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, V_f (mm/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HSAM2

| Material					Recommended Cutting Values – Side Cutting HSM (Light)							
Group		Material Description	Width of Cut, a_e	Depth of Cut, a_p	Parameter	Tool Diameter (mm)						
ISO	VDI 3323					3	6	10	12	16	20	25
N	21-22	Aluminum-Wrought Alloy	0.05D	2.0D	Vc, SMM	1006	1006	1006	1006	1006	1006	1006
					Fz, MMPT	0.053	0.14	0.267	0.356	0.381	0.419	0.495
					n, RPM	106740	53370	32022	26685	20014	16011	12809
					Vf, MMPM	17080	22367	25621	28467	22876	20131	19033
	23-25	Aluminum-Cast Alloy	0.05D	2.0D	Vc, SMM	366	366	366	366	366	366	366
					Fz, MMPT	0.053	0.14	0.267	0.356	0.381	0.419	0.495
					n, RPM	38834	19417	11650	9708	7281	5825	4660
					Vf, MMPM	6214	8138	9321	10357	8323	7324	6924
	26-28	Copper and Copper Alloys (Bronze/Brass)	0.05D	2.0D	Vc, SMM	564	564	564	564	564	564	564
					Fz, MMPT	0.043	0.114	0.216	0.292	0.33	0.356	0.406
					n, RPM	59842	29921	17953	14961	11220	8976	7181
					Vf, MMPM	7752	10260	11628	13110	11115	9576	8755
	29.1	Non-Metallic Materials (Duroplastic)	0.05D	2.0D	Vc, SMM	1021	1021	1021	1021	1021	1021	1021
					Fz, MMPT	0.086	0.229	0.432	0.584	0.635	0.699	0.813
					n, RPM	108331	54166	32499	27083	20312	16250	13000
					Vf, MMPM	28066	37147	42100	47465	38695	34051	31699

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.
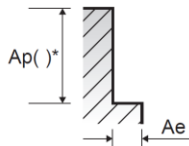*Reduce speed and feed recommendations for materials harder than listed.*
Reduce cut depth and feed by 50% for long-flute or long-reach tools.
Above recommendations are based on ideal conditions. Adjust parameters accordingly for smaller taper machining centers or less rigid conditions. HSM = high speed machining.
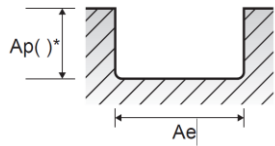
Slotting

Side Cutting

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:
        (Recommended Feed MMPM / Recommended RPM) X Available RPM = MMPM

TESTED. PROVEN.

Speed 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 (mm³/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	(mm³/min)
a_e	Radial depth of cut	mm
a_p	Axial depth of cut	mm
Z	Number of teeth/flutes	