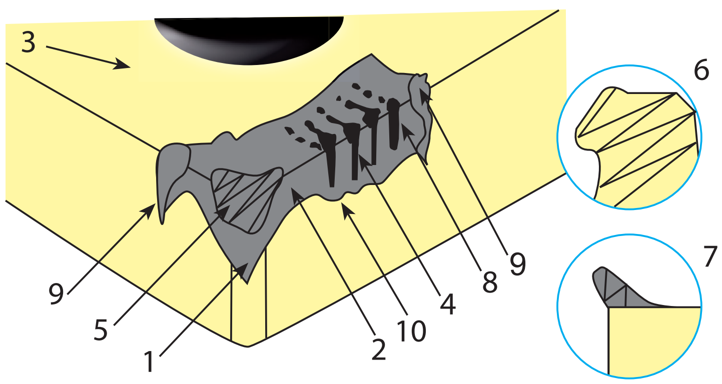
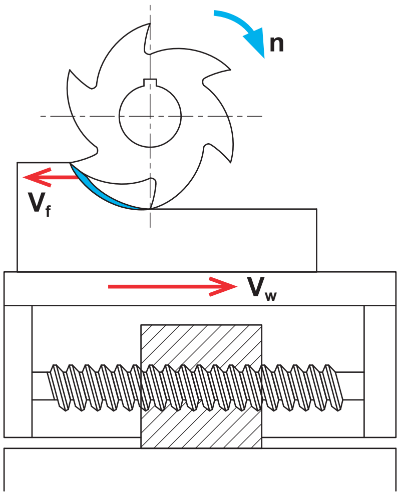
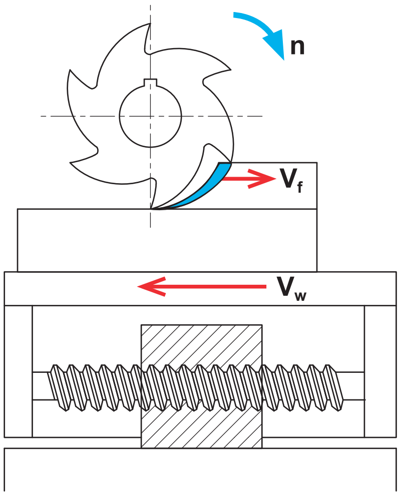
Trouble shooting – indexable milling

Fig.	Type of wear	Effects	Reason	Countermeasure
1+2	Flank wear	– Bad surface quality and dimensional stability – Increase of cutting force	– Grade not wear-resistant enough – Cutting speed too high – Clearance angle too small – Feed rate too low	– Grade with higher wear-resistance – Reduce cutting speed – Increase clearance angle – Reduce feed rate
3	Crater wear	– Bad surface quality and chip control	– Grade not wear-resistant enough – Cutting speed too high – Feed rate too low	– Grade with higher wear-resistance – Reduce cutting speed – Reduce feed rate
4	Chipping	– Unstable tool life – Sudden breakage of cutting edge	– Grade too hard – Feed rate too high – Cutting edge not stable enough – Stability of the holder or tension insufficient	– Grade with higher toughness – Reduce feed rate – Change honing of cutting edge – Use a more stable tool holder
5	Breakage	– Increase of cutting force – Bad surface quality and dimensional stability	– Grade too hard – Feed rate too high – Cutting edge not stable enough – Stability of the holder or tension insufficient	– Grade with higher toughness – Reduce feed rate – Change honing of cutting edge – Use a more stable tool holder
6	Plastic deformation	– Bad dimensional stability – Damage to cutting edge	– Grade not wear-resistant enough – Cutting speed too high – Cutting depth and/or feed rate too high – Temperature on the cutting edge too high	– Grade with higher toughness – Reduce cutting speed – Reduce cutting depth and feed rate – Grade with higher heat-resistance
7	Welding	– Increase of cutting force – Bad surface quality	– Cutting speed too low – Cutting edge not sharp enough – Grade not suitable	– Increase cutting speed – Increase rake angle – Use a more suitable grade
8	Thermal cracks	– Breakage due to thermal interaction, often caused when cutting is interrupted (milling)	– Temperature fluctuation when machining – Grade too hard	– Dry machining – Grade with higher toughness
9	Notch wear	– Burr formation – Increase of cutting force	– Damage through chips (jagged edges) – Feed rate and cutting speed too high	– Grade with higher wear-resistance – Increase rake angle to get a sharper cutting edge – Reduce cutting speed
10	Flaking (coating)	– Often appears when machining hardened materials or caused by vibration	– Cutting edge adhesion and chipping – Bad chip removal	– Increase rake angle to get a sharper cutting edge – Chip breaker with bigger chip space



Indexable milling

Difference between up-milling and down-milling

	 V_f Feed rate tool V_w Feed rate work piece n Rotation
Up-milling	Down-milling
Up-milling: the feed direction of the work piece is opposite to that of the milling rotation at the connecting position.	Down-milling: the feed direction of the work piece is the same as that of the milling rotation at the connecting position.

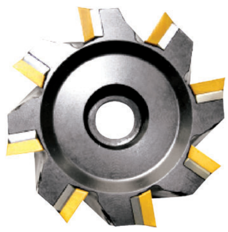
Advantages and disadvantages

Direction	Advantages	Disadvantages
Up-milling	– Prevents hooking of tool – More smooth cut	– Bigger stress on cutting edge – Shorter tool life
Down-milling	– Higher tool life – Less thermal stress	– Hooking of tool possible

Indexable milling

Pitch selection

The pitch is the distance between one point on one cutting edge and the same point on the next edge.
Milling cutters are mainly classified into wide, normal and fine pitches.

Operational stability		
L (low)	M (medium)	H (high)
Wide pitch	Normal pitch	Fine pitch
		
When the milling width is equal to the diameter of the cutter, the machining system is stable and main power of machine is sufficient, selecting a wide pitch can achieve high productive efficiency.	General milling function and multiple mixed productions.	When the milling width is less than the diameter of cutter, cutting by maximum edges can achieve high productive efficiency.

Approach angle

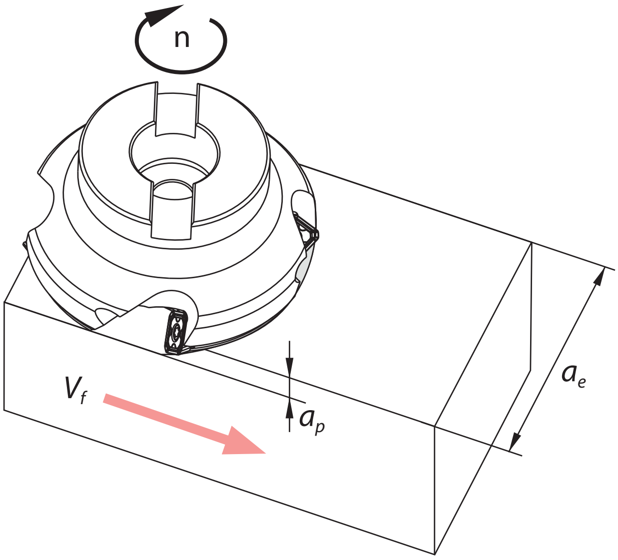
The approach angle is composed by insert. Tool body, chip thickness, cutting forces and tool life are affected especially by the approach angle. Decreasing the approach angle reduces chip thickness and spreads the cutting area between cutting edge and work piece for a given feed rate.
A smaller approach angle also guarantees stable entering or exiting the work piece, to protect the cutting edge and extend tool life. However this will increase higher axial cutting forces on the work piece, thus it is not suitable for machining thin work piecees such as thin plates.

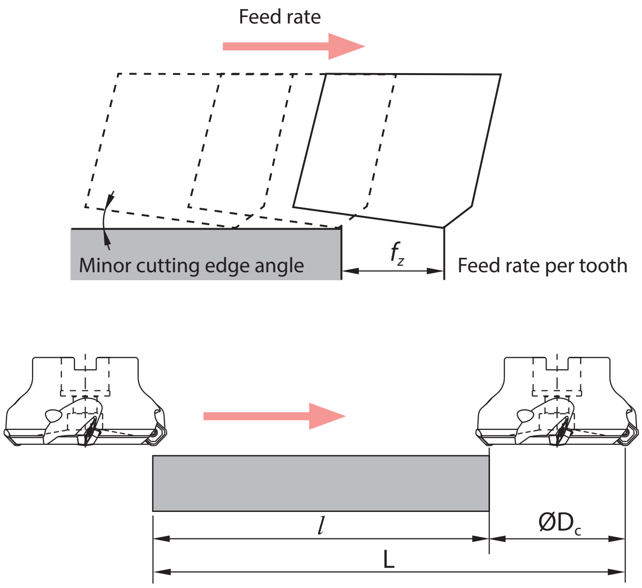
Approach angle	Feed rate per tooth	Max. cutting depth
90°	f_z	$h_{ex} = f_z \times sin_{kr}$
75°		$h_{ex} = 0{,}96 \times f_z$
60°		$h_{ex} = 0{,}86 \times f_z$
45°		$h_{ex} = 0{,}707 \times f_z$
Round		$h_{ex} = \frac{\sqrt{iC^2 \times (iC - 2a_p)^2}}{iC} \times f_z$

A Turning
B Milling
C Drilling
D Technical Information
E Index

Indexable milling

General formulas



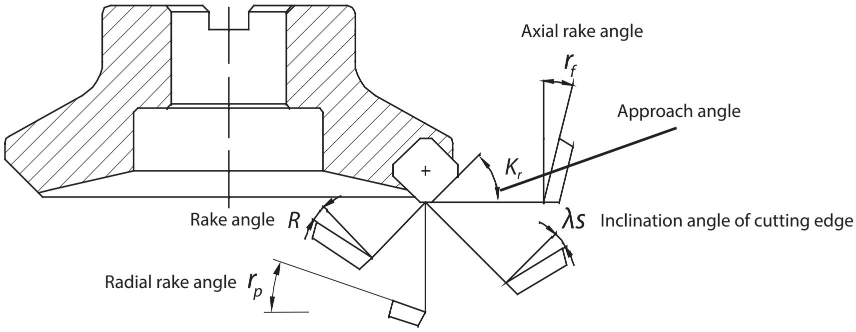


V_c: Feed rate [m/min]	V_f: Feed rate of worktable (feed speed) [mm/min]
D_c: Nominal diameter of milling tool [mm]	f_z: Feed rate per tooth [mm/z]
n: Spindle speed [u/min]	π: ~3,14
z_n: Number of teeth	T_c: Machining time [min]
Q: Metal removal rate [cm^3/min]	f_n: Feed rate per revolution [mm/u]

Cutting speed	$V_c = \frac{\pi \times D_c \times n}{1000}$ [m/min]
Spindle speed	$n = \frac{1000 \times V_c}{\pi \times D_c}$ [rev/min]
Feed rate of work table	$V_f = f_z \times n \times z_n$ [mm/min]
Feed rate per tooth	$f_z = \frac{V_f}{n \times Z_n}$ [mm/z]
Feed rate per revolution	$f_n = \frac{V_f}{n}$ [mm/rev]
Machining time	$T_c = \frac{1000 \times V_c}{\pi \times D_c}$ [min]
Metal removal rate	$Q = \frac{a_p \times a_e \times V_f}{1000}$ [cm^3/min]

Indexable milling

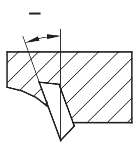
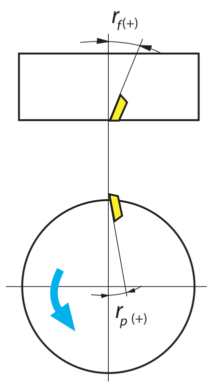
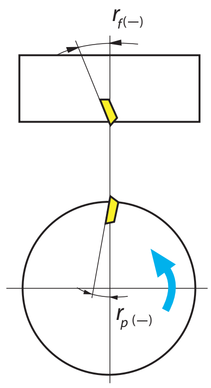
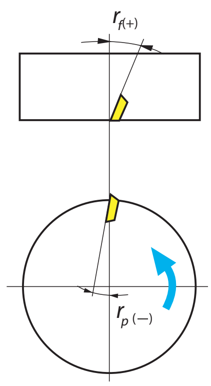
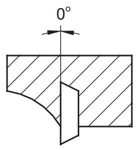
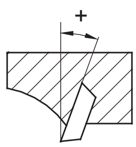
Function of angles when face milling



Main angles

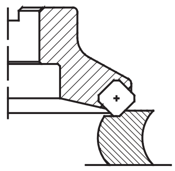
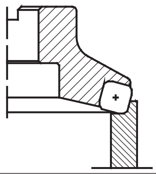
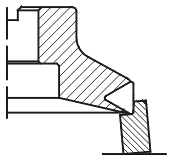
Angle	Feature	Effet		
Axial rake angle r_f	Influences chip direction	Negative angle, good chip removal		
Radial rake angle r_p	Influences cutting edge sharpness	Positive angle, good cutting performance		
Approach angle ***Kr***	Influences chip thickness	Kr↑, chip thickness↑, Kr↓, chip thickness↓;		
Rake angle ***R***	Influences cutting force	Poor cutting performance, stable cutting edge	(−)←0→(+)	Good cutting performance, unstable cutting edge
Inclination angle ***λs***	Influences chip flow direction	Poor cutting performance, stable cutting edge	(−)←0→(+)	Good cutting performance, unstable cutting edge

Combination of different rake angles

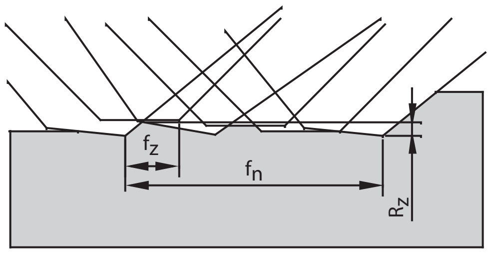
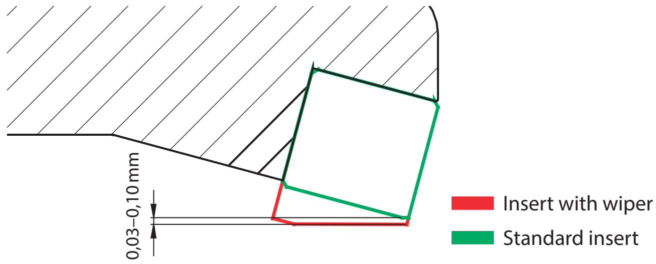
		Double positive	Double negative	Positive/Negative
Negative rake angle				
Neutral angle				
Positive angle				
Axial rake angle r_f		+	–	+
Radial rake angle r_p		+	–	–
Application field	**P**	√		√
	M	√		√
	K		√	√
	N	√		
	S	√		√

Indexable milling

Cutting performances of different approach angles

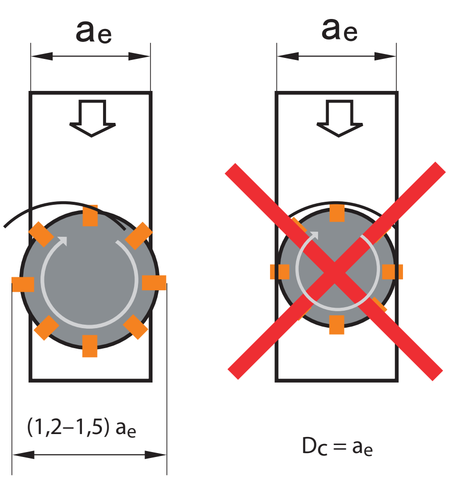
Approach angle	Depiction	Explanation
45°		Axial force is largest. It will bend when machining thin-wall work piece, and reduces the precision of work piece. It is benefit to avoid fringe breakage of work piece when machining cast iron.
75°		The main purpose is to resolve the radial cutting force, it is often used for general face milling.
90°		The axial force is zero in theory, suitable for milling thin plate workpiece.

Inserts with wiper

Using standard inserts	Using inserts with wiper
	
Normal surface quality	High surface quality

The wiper insert must protrude below the other inserts by 0.03–0.10 mm at axial direction, only that the wiping function can take into effect. Generally speaking, a cutter can assemble only one wiper insert. If the diameter of cutter is much bigger or cutter's feed rate per revolution is bigger than the length of wiper edge, 2 to 3 wiper inserts can be assembled.

Cutting width



Generally speaking, the relation between cutting width and tool cutting diameter is $D_c = (1.2–1.5)\ a_e$.

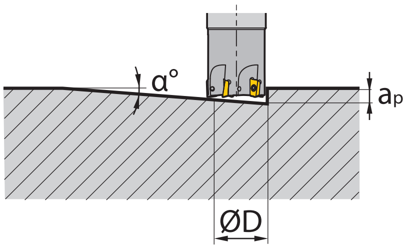
In the machining practice, it needs to avoid coincidence of tool center and workpiece center as much as possible.

D_c: Tool diameter
a_e: Lateral infeed

Indexable milling

Plunging and circular milling with insert APKT

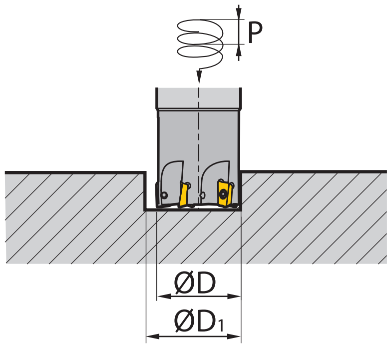
Plunging



$$L_m = \frac{a_p}{\tan_\alpha}$$

α : Angle de plongée

Circular milling



$$P = \tan_\alpha \times \pi \times D_1$$

α : Angle d'hélice

Insert	Diameter ØD [mm]	Max. cutting depth a_p [mm]	Max. plunge angle α°	Min. diameter $ØD_1$ [mm]	Max. diameter [mm]
AP**11**	16	10	10	20	30
	20	10	5	28	38
	25	10	4	40	48
	32	10	3	56	60
	40	10	2	70	76

Reduce the feed rate when plunging and circular milling.
For drilling operations (axial) set the feed rate under 0.2 mm.
„Attention“ – drilling can form long chips.

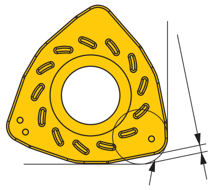
Indexable milling

Plunging and circular milling with insert WPGT or SDMT

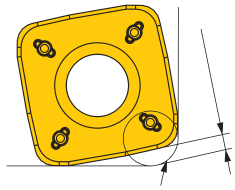
Approx. programmed radius

Insert	approx. R [mm]	Residual material K [mm]
WPGT050315ZSR	2	0,5
WPGT060415ZSR	2,5	0,7
WPGT080615ZSR	2,5	0,7
WPGT090725ZSR	4,5	1,2
SDMT06T208	1,6	0,5
SDMT09T312	2,5	0,87
SDMT120412	4,0	0,93
SDMT150620	4,0	1,38

WPGT

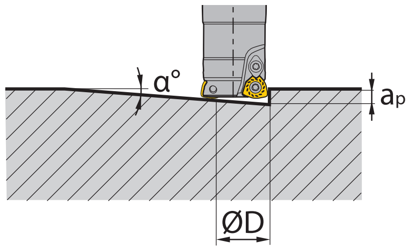


SDMT



Insert WPGT

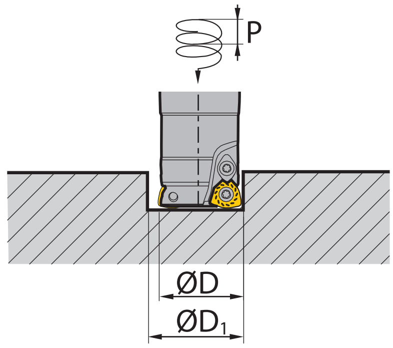
Plunging



$$L_m = \frac{a_p}{\tan_\alpha}$$

α : Angle de plongée

Circular milling



$$P = \tan_\alpha \times \pi \times D_1$$

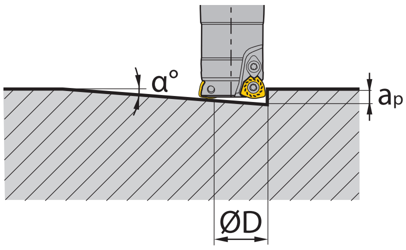
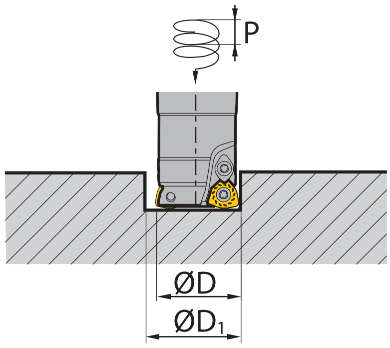
α : Angle d'hélice

Insert	Diameter ØD [mm]	Plunging: Max. cutting depth a_p [mm]	Plunging: Max. plunge angle α°	Circular milling: Min. diameter $ØD_1$ [mm]	Circular milling: Max. diameter [mm]
WP**05**	20	1,5	12	24	37
WP**06**	25	1,5	8,8	31	47
	32	1,5	5	45	61
	40	1,5	3,2	61	77
	50	1,5	2,8	81	97
WP**08**	40	1,5	9	52	77
	50	1,5	5,4	71	97
	63	1,5	4,3	97	123
	80	1,5	2,9	131	157
	100	1,5	2,1	171	197
	125	1,5	1,3	221	247
	160	1,5	1,1	291	317
WP**09**	50	3,0	7,2	70	96
	63	3,0	4,5	96	122
	80	3,0	2,8	130	156
	100	3,0	2,2	170	196
	125	3,0	1,6	220	246
	160	3,0	1,2	290	316

Reduce the feed rate when plunging and circular milling.
For drilling operations (axial) set the feed rate under 0.2 mm.
„Attention" – drilling can form long chips.

Indexable milling

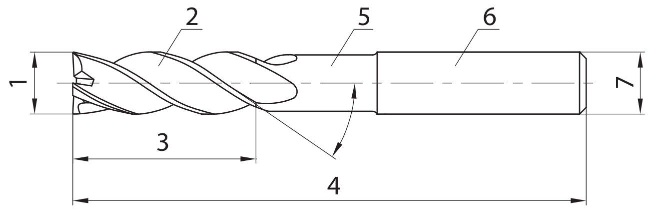
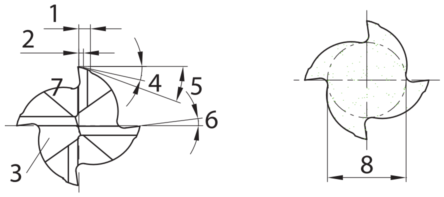
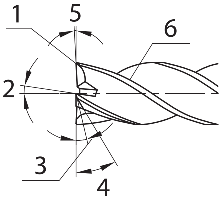
Insert SDMT

		Plunging		Circular milling	
		 $L_m = \frac{a_p}{\tan_\alpha}$ α: Plunge angle		 $P = \tan_\alpha \times \pi \times D_1$ α: Helix angle	
Insert	Diameter ØD (mm)	Max. cutting depth a_p [mm]	Max. plunge angle α°	Min. diameter $ØD_1$ [mm]	Max. diameter [mm]
SD**06**	20	0,8	3,6	30	38
	25	0,8	2,8	40	48
	32	0,8	1,6	52	60
	40	0,8	1,1	70	78
	50	0,8	0,8	90	98
	63	0,8	0,7	114	122
SD**09**	25	1,4	6,5	34	48
	32	1,4	4,5	48	62
	35	1,4	3,6	54	68
	50	1,4	1,8	84	98
	63	1,4	1,3	110	124
SD**12**	32	1,8	10,4	44	60
	40	1,8	5,7	60	76
	50	1,8	3,5	80	96
	63	1,8	2,5	106	122
	80	1,8	1,6	140	156
	100	1,8	1,2	180	196
SD**15**	40	2,2	7,3	54	76
	80	2,2	1,4	134	156
	100	2,2	1,0	174	196
	125	2,2	0,9	234	246
	160	2,2	0,6	304	316

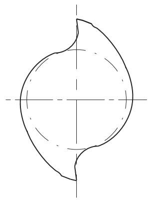
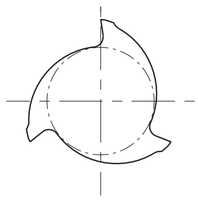
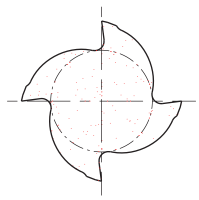
Reduce the feed rate when plunging and circular milling.
For drilling operations (axial) set the feed rate under 0,2 mm.
„Attention" – drilling can form long chips.

Solid carbide mills

Terminology

A		1. Cutting edge diameter 2. Chip pocket 3. Length of cutting edge 4. Total length 5. Neck 6. Shank 7. Shank diameter
B		1. Chamfer width, main cutting edge 2. Chamfer width, diameter 3. Neck, front side 4. Primary radial clearance angle 5. Secondary radial clearance angle 6. Radial rake angle 7. Axial main cutting edge 8. Core diameter
C		1. Cutting edge 2. Axial rake angle 3. Primary axial clearance angle 4. Secondary axial clearance angle 5. Inclination angle 6. Radial cutting edge

Teeth, chip pocket and tool rigidity

Teeth	2 flutes	3 flutes	4 flutes
Cross section			
Cutting edge ratio	**54 %**	**56 %**	**60 %**
Advantages	– Large chip pocket – Good chip removal	– Good chip removal – Good surface quality	– Good rigidity – Good surface
Application	– Slot milling – Square shoulder milling – Drilling	– Slot milling – Square shoulder milling – Finishing	– Slot milling (flat) – Square shoulder milling – Finishing

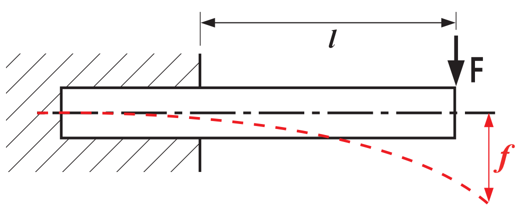
Length of cutting edge (overhang) and cutting diameter

The shorter the overhang, the stronger the rigidity. Thus isn't easy to generate. Bend and vibration in the cutting process may occur.

Length (overhang) increases by 1 time, the deflection degree (f) will be 8 times of the former one.

Reduce the overhang by 20 %
the deflection degree (f) will decrease by 50 %

Increase the diameter by 20 %
the deflection degree (f) will decrease by 50 %



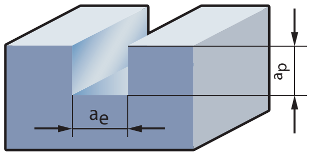
$$f = \frac{F \times l^3}{3 \times E \times I} = \frac{F \times l^3 \times 64}{3 \times E \times I}$$

A Turning
B Milling
C Drilling
D Technical Information
E Index

Solid carbide mills

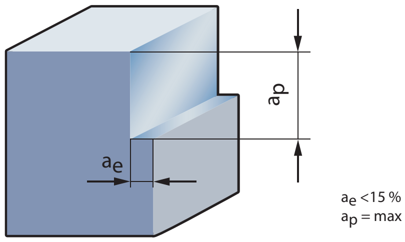
Machining strategy – HPC/UM (HSC) milling cutters

HPC = High Performance Cutting
Machining with significantly increased metal removal rate through higher cutting speeds and feed rates compared with conventional machine cutting processes.

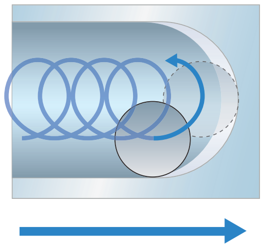


Full-slot milling

HSC (UM) = High Speed Cutting
High cutting speeds and feed rates in combination with low cutting depths lead to lower chip thickness as in normal machining.

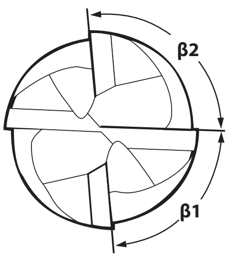


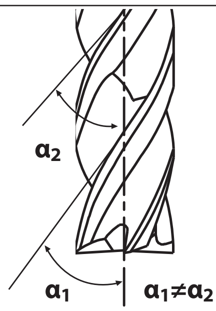
Profiling



Trochoidal milling

The UM milling cutter is specifically optimised for HSC machining.



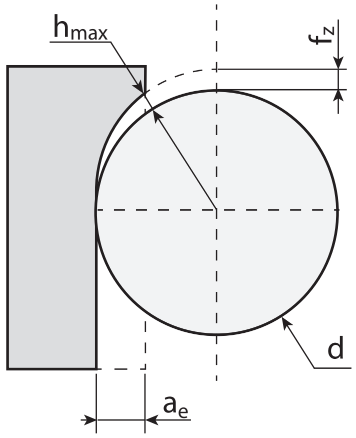


High metal removal rates can be realised with this tool.
Especially on highly dynamic machines with optimised tool paths this milling cutter shows its full potential.

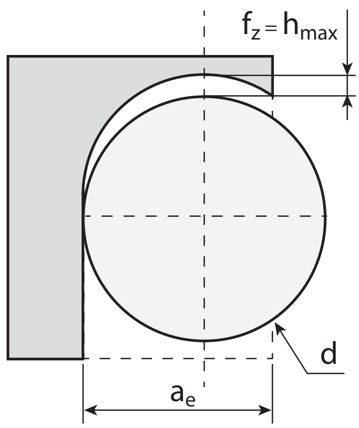
Solid carbide mills

HSC strategy

It's important to use the right strategy. When programming make sure the width of cut is kept. The width of cut is usually not higher than 15 %. Regarding the depth of cut, the total length of the cutting edge can be used.



$a_e < d/2 \quad h_{max} < f_z$

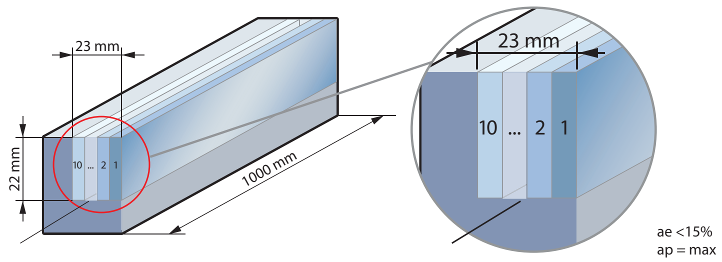


$a_e \geq d/2 \quad h_{max} = f_z$

$$h_{max} = 2f_z\sqrt{\frac{a_e}{d}\left(1-\frac{a_e}{d}\right)}$$

When changing the width of cut the cutting data needs to be adjusted.
As calculatory size applies a chip thickness from approx. 0.15 –0.2 mm on basis of the usual steel types.

Example

Tool	Machining
 UM-4E-D20.0-W KMG405	 HSC strategy

Workpiece material
16MnCr5 (1.7131) ca. 700 N/mm³

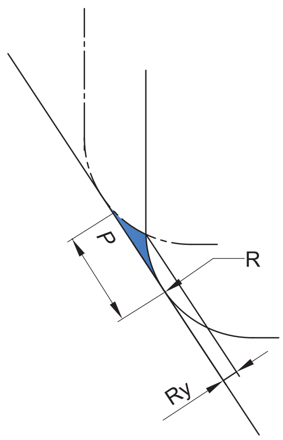
Cutting data	
V_c	550 m/min
n	8750 1/min
f_z	0,3 mm (**h_{max} = 0,19 mm**)
V_f	10500 mm/min
a_p	22 mm
a_e	2,3 mm

Result
Chip removal rate **530 cm³/min**! Machining time 58 seconds! The maximum chip thickness is 0.19 mm.

A Turning B Milling C Drilling D Technical Information E Index

Solid carbide mills

Feed rate selecting table for profile machining with ball nose cutters and torus mills



$$Ry = R \times \{1\text{-}\cos[\arcsin(fr/2R)]\}$$

Ry: Theoretical values of surface quality
P: Feed rate
R: Radius of the ball nose cutter or torus mill

R \ Ry	Feed rate									
	0,1	0,2	0,3	0,4	0,5	0,6	0,7	0,8	0,9	1,0
0,5	0,003	0,010	0,023	0,042	0,067	0,100				
1,0	0,001	0,005	0,011	0,020	0,032	0,046	0,063	0,083	0,107	
1,5	0,001	0,003	0,008	0,013	0,021	0,030	0,041	0,054	0,069	0,086
2,0	0,001	0,003	0,006	0,010	0,015	0,023	0,031	0,040	0,051	0,064
2,5	0,001	0,002	0,005	0,008	0,013	0,018	0,025	0,032	0,041	0,051
3,0		0,001	0,004	0,007	0,010	0,015	0,020	0,027	0,034	0,042
4,0		0,001	0,003	0,005	0,008	0,011	0,015	0,020	0,025	0,031
5,0		0,001	0,002	0,004	0,006	0,009	0,012	0,016	0,020	0,025
6,0			0,002	0,003	0,005	0,008	0,010	0,013	0,017	0,021
8,0			0,001	0,003	0,004	0,006	0,008	0,010	0,013	0,016
10,0			0,001	0,002	0,003	0,005	0,006	0,008	0,010	0,013
12,5			0,001	0,002	0,003	0,004	0,005	0,006	0,008	0,010

R \ Ry	Feed rate									
	1,1	1,2	1,3	1,4	1,5	1,6	1,7	1,8	1,9	2,0
0,5										
1,0										
1,5	0,104									
2,0	0,077	0,092	0,109							
2,5	0,061	0,073	0,086	0,100						
3,0	0,051	0,061	0,071	0,083	0,095	0,109				
4,0	0,038	0,045	0,053	0,062	0,071	0,081	0,091	0,103		
5,0	0,030	0,036	0,042	0,049	0,057	0,064	0,073	0,082	0,091	0,101
6,0	0,025	0,030	0,035	0,041	0,047	0,054	0,061	0,068	0,076	0,084
8,0	0,019	0,023	0,026	0,031	0,035	0,040	0,045	0,051	0,057	0,063
10,0	0,015	0,018	0,021	0,025	0,028	0,032	0,036	0,041	0,045	0,050
12,5	0,012	0,014	0,017	0,020	0,023	0,026	0,029	0,032	0,036	0,040

Nonstandard – solid carbide end mills

Name/Company:	ZCC·CT
Address:	
Tel.:	**Wanheimer Str. 57** **40472 Düsseldorf, Germany**
Fax:	**Fax:** **+49-(0)211-989240-111**
E-mail:	**E-mail:** **technik@zccct-europe.com**

Material	
Material	
Tensile strength (N/mm^2)	
Hardness	

Coating			
Yes	☐	No	☐

Centre cutting			
Yes	☐	No	☐

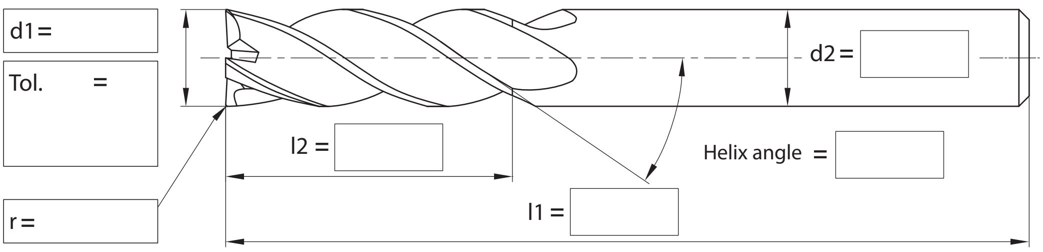
Series			
GM		NM	
PM		AL	
UM		VSM	
HM		HPC	

Machining operations		
☐ Slot milling	☐ Square shoulder milling	☐ Profile milling

Tool holder type			
DIN6535			
Form HA	Form HB	Normal straight shaft	Special type
☐	☐	☐	☐

Type		
☐ Square shoulder mill	☐ Ball nose cutter	☐ Torus mill

Number of teeth



Remarks:	
Order quantity:	Desired delivery date:
Date:	Signature:

A Turning
B Milling
C Drilling
D Technical Information
E Index

A Turning
B Milling
C Drilling
D Technical Information
E Index

Notes