

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.

Material					Recommended Cutting Values – Side Milling				
Group		Material Description	Width of Cut a _e	Depth of Cut a _p	Parameter	Tool Diameter (in)			
ISO	VDI 3323					1/16	5/64	3/32	1/8
N	21-22	Aluminum-Wrought Alloy	0.1D	1.0D	Vc, SFM	900 - 1800	900 - 1800	900 - 1800	900 - 1800
					Fz, IPT	0.00102	0.00129	0.00153	0.00206
					n, RPM	55032 - 110064	44040 - 88079	36668 - 73337	27516 - 55032
					Vf, IPM	168.4 - 336.8	170.43 - 340.87	168.31 - 336.62	170.05 - 340.1
	23-25	Aluminum-Cast Alloy	0.1D	1.0D	Vc, SFM	960 - 1200	960 - 1200	960 - 1200	960 - 1200
					Fz, IPT	0.00102	0.00129	0.00153	0.00206
					n, RPM	58701 - 73376	46976 - 58719	39113 - 48891	29350 - 36688
					Vf, IPM	179.63 - 224.53	181.8 - 227.24	179.53 - 224.41	181.38 - 226.73
	26-28	Copper and Copper Alloys (Bronze/Brass)	0.1D	1.0D	Vc, SFM	270 - 660	270 - 660	270 - 660	270 - 660
					Fz, IPT	0.00082	0.00103	0.00123	0.00165
					n, RPM	16510 - 40357	13212 - 32296	11001 - 26890	8255 - 20178
					Vf, IPM	50.52 - 123.49	51.13 - 124.99	50.49 - 123.43	51.02 - 124.7

NOTE: All cutting data are target values.

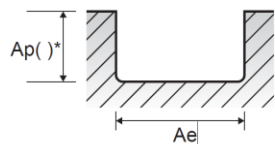
Maximum recommended depth shown.

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.

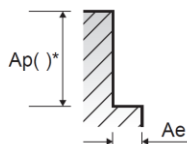
Reduce speed and feed recommendations for materials harder than listed.

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.



Slotting



Side Cutting

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:

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



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.

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					1/16	5/64	3/32	1/8
N	21-22	Aluminum-Wrought Alloy	0.1D	0.4D	Vc, SFM	900 - 1800	900 - 1800	900 - 1800	900 - 1800
					Fz, IPT	0.00102	0.00129	0.00153	0.00206
					n, RPM	55032 - 110064	44040 - 88079	36668 - 73337	27516 - 55032
					Vf, IPM	168.4 - 336.8	170.43 - 340.87	168.31 - 336.62	170.05 - 340.1
	23-25	Aluminum-Cast Alloy	0.1D	0.4D	Vc, SFM	960 - 1200	960 - 1200	960 - 1200	960 - 1200
					Fz, IPT	0.00102	0.00129	0.00153	0.00206
					n, RPM	58701 - 73376	46976 - 58719	39113 - 48891	29350 - 36688
					Vf, IPM	179.63 - 224.53	181.8 - 227.24	179.53 - 224.41	181.38 - 226.73
	26-28	Copper and Copper Alloys (Bronze/Brass)	0.1D	0.4D	Vc, SFM	270 - 660	270 - 660	270 - 660	270 - 660
					Fz, IPT	0.00082	0.00103	0.00123	0.00165
					n, RPM	16510 - 40357	13212 - 32296	11001 - 26890	8255 - 20178
					Vf, IPM	50.52 - 123.49	51.13 - 124.99	50.49 - 123.43	51.02 - 124.7

NOTE: All cutting data are target values.

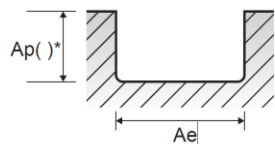
Maximum recommended depth shown.

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.

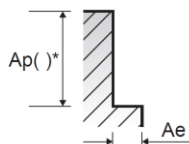
Reduce speed and feed recommendations for materials harder than listed.

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.



Slotting



Side Cutting

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:

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



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	<i>in/min</i>
f_n	Feed per revolution	<i>in/rev</i>
f_z	Feed per tooth	<i>in</i>
v_c	Cutting speed	<i>ft/min (SFM)</i>
n	Spindle speed	<i>rev/min (RPM)</i>
D_{tool}	Tool cutting diameter	<i>in</i>
MMR	Material removal rate	<i>(in³/min)</i>
a_e	Radial depth of cut	<i>in</i>
a_p	Axial depth of cut	<i>in</i>
Z	Number of teeth/flutes	