

# Drill & Cut Catalogue (HDPE & Acrylic)

Wednesday, August 12, 2015 11:05 AM

## Basal Feeds and Speeds

Applicable to: 2016-008 Car Tablet Case and others using HDPE

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

- Cuts are in Hgh-Density Polyethylene, UHMW, nominal hardness, or in Cast Acrylic Lexan/Plexiglass as indicated
  - HDPE does in fact wear carbide and HSS bits. When thin burrs and seeming melted fringes start to appear, it means the bit is just about at the end of its useful sharpness, and should be replaced at the next practical opportunity.
- 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.
- Bold Blue** text in the table below indicates values validated in GW
- Bold Purple** text in the table below indicates the preferred uniform-RPM cut parameters for the bit.
- Peach** background table cells indicate information unmodified from the original 6061 cut catalogue
- 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 Engage-ment (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>1FL End Mills</b>							
1A: 1/4" 1FL HSS 0.7LOC 1B: 1/4" 1FL HSS 1.0LOC				@16% (0.030"SO): 33.1, 21.5(10),14.3k; 0.075"DOC; CL		@100% (0.250"SO): 21.1; 21(10), 13.9k; 0.075"DOC; CL	Supposedly VERY safe numbers, owing to the softness of the plastic. Need to watch for melting. Hopefully the relief on these O-Flute bits will prevent material heating, unlike the 2-flute end mills that actually have a wiper after the cutting edge.  0.130" DOC even worked with no seeming problem.  When working in hdpe, realize that "melting" is also a sign of a dulling bit making chips that are too small....
1C: 1/8" 1FL HSS 0.625LOC				@16% (0.020"SO): 30.7, 22.5(1.0),28.7k; 0.030"DOC; CL  Experimental long: @16% (0.020"SO): 30.0, 21.9(10), 27.9k; 0.0725"DOC, CL		@100% (0.125"SO): 22.5, 22.5 (10), 28.7k; 0.030"DOC; CL	...what he said
1D: 1/16" 1FL CARB 0.25LOC 1/8"SH						@100% (0.0625"SO): ??? ?	...what he said
1E: 3/16" 1FL HSS 0.75LOC 1F: 3/16" 1FL HSS				@16% (0.025"SO): 30.7, 20.9 (10.0), 19.1k; 0.050"DOC;		@100% (0.1875"SO): 20.7, 20.7(10),	...what he said

0.625LOC				CL		18.9k; 0.050"DOC; CL	
2 FL End Mills							
1/16" 2FL Carbide EM	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
1/8" 2FL Carbide EM:  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.
1/8" 2FL Carbide EM  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.
1/8" 2FL HSS EM  0.75 LOC, 0.0 LBS	0.005-0.040"		NR	@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.
3/16" 2FL Carbide EM 0.312 LOC, 1.50 LBS 0.312 LOC, 2.00 LBS	0.010 - 0.040"	0.035 DOC, 0.030 SO CL (16%) 0.025 SO (13%) CV 0.022 min RAD	NR	@16% (0.030): 43.2; 15.8 (11.0); 22.0k; 0.030; CL	NR	NR	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.
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  17.4k@64/25pl (0.050/0.050)	@20% (0.050"): 37.44; 15.0 (11.0); 15.6k; 0.040; CL (coat)  @20% (0.050"): 39.6; 15.8; 16.2k; 0.020; CL (for small pockets)	NR	@100% (0.250"): 38;15.1;15.6k;0. 030;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.
Radiused End Mills							
1/8" 2FL 0.015R 0.5DOC 3/16" 2FL 0.020R 0.625DOC 1/4" 2FL 0.020R 0.75DOC							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  @5.3% (0.010"): 42.6; 6.4; 19.3k; 0.380; CL (coat)	@16% (0.30): 64.9; 15.8; 22.0k; 0.030; CL  @20% (0.0375"): 25.7; 6.9; 20.8k; 0.30; CL (coat)	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.
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"):	NR	NR	NR	Only a finisher bit

			80; 6.5; 14.8k; 0.380; CL				
<b>Ball End Mills</b>							
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
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)	NR	NR	For 3D contour finishing only
<b>Roll-form Tap Drills</b>							
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)
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.
<b>Cutting Tap Drills</b>							
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 (p=0.75D)	#10 cutting tap hole
<b>Clearance Drills</b>							
Op1420 2FL 160deg (#6 clearance hole)	DRILL		NR	NR	NR	@DRILL-Op1420D: 11.1; 15.4k; 0.110 (p=0.75D)	#6 Clearance hole
Op1693 2FL 160deg (#8 clearance hole)	DRILL		NR	NR	NR	@DRILL-Op1693D: 10.6; 13k; 0.127 (p=0.75D)	#8 Clearance hole
Op1935 2FL 160deg (#10 clearance hole)	DRILL		NR	NR	NR	@DRILL-Op1935D: 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.