

End Mill Orada a

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.

		Material					Recommended Cutting Val	ues – Slotting			
G	oup							Tool Diameter (in)			
ISO	VDI 3323	Material Description	Width of Cut, a _e	Depth of Cut, ap	Parameter	1/4	3/8	1/2	5/8	3/4	
					Vc, SFM	275	275	275	275	275	
	4.5	Steel - Non-Alloy, Cast, & Free	1.00	1.00	Fz, IPT	0.0013	0.0018	0.0024	0.0029	0.0034	
	1-5	125 - 270 HB	1.00	1.00	n, RPM	4200	2800	2100	1680	1400	
		120 210 112			Vf, IPM	27	25	25	24	24	
					Vc, SFM	275	275	275	275	275	
	6.0	Steel - Low alloy & cast	1.00	1.00	Fz, IPT	0.0013	0.0018	0.0024	0.0029	0.0034	
	0-0	180 - 275 HB	1.00	1.0D	n, RPM	4200	2800	2100	1680	1400	
					Vf, IPM	27	25	25	24	24	
		Steel - Low alloy & cast 300 - 350 HB	1.0D	1.0D	Vc, SFM	275	275	275	275	275	
D	•				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	
					Vc, SFM	230	230	230	230	230	
	10	Steel - High Alloy,Cast, & Tool	1.00	0.75D	Fz, IPT	0.0013	0.0018	0.0024	0.0029	0.0034	
	10	200 HB	1.00	0.750	n, RPM	3510	2340	1760	1410	1170	
					Vf, IPM	23	21	21	20	20	
					Vc, SFM	250	250	250	250	250	
	11 1_	Steel - Bainitic Ultra-High-Carbon	1.0D	0.75D	Fz, IPT	0.001	0.0014	0.0019	0.0023	0.0026	
		260 - 480 HB	1.00	0.750	n, RPM	3820	2550	1910	1530	1270	
		200 - 400 NB			Vf, IPM	19	18	18	18	17	



Slotting

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





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 Val	ues – Slotting		
G	roup							Tool Diameter (in)		
ISO	VDI 3323	Material Description	Width of Cut, a _e	Depth of Cut, ap	Parameter	1/4	3/8	1/2	5/8	3/4
		Stainless Steel,			Vc, SFM	225	225	225	225	225
	10.10	Ferritic/Martensitic, Hardened and Tempered,	1.00	0.5D	Fz, IPT	0.0008	0.0012	0.0017	0.0019	0.0022
			1.00	0.50	n, RPM	3440	2290	1720	1380	1150
		200 - 240 HB			Vf, IPM	14	14	15	13	13
					Vc, SFM	250	250	250	250	250
м	14 1	Stainless Steel, Austenitic, Procipitation Hardoned	1.00	0.5D	Fz, IPT	0.0010	0.0014	0.0021	0.0023	0.0025
141		250 HB	1.00	0.50	n, RPM	3820	2550	1910	1530	1270
					Vf, IPM	19	18	20	18	16
		Stainless Steel, Austenitic-Ferritic,	1.0D	0.5D	Vc, SFM	200	200	200	200	200
	14.2				Fz, IPT	0.0008	0.0011	0.0017	0.0018	0.0020
		250 HB	1.00		n, RPM	3060	2040	1530	1220	1020
					Vf, IPM	12	11	13	11	10
		Oracle have a Oracle	1.0D	1.0D	Vc, SFM	260	260	260	260	260
	15-16	Cast Iron - Gray; Ferritic / Pearlitic Pearlitic			Fz, IPT	0.0011	0.0015	0.0021	0.0026	0.0030
	13-10	180 -260 HB			n, RPM	3970	2650	1990	1590	1320
					Vf, IPM	22	20	21	21	20
		Dustile kan Nedular Craphiter			Vc, SFM	260	260	260	260	260
к	17-18	Eerritic Pearlitic	1.0D	1.0D	Fz, IPT	0.0011	0.0015	0.0021	0.0026	0.0030
	17-18	160 - 250 HB	1.00	1.00	n, RPM	3970	2650	1990	1590	1320
					Vf, IPM	22	20	21	21	20
		Cost Iron Mollophia:			Vc, SFM	260	260	260	260	260
	19-20	Eerritic Pearlitic	1.0D	1.0D	Fz, IPT	0.0011	0.0015	0.0021	0.0026	0.0030
	10-20	130 - 230 HB	1.00	1.00	n, RPM	3970	2650	1990	1590	1320
		130 - 230 HB			Vf, IPM	22	20	21	21	20





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 – HTM	
		Material				Recommended Cutting Values – Slotting					
G	iroup							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	
	31-35 Heat Resist 200 H			0.3D	Vc, SFM	64	64	64	64	64	
		Heat Resistant Super Alloys 200 HB - 350 HB	1.0D		Fz, IPT	0.0008	0.0011	0.0015	0.0017	0.0019	
					n, RPM	980	650	490	390	330	
e					Vf, IPM	4	4	4	3	3	
3					Vc, SFM	160	160	160	160	160	
	26.27		1.00	0.50	Fz, IPT	0.0008	0.0011	0.0015	0.0017	0.0019	
	30-37	litanium Alloys	1.00	0.5D	n, RPM	2440	1630	1220	980	810	
					Vf, IPM	10	9	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: (Recommended Feed IPM / Recommended RPM) X Available RPM = IPM





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 - HIM
		Material					Recommended Cutting Values -	Heavy Side Cutting		
G	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
					Vc, SFM	500	500	500	500	500
	1.5	Steel - Non-Alloy, Cast, & Free	0.50	1 50	Fz, IPT	0.0016	0.0022	0.0031	0.0037	0.0043
	1-5	125 - 270 HB	0.50	1.50	n, RPM	7640	5090	3820	3060	2550
					Vf, IPM	61	56	59	57	55
					Vc, SFM	500	500	500	500	500
	c 0	Steel - Low alloy & cast	0.50	1 50	Fz, IPT	0.0016	0.0022	0.0031	0.0037	0.0043
	0-0	180 - 275 HB	0.5D	1.50	n, RPM	7640	5090	3820	3060	2550
					Vf, IPM	61	56	59	57	55
		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
					Vc, SFM	450	450	450	450	450
	10	Steel - High Alloy,Cast, & Tool	0.5D	150	Fz, IPT	0.0016	0.0022	0.0031	0.0037	0.0043
	10	200 HB	0.50	1.50	n, RPM	6880	4580	3440	2750	2290
					Vf, IPM	55	50	53	51	49
					Vc, SFM	400	400	400	400	400
	11.1	Steel - Bainitic Ultra-High-Carbon	0.5D	150	Fz, IPT	0.0012	0.0017	0.0024	0.0028	0.0033
		260 - 480 HB	0.50	1.50 -	n, RPM	6110	4070	3060	2440	2040
					Vf, IPM	37	35	37	34	34



Slotting

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

Ae





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 – HIM	
		Material					Recommended Cutting Values -	Heavy Side Cutting		
Gi	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,		1.50	Vc, SFM	250	250	250	250	250
	10.10	Ferritic/Martensitic, Hardened and	0.50		Fz, IPT	0.0010	0.0015	0.0021	0.0024	0.0028
	12-13	Tempered,	0.50	1.50	n, RPM	3820	2550	1910	1530	1270
		200 - 240 HB			Vf, IPM	19	19	20	18	18
					Vc, SFM	300	300	300	300	300
м	14.1	Stainless Steel, Austenitic,	0.50	150	Fz, IPT	0.0013	0.0018	0.0026	0.0028	0.0031
IVI	196.1	250 HB	0.50	1.50	n, RPM	4580	3060	2290	1830	1530
		200112			Vf, IPM	30	28	30	26	24
	Stainless Steel, Austenitic-Ferritic	Otainlana Otaal Avatanitia Familia	0.5D	1.5D	Vc, SFM	200	200	200	200	200
	14.2	Stainless Steel, Austenitic-Ferritic, Solution Annealed 250 HB			Fz, IPT	0.0010	0.0014	0.0021	0.0022	0.0025
	14.2				n, RPM	3060	2040	1530	1220	1020
					Vf, IPM	15	14	16	13	13
		Cost Iron Crow	0.50	1.5D	Vc, SFM	370	370	370	370	370
	15-16	Cast Iron - Gray; Ferritic / Pearlitic Pearlitic			Fz, IPT	0.0014	0.0019	0.0026	0.0032	0.0037
	13-10	180 -260 HB	0.50		n, RPM	5650	3770	2830	2260	1880
					Vf, IPM	40	36	37	36	35
		Dustile Iron Nedular Cranhiter			Vc, SFM	370	370	370	370	370
к	17-18	Eerritic Pearlitic	0.5D	1 5D	Fz, IPT	0.0014	0.0019	0.0026	0.0032	0.0037
		160 - 250 HB	0155	1.00	n, RPM	5650	3770	2830	2260	1880
					Vf, IPM	40	36	37	36	35
		Cast Iron - Malloable:			Vc, SFM	370	370	370	370	370
	19-20	Ferritic, Pearlitic	0.5D	1.5D	Fz, IPT	0.0014	0.0019	0.0026	0.0032	0.0037
		130 - 230 HB	0.00	1.00	n, RPM	5650	3770	2830	2260	1880
		130 - 230 HB			Vf, IPM	40	36	37	36	35



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





- 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 - HIM				
		Material												
G	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				
	31-35			1.5D					Vc, SFM	90	90	90	90	90
		Heat Resistant Super Alloys 200 HB - 350 HB	0.2D		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				
3					Vc, SFM	160	160	160	160	160				
	26.27		0.50	1 50	Fz, IPT	0.001	0.0014	0.0019	0.0021	0.0023				
	30-37	Titanium Alloys	0.5D	1.50	n, RPM	2440	1630	1220	980	810				
					Vf, IPM	12	11	12	10	9				





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





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 – HIM
		Material					Recommended Cutting Value	s – Peel Milling		
G	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
					Vc, SFM	650	650	650	650	650
	1.5	Steel - Non-Alloy, Cast, & Free	0.09D	2.00	Fz, IPT	0.0022	0.0031	0.0043	0.0051	0.006
	1-0	125 - 270 HB	0.06D	2.00	n, RPM	9930	6620	4970	3970	3310
		120 210112			Vf, IPM	109	103	107	101	99
					Vc, SFM	650	650	650	650	650
	6.0	Steel - Low alloy & cast	0.09D	2.00	Fz, IPT	0.0022	0.0031	0.0043	0.0051	0.006
	0-0	180 - 275 HB	0.06D	2.00	n, RPM	9930	6620	4970	3970	3310
					Vf, IPM	109	103	107	101	99
		Steel - Low alloy & cast 300 - 350 HB	0.08D	2.0D	Vc, SFM	650	650	650	650	650
Б	0				Fz, IPT	0.0017	0.0024	0.0033	0.004	0.0046
	3				n, RPM	9930	6620	4970	3970	3310
					Vf, IPM	84	79	82	79	76
					Vc, SFM	580	580	580	580	580
	10	Steel - High Alloy,Cast, & Tool	0.09D	2.00	Fz, IPT	0.0022	0.0031	0.0043	0.0051	0.006
	10	200 HB	0.080	2.00	n, RPM	8860	5910	4430	3540	2950
					Vf, IPM	97	92	95	90	89
					Vc, SFM	550	550	550	550	550
	11.1	Steel - Bainitic Ultra-High-Carbon	0.09D	2.05	Fz, IPT	0.0017	0.0024	0.0033	0.004	0.0046
		260 - 480 HB	0.060	2.00	n, RPM	8400	5600	4200	3360	2800
		200 - 400 115			Vf, IPM	71	67	69	67	64



Slotting

More: All cut Maximum rec Finish cuts tvr

Ae

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





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

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. 2)

										End WIII Series - HIM
		Material					Recommended Cutting Value	es – Peel Milling		
G	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	350	350	350	350	350
	12-12	Ferritic/Martensitic, Hardened and	0.06D	3.05	Fz, IPT	0.0015	0.0021	0.0029	0.0034	0.0039
	12-13	Tempered,	0.00D	2.00	n, RPM	5350	3570	2670	2140	1780
		200 - 240 HB			Vf, IPM	40	37	39	36	35
					Vc, SFM	425	425	425	425	425
м	14.1	Stainless Steel, Austenitic, Precipitation Hardened	0.06D	2 00	Fz, IPT	0.0018	0.0025	0.0036	0.0039	0.0044
		250 HB	0.000	2.00	n, RPM	6490	4330	3250	2600	2160
					Vf, IPM	58	54	59	51	48
		Stainless Steel, Austenitic-Ferritic, Solution Annealed 250 HB	0.06D	2.0D	Vc, SFM	300	300	300	300	300
	14.2				Fz, IPT	0.0014	0.0020	0.0029	0.0031	0.0035
	14.2				n, RPM	4580	3060	2290	1830	1530
					Vf, IPM	32	31	33	28	27
		Cost Iron Crown	0.07D	2.0D	Vc, SFM	550	550	550	550	550
	15-16	Cast Iron - Gray; Ferritic / Pearlitic Pearlitic			Fz, IPT	0.0020	0.0027	0.0037	0.0045	0.0052
	10 10	180 -260 HB			n, RPM	8400	5600	4200	3360	2800
					Vf, IPM	84	76	78	76	73
		Ductile Iron Nedular Cranhite:			Vc, SFM	550	550	550	550	550
к	17-18	Eerritic Pearlitic	0.07D	2.0D	Fz, IPT	0.0020	0.0027	0.0037	0.0045	0.0052
		160 - 250 HB	0.07.5	2.05	n, RPM	8400	5600	4200	3360	2800
					Vf, IPM	84	76	78	76	73
		Cast Iron - Malloable:			Vc, SFM	550	550	550	550	550
	19-20	Ferritic Pearlitic	0.07D	2.0D	Fz, IPT	0.0020	0.0027	0.0037	0.0045	0.0052
		130 - 230 HB	0.072	2.00	n, RPM	8400	5600	4200	3360	2800
	130 - 230 HB			Vf, IPM	84	76	78	76	73	

Side Cutting Slotting



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 – HTM
		Material								
G	roup							Tool Diameter (in)		
ISO	VDI 3323	Material Description	Width of Cut, a _e	e Depth of Cut, a _p	Parameter	1/4	3/8	1/2	5/8	3/4
	31-35 Heat Res 200				Vc, SFM	120	120	120	120	120
		Heat Resistant Super Alloys 200 HB - 350 HB	0.04D	2.0D	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
8					Vc, SFM	300	300	300	300	300
	26.27		0.05D	2.00	Fz, IPT	0.001	0.0014	0.0019	0.0021	0.0023
	30-37	Intanium Alloys	0.05D	2.00	n, RPM	4580	3060	2290	1830	1530
					Vf, IPM	23	21	22	19	18





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 (in³/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