



Calculations

n = Revolutions per minute (rpm)
 vc = Cutting speed (m/min)
 d = Tool diameter in millimeters (mm)

z = Number of teeth
 fz = Tooth feed (mm/tooth)
 vf = Feed rate (mm/min)

The end mill speed is calculated with the following formula:

$$n \text{ [rpm]} = (vc \text{ [m/min]} * 1000) / 3.14 * \phi d1 \text{ [mm]}$$

Example calculation:

vc = 500 m/min (selected from chart)
 d = ϕ 8 mm

$$19904 \text{ rpm} = (500 * 1000) / (3.14 * 8)$$

If the maximum speed of the milling motor is lower than the calculated value, the maximum speed of the milling motor needs to be inserted into the formula for the feed calculation.

The end mill feed rate is calculated with the following formula:

$$vf = n * z * fz$$

Example calculation for aluminum (wrought alloy) with 8 mm end mill 2-flute:

n = 15923 rpm from upper formula
 fz = 0,064 from chart
 z = 2

$$2547,77 \text{ mm/min} = 19904 * 2 * 0,064$$

Benchmark for Speed and Feed

	Ø Cutting Speed m/min.*	End Mill Diameter								
		Ø 1 mm	Ø 2 mm	Ø 3 mm	Ø 4 mm	Ø 5 mm	Ø 6 mm	Ø 8 mm	Ø 10 mm	Ø 12 mm
		Feed Rate in mm / Tooth / Revolution								
Cast Aluminum > 6 % Si	200	0,010	0,010	0,010	0,015	0,015	0,025	0,030	0,038	0,050
Wrought Aluminum Alloy	500	0,010	0,020	0,025	0,050	0,050	0,050	0,064	0,080	0,100
Soft Plastic	600	0,025	0,030	0,035	0,045	0,065	0,090	0,100	0,200	0,300
Hard Plastic	550	0,015	0,020	0,025	0,050	0,060	0,080	0,089	0,100	0,150
Hard Wood	450	0,020	0,025	0,030	0,055	0,065	0,085	0,095	0,095	0,155
Soft Wood	500	0,025	0,030	0,035	0,060	0,070	0,090	0,100	0,110	0,160
MDF	450	0,050	0,070	0,100	0,150	0,200	0,300	0,400	0,500	0,600
Brass, Copper, Bronze	365	0,015	0,020	0,025	0,025	0,030	0,050	0,056	0,065	0,080
Steel	75	0,010	0,010	0,012	0,025	0,030	0,038	0,045	0,050	0,080

*The stated cutting speeds are average values. In result of the milling process and the type of end mill adjustments might be necessary.

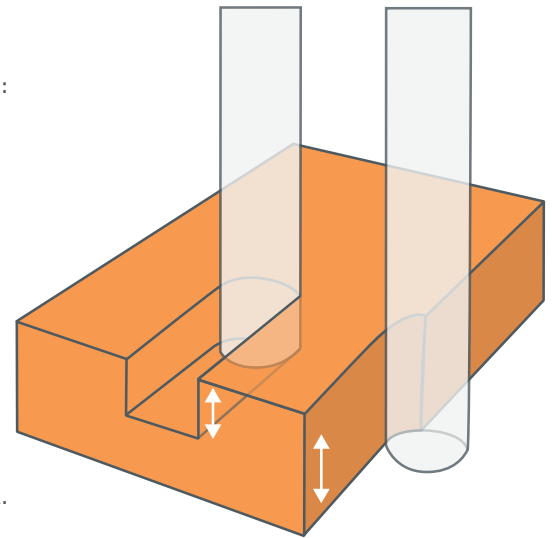
Roughing: Reduction of up to 25 % – **Finishing:** Increase of up to 25 % – **HSS end mill:** Reduction of up to 50 % (hard materials) – **VHM end mill:** Increase of up to 25 %

Practical Tip

Immersion Depth:

While milling a groove we recommend the following immersion depth:

Material	D-Series	M-Series	Q-Series
Non-ferrous Metals	up to a diameter 0,05 times higher	up to a diameter 0,2 times higher	up to a diameter 0,3 times higher
Wood, Plastics	up to a diameter 0,5 times higher	up to a diameter 0,75 times higher	up to a diameter 1,0- times higher
Rigid Foam	up to a diameter 3,5 times higher	up to a diameter 3,5 times higher	up to a diameter 5 times higher

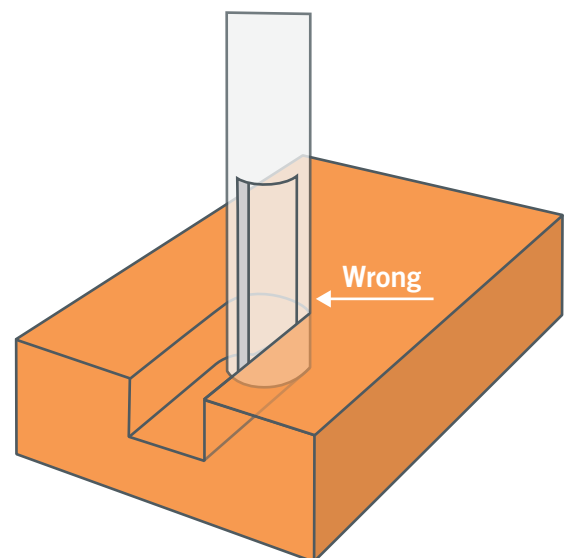
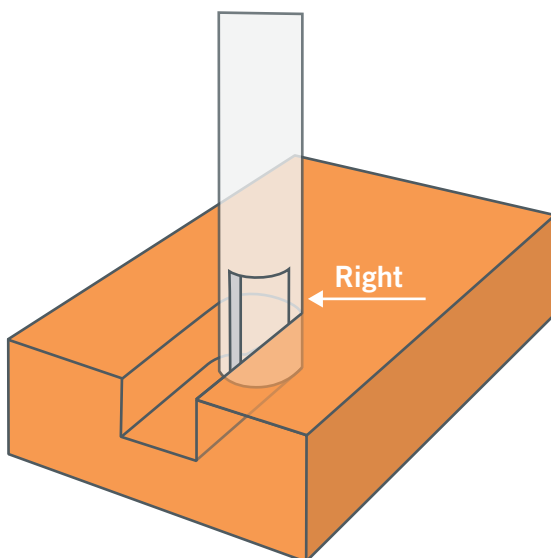


During contour milling we recommend a lateral infeed of approx. 25 % of the end mill diameter while applying 100 % immersion depth.

The immersion depth is conditional to the milling motor, the structure and the stability of the machine. This means a reduction of the values is required, when using a less powerful and / or lighter milling motor.

End Mill Length Selection:

To avoid increasing vibration of the end mill we recommend clamping the bit as short as possible or as long as necessary.



Cooling / Lubricating:

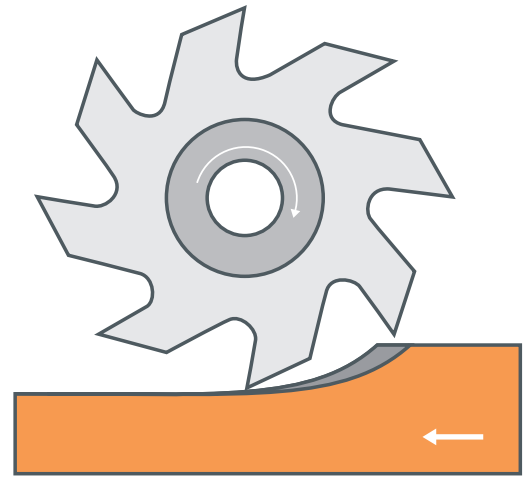
The cooling of **non-ferrous metals** occurs in best case with a **lubrication system** in combination with lubricant. Furthermore the lubricating improves the surface quality and the service life of the tool.

Lubricating with a **soap solution** is suitable on **acrylic glass**. This improves an excellent surface.

Downcut Milling:

In downcut milling, the end mill pulls itself into the workpiece, which can lead to the gantry or the Z-axis being pulled uncontrolled (backlash of lead screw) in the direction of the workpiece during the removal of larger chips. This causes a less precise milling pattern and can even result in the breakage of the end mill, if the chip building is too large.

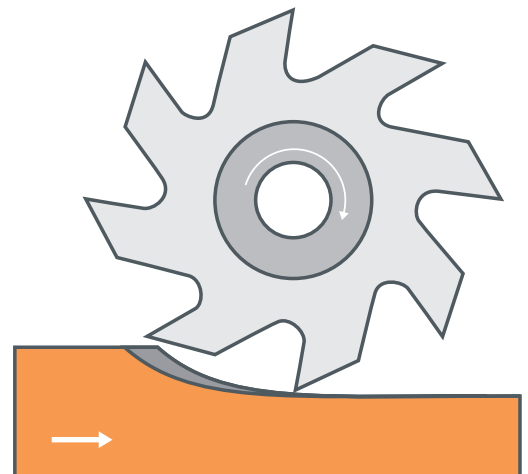
Assuming backlash-free ball screws are used, the downcut is preferred over the upcut milling.



Upcut Milling:

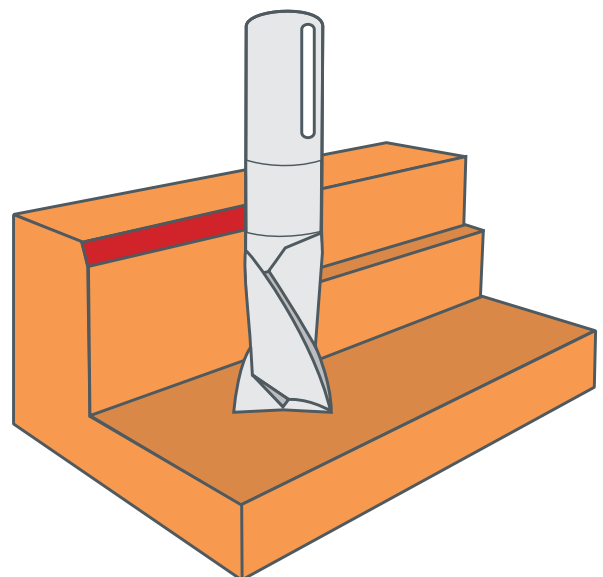
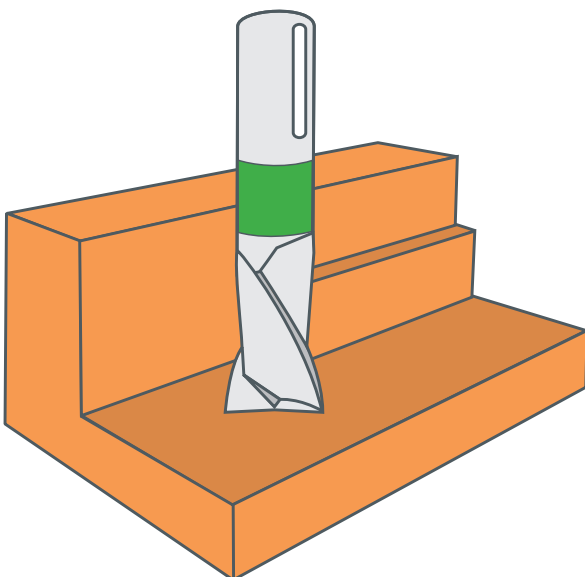
During upcut milling, the end mill pushes away from the workpiece, which, when only removing little chips, quickly causes the cutting edge to push out of the workpiece. This leads to chatter marks which create an unclean surface and reduce the endurance of the end mill.

The upcut milling is favored on machines with threaded lead screws which contain backlash.



Relief-grinded End Mills:

The maximum possible infeed is usually reduced to the spiral length of the end mill, otherwise the shank will rub on the workpiece. Due to the relief-grinded shank, depths over several infeeds up to the maximum effective lengths are possible, thus, exceeding the spiral length.



Lubrication System 20-50 ml/h:

Dependence of lubrication system on production process

Milling
Drilling
Grinding
Lapping
Turning
Grating
Honing



increasing lube

Suitability of lubrication system for cutting materials

Cu Alloy
Al Casting Alloy
Steel ferritic
Mg Alloy
Wrought Aluminum Alloy
Pearlitic
Cast Iron
Stainless Steels



increasing material qualification