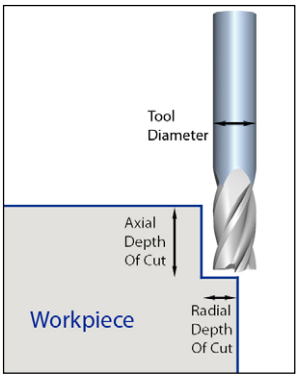
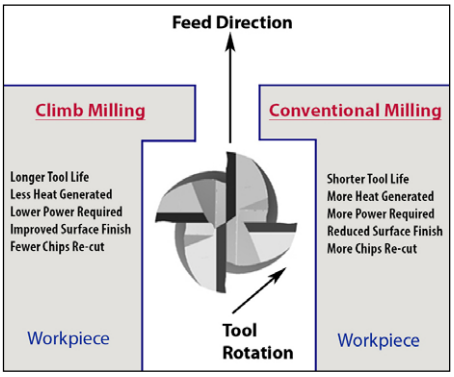
SPEED & FEED RECOMMENDATIONS

GENERAL PURPOSE

Material		Speed (SFM)		Feed Per Tooth By End Mill Diameter							
		Uncoated	TiALN Coated	1/4"	5/16"	3/8"	7/16"	1/2"	5/8"	3/4"	1"
Aluminum & Aluminum Alloys	N	600-1200	900-1800	.0020	.0025	.0030	.0035	.0040	.0050	.0060	.0080
Copper & Copper Alloys	N	350-850	525-1275	.0020	.0025	.0025	.0030	.0030	.0035	.0040	.0060
Brass & Bronze	N	250-400	375-600	.0020	.0025	.0025	.0030	.0030	.0035	.0040	.0050
Graphite	N	500-800	500-1200	.0030	.0035	.0025	.0030	.0030	.0040	.0050	.0070
Plastics	N	600-1100	600-1650	.0030	.0035	.0040	.0050	.0060	.0080	.0100	.0150
Iron, Cast (soft)	K	250-450	375-650	.0020	.0022	.0025	.0027	.0030	.0045	.0060	.0080
Iron, Cast (hard)	K	100-250	100-375	.0008	.0010	.0015	.0017	.0020	.0025	.0030	.0040
Iron, Ductile	K	80-400	100-600	.0010	.0012	.0015	.0017	.0020	.0030	.0040	.0060
Iron, Malleable	K	150-500	225-650	.0010	.0015	.0020	.0025	.0030	.0040	.0050	.0070
Carbon Steels, Low	P	200-400	300-600	.0010	.0015	.0020	.0025	.0030	.0040	.0050	.0070
Carbon Steels, Medium	P	100-250	150-375	.0015	.0016	.0017	.0018	.0020	.0030	.0040	.0050
Carbon Steels Hardened to 35 Rc	P	130-230	130-345	.0010	.0011	.0012	.0013	.0015	.0017	.0020	.0030
Carbon Steels Hardened to 50 Rc	P	70-130	70-160	.0007	.0007	.0008	.0009	.0010	.0015	.0020	.0030
Carbon Steels Hardened to 60 Rc	P	30-70	30-90	.0005	.0006	.0007	.0009	.0010	.0012	.0015	.0020
Steels, Mold	P	200-350	300-525	.0010	.0012	.0015	.0017	.0020	.0025	.0030	.0040
Steels, Tool	P	100-250	150-375	.0010	.0012	.0015	.0017	.0020	.0025	.0030	.0040
Stainless Steels, Soft	M	200-350	300-450	.0010	.0012	.0015	.0012	.0020	.0030	.0040	.0060
Stainless Steels, Hard	M	100-200	150-300	.0005	.0006	.0007	.0008	.0010	.0020	.0030	.0050
Monel & High Nickel Steel	S	75-175	75-200	.0010	.0012	.0015	.0017	.0020	.0025	.0030	.0040
Titanium, Soft	S	125-300	125-375	.0010	.0012	.0015	.0017	.0020	.0030	.0040	.0060
Titanium, Hard	S	50-150	50-175	.0005	.0006	.0007	.0008	.0010	.0015	.0020	.0020
Nickel Based High Temp Alloys	S	50-100	50-125	.0008	.0008	.0009	.0009	.0010	.0012	.0015	.0020

- Higher Feed Per Tooth should be used to start for radial depths of cut less than 25% of the tool diameter. Lower Feed Per Tooth should be used to start for radial depths of cut greater than 25% of the tool diameter.
- The above recommendations are for axial lengths of cut not to exceed 1 times the tool diameter for profiling and .5 times the diameter for full slotting.
- The above parameters are recommended starting points only. If the tool is working well, without vibrations or significant noise, increase the SFM and/or Feed Per Tooth in 5-10% increments.
- Optimum speeds & feeds will depend upon material, setup, machine conditions & tool deflection. Higher or lower parameters may be required to achieve optimum machining conditions.
- For Light Radial Depths of cut, make certain to increase the feed rate to compensate for Radial Chip Thinning Factor (RCTF). Consult a formula or app to calculate.
- Climb Milling is preferred to Conventional Milling

$$RPM = \frac{SFM}{(3.146 * Cutter\ Diam.) / 12}$$

$$IPM = RPM * Feed\ Per\ Tooth * \#\ of\ Teeth$$

(Inches Per Minute)