

# Drill & Cut Catalogue (6061-T651 Aluminum)

Wednesday, August 12, 2015 11:05 AM

## Basal Feeds and Speeds

Applicable to: 2015-001 DragKnife, 2015-002 Laser Mount, 2016-001, and others

**IMPORTANT: Use ONLY approved cuts as defined in the table below. Do not tweak toolpath cut values. Instead, normalize to the approved cuts.**

- All cuts are in 6061-T651 Aluminum plate or bar, 95Bn hardness.
- These are for square-end end mills for the most part, and might need adjustment for all end-mills.
- F&S indicated as:

**End Mills:** @[EA%] ([EAin"]): [FR]; [PR]; [RPM]; [DOC]; [CL | CV]; [3DSO]

**Drills:** @DRILL: [PR]; [RPM]; [DOC]

- EA = Engagement Angle, as a % of the bit diameter (EA%) and inches (EAin) for that bit
- FR = linear feed rate in IPM
- PR = plunge rate in IPM
- RPM = spindle speed in RPM
- DOC = Max intended depth individual cut or drill peck, in inches
- CL = Climb milling (meaningless for drills)
- CV = Conventional milling (default; meaningless for drills)
- 3DSO = 3D-carving step-over, for minimizing scalloping. Optional.
- "NR" = "Not Recommended" on this machine, or with that bit
- LBS = Length Behind Shank, or the diameter-reduced length of the shank behind the flutes.
- When a bit is expected to do multiple cuts, calculate for uniform spindle RPM so in-program speed-control isn't needed.
- **Blue** text in the table below indicates values validated in GW
- **Purple** text in the table below indicates the preferred uniform-RPM cut parameters for the bit.
- The Heavy Roughing column, corresponding to 50-85% radial engagement is not feasible on the CNC shark, so is left off
- For drill-cycle dwell time, units are fractional seconds in BobCAD.

Tool Workflow # & Description	Axial Engagement (DOC) for this cut	HSM (overrides Lt Rough values)	Finish (0-10% of cutter diameter)	Lt Rough (5-25%)	Med Rough (25-50%)	Full Slot/Drill (85-100%)	Comments
<b>2 FL End Mills</b>							
<b>1/16" 2FL Carbide EM</b>	0.005 - 0.010"		NR	@16% (0.010"): 18.7;6.9;24k;0.010 ;CV	NR	NR	Very delicate bit; be very careful with it
<b>1/8" 2FL Carbide EM:</b>  0.50 LOC 0.0 LBS, 0.25 LOC 0.50 LBS, 0.25 LOC 0.75 LBS, 0.25 LOC 1.00 LBS 0.25 LOC 1.25 LBS,	0.005 - 0.040"	0.035 DOC, 0.025 SO CL (20%) 0.020 SO CV 0.022 min RAD	@5% (0.006"): 54;11;24k;0.380; CL  @8% (0.010"): 42.5; 11.5; 24k; 0.380; CL (use coating if available)	@20% (0.025"): 44.4; 17.8 (9.0);24k; 0.030; CL  alternatively...  @20% (0.025"): 28.8; 11.5 (9); 24k; 0.030/0.035; CL	@40% (0.050"): 36.5;18;24k ;0.030;CL	@100% (0.125"): 35;17;24k;0.020; CV	Smaller end-mill; use a newer mill for finish cuts; Medium Rough cut is only for use in top-side profile cuts where 2+ horizontal paths are used.
<b>1/8" 2FL Carbide EM</b>  0.75 LOC, 0.0 LBS	0.005 - 0.020"	0.035 DOC, 0.025 SO CL (20%) 0.020 SO CV 0.022 min RAD	NR	@20% (0.025"): 26.0; 11.5; 24.0k; 0.020; CL	NR	NR	EXPERIMENTAL: Be VERY careful with this bit, if you have any. They shatter very easily with even the slightest horizontal deflection. Be sure they are properly engaged in the chuck insert. Non-HSM toolpaths are not recommended. True slotting of any significant depth is to be avoided.
<b>1/8" 2FL HSS EM</b>  0.75 LOC, 0.0 LBS	0.005-0.040 "			@20% (0.025"): 34.5;13.8 (9); 24k; 0.030; CL			This is completely experimental, trying to get the SFM for the cutter in AL 6061-T6 within the 24K RPM limit for the spindle. The carbide tools use the max RPM, but really to get the 1200 SFM it needs to be more like 33K RPM.
<b>3/16" 2FL Carbide EM</b> 0.312 LOC, 1.50 LBS 0.312 LOC, 2.00 LBS (including radiused)	0.010 - 0.040"	0.035 DOC, 0.030 SO CL (16%) 0.025 SO (13%) CV 0.022 min RAD		@16% SO (0.030"): 43.2 IPM; 15.8 PPM (11.0); 22.0k RPM; 0.030 DOC; CL	NR	@100% SO (0.1875"): 22.9 IPM, 11.4 IPPM (10.0); 22.0k RPM; 0.020 DOC; CV	For medium and light roughing, preferred when minimum-radius permits due to Aluminum SFM of ~1000 can be achieved within the limits of this bit.  This slotting cut, especially with the extended reach bits, is a very difficult cut for current capabilities, so be very careful.

			@5% (0.010") SO 37.1 IPM 8.4 (7.8) IPPM 21.5K RPM 0.1" DOC (GW claims excessive deflection with this?)			@100% SO (0.1875"): 22.32 IPM, 7.9 IPPM (6.0), 22.0K RPM, 0.030 DOC	EXPERIMENTAL
1/4" 2FL Carbide EM  (constant 15.6k RPM for merging across multiple cut types)	0.005 - 0.040"	0.050 DOC, 0.050 SO CL (20%) 0.040 SO CV 0.022 min RAD	@2% (0.005): 80; 15; 15.6k; 0.380; CL  @4% (0.010"): 76.6; 15; 15.6k, 0.380; CL (coat)  @5% (0.0125"): 69; 15; 15.6k; 0.380; CL	@20% (0.050"): 37.44; 15.0 (11.0); 15.6k; 0.040; CL (coat) slider:34% [0.0749mrr, 0.0144hp, 0.000054defl, 0.93 in#, 1021 SFM]  @20% (0.050"): 39.6; 15.8; 16.2k; 0.020; CL (for small pockets)  17.4k@64/25pl (0.050/0.050)	NR	@100% (0.250"): 38 IPM; 15.1 IPPM; 15.6k RPM, 0.030 DOC; CV (yeah right)  @100% (0.250" SO) 21.1 IPM, 10.6 (5) IPPM; 15.6k RPM; 0.020 DOC CV	Medium end-mill; use a newer mill for finish cuts. Between Lt. and Med. Rough, only go with medium if the cut is thin, like 10-20mils. Lt roughing can take deeper cuts, but needs to keep a smaller engagement angle.  As for the slot numbers, the first set are hardly believable, but the bottom seems more in line with a very safe cut.
		HSM experimental 0.150 DOC 0.020 SO 24% slider 16.1K RPM 47.5 IPM 12.9 (11.0) IPPM  0.150 DOC 0.015 SO 24% slider 16.1K RPM 54.3 IPM 12.9 (11.0) IPPM [0.1224mrr, 0.0235hp, 0.000085defl, 1.47 in#, 1054 SFM]  0.150 DOC 0.015/0.012 SO 24% slider 19k RPM 64.2 IPM 15.2 (13.0) IPPM [0.1444mrr, 0.0277hp, 0.000085defl, 1.47 in#, 1244 SFM] (!!!!)				@100% (0.250" SO) 14.8 IPM, 7.4 (5) IPPM, 15.6K RPM; 0.050 DOC, CV	EXPERIMENTAL
Radiused End Mills							
1/8" 2FL 0.015R 0.5DOC 3/16" 2FL 0.020R 0.625DOC 1/4" 2FL 0.020R 0.75DOC 1/4" 2FL 0.015R 0.375 LOC 1.5LBS							F&S calculators seem to indicate that, at least for these small corner radii, the F&S numbers are essentially the same as for a sharp-pointed end mill. Just be sure to input the geometry correctly, and don't use the same tool definition for a square-end and radiused tool.
3+ FL End Mills							
1/8" 3FL Carbide EM	0.005 - 0.010"		@8% (0.010"): 64; 11.5; 24k; 0.380; CL (coat)  @5%(0.006"): 37; 5.3; 24k; 0.380; CL	NR	NR	NR	For finishing passes on profiles and inside geometry.
3/16" 3FL Carbide EM	0.005 - 0.010"		@3.2% (0.006"): 59.5; 6.4; 19.3k; 0.380; CL	@16% (0.30): 64.9; 15.8; 22.0k; 0.030; CL	NR	NR	For finishing passes on profiles. Note that profiles roughed with an 1/8" bit might have areas inaccessible to the 3/16 bit.

			@5.3% (0.010"): 42.6; 6.4; 19.3k; 0.380; CL (coat)	@20% (0.0375"): 25.7; 6.9; 20.8k; 0.30; CL (coat)			
1/4" 4FL Carbide Finisher	Full height (0.375)		@4% (0.010"): 66.37; 6.5; 14.7k; 0.380; CL (coat)  @2% (0.005"): 80; 6.5; 14.8k; 0.380; CL	NR	NR	NR	Only a finisher bit
Ball End Mills							
1/8" 2-FL Ball Nose Carbide EM, 0.25 LOC 1.25 LBS			@8% (0.010"): 39.1; 13.9; 24.0k; 0.010; CL (coat)	@16% (0.020"): 33.6; 13.9; 24.0k; 0.020; CL (coat)	NR	NR	For 3D contour finishing only
1/8" 3-FL Ball Nose Carbide EM, 0.5 LOC 0 LBS			@8% (0.010"): 58.7; 13.9; 24.0k; 0.010; CL (coat)	@16% (0.020"): 50.4; 13.9; 24.0k; 0.020; CL (coat)	NR	NR	For 3D contour finishing only
3/16" 2-FL Ball Nose Carbide EM 0.312 LOC 1.25 LBS			@8% (0.015"): 58.7; 20.8; 24.0k; 0.015; CL (coat)	@16% (0.030"): 41.8; 17.3; 24.0k; 0.030; CL (coat)	NR	NR	For 3D contour finishing only
0.250" 2-FL Ball Nose Carbide EM 0.375LOC 2.0LBS			5.4% (0.010") SO 0.040" DOC (16% slider) 65.8 IPM 14.9 (12) IPPM 22.1k RPM  (4% slider) 47.417 IPM 18.8 (9.0) IPPM 22.1k RPM				13 Mar 2017: somewhat experimental, since we've never used a 2fl 1/4" ball nose
1/4" 3-FL Ball Nose Carbide EM 0.75 LOC 0.0 LBS			@8% (0.020"): 80.0; 23.0; 24.0k; 0.020; CL (coat)	@16% (0.040"): 78.5; 21.6; 22.5k; 0.040; CL (coat) ????? Seems FAST	NR	NR	For 3D contour finishing only
Roll-form Tap Drills							
Op1004 2FL 160deg (#4 roll tap, tight)	DRILL		NR	NR	NR	@DRILL-Op1004D: 11.0; 21.8k; 0.075 (p=0.75D)	#4-40 roll-form tap hole tight fit (2.55mm)
Op1024 2FL 160deg (#4 roll tap, loose)	DRILL		NR	NR	NR	@DRILL-Op1024D: 11.0; 21.4k; 0.076 (p=0.75D)	#4-40 roll-form tap hole loose fit (2.60mm)
Op1240 2FL 160deg (#6 roll tap, tight)	DRILL		NR	NR	NR	@DRILL-Op1240D: 11.0; 17.7k; 0.093 (p=0.75D)	#6-32 roll-form tap hole tight fit (3.15mm)
Op1260 2FL 160deg (#6 roll tap, loose)	DRILL		NR	NR	NR	@DRILL-Op1260D: 11.0; 17.4; 0.094 (p=0.75D)	#6-32 roll-form tap hole loose fit (3.20mm)
Op1496 2FL 160deg (#8 roll tap, tight)	DRILL		NR	NR	NR	@DRILL-Op1496D: 10.9; 14.6k; 0.1122 (p=0.75D)	#8-32 roll-form tap hole right fit (3.80mm)  NOTE: Running this at 50% FRO with 5.0 plunge rate produced almost zero head deflection, and very good holes. Consider lowering the plunge rate further.
Op1520 2FL 160deg (#8 roll tap, loose)	DRILL		NR	NR	NR	@DRILL-Op1520D: 10.9; 14.4k; 0.1140 (p=0.75D)	#8-32 roll-form tap hole loose fit (#24)
Op1713 2FL 160deg (#10 roll tap, tight)	DRILL		NR	NR	NR	@DRILL-Op1713D: 10.4; 12.8k; 0.1285 (p=0.75D)	#10-24 roll-form tap hole tight fit (4.35mm)
Op1732 2FL 160deg (#10 roll tap, loose)	DRILL		NR	NR	NR	@DRILL-Op1732D: 10.5; 12.6k; 0.1299 (p=0.75D)	#10-24 roll-form tap hole loose fit (4.40mm)  None on hand at this time; MSC shows \$22 EACH!, and PreciseBits will not be getting/making more.  This one might be better machined with a normal 0.125 end mill.
Cutting Tap Drills							
Op1130 2FL 160deg (#6 cutting tap)	DRILL		NR	NR	NR	@DRILL-Op1130D: 18.6; 19.3k; 0.085 (p=0.75D)	#6 cutting tap hole
Op1380 2FL 160deg (#8 cutting tap)	DRILL		NR	NR	NR	@DRILL-Op1380D: 11.2; 15.8k; 0.100 (p=0.75D)	#8 cutting tap hole
Op1540 2FL 160deg (#10 cutting tap)	DRILL		NR	NR	NR	@DRILL-Op1540D: 10.8; 14.2k; 0.115	#10 cutting tap hole

						(p=0.75D)	
<b>Clearance Drills</b>							
Op1420 2FL 160deg (#6 clearance hole)	DRILL		NR	NR	NR	@DRILL0p1420D-: 11.1; 15.4k; 0.110 (p=0.75D)	#6 Clearance hole
Op1693 2FL 160deg (#8 clearance hole)	DRILL		NR	NR	NR	@DRILL-0p1693D: 10.6; 13k; 0.127 (p=0.75D)	#8 Clearance hole
Op1935 2FL 160deg (#10 clearance hole)	DRILL		NR	NR	NR	@DRILL-0p1935D: 10.4; 11.3k; 0.145 (p=0.75D)	#10 Clearance hole
<b>Other Drills</b>							
<b>Other Bit Types</b>							
0.75" HSS Face Mill	0.005 - 0.010"		@10% (0.075"):- (fine) 30.0; 9.0 (4.2) 5.6k  @10% (0.075"):- (fine) 13.9; 4.2; 2.6k; 0.002-0.008"; CL  (finish)32;9.6;2.7k;CL	NR	NR	NR	LOL...we actually used 66%? For a FINISH job? Hahaha....no wonder it looked like poopie! Moving from 66% to 10% should allow the FR to be higher, so not exactly 6.6x the time.
60-deg V-bit 1/8" 0.005" tip	0.005"		@0.005": 12.7; 6.3; 24k; 0.005; CV	NR	NR	NR	Text and scale engraving bit; always use 0.005" engraving depth.
1/8" 90-deg 2FL Chamfer	0.020" (L) 0.014" (XYZ)		0.020": 27.7; 13.9; 24k; 0.020; CL	NR	NR	NR	Edge and hole chamfering
1/4" 90-deg 2FL Chamfer	0.020" (L) 0.014" (XYZ)		0.020": 55.5; 27.7;24k;0.020;CL	NR	0.040": 55.5; 27.7;24k;0.020;CL	NR	Edge and hole chamfering; the plunge numbers seem very high.
1/2" 82-deg 2FL Chamfer	Varies		#8 @ 82deg: 0.5752*: 31; 15.4; 16.5k; 0.359d= 0.2065 cut-Z; CL				Countersinking a #8 screw requires a 3/8 or 1/2". Unless BobCAD can spiral it in or something?
1/4" 90-degree spot drill	0.020" (L)		0.020": 5.0, 9.3k, 0.020"  0.020": 5.0, 18.0k	NR	NR	NR	Before drilling to correct diameter. Helps prevent drill bit wandering. However, the machine is so floppy that is moves a LOT on plunge use a slow rate, and a high RPMs. Use dwell when feasible.
0.020R 2FL round over bit Carbide 1/8" shank	0.020" (Radius)		0.020": 27.7; 13.9; 24k; 0.020; CL	NR	NR	NR	Round over bit for touched edges. Values are copied from the 1/8" 90-degree chamfer bit, as there is no such bit profile provided in GWizard.