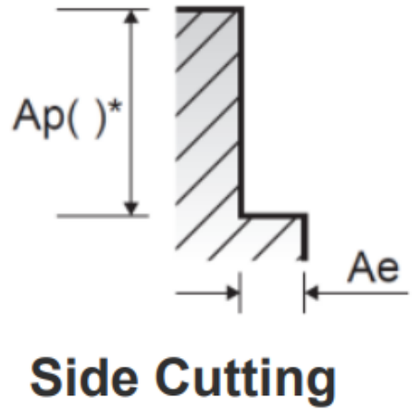
Side Cutting

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

End Mill Series - HM42

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					1/4	5/16	3/8	7/16	1/2	9/16	5/8	3/4	1"
N	21	Aluminum-wrought alloy	0.5D	1.5D	Vc, SFM	445	425	500	575	590	515	575	450	525
					Fz, IPT	0.001	0.002	0.002	0.003	0.003	0.004	0.004	0.006	0.007
					n, RPM	6800	5200	5100	5000	4500	3500	3500	2300	2000
					Vf, IPM	28	35	37	38	38	40	40	42	42
	22		0.5D	1.5D	Vc, SFM	445	425	500	575	590	515	575	450	525
					Fz, IPT	0.001	0.002	0.002	0.003	0.003	0.004	0.004	0.006	0.007
					n, RPM	6800	5200	5100	5000	4500	3500	3500	2300	2000
					Vf, IPM	28	35	37	38	38	40	40	42	42
	23	Aluminum-cast, alloyed	0.5D	1.5D	Vc, SFM	445	425	500	575	590	515	575	450	525
					Fz, IPT	0.001	0.002	0.002	0.003	0.003	0.004	0.004	0.006	0.007
					n, RPM	6800	5200	5100	5000	4500	3500	3500	2300	2000
					Vf, IPM	28	35	37	38	38	40	40	42	42
	24		0.5D	1.5D	Vc, SFM	445	425	500	575	590	515	575	450	525
					Fz, IPT	0.001	0.002	0.002	0.003	0.003	0.004	0.004	0.006	0.007
					n, RPM	6800	5200	5100	5000	4500	3500	3500	2300	2000
					Vf, IPM	28	35	37	38	38	40	40	42	42
	25		0.5D	1.5D	Vc, SFM	445	425	500	575	590	515	575	450	525
					Fz, IPT	0.001	0.002	0.002	0.003	0.003	0.004	0.004	0.006	0.007
					n, RPM	6800	5200	5100	5000	4500	3500	3500	2300	2000
					Vf, IPM	28	35	37	38	38	40	40	42	42



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

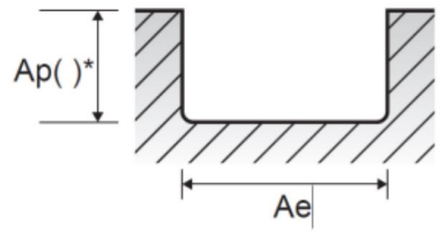
Slotting

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 (in/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	Tool Diameter (in)								
ISO	VDI 3323					1/4	5/16	3/8	7/16	1/2	9/16	5/8	3/4	1"
N	21	Aluminum-wrought alloy	1D	0.5D	Vc, SFM	445	425	500	575	590	515	575	450	525
					Fz, IPT	0.001	0.002	0.002	0.003	0.003	0.004	0.004	0.006	0.007
					n, RPM	6800	5200	5100	5000	4500	3500	3500	2300	2000
					Vf, IPM	28	35	37	38	38	40	40	42	42
	22		1D	0.5D	Vc, SFM	445	425	500	575	590	515	575	450	525
					Fz, IPT	0.001	0.002	0.002	0.003	0.003	0.004	0.004	0.006	0.007
					n, RPM	6800	5200	5100	5000	4500	3500	3500	2300	2000
					Vf, IPM	28	35	37	38	38	40	40	42	42
	23	Aluminum-cast, alloyed	1D	0.5D	Vc, SFM	445	425	500	575	590	515	575	450	525
					Fz, IPT	0.001	0.002	0.002	0.003	0.003	0.004	0.004	0.006	0.007
					n, RPM	6800	5200	5100	5000	4500	3500	3500	2300	2000
					Vf, IPM	28	35	37	38	38	40	40	42	42
	24		1D	0.5D	Vc, SFM	445	425	500	575	590	515	575	450	525
					Fz, IPT	0.001	0.002	0.002	0.003	0.003	0.004	0.004	0.006	0.007
					n, RPM	6800	5200	5100	5000	4500	3500	3500	2300	2000
					Vf, IPM	28	35	37	38	38	40	40	42	42
	25		1D	0.5D	Vc, SFM	445	425	500	575	590	515	575	450	525
					Fz, IPT	0.001	0.002	0.002	0.003	0.003	0.004	0.004	0.006	0.007
					n, RPM	6800	5200	5100	5000	4500	3500	3500	2300	2000
					Vf, IPM	28	35	37	38	38	40	40	42	42



Slotting

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, ae 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: (Recommended Feed IPM / Recommended RPM) X Available RPM = IPM

Speeds and Feeds

Feed Rate, Per Revolution (in/min)

$$v_f = f_n \cdot n$$

Feed Rate, Per Tooth (in/min)

$$v_f = f_z \cdot n \cdot Z$$

Feed Per Revolution (in/rev)

$$f_n = \frac{v_f}{n}$$

Feed Per Tooth (in)

$$f_z = \frac{v_f}{n \cdot Z}$$

Cutting Speed (ft/min)

$$v_c = \frac{\pi \cdot D_{tool} \cdot n}{12}$$

Spindle Speed (rev/min)

$$n = \frac{v_c \cdot 12}{\pi \cdot D_{tool}}$$

Material Removal Rate (in³/min)

$$MMR = a_p \cdot a_e \cdot v_f$$

Inch

Symbol	Definition	Unit
v_f	Feed rate	*in/min*
f_n	Feed per revolution	*in/rev*
f_z	Feed per tooth	*in*
v_c	Cutting speed	*ft/min (SFM)*
n	Spindle speed	*rev/min (RPM)*
D_{tool}	Tool cutting diameter	*in*
MMR	Material removal rate	*(in³/min)*
a_e	Radial depth of cut	*in*
a_p	Axial depth of cut	*in*
Z	Number of teeth/flutes	