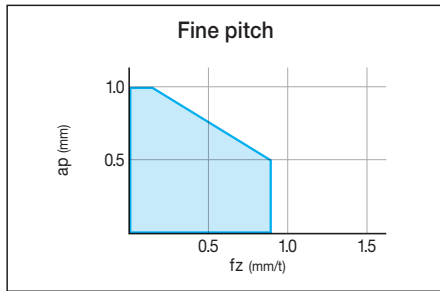
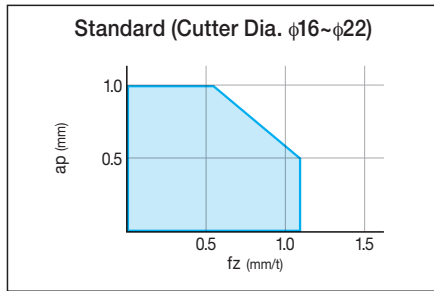


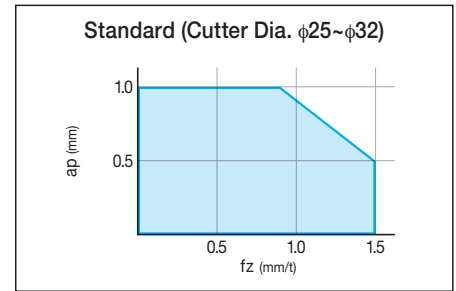
Cutting Performance (GM)



MFH20----4T, MFH22----4T, MFH25----5T
MFH28----5T, MFH32----6T



MFH16----2T, MFH17----2T, MFH18----2T
MFH20----3T, MFH22----3T



MFH25----4T, MFH28----4T, MFH32----5T

When using fine pitch type, reduce the cutting conditions compared with standard type.

Recommended Cutting Conditions

Insert Type	Workpiece Material	Toolholder Description and Feed Rate (fz [mm/t]) * Recommended feed rate (Reference value) : ap=0.5mm							Recommended Insert Grades (Cutting Speed Vc [m/min])			
		MFH16 ----2T	MFH20 ----3T	MFH20 ----4T	MFH25 ----4T	MFH25 ----5T	MFH32 ----5T	MFH32 ----6T	MEGACOAT NANO			CVD Coated Carbide
									PR1535	PR1525	PR1510	CA6535
GM	Carbon Steel	0.2-0.7-1.2	0.2-0.5-0.8	0.2-0.8-1.5	0.2-0.5-0.8	0.2-0.8-1.5	0.2-0.5-0.8	0.2-0.5-0.8	☆ 120-180-250	★ 120-180-250	—	—
	Alloy Steel								☆ 100-160-220	★ 100-160-220	—	—
	Mold Steel (~40HRC)	0.2-0.5-0.9	0.2-0.4-0.6	0.2-0.6-1.2	0.2-0.4-0.6	0.2-0.6-1.2	0.2-0.4-0.6	0.2-0.4-0.6	☆ 80-140-180	★ 80-140-180	—	—
	Mold Steel (40~50HRC)	0.2-0.3-0.5	0.2-0.25-0.3	0.2-0.3-0.6	0.2-0.25-0.3	0.2-0.3-0.6	0.2-0.25-0.3	0.2-0.25-0.3	☆ 60-100-130	★ 60-100-130	—	—
	Stainless Steel (Austenitic related)	0.2-0.5-0.9	0.2-0.4-0.6	0.2-0.6-1.2	0.2-0.4-0.6	0.2-0.6-1.2	0.2-0.4-0.6	0.2-0.4-0.6	★ 100-160-200	☆ 100-160-200	—	—
	Stainless Steel (Martensitic related)								☆ 150-200-250	—	—	★ 180-240-300
	Stainless Steel (Precipitation Hardening)								★ 90-120-150	—	—	—
	Gray Cast Iron	0.2-0.7-1.2	0.2-0.5-0.8	0.2-0.8-1.5	0.2-0.5-0.8	0.2-0.8-1.5	0.2-0.5-0.8	0.2-0.5-0.8	—	—	★ 120-180-250	—
	Nodular Cast Iron	0.2-0.5-0.9	0.2-0.4-0.6	0.2-0.6-1.2	0.2-0.4-0.6	0.2-0.6-1.2	0.2-0.4-0.6	0.2-0.4-0.6	—	—	★ 100-150-200	—
	Ni-base heat-resistant alloys	0.2-0.3-0.6	0.2-0.25-0.4	0.2-0.4-0.8	0.2-0.25-0.4	0.2-0.4-0.8	0.2-0.25-0.4	0.2-0.25-0.4	☆ 20-30-50	—	—	★ 20-30-50
	Titanium Alloys								★ 40-60-80	—	☆ 30-50-70	—

Standard Fine pitch ★:1st Recommendation ☆:2nd Recommendation

* Machining with coolant is recommended for Ni-base heat-resistant alloys and titanium alloys.

The bold-faced number indicates a center value of recommended cutting condition. Adjust the cutting speed and the feed rate within the above conditions according to the actual machining situation. For machining center equivalent to BT30, reduce feed rate to 25% or less of the recommended condition. For slotting, internal coolant or center through coolant is recommended.

Note for Machining Program (Approx. R)

Shape	Description	Insert Type	Cutting angle γ	Approx. R (mm)	Unmachined part K (mm)	Max. inclination angle of workpiece at contouring
	MFH...-03-...	GM	12°	1.6	0.39	90°

Reference data for Ramping

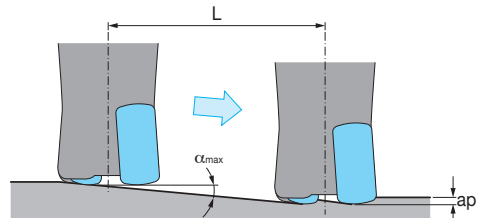
Description	Cutter Dia. ϕD (mm)	16	17	18	20	22	25	28	32
MFH...-03-...	Max. ramping angle α_{\max}	2.8°	2.5°	2.1°	1.7°	1.4°	1.2°	1°	0.8°
	$\tan \alpha_{\max}$	0.049	0.042	0.037	0.03	0.024	0.021	0.017	0.014

Guide for Ramping (Slant Milling)

- Ramping angle should be under α_{\max} (Maximum ramping angle) in the above cutting conditions.
- Feed rate should be under 70% of the above cutting conditions.

Formula of the cutting length "L" at Max. ramping angle

$$L = \frac{ap}{\tan \alpha_{\max}}$$

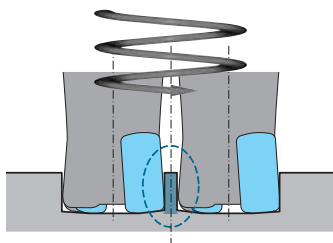


Guide for Helical milling

- For helical milling, use between Min. cutting dia. and Max. cutting dia.

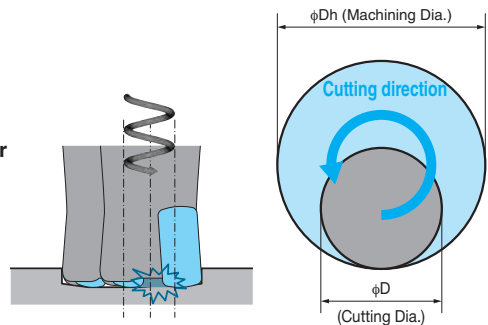
Over Max. Cutting Dia.

Center core part remains after machining



Under Min. Cutting Dia.

Center core part interferes with toolholder

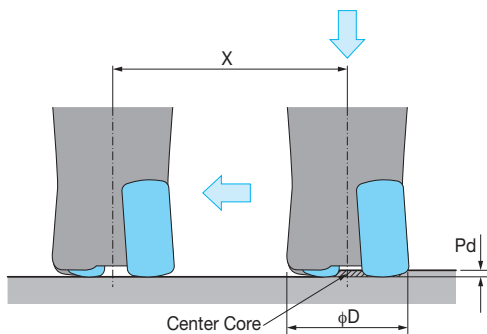


Description	Min. Cutting Dia. $\phi Dh1$	Max. Cutting Dia. $\phi Dh2$
MFH...-03-...	$2 \times D - 8$	$2 \times D - 2$

Unit : mm

- Sinking depth at helical milling should be under Max. ap (1mm).
- Down-cut milling is recommended (refer to the figure above)
- Feed rate should be under 50% of the recommended cutting conditions.
- Be careful to machine in a safe environment to avoid accident caused by long chips.

Guide for Drilling



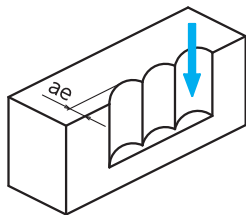
Description	GM	
	Max. cutting depth Pd	Min. cutting length X for flat bottom surface
MFH...-03-...	1.0	$\phi D - 9$

Unit : mm

- * When traversing directly after drilling, set the table feed at up to 25% of the recommended cutting conditions.
- * When drilling, reduce feed rate per revolution to under $f=0.2\text{mm/rev}$.

Vertical milling (Plunging)

Vertical milling (Plunging)



- Available for vertical milling (plunging).

Insert Description	Max. Width of Cut (ae)
LOGU03	3.5mm

- For vertical milling (plunging), reduce feed rate to $fz=0.2\text{mm/t}$ or less.

