

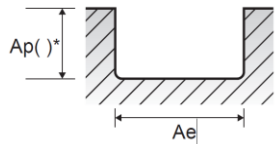
# Speeds and Feeds



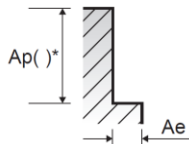
- 1) Select your material in the ISO colored chart with respect to material description.
- 2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V<sub>f</sub> (mm/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – **HTM**

Material		Recommended Cutting Values – <b>Slotting</b>								
Group	Material Description	Width of Cut, a <sub>e</sub>	Depth of Cut, a <sub>p</sub>	Parameter	Tool Diameter (mm)					
ISO					VDI 3323	6	10	12	16	20
P	1-5	Steel - Non-Alloy, Cast, & Free Cutting 125 - 270 HB	1.0D	1.0D	Vc, SMM	84	84	84	84	84
					Fz, MMPT	0.03048	0.04826	0.05842	0.08128	0.09144
					n, RPM	4450	2670	2220	1670	1330
					Vf, MMPM	686	635	660	686	610
	6-8	Steel - Low alloy & cast 180 - 275 HB	1.0D	1.0D	Vc, SMM	84	84	84	84	84
					Fz, MMPT	0.03048	0.04826	0.05842	0.08128	0.09144
					n, RPM	4450	2670	2220	1670	1330
					Vf, MMPM	686	635	660	686	610
	9	Steel - Low alloy & cast 300 - 350 HB	1.0D	1.0D	Vc, SMM	84	84	84	84	84
					Fz, MMPT	0.02286	0.0381	0.04572	0.06604	0.06858
					n, RPM	4450	2670	2220	1670	1330
					Vf, MMPM	508	508	508	559	457
	10	Steel - High Alloy, Cast, & Tool 200 HB	1.0D	0.75D	Vc, SMM	70	70	70	70	70
					Fz, MMPT	0.03048	0.04826	0.05842	0.08128	0.09144
					n, RPM	3720	2230	1860	1390	1120
					Vf, MMPM	559	533	533	559	508
11.1	Steel - Bainitic Ultra-High-Carbon 260 - 480 HB	1.0D	0.75D	Vc, SMM	76	76	76	76	76	
				Fz, MMPT	0.02286	0.0381	0.04572	0.06604	0.06858	
				n, RPM	4040	2430	2020	1520	1210	
				Vf, MMPM	508	508	508	559	457	



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<sub>e</sub> 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:

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



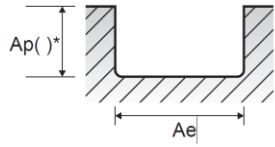
# Speeds and Feeds



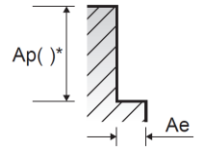
- 1) Select your material in the ISO colored chart with respect to material description.
- 2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V<sub>f</sub> (mm/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTM

Group		Material Description	Width of Cut, a <sub>e</sub>	Depth of Cut, a <sub>p</sub>	Parameter	Recommended Cutting Values – Slotting					
ISO	VDI 3323					Tool Diameter (mm)					
							6	10	12	16	20
M	12-13	Stainless Steel, Ferritic/Martensitic, Hardened and Tempered, 200 - 240 HB	1.0D	0.5D	Vc, SMM	69	69	69	69	69	
					Fz, MMPT	0.02032	0.03302	0.04064	0.05334	0.05842	
					n, RPM	3640	2180	1820	1360	1090	
					Vf, MMPM	381	356	381	356	330	
	14.1	Stainless Steel, Austenitic, Precipitation Hardened 250 HB	1.0D	0.5D	Vc, SMM	76	76	76	76	76	
					Fz, MMPT	0.02286	0.0381	0.0508	0.06604	0.06604	
					n, RPM	4040	2430	2020	1520	1210	
	14.2	Stainless Steel, Austenitic-Ferritic, Solution Annealed 250 HB	1.0D	0.5D	Vc, SMM	61	61	61	61	61	
					Fz, MMPT	0.02032	0.03048	0.04064	0.0508	0.05334	
n, RPM					3230	1940	1620	1210	970		
K	15-16	Cast Iron - Gray; Ferritic / Pearlitic, Pearlitic 180 -260 HB	1.0D	1.0D	Vc, SMM	79	79	79	79	79	
					Fz, MMPT	0.0254	0.04064	0.0508	0.07366	0.07874	
					n, RPM	4200	2520	2100	1580	1260	
	17-18	Ductile Iron - Nodular Graphite; Ferritic, Pearlitic 160 - 250 HB	1.0D	1.0D	Vc, SMM	79	79	79	79	79	
					Fz, MMPT	0.0254	0.04064	0.0508	0.07366	0.07874	
					n, RPM	4200	2520	2100	1580	1260	
	19-20	Cast Iron - Malleable; Ferritic, Pearlitic 130 - 230 HB	1.0D	1.0D	Vc, SMM	79	79	79	79	79	
					Fz, MMPT	0.0254	0.04064	0.0508	0.07366	0.07874	
					n, RPM	4200	2520	2100	1580	1260	
					Vf, MMPM	533	508	533	584	508	



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<sub>e</sub> 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 MMPM / Recommended RPM) X Available RPM = MMPM



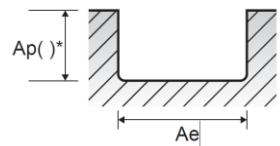
# Speeds and Feeds



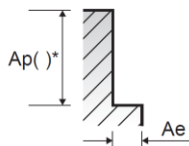
- 1) Select your material in the ISO colored chart with respect to material description.
- 2) Start with a middle/average value for spindle speed,  $n$  (RPM) and feed rate,  $V_f$  (mm/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

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	0.3D	$V_c$ , SMM	20	20	20	20	20
					$F_z$ , MMPT	0.02032	0.03302	0.0381	0.0381	0.04826
					$n$ , RPM	1030	620	520	390	310
					$V_f$ , MMPM	102	102	102	76	76
	36-37	Titanium Alloys	1.0D	0.5D	$V_c$ , SMM	49	49	49	49	49
					$F_z$ , MMPT	0.02032	0.03048	0.03556	0.04826	0.0508
					$n$ , RPM	2590	1550	1290	970	780
					$V_f$ , MMPM	254	229	229	229	203

End Mill Series - HTM



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:

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



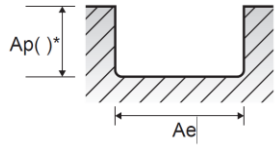
# Speeds and Feeds



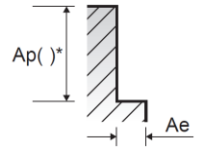
- 1) Select your material in the ISO colored chart with respect to material description.
- 2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V<sub>f</sub> (mm/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTM

Material		Recommended Cutting Values – Heavy Side Cutting								
Group	Material Description	Width of Cut, a <sub>e</sub>	Depth of Cut, a <sub>p</sub>	Parameter	Tool Diameter (mm)					
ISO					VDI 3323	6	10	12	16	20
P	1-5	Steel - Non-Alloy, Cast, & Free Cutting 125 - 270 HB	0.5D	1.5D	Vc, SMM	152	152	152	152	152
					Fz, MMPT	0.04064	0.05588	0.07874	0.09398	0.10922
					n, RPM	7640	5090	3820	3060	2550
					Vf, MMPM	1549	1422	1499	1448	1397
	6-8	Steel - Low alloy & cast 180 - 275 HB	0.5D	1.5D	Vc, SMM	152	152	152	152	152
					Fz, MMPT	0.04064	0.05588	0.07874	0.09398	0.10922
					n, RPM	7640	5090	3820	3060	2550
					Vf, MMPM	1549	1422	1499	1448	1397
	9	Steel - Low alloy & cast 300 - 350 HB	0.5D	1.5D	Vc, SMM	122	122	122	122	122
					Fz, MMPT	0.03048	0.04318	0.06096	0.07112	0.08382
					n, RPM	6110	4070	3060	2440	2040
					Vf, MMPM	940	889	940	864	864
	10	Steel - High Alloy, Cast, & Tool 200 HB	0.5D	1.5D	Vc, SMM	137	137	137	137	137
					Fz, MMPT	0.04064	0.05588	0.07874	0.09398	0.10922
					n, RPM	6880	4580	3440	2750	2290
					Vf, MMPM	1397	1270	1346	1295	1245
	11.1	Steel - Bainitic Ultra-High-Carbon 260 - 480 HB	0.5D	1.5D	Vc, SMM	122	122	122	122	122
					Fz, MMPT	0.03048	0.04318	0.06096	0.07112	0.08382
					n, RPM	6110	4070	3060	2440	2040
					Vf, MMPM	940	889	940	864	864



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<sub>e</sub> 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 MMPM / Recommended RPM) X Available RPM = MMPM



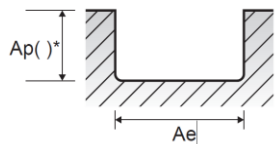
# Speeds and Feeds



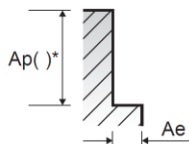
- 1) Select your material in the ISO colored chart with respect to material description.
- 2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V<sub>f</sub> (mm/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTM

Material		Recommended Cutting Values – Heavy Side Cutting								
ISO	VDI 3323	Material Description	Width of Cut, a <sub>e</sub>	Depth of Cut, a <sub>p</sub>	Parameter	Tool Diameter (mm)				
						6	10	12	16	20
M	12-13	Stainless Steel, Ferritic/Martensitic, Hardened and Tempered, 200 - 240 HB	0.5D	1.5D	Vc, SMM	76	76	76	76	76
					Fz, MMPT	0.0254	0.0381	0.05334	0.06096	0.07112
					n, RPM	3820	2550	1910	1530	1270
	14.1	Stainless Steel, Austenitic, Precipitation Hardened 250 HB	0.5D	1.5D	Vf, MMPM	483	483	508	457	457
					Vc, SMM	91	91	91	91	91
					Fz, MMPT	0.03302	0.04572	0.06604	0.07112	0.07874
	14.2	Stainless Steel, Austenitic-Ferritic, Solution Annealed 250 HB	0.5D	1.5D	n, RPM	4580	3060	2290	1830	1530
					Vf, MMPM	762	711	762	660	610
					Vc, SMM	61	61	61	61	61
K	15-16	Cast Iron - Gray; Ferritic / Pearlitic, Pearlitic 180 -260 HB	0.5D	1.5D	Fz, MMPT	0.0254	0.03556	0.05334	0.05588	0.0635
					n, RPM	3060	2040	1530	1220	1020
					Vf, MMPM	381	356	406	330	330
	17-18	Ductile Iron - Nodular Graphite; Ferritic, Pearlitic 160 - 250 HB	0.5D	1.5D	Vc, SMM	113	113	113	113	113
					Fz, MMPT	0.03556	0.04826	0.06604	0.08128	0.09398
					n, RPM	5650	3770	2830	2260	1880
	19-20	Cast Iron - Malleable; Ferritic, Pearlitic 130 - 230 HB	0.5D	1.5D	Vf, MMPM	1016	914	940	914	889
					Vc, SMM	113	113	113	113	113
					Fz, MMPT	0.03556	0.04826	0.06604	0.08128	0.09398
					n, RPM	5650	3770	2830	2260	1880
					Vf, MMPM	1016	914	940	914	889



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<sub>e</sub> 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:

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



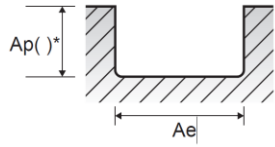
# Speeds and Feeds



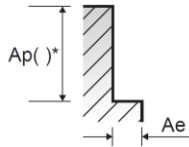
- 1) Select your material in the ISO colored chart with respect to material description.
- 2) Start with a middle/average value for spindle speed,  $n$  (RPM) and feed rate,  $V_f$  (mm/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTM

Material			Recommended Cutting Values – Heavy 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
S	31-35	Heat Resistant Super Alloys 200 HB - 350 HB	0.2D	1.5D	$V_c$ , SMM	27	27	27	27	27
					$F_z$ , MMPT	0.0254	0.03556	0.04826	0.05334	0.05842
					$n$ , RPM	1380	920	690	550	460
					$V_f$ , MMPM	178	152	178	152	127
	36-37	Titanium Alloys	0.5D	1.5D	$V_c$ , SMM	49	49	49	49	49
					$F_z$ , MMPT	0.0254	0.03556	0.04826	0.05334	0.05842
					$n$ , RPM	2440	1630	1220	980	810
					$V_f$ , MMPM	305	279	305	254	229



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



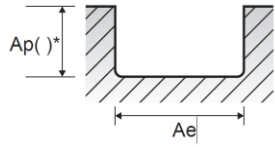
# Speeds and Feeds



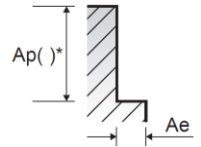
- 1) Select your material in the ISO colored chart with respect to material description.
- 2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V<sub>f</sub> (mm/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTM

Material		Recommended Cutting Values – Peel Cutting								
Group	Material Description	Width of Cut, a <sub>e</sub>	Depth of Cut, a <sub>p</sub>	Parameter	Tool Diameter (mm)					
ISO					VDI 3323	6	10	12	16	20
P	1-5	Steel - Non-Alloy, Cast, & Free Cutting 125 - 270 HB	0.08D	2.0D	Vc, SMM	198	198	198	198	198
					Fz, MMPT	0.05588	0.07874	0.10922	0.12954	0.1524
					n, RPM	9930	6620	4970	3970	3310
					Vf, MMPM	2769	2616	2718	2565	2515
	6-8	Steel - Low alloy & cast 180 - 275 HB	0.08D	2.0D	Vc, SMM	198	198	198	198	198
					Fz, MMPT	0.05588	0.07874	0.10922	0.12954	0.1524
					n, RPM	9930	6620	4970	3970	3310
					Vf, MMPM	2769	2616	2718	2565	2515
	9	Steel - Low alloy & cast 300 - 350 HB	0.08D	2.0D	Vc, SMM	198	198	198	198	198
					Fz, MMPT	0.04318	0.06096	0.08382	0.1016	0.11684
					n, RPM	9930	6620	4970	3970	3310
					Vf, MMPM	2134	2007	2083	2007	1930
	10	Steel - High Alloy, Cast, & Tool 200 HB	0.08D	2.0D	Vc, SMM	177	177	177	177	177
					Fz, MMPT	0.05588	0.07874	0.10922	0.12954	0.1524
					n, RPM	8860	5910	4430	3540	2950
					Vf, MMPM	2464	2337	2413	2286	2261
	11.1	Steel - Bainitic Ultra-High-Carbon 260 - 480 HB	0.08D	2.0D	Vc, SMM	168	168	168	168	168
					Fz, MMPT	0.04318	0.06096	0.08382	0.1016	0.11684
					n, RPM	8400	5600	4200	3360	2800
					Vf, MMPM	1803	1702	1753	1702	1626



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<sub>e</sub> 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 MMPM / Recommended RPM) X Available RPM = MMPM

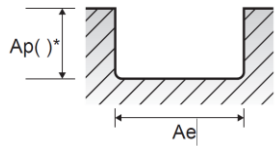


# Speeds and Feeds

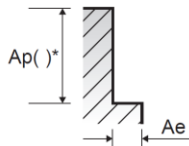


- 1) Select your material in the ISO colored chart with respect to material description.
- 2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V<sub>f</sub> (mm/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

Material		Recommended Cutting Values – Peel Cutting								End Mill Series – HTM
Group	Material Description	Width of Cut, a <sub>e</sub>	Depth of Cut, a <sub>p</sub>	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	0.06D	2.0D	Vc, SMM	107	107	107	107	107
					Fz, MMPT	0.0381	0.05334	0.07366	0.08636	0.09906
					n, RPM	5350	3570	2670	2140	1780
	14.1	Stainless Steel, Austenitic, Precipitation Hardened 250 HB	0.06D	2.0D	Vc, SMM	130	130	130	130	130
					Fz, MMPT	0.04572	0.0635	0.09144	0.09906	0.11176
					n, RPM	6490	4330	3250	2600	2160
	14.2	Stainless Steel, Austenitic-Ferritic, Solution Annealed 250 HB	0.06D	2.0D	Vc, SMM	91	91	91	91	91
					Fz, MMPT	0.03556	0.0508	0.07366	0.07874	0.0889
					n, RPM	4580	3060	2290	1830	1530
K	15-16	Cast Iron - Gray; Ferritic / Pearlitic, Pearlitic 180 - 260 HB	0.07D	2.0D	Vc, SMM	168	168	168	168	168
					Fz, MMPT	0.0508	0.06858	0.09398	0.1143	0.13208
					n, RPM	8400	5600	4200	3360	2800
	17-18	Ductile Iron - Nodular Graphite; Ferritic, Pearlitic 160 - 250 HB	0.07D	2.0D	Vc, SMM	168	168	168	168	168
					Fz, MMPT	0.0508	0.06858	0.09398	0.1143	0.13208
					n, RPM	8400	5600	4200	3360	2800
	19-20	Cast Iron - Malleable; Ferritic, Pearlitic 130 - 230 HB	0.07D	2.0D	Vc, SMM	168	168	168	168	168
					Fz, MMPT	0.0508	0.06858	0.09398	0.1143	0.13208
					n, RPM	8400	5600	4200	3360	2800
					Vf, MMPM	2134	1930	1981	1930	1854



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<sub>e</sub> 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:

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



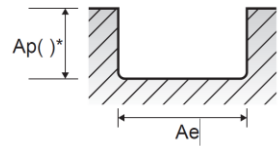


# Speeds and Feeds

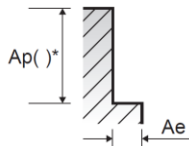


- 1) Select your material in the ISO colored chart with respect to material description.
- 2) Start with a middle/average value for spindle speed,  $n$  (RPM) and feed rate,  $V_f$  (mm/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

Material			Recommended Cutting Values - Peel Cutting							End Mill Series - HTM
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	0.04D	2.0D	$V_c$ , SMM	37	37	37	37	37
					$F_z$ , MMPT	0.0254	0.03556	0.04826	0.05334	0.05842
					$n$ , RPM	1830	1220	920	730	610
					$V_f$ , MMPM	229	229	229	203	178
	36-37	Titanium Alloys	0.05D	2.0D	$V_c$ , SMM	91	91	91	91	91
					$F_z$ , MMPT	0.0254	0.03556	0.04826	0.05334	0.05842
					$n$ , RPM	4580	3060	2290	1830	1530
					$V_f$ , MMPM	584	533	559	483	457



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 <sup>3</sup> /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 <sup>3</sup> /min)
$a_e$	Radial depth of cut	mm
$a_p$	Axial depth of cut	mm
$Z$	Number of teeth/flutes	