

END MILL SPEED AND FEED RECOMMENDATIONS

	Material	Speed SFM	Feed (F.P.T.)					
			1/8	1/4	1/2	3/4	1"	
Non Ferrous	Aluminum / Aluminum Alloy	1000-2000	.001	.002	.004	.006	.008	
	Aluminum Alloy +10%	700-1200	.001	.002	.004	.006	.008	
	Brass	300-450	.001	.002	.003	.004	.005	
	Bronze	250-350	.001	.002	.003	.004	.005	
	Copper / Copper Alloys	500-900	.001	.002	.003	.005	.007	
	Composites:	Fiberglass	200-600	.001	.002	.003	.004	.005
		Graphites	200-400	.001	.006	.010	.015	.020
		Graphite / Epoxy	200-400	.001	.002	.003	.004	.005
Magnesium	1000 min	.001	.002	.004	.006	.008		
Stainless Steel	Free Machine	300-400	.0005	.001	.002	.004	.006	
	Work Hardening	150-300	.0005	.0005	.001	.003	.005	
High Temp Alloys	Nickel Base	70-100	.0005	.001	.001	.0015	.002	
Cast Iron	Ductile	250-400	.0005	.0015	.002	.004	.006	
	Gray	350-500	.0005	.002	.004	.006	.008	
	Malleable	200-350	.0005	.002	.004	.006	.008	
Steels	Low Alloy	350-600	.0005	.001	.002	.004	.006	
	Medium Alloy	200-400	.0005	.001	.002	.004	.006	
	High Alloy Mold-Die	175-250	.0005	.001	.002	.004	.006	
	High Strength	75-150	.0005	.0005	.001	.003	.004	
Titanium	Soft	150-300	.0005	.001	.002	.004	.006	
	Hard	50-150	.0005	.0005	.001	.002	.004	

All speed and feed recommendations should be considered only as a starting point. Variations may be needed to achieve optimum results.

General Recommendations

The above suggested parameters are recommended for uncoated tools only. When various coatings are applied, SFM may be increased accordingly. These general percentages are as follows under optimal conditions:

TiN = +15% • TiCN = +25% • TiAlN = +40%

Rigidity

Maximize rigidity to reduce chatter and increase tool life. Ways to improve rigidity include, choosing the largest diameter possible to perform your milling task, use the shortest LOC (Length of Cut) available, and always use the tool holder which offers the shortest gage line (Shortest Tool Holder Length).

Chatter

If chatter is present increase feed or reduce speed.

Extra Long End Mills

For extra long end mills the SFM should be reduced by 25%

Cutter Direction

Cutter direction should be in a climb milling direction whenever possible. Conventional tool paths may be employed when the tool is in use on older manual equipment to minimize backlash or to remove scale to reach the softer parent material underneath.

TIR or Total Indicator Runout

This condition should be held to a minimum at all times. As this value increases, the effectiveness of the tools performance decreases thus reducing tool life.

Recutting Chips

Use a coolant or air blast to evacuate chips to avoid premature damage to your carbide cutting tool, which may occur if chips are recut.