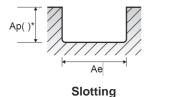


End Mill Conice

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.

84-1										End Mill Series – HTPM
		Material					Recommended Cutting Val	ues – Slotting		
Group				Tool Diameter (in)						
ISO	VDI 3323	Material Description	Width of Cut, a _e	Depth of Cut, a _p	Parameter	1/4	3/8	1/2	5/8	3/4
					Vc, SFM	500	550	550	550	550
	1-4	Steel - Non-Alloy, Cast, & Free Cutting	1.0D	0.8D	Fz, IPT	0.0006	0.0015	0.0019	0.0021	0.0026
	1-4	125 - 270 HB	1.00	0.60	n, RPM	7620	5620	4210	3370	2800
					Vf, IPM	19	34	31	28	29
					Vc, SFM	350	385	385	385	385
	5	Steel - Non-Alloy, Cast, & Free Cutting	1.0D	0.8D	Fz, IPT	0.0006	0.0015	0.0019	0.0021	0.0026
	3	300 HB	1.0D	0.8D	n, RPM	5360	3910	2930	2350	1960
		000112			Vf, IPM	14	23	22	20	20
		Steel - Low alloy & cast 180 - 275 HB	1.0D	0.8D	Vc, SFM	500	550	550	550	550
	6-7				Fz, IPT	0.0006	0.0015	0.0019	0.0021	0.0026
	0-7				n, RPM	7620	5620	4210	3370	2800
Р					Vf, IPM	19	34	31	28	29
			1.0D	0.8D	Vc, SFM	350	385	385	385	385
	8-9	Steel - Low alloy & cast			Fz, IPT	0.0006	0.0015	0.0019	0.0021	0.0026
	0-0	300 - 350 HB			n, RPM	5360	3910	2930	2350	1960
					Vf, IPM	14	23	22	20	20
					Vc, SFM	210	230	230	230	230
	10	Steel - High Alloy,Cast, & Tool	1.0D	0.8D	Fz, IPT	0.0004	0.0011	0.0013	0.0015	0.0018
		200 HB	1.00	0.00	n, RPM	3210	2340	1760	1410	1170
					Vf, IPM	6	10	9	8	8
					Vc, SFM	210	230	230	230	230
	11.1	Steel - Bainitic Ultra-High-Carbon	1.0D	0.8D	Fz, IPT	0.0004	0.0011	0.0013	0.0015	0.0018
		260 - 480 HB		0.00	n, RPM	3210	2340	1760	1410	1170
					Vf, IPM	6	10	9	8	8



Ap()*

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:

Side Cutting



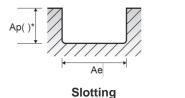


End Mill Orada

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 – HTPM	
		Material					Recommended Cutting Va	lues – Slotting			
0	roup								Tool Diameter (in)		
ISO	VDI 3323	Material Description	Width of Cut, a _e	Depth of Cut, a _p	Parameter	1/4	3/8	1/2	5/8	3/4	
		Stainless Steel,			Vc, SFM	485	485	485	485	485	
	12-13	Ferritic/Martensitic, Hardened and	1.0D	0.8D	Fz, IPT	0.0005	0.0013	0.0015	0.0018	0.0022	
	12-13	Tempered,	1.00	0.60	n, RPM	7430	4950	3710	2970	2480	
		200 - 240 HB			Vf, IPM	15	27	23	21	21	
					Vc, SFM	350	350	350	350	350	
м	14.1	Stainless Steel, Austenitic, Precipitation Hardened	1.0D	0.8D	Fz, IPT	0.0007	0.0019	0.0022	0.0024	0.0030	
IVI	196.1	250 HB	1.00	0.8D	n, RPM	5317	3545	2659	2127	1772	
					Vf, IPM	15	27	23	21	21	
		Stainless Steel, Austenitic-Ferritic, Solution Annealed 250 HB	1.0D 1.0D	0.8D	Vc, SFM	310	310	310	310	310	
	14.2				Fz, IPT	0.0007	0.0019	0.0022	0.0024	0.0030	
	14.2				n, RPM	4767	3178	2384	1907	1589	
					Vf, IPM	14	24	21	19	19	
					Vc, SFM	365	405	405	405	405	
	15-16	Cast Iron - Gray; Ferritic / Pearlitic, Pearlitic			Fz, IPT	0.0008	0.0019	0.0023	0.0026	0.0032	
	13-10	180 -260 HB			n, RPM	5608	4115	3087	2469	2058	
					Vf, IPM	18	31	28	25	26	
		Dustile lass Neddas Orachites			Vc, SFM	365	405	405	405	405	
к	17-18	Ductile Iron - Nodular Graphite; Ferritic, Pearlitic	1.0D	0.8D	Fz, IPT	0.0008	0.0019	0.0023	0.0026	0.0032	
Ň	17-10	160 - 250 HB	1.00	0.80	n, RPM	5608	4115	3087	2469	2058	
					Vf, IPM	18	31	28	25	26	
		Cost Iron Mollocki			Vc, SFM	365	405	405	405	405	
	19-20	Cast Iron - Malleable; Ferritic, Pearlitic	1.0D	0.8D	Fz, IPT	0.0008	0.0019	0.0023	0.0026	0.0032	
	13-20	130 - 230 HB	1.00	0.00	n, RPM	5608	4115	3087	2469	2058	
					Vf, IPM	18	31	28	25	26	





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:

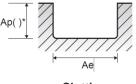
Side Cutting





- 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 – HTPM								
	Material Recommended Cutting Values – Slotting																		
	Gro	bup							Tool Diameter (in)										
15		VDI 3323	Material Description	Width of Cut, a _e	Depth of Cut, a _p	Parameter	1/4	3/8	1/2	5/8	3/4								
				1.0D	0.5D	Vc, SFM	85	85	85	85	85								
		31-35	Heat Resistant Super Alloys 200 HB - 350 HB			Fz, IPT	0.0005	0.0013	0.0015	0.0017	0.0021								
						0.5D	0.5D	0.50	0.50	0.50	0.50	0.50	0.50	0.50	n, RPM	1299	866	649	520
						Vf, IPM	2	5	4	4	4								
	ິ					Vc, SFM	190	190	190	190	190								
		36-37		1.0D	0.5D	Fz, IPT	0.0006	0.0017	0.0020	0.0022	0.0027								
		30-37	Titanium Alloys	1.00	0.5D	n, RPM	2903	1935	1452	1161	968								
						Vf, IPM	7	13	11	10	10								





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

Side Cutting

Ae





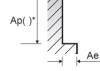
End Mill Series HTPM

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 – HTPM
		Material					Recommended Cutting Value	es – Side Cutting		
Group							Tool Diameter (in)			
ISO	VDI 3323	Material Description	Width of Cut, a _e	Depth of Cut, a _p	Parameter	1/4	3/8	1/2	5/8	3/4
					Vc, SFM	500	550	550	550	550
	1-4	Steel - Non-Alloy, Cast, & Free Cutting	0.5D	1.2D	Fz, IPT	0.0006	0.0015	0.0019	0.0021	0.0026
	1-4	125 - 270 HB	0.50	1.20	n, RPM	7620	5620	4210	3370	2800
		120 210112			Vf, IPM	19	34	31	28	29
					Vc, SFM	350	385	385	385	385
	5	Steel - Non-Alloy, Cast, & Free Cutting	0.5D	1.2D	Fz, IPT	0.0006	0.0015	0.0019	0.0021	0.0026
	3	300 HB	0.50	1.20	n, RPM	5360	3910	2930	2350	1960
		000112			Vf, IPM	14	23	22	20	20
		Steel - Low alloy & cast 180 - 275 HB	0.5D	1.2D	Vc, SFM	500	550	550	550	550
	6-7				Fz, IPT	0.0006	0.0015	0.0019	0.0021	0.0026
	0-7				n, RPM	7620	5620	4210	3370	2800
Р					Vf, IPM	19	34	31	28	29
			0.5D	1.2D	Vc, SFM	350	385	385	385	385
	8-9	Steel - Low alloy & cast			Fz, IPT	0.0006	0.0015	0.0019	0.0021	0.0026
	0-3	300 - 350 HB			n, RPM	5360	3910	2930	2350	1960
					Vf, IPM	14	23	22	20	20
					Vc, SFM	210	230	230	230	230
	10	Steel - High Alloy, Cast, & Tool	0.5D	1.2D	Fz, IPT	0.0004	0.0011	0.0013	0.0015	0.0018
		200 HB	0.55	1.20	n, RPM	3210	2340	1760	1410	1170
					Vf, IPM	6	10	9	8	8
					Vc, SFM	210	230	230	230	230
	11.1	Steel - Bainitic Ultra-High-Carbon	0.5D	1.2D	Fz, IPT	0.0004	0.0011	0.0013	0.0015	0.0018
		260 - 480 HB	0.00	1.20	n, RPM	3210	2340	1760	1410	1170
					Vf, IPM	6	10	9	8	8

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:

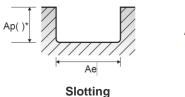




1) Select your material in the ISO colored chart with respect to material description.

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 – HTPM	
		Material					Recommended Cutting Value	es – Side Cutting			
Group									Tool Diameter (in)		
ISO	VDI 3323	Material Description	Width of Cut, a _e	Depth of Cut, a _p	Parameter	1/4	3/8	1/2	5/8	3/4	
		Stainless Steel,			Vc, SFM	485	485	485	485	485	
	12-13	Ferritic/Martensitic, Hardened and	0.5D	1.2D	Fz, IPT	0.0005	0.0013	0.0015	0.0018	0.0022	
	12-13	Tempered,	0.5D	1.20	n, RPM	7430	4950	3710	2970	2480	
		200 - 240 HB			Vf, IPM	15	27	23	21	21	
		Otaislana Otaal. Avatasitis			Vc, SFM	350	350	350	350	350	
м	14.1	Stainless Steel, Austenitic, Precipitation Hardened	0.5D	1.2D	Fz, IPT	0.0007	0.0019	0.0022	0.0024	0.0030	
141	14.1	250 HB	0.50	1.20	n, RPM	5317	3545	2659	2127	1772	
					Vf, IPM	15	27	23	21	21	
		Stainless Steel, Austenitic-Ferritic, Solution Annealed 250 HB	0.5D	1.2D	Vc, SFM	310	310	310	310	310	
	14.2				Fz, IPT	0.0007	0.0019	0.0022	0.0024	0.0030	
	14.2				n, RPM	4767	3178	2384	1907	1589	
					Vf, IPM	14	24	21	19	19	
		Oracle Income Oracle			Vc, SFM	365	405	405	405	405	
	15-16	Cast Iron - Gray; Ferritic / Pearlitic, Pearlitic	0.5D	1.2D	Fz, IPT	0.0008	0.0019	0.0023	0.0026	0.0032	
	13-10	180 -260 HB	0.50	1.20	n, RPM	5608	4115	3087	2469	2058	
					Vf, IPM	18	31	28	25	26	
		Ductile Iron - Nodular Graphite;			Vc, SFM	365	405	405	405	405	
к	17-18	Ferritic, Pearlitic	0.5D	1.2D	Fz, IPT	0.0008	0.0019	0.0023	0.0026	0.0032	
		160 - 250 HB	0.00	1.20	n, RPM	5608	4115	3087	2469	2058	
					Vf, IPM	18	31	28	25	26	
		Cast Iron - Malleable:			Vc, SFM	365	405	405	405	405	
	19-20	Ferritic. Pearlitic	0.5D	1.2D	Fz, IPT	0.0008	0.0019	0.0023	0.0026	0.0032	
		130 - 230 HB	0.00	20	n, RPM	5608	4115	3087	2469	2058	
					Vf, IPM	18	31	28	25	26	





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:

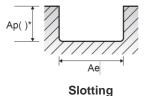
Side Cutting





- 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 – HTPM						
	Material Recommended Cutting Values - Side Cutting															
	Group							Tool Diameter (in)								
IS	O VDI 3323	Material Description	Width of Cut, a _e	Depth of Cut, a _p	Parameter	1/4	3/8	1/2	5/8	3/4						
				1.0D	Vc, SFM	85	85	85	85	85						
	31-3	Heat Resistant Super Alloys	0.25D		Fz, IPT	0.0005	0.0013	0.0015	0.0017	0.0021						
	31-3	200 HB - 350 HB			1.00	1.00	1.00	1.00	1.00	1.00	n, RPM	1299	866	649	520	433
					Vf, IPM	2	5	4	4	4						
					Vc, SFM	190	190	190	190	190						
	36-3	Titanium Alloys	0.25D	1.00	Fz, IPT	0.0006	0.0017	0.0020	0.0022	0.0027						
	30-3	Intanium Alloys	0.35D	1.0D	n, RPM	2903	1935	1452	1161	968						
					Vf, IPM	7	13	11	10	10						





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.

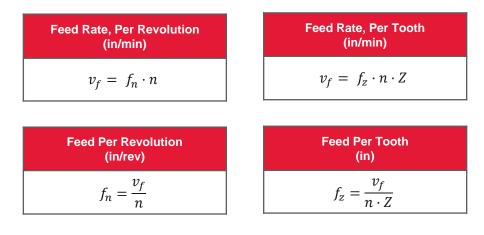
Reduce cut depth and feed by 50% for long-flute or long-reach tools.

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







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 (in ³ /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)
п	Spindle speed	rev/min (RPM)
D _{tool}	Tool cutting diameter	in
MRR	Material removal rate	(in³/min)
a _e	Radial depth of cut	in
a_p	Axial depth of cut	in
Ζ	Number of teeth/flutes	



HaasTooling.com | HaasCNC.com