}{1000} = 172\ cm^3/min$
calculation of specific cutting force:	$k_c = 0.15^{-0.24} \bullet 1615 = 2546$ MPa
calculation of required spindle power:	$P_c = \frac{4 \bullet 50 \bullet 859 \bullet 2546}{60 \bullet 10^6} = 7.3$ kW
calculation of motor power:	$P_{mot} = \frac{7.3}{0.8} = 9.1$ kW

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 Innotool partner.

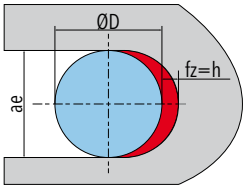


fig. 1: full slot

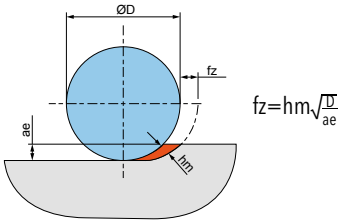


fig. 2: trim-milling

Since the chip is tapered in a comma-shaped form when the cutting width **ae** is reduced, tooth feed must be compensated by the formula shown under ***fig. 2*** whenever the width of cut is less than 1/3 of the cutter diameter. This is often the case at trimming operations ***(fig. 2)*** or when slot mills are used.

For the machining of full slots ***(fig.1)*** or for cutting widths of more than 1/3 of the cutter diameter, the application of this formula is not required. Optimal average chip thickness **(hm)** or tooth feeds **(fz)** of Ingersoll inserts are indicated on the respective cutting data recommendations and vary according to the design of the cutting edge.

In simplified terms, inserts with large protective chamfers can or rather have to be used with higher cutting force at the cutting edges than inserts with sharp cutting edges. Using the inserts with insufficient chip thickness can lead to poor chip formation and increased friction or heat development. Reduced tool life is the result.

To overload the insert with too large chip thicknesses can lead to break-outs or forced fractures of the cutting edge. For a convincing machining result, an optimal load of the cutting edge adapted to the respective insert is therefore essential.

In addition to improoved tool life, the use of the above-mentioned formula also leads to a higher productivity in trim-milling.

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 Innotool partner.



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

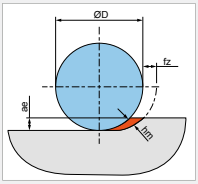
Recommended cutting data:

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		
unalloyed steel	IN2505	250 – 290	IN2530	200 – 240	0.06 – 0.12
alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0.06 – 0.10
alloyed steel 1100 N/mm²	IN2505	160 – 180	IN2530	110 – 130	0.06
stainless steel	IN2035	120 – 180	IN2035	80 – 130	0.06 – 0.12
gray cast iron	IN2504	180 – 250	IN2530	150 – 200	0.06 – 0.12
nodular cast iron	IN2504	140 – 210	IN2530	110 – 160	0.06 – 0.10
aluminum	IN90D	800 – 1500	IN05S	500 – 800	0.05 – 0.12
high temperature alloys	IN2035	110 – 125	IN2035	60 – 80	0.06
titanium alloys	IN2505	40 – 50	IN2035	30 – 40	0.06
hard machining < 54 HRC	IN2504	30 – 40	–	–	0.06
hard machining < 63 HRC	–	–	–	–	–

Tips:

- For tightening the insert screws please always use a torque driver (0.5 Nm).
- For difficult to machine materials please use a max. depth of cut ap = 2 mm and a feed per tooth fz = 0.06 mm.
- If tool engagement is less than 1/3 of cutting tool diameter, the feed per tooth should be calculated with the following formula:

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



Ramping data and circular interpolation:

tool diameter [mm]	max. ramping angle [°]	min. bore dia. uneven ground [mm]	max. ap/rev. [mm]	min. bore dia. even ground [mm]	max. ap/rev. [mm]	max. bore dia. even ground [mm]	max. ap/rev. [mm]
9.5	10.5	11	0.9	17	4.4	18	4.5
10.0	10.0	12	1.1	18	4.4	19	4.5
11.5	7.0	15	1.4	21	3.7	22	4.1
12.0	6.5	16	1.4	22	3.6	23	3.9
13.5	5.5	19	1.5	25	3.5	26	3.8
14.0	5.2	20	1.5	26	3.4	27	3.7
15.0	4.4	22	1.6	28	3.1	29	3.4
16.0	4.0	24	1.6	30	3.1	31	3.3
19.0	2.6	30	1.6	36	2.4	37	2.6
20.0	2.5	32	1.7	38	2.5	39	2.6
22.0	2.3	36	1.7	42	2.5	43	2.6
25.0	2.0	42	1.7	48	2.5	49	2.6
30.0	1.7	52	1.7	58	2.6	59	2.7
32.0	1.6	56	1.7	62	2.6	63	2.7
35.0	1.4	62	1.7	68	2.5	69	2.6
40.0	1.2	72	1.7	78	2.5	79	2.6

General information:

insert screw: **SM18-041-00** torque: **0.5 Nm**
torque wrench: **DTN005S with bit DS-TP06TB**

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

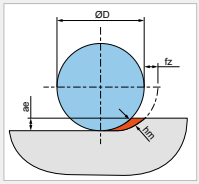
Recommended cutting data:

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		
unalloyed steel	IN4030	250 – 290	IN2530	200 – 240	0.10 – 0,15
alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0.10 – 0.12
alloyed steel 1100 N/mm²	IN2505	160 – 180	IN2530	110 – 130	0.10
stainless steel	IN2035	120 – 180	IN2035	80 – 130	0.10 – 0.12
gray cast iron	IN2504	180 – 250	IN4030	150 – 200	0.10 – 0.15
nodular cast iron	IN2504	140 – 210	IN4030	110 – 160	0.10 – 0.12
aluminum	IN90D	800 – 1500	IN10K	500 – 800	0.05 – 0.20
high temperature alloys	IN2035	110 – 125	IN2035	60 – 80	0.10
titanium alloys	IN2505	40 – 50	IN2035	30 – 40	0.10
hard machining < 54 HRC	IN2504	130 – 150	–	–	0.10
hard machining < 63 HRC	IN2504	110 – 130	–	–	0.10

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:

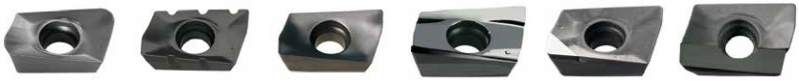
tool diameter [mm]	max. ramping angle [°]	min. bore dia. uneven ground [mm]	max. ap/rev. [mm]	min. bore dia. even ground [mm]	max. ap/rev. [mm]	max. bore dia. even ground [mm]	max. ap/rev. [mm]
12	1,5	16	0,3	21,2	0,7	23	0,9
15	11.0	18	1.8	26.8	7.2	29	8.5
16	10.0	20	2.2	28.8	7.0	31	8.5
20	7.0	28	3.0	36.7	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 Innotool partner.



insert:	BOMT1304_R	ZOMT1304_R	BOMT1304_R-HS	BOCT1304_FR-P	BOMT1304_R-DT1	BOMT1304_R-DT2
average chip thickness:	hm = 0.12 mm	hm = 0.12 mm	hm = 0.08 mm	hm = 0.05 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 = 4 mm	ap = 12 mm

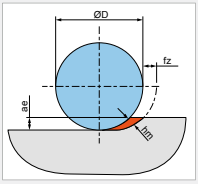
Recommended cutting data:

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		
unalloyed steel	IN4030	250 – 290	IN2530	200 – 240	0.12 – 0.20
alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0.12 – 0.15
alloyed steel 1100 N/mm²	IN2505	160 – 180	IN2530	110 – 130	0.12
stainless steel	IN2035	120 – 180	IN2035	80 – 130	0.12 – 0.20
gray cast iron	IN2504	180 – 250	IN4030	150 – 200	0.12 – 0.20
nodular cast iron	IN2504	140 – 210	IN4030	110 – 160	0.12 – 0.15
aluminum	IN90D	800 – 1500	IN10K	500 – 800	0.12 – 0.20
high temperature alloys	IN2035	110 – 125	IN2035	60 – 80	0.12
titanium alloys	IN2505	40 – 50	IN2035	30 – 40	0.12
hard machining < 54 HRC	IN2504	30 – 40	–	–	0.12
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	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 Innotool partner.

insert:	TIMC				
cutting depth:	1.6 mm	2 mm	3 mm	4 mm	5 mm
average chip thickness:	hm = 0.050 mm	hm = 0.055 mm	hm = 0.065 mm	hm = 0.075 mm	hm = 0.075 mm

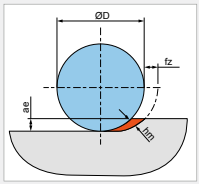
Recommended cutting data:

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		
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	–
stainless steel	IN2005	70 – 150	IN2005	60 – 130	–
gray cast iron	IN2005	110 – 140	IN2005	90 – 120	–
nodular cast iron	IN2005	55 – 110	IN2005	45 – 90	–
aluminum	IN2005	250	IN2005	250	–
high temperature alloys	IN2005	20 – 40	IN2005	20 – 40	–
titanium alloys	IN2005	30 – 60	IN2005	30 – 60	–
hard machining < 54 HRC	–	–	–	–	–
hard machining < 63 HRC	–	–	–	–	–

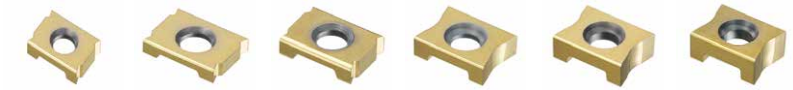
Tips:

- 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 Innotool partner.



insert:	IEE211	IEE311	IEE312	IXE412	IXE413	IXE414
average chip thickness:	hm = 0.05 mm	hm = 0.06 mm	hm = 0.07 mm	hm = 0.10 mm	hm = 0.15 mm	hm = 0.15 mm

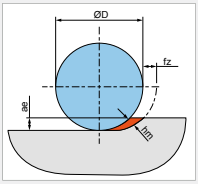
Recommended cutting data:

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		
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
stainless steel	IN2530	80 – 160	IN2530	80 – 160	hm x 1.2
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
aluminum	IN05S	500 – 1200	IN05S	400 – 800	hm x 1.3
high temperature alloys	IN2535	50 – 80	IN4035	40 – 70	hm x 0.9
titanium alloys	–	–	IN4035	30 – 40	hm x 1.0
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: **0.6 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**

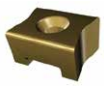
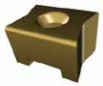
insert screw size 312: **SM35-042-50** torque: **2.0 Nm**
torque wrench: **DTN020S with bit DS-T09TB**

insert screw size 412: **SM40-040-50** torque: **4.5 Nm**
torque wrench: **DT-40-01 with bit DS-T15B**

insert screw size 413: **SM40-070-50** torque: **4.5 Nm**
torque wrench: **DT-40-01 with bit DS-T15B**

insert screw size 414: **SM40-080-50** torque: **4.5 Nm**
torque wrench: **DT-40-01 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 Innotool partner.

insert:	IXH415	IXH416
average chip thickness:	hm = 0.15 mm	hm = 0.15 mm

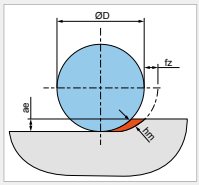
Recommended cutting data:

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		
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
stainless steel	IN4035	80 – 160	IN4035	80 – 160	hm x 1.2
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
aluminum	IN05S	500 – 1200	IN05S	400 – 800	hm x 1.3
high temperature alloys	IN4005	50 – 80	IN4030	40 – 70	hm x 0.9
titanium alloys	–	–	IN4005	30 – 40	hm x 1.0
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:

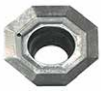
insert screw size 415: **SM40-090-00** torque: **4.5 Nm**
torque wrench: **DT-40-01 with bit DS-T15B**

insert screw size 416: **SM40-110-00** torque: **4.5 Nm**
torque wrench: **DT-40-01 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 Innotool partner.







insert:	OFMT05T3AFN-HR	OFCT05T3AFTN-HR	OFCT05T3TN	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

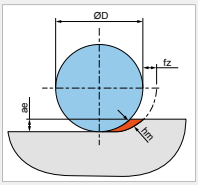
Recommended cutting data:

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		
unalloyed steel	IN4030	250 – 290	IN2530	200 – 240	0.12 – 0.25
alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0.12 – 0.20
alloyed steel 1100 N/mm²	IN2505	160 – 180	IN2530	110 – 130	0.12
stainless steel	IN6535	120 – 180	IN2035	80 – 130	0.12 – 0.25
gray cast iron	IN2510	180 – 250	IN4030	150 – 200	0.20 – 0.40
nodular cast iron	IN2510	140 – 210	IN4030	110 – 160	0.20 – 0.30
aluminum	IN10K	800 – 1500	IN10K	500 – 800	0.05 – 0.30
high temperature alloys	IN6535	110 – 125	IN2035	60 – 80	0.12
titanium alloys	IN2505	40 – 50	IN2035	30 – 40	0.12
hard machining < 54 HRC	IN2505	30 – 40	–	–	0.20
hard machining < 63 HRC	–	–	–	–	–

Tips:

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

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



Ramping data and circular interpolation:

tool diameter [mm]	max. ramping angle [°]	min. bore dia. uneven ground [mm]	max. ap/rev. [mm]	min. bore dia. even ground [mm]	max. ap/rev. [mm]	max. bore dia. even ground [mm]	max. ap/rev. [mm]	min. 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 wide pitch: **SM40-093-20**
insert screw fine pitch: **SM40-100-R0**
torque: **4.5 Nm**
torque wrench: **DT-40-01 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 Innotool partner.

insert:	OFMT0705AFR-HR	OFCT0705AFTN-HR	OFMT0705AFTN	OFMW0705AFTN	OFCT0705AFFN-P	OFCT0705AFFR-W
average chip thickness:	hm = 0.15 mm	hm = 0.18 mm	hm = 0.15 mm	hm = 0.25 mm	hm = 0.05 mm	fu max = 3.8 mm
max. cutting depth:	ap = 4.8 mm	ap = 4.8 mm	ap = 4.8 mm	ap = 4.8 mm	ap = 4.8 mm	ap = 4.8 mm

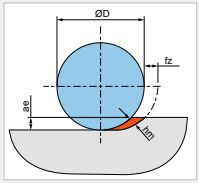
Recommended cutting data:

material	cutting speed Vc [m/min]				feed per tooth fz [mm]
	1st choice dry machining resp. wear resistant material		1st choice wet machining resp. tough carbide		
unalloyed steel	IN4030	250 – 290	IN4030	200 – 240	0.15 – 0.30
alloyed steel 800 N/mm²	IN4005	210 – 250	IN4030	160 – 200	0.15 – 0.25
alloyed steel 1100 N/mm²	IN4005	160 – 180	IN4030	110 – 130	0.15
stainless steel	IN6535	120 – 180	IN2035	80 – 130	0.18 – 0.30
gray cast iron	IN4005	180 – 250	IN4030	150 – 200	0.25 – 0.50
nodular cast iron	IN4005	140 – 210	IN4030	110 – 160	0.25 – 0.40
aluminum	IN05S	800 – 1500	IN05S	500 – 800	0.05 – 0.30
high temperature alloys	IN6535	110 – 125	IN2035	60 – 80	0.18
titanium alloys	IN4005	40 – 50	IN2035	30 – 40	0.18
hard machining < 54 HRC	IN4005	30 – 40	–	–	0.25
hard machining < 63 HRC	–	–	–	–	–

Tips:

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

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



Ramping data and circular interpolation:

tool diameter [mm]	max. ramping angle [°]	min. bore dia. uneven ground [mm]	max. ap/rev. [mm]	min. bore dia. even ground [mm]	max. ap/rev. [mm]	max. bore dia. even ground [mm]	max. ap/rev. [mm]	max. bore dia. uneven ground [mm]	max. ap/rev. [mm]
50	7.1	90.8	4.8	108.9	4.8	112	4.8	124.6	4.8
63	5.1	116.6	4.8	134.8	4.8	137.9	4.8	150.5	4.8
80	3.7	150.4	4.8	168.8	4.8	171.9	4.8	184.4	4.8
100	2.8	190.2	4.8	208.6	4.8	211.7	4.8	224.2	4.8
125	2.2	240.3	4.8	258.7	4.8	261.8	4.8	274.3	4.8
160	1.6	310.1	4.8	328.6	4.8	331.7	4.8	344.1	4.8

General information:

insert screw wide pitch: **SM50-120-30**
insert screw fine pitch: **SM50-130-R0**
torque: **7.5 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 Innotool partner.






insert:	ONCU0505ANTN-HR	ONCU050520TN	ONCU0505ANEN	ONCU0505ANFN-P	ONCU0505ANN	ONCU0505ANTN-W
average chip thickness:	hm = 0.22 mm	hm = 0.25 mm	hm = 0.08 mm	hm = 0.05 mm	hm = 0.15 mm	fu max = 2.4
max. cutting depth:	ap = 3 mm	ap = 1.5 – 2.5 mm	ap = 3 mm	ap = 3 mm	ap = 3 mm	ap = 3 mm

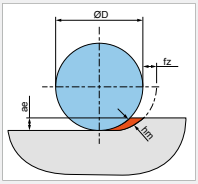
Recommended cutting data:

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		
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
stainless steel	IN6535	120 – 180	IN2035	80 – 130	0.08 – 0.30
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
aluminum	IN10K	800 – 1500	IN10K	500 – 800	0.05 – 0.30
high temperature alloys	IN6535	110 – 125	IN2035	60 – 80	0.08
titanium alloys	IN2505	40 – 50	IN2035	30 – 40	0.22
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: **DT-40-01 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 Innotool 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 = 0.08 mm	hm = 0.15 mm	hm = 0.05 mm	fu max = 2.4	fu max = 2.4
max. cutting depth:	ap = 5 mm	ap = 5 mm	ap = 5 mm	ap = 3.9 mm	ap = 3.8 mm

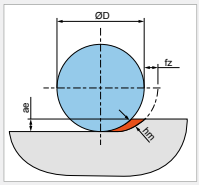
Recommended cutting data:

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		
unalloyed steel	IN4030	250 – 290	IN4030	200 – 240	0.30 – 0.40
alloyed steel 800 N/mm²	IN4005	210 – 250	IN4030	160 – 200	0.30 – 0.35
alloyed steel 1100 N/mm²	IN4005	160 – 180	IN4030	110 – 130	0.30
stainless steel	IN2035	120 – 180	IN2035	80 – 130	0.08 – 0.30
gray cast iron	IN70N	600 – 900	IN4030	150 – 200	0.08 – 0.40
nodular cast iron	IN4010	140 – 210	IN4030	110 – 160	0.30 – 0.35
aluminum	IN10K	800 – 1500	IN10K	500 – 800	0.05 – 0.40
high temperature alloys	IN2035	110 – 125	IN2035	60 – 80	0.08
titanium alloys	IN4005	40 – 50	IN2035	30 – 40	0.30
hard machining < 54 HRC	IN4005	30 – 40	–	–	0.30
hard machining < 63 HRC	–	–	–	–	–

Tips:

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

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



General information:

for 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 Innotool partner.






insert:	PNCU0805GNTR	PNCU0805GNFR-HS	PNCQ0804GNTN	PNCU0805GNFR-P	PNCU0805GNR	PNCU0805GNTR-W
average chip thickness:	hm = 0.20 mm	hm = 0.08 mm	hm = 0.20 mm	hm = 0.05 mm	hm = 0.10 mm	fu max = 3.6
max. cutting depth:	ap = 6 mm	ap = 6 mm	ap = 6 mm	ap = 6 mm	ap = 6 mm	ap = 6 mm

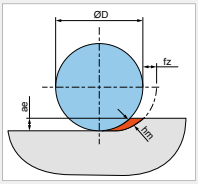
Recommended cutting data:

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		
unalloyed steel	IN4030	250 – 290	IN4030	200 – 240	0.20 – 0.40
alloyed steel 800 N/mm²	IN4005	210 – 250	IN4030	160 – 200	0.20 – 0.30
alloyed steel 1100 N/mm²	IN4005	160 – 180	IN4030	110 – 130	0.20
stainless steel	IN2035	120 – 180	IN2035	80 – 130	0.08 – 0.30
gray cast iron	IN70N	600 – 900	IN4030	150 – 200	0.10 – 0.40
nodular cast iron	IN4015	140 – 210	IN4030	110 – 160	0.20 – 0.30
aluminum	IN05S	800 – 1500	IN05S	500 – 800	0.05 – 0.40
high temperature alloys	IN2035	110 – 125	IN2035	60 – 80	0.08
titanium alloys	IN2505	40 – 50	IN2035	30 – 40	0.20
hard machining < 54 HRC	IN2505	30 – 40	–	–	0.20
hard machining < 63 HRC	–	–	–	–	–

Tips:

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

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



General information:

insert screw: **SM40-100-10**
torque: **4.5 Nm**
torque wrench: **DT-40-01 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 Innotool 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

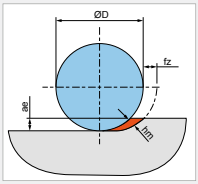
Recommended cutting data:

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		
unalloyed steel	IN2505	250 – 290	IN2505	200 – 240	0.06 – 0.12
alloyed steel 800 N/mm²	IN2505	210 – 250	IN2505	160 – 200	0.06 – 0.10
alloyed steel 1100 N/mm²	IN2505	160 – 180	IN2505	110 – 130	0.06
stainless steel	IN2505	120 – 180	IN2505	80 – 130	0.06 – 0.12
gray cast iron	IN2505	180 – 250	IN2505	150 – 200	0.06 – 0.15
nodular cast iron	IN2505	140 – 210	IN2505	110 – 160	0.06 – 0.12
aluminum	IN10K	800 – 1500	IN10K	500 – 800	0.05- 0.15
high temperature alloys	IN2505	110 – 125	IN2505	60 – 80	0.06
titanium alloys	IN2505	40 – 50	IN2505	30 – 40	0.06
hard machining < 54 HRC	IN2505	30 – 40	–	–	0.06
hard machining < 63 HRC	–	–	–	–	–

Tips:

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

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



General information:

insert screw: **SM20-043-00**
torque: **0.7 Nm**
torque wrench: **DTNV01S with bit DS-TP06TB**

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 Innotool partner.


insert:	SDE-_	SEE-_
average chip thickness:	hm = 0.13 mm	hm = 0.05 mm
max. cutting width:	4 – 15 mm	4 – 15 mm

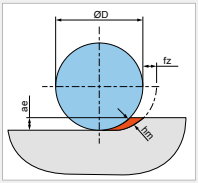
Recommended cutting data:

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		
unalloyed steel	IN4030	250 – 290	IN4030	200 – 240	0.13 – 0.25
alloyed steel 800 N/mm²	IN4030	210 – 250	IN4030	160 – 200	0.13 – 0.20
alloyed steel 1100 N/mm²	IN4030	160 – 180	IN4030	110 – 130	0.13
stainless steel	IN4030	120 – 180	IN4030	80 – 130	0.13 – 0.20
gray cast iron	IN4030	180 – 250	IN4030	150 – 200	0.13 – 0.25
nodular cast iron	IN4030	140 – 210	IN4030	110 – 160	0.13 – 0.20
aluminum	IN30M	500 – 800	IN30M	500 – 800	0.13 – 0.25
high temperature alloys	IN4030	110 – 125	IN4030	60 – 80	0.13
titanium alloys	IN4030	40 – 50	IN4030	30 – 40	0.13
hard machining < 54 HRC	IN4030	30 – 40	–	–	0.13
hard machining < 63 HRC	–	–	–	–	–

Tips:

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

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



General information:

for cutting width	insert screw	torque wrench	with bit	torque
4 mm	SM35-034-50	DTNV02S	DS-T09TB	1.4 Nm
5 mm	SM35-042-50	DTNV02S	DS-T09TB	1.4 Nm
6 mm	SM40-050-50	DT-40-01	DS-T15B	4.5 Nm
7 and 8 mm	SM40-060-50	DT-40-01	DS-T15B	4.5 Nm
9 and 10 mm	SM40-080-50	DT-40-01	DS-T15B	4.5 Nm
12 / 13 / 14 and 15 mm	SM40-106-50	DT-40-01	DS-T15B	4.5 Nm

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 Innotool partner.

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

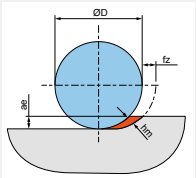
Recommended cutting data:

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		
unalloyed steel	IN2505	250 – 290	IN2530	200 – 240	0.10 – 0.35
alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0.10 – 0.30
alloyed steel 1100 N/mm²	IN2505	160 – 180	IN2530	110 – 130	0.10 – 0.25
stainless steel	IN4035	120 – 180	IN2530	80 – 130	0.08 – 0.25
gray cast iron	IN2505	180 – 250	IN4030	150 – 200	0.10 – 0.35
nodular cast iron	IN2505	140 – 210	IN4030	110 – 160	0.10 – 0.35
aluminum	IN2505	800 – 1500	IN2505	500 – 800	0.08 – 0.25
high temperature alloys	IN4035	110 – 125	IN2035	60 – 80	0.08 - 0,18
titanium alloys	IN4035	40 – 50	IN2035	30 – 40	0.08 - 0,18
hard machining < 54 HRC	IN2505	30 – 40	–	–	0.08
hard machining < 63 HRC	–	–	–	–	–

Tips:

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

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



Ramping data and circular interpolation:

tool diameter [mm]	max. ramping angle [°]	min. bore dia. [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: **DT-40-01 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 Innotool partner.




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

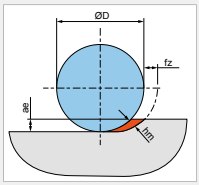
Recommended cutting data:

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		
unalloyed steel	IN4030	250 – 290	IN4030	200 – 240	0.13 – 0.20
alloyed steel 800 N/mm²	IN4030	210 – 250	IN4030	160 – 200	0.13 – 0.15
alloyed steel 1100 N/mm²	IN4030	160 – 180	IN4030	110 – 130	0.13
stainless steel	IN4030	120 – 180	IN4030	80 – 130	0.13 – 0.20
gray cast iron	IN4030	180 – 250	IN4030	150 – 200	0.13 – 0.20
nodular cast iron	IN4030	140 – 210	IN4030	110 – 160	0.13 – 0.15
aluminum	IN05S	800 – 1500	IN05S	500 – 800	0.05 – 0.20
high temperature alloys	IN2505	110 – 125	IN2505	60 – 80	0.13
titanium alloys	IN2505	40 – 50	IN2505	30 – 40	0.13
hard machining < 54 HRC	IN2505	30 – 40	–	–	0.13
hard machining < 63 HRC	–	–	–	–	–

Tips:

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

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



Ramping data and circular interpolation:

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

insert:	SGM-44R_
average chip thickness:	hm = 0.18 mm
max. cutting depth:	ap = 8.7 mm

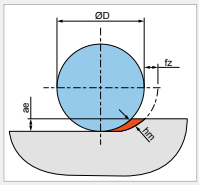
Recommended cutting data:

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		
unalloyed steel	IN2505	150 – 250	IN2530	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
stainless steel	IN4030	80 – 160	IN2530	80 – 160	hm x 1.2
gray cast iron	IN4015	160 – 250	IN4030	140 – 200	hm x 1.2
nodular cast iron	IN4015	120 – 200	IN4015	100 – 180	hm x 1.0
aluminum	–	–	–	–	–
high temperature alloys	IN2530	50 – 80	IN2530	40 – 70	hm x 0.9
titanium alloys	–	–	IN2530	30 – 40	hm x 1.0
hard machining < 54 HRC	–	–	–	–	–
hard machining < 63 HRC	–	–	–	–	–

Tips:

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

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



General information:

insert screw: **SM40-120-20**
torque: **4.5 Nm**
torque wrench: **DT-40-01 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 Innotool partner.

insert: SHET1105_FR-P

average chip thickness: **hm = 0.05 mm**

max. cutting depth: **ap = 8.4 mm**

Recommended cutting data:

material	remark	hardness [HB]	carbide grade	cutting speed Vc [m/min]	feed per tooth fz [mm]
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. RPM:

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: **SM40-093-20**
torque: **4.5 Nm**
torque wrench: **DT-40-10 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 Innotool partner.

insert: TCHW110204R-W

average chip thickness: **hm = 0.08 mm**

max. cutting depth: **ap = 1.5 mm**

Recommended cutting data:

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		
unalloyed steel	IN2504	250 – 290	IN4004	200 – 240	0.08 – 0.15
alloyed steel 800 N/mm²	IN2504	210 – 250	IN4004	160 – 200	0.08 – 0.10
alloyed steel 1100 N/mm²	IN2504	160 – 180	IN4004	110 – 130	0.08
stainless steel	IN2504	120 – 180	IN4004	80 – 130	0.08 – 0.15
gray cast iron	IN2504	180 – 250	IN4004	150 – 200	0.08 – 0.15
nodular cast iron	IN2504	140 – 210	IN4004	110 – 160	0.08 – 0.10
aluminum	IN2504	800 – 1500	IN4004	500 – 800	0.08 – 0.15
high temperature alloys	IN2504	110 – 125	IN4004	60 – 80	0.08
titanium alloys	IN2504	40 – 50	IN4004	30 – 40	0.08
hard machining < 54 HRC	IN2504	70 – 100	–	–	0.08
hard machining < 63 HRC	IN2504	50 – 80	–	–	0.08

Tips:

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

General information:

insert screw: **SM25-064-00**
torque: **1.1 Nm**
torque wrench: **DTN011S with bit DS-T08TB**

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 Innotool partner.

insert:	WNGU1306_R	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

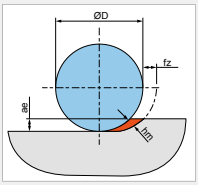
Recommended cutting data:

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		
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
stainless steel	IN2505	120 – 180	IN2530	80 – 130	0.10 – 0.25
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
aluminum	IN10K	800 – 1500	IN10K	500 – 800	0.05 – 0.35
high temperature alloys	IN2505	110 – 125	IN2530	60 – 80	0.10
titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0.10
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: **DT-40-01 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 Innotool partner.


insert:	WNMU04T3_R	WNCU04T3_FN-P
average chip thickness:	hm = 0.07mm	hm = 0.05 mm
max. cutting depth:	ap = 3.8 mm	ap = 3.8 mm

Recommended cutting data:

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		
unalloyed steel	IN4030	250 – 290	IN2530	200 – 240	0.07 – 0.18
alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0.07 – 0.13
alloyed steel 1100 N/mm²	IN2505	160 – 180	IN2530	110 – 130	0.07
stainless steel	IN2035	120 – 180	IN2035	80 – 130	0.07 – 0.18
gray cast iron	IN2504	180 – 250	IN4030	150 – 200	0.07 – 0.18
nodular cast iron	IN2504	140 – 210	IN4030	110 – 160	0.07 – 0.13
aluminum	IN10K	800 – 1500	IN10K	500 – 800	0.05 – 0.20
high temperature alloys	IN2035	110 – 125	IN2530	60 – 80	0.07
titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0.07
hard machining < 54 HRC	IN2504	30 – 40	–	–	0.07
hard machining < 63 HRC	–	–	–	–	–

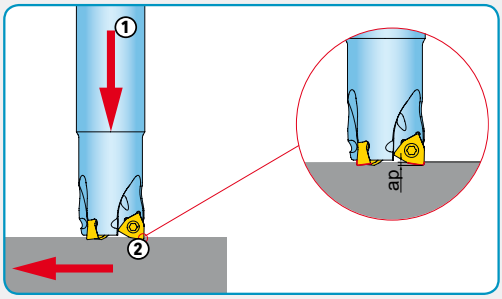
Ramping data and circular interpolation:

tool diameter [mm]	max. ramping angle [°]	min. bore dia. [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 Innotool partner.


insert:	WNMU0606_R	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:

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		
unalloyed steel	IN4030	250 – 290	IN2530	200 – 240	0.13 – 0.25
alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0.13 – 0.20
alloyed steel 1100 N/mm²	IN2505	160 – 180	IN2530	110 – 130	0.13
stainless steel	IN2035	120 – 180	IN2035	80 – 130	0.13 – 0.25
gray cast iron	IN2504	180 – 250	IN4030	150 – 200	0.13 – 0.25
nodular cast iron	IN2504	140 – 210	IN4030	110 – 160	0.13 – 0.20
aluminum	IN10K	800 – 1500	IN10K	500 – 800	0.13 – 0.35
high temperature alloys	IN2035	110 – 125	IN2530	60 – 80	0.13
titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0.13
hard machining < 54 HRC	IN2504	30 – 40	–	–	0.13
hard machining < 63 HRC	–	–	–	–	–

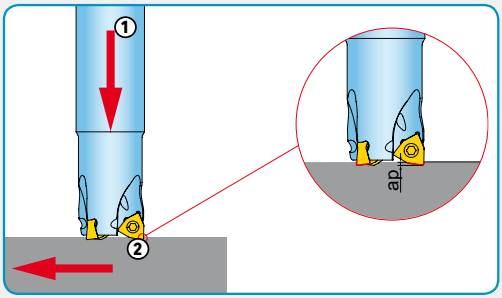
Ramping data and circular interpolation:

tool diameter [mm]	max. ramping angle [°]	min. bore dia. [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

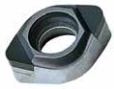


General information:

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 Innotool partner.




insert:	CNHU060310N	CNHU060315N	CNHU060304N-001	CNHU060310N-001	CNHU110420N
max. cutting depth = corner radius:	ap = 1 mm	ap = 1.5 mm	ap = 0.4 mm	ap = 1 mm	ap = 2 mm

Recommended cutting data:

material	cutting speed Vc [m/min]				feed per tooth fz [mm]
	semi-finishing		finishing		
unalloyed steel	IN2505	200 – 250	IN2005	300 – 500	0.1 – 0.3
alloyed steel 800 N/mm²	IN2505	180 – 250	IN2005	300 – 450	0.1 – 0.3
alloyed steel 1100 N/mm²	IN2505	180 – 220	IN2005	250 – 400	0.1 – 0.3
stainless steel	IN2505	100 – 180	IN2005	200 – 250	0.1 – 0.25
gray cast iron	IN2505	220 – 280	IN2005	300 – 600	0.15 – 0.3
nodular cast iron	IN2505	180 – 250	IN2005	250 – 450	0.1 – 0.3
aluminum	IN05S	500 – 1000	IN05S	800 – 1200	0.1 – 0.15
high temperature alloys	IN2005	40 – 100	IN2005	50 – 150	0.1 – 0.2
titanium alloys	IN2005	40 – 50	IN2005	50 – 80	0.1 – 0.2
hard machining < 54 HRC	IN2006	100 – 150	IN2006	120 – 180	0.08 – 0.15
hard machining < 63 HRC	IN2006	80 – 130	IN2006	100 – 150	0.07 – 0.12

Tips:

- For extreme cavities plunge milling is recommended, for finishing 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**

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 Innotool partner.

insert: PEMT0502ZCTR-HR

max. cutting depth: **ap = 1.0 mm**

Recommended cutting data:

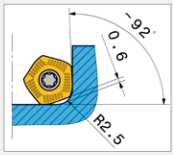
material	cutting speed Vc [m/min]				feed per tooth fz [mm]
	1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide		
unalloyed steel	IN2505	250 – 290	IN2530	200 – 240	0.40 – 1.50
alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0.40 – 1.00
alloyed steel 1100 N/mm²	IN2505	160 – 180	IN2530	110 – 130	0.40
stainless steel	IN2035	120 – 180	IN2035	80 – 130	0.40 – 1.50
gray cast iron	IN2504	180 – 250	IN2530	150 – 200	0.40 – 1.50
nodular cast iron	IN2504	140 – 210	IN2530	110 – 160	0.40 – 1.00
aluminum	–	–	–	–	–
high temperature alloys	IN2035	110 – 125	IN2035	60 – 80	0.40
titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0.40
hard machining < 54 HRC	IN2504	130 – 150	–	–	0.40
hard machining < 63 HRC	IN2504	110 – 130	–	–	0.40

Tips:

- For tightening the insert screws please always use a torque driver (1.1 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 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. uneven ground [mm]
20 R2.5	9.6	1	27.3	30	30.5	39
25 R2.5	5.2	1	37.3	40	40.5	49
32 R2.5	3.4	1	51.3	54	54.5	63
35 R2.5	3.0	1	57.3	60	60.5	69
40 R2.5	2.5	1	67.3	70	70.5	79
42 R2.5	2.3	1	71.3	74	74.5	83
50 R2.5	1.8	1	87.3	90	90.5	99
52 R2.5	1.7	1	91.3	94	94.5	103
63 R2.5	1.4	1	113.3	116	116.5	125
66 R2.5	1.4	1	119.3	122	122.5	131

General information:

insert screw: **SM25-064-00**
torque: **1.1 Nm**
torque wrench: **DTN011S with bit DS-T08TB**

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

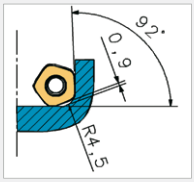

insert:	PNCQ0804ZNTN	PNCT0804ZNN-HR
max. cutting depth:	ap = 1.5 mm	ap = 1.5 mm

Recommended cutting data:

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		
unalloyed steel	IN4005	250 – 290	IN4030	200 – 240	0.50 – 2.50
alloyed steel 800 N/mm²	IN4005	210 – 250	IN4030	160 – 200	0.50 – 1.50
alloyed steel 1100 N/mm²	IN4005	160 – 180	IN4030	110 – 130	0.50 – 1.00
stainless steel	IN2505	120 – 180	IN2505	80 – 130	0.50 – 1.50
gray cast iron	IN4005	180 – 250	IN4030	150 – 200	0.50 – 2.50
nodular cast iron	IN4005	140 – 210	IN4030	110 – 160	0.50 – 1.50
aluminum	–	–	–	–	–
high temperature alloys	IN2505	110 – 125	IN2505	60 – 80	0.50 – 1.50
titanium alloys	IN2505	40 – 50	IN2505	30 – 40	0.50 – 1.50
hard machining < 54 HRC	IN4005	30 – 40	–	–	0.50
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.
- Programming radius R4.5.



Ramping angle and circular interpolation:

tool diameter [mm]	max. ramping angle [°]	min. bore dia. uneven ground [mm]	max. ap/rev. [mm]	min. bore dia. even ground [mm]	max. ap/rev. [mm]	max. bore dia. even ground [mm]	max. ap/rev [mm]	max. bore dia. uneven ground [mm]	max. ap/rev. [mm]
35 R4.5	0.15	54.9	0.1	55.5	0.1	56.5	0.1	68.7	0.2
42 R4.5	0.20	68.7	0.2	69.5	0.3	70.5	0.3	82.7	0.4
50 R4.5	0.20	84.4	0.3	85.5	0.3	86.4	0.3	98.7	0.5
52 R4.5	0.60	88.4	1.1	89.5	1.2	90.4	1.2	102.7	1.5
63 R4.5	0.70	109.1	1.5	111.4	1.5	112.4	1.5	124.7	1.5
66 R4.5	0.70	114.9	1.5	117.4	1.5	118.4	1.5	130.7	1.5
80 R4.5	0.75	142.4	1.5	145.4	1.5	146.4	1.5	158.7	1.5
100 R4.5	0.75	181.8	1.5	185.4	1.5	186.4	1.5	198.7	1.5
125 R4.5	0.95	230.7	1.5	235.4	1.5	236.4	1.5	248.7	1.5
160 R4.5	0.70	300.7	1.5	305.4	1.5	306.3	1.5	318.7	1.5

General information:

insert screw: **SM40-093-20**
torque: **4.5 Nm**
torque wrench: **DT-40-01 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 Innotool partner.






insert:	RCLT_CP	RCLT_CC	RCLT_CC1	RCLT_CC2	RCLT_PH	RCLT_PH2
factor for feed per tooth fz:	1	0.8	1	1.2	2	4
recom. cutting depth Ø12:	ap = 4 mm	–	ap = 3 mm	ap = 3 mm	ap = 1.5 mm	ap = 2 mm
recom. cutting depth Ø16:	ap = 6 mm	ap = 4 mm	ap = 4 mm	–	ap = 2 mm	ap = 2.5 mm

Recommended cutting data:

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		
unalloyed steel	IN4040	160 – 220	IN4030	130 – 180	0.15 – 0.25
alloyed steel 800 N/mm²	IN4040	140 – 200	IN4030	110 – 160	0.12 – 0.22
alloyed steel 1100 N/mm²	IN2005	120 – 180	IN4030	100 – 150	0.1 – 0.2
stainless steel	IN2035 / IN4030	90 – 150	IN2035 / IN4030	80 – 130	0.1 – 0.2
gray cast iron	IN4015	160 – 250	IN4015	140 – 200	0.15 – 0.25
nodular cast iron	IN4015	140 – 200	IN4015	120 – 170	0.12 – 0.22
aluminum	IN05S	500 – 1200	IN05S	500 – 1200	0.15 – 0.25
high temperature alloys	IN2035 / IN4030	50 – 80	IN2035 / IN4030	50 – 70	0.1 – 0.18
titanium alloys	IN2005	40 – 50	IN2035 / IN4030	30 – 40	0.1 – 0.15
hard machining < 54 HRC	–	–	–	–	–
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.

- 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 Innotool partner.

Ramping data and circular interpolation:

tool diameter [mm]	max. ramping angle [°]	min. bore dia. [mm]	max. feed per rev. [mm]	even bore dia. [mm]	max. feed per rev. [mm]	max. bore dia. [mm]	max. feed per 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: **DT-40-01 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 Innotool partner.






insert:	RH_06_	RH_08_	RH_10_	RH_12_	RH_16_	RH_20_
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 mm	fz = 0.6-1.5 mm
recom. cutting depth:	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	ap = 2.0-5.0 mm

Recommended cutting data:

material	cutting speed Vc [m/min]				insert type
	1st choice for inserts Ø 6 / 8 / 10 and 12		1st choice for inserts Ø 16 and 20		
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
stainless steel	IN2035 / IN4035 / IN7035	90 – 150	IN2035 / IN4035 / IN7035	80 – 130	RHHT / RHKT
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
aluminum	IN05S	500 – 1200	IN05S	500 – 1000	RHHT…P
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
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 Innotool partner.

Ramping data and circular interpolation:

tool dia. [mm]	max. ramping angle [°]	min. boring dia. [mm]	max. feed per rev. [mm]	even boring dia. [mm]	max. feed per rev. [mm]	max. boring dia. [mm]	max. feed per 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: **DT-40-01 with bit DS-T15B**
insert screw for RH_12_:	**SM40-080-10**	torque: **4.5 Nm**	torque wrench: **DT-40-01 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 Innotool partner.




insert: geometry:	RPLX10T3MON-HR positive geometry	RPLX10T3MOTN-HR pos. geometry, neg.chamfered	RPLX1204MON-HR1 positive geometry	RPLX1204MOTN-HR pos. geometry, neg.chamfered
recomm. cutting depth:	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 = 3.0 mm	ap = 3.0 mm

Recommended cutting data:

material		grade	cutting speed Vc [m/min]		feed per tooth fz [mm]		cutting depth ap [mm]	
			dry machining	wet machining	RPLX10	RPLX12	RPLX10	RPLX12
unalloyed steel		IN7035	150 - 350	120 - 250	0.15 - 0.40	0.25 - 0.50	0.80 - 2.50	1.00 - 3.00
alloyed steel 1100 N/mm²		IN7035	130 - 280	110 - 220	0.15 - 0.35	0.25 - 0.40	0.80 - 2.50	1.00 - 3.00
stainless steel	1.4021 X20Cr13	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	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	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	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	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	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	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	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	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	IN7035	50 - 150	40 - 80	0.10 - 0.30	0.15 - 0.40	0.80 - 2.50	1.00 - 3.00
high temperature materials	1.4826 GX40CrNiSi22-10	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	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	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	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	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	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	IN7035	120 - 210	90 - 160	0.10 - 0.30	0.15 - 0.40	0.80 - 2.50	1.00 - 3.00
	1.4939 X12CrNiMo12	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	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	IN7035	50 - 110	40 - 80	0.10 - 0.30	0.12 - 0.40	0.80 - 2.50	1.00 - 3.00
titanium alloys		IN7035	-	40 - 60	0.08 - 0.25	0.10 - 0.35	0.80 - 2.50	1.00 - 3.00
nickel alloys		IN7035	-	20 - 50	0.08 - 0.25	0.10 - 0.35	0.80 - 2.50	1.00 - 3.00

At max extension lengths the cutting speed 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 Innotool partner.

Ramping angle and circular interpolation:

tool diameter [mm]	max. ramping angle [°]	min. boring dia. [mm]	max. boring dia. [mm]	boring dia. even ground [mm]	max. feed per revolution [mm]	recom. feed per revolution [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: **SM35-076-10** torque: **3 Nm**
torque wrench: **DS-T15S**

insert screw: **SM35-090-00** torque: **3 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 Innotool partner.

insert:	SDMS13_-PH SDXS13_-PH	SDES13_MDR_	SDES13_MPR_ SDXS13_-MPR-MR	SDMS19_-PH	SDES19_MDR_	SDES19_MPR_ SDXS19_-MPR-MR
recom. cutting depth:	ap = 1.5 mm	ap = 1.8 mm	ap = 1.8 mm	ap = 2 mm	ap = 2.5 mm	ap = 2.5 mm
max. cutting depth:	ap = 2 mm	ap = 2 mm	ap = 2.2 mm	ap = 3 mm	ap = 3 mm	ap = 3.7 mm
ap_{max} with undercut at 90° shoulder:	ap_{max} = 2 mm	ap_{max} = 2 mm	ap_{max} = 2.2 mm	ap_{max} = 3 mm	ap_{max} = 3 mm	ap_{max} = 3.7 mm
ap_{max} without undercut at 90° shoulder:	ap_{max} = 0.5 mm	ap_{max}=0.95 mm	ap_{max} =1.45 mm	ap_{max} =1.1 mm	ap_{max} =1.1 mm	ap_{max} =1.5 mm
programming radius:	R 3.2	R 3.2	R 3.5	R 4.5	R 4.5	R 5.5

Recommended cutting data:

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		
unalloyed steel	IN4005	160 – 220	IN4030	130 – 180	1.5 – 3
alloyed steel 800 N/mm²	IN4005	140 – 200	IN4030	110 – 160	1.2 – 3
alloyed steel 1100 N/mm²	IN4005	120 – 180	IN4030	100 – 150	1 – 2.5
stainless steel	IN4035	90 – 150	IN4035	80 – 130	1 – 2
gray cast iron	IN2505	160 – 250	IN4030	140 – 200	1.5 – 3
nodular cast iron	IN2505	140 – 200	IN4030	120 – 170	1.2 – 3
aluminum	–	–	–	–	–
high temperature alloys	IN4035	50 – 80	IN4035	50 – 70	0.8 – 1.8
titanium alloys	–	–	IN4035	30 – 40	0.5 – 1.5
hard machining < 54 HRC	IN2505	60 – 100	–	–	0.8 – 2
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

Ramping angle and circular interpolation:

tool diameter [mm]	MDR / N and R geometry					MPR / MPR-MR geometry				
	max. ramp. angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. [mm]	max. ap/rev. [mm]	max. ramp. angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. [mm]	max. ap/rev. [mm]
32	15.0	38.1	2.0	64	2.0	10.8	38.1	2.2	64	2.2
35	10.8	43.3	2.0	70	2.0	8.1	44.1	2.2	70	2.2
40	7.4	54.0	2.0	80	2.0	5.7	54.0	2.2	80	2.2
42	6.6	58.0	2.0	84	2.0	5.1	58.0	2.2	84	2.2
50	4.7	74.0	2.0	100	2.0	3.8	73.9	2.2	100	2.2
52	4.3	78.0	2.0	104	2.0	3.4	77.9	2.2	104	2.2
63	3.1	100.0	2.0	126	2.0	2.5	99.9	2.2	126	2.2
66	2.9	106.1	2.0	132	2.0	2.3	105.9	2.2	132	2.2
80	1.8	134.1	2.0	160	2.0	1.3	134.3	2.2	160	2.2
100	1.3	174.1	2.0	200	2.0	1.0	174.3	2.2	200	2.2
80	4.3	121.8	3.0	160	3.0	3.6	121.9	3.7	160	3.7
100	3.0	161.8	3.0	200	3.0	2.5	161.9	3.7	200	3.7
125	2.1	211.8	3.0	250	3.0	1.8	211.9	3.7	250	3.7
160	1.5	281.8	3.0	320	3.0	1.3	281.9	3.7	320	3.7

General information:

insert size 13

insert screw: **SM40-100-R0**
torque: **4 Nm**
torque wrench: **DTNV00S with bit DS-T15TB**

insert size 19

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 Innotool partner.

insert:	 SDMS130516R-PP	 SDES130516N-PF1	 SDES130516N-PF	 SDES1305MPR SDXS1305MPR-MR SDES1305MPR-001
max. cutting depth ap:	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
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
programming radius:	6.4	6.4	6.4	6.14

Recommended cutting data:

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		
unalloyed steel	IN4005	160 – 220	IN4030	130 – 180	0.2 – 0.5
alloyed steel 800 N/mm²	IN4005	140 – 200	IN4030	110 – 160	0.2 – 0.5
alloyed steel 1100 N/mm²	IN4005	120 – 180	IN4030	100 – 150	0.2 – 0.5
stainless steel	IN4035	90 – 150	IN4035	80 – 130	0.2 – 0.45
gray cast iron	IN2505	160 – 250	IN4030	140 – 200	0.2 – 0.6
nodular cast iron	IN2505	140 – 200	IN4030	120 – 170	0.2 – 0.6
aluminum	–	–	–	–	–
high temperature alloys	IN4035	50 – 80	IN4035	50 – 70	0.2 – 0.45
titanium alloys	–	–	IN4035	30 – 40	0.2 – 0.45
hard machining < 54 HRC	IN4005	60 – 100	–	–	0.2 – 0.4
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

Ramping angle and circular interpolation:

tool diameter [mm]	PF / PF1 / PP geometry					MPR / MPR-001 geometry				
	max. ramp. angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. even ground [mm]	max. ap/rev. [mm]	max. ramp. 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 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 Innotool partner.

				
insert:	SDES130508N-PF SDES130508N-PF1	SDES130516N-PF SDES130516N-PF1	SDMS130512R-PP	SDMS130516R-PP
feed rate fz:	0.1 – 0.3 mm	0.1 – 0.3 mm	0.1 – 0.3 mm	0.1 – 0.3 mm
recommended width of cut ae:	9 mm	8 mm	8 mm	8 mm
max. width of cut ae:	11.9 mm	11.1 mm	11.6 mm	11.1 mm

Recommended cutting data:

material	SDES1305...		SDMS1305...	
	grade	cutting speed Vc [m/min]	grade	cutting speed Vc [m/min]
unalloyed steel	IN4005	150 - 200	–	–
alloyed steel 800 N/mm²	IN4005	130 - 180	–	–
alloyed steel 1100 N/mm²	IN4005	110 - 170	–	–
stainless steel	–	–	IN4035 / IN4030	80 - 150
gray cast iron	IN4015 / IN4005	160 - 220	–	–
nodular cast iron	IN4015 / IN4030	140 - 200	–	–
aluminum	–	–	–	–
high temperature alloys	–	–	IN4035	40 - 70
titanium alloys	IN4035	30 - 50	–	–
hard machining < 54 HRC	–	–	–	–
hard machining < 63 HRC	–	–	–	–

Tips:

- The worse the material machinability, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- The longer the extension length, the lower the cutting speed.
- 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 size 13:

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 Innotool partner.

insert:	 SDMS1906ZPR-PP	 SDMS190620R-PH	 SDES1906ZPR-PF	 SDES190620N SDES190620N-001	 SDES1906MPR SDXS1906MPR-MR SDES1906MPR-001
max. cutting depth ap:	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:

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		
unalloyed steel	IN4005	160 – 220	IN4030	130 – 180	0.25 – 0.6
alloyed steel 800 N/mm²	IN4005	140 – 200	IN4030	110 – 160	0.25 – 0.6
alloyed steel 1100 N/mm²	IN4005	120 – 180	IN4030	100 – 150	0.25 – 0.6
stainless steel	IN4035	90 – 150	IN4035	80 – 130	0.2 – 0.5
gray cast iron	IN2505	160 – 250	IN4030	140 – 200	0.25 – 0.8
nodular cast iron	IN2505	140 – 200	IN4030	120 – 170	0.25 – 0.8
aluminum	–	–	–	–	–
high temperature alloys	IN4035	50 – 80	IN4035	50 – 70	0.2 – 0.55
titanium alloys	–	–	IN4035	30 – 40	0.2 – 0.55
hard machining < 54 HRC	IN4005	60 – 100	–	–	0.2 – 0.5
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

Ramping angle and circular interpolation:

tool diameter [mm]	PF / PP geometry					PH / N / N-001 geometry				
	max. ramp. angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. even ground [mm]	max. ap/rev. [mm]	max. ramp. angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. even ground [mm]	max. ap/rev. [mm]
80	2.4	126.0	0.5 (PF)/1.3 (PP)	160.0	0.5 (PF)/1.3 (PP)	2.6	126.0	0.5	160.0	0.5
100	1.8	166.0	0.5 (PF)/1.3 (PP)	200.0	0.5 (PF)/1.3 (PP)	1.9	166.0	0.5	200.0	0.5
125	1.3	216.0	0.5 (PF)/1.3 (PP)	250.0	0.5 (PF)/1.3 (PP)	1.4	216.0	0.5	250.0	0.5
160	0.9	286.0	0.5 (PF)/1.3 (PP)	320.0	0.5 (PF)/1.3 (PP)	1.0	286.0	0.5	320.0	0.5

tool diameter [mm]	MPR / MPR-001 geometry				
	max. ramp. angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. even ground [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 Innotool 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:

material	cutting speed Vc [m/min]				recommended cutting depth ap [mm]	feed per tooth fz [mm]
	1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide			
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	120 – 180	IN4030	100 – 150	0.8 – 1.5	0.5 – 1.6
stainless steel	IN4035 / IN7035	90 – 150	IN4035 / IN7035	80 – 130	0.8 – 1.5	0.5 – 1.4
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
aluminum	–	–	–	–	–	–
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
hard machining < 48 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 machining, the smaller the tool engagement should be chosen.
- The smaller the cutting tool diameter, the higher the cutting speed can be.
- The starting feed rate should be reduced by 30%.
- 4-edged insert

Ramping data and circular interpolation:

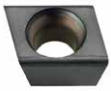
tool diameter [mm]	SDXS0904MPR-MR, -MM, -MR1 and -MRH geometry			
	max. ramping angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. [mm]
25	5.5	32.5	1.5	50.0
30	3.5	42.5	1.5	60.0
32	3.3	46.5	1.5	64.0
35	2.6	52.5	1.5	70.0
40	2.2	62.5	1.5	80.0
42	2.0	66.5	1.5	84.0
50	1.5	82.5	1.5	100.0
52	1.3	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 Innotool partner.




insert:	SHLT06_/SHGT06_	SPLT07_/SHGT07_	SHLT09_/SHGT09_	CDE313L_	DPM324L_
feed per tooth fz [mm]:	0.1 – 0.2	0.12 – 0.22	0.12 – 0.25	0.08 – 0.18	0.12 – 0.2
recomm. cutting depth [mm]:	step 3	step 4	step 6	step 5	step 8
max. cutting depth [mm]:	step 5	step 7	step 8	step 7	step 10

Recommended cutting data:

material	cutting speed Vc [m/min]					
	SHLT / SHGT / SPLT / SDGT		CDE313L...		DPM324L...	
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
stainless steel	IN2005	90 – 150	IN2005	90 – 150	IN2035	80 – 150
gray cast iron	IN2010	140 – 200	IN4015	140 – 200	IN4030	120 – 180
nodular cast iron	IN2010	120 – 180	IN4015	120 – 180	IN4030	100 – 160
aluminum	IN10K	300 – 800	IN05S	300 – 800	IN05S	300 – 800
high temperature alloys	IN2005	40 – 70	IN2005	40 – 70	IN2035	40 – 70
titanium alloys	IN2530	30 – 40	IN2005	30 – 40	IN2035	30 – 40
hard machining < 54 HRC	–	–	–	–	–	–
hard machining < 63 HRC	–	–	–	–	–	–

Tips:

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

General information SHLT06 / SHGT06:

insert screw:	**SM22-052-00**
torque:	**0.8 Nm**
torque wrench:	**DTNV01S with bit DS-T07TB**

General information SPLT07 / SDGT07:

insert screw:	**SM25-064-00**
torque:	**1.1 Nm**
torque wrench:	**DTN011S with bit DS-T08TB**

General information SHLT09 / SHGT09:

insert screw:	**SM35-088-60**
torque:	**3 Nm**
torque wrench:	**DTN030S with bit DS-T10TB**

General information DPM324L:

insert screw:	**SM40-120-20**
torque:	**4.5 Nm**
torque wrench:	**DT-40-01 with bit DS-T15B1**

General information CDE313L:

insert screw:	**SM40-090-00**
torque:	**4.5 Nm**
torque wrench:	**DT-40-01 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 Innotool partner.

insert: UOMT0602TR

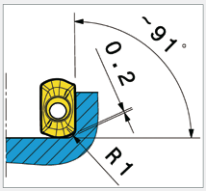
max. cutting depth: **ap = 0.5 mm**

Recommended cutting data:

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		
unalloyed steel	IN2505	250 – 290	IN2530	200 – 240	0.30 – 0.80
alloyed steel 800 N/mm²	IN2505	210 – 250	IN2530	160 – 200	0.30 – 0.50
alloyed steel 1100 N/mm²	IN2505	160 – 180	IN2530	110 – 130	0.30
stainless steel	IN2035	120 – 180	IN2035	80 – 130	0.30 – 0.80
gray cast iron	IN2504	180 – 250	IN2530	150 – 200	0.30 – 0.80
nodular cast iron	IN2504	140 – 210	IN2530	110 – 160	0.30 – 0.50
aluminum	–	–	–	–	–
high temperature iron	IN2035	110 – 125	IN2035	60 – 80	0.30
titanium alloys	IN2505	40 – 50	IN2530	30 – 40	0.30
hard machining < 54 HRC	IN2504	130 – 150	–	–	0.30
hard machining < 63 HRC	IN2504	110 – 130	–	–	0.30

Tips:

- For mounting the inserts please use only a torque wrench (0.5 Nm).
- The worse the material machinability, the less 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 diameter even ground [mm]	max. bore diameter even ground [mm]	max. bore dia. uneven ground [mm]
9.5	10.5	0.5	11	14.25	18
10 R1	10.0	0.5	12	15.25	19
11.5	7.0	0.5	15	18.25	22
12 R1	6.5	0.5	16	19.25	23
13.5	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 Innotool 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:

material	cutting speed Vc [m/min]				recommended cutting depth ap [mm]	feed per tooth fz [mm]
	1st choice dry machining resp. wear resistant carbide		1st choice wet machining resp. tough carbide			
unalloyed steel	IN2505	160 – 220	IN2530	130 – 180	0.5 – 0.8	0.5 – 1.0
alloyed steel 800 N/mm²	IN2505	140 – 200	IN2530	110 – 160	0.5 – 0.8	0.5 – 1.0
alloyed steel 1100 N/mm²	IN2505	120 – 180	IN2530	100 – 150	0.5 – 0.8	0.5 – 1.0
stainless steel	IN2530 / IN4035	90 – 150	IN2530 / IN4035	80 – 130	0.5 – 0.8	0.5 – 0.9
gray cast iron	IN2505	160 – 250	IN4030	140 – 200	0.5 – 0.8	0.5 – 1.1
nodular cast iron	IN2505	140 – 200	IN4030	120 – 170	0.5 – 0.8	0.5 – 1.1
aluminum	–	–	–	–	–	–
high temperature alloys	–	–	IN4035	50 – 70	0.5 – 0.7	0.5 – 0.9
titanium alloys	–	–	IN4035	30 – 40	0.5 – 0.7	0.5 – 0.9
hard machining < 48 HRC	IN2504	60 – 100	–	–	0.3 – 0.6	0.5 – 0.7
hard machining < 63 HRC	–	–	–	–	–	–

Tips:

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

Ramping data and circular interpolation:

tool diameter [mm]	max. ramping angle [°]	min. bore dia. [mm]	max. ap/rev. [mm]	max. bore dia. [mm]
16	14.4	19.8	0.7	32.0
20	5.9	27.6	0.7	40.0
25	5.3	37.6	0.7	50.0
30	3.5	47.6	0.7	60.0
32	3.1	51.6	0.7	64.0
35	2.2	57.6	0.7	70.0
40	2.1	67.6	0.7	80.0
42	1.6	71.6	0.7	84.0
50	1.3	87.6	0.7	100.0
52	1.2	91.6	0.7	104.0

General information:

insert screw: **SM25-054-00**
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 Innotool partner.

					
insert:	WNMU04T302N	WNMU04T304N	WNCU04T302FN-P	WNCU04T304FN-P	WNCU04T308FN-P
feed per tooth fz [mm]:	0.07 – 0.20	0.07 – 0.20	0.07 – 0.20	0.07 – 0.20	0.07 – 0.20
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
effective tool diameter [mm]:	nom. Ø +0.1	nom. Ø	nom. Ø +0.1	nom. Ø	nom. Ø –0.1

Recommended cutting data:

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		
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
stainless steel	IN2505	120 – 180	IN4030 / IN6535	80 – 130	0.07 – 0.18
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
aluminum	IN10K	800 – 1500	IN10K	500 – 800	0.07 – 0.20
high temperature alloys	IN2505	110 – 125	IN4030 / IN6535	60 – 80	0.07
titanium alloys	IN2505	40 – 50	IN4030 / IN6535	30 – 40	0.07
hard machining < 54 HRC	IN2504	30 – 40	–	–	0.07
hard machining < 63 HRC	–	–	–	–	–

Tips:

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

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



Recommended cutting data:

material	DC [mm]	cutting speed vc [m / min]	feed rate per tooth fz [mm]	cutting depth ap max [mm]	grade
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 < 800N/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 < 1100N/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
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
gray cast iron cast alloy	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
non-ferrous metals plastics	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
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

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 Innotool partner.

Z = 3/4/5/7



Recommended cutting data:

material	DC [mm]	cutting speed vc [m / min]	feed rate per tooth fz [mm]	cutting depth ap max [mm]	grade
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 < 800N/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 < 1100N/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
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
gray cast iron cast alloy	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
non-ferrous metals plastics	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
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

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 Innotool partner.

Recommended cutting data:

material	DC [mm]	cutting speed vc [m / min]	feed rate per tooth fz [mm]	cutting depth ap max [mm]	grade
unalloyed steel	3	140 – 200	0.02 – 0.04	1.5 x Ø	IN2005
	4	140 – 200	0.02 – 0.05	1.5 x Ø	IN2005
	5	140 – 200	0.025 – 0.06	1.5 x Ø	IN2005
	6	140 – 200	0.025 – 0.07	1.5 x Ø	IN2005
	8	140 – 200	0.03 – 0.09	1.5 x Ø	IN2005
	10	140 – 200	0.03 – 0.10	1.5 x Ø	IN2005
	12	140 – 200	0.035 – 0.11	1.5 x Ø	IN2005
	16	140 – 200	0.05 – 0.13	1.5 x Ø	IN2005
	20	140 – 200	0.05 – 0.17	1.5 x Ø	IN2005
	25	140 – 200	0.07 – 0.20	1.5 x Ø	IN2005
alloyed steel < 800N/mm²	3	140 – 200	0.02 – 0.04	1.5 x Ø	IN2005
	4	140 – 200	0.02 – 0.05	1.5 x Ø	IN2005
	5	140 – 200	0.025 – 0.06	1.5 x Ø	IN2005
	6	140 – 200	0.025 – 0.07	1.5 x Ø	IN2005
	8	140 – 200	0.03 – 0.09	1.5 x Ø	IN2005
	10	140 – 200	0.03 – 0.10	1.5 x Ø	IN2005
	12	140 – 200	0.035 – 0.11	1.5 x Ø	IN2005
	16	140 – 200	0.05 – 0.13	1.5 x Ø	IN2005
	20	140 – 200	0.05 – 0.17	1.5 x Ø	IN2005
	25	140 – 200	0.07 – 0.20	1.5 x Ø	IN2005
alloyed steel < 1100N/mm²	3	120 – 180	0.02 – 0.04	1.5 x Ø	IN2005
	4	120 – 180	0.02 – 0.05	1.5 x Ø	IN2005
	5	120 – 180	0.025 – 0.06	1.5 x Ø	IN2005
	6	120 – 180	0.025 – 0.07	1.5 x Ø	IN2005
	8	120 – 180	0.03 – 0.09	1.5 x Ø	IN2005
	10	120 – 180	0.03 – 0.10	1.5 x Ø	IN2005
	12	120 – 180	0.035 – 0.11	1.5 x Ø	IN2005
	16	120 – 180	0.05 – 0.13	1.5 x Ø	IN2005
	20	120 – 180	0.05 – 0.17	1.5 x Ø	IN2005
	25	120 – 180	0.07 – 0.20	1.5 x Ø	IN2005
stainless steel	3	60 – 120	0.02 – 0.03	0.5 – 0.8 x Ø	IN2005
	4	60 – 120	0.02 – 0.035	0.5 – 0.8 x Ø	IN2005
	5	60 – 120	0.02 – 0.04	0.5 – 0.8 x Ø	IN2005
	6	60 – 120	0.02 – 0.05	0.5 – 0.8 x Ø	IN2005
	8	60 – 120	0.02 – 0.07	0.5 – 0.8 x Ø	IN2005
	10	60 – 120	0.02 – 0.08	0.5 – 0.8 x Ø	IN2005
	12	60 – 120	0.03 – 0.09	0.5 – 0.8 x Ø	IN2005
	16	60 – 120	0.04 – 0.10	0.5 – 0.8 x Ø	IN2005
	20	60 – 120	0.04 – 0.13	0.5 – 0.8 x Ø	IN2005
	25	60 – 120	0.05 – 0.15	0.5 – 0.8 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 Innotool partner.

Recommended cutting data:

gray cast iron cast alloy	3	160 – 220	0.02 – 0.04	1.5 x Ø	IN2005
	4	160 – 220	0.02 – 0.05	1.5 x Ø	IN2005
	5	160 – 220	0.025 – 0.06	1.5 x Ø	IN2005
	6	160 – 220	0.025 – 0.07	1.5 x Ø	IN2005
	8	160 – 220	0.03 – 0.09	1.5 x Ø	IN2005
	10	160 – 220	0.03 – 0.10	1.5 x Ø	IN2005
	12	160 – 220	0.035 – 0.11	1.5 x Ø	IN2005
	16	160 – 220	0.05 – 0.13	1.5 x Ø	IN2005
	20	160 – 220	0.05 – 0.17	1.5 x Ø	IN2005
	25	160 – 220	0.07 – 0.20	1.5 x Ø	IN2005
super alloys	3	40 – 80	0.02 – 0.03	0.5 – 0.8 x Ø	IN2005
	4	40 – 80	0.02 – 0.035	0.5 – 0.8 x Ø	IN2005
	5	40 – 80	0.02 – 0.04	0.5 – 0.8 x Ø	IN2005
	6	40 – 80	0.02 – 0.05	0.5 – 0.8 x Ø	IN2005
	8	40 – 80	0.02 – 0.07	0.5 – 0.8 x Ø	IN2005
	10	40 – 80	0.02 – 0.08	0.5 – 0.8 x Ø	IN2005
	12	40 – 80	0.03 – 0.09	0.5 – 0.8 x Ø	IN2005
	16	40 – 80	0.04 – 0.10	0.5 – 0.8 x Ø	IN2005
	20	40 – 80	0.04 – 0.13	0.5 – 0.8 x Ø	IN2005
	25	40 – 80	0.05 – 0.15	0.5 – 0.8 x Ø	IN2005
hardened steel < 54 HRC	3	80 – 140	0.01 – 0.03	0.3 x Ø	IN2005
	4	80 – 140	0.01 – 0.035	0.3 x Ø	IN2005
	5	80 – 140	0.01 – 0.04	0.3 x Ø	IN2005
	6	80 – 140	0.01 – 0.04	0.3 x Ø	IN2005
	8	80 – 140	0.02 – 0.04	0.3 x Ø	IN2005
	10	80 – 140	0.05 – 0.05	0.3 x Ø	IN2005
	12	80 – 140	0.02 – 0.07	0.3 x Ø	IN2005
	16	80 – 140	0.03 – 0.08	0.3 x Ø	IN2005
	20	80 – 140	0.03 – 0.10	0.3 x Ø	IN2005
	25	80 – 140	0.05 – 0.12	0.3 x Ø	IN2005

Cutting data for finishing:

Cutting speed	vc = 20 – 30%	increase
Feed rate per tooth	lowest value in table	
Width of cut	ae = 0.1 – 0.3mm	

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 Innotool partner.

Recommended cutting data:

material	DC [mm]	cutting speed vc [m / min]	feed rate per tooth fz [mm]	cutting depth ap max [mm]	length of cut [mm]	grade
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	0.04 – 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 < 800N/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
unalloyed steel < 1100N/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
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
gray cast iron cast alloy	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
non-ferrous metals plastics	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
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
hardened steel < 54 HRC	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 Innotool partner.

Indexable Drills



Recommended cutting data:

material	Vc [m/min]	feed rate per tooth fz [mm/tooth]			1st choice grade peripheral / center
		SCLT050204N-PH SHGT050204-HP SCLT050204N	SHLT060204N-PH SHGT060204-HP SHLT060204N	SPLT07T308N-PH SDGT07T308-HP SPLT07T308N	
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
stainless steel	150 – 220	0.05 – 0.11	0.06 – 0.12	0.08 – 0.15	IN2005
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
aluminum	300 – 600	0.05 – 0.12	0.08 – 0.15	0.10 – 0.20	IN10K
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

material	Vc [m/min]	feed rate per tooth fz [mm/tooth]			1st choice grade peripheral / center
		SHLT090408N-PH1 SHGT090408-HP SHLT090408N	SHLT110408N-PH1 SHGT110408-HP SHLT110408N	SPLT140512N-PH SDGT140512-HP	
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
stainless steel	150 – 220	0.09 – 0.16	0.10 – 0.17	0.11 – 0.19	IN2005
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
aluminum	300 – 600	0.12 – 0.22	0.14 – 0.23	0.15 – 0.26	IN10K
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

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**

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 Innotool partner.

STANDARD PLUS

Innotool'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, drills, solid carbide, adaptions, set-up equipment and indexable inserts.

The development and production of special-purpose tools according to customer-specific requirements is another important factor for Innotool.

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.

INNOTOOL

INNOTOOL

INNOVATIVE CUTTING TOOLS
Florianstraße 13–17
71665 Vaihingen-Horrheim, Germany
Tel.: +49 (0)7042-8316-0
Fax: +49 (0)7042-8316-26
E-Mail: info@innotool.de

www.innotool.de

Order-No.: 3060189 / version 07/19