

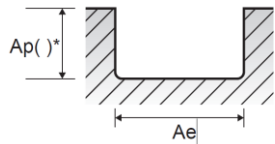
Speeds and Feeds



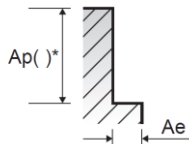
- 1) Select your material in the ISO colored chart with respect to material description.
- 2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (mm/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTPM

Material		Recommended Cutting Values – Slotting								
Group	Material Description	Width of Cut, a _e	Depth of Cut, a _p	Parameter	Tool Diameter (mm)					
ISO					VDI 3323	6	10	12	16	20
P	1-4	Steel - Non-Alloy, Cast, & Free Cutting 125 - 270 HB	1.0D	0.8D	Vc, SMM	152	168	168	168	168
					Fz, MMPT	0.0160	0.0380	0.0470	0.0530	0.0650
					n, RPM	8060	5350	4460	3340	2670
					Vf, MMPM	516	813	838	709	695
	5	Steel - Non-Alloy, Cast, & Free Cutting 300 HB	1.0D	0.8D	Vc, SMM	107	117	117	117	117
					Fz, MMPT	0.0160	0.0380	0.0469	0.0529	0.0650
					n, RPM	5680	3720	3100	2330	1860
					Vf, MMPM	363	566	583	493	484
	6-7	Steel - Low alloy & cast 180 - 275 HB	1.0D	0.8D	Vc, SMM	152	168	168	168	168
					Fz, MMPT	0.0160	0.0380	0.0470	0.0530	0.0650
					n, RPM	8060	5350	4460	3340	2670
					Vf, MMPM	516	813	838	709	695
	8-9	Steel - Low alloy & cast 300 - 350 HB	1.0D	0.8D	Vc, SMM	107	117	117	117	117
					Fz, MMPT	0.0160	0.0380	0.0469	0.0529	0.0650
					n, RPM	5680	3720	3100	2330	1860
					Vf, MMPM	363	566	583	493	484
	10	Steel - High Alloy, Cast, & Tool 200 HB	1.0D	0.8D	Vc, SMM	64	70	70	70	70
					Fz, MMPT	0.0110	0.0270	0.0320	0.0370	0.0451
					n, RPM	3400	2230	1860	1390	1110
					Vf, MMPM	149	241	238	206	201
	11.1	Steel - Bainitic Ultra-High-Carbon 260 - 480 HB	1.0D	0.8D	Vc, SMM	64	70	70	70	70
					Fz, MMPT	0.0110	0.0270	0.0320	0.0370	0.0451
					n, RPM	3400	2230	1860	1390	1110
					Vf, MMPM	149	241	238	206	201



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:

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



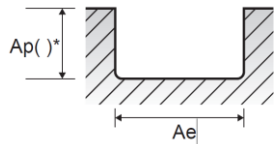
Speeds and Feeds



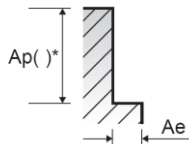
- 1) Select your material in the ISO colored chart with respect to material description.
- 2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (mm/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTPM

Material		Recommended Cutting Values – Slotting								
Group	Material Description	Width of Cut, a _e	Depth of Cut, a _p	Parameter	Tool Diameter (mm)					
ISO	VDI 3323				6	10	12	16	20	
M	12-13	Stainless Steel, Ferritic/Martensitic, Hardened and Tempered, 200 - 240 HB	1.0D	0.8D	Vc, SMM	148	148	148	148	148
					Fz, MMPT	0.0130	0.0340	0.0390	0.0450	0.0550
					n, RPM	7850	4710	3930	2940	2360
	14.1	Stainless Steel, Austenitic, Precipitation Hardened 250 HB	1.0D	0.8D	Vf, MMPM	408	641	612	530	518
					Vc, SMM	107	107	107	107	107
					Fz, MMPT	0.0180	0.0480	0.0550	0.0620	0.0771
	14.2	Stainless Steel, Austenitic-Ferritic, Solution Annealed 250 HB	1.0D	0.8D	n, RPM	5620	3370	2810	2110	1690
					Vf, MMPM	405	647	619	523	521
					Vc, SMM	94	94	94	94	94
K	15-16	Cast Iron - Gray; Ferritic / Pearlitic, Pearlitic 180 -260 HB	1.0D	0.8D	Fz, MMPT	0.0200	0.0480	0.0580	0.0650	0.0809
					n, RPM	5940	3920	3260	2450	1960
					Vf, MMPM	475	752	757	636	634
	17-18	Ductile Iron - Nodular Graphite; Ferritic, Pearlitic 160 - 250 HB	1.0D	0.8D	Vc, SMM	111	123	123	123	123
					Fz, MMPT	0.0200	0.0480	0.0580	0.0650	0.0809
					n, RPM	5940	3920	3260	2450	1960
	19-20	Cast Iron - Malleable; Ferritic, Pearlitic 130 - 230 HB	1.0D	0.8D	Vf, MMPM	475	752	757	636	634
					Vc, SMM	111	123	123	123	123
					Fz, MMPT	0.0200	0.0480	0.0580	0.0650	0.0809
					n, RPM	5940	3920	3260	2450	1960
					Vf, MMPM	475	752	757	636	634



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:

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



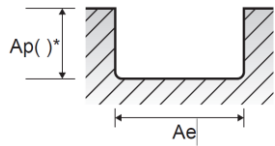
Speeds and Feeds



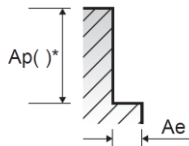
- 1) Select your material in the ISO colored chart with respect to material description.
- 2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (mm/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTPM

Material			Recommended Cutting Values – Slotting							
Group		Material Description	Width of Cut, a_e	Depth of Cut, a_p	Parameter	Tool Diameter (mm)				
ISO	VDI 3323					6	10	12	16	20
S	31-35	Heat Resistant Super Alloys 200 HB - 350 HB	1.0D	0.5D	V_c , SMM	26	26	26	26	26
					F_z , MMPT	0.0120	0.0329	0.0380	0.0430	0.0537
					n , RPM	1380	830	690	520	410
					V_f , MMPM	66	109	105	89	89
	36-37	Titanium Alloys	1.0D	0.5D	V_c , SMM	58	58	58	58	58
					F_z , MMPT	0.0160	0.0420	0.0501	0.0550	0.0680
					n , RPM	3080	1850	1540	1150	920
					V_f , MMPM	197	310	308	254	251



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:

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



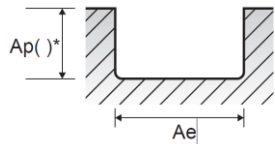
Speeds and Feeds



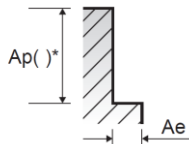
- 1) Select your material in the ISO colored chart with respect to material description.
- 2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (mm/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTPM

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	10	12	16	20	
P	1-4	Steel - Non-Alloy, Cast, & Free Cutting 125 - 270 HB	0.5D	1.2D	Vc, SMM	152	168	168	168	168	
					Fz, MMPT	0.0160	0.0380	0.0470	0.0530	0.0650	
					n, RPM	8060	5350	4460	3340	2670	
	5	Steel - Non-Alloy, Cast, & Free Cutting 300 HB	0.5D	1.2D	Vf, MMPM	516	813	838	709	695	
					Vc, SMM	107	117	117	117	117	
					Fz, MMPT	0.0160	0.0380	0.0469	0.0529	0.0650	
	6-7	Steel - Low alloy & cast 180 - 275 HB	0.5D	1.2D	n, RPM	5680	3720	3100	2330	1860	
					Vf, MMPM	363	566	583	493	484	
					Vc, SMM	152	168	168	168	168	
	8-9	Steel - Low alloy & cast 300 - 350 HB	0.5D	1.2D	Fz, MMPT	0.0160	0.0380	0.0470	0.0530	0.0650	
					n, RPM	8060	5350	4460	3340	2670	
					Vf, MMPM	516	813	838	709	695	
	10	Steel - High Alloy, Cast, & Tool 200 HB	0.5D	1.2D	Vc, SMM	107	117	117	117	117	
					Fz, MMPT	0.0110	0.0270	0.0320	0.0370	0.0451	
					n, RPM	3400	2230	1860	1390	1110	
	11.1	Steel - Bainitic Ultra-High-Carbon 260 - 480 HB	0.5D	1.2D	Vf, MMPM	149	241	238	206	201	
					Vc, SMM	64	70	70	70	70	
					Fz, MMPT	0.0110	0.0270	0.0320	0.0370	0.0451	
						n, RPM	3400	2230	1860	1390	1110
						Vf, MMPM	149	241	238	206	201



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:

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



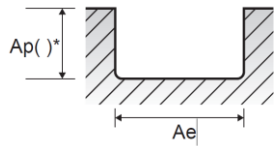
Speeds and Feeds



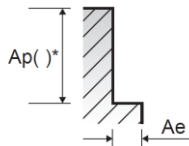
- 1) Select your material in the ISO colored chart with respect to material description.
- 2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (mm/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTPM

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	10	12	16	20
M	12-13	Stainless Steel, Ferritic/Martensitic, Hardened and Tempered, 200 - 240 HB	0.5D	1.2D	Vc, SMM	148	148	148	148	148
					Fz, MMPT	0.0130	0.0340	0.0390	0.0450	0.0550
					n, RPM	7850	4710	3930	2940	2360
	14.1	Stainless Steel, Austenitic, Precipitation Hardened 250 HB	0.5D	1.2D	Vc, SMM	107	107	107	107	107
					Fz, MMPT	0.0180	0.0480	0.0550	0.0620	0.0771
					n, RPM	5620	3370	2810	2110	1690
	14.2	Stainless Steel, Austenitic-Ferritic, Solution Annealed 250 HB	0.5D	1.2D	Vc, SMM	94	94	94	94	94
					Fz, MMPT	0.0180	0.0480	0.0550	0.0620	0.0761
					n, RPM	5040	3020	2520	1890	1510
K	15-16	Cast Iron - Gray; Ferritic / Pearlitic, Pearlitic 180 - 260 HB	0.5D	1.2D	Vc, SMM	111	123	123	123	123
					Fz, MMPT	0.0200	0.0480	0.0580	0.0650	0.0809
					n, RPM	5940	3920	3260	2450	1960
	17-18	Ductile Iron - Nodular Graphite; Ferritic, Pearlitic 160 - 250 HB	0.5D	1.2D	Vc, SMM	111	123	123	123	123
					Fz, MMPT	0.0200	0.0480	0.0580	0.0650	0.0809
					n, RPM	5940	3920	3260	2450	1960
	19-20	Cast Iron - Malleable; Ferritic, Pearlitic 130 - 230 HB	0.5D	1.2D	Vc, SMM	111	123	123	123	123
					Fz, MMPT	0.0200	0.0480	0.0580	0.0650	0.0809
					n, RPM	5940	3920	3260	2450	1960
					Vf, MMPM	475	752	757	636	634



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:

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



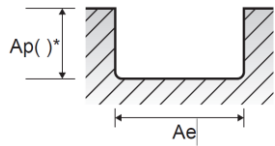
Speeds and Feeds



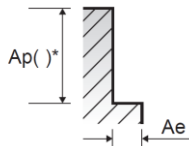
- 1) Select your material in the ISO colored chart with respect to material description.
- 2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (mm/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTPM

Material		Recommended Cutting Values – Side Cutting								
Group		Width of Cut, a_e	Depth of Cut, a_p	Parameter	Tool Diameter (mm)					
ISO	VDI 3323				6	10	12	16	20	
S	31-35	Heat Resistant Super Alloys 200 HB - 350 HB	0.25D	1.0D	Vc, SMM	26	26	26	26	26
					Fz, MMPT	0.0120	0.0329	0.0380	0.0430	0.0537
					n, RPM	1380	830	690	520	410
					Vf, MMPM	66	109	105	89	89
	36-37	Titanium Alloys	0.35D	1.0D	Vc, SMM	58	58	58	58	58
					Fz, MMPT	0.0160	0.0420	0.0501	0.0550	0.0680
					n, RPM	3080	1850	1540	1150	920
					Vf, MMPM	197	310	308	254	251



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	