Troubleshooting

General notes

All the cutting rate recommendations specified in this catalogue are standard values valid exclusively for new tools or tools re-ground to Guhring specifications. Pre-requisites are stable machines, optimal cooling, optimal tool clamping and maximum concentricity of the tool and the machine spindle. Our recommended cutting rates must be reduced if the conditions deviate. The values may also be adjusted to influence Surface finish quality, machining rate or tool life.
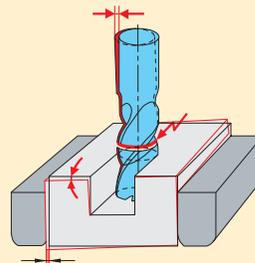
1. Workpiece clamping
Loss of tool life or tool breakage through unstable clamping

➤ improve workpiece clamping

Alternative:
➤ reduce feed
➤ reduce cutting width or depth

2. Tool clamping
Loss of tool life or tool breakage through unstable, worn or too small/long/thin tool holder

➤ apply new or larger tool holder or holder with increased clamping force and increased concentricity

Alternative:
➤ reduce cutting rates
➤ reduce clamping length
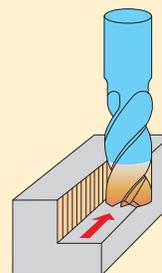➤ apply tool with smaller diameter
➤ check clamping screws for wear
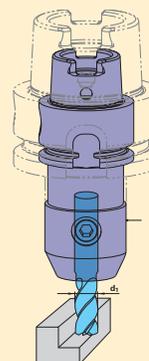
3. Surface finish quality
Excessive peak-to-valley height Ra/Rz at the tool Surface finish through excessive feed and feed rates or vibrations

➤ improve workpiece clamping and tool clamping (see points 1 and 2)

Alternative:
➤ reduce feed and feed rate
➤ increase cutting speed

4. Vibrations
High tool wear, insufficient workpiece Surface finish quality and insufficient dimensional accuracy through vibration

➤ improve workpiece and tool clamping (see points 1 and 2)
➤ increase tooth feed, because the chip centre thickness is too small
➤ modify speed
➤ modify milling strategy, i.e. select alternative cutting distribution
➤ change tool selection, i.e. reduce no. of teeth or spiral
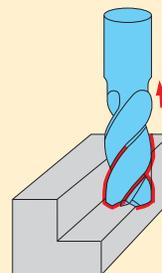
5. Chip congestion/cooling

Significant reduction in tool life, crumbling on cutting lips, edge build-up or conglutination of flutes through insufficient chip evacuation

➤ select milling cutters with internal cooling

Alternative:
➤ peripheral cooling via GM 300 chuck
➤ increase volume flow
➤ adjust coolant flow
➤ apply compressed air cooling (according to tool and material)
➤ reduce feed rate
➤ modify cutting distribution
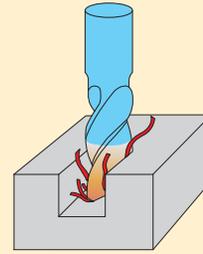
6. Pecking when drilling

Significant reduction in tool life as well as crumbling of cutting lips through insufficient chip evacuation and thermal stresses

➤ select milling cutter with internal cooling
 with drilling depths > 0.5 x D pecking in stages
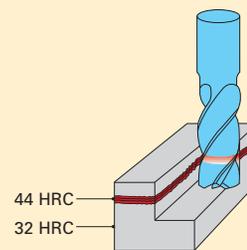
Alternative:
➤ peripheral cooling via GM 300 chuck
➤ increase volume flow
➤ adjust coolant flow
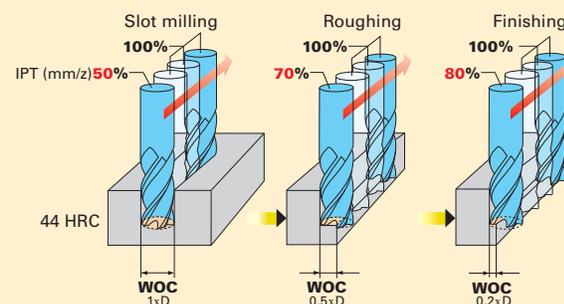➤ reduce feed rate
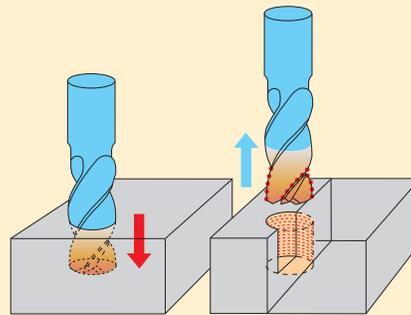
7. Thermal influence on materials

Through welding or torch cutting, the material characteristics at the parting line do not correspond with the specified material class

➤ reduce cutting rates
➤ select tool for materials with a higher tensile strength

44 HRC
32 HRC

8. Entry in hardened materials

For entering materials over 44 HRC, reduce the feed rate vf (mm/min) in accordance with the illustration on the right

Slot milling   Roughing   Finishing
100%   100%   100%
IPT (mm/z) 50%   70%   80%

44 HRC

WOC   WOC   WOC
1xD   0,5xD   0,2xD

FEEDS & SPEEDS

Troubleshooting

9. Loss in tool life with interrupted cutting
Significant loss in tool life through interrupted cutting
(especially with milling angles of 90º)

➤ modify cutting distribution
➤ reduce feed rate for entry and exit
➤ reduce approach angle
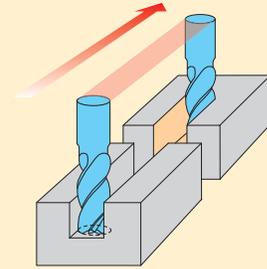
10. Feed rate adjustment: Modifying the cutting width
➤ when modifying the cutting width WOC, the feed rate
   must be reduced in accordance with the illustration on
   the right
➤ cutting speed or revolutions remain unchanged
➤ double reduction applies when also modifying the cut-
   ting depth DOC!

| WOC = 1 x D | WOC = 0.5 x D | WOC = 0.25 x D |
| IPT = 25 % | IPT = 50 % | IPT = 100 % |

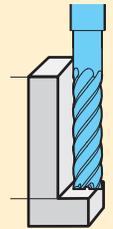**11. Feed rate adjustment: Modifying the cutting depth**
➤ when modifying the cutting depth DOC, the feed rate
   must be reduced in accordance with the illustration
   on the right
➤ cutting speed or revolutions remain unchanged up to
   cutting depths of 3 x D, must only be adapted over 3 x D
➤ double reduction applies when also modifying the cut-
   ting width ae!
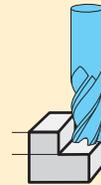
| DOC = 1 x D | DOC = 2 x D | DOC = 3 x D |
| IPT = 100 % | IPT = 50 % | IPT = 25 % |

12. Plunging strategies
for drilling:
➤ reduce feed rate v_f (mm/min.)
➤ additional pecking for drilling depths > 0.5 x D or tran-
   sition to radial machining
Attention: Danger of breakage through abrupt load
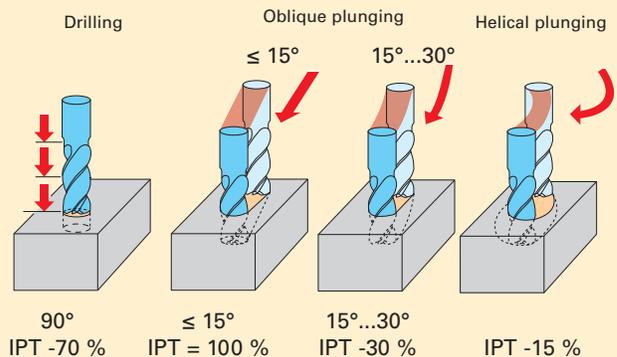increase!

Oblique plunging up to 15º (preferred):
➤ reduction in feed rate v_f (mm/min.) not required

Oblique plunging between 15º and 30º:
➤ reduce feed rate IPT in accordance with the illustration
   on the right

Helical plunging:
➤ for helical plunging on a milling cycle, we recommend
   a feed of 0.1 to 0.2 per cycle
➤ reduce feed rate v_f (mm/min.) in accordance with the
   illustration on the right
➤ select preferred hole diameter 1.8 x D

Drilling   Oblique plunging   Helical plunging
                  ≤ 15°   15°...30°

| 90° | ≤ 15° | 15°...30° | |
| IPT -70 % | IPT = 100 % | IPT -30 % | IPT -15 % |

Troubleshooting

13. Copy milling

For cutting depths DOC<0.5xD, the engaged effective diameter D_{eff} must be applied to calculate the speed. With the spindle not engaged, the effective diameter is calculated according to the illustration below. To increase tool life, we recommend machining with tilted spindle. The tilt angle must be taken into account when calculating the effective diameter D_{eff}.
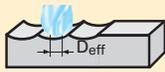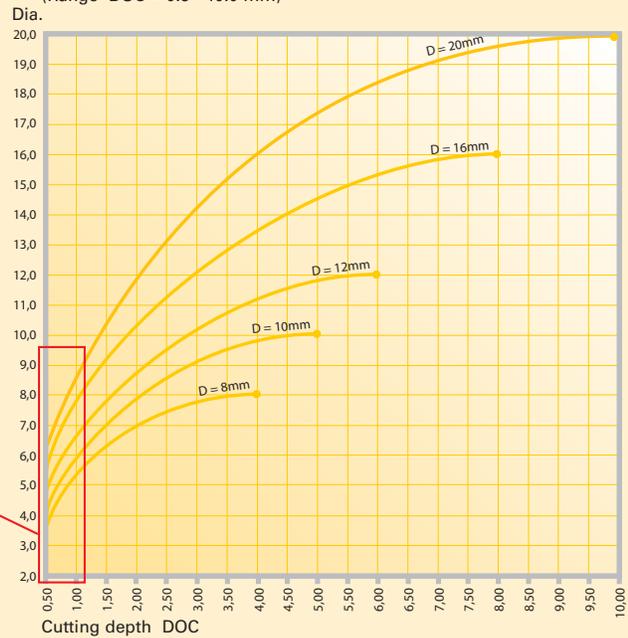
$$D_{(eff)} = 2 \cdot \sqrt{D \cdot DOC - DOC^2}$$
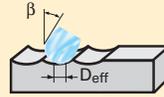
$$D_{(eff)} = D \cdot \sin\left[\beta + \arccos\left(\frac{D - 2DOC}{D}\right)\right]$$

(Range DOC = 0.5 - 10.0 mm)

(Range DOC = 0.1 - 0.5 mm)

Modifying the cutting width WOC results in improved Surface finish quality of the workpiece (reduced peak-to-valley height)

$$R_{th} = \frac{D}{2} - \sqrt{\frac{D^2 - WOC^2}{4}}$$

Overmeasure

Nom. size

WOC

Act. contour

WOC