Cutting data recommendations

Uddeholm Dievar®

44-46 HRC

UDDEHOLM
a voestalpine company

Cutting data formulae

Turning

$$Cutting\ speed,\ v_c = \frac{\pi \cdot D \cdot n}{1000} \quad (m / min)$$

$$Spindle\ speed,\ n = \frac{1000 \cdot v_c}{\pi \cdot D} \quad (rev / min)$$

$$Material\ removal\ rate,\ Q = v_c \cdot a_p \cdot f \quad (cm^3 / min)$$

$$Surface\ roughness,\ R_a \approx \frac{f^2 \cdot 50}{r_\varepsilon} \quad (\mu m)$$

Legend

v_c = Cutting speed (m/min)
n = Spindle speed (rev/min)
f = Feed per rev (mm/rev)
a_p = Axial depth of cut (mm)
D = Workpiece diameter (mm)
Q = Material removal rate (cm^3/min)
R_a = Surface roughness (µm)
r_e = Nose radius (mm)

Milling

$$v_c = \frac{\pi \cdot D \cdot n}{1000} (m/\mathrm{min})$$

$$n = \frac{1000 \cdot vc}{\pi \cdot D} (\mathrm{rev/min})$$

$$vf = fz \cdot z \cdot n = f \cdot n (\mathrm{mm/min})$$

$$D_{eff} = 2 \cdot \sqrt{ap\,(D - ap)}\,(\mathrm{mm})$$

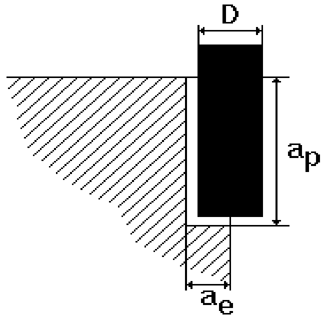
$$D_{eff} = 2 \cdot \sqrt{ap\,(D_i - ap)} + D - D_i\,(\mathrm{mm})$$

$$h_m = f_z \cdot \sqrt{\frac{a_e}{D}}\,(\mathrm{mm}) \quad \frac{a_e}{D} < 0{,}3$$

$$Q = \frac{a_p \cdot a_e \cdot v_f}{1000} (\mathrm{cm^3/min})$$

Legend

v_c = Cutting speed (m/min)
n = Spindle speed (rev/min)
v_f = Feed speed (mm/min)
a_p = Axial depth of cut (mm)
a_e = Radial depth of cut (mm)
f = Feed per rev (mm/rev)
z = Number of teeth
f_z = Feed per tooth (mm/tooth)
D = Cutter diameter (mm)
D_{eff} = Effective cutter diameter (mm)
D_i = Diameter of insert (mm)
h_m = Average chip thickness (mm)
Q = Material removal rate (cm^3/min)



Drilling

$$Cutting\ speed,\ v_c = \frac{\pi \cdot D \cdot n}{1000} \quad (m / min)$$

$$Spindle\ speed,\ n = \frac{1000 \cdot v_c}{\pi \cdot D} \quad (rev / min)$$

$$Feed\ speed,\ v_f = f \cdot n \quad (mm / min)$$

$$Feed\ per\ rev,\ f = \frac{v_f}{n} \quad (mm / rev)$$

Legend

v_c = Cutting speed (m/min)
n = Spindle speed (rev/min)
v_f = Feed speed (mm/min)
D = Drill diameter (mm)
f = Feed per rev (mm/rev)

Turning

Turning	Cemented carbide	
	Roughing	Finishing
Cutting speed, v_c (m/min)	40-60	70-90
Feed, f (mm/rev)	0,2-0,4	0,05-0,2
Depth of cut, a_p (mm)	2-4	0,5-2
Suitable grades	P20-P30 coated carbide	P10 coated carbide or mixed ceramic

Remarks:

1. Cutting fluid is recommended.
2. For turning with interrupted cut or face turning of large workpieces use a thougher cemented carbide grade.

Face milling

Face milling	Cemented carbide	
	Roughing	
Cutting speed, v_c (m/min)	40-60	60-90
Feed, f_z (mm/tooth)	0,2-0,3	0,1-0,2
Depth of cut, a_p (mm)	1-2	0,5-1
Suitable grades	P20-P40 coated carbide	P10-P20 coated carbide or cermet

Remarks:

1. Use a milling cutter with a positive-negative or positive-positive geometry.
2. Climb milling should generally be used.
3. Milling should generally be done without coolant.
   If a high surface finish is required coolant may be used.
4. Cermets can be of use when finishing under stable conditions.

Square shoulder milling

Square shoulder milling with cemented carbide	a_e = 0.1 x D	a_e = 0.5 x D	a_e = 1 x D
Cutting speed, v_c (m/min)	80-100	60-80	40-60
Feed, f_z (mm/tooth)	0,2-0,3	0,1-0,2	0,05-0,1
Suitable grades	P15-P40 coated carbide		

Remarks:

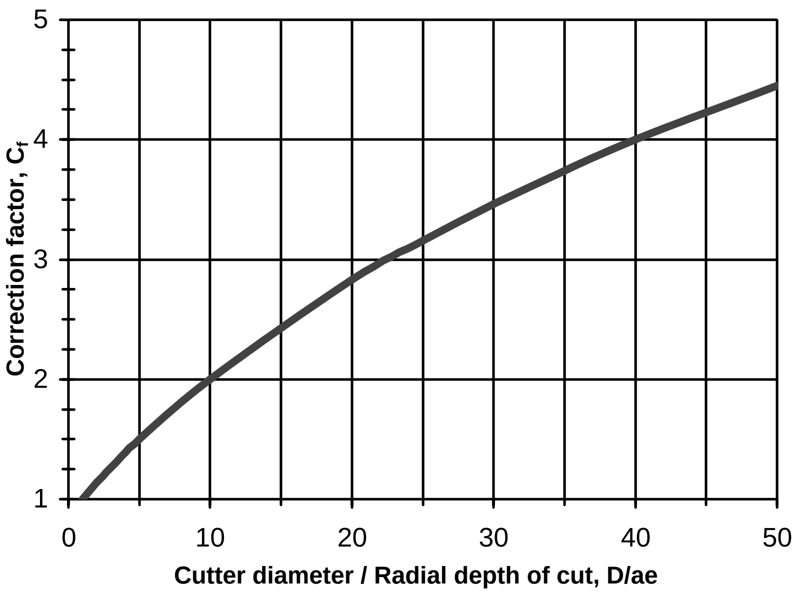
1. Climb milling should generally be used.
2. Choose the cutter diameter (D) and the radial depth of cut (a_e) so that at least two cutting edges are engaged simultaneously.
3. If the machine tool power is inadequate for the data given reduce the depth of cut, but do not reduce the feed.

End milling

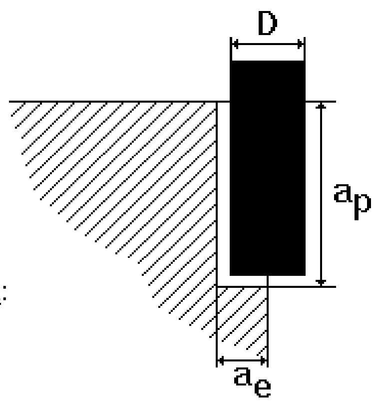
Slot milling **Axial depth of cut, a_p = 1 x D**		**Cutter diameter (mm)** 3 - 5	5 - 10	10 - 20	20 - 30	30 - 40
Coated HSS [1-4]	Cutting speed, v_c (m/min)	5-10				
	Feed, f_z (mm/tooth)	0,01-0,03	0,03-0,04	0,04-0,05	0,05-0,06	0,06-0,09
Solid cemented carbide [5-8]	Cutting speed, v_c (m/min)	50-70				
	Feed, f_z (mm/tooth)	0,006-0,01	0,01-0,02	0,02-0,04		
Indexable insert [6-8] (cemented carbide inserts)	Cutting speed, v_c (m/min)			50-70		
	Feed, f_z (mm/tooth)			0,06-0,08	0,08-0,10	0,10-0,12
	Suitable grades			P15-P40 coated carbide		
Side milling **Axial depth of cut, a_p = 1.5 x D**		For side milling the same cutting speed as for slot milling can be used, but the feeds must be adjusted in order to obtain a suitable average chip thickness.				

Correction factor for side milling

Divide the cutter diameter with the radial depth of cut. See in the chart below which correction factor, C_f, this corresponds to, and multiply the chosen feed in the table for slot milling with this factor.



Example:



<u>Tool</u>:
CC insert

<u>Cutter diameter</u>:
D = 40 mm

<u>Radial depth of cut</u>:
a_e = 2 mm

D/a_e = 40/2 = 20
Feed acc. to table slot milling = 0.11 mm/tooth
Correction factor acc. to chart: C_f = 2.8

Feed for side milling:
f_z = 2.8 x 0.11 = 0.31 mm/tooth

Remarks: (slot and side milling)

1. Climb milling is generally recommended.
2. Use a cutter with chipbreaker when side milling with radial depths of cut, a_e > 0.3 xD.
3. When side milling with small radial depths of cut (a_e) the cutting speed can be increased by up to 15%.
4. Use liberal amounts of cutting fluid.
5. It is recommended to use a TiCN coated cutter when milling with solid cemented carbide tools. The axial depth of cut should not exceed the cutter diameter when slot milling.
6. Climb milling is generally recommended.
7. When side milling with small radial depths of cut (a_e) the cutting speed can be increased by up to 50%.
8. The radial run-out, at the cutting edges, must be small and not exceed 0.03 mm.

Uddeholm Dievar

44-46 HRC

Cavity milling

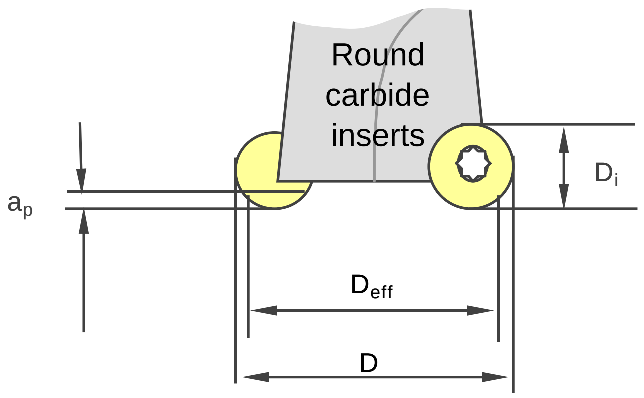
Rough milling with round carbide inserts		<20	21-30	31-40	41-50	>50
		Diameter of cutter, D (mm)				
Axial depth of cut, $ap = 0{,}2 \times D_i$	Cutting speed v_c (m/min)	80-100		60-80		
	Feed f_z (mm/tooth)	-0,15	0,15-0,18	0,18-0,22	0,22-0,25	0,25-0,28
Axial depth of cut, $ap = 0{,}15 \times D_i$	Cutting speed vc (m/min)	80-100		60-80		
	Feed f_z (mm/tooth)	-0,17	0,18-0,21	0,21-0,25	0,25-0,27	0,27-0,30
Axial depth of cut, $ap = 0{,}1 \times D_i$	Cutting speed vc (m/min)	100-120		80-100		
	Feed f_z (mm/tooth)	-0,2	0,20-0,25	0,25-0,28	0,28-0,32	0,32-0,35
Axial depth of cut, $ap = 0{,}05 \times D_i$	Cutting speed vc (m/min)	120-140		100-120		
	Feed f_z (mm/tooth)	-0,25	0,25-0,3	0,3-0,35	0,35-0,38	0,38-0,42

D_i = Diameter of the insert

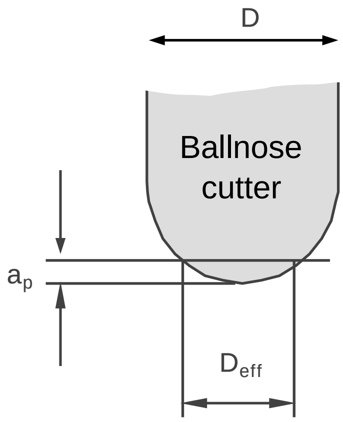
Rough milling with high feed cutters		<20	21-30	31-40	41-50	>50
		Diameter of cutter, D (mm)				
Axial depth of cut, ap = 70% of max[1]	Cutting speed vc (m/min)	70-100		50-70		
	Feed f_z (mm/tooth)	-0,3	0,3-0,5	0,5-0,7	0,7-1,0	0,7-1,0
Axial depth of cut, ap = 50% of max[1]	Cutting speed vc (m/min)	80-110		70-100		
	Feed f_z (mm/tooth)	-0,5	0,5-0,7	0,7-0,9	0,9-1,1	1,0-1,2

[1] Per centage of maximum depth of cut allowed (according to milling tool supplier)

Semi finishing and finishing milling with ballnose cutters		<6	6-8	8-10	10-12	>12
		Diameter of cutter, D (mm)				
Semi finishing Axial depth of cut, ap = 5% of D (Ø cutter)	Cutting speed vc (m/min)	120-140				
	Feed f_z (mm/tooth)	-0,08	0,08-0,1	0,1-0,12	0,12-0,14	0,14-
Finishing Axial depth of cut, ap = 2% of D (Ø cutter)	Cutting speed vc (m/min)	140-160				
	Feed f_z (mm/tooth)	-0,1	0,1-0,12	0,12-0,14	0,14-0,16	0,16-



$$D_{eff} = 2 \cdot \sqrt{ap(D_i - ap)} + D - D_i \text{ (mm)}$$



$$D_{eff} = 2 \cdot \sqrt{ap\,(D - ap)} \text{ (mm)}$$

Remarks cavity milling:

1. Down milling strategy is recommended
2. Recommended cutting speeds are at the effective cutter diameter (D_{eff})
3. Reduce the cutting speed and feed rate by 20% when using tool overhang >5xD
4. The radial depht of cut (ae) should be maximum 70% of the effective cutter diameter (D_{eff})
5. A tough PVD coated carbide grade with sharp edge geometry is recommended

Uddeholm Dievar

44-46 HRC

Drilling

Drilling 		Drill diameter (mm)				
		1 - 5	5 - 10	10 - 20	20 - 30	30 - 40
Coated HSS [1-2)]	Cutting speed, v_c (m/min)	6-8				
	Feed, f (mm/rev)	0,05-0,1	0,1-0,15	0,15-0,2	0,2-0,25	0,25-0,3
Indexable insert [3-4)]	Cutting speed, v_c (m/min)				70-90	
(cem. carbide inserts)	Feed, f (mm/rev)				0,05-0,10	0.10-0,15
Solid cemented	Cutting speed, v_c (m/min)		60-80			
carbide [5-7)]	Feed, f (mm/rev)		0,08-0,10	0,10-0,18	0,18-0,26	0,26-0,3
Carbide tipped [5-7)]	Cutting speed, v_c (m/min)			60-80		
	Feed, f (mm/rev)			0,12-0,2	0,20,28	0,28-0,35

Remarks:

1. The cutting fluid should be ample and directed at the tool.
2. When drilling with short "NC drills" the feed may be increased by up to 20%.
   For extra long drills the feed must be decreased.
3. Use insert grades in the range of ISO P20-P30.
   Under unstable conditions a tougher carbide grade should be used for the centre position.
4. Use a high cutting fluid pressure and flow rate for a good chip removal.
5. If machining with solid carbide or carbide tipped drills, a rigid set-up and stable working conditions are required.
6. The use of drills with internal cooling channels is recommended.
7. Use a cutting fluid concentration of 15-20 %.

Tapping with HSS

Cutting speed, v_c =	2-3 m/min

Remarks:

1. Threading compound or cutting oil gives a longer tool life than emulsion.
2. TiCN coated taps are recommended.
3. Straight fluted taps are recommended for both through holes and blind holes.