

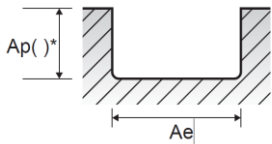
Speeds and Feeds



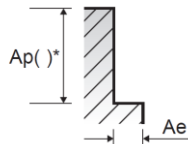
End Mill Series -
HM42

- 1) Select your material in the ISO colored chart with respect to material description.
- 2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (mm/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

Material			Recommended Cutting Values – Side Cutting									
Group		Material Description	Width of Cut, a_e	Depth of Cut, a_p	Parameter	Tool Diameter (mm)						
ISO	VDI 3323					6	8	10	12	16	20	25
P	1	Steel - Non-alloy, Cast & Free Cutting 125 HB	0.1D	1.5D	Vc, SMM	35	35	35	35	35	35	35
					Fz, MMPT	0.025	0.036	0.046	0.061	0.079	0.089	0.099
					n, RPM	1860	1390	1110	930	700	560	450
					Vf, MMPM	178	203	203	229	229	203	178
	2	Steel - Non-alloy, Cast & Free Cutting 190 HB	0.1D	1.5D	Vc, SMM	30	30	30	30	30	30	30
					Fz, MMPT	0.023	0.036	0.043	0.056	0.071	0.089	0.091
					n, RPM	1600	1190	960	800	600	480	380
	3-4	Steel - Non-alloy, Cast & Free Cutting 250 - 270 HB	0.1D	1.5D	Vf, MMPM	152	178	178	178	178	178	152
					Vc, SMM	25	25	25	25	25	25	25
					Fz, MMPT	0.018	0.028	0.038	0.048	0.058	0.066	0.074
					n, RPM	1330	1000	800	660	500	400	320
	5	Steel - Non-alloy, Cast & Free Cutting 300 HB	0.1D	1.5D	Vf, MMPM	102	127	127	127	127	102	102
					Vc, SMM	15	15	15	15	15	15	15
					Fz, MMPT	0.018	0.028	0.036	0.046	0.058	0.074	0.069
					n, RPM	800	600	480	400	300	240	190
	6	Steel - Low Alloy & Cast 180 HB	0.1D	1.5D	Vf, MMPM	51	76	76	76	76	76	76
					Vc, SMM	30	30	30	30	30	30	30
					Fz, MMPT	0.023	0.036	0.043	0.056	0.071	0.089	0.091
					n, RPM	1600	1190	960	800	600	480	380
						Vf, MMPM	152	178	178	178	178	178



Slotting



Side Cutting

NOTE: All cutting data are target values.
Maximum recommended depth shown.

Finish cuts typically require reduced feed rates and/or higher spindle speed, with a radial depth of cut, a_e of (2%)XD or less.

Reduce speed and feed recommendations for materials harder than listed.

Reduce cut depth and feed by 50% for long-flute or long-reach tools.

Above recommendations are based on ideal conditions. Adjust parameters accordingly for smaller taper machining centers or less rigid conditions.

Tech Tips: The tables above are based on common machining calculators.

We realize that shops may not have the RPM capability shown in the tables.

To adapt the tables to the machining conditions available, use the following calculation:

$$(\text{Recommended Feed MMPM} / \text{Recommended RPM}) \times \text{Available RPM} = \text{MMPM}$$



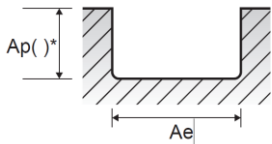
Speeds and Feeds



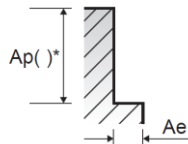
End Mill Series -
HM42

- 1) Select your material in the ISO colored chart with respect to material description.
- 2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (mm/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

Group		Material Description	Width of Cut, a _e	Depth of Cut, a _p	Parameter	Recommended Cutting Values – Side Cutting								
ISO	VDI 3323					Tool Diameter (mm)								
								6	8	10	12	16	20	25
P	7	Steel - Low Alloy & Cast 275 HB	0.1D	1.5D	Vc, SMM	25	25	25	25	25	25	25	25	
					Fz, MMPT	0.018	0.028	0.038	0.048	0.058	0.066	0.074		
					n, RPM	1330	1000	800	660	500	400	320		
					Vf, MMPM	102	127	127	127	127	102	102		
	8-9	Steel - Low Alloy & Cast 300 - 350 HB	0.1D	1.5D	Vc, SMM	15	15	15	15	15	15	15	15	
					Fz, MMPT	0.018	0.028	0.036	0.046	0.058	0.074	0.069		
					n, RPM	800	600	480	400	300	240	190		
					Vf, MMPM	51	76	76	76	76	76	76		
	10	Steel - High Alloy, Cast & Tool 200 HB	0.1D	1.5D	Vc, SMM	30	30	30	30	30	30	30	30	
					Fz, MMPT	0.023	0.036	0.043	0.056	0.071	0.089	0.091		
					n, RPM	1600	1190	960	800	600	480	380		
					Vf, MMPM	152	178	178	178	178	178	152		
11.1	Steel - Bainitic Ultra-High-Carbon 260 - 480 HB	0.1D	1.5D	Vc, SMM	15	15	15	15	15	15	15	15		
				Fz, MMPT	0.018	0.028	0.036	0.046	0.058	0.074	0.069			
				n, RPM	800	600	480	400	300	240	190			
				Vf, MMPM	51	76	76	76	76	76	76			
N	21-22	Aluminum - Wrought Alloys 60 - 100 HB	0.1D	1.5D	Vc, SMM	105	100	95	95	100	100	95		
					Fz, MMPT	0.02	0.036	0.048	0.056	0.066	0.076	0.089		
					n, RPM	5570	3980	3020	2520	1990	1590	1210		
					Vf, MMPM	457	559	584	584	533	483	432		
	23-24	Aluminum - Cast Alloys 75 - 90 HB	0.1D	1.5D	Vc, SMM	68	65	62	62	65	65	62		
					Fz, MMPT	0.02	0.036	0.048	0.056	0.066	0.074	0.089		
					n, RPM	3610	2590	1970	1650	1290	1040	790		
					Vf, MMPM	305	381	381	381	330	305	279		



Slotting



Side Cutting

NOTE: All cutting data are target values.
Maximum recommended depth shown.

Finish cuts typically require reduced feed rates and/or higher spindle speed, with a radial depth of cut, a_e of (2%)XD or less.

Reduce speed and feed recommendations for materials harder than listed.

Reduce cut depth and feed by 50% for long-flute or long-reach tools.

Above recommendations are based on ideal conditions. Adjust parameters accordingly for smaller taper machining centers or less rigid conditions.

Tech Tips: The tables above are based on common machining calculators.

We realize that shops may not have the RPM capability shown in the tables.

To adapt the tables to the machining conditions available, use the following calculation:

$$(\text{Recommended Feed MMPM} / \text{Recommended RPM}) \times \text{Available RPM} = \text{MMPM}$$



Speeds and Feeds



Feed Rate, Per Revolution (mm/min)
$v_f = f_n \cdot n$

Feed Rate, Per Tooth (mm/min)
$v_f = f_z \cdot n \cdot Z$

Feed Per Revolution (mm/rev)
$f_n = \frac{v_f}{n}$

Feed Per Tooth (mm)
$f_z = \frac{v_f}{n \cdot Z}$

Cutting Speed (m/min)
$v_c = \frac{\pi \cdot D_{tool} \cdot n}{1000}$

Spindle Speed (rev/min)
$n = \frac{v_c \cdot 1000}{\pi \cdot D_{tool}}$

Material Removal Rate (mm ³ /min)
$MMR = \frac{a_p \cdot a_e \cdot v_f}{1000}$

Metric

Symbol	Definition	Unit
v_f	Feed rate	mm/min
f_n	Feed per revolution	mm/rev
f_z	Feed per tooth	mm
v_c	Cutting speed	m/min (SMM)
n	Spindle speed	rev/min (RPM)
D_{tool}	Tool cutting diameter	mm
MMR	Material removal rate	(mm ³ /min)
a_e	Radial depth of cut	mm
a_p	Axial depth of cut	mm
Z	Number of teeth/flutes	