

Side Cutting

Speeds and Feeds

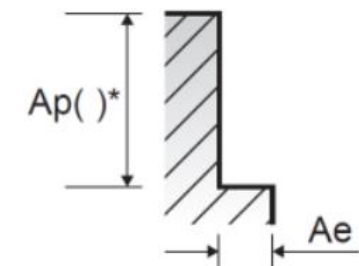


- 1) Select your material in the ISO colored chart with respect to material description.
- 2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, Vf (mm/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series - HM42

Material		Recommended Cutting Values - Side Cutting											
ISO	VDI 3323	Material Description	Width of Cut, a_e	Depth of Cut, a_p	Parameter	Cutting Diameter (mm)							
						6	8	10	12	14	16	20	25
N	21	Aluminum-Wrought Alloy	$\varnothing 6 \sim \varnothing 10 = 0.25D$ $\varnothing 12 \sim \varnothing 25 = 0.5D$	1.0D	Vc, SMM	425	490	510	525	510	575	475	475
					Fz, MMPT	0.0025	0.0036	0.006	0.007	0.009	0.009	0.015	0.019
					n, RPM	6900	5970	4930	4230	3520	3480	2310	1840
					Vf, MMPM	35	43	58	60	64	63	67	71
	22	Aluminum-Wrought Alloy	$\varnothing 6 \sim \varnothing 10 = 0.25D$ $\varnothing 12 \sim \varnothing 25 = 0.5D$	1.0D	Vc, SMM	425	490	510	525	510	575	475	475
					Fz, MMPT	0.0025	0.0036	0.006	0.007	0.0090	0.0090	0.015	0.019
					n, RPM	6900	5970	4930	4230	3520	3480	2310	1840
					Vf, MMPM	35	43	58	60	64	63	67	71
	23	Aluminum-Cast Alloy	$\varnothing 6 \sim \varnothing 10 = 0.25D$ $\varnothing 12 \sim \varnothing 25 = 0.5D$	1.0D	Vc, SMM	280	320	330	340	330	375	310	314
					Fz, MMPT	0.0025	0.0036	0.006	0.007	0.009	0.0090	0.015	0.019
					n, RPM	4510	3900	3220	2760	2300	2270	1500	1220
					Vf, MMPM	23	28	38	39	41	41	44	47
24	Aluminum-Cast Alloy	$\varnothing 6 \sim \varnothing 10 = 0.25D$ $\varnothing 12 \sim \varnothing 25 = 0.5D$	1.0D	Vc, SMM	280	320	330	340	330	375	310	314	
				Fz, MMPT	0.0025	0.0036	0.006	0.007	0.009	0.0090	0.015	0.019	
				n, RPM	4510	3900	3220	2760	2300	2270	1500	1220	
				Vf, MMPM	23	28	38	39	41	41	44	47	

NOTE: Feed to be reduced by approximately 50% if LOC (length of cut) is over 3XD. The above recommendations are based on ideal conditions; for smaller taper machining centers or less rigid conditions, please adjust parameters accordingly on diameters greater than 12mm. In profile operations, engaging more than 2XD, reduce the radial depth of cut, a_e by 50%-60%. Finish cuts typically require reduced cutting feeds and speeds; also, it is recommended the radial depth of cut, a_e should not exceed (2%)XD.



Side Cutting





Slotting

Speeds and Feeds

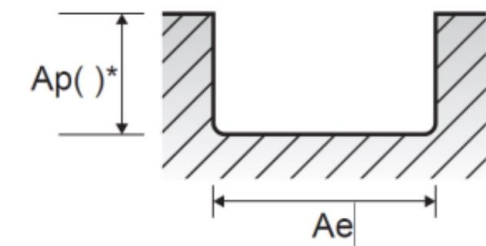


- 1) Select your material in the ISO colored chart with respect to material description.
- 2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, Vf (mm/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series - HM42

Material		Recommended Cutting Values - Slotting											
Group		Material Description	Width of Cut, a_e	Depth of Cut, a_p	Parameter	Cutting Diameter (mm)							
ISO	VDI 3323					6	8	10	12	14	16	20	25
N	21	Aluminum-Wrought Alloy	1.0D	0.5D	Vc, SMM	425	490	510	525	510	575	475	475
					Fz, MMPT	0.0020	0.0028	0.005	0.006	0.007	0.007	0.011	0.014
					n, RPM	6900	5970	4930	4230	3520	3480	2310	1840
					Vf, MMPM	27	33	47	47	49	49	51	53
			1.0D	0.5D	Vc, SMM	425	490	510	525	510	575	475	475
					Fz, MMPT	0.0020	0.0028	0.005	0.006	0.0070	0.0070	0.011	0.014
					n, RPM	6900	5970	4930	4230	3520	3480	2310	1840
					Vf, MMPM	27	33	47	47	49	49	51	53
	23	Aluminum-Cast Alloy	1.0D	0.5D	Vc, SMM	280	320	330	340	330	375	310	305
					Fz, MMPT	0.0020	0.0028	0.005	0.005	0.006	0.0070	0.011	0.014
					n, RPM	4510	3900	3220	2760	2300	2270	1500	1180
					Vf, MMPM	18	22	30	31	32	32	33	34
			1.0D	0.5D	Vc, SMM	280	320	330	340	330	375	310	305
					Fz, MMPT	0.0020	0.0028	0.005	0.005	0.006	0.0070	0.011	0.014
					n, RPM	4510	3900	3220	2760	2300	2270	1500	1180
					Vf, MMPM	18	22	30	31	32	32	33	34

NOTE: Feed to be reduced by approximately 50% if LOC (length of cut) is over 3XD. The above recommendations are based on ideal conditions; for smaller taper machining centers or less rigid conditions, please adjust parameters accordingly on diameters greater than 12mm. In profile operations, engaging more than 2XD, reduce the radial depth of cut, a_e by 50%-60%. Finish cuts typically require reduced cutting feeds and speeds; also, it is recommended the radial depth of cut, a_e should not exceed (2%)XD.



Slotting



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 (cm³/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	(cm ³ /min)
a_e	Radial depth of cut	mm
a_p	Axial depth of cut	mm
Z	Number of teeth/flutes	