

Metric Performance Series' by ULTRATOOL



Series 355 Ultra-Grain® Carbide End Mill
Five Flute • 45 Degree RH Spiral
with Square End or Corner Radius



SFR 355 Series Specs (Metric):
Cutting Diam +0.000/-0.051mm
Shank Diam +0.000/-0.007mm

ULTRATOOL
PERFORMANCE
SERVICES
Metric Sizes

Diam	LOC	OAL	Shank	Square#	Radius#	Rad	AT Coated
3.0mm	12	38	3.0	31570AT	31580AT	0.2	\$12.90
4.0mm	14	50	4.0	31571AT	31581AT	0.2	\$16.90
5.0mm	16	50	5.0	31572AT	31582AT	0.3	\$18.60
6.0mm	19	50	6.0	31573AT	31583AT	0.3	\$21.90
8.0mm	20	63	8.0	31574AT	31584AT	0.3	\$27.90
10.0mm	22	70	10.0	31575AT	31585AT	0.5	\$42.60
12.0mm	25	74	12.0	31576AT	31586AT	0.5	\$54.00
16.0mm	38	89	16.0	31577AT	31587AT	0.5	\$101.80
20.0mm	38	100	20.0	31578AT	31588AT	0.5	\$194.30
25.0mm	38	100	25.0	31579AT	31589AT	0.5	\$208.00



Series 323 Patented SmoothFlute®
Four Flute Variable Helix End Mill
with standard Corner Radius



SFR 323 Series End Mill Specs (Metric):
Cutting Diam +0.000/-0.051mm
Shank Diam +0.000/-0.007mm

Diam	LOC	OAL	Shank	Radius	EDP#	AT Coated
3.0mm	12	38	3.0	0.2	23080AT	\$12.90
4.0mm	14	50	4.0	0.2	23081AT	\$16.90
5.0mm	16	50	5.0	0.3	23082AT	\$18.60
6.0mm	19	50	6.0	0.3	23083AT	\$21.90
8.0mm	20	63	8.0	0.3	23084AT	\$27.90
10.0mm	22	70	10.0	0.5	23085AT	\$42.60
12.0mm	16	74	12.0	0.5	23086AT	\$51.70
12.0mm	30	74	12.0	0.5	23087AT	\$54.00
14.0mm	32	89	14.0	0.5	23088AT	\$88.80
16.0mm	38	89	16.0	0.5	23089AT	\$101.80
20.0mm	38	100	20.0	0.5	23090AT	\$194.30
25.0mm	38	100	25.0	0.5	23091AT	\$208.00



Series 365 Patented SmoothFlute®
Six Flute Variable Helix End Mill
with standard Corner Radius



SFR 365 Series End Mill Specs (Metric):
Cutting Diam +0.000/-0.051mm
Shank Diam +0.000/-0.007mm

Diam	LOC	OAL	Shank	Radius	EDP#	AT Coated
6.0mm	10	50	6.0	0.25	27720AT	\$21.90
6.0mm	19	63	6.0	0.5	27721AT	\$25.70
8.0mm	20	63	8.0	0.5	27722AT	\$30.40
10.0mm	16	70	10.0	0.5	27723AT	\$39.70
10.0mm	22	70	10.0	1.0	27724AT	\$42.60
12.0mm	16	74	12.0	0.5	27725AT	\$52.70
12.0mm	30	74	12.0	1.0	27726AT	\$60.70
12.0mm	45	100	12.0	2.0	27727AT	\$80.80
16.0mm	38	89	16.0	0.5	27728AT	\$101.80
16.0mm	57	125	16.0	1.0	27729AT	\$149.00
20.0mm	38	100	20.0	1.0	27730AT	\$194.30
25.0mm	38	100	25.0	1.0	27731AT	\$208.00

new!

new!

TC UC A1
Choose from UnCoated (UC), TiCN (TC), or TiB2 (A1) surface hardcoatings. Simply add the appropriate suffix, UC, TC, or A1, after the EDP#. SmoothEdge is included!

Series 330AL Aluminum End Mill
Two Flute • 45 Degree RH Spiral



UC TC A1 SE

Diam	LOC	OAL	Shank	0.5mm		1.0mm		3.0mm		UnCoated	TC or A1 Coated
				Square EDP#	Rad EDP#	Square EDP#	Rad EDP#	Square EDP#	Rad EDP#		
3.0mm	6	38	3.0	11504	11701					\$9.20	\$11.50
3.0mm	12	38	3.0	11525	11702					\$9.30	\$11.70
4.0mm	9	50	4.0	11507	11703					\$11.10	\$14.20
4.0mm	12	50	4.0	11528	11704					\$11.40	\$14.50
5.0mm	9	50	5.0	11510	11705					\$13.40	\$17.10
5.0mm	14	50	5.0	11531	11706					\$13.80	\$17.40
6.0mm	12	63	6.0	11513	11707	11720				\$19.30	\$24.90
6.0mm	25	63	6.0	39760	11708	11721				\$21.60	\$27.20
6.0mm	50	100	6.0	39761	11709	11722				\$36.30	\$43.70
8.0mm	14	63	8.0	11516	11710	11723				\$25.20	\$32.60
8.0mm	25	63	8.0	39716	11711	11724				\$26.10	\$33.50
8.0mm	50	100	8.0	39762	11712	11725				\$43.10	\$52.40
10.0mm	16	70	10.0	11519	11713	11726				\$37.00	\$45.70
10.0mm	30	70	10.0	39717	11714	11727				\$38.60	\$47.30
10.0mm	50	100	10.0	39763	11715	11728				\$54.40	\$67.90
12.0mm	19	74	12.0	11522	11716	11729				\$42.20	\$52.80
12.0mm	32	74	12.0	39718	11717	11730				\$44.80	\$55.30
12.0mm	50	100	12.0	39764	11718	11731				\$66.60	\$80.80
14.0mm	44	89	14.0	39765	11719	11732				\$74.60	\$89.40
16.0mm	44	89	16.0	39719		11733	11740			\$84.30	\$101.00
16.0mm	64	125	16.0	39766		11734	11741			\$130.40	\$156.30
18.0mm	50	100	18.0	39767		11735	11742			\$129.20	\$147.40
20.0mm	50	100	20.0	39720		11736	11743			\$142.80	\$162.50
20.0mm	76	150	20.0	39768		11737	11744			\$229.60	\$263.60
25.0mm	50	100	25.0	39721		11738	11745			\$188.30	\$216.10
25.0mm	76	150	25.0	39769		11739	11746			\$291.80	\$331.90

Series 333AL Aluminum End Mill
Three Flute • 45 Degree RH Spiral



UC TC A1 SE

Diam	LOC	OAL	Shank	0.5mm		1.0mm		3.0mm		UnCoated	TC or A1 Coated
				Square EDP#	Rad EDP#	Square EDP#	Rad EDP#	Square EDP#	Rad EDP#		
3.0mm	6	38	3.0	11568	11594					\$10.20	\$12.50
3.0mm	12	38	3.0	11569	11595					\$10.30	\$12.60
4.0mm	9	50	4.0	11570	11596					\$12.30	\$15.40
4.0mm	12	50	4.0	11571	11597					\$12.70	\$15.80
5.0mm	9	50	5.0	11572	11598					\$14.60	\$18.30
5.0mm	14	50	5.0	11573	11599					\$15.10	\$18.80
6.0mm	12	63	6.0	11574	11600	11613				\$21.30	\$26.80
6.0mm	25	63	6.0	11575	11601	11614				\$23.70	\$29.30
6.0mm	50	100	6.0	11576	11602	11615				\$39.90	\$47.30
8.0mm	14	63	8.0	11577	11603	11616				\$27.80	\$35.20
8.0mm	25	63	8.0	11578	11604	11617				\$28.80	\$36.10
8.0mm	50	100	8.0	11579	11605	11618				\$47.50	\$57.60
10.0mm	16	70	10.0	11580	11606	11619				\$40.70	\$49.30
10.0mm	30	70	10.0	11581	11607	11620				\$42.60	\$51.20
10.0mm	50	100	10.0	11582	11608	11621				\$59.80	\$73.40
12.0mm	19	74	12.0	11583	11609	11622				\$46.50	\$57.00
12.0mm	32	74	12.0	11584	11610	11623				\$49.30	\$59.80
12.0mm	50	100	12.0	11585	11611	11624				\$73.10	\$87.30
14.0mm	44	89	14.0	11586	11612	11625				\$81.40	\$96.80
16.0mm	44	89	16.0	11587		11626	11633			\$92.90	\$109.50
16.0mm	64	125	16.0	11588		11627	11634			\$143.40	\$169.40
18.0mm	50	100	18.0	11589		11628	11635			\$142.40	\$160.90
20.0mm	50	100	20.0	11590		11629	11636			\$157.20	\$176.80
20.0mm	76	150	20.0	11591		11630	11637			\$252.60	\$286.50
25.0mm	50	100	25.0	11592		11631	11638			\$207.20	\$235.00
25.0mm	76	150	25.0	11593		11632	11639			\$321.20	\$361.40

SFR 330AL Series Specs (Metric):
Cutting Diam +0.000/-0.007mm
Shank Diam +0.000/-0.007mm

SFR 333AL Series Specs (Metric):
Cutting Diam +0.000/-0.007mm
Shank Diam +0.000/-0.007mm

Application Data for High Performance Series 323, 355, 377, & 365 Series ULTRATOOL End Mills

The milling data presented below is for the 323, 355, 377, and 365 Series of Ultra end mills. When using SmoothCoat & SmoothEdge surface treatments, Surface Feet or Meters Per Minute can be increased from the stated levels by at least 25%.

Do not use a radial DOC exceeding more than 25% of diameter for Series 355 only.



Peripheral Milling data based on axial depth ≤ 100% of tool diameter & radial depth of ≤ 25% of tool diameter.



Slot Milling data based on axial depth of cut = 50% of tool diameter.

End Mill Specifications:

Diameter: +.000 / -.002
Shank Diameter: +.0000 / -.0003
LOC: +.060 / -.000
OAL: ± .060
Helix: ± 2°

Milling;
Fractional

Material	SFPM	SFPM	1/8"	3/16"	1/4"	5/16"	3/8"	7/16"	1/2"	5/8"	3/4"	1"
Steel	Peripheral	Slotting	Feed Per Tooth (FPT)									
1018 / 1020	300 to 600	200 to 400	.0007	.0012	.0015	.0018	.0020	.0025	.0030	.0035	.0040	.0045
4140 / 4340 / P20	250 to 500	200 to 350	.00065	.0010	.0012	.0015	.0018	.0022	.0025	.0030	.0035	.0040
Stainless Steel												
303 / 304 / 316	250 to 400	200 to 350	.0006	.0008	.0010	.0012	.0018	.0022	.0025	.0030	.0035	.0038
410 / 420 / 440C	200 to 300	150 to 250	.0006	.0008	.0010	.0012	.0018	.0022	.0025	.0030	.0035	.0038
15-5/17-4 ≤ 32HRc	200 to 350	150 to 300	.0006	.0008	.0010	.0012	.0018	.0022	.0025	.0030	.0035	.0038
15-5/17-4 ≥ 32HRc	150 to 250	150 to 250	.0004	.0006	.0008	.0010	.0015	.0020	.0020	.0025	.0030	.0035
Tool Steel												
A2/D2/H13 ≤ 32HRc	200 to 300	150 to 250	.0005	.0008	.0010	.0012	.0018	.0022	.0025	.0030	.0035	.0035
A2/D2/H13 ≥ 32HRc	150 to 250	100 to 200	.0004	.0006	.0008	.0010	.0015	.0020	.0020	.0025	.0030	.0035
Titanium												
6Al-4V	150 to 300	125 to 225	.0005	.0008	.0010	.0010	.0012	.0020	.0025	.0025	.0030	.0040
High Temp Alloys												
Inconel 625	100 to 150	75 to 125	.0005	.0007	.0010	.0012	.0012	.0018	.0020	.0020	.0025	.0030
Inconel 718	70 to 150	50 to 100	.0005	.0007	.0008	.0009	.0012	.0018	.0020	.0020	.0030	.0040
Cast Iron												
Gray Iron ≤ 32HRc	150 to 400	150 to 300	.0005	.0007	.0010	.0012	.0015	.0018	.0020	.0030	.0040	.0045

Application Data for Series 323, 355, 377, and 365 High Performance End Mills (continued); Peel Milling



Recommendations are based upon a radial cut depth of 10% of the end mill's diameter and axial cut depth of 50-85% of the tool's LOC.

Peel milling can be performed wet or dry (with AT coating); please consult technical@toolalliance.com for specific application data.



See it run now!

Scan the Quick Code and watch the Series 365 milling various materials on the Tool Alliance YouTube channel.

Series 323, 355, 377, and 365 Peel Milling

Surface Feet Per Minute (SFPM) and Feed Per Tooth (FPT) recommendations by tool diameter and material:

Material	SFPM	1/8"	3/16"	1/4"	5/16"	3/8"	7/16"	1/2"	5/8"	3/4"	1"
Steel	Peripheral	Feed Per Tooth (FPT)									
1018 / 1020	400 to 600	.001-.003	.001-.004	.0015-.005	.002-.008	.002-.008	.003-.010	.003-.010	.003-.010	.004-.012	.004-.012
4140 / 4340 / P20	350 to 500	.001-.002	.001-.003	.001-.004	.0015-.006	.0015-.006	.002-.007	.002-.007	.002-.007	.0025-.008	.0025-.008
Stainless Steel											
303 / 304 / 316	300 to 500	.001-.002	.001-.003	.0015-.004	.002-.006	.002-.006	.003-.008	.003-.008	.003-.008	.003-.010	.003-.010
410 / 420 / 440C	250 to 400	.001-.002	.001-.003	.0015-.004	.002-.006	.002-.006	.003-.008	.003-.008	.003-.008	.003-.010	.003-.010
15-5/17-4 ≤ 32HRc	300 to 500	.001-.002	.001-.003	.0015-.004	.002-.006	.002-.006	.003-.008	.003-.008	.003-.008	.003-.010	.003-.010
15-5/17-4 ≥ 32HRc	200 to 300	.0005-.002	.0005-.002	.001-.003	.0015-.005	.0015-.005	.002-.006	.002-.006	.002-.006	.003-.008	.003-.008
Tool Steel											
A2/D2/H13 ≤ 32HRc	250 to 350	.001-.002	.001-.003	.0015-.004	.002-.006	.002-.006	.003-.008	.003-.008	.003-.008	.003-.010	.003-.010
A2/D2/H13 ≥ 32HRc	200 to 300	.001-.002	.001-.003	.0015-.004	.002-.006	.002-.006	.003-.008	.003-.008	.003-.008	.003-.010	.003-.010
Titanium											
6Al-4V	250 to 300	.001-.002	.001-.003	.0015-.004	.002-.006	.002-.006	.003-.008	.003-.008	.003-.008	.003-.010	.003-.010
High Temp Alloys											
Inconel 625	125 to 200	.0005-.002	.0005-.002	.001-.003	.0015-.005	.0015-.005	.002-.006	.002-.006	.002-.006	.003-.008	.003-.008
Inconel 718	100 to 150	.0005-.002	.0005-.002	.001-.003	.0015-.005	.0015-.005	.002-.006	.002-.006	.002-.006	.003-.008	.003-.008
Cast Iron											
Gray Iron ≤ 32HRc	250 to 500	.001-.002	.001-.003	.001-.004	.0015-.006	.0015-.006	.002-.007	.002-.007	.002-.007	.0025-.008	.0025-.008

1

ULTRA-Grain®

Components of Guaranteed Quality

COMPONENT #1: Carbide Substrate From being the first Company to introduce MicroGrain carbide to the mass-market round tool industry through the present day, Tool Alliance® has consistently innovated new powder and grade combinations for demanding applications. We recognize that our material is the very first Significant Characteristic. By creating partnerships with a limited number of tungsten powder and cemented-carbide material suppliers, we are able to guarantee that our customers receive precision-tolerance tools ground from only the purest, finest grades available worldwide. The following photographs of Ultra-Carb® 1 and Ultra-Grain® 1 respectively demonstrate the complexity of the compound we commonly refer to as Cemented Carbide. Taken at magnification of 10,000 X through an SEM (Scanning Electron Microscope), the visible grains are tungsten while the cobalt binder appears as dark shadows. The largest tungsten grains appearing in the Ultra-Carb photo are less than one micron in size. Note that these grades are two samples representing more than a dozen different substrates we use throughout our product lines, each having a particular application niche. Compared to other industry participants, you will find that Tool Alliance offers the best month-to-month and year-to-year consistency in carbide grain structure.



Ultra-Carb® 1
Cobalt Percentage: 6%
Grain Size (µm): ≤ 0.8
Hardness: 93.5 HRA
Fracture Toughness (K1c): 6.6
TRS (GPa): 3.8
Density (gm/cc): 14.90



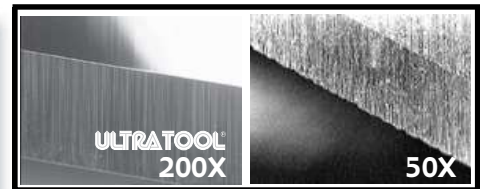
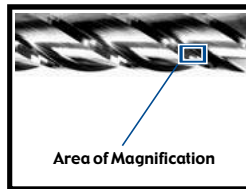
Ultra-Grain® 1
Cobalt Percentage: 10%
Grain Size (µm): ≤ 0.7
Hardness: 92.7 HRA
Fracture Toughness (K1c): 7.9
TRS (GPa): 4.1
Density (gm/cc): 14.30



2

SmoothGrind®

COMPONENT #2: The Grinding Process After selecting the best material available, Tool Alliance has perfected the manufacturing technology to optimize 100% of its physical properties. We call this process SmoothGrind®. Years in development, SmoothGrind is the result of a proprietary combination of material, abrasive, coolant, machine-tool, software, and grinding method technologies that produce cutting tools with superior qualitative characteristics. Sharper and longer lasting cutting edges, enhanced work piece finishes, and much improved lubricity are just some of the benefits brought to you by the latest solid carbide rotary tooling advances from Tool Alliance. The two photos above display an Ultra-Tool end mill primary relief featuring SmoothGrind (left) versus a major competitor's product (right). To fully demonstrate the difference, the Ultra end mill is shown at double the magnification. Note the straight line of our end mill's primary relief in comparison to the jagged edge of the competing product. Keep in mind the competitive end mill is a very good product that has a large following, yet the difference is substantial.



SmoothGrind® Competitor's

3

SmoothContricity®



COMPONENT #3: The Tooling Process All the best physical ingredients are wasted unless they are all pulled together in a comprehensive system that maximizes their respective attributes. Tool Alliance calls this process SmoothContricity®. Our customer base represents the leading edge of machine tool utilization, and SmoothContricity ensures that optimum results can be obtained in a variety of ways; minimized run-out (TIR), industry-leading tolerances on diameter & radius, and 100% Shrink Fit Ready (SFR) shanks. Combined, these attributes allow our consumers to reach full machining potential and position the cutting tool as a systematic contributor to process consistency and repeatability.



Shrink Fit Ready

4

SmoothEdge®

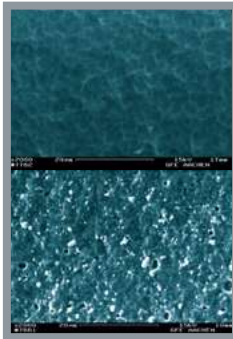


.0001 SmoothEdge atop cylindrical margin atop primary relief.



COMPONENT #4: The Edge Preparation Process

Our cutting edges are literally too sharp for certain materials. For our carbide inserts and now increasingly for our solid carbide round tools, proper edge preparation can yield huge productivity improvements to "out of the box" tool application. Using a treatment we call SmoothEdge® and performed on machine tools developed in our own R&D lab, we've taken the mystery out of tool "break-in" and provided a consistency that can be counted on time and again. The processes range from a microblasting treatment using extremely fine aluminum oxide powder to a diamond-lapping compound to brushes. All are application-specific to sound and run smooth from the first cut and protect your tooling investment from unnecessary potential for chipping during your initial tooling paths. Big productivity gains can be achieved in certain applications as well due to improved chip formation and evacuation. Learn more about SmoothEdge on Page #55.



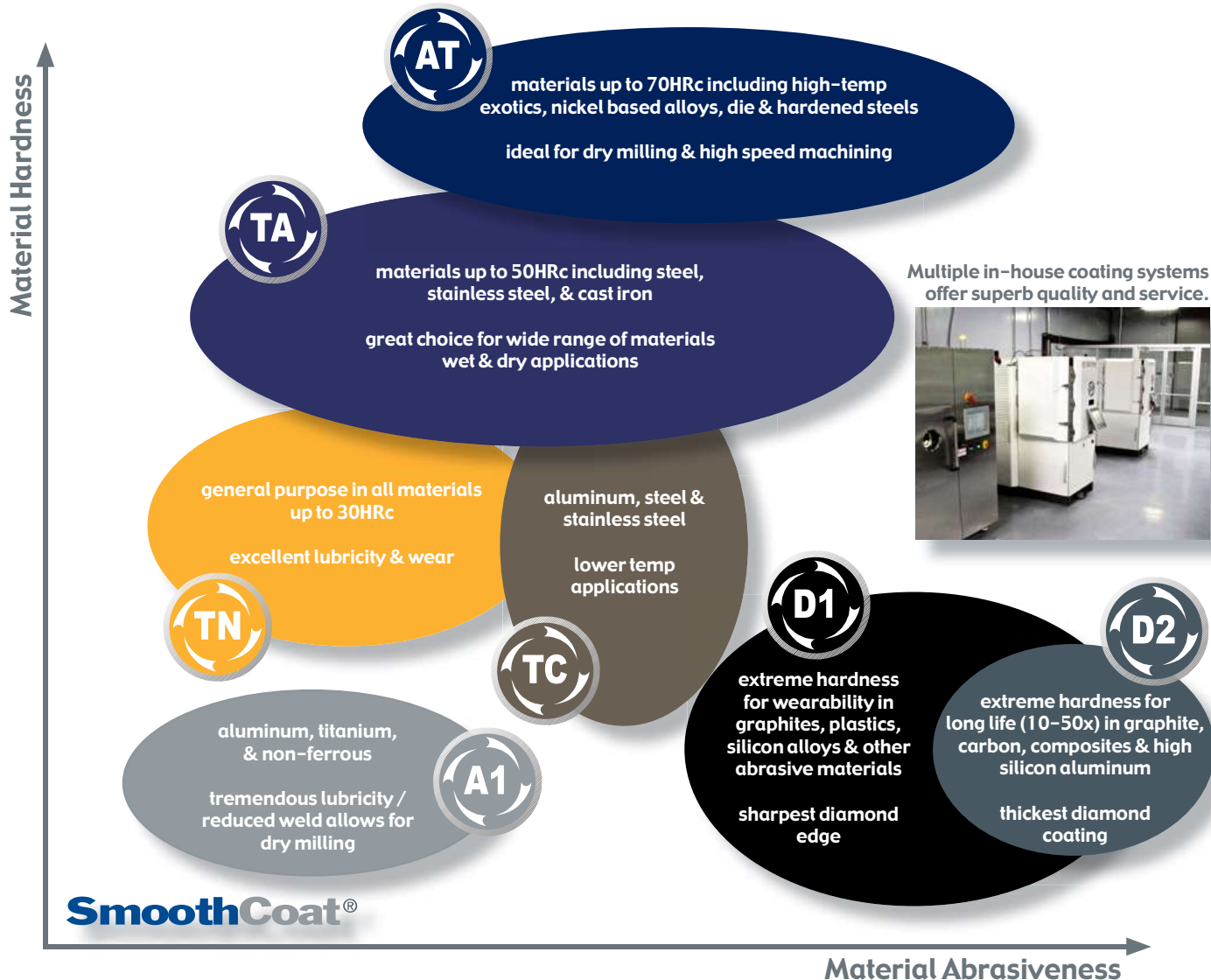
Our coating @ 2,000X (top).
Everybody else's (bottom).

SmoothCoat® 5

COMPONENT #5: The Coating Process The challenge of finding a coating method to leverage 100% of the inherent assets of our carbide grade and grinding technologies was difficult. What we finally discovered was such a perfect fit and so logical for our product lines that we invested heavily into the process we now call SmoothCoat®. Much more than simply the standard arc-deposited PVD coating, SmoothCoat involves sputter multi-layering and a multi-step prep & post operation called Micro-Blasting. The advantages of this procedure include relieving of tensile stresses underneath the cutting edge, increased stability of the coating surface, and perhaps most importantly, elevating SmoothGrind even another notch by leveling and activating the cemented carbide substrate. The result is a smooth, shiny, tough, and durable surface that can withstand tomorrow's machining requirements and outlast competitive coatings. Additionally, we've made it a standard feature on thousands of our standard catalog items. Our coating services are performed within our own factories for quality & extremely quick turnaround times.

Coating Availability

Order by adding the suffix TA, TN, AT, TC, A1, D1, or D2 to the EDP #.



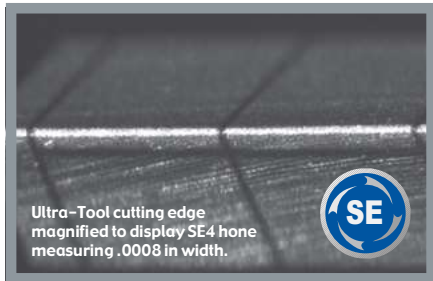
SmoothCoat®

ULTRATOOL® Technical Data

SmoothEdge®

The Edge Preparation Process

Our cutting edges are literally too sharp for certain materials. For our carbide inserts and now increasingly for our solid carbide round tools, proper edge preparation can yield huge productivity improvements to “out of the box” tool application. Using a process we call **SmoothEdge®** and performed on machine tools developed in our own R&D lab, we’ve taken the mystery out of tool “break-in” and provided a consistency that can be counted on time and again. All five types of **SmoothEdge** will yield different benefits dependent upon application. **SmoothEdge** will make your tools sound and run smooth from the first cut and protect your tooling investment from unnecessary potential for chipping during initial tool paths.



Combine SmoothEdge with our other value added features to design the ultimate cutting solution.

SmoothGrind®

- Lubricity
- Sharpness
- Polished Cutting Edges
- Hardness & Adhesion
- Masked Shanks
- Coating Uniformity
- Minimized TIR
- Shrink Fit Ready (SFR)
- Tight Tolerances

SmoothCoat®

SmoothContricity®

Primary SmoothCoat recommendations:



A1 for SE2



TA for SE4



AT for SE5

Our newest technology can achieve incredible productivity increases in specific applications. Many of our new Series include SmoothEdge as a standard feature, while on others it can be added as a same day post treatment for a small charge. Ask your Inside Sales representative about SmoothEdge today!



SmoothEdge 1

A microblasting treatment using extremely fine aluminum oxide powder to smooth the carbide surface while generating a very light edge preparation. This feature comes standard with any SmoothCoat® coating.

Uses: Highly recommended for most milling and drilling applications.



SmoothEdge 2

A lapping treatment to create extreme lubricity & smoothness with minimal edge prep on uncoated tools.

Uses: Highly recommended for milling and drilling of aluminum and other non-ferrous applications using UnCoated, A1, or TC coated tools.



SmoothEdge 3

Combines microblasting and lapping for a light hone with extreme lubricity.

Uses: Highly recommended for a wide range of general purpose machining applications using coated tools.



SmoothEdge 4

Adds a proprietary hone to the blasting and lapping cycles for a medium edge prep with excellent lubricity.

Uses: Highly recommended for milling and drilling applications involving general steels, stainless, and cast iron.



SmoothEdge 5

Doubles the honing and lapping cycle for maximum edge strength; a robust edge preparation combined with excellent lubricity characteristics.

Uses: Highly recommended for milling and drilling applications involving stainless, high-temp alloys, and exotics.

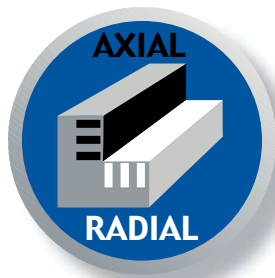
ULTRATOOL Technical Data

With so many variables present in the machining process, it is essential to optimize every possible factor to achieve world-class efficiency. Your choice of a genuine Ultra-Tool® Solid Carbide product is an excellent first step in the process. Ultra-Tool® Solid Carbide products are high-performance tools that will perform best in a machining environment characterized by rigid fixturing and minimal spindle runout. Attention to proper speed and feed will eliminate vibration, chatter, and overheating as well as extending tool life. Generally speaking, the peripheral speed of solid carbide tools will vary with the hardness of the material being cut. The harder the material, the slower the speed. High speed and insufficient feed will cause work surface glazing and poor tool life. Chipping of cutting edges is an indication of chatter which can be caused by too high of speed, too light of cut, or improper support of the tool or workpiece. Handling is also very important; sharpened cutting edges should never be allowed to come into contact with any hard object (or another tool) in a non-machining environment as they will chip easily. Keep your Ultra-Tool® products in their original protective packaging until ready for use.

The guidelines on the following pages are generalities designed to demonstrate the operating window within which you may experience the best results. The charts and information provided should prove valuable in longer tool life with greatly reduced operational costs. This information is for uncoated product: SmoothCoat products will have significantly higher speed and feed rates. For more information contact an Ultra-Tool® Factory Engineer, Sales Manager or consult our websites at ultra-tool.com and toolalliance.com. eMails can be sent to technical@toolalliance.com.

Ultra-Tool International, Inc. is constantly striving to improve its processes, specifications, and tolerances. As such, products are subject to change without prior notice.

WARNING: Grinding or other use of this tool may produce hazardous dust and fumes which may endanger health. Grinding or modification should be done by professionals only. To avoid adverse health effects, read the material safety data sheet for this product. Utilize adequate ventilation and appropriate protection. Cutting tools may shatter when broken; eye protection in vicinity of use is strongly advised. MSDS available at www.ultra-tool.com.



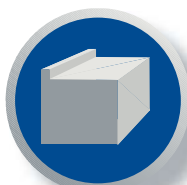
Commonly Used Formulas:

Surface Feet Minute (SFM)=RPM x Diam. x .262
 Revolutions Per Minute (RPM)=3.82 x (SFM / Diam.)
 Feed Rate (IPM)=IPT x #teeth x RPM
 Drilling (IPM)=IPR x RPM
 Feed Per Tooth (IPT)=IPM / (#teeth x RPM)
 Convert Inches to millimeters: Multiply by 25.4
 Convert millimeters to Inches: Multiply by .03937

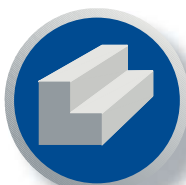
Tech Help Call, eMail us at technical@toolalliance.com, or copy / fax us this page for detailed assistance beyond what printed materials can provide. Please have the following information available to assure we can promptly process a response.

Checklist:

- Tool Description
- Application Description
- Work Piece Material
- Hardness (HRc)
- Current Speed (RPM or SFPM)
- Current Feed (CPT or IPM or FPR)
- Axial DOC
- Radial DOC
- Hole Depth (drilling)
- Machine Tool



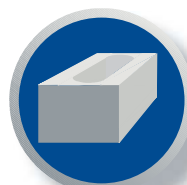
Face Milling



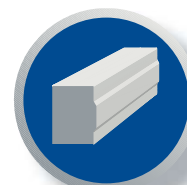
End Milling



Slot Milling



Pocket Milling



Peripheral Milling



Application Tips for ULTRATOOL® Solid Carbide Products

Trouble Shooting for Ultra-Tool® Carbide End Mills

Problem	Cause	Solution
Chipping	<ul style="list-style-type: none"> Feed rate too high Up milling (conventional) Cutting edge too sharp Chattering Loose tool Workpiece rigidity Tool rigidity Low cutting speed Loose toolholder 	<ul style="list-style-type: none"> Reduce feed rate Change to down milling (climb) Hone cutting edge or allow break-in Reduce RPM Remove, clean, and retighten Tighten workpiece holding method Shorten LOC, place shank further up holder Increase RPM Remove from spindle, clean and replace
Wear	<ul style="list-style-type: none"> High cutting speed Low feed rate Up milling (conventional) Hard material Poor chip evacuation Improper cutter helix Poor coolant 	<ul style="list-style-type: none"> Reduce RPM Increase feed rate Change to down milling (climb) Use coated tool Reposition coolant lines, use air blasting Change to recommended helix angle Replace coolant or correct mixture
Breakage	<ul style="list-style-type: none"> Feed rate too high Depth of cut too large Poor tool rigidity Tool wear Poor chip evacuation 	<ul style="list-style-type: none"> Reduce feed rate Reduce depth of cut Shorten LOC, place shank further up holder Replace/regrind sooner Reposition coolant lines, use air blasting
Chattering	<ul style="list-style-type: none"> Speed and feed too high Poor toolholder rigidity Poor spindle rigidity Workpiece rigidity Relief angle too high Depth of cut too large Poor tool rigidity 	<ul style="list-style-type: none"> Reduce feed rate Replace with shorter/more rigid holder Use larger spindle or different machine tool Tighten workpiece holding method Regrind with smaller relief angle Reduce depth of cut Shorten LOC, place shank further up holder
Short Life	<ul style="list-style-type: none"> Cutter/workpiece friction Hard material Poor material condition Improper cutter angle Poor coolant 	<ul style="list-style-type: none"> Use coated tool Use coated tool, clean material surface Regrind with proper primary relief angle Replace coolant or correct mixture
Chip Packing	<ul style="list-style-type: none"> Feed rate too high Low cutting speed Insufficient chip room Insufficient coolant 	<ul style="list-style-type: none"> Reduce feed rate or increase speed Increase RPM or reduce feed rate Use tool with less flutes, increase helix Increase volume of coolant
Poor Surface Finish	<ul style="list-style-type: none"> Feed rate too high Low cutting speed Tool wear Edge build up Depth of cut too large Chip welding 	<ul style="list-style-type: none"> Reduce feed rate Increase RPM Replace or regrind tool Increase RPM, switch to higher helix tool Reduce depth of cut Increase volume of coolant
Burring or Workpiece Chipping	<ul style="list-style-type: none"> Tool wear Improper helix angle Feed rate too high Depth of cut too large 	<ul style="list-style-type: none"> Replace or regrind tool Change to recommended helix angle Reduce feed rate Reduce depth of cut
Workpiece Inaccuracy	<ul style="list-style-type: none"> Loose/worn toolholder Poor toolholder rigidity Poor spindle rigidity Insufficient number of flutes Tool deflection 	<ul style="list-style-type: none"> Repair or replace Replace with shorter/more rigid toolholder Use larger spindle or different machine tool Use tool with higher flute quantity Shorten LOC, place shank further up holder

Trouble Shooting for Ultra-Tool® Carbide Drills

Problem	Cause	Solution (see key below)
Heavy Wear at Outer Edge	<ul style="list-style-type: none"> Insufficient coolant Incorrect speed & feed 	<ul style="list-style-type: none"> 5, 6 1, 2, 8
Chipping at Outer Cutting Edge	<ul style="list-style-type: none"> Loose tool, tool movement Workpiece movement Poor coolant conditions Incorrect speed & feed 	<ul style="list-style-type: none"> 8, 10, 11, 12, 14, 16, 17, 21 8, 12, 13, 21 5, 6 1, 2, 3, 4
Drill Point Chipping	<ul style="list-style-type: none"> Loose tool, tool movement Incorrect speed & feed Drill centering 	<ul style="list-style-type: none"> 10, 11, 12, 14 1, 2, 3, 4 8, 10, 11, 12, 21
Margin Wear	<ul style="list-style-type: none"> Drill margin rubbing wall Poor chip evacuation Poor coolant conditions Workpiece movement 	<ul style="list-style-type: none"> 20 (check drill for backtaper) 5, 6, 8, 20 5, 6 8, 13, 21
Tool Breakage	<ul style="list-style-type: none"> Loose tool, tool movement Workpiece movement Wrong drill type Poor coolant conditions Incorrect speed & feed 	<ul style="list-style-type: none"> 8, 10, 11, 12, 14, 16, 17, 21 8, 12, 13, 21 9, 15, 16, 18, 19, 20 5, 6 1, 2, 3, 4
Poor Tool Life	<ul style="list-style-type: none"> Incorrect speed & feed Poor coolant conditions Wrong drill point 	<ul style="list-style-type: none"> 1, 2, 3, 4 5, 6 8, 21
Drill Walk	<ul style="list-style-type: none"> Incorrect speed & feed Tool wear Wrong drill point Material condition 	<ul style="list-style-type: none"> 1, 2 7, 8, 21 8, 10, 11, 21 11, 12, 15, 16, 17
Chip Welding	<ul style="list-style-type: none"> Poor coolant conditions Wrong drill type 	<ul style="list-style-type: none"> 5, 6 19, 20
Hole Size Inaccuracy	<ul style="list-style-type: none"> Incorrect speed & feed Poor coolant conditions Loose tool Wrong drill type 	<ul style="list-style-type: none"> 1, 2, 3, 4 5, 6 14 9, 18
Non-Cylindrical Hole	<ul style="list-style-type: none"> Loose tool, tool movement Workpiece movement Incorrect speed & feed Wrong drill type 	<ul style="list-style-type: none"> 8, 10, 11, 12, 14, 16, 17 13 1, 2 18, 21
Heavy Burr	<ul style="list-style-type: none"> Incorrect speed & feed Incorrect drill point 	<ul style="list-style-type: none"> 1, 2 8, 21
Blue Chips	<ul style="list-style-type: none"> Poor coolant conditions Tool wear 	<ul style="list-style-type: none"> 5, 6 7, 8
Long Chips	<ul style="list-style-type: none"> Poor point grind Incorrect speed & feed 	<ul style="list-style-type: none"> 8 1, 2
Solutions Key for Drills	<ul style="list-style-type: none"> 1) Reduce RPM 2) Increase feed 3) Increase RPM 4) Reduce feed 5) Increase coolant 6) Increase mixture 7) Add negative hone 8) Repoint drill 9) Correct drill type/size 10) Use self-centering drill 11) Spot/center drill 12) Clean surface 13) Improve rigidity/clamp 14) Tighten holder 15) Use straight flute 16) Use stub length 17) Place further up holder 18) Use three-flute 19) Use slower helix 20) Use parabolic design 21) Change point style 	

Trouble Shooting for Ultra-Tool® Carbide Reamers

Problem	Cause	Solution
Chatter	<ul style="list-style-type: none"> High cutting speed Feed rate too low Workpiece movement Toolholder rigidity Tool rigidity 	<ul style="list-style-type: none"> Lower RPM or increase feed rate Increase feed rate Tighten workpiece rigidity Tighten toolholder or reduce float Use shorter tool, place further up holder
Tool Wear / Chipping	<ul style="list-style-type: none"> Incorrect feed rate Incorrect speed Poor hole condition Abrasive material Poor chip evacuation Poor coolant Insufficient coolant Workpiece alignment Excessive stock removal 	<ul style="list-style-type: none"> Increase feed rate (typically) Reduce speed (typically) Work-hardened hole; change drilling type Use proper coolant, coated reamer Use/increase coolant, use helical reamer Replace coolant or correct mixture Increase coolant volume Use bushing, floating holder, lead chamfer Use larger diameter starter drill
Tool Breakage	<ul style="list-style-type: none"> Incorrect feed rate Incorrect speed Tool wear Bottoming of hole Coolant conditions Insufficient stock removal Poor set up Excessive stock removal 	<ul style="list-style-type: none"> Increase feed rate (typically) Reduce speed (typically) Sharpen or replace reamer Adjust stop depth, check preset Increase, replace, or correct coolant Use smaller diameter starter drill Use bushing, floating toolholder Use larger diameter starter drill

Problem	Cause	Solution
Poor Finish	<ul style="list-style-type: none"> Feed rate too low Insufficient stock removal Poor hole condition Poor coolant Insufficient coolant 	<ul style="list-style-type: none"> Increase feed rate Use smaller diameter starter drill Work-hardened hole; change drilling type Replace/correct coolant mixture Increase coolant volume
Hole Tolerance	<ul style="list-style-type: none"> Workpiece alignment Incorrect tool size Material shrinkage Tool wear Toolholder runout 	<ul style="list-style-type: none"> Use bushing, floating toolholder Check diameter of tool Adjust diameter for shrinkage; more coolant Sharpen or replace tool Adjust or replace toolholder