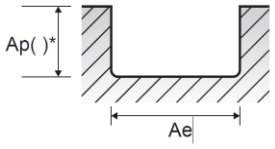
Speeds and Feeds

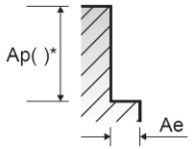
1) Select your material in the ISO colored chart with respect to material description.
2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (in/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTPM

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	3/8	1/2	5/8	3/4
P	1-4	Steel - Non-Alloy, Cast, & Free Cutting 125 - 270 HB	1.0D	0.8D	Vc, SFM	500	550	550	550	550
					Fz, IPT	0.0006	0.0015	0.0019	0.0021	0.0026
					n, RPM	7620	5620	4210	3370	2800
					Vf, IPM	19	34	31	28	29
	5	Steel - Non-Alloy, Cast, & Free Cutting 300 HB	1.0D	0.8D	Vc, SFM	350	385	385	385	385
					Fz, IPT	0.0006	0.0015	0.0019	0.0021	0.0026
					n, RPM	5360	3910	2930	2350	1960
					Vf, IPM	14	23	22	20	20
	6-7	Steel - Low alloy & cast 180 - 275 HB	1.0D	0.8D	Vc, SFM	500	550	550	550	550
					Fz, IPT	0.0006	0.0015	0.0019	0.0021	0.0026
					n, RPM	7620	5620	4210	3370	2800
					Vf, IPM	19	34	31	28	29
	8-9	Steel - Low alloy & cast 300 - 350 HB	1.0D	0.8D	Vc, SFM	350	385	385	385	385
					Fz, IPT	0.0006	0.0015	0.0019	0.0021	0.0026
					n, RPM	5360	3910	2930	2350	1960
					Vf, IPM	14	23	22	20	20
	10	Steel - High Alloy,Cast, & Tool 200 HB	1.0D	0.8D	Vc, SFM	210	230	230	230	230
					Fz, IPT	0.0004	0.0011	0.0013	0.0015	0.0018
					n, RPM	3210	2340	1760	1410	1170
					Vf, IPM	6	10	9	8	8
	11.1	Steel - Bainitic Ultra-High-Carbon 260 - 480 HB	1.0D	0.8D	Vc, SFM	210	230	230	230	230
					Fz, IPT	0.0004	0.0011	0.0013	0.0015	0.0018
					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.

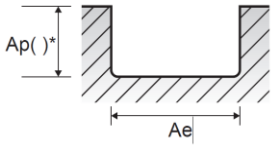
Tech Tips: *The tables above are based on common machining calculators.*
We realize that shops may not have the RPM capability shown in the tables.
To adapt the tables to the machining conditions available, use the following calculation:
(Recommended Feed IPM / Recommended RPM) X Available RPM = IPM

Speeds and Feeds

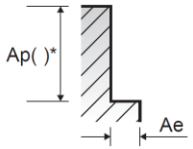
1) Select your material in the ISO colored chart with respect to material description.
2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (in/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTPM

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	3/8	1/2	5/8	3/4
M	12-13	Stainless Steel, Ferritic/Martensitic, Hardened and Tempered, 200 - 240 HB	1.0D	0.8D	Vc, SFM	485	485	485	485	485
					Fz, IPT	0.0005	0.0013	0.0015	0.0018	0.0022
					n, RPM	7430	4950	3710	2970	2480
					Vf, IPM	15	27	23	21	21
	14.1	Stainless Steel, Austenitic, Precipitation Hardened 250 HB	1.0D	0.8D	Vc, SFM	350	350	350	350	350
					Fz, IPT	0.0007	0.0019	0.0022	0.0024	0.0030
					n, RPM	5317	3545	2659	2127	1772
					Vf, IPM	15	27	23	21	21
	14.2	Stainless Steel, Austenitic-Ferritic, Solution Annealed 250 HB	1.0D	0.8D	Vc, SFM	310	310	310	310	310
					Fz, IPT	0.0007	0.0019	0.0022	0.0024	0.0030
					n, RPM	4767	3178	2384	1907	1589
					Vf, IPM	14	24	21	19	19
K	15-16	Cast Iron - Gray; Ferritic / Pearlitic, Pearlitic 180 -260 HB	1.0D	0.8D	Vc, SFM	365	405	405	405	405
					Fz, IPT	0.0008	0.0019	0.0023	0.0026	0.0032
					n, RPM	5608	4115	3087	2469	2058
					Vf, IPM	18	31	28	25	26
	17-18	Ductile Iron - Nodular Graphite; Ferritic, Pearlitic 160 - 250 HB	1.0D	0.8D	Vc, SFM	365	405	405	405	405
					Fz, IPT	0.0008	0.0019	0.0023	0.0026	0.0032
					n, RPM	5608	4115	3087	2469	2058
					Vf, IPM	18	31	28	25	26
	19-20	Cast Iron - Malleable; Ferritic, Pearlitic 130 - 230 HB	1.0D	0.8D	Vc, SFM	365	405	405	405	405
					Fz, IPT	0.0008	0.0019	0.0023	0.0026	0.0032
					n, RPM	5608	4115	3087	2469	2058
					Vf, IPM	18	31	28	25	26



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.

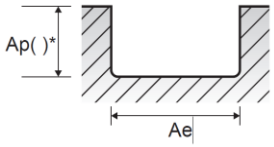
Tech Tips: *The tables above are based on common machining calculators.*
We realize that shops may not have the RPM capability shown in the tables.
To adapt the tables to the machining conditions available, use the following calculation:
(Recommended Feed IPM / Recommended RPM) X Available RPM = IPM

Speeds and Feeds

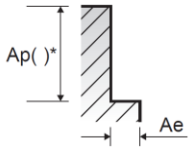
1) Select your material in the ISO colored chart with respect to material description.
2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (in/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTPM

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	3/8	1/2	5/8	3/4
S	31-35	Heat Resistant Super Alloys 200 HB - 350 HB	1.0D	0.5D	Vc, SFM	85	85	85	85	85
					Fz, IPT	0.0005	0.0013	0.0015	0.0017	0.0021
					n, RPM	1299	866	649	520	433
					Vf, IPM	2	5	4	4	4
	36-37	Titanium Alloys	1.0D	0.5D	Vc, SFM	190	190	190	190	190
					Fz, IPT	0.0006	0.0017	0.0020	0.0022	0.0027
					n, RPM	2903	1935	1452	1161	968
					Vf, IPM	7	13	11	10	10



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.

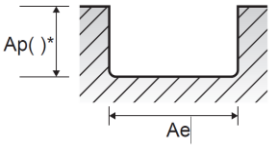
Tech Tips: *The tables above are based on common machining calculators.*
We realize that shops may not have the RPM capability shown in the tables.
To adapt the tables to the machining conditions available, use the following calculation:
(Recommended Feed IPM / Recommended RPM) X Available RPM = IPM

Speeds and Feeds

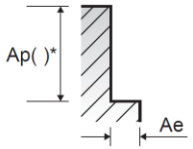
1) Select your material in the ISO colored chart with respect to material description.
2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (in/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTPM

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	3/8	1/2	5/8	3/4
P	1-4	Steel - Non-Alloy, Cast, & Free Cutting 125 - 270 HB	0.5D	1.2D	Vc, SFM	500	550	550	550	550
					Fz, IPT	0.0006	0.0015	0.0019	0.0021	0.0026
					n, RPM	7620	5620	4210	3370	2800
					Vf, IPM	19	34	31	28	29
	5	Steel - Non-Alloy, Cast, & Free Cutting 300 HB	0.5D	1.2D	Vc, SFM	350	385	385	385	385
					Fz, IPT	0.0006	0.0015	0.0019	0.0021	0.0026
					n, RPM	5360	3910	2930	2350	1960
					Vf, IPM	14	23	22	20	20
	6-7	Steel - Low alloy & cast 180 - 275 HB	0.5D	1.2D	Vc, SFM	500	550	550	550	550
					Fz, IPT	0.0006	0.0015	0.0019	0.0021	0.0026
					n, RPM	7620	5620	4210	3370	2800
					Vf, IPM	19	34	31	28	29
	8-9	Steel - Low alloy & cast 300 - 350 HB	0.5D	1.2D	Vc, SFM	350	385	385	385	385
					Fz, IPT	0.0006	0.0015	0.0019	0.0021	0.0026
					n, RPM	5360	3910	2930	2350	1960
					Vf, IPM	14	23	22	20	20
	10	Steel - High Alloy,Cast, & Tool 200 HB	0.5D	1.2D	Vc, SFM	210	230	230	230	230
					Fz, IPT	0.0004	0.0011	0.0013	0.0015	0.0018
					n, RPM	3210	2340	1760	1410	1170
					Vf, IPM	6	10	9	8	8
	11.1	Steel - Bainitic Ultra-High-Carbon 260 - 480 HB	0.5D	1.2D	Vc, SFM	210	230	230	230	230
					Fz, IPT	0.0004	0.0011	0.0013	0.0015	0.0018
					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.

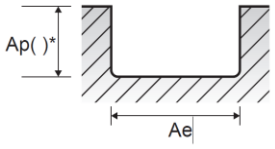
Tech Tips: *The tables above are based on common machining calculators.*
We realize that shops may not have the RPM capability shown in the tables.
To adapt the tables to the machining conditions available, use the following calculation:
(Recommended Feed IPM / Recommended RPM) X Available RPM = IPM

Speeds and Feeds

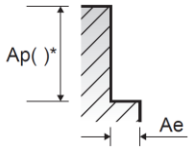
1) Select your material in the ISO colored chart with respect to material description.
2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (in/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTPM

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	3/8	1/2	5/8	3/4
M	12-13	Stainless Steel, Ferritic/Martensitic, Hardened and Tempered, 200 - 240 HB	0.5D	1.2D	Vc, SFM	485	485	485	485	485
					Fz, IPT	0.0005	0.0013	0.0015	0.0018	0.0022
					n, RPM	7430	4950	3710	2970	2480
					Vf, IPM	15	27	23	21	21
	14.1	Stainless Steel, Austenitic, Precipitation Hardened 250 HB	0.5D	1.2D	Vc, SFM	350	350	350	350	350
					Fz, IPT	0.0007	0.0019	0.0022	0.0024	0.0030
					n, RPM	5317	3545	2659	2127	1772
					Vf, IPM	15	27	23	21	21
	14.2	Stainless Steel, Austenitic-Ferritic, Solution Annealed 250 HB	0.5D	1.2D	Vc, SFM	310	310	310	310	310
					Fz, IPT	0.0007	0.0019	0.0022	0.0024	0.0030
					n, RPM	4767	3178	2384	1907	1589
					Vf, IPM	14	24	21	19	19
K	15-16	Cast Iron - Gray; Ferritic / Pearlitic, Pearlitic 180 -260 HB	0.5D	1.2D	Vc, SFM	365	405	405	405	405
					Fz, IPT	0.0008	0.0019	0.0023	0.0026	0.0032
					n, RPM	5608	4115	3087	2469	2058
					Vf, IPM	18	31	28	25	26
	17-18	Ductile Iron - Nodular Graphite; Ferritic, Pearlitic 160 - 250 HB	0.5D	1.2D	Vc, SFM	365	405	405	405	405
					Fz, IPT	0.0008	0.0019	0.0023	0.0026	0.0032
					n, RPM	5608	4115	3087	2469	2058
					Vf, IPM	18	31	28	25	26
	19-20	Cast Iron - Malleable; Ferritic, Pearlitic 130 - 230 HB	0.5D	1.2D	Vc, SFM	365	405	405	405	405
					Fz, IPT	0.0008	0.0019	0.0023	0.0026	0.0032
					n, RPM	5608	4115	3087	2469	2058
					Vf, IPM	18	31	28	25	26



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.

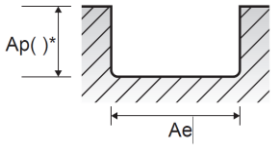
Tech Tips: *The tables above are based on common machining calculators.*
We realize that shops may not have the RPM capability shown in the tables.
To adapt the tables to the machining conditions available, use the following calculation:
(Recommended Feed IPM / Recommended RPM) X Available RPM = IPM

Speeds and Feeds

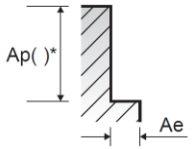
1) Select your material in the ISO colored chart with respect to material description.
2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (in/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTPM

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	3/8	1/2	5/8	3/4
S	31-35	Heat Resistant Super Alloys 200 HB - 350 HB	0.25D	1.0D	Vc, SFM	85	85	85	85	85
					Fz, IPT	0.0005	0.0013	0.0015	0.0017	0.0021
					n, RPM	1299	866	649	520	433
					Vf, IPM	2	5	4	4	4
	36-37	Titanium Alloys	0.35D	1.0D	Vc, SFM	190	190	190	190	190
					Fz, IPT	0.0006	0.0017	0.0020	0.0022	0.0027
					n, RPM	2903	1935	1452	1161	968
					Vf, IPM	7	13	11	10	10



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 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 (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)*
n	Spindle speed	*rev/min (RPM)*
D_{tool}	Tool cutting diameter	*in*
MRR	Material removal rate	*(in^3/min)*
a_e	Radial depth of cut	*in*
a_p	Axial depth of cut	*in*
Z	Number of teeth/flutes	