ENGINEERED SYNTACTIC SYSTEMS

PLEASE – BEFORE YOU TRY IT YOUR WAY, TRY IT OURS!

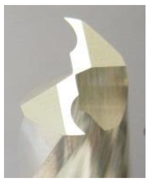
ESS-TOOL™ Tool Board Machining Guide

Follow the guidelines below to minimize dust, maximize tool life and optimize surface finish when machining ESS-TOOL.

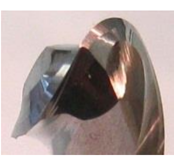
Cutter Type	• Solid Carbide. (Various coatings are available to further extend tool life.) • **2 Flute, Plastic Cutting Tools** • **SHARP TOOLS** are required. Syntactic foams are abrasive. Check cutting edges and monitor tool board surface for evidence of dull tooling.
Speed and Feed	• Syntactic foam is unique with no similarity to machining metal or other items you may have experienced. Use of cutting tools and feed/speed rates for metal will result in dust and poor surface quality. • **"Chip Load"** (the measurement of thickness of material removed by each cutting edge during a cut) provides a method to calculate specific feed/speed rates. For the recommended tool styles shown on page 2, use Chip Load as shown for the appropriate size tool, plug in your CNC spindle speed and calculate feed rate using the formula: ***Feed Rate = Chip Load x Spindle RPM x 2***
Climb vs Conventional Milling	• While conventional and climb milling table feed direction both provide good results with the tool types shown in our guide, climb milling is often preferred to extend tool life. • With climb milling, chip width starts at maximum and decreases, transferring heat to the chip and creating a cleaner shear plane resulting in less tool rub and increasing tool life. Chips are removed behind the cutter, reducing the chance of re-cutting.
Optimization techniques	**1.** Experiment with the maximum possible chip size. Use feed rate as determined from the chip load rating and your machine RPM. **2.** Increase feed rate until the part finish begins to deteriorate. Decrease feed rate 10%. **3.** Decrease RPM by some set increment until surface finish begins to deteriorate. Once this happens, increase RPM until finish is again acceptable. Speed and feed are now optimized in your process. **4.** Usage of separate tools for roughing and finishing allows rotation of finish tool into roughing position when part finish deteriorates. **5.** Clear removed chips to prevent premature tool wear. **NOTE: Too low a feed rate will generate excess heat and reduce tool life. Proper settings will result in a tool operating at or near room temperature. Too high a feed rate will cause poor surface finish or part movement during machining.**
Coolant	• **None required/air. Chips/dust generated must be cleared from the tool area. Recutting chips will quickly dull a tool and may create a fire hazard.**
Protection	• Enclose chip space, dust extraction, safety goggles, dust mask, protective gloves

ESS-TOOL™ Tool Board Machining Guide

Double Flute Upcut Spiral - Solid Carbide: High helix geometry with a special point for upward chip flow, smooth sidewall and improved bottom finish.

Examples shown are Onsrud Tool 52-7xx series	Cutting Diameter (D)	Roughing Parameters RDOC[1] = 100% ADOC[2] = up to 1xD	Finishing Parameters: Walls RDOC[1] = see below ADOC[2] = up to 4xD		Finishing Parameters: Floors RDOC[1] = 40 – 65% ADOC[2] = see below	
		Chip load	Chip load	RDOC[1]	Chip load	ADOC[2]
 	> 3/8"	.010 - .020	.005	.05	.005	.02

High Finish Ball Nose – Solid Carbide: 3D contouring of ESS-TOOL materials. Unique geometry and highly polished surface result in a smooth surface without tool marks.

Examples shown are Onsrud Tool 65-2xxB series	Cutting Diameter (D)	Roughing Parameters RDOC[1] = 33% of D ADOC[2] = up to 2xD	Finishing Parameters		
		Chip load	Chip load	RDOC[1]	ADOC[2]
 	>1/4"	.005 - .007	.003	.002-.003	.01

[1]RDOC: Radial Depth of Cut – the depth of the tool along its radius in the work piece as it makes its cut.
[2]ADOC: Axial Depth of Cut – the depth of the tool along its axis in the work piece as it makes its cut.

For quick reference, use the chart below to determine a good starting point for machining of ESS-TOOL tool board.

- For similar tool designs to those shown, select the chip load based on the tool size and requirement.
- Look across the top of the chart for the spindle speed typically used in your machining center.
- Select the Feed Rate (inches/minute) for your program from the corresponding bold number below.

	Spindle Speed							
	2500	5000	7500	10000	12500	15000	17500	20000
Chip Load	Feed Rate							
0.005	***25***	***50***	***75***	***100***	***125***	***150***	***175***	***200***
0.010	***50***	***100***	***150***	***200***	***250***	***300***	***350***	***400***
0.015	***75***	***150***	***225***	***300***	***375***	***450***	***525***	***600***
0.020	***100***	***200***	***300***	***400***	***500***	***600***	***700***	***800***