



Speeds and Feeds for **1011** style end mills. Solid carbide 3 flute fine pitch rougher high helix uncoated. End Mills are made in the USA.

Material	Hardness		Feed Per Tooth					
	Bhn	SFM	1/8"dia	1/4"	3/8"	1/2"	3/4"	1"
Aluminum 2011,2024,6061,7075	--	<b>800</b>	.0013	.0022	.0038	.005	.007	.009
Free Machining 12L14, 1215,11L17	80-150	<b>250</b>	.0006	.0015	.0022	.004	.005	.0055
Low Carbon Steel 1008, 1018, 1020	*	*						
Medium Carbon Steel 4140, 8620	*	*						
Tool Steel, H.S.S A2,D2,M2,M42	*	*						
Cast Iron	120-175	<b>300</b>	.0005	.0014	.002	.0035	.005	.0052
Stainless Steel 300 Series	*	*						
Stainless Steel 400 Series	*	*						
Stainless Steel 15-5, 17-4	*	*						
Titanium 6AL-4V	*	*						
Inconel 625, 718	*	*						
Copper, Bronze	120-160	<b>450</b>	.0009	.0018	.0026	.0045	.006	.0065
Brass (360 half hard)	--	<b>700</b>	.0013	.0022	.0038	.005	.007	.009

\* please use our 4 flute rougher with TiAlN coating series 1010

Feed Per Tooth is based on.....

**slotting** depth of 1.0 x Cutting Diameter, for deeper slotting reduce feed and speed

**profiling** with Axial depth up to 2x Cutting Diameter and Radial Depth of .25 x Cutting Diameter, when profiling less than .25 cutting diameter increase feed and speed

When using long flute length roughers reduce SFM and Feed Per Tooth accordingly. Normally SFM only 10-20%, but Feed Per Tooth should be reduced 30-60%.

#### 6 guidelines for best cutting results

keep overall gage length short	keep tool concentricity low	climb milling is generally best
workpiece rigidity is important	avoid double cutting chips	avoid thermal shock

**Note:** Cutting speeds, feeds, and lubrication data are given as starting values only. Parameters should be varied to suite your particular conditions. MariTool is constantly improving products, tolerances, and reliability. As such, these speeds and feeds are subject to change without prior notice. Cutting tools may shatter and or produce dangerous fumes when being used. Appropriate protection is advised.

Rev B