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 cutting speed, V_c (ft/min) and feed per tooth, f_z (in/tooth). Adjust the cutting speed and/or feed based on your cutting conditions.

Milling Series - HIEX/HPSM

Material			Recommended Cutting Values			
ISO	Material Description	Grade	Vc, SFM	Fz, IPT (Face/Shoulder Milling)	Fz, IPT (Grooving, Ramping, Helical Milling)	Depth of Cut, a_p
P	Steel	HU30	492 - 787	0.002 - 0.012	0.002 - 0.012	0.649 in
		HP25	426 - 688	0.002 - 0.012	0.002 - 0.012	
		MKP30	820 - 1148	0.002 - 0.012	0.002 - 0.012	
		HKP40	492 - 820	0.002 - 0.012	0.002 - 0.012	
M	Stainless Steel	HU30	295 - 492	0.002 - 0.010	0.002 - 0.006	
K	Cast Iron	HU30	393 - 656	0.003 - 0.014	0.003 - 0.008	
		MKP30	656 - 984	0.003 - 0.014	0.003 - 0.008	
		HKP40	492 - 820	0.003 - 0.014	0.003 - 0.008	
N	Aluminum & Non-Ferrous	HN30	1640 - 4920	0.003 - 0.010	0.003 - 0.010	

NOTE: When surface and shoulder milling, the data refers to general cutting conditions and can be adjustable up to 1150 SFM and 0.016 IPT depending on user environment. In deep grooving, set the ap under 0.197 in and use coolant + air."

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	