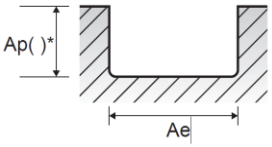
Speeds and Feeds

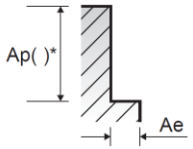
1) Select your material in the ISO colored chart with respect to material description.
2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (in/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTM

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	1-5	Steel - Non-Alloy, Cast, & Free Cutting 125 - 270 HB	1.0D	1.0D	Vc, SFM	275	275	275	275	275
					Fz, IPT	0.0013	0.0018	0.0024	0.0029	0.0034
					n, RPM	4200	2800	2100	1680	1400
					Vf, IPM	27	25	25	24	24
	6-8	Steel - Low alloy & cast 180 - 275 HB	1.0D	1.0D	Vc, SFM	275	275	275	275	275
					Fz, IPT	0.0013	0.0018	0.0024	0.0029	0.0034
					n, RPM	4200	2800	2100	1680	1400
					Vf, IPM	27	25	25	24	24
	9	Steel - Low alloy & cast 300 - 350 HB	1.0D	1.0D	Vc, SFM	275	275	275	275	275
					Fz, IPT	0.001	0.0014	0.0019	0.0023	0.0026
					n, RPM	4200	2800	2100	1680	1400
					Vf, IPM	21	20	20	19	18
	10	Steel - High Alloy,Cast, & Tool 200 HB	1.0D	0.75D	Vc, SFM	230	230	230	230	230
					Fz, IPT	0.0013	0.0018	0.0024	0.0029	0.0034
					n, RPM	3510	2340	1760	1410	1170
					Vf, IPM	23	21	21	20	20
	11.1	Steel - Bainitic Ultra-High-Carbon 260 - 480 HB	1.0D	0.75D	Vc, SFM	250	250	250	250	250
					Fz, IPT	0.001	0.0014	0.0019	0.0023	0.0026
					n, RPM	3820	2550	1910	1530	1270
					Vf, IPM	19	18	18	18	17



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.

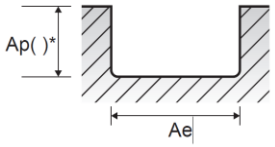
Tech Tips: *The tables above are based on common machining calculators.*
We realize that shops may not have the RPM capability shown in the tables.
To adapt the tables to the machining conditions available, use the following calculation:
(Recommended Feed IPM / Recommended RPM) X Available RPM = IPM

Speeds and Feeds

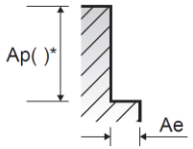
1) Select your material in the ISO colored chart with respect to material description.
2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (in/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTM

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	0.5D	Vc, SFM	225	225	225	225	225
					Fz, IPT	0.0008	0.0012	0.0017	0.0019	0.0022
					n, RPM	3440	2290	1720	1380	1150
					Vf, IPM	14	14	15	13	13
	14.1	Stainless Steel, Austenitic, Precipitation Hardened 250 HB	1.0D	0.5D	Vc, SFM	250	250	250	250	250
					Fz, IPT	0.0010	0.0014	0.0021	0.0023	0.0025
					n, RPM	3820	2550	1910	1530	1270
					Vf, IPM	19	18	20	18	16
	14.2	Stainless Steel, Austenitic-Ferritic, Solution Annealed 250 HB	1.0D	0.5D	Vc, SFM	200	200	200	200	200
					Fz, IPT	0.0008	0.0011	0.0017	0.0018	0.0020
					n, RPM	3060	2040	1530	1220	1020
					Vf, IPM	12	11	13	11	10
K	15-16	Cast Iron - Gray; Ferritic / Pearlitic, Pearlitic 180 -260 HB	1.0D	1.0D	Vc, SFM	260	260	260	260	260
					Fz, IPT	0.0011	0.0015	0.0021	0.0026	0.0030
					n, RPM	3970	2650	1990	1590	1320
					Vf, IPM	22	20	21	21	20
	17-18	Ductile Iron - Nodular Graphite; Ferritic, Pearlitic 160 - 250 HB	1.0D	1.0D	Vc, SFM	260	260	260	260	260
					Fz, IPT	0.0011	0.0015	0.0021	0.0026	0.0030
					n, RPM	3970	2650	1990	1590	1320
					Vf, IPM	22	20	21	21	20
	19-20	Cast Iron - Malleable; Ferritic, Pearlitic 130 - 230 HB	1.0D	1.0D	Vc, SFM	260	260	260	260	260
					Fz, IPT	0.0011	0.0015	0.0021	0.0026	0.0030
					n, RPM	3970	2650	1990	1590	1320
					Vf, IPM	22	20	21	21	20



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.

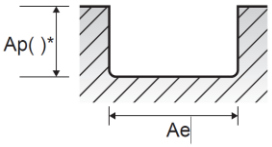
Tech Tips: *The tables above are based on common machining calculators.*
We realize that shops may not have the RPM capability shown in the tables.
To adapt the tables to the machining conditions available, use the following calculation:
(Recommended Feed IPM / Recommended RPM) X Available RPM = IPM

Speeds and Feeds

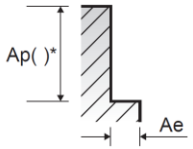
1) Select your material in the ISO colored chart with respect to material description.
2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (in/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTM

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	0.3D	Vc, SFM	64	64	64	64	64
					Fz, IPT	0.0008	0.0011	0.0015	0.0017	0.0019
					n, RPM	980	650	490	390	330
					Vf, IPM	4	4	4	3	3
	36-37	Titanium Alloys	1.0D	0.5D	Vc, SFM	160	160	160	160	160
					Fz, IPT	0.0008	0.0011	0.0015	0.0017	0.0019
					n, RPM	2440	1630	1220	980	810
					Vf, IPM	10	9	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.

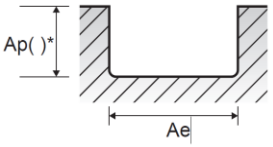
Tech Tips: *The tables above are based on common machining calculators.*
We realize that shops may not have the RPM capability shown in the tables.
To adapt the tables to the machining conditions available, use the following calculation:
(Recommended Feed IPM / Recommended RPM) X Available RPM = IPM

Speeds and Feeds

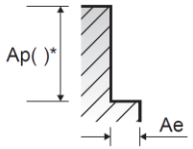
1) Select your material in the ISO colored chart with respect to material description.
2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (in/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTM

Material			Recommended Cutting Values – Heavy 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	1-5	Steel - Non-Alloy, Cast, & Free Cutting 125 - 270 HB	0.5D	1.5D	Vc, SFM	500	500	500	500	500
					Fz, IPT	0.0016	0.0022	0.0031	0.0037	0.0043
					n, RPM	7640	5090	3820	3060	2550
					Vf, IPM	61	56	59	57	55
	6-8	Steel - Low alloy & cast 180 - 275 HB	0.5D	1.5D	Vc, SFM	500	500	500	500	500
					Fz, IPT	0.0016	0.0022	0.0031	0.0037	0.0043
					n, RPM	7640	5090	3820	3060	2550
					Vf, IPM	61	56	59	57	55
	9	Steel - Low alloy & cast 300 - 350 HB	0.5D	1.5D	Vc, SFM	400	400	400	400	400
					Fz, IPT	0.0012	0.0017	0.0024	0.0028	0.0033
					n, RPM	6110	4070	3060	2440	2040
					Vf, IPM	37	35	37	34	34
	10	Steel - High Alloy,Cast, & Tool 200 HB	0.5D	1.5D	Vc, SFM	450	450	450	450	450
					Fz, IPT	0.0016	0.0022	0.0031	0.0037	0.0043
					n, RPM	6880	4580	3440	2750	2290
					Vf, IPM	55	50	53	51	49
	11.1	Steel - Bainitic Ultra-High-Carbon 260 - 480 HB	0.5D	1.5D	Vc, SFM	400	400	400	400	400
					Fz, IPT	0.0012	0.0017	0.0024	0.0028	0.0033
					n, RPM	6110	4070	3060	2440	2040
					Vf, IPM	37	35	37	34	34



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.

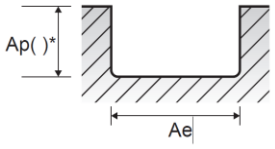
Tech Tips: *The tables above are based on common machining calculators.*
We realize that shops may not have the RPM capability shown in the tables.
To adapt the tables to the machining conditions available, use the following calculation:
(Recommended Feed IPM / Recommended RPM) X Available RPM = IPM

Speeds and Feeds

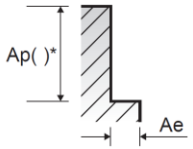
1) Select your material in the ISO colored chart with respect to material description.
2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (in/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTM

Material			Recommended Cutting Values – Heavy 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	1.5D	Vc, SFM	250	250	250	250	250
					Fz, IPT	0.0010	0.0015	0.0021	0.0024	0.0028
					n, RPM	3820	2550	1910	1530	1270
					Vf, IPM	19	19	20	18	18
	14.1	Stainless Steel, Austenitic, Precipitation Hardened 250 HB	0.5D	1.5D	Vc, SFM	300	300	300	300	300
					Fz, IPT	0.0013	0.0018	0.0026	0.0028	0.0031
					n, RPM	4580	3060	2290	1830	1530
					Vf, IPM	30	28	30	26	24
	14.2	Stainless Steel, Austenitic-Ferritic, Solution Annealed 250 HB	0.5D	1.5D	Vc, SFM	200	200	200	200	200
					Fz, IPT	0.0010	0.0014	0.0021	0.0022	0.0025
					n, RPM	3060	2040	1530	1220	1020
					Vf, IPM	15	14	16	13	13
K	15-16	Cast Iron - Gray; Ferritic / Pearlitic, Pearlitic 180 -260 HB	0.5D	1.5D	Vc, SFM	370	370	370	370	370
					Fz, IPT	0.0014	0.0019	0.0026	0.0032	0.0037
					n, RPM	5650	3770	2830	2260	1880
					Vf, IPM	40	36	37	36	35
	17-18	Ductile Iron - Nodular Graphite; Ferritic, Pearlitic 160 - 250 HB	0.5D	1.5D	Vc, SFM	370	370	370	370	370
					Fz, IPT	0.0014	0.0019	0.0026	0.0032	0.0037
					n, RPM	5650	3770	2830	2260	1880
					Vf, IPM	40	36	37	36	35
	19-20	Cast Iron - Malleable; Ferritic, Pearlitic 130 - 230 HB	0.5D	1.5D	Vc, SFM	370	370	370	370	370
					Fz, IPT	0.0014	0.0019	0.0026	0.0032	0.0037
					n, RPM	5650	3770	2830	2260	1880
					Vf, IPM	40	36	37	36	35



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.

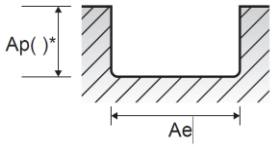
Tech Tips: *The tables above are based on common machining calculators.*
We realize that shops may not have the RPM capability shown in the tables.
To adapt the tables to the machining conditions available, use the following calculation:
(Recommended Feed IPM / Recommended RPM) X Available RPM = IPM

Speeds and Feeds

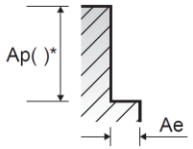
1) Select your material in the ISO colored chart with respect to material description.
2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (in/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTM

Material			Recommended Cutting Values – Heavy 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	0.2D	1.5D	Vc, SFM	90	90	90	90	90
					Fz, IPT	0.0010	0.0014	0.0019	0.0021	0.0023
					n, RPM	1380	920	690	550	460
					Vf, IPM	7	6	7	6	5
	36-37	Titanium Alloys	0.5D	1.5D	Vc, SFM	160	160	160	160	160
					Fz, IPT	0.001	0.0014	0.0019	0.0021	0.0023
					n, RPM	2440	1630	1220	980	810
					Vf, IPM	12	11	12	10	9



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.

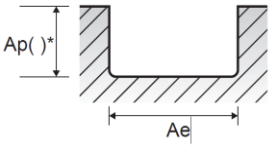
Tech Tips: *The tables above are based on common machining calculators.*
We realize that shops may not have the RPM capability shown in the tables.
To adapt the tables to the machining conditions available, use the following calculation:
(Recommended Feed IPM / Recommended RPM) X Available RPM = IPM

Speeds and Feeds

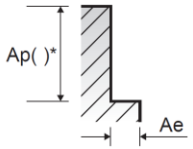
1) Select your material in the ISO colored chart with respect to material description.
2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (in/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTM

Material			Recommended Cutting Values – Peel Milling							
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	1-5	Steel - Non-Alloy, Cast, & Free Cutting 125 - 270 HB	0.08D	2.0D	Vc, SFM	650	650	650	650	650
					Fz, IPT	0.0022	0.0031	0.0043	0.0051	0.006
					n, RPM	9930	6620	4970	3970	3310
					Vf, IPM	109	103	107	101	99
	6-8	Steel - Low alloy & cast 180 - 275 HB	0.08D	2.0D	Vc, SFM	650	650	650	650	650
					Fz, IPT	0.0022	0.0031	0.0043	0.0051	0.006
					n, RPM	9930	6620	4970	3970	3310
					Vf, IPM	109	103	107	101	99
	9	Steel - Low alloy & cast 300 - 350 HB	0.08D	2.0D	Vc, SFM	650	650	650	650	650
					Fz, IPT	0.0017	0.0024	0.0033	0.004	0.0046
					n, RPM	9930	6620	4970	3970	3310
					Vf, IPM	84	79	82	79	76
	10	Steel - High Alloy,Cast, & Tool 200 HB	0.08D	2.0D	Vc, SFM	580	580	580	580	580
					Fz, IPT	0.0022	0.0031	0.0043	0.0051	0.006
					n, RPM	8860	5910	4430	3540	2950
					Vf, IPM	97	92	95	90	89
	11.1	Steel - Bainitic Ultra-High-Carbon 260 - 480 HB	0.08D	2.0D	Vc, SFM	550	550	550	550	550
					Fz, IPT	0.0017	0.0024	0.0033	0.004	0.0046
					n, RPM	8400	5600	4200	3360	2800
					Vf, IPM	71	67	69	67	64



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.

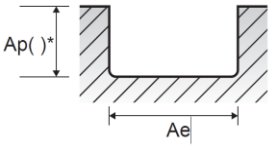
Tech Tips: *The tables above are based on common machining calculators.*
We realize that shops may not have the RPM capability shown in the tables.
To adapt the tables to the machining conditions available, use the following calculation:
(Recommended Feed IPM / Recommended RPM) X Available RPM = IPM

Speeds and Feeds

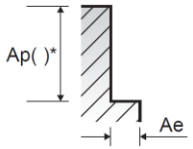
1) Select your material in the ISO colored chart with respect to material description.
2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (in/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTM

Material			Recommended Cutting Values – Peel Milling							
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	0.06D	2.0D	Vc, SFM	350	350	350	350	350
					Fz, IPT	0.0015	0.0021	0.0029	0.0034	0.0039
					n, RPM	5350	3570	2670	2140	1780
					Vf, IPM	40	37	39	36	35
	14.1	Stainless Steel, Austenitic, Precipitation Hardened 250 HB	0.06D	2.0D	Vc, SFM	425	425	425	425	425
					Fz, IPT	0.0018	0.0025	0.0036	0.0039	0.0044
					n, RPM	6490	4330	3250	2600	2160
					Vf, IPM	58	54	59	51	48
	14.2	Stainless Steel, Austenitic-Ferritic, Solution Annealed 250 HB	0.06D	2.0D	Vc, SFM	300	300	300	300	300
					Fz, IPT	0.0014	0.0020	0.0029	0.0031	0.0035
					n, RPM	4580	3060	2290	1830	1530
					Vf, IPM	32	31	33	28	27
K	15-16	Cast Iron - Gray; Ferritic / Pearlitic, Pearlitic 180 -260 HB	0.07D	2.0D	Vc, SFM	550	550	550	550	550
					Fz, IPT	0.0020	0.0027	0.0037	0.0045	0.0052
					n, RPM	8400	5600	4200	3360	2800
					Vf, IPM	84	76	78	76	73
	17-18	Ductile Iron - Nodular Graphite; Ferritic, Pearlitic 160 - 250 HB	0.07D	2.0D	Vc, SFM	550	550	550	550	550
					Fz, IPT	0.0020	0.0027	0.0037	0.0045	0.0052
					n, RPM	8400	5600	4200	3360	2800
					Vf, IPM	84	76	78	76	73
	19-20	Cast Iron - Malleable; Ferritic, Pearlitic 130 - 230 HB	0.07D	2.0D	Vc, SFM	550	550	550	550	550
					Fz, IPT	0.0020	0.0027	0.0037	0.0045	0.0052
					n, RPM	8400	5600	4200	3360	2800
					Vf, IPM	84	76	78	76	73



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.

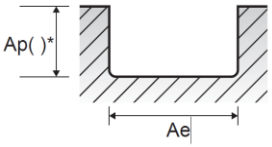
Tech Tips: *The tables above are based on common machining calculators.*
We realize that shops may not have the RPM capability shown in the tables.
To adapt the tables to the machining conditions available, use the following calculation:
(Recommended Feed IPM / Recommended RPM) X Available RPM = IPM

Speeds and Feeds

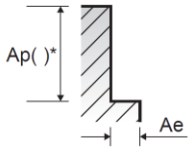
1) Select your material in the ISO colored chart with respect to material description.
2) Start with a middle/average value for spindle speed, n (RPM) and feed rate, V_f (in/min). Adjust the spindle speed and/or feed rate based on your cutting conditions.

End Mill Series – HTM

Material			Recommended Cutting Values – Peel Milling							
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	0.04D	2.0D	Vc, SFM	120	120	120	120	120
					Fz, IPT	0.0010	0.0014	0.0019	0.0021	0.0023
					n, RPM	1830	1220	920	730	610
					Vf, IPM	9	9	9	8	7
	36-37	Titanium Alloys	0.05D	2.0D	Vc, SFM	300	300	300	300	300
					Fz, IPT	0.001	0.0014	0.0019	0.0021	0.0023
					n, RPM	4580	3060	2290	1830	1530
					Vf, IPM	23	21	22	19	18



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	