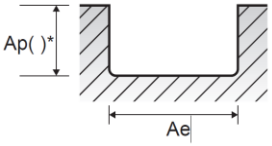
Speeds and Feeds

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

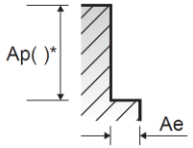
End Mill Series – HSAM2

Material			Recommended Cutting Values – Slotting								
Group		Material Description	Width of Cut, a_e	Depth of Cut, a_p	Parameter	Tool Diameter (in)						
ISO	VDI 3323					1/8	3/16	1/4	3/8	1/2	5/8	3/4
N	21-22	Aluminum-Wrought Alloy	1.0D	1.0D	Vc, SFM	2000	2000	2000	2000	2000	2000	2000
					Fz, IPT	0.001	0.002	0.003	0.0045	0.006	0.0066	0.0075
					n, RPM	61100	40750	30500	20400	15300	12200	10200
					Vf, IPM	183	245	275	275	275	242	230
	23-25	Aluminum-Cast Alloy	1.0D	1.0D	Vc, SFM	600	600	600	600	600	600	600
					Fz, IPT	0.001	0.002	0.003	0.0045	0.006	0.0066	0.0075
					n, RPM	18340	12220	9170	6110	4580	3670	3060
					Vf, IPM	55	73	83	83	83	73	69
	26-28	Copper and Copper Alloys (Bronze/Brass)	1.0D	1.0D	Vc, SFM	880	880	880	880	880	880	880
					Fz, IPT	0.0008	0.0014	0.002	0.004	0.005	0.0055	0.006
					n, RPM	26890	17930	13450	8960	6720	5380	4480
					Vf, IPM	65	75	81	108	101	89	81
	29.1	Non-Metallic Materials (Duroplastic)	1.0D	1.0D	Vc, SFM	1670	1670	1670	1670	1670	1670	1670
					Fz, IPT	0.0015	0.0028	0.004	0.0075	0.01	0.011	0.012
					n, RPM	51040	34020	25520	17010	12760	10210	8510
					Vf, IPM	230	285	306	383	383	337	306

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.



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 IPM / Recommended RPM) X Available RPM = IPM

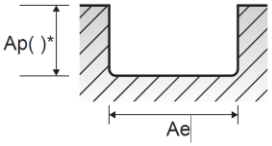
Speeds and Feeds

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

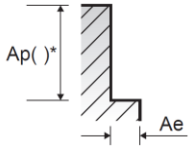
End Mill Series – HSAM2

Material			Recommended Cutting Values – Side Cutting								
Group		Material Description	Width of Cut, a_e	Depth of Cut, a_p	Parameter	Tool Diameter (in)						
ISO	VDI 3323					1/8	3/16	1/4	3/8	1/2	5/8	3/4
N	21-22	Aluminum-Wrought Alloy	0.5D	1.5D	Vc, SFM	3000	3000	3000	3000	3000	3000	3000
					Fz, IPT	0.001	0.002	0.003	0.0045	0.006	0.0066	0.0075
					n, RPM	91700	61120	45800	30600	23000	18300	15300
					Vf, IPM	275	367	412	413	414	362	344
	23-25	Aluminum-Cast Alloy	0.5D	1.5D	Vc, SFM	800	800	800	800	800	800	800
					Fz, IPT	0.001	0.002	0.003	0.0045	0.006	0.0066	0.0075
					n, RPM	24450	16300	12220	8150	6110	4890	4080
					Vf, IPM	73	98	110	110	110	97	92
	26-28	Copper and Copper Alloys (Bronze/Brass)	0.5D	1.5D	Vc, SFM	1150	1150	1150	1150	1150	1150	1150
					Fz, IPT	0.0008	0.0014	0.002	0.004	0.005	0.0055	0.006
					n, RPM	35140	23430	17570	11720	8790	7030	5860
					Vf, IPM	84	98	105	141	132	116	105
	29.1	Non-Metallic Materials (Duroplastic)	0.5D	1.5D	Vc, SFM	2070	2070	2070	2070	2070	2070	2070
					Fz, IPT	0.0015	0.0028	0.004	0.0075	0.01	0.011	0.012
					n, RPM	63260	42170	31630	21090	15820	12650	10540
					Vf, IPM	285	354	380	474	474	418	380

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.



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 IPM / Recommended RPM) X Available RPM = IPM

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	