

ULTRATOOL® Two & Three Flute Specialty End Mills for Aluminum

Series 330AL
Ultra-Grain® End Mill
Two Flute • 45 Degree RH Spiral

The Ultra-Tool® Series 330AL End Mill is designed specifically for the highest possible efficiency in the milling of aluminum. The 330AL will allow full machine capacity roughing and close tolerance finishing of aluminum and other non-ferrous materials. SmoothEdge now included with TC coating!



SFR 330AL Series End Mill Specs:
Cutting Diam +.0000/-0.0003
Shank Diam +.0000/-0.0003



Clearance neck available via Express Service

ULTRATOOL PERFORMANCE S E R V I C E S



Diam	LOC	OAL	Shank	Square EDP#	.010 Rad EDP#	.020 Rad EDP#	.030 Rad EDP#	.060 Rad EDP#	.090 Rad EDP#	.125 Rad EDP#	.190 Rad EDP#	.250 Rad EDP#	Ball EDP#	Coating		
														UnCoated	TC or A1	
1/8	1/4	1-1/2	1/8	11104	11747										\$9.20	\$11.50
1/8	1/2	1-1/2	1/8	11139	11748								11808		\$9.30	\$11.70
3/16	3/8	2"	3/16	11107	11749										\$12.70	\$15.80
3/16	5/8	2"	3/16	11142	11750								11809		\$13.00	\$16.10
1/4	3/8	2-1/2	1/4	11110		11751	11765	11781							\$18.00	\$23.60
1/4	1/2	2"	1/4	39750		11752	11766	11782							\$17.10	\$22.70
1/4	1"	2-1/2	1/4	39710		11753	11767	11783					11810		\$21.60	\$27.10
1/4	2"	4"	1/4	39751		11754	11768	11784							\$36.30	\$43.70
5/16	1/2	2-1/2	5/16	11113		11755	11769								\$23.40	\$30.80
5/16	3/4	2-1/2	5/16	11148		11756	11770						11811		\$24.00	\$31.40
5/16	1-3/8	3"	5/16	39752		11757	11771								\$37.50	\$44.80
3/8	5/8	2-1/2	3/8	11116		11758	11772	11785							\$25.40	\$32.80
3/8	3/4	2-1/2	3/8	39753		11759	11773	11786					11812		\$26.20	\$33.60
3/8	1-1/8	2-1/2	3/8	39711		11760	11774	11787							\$29.70	\$37.10
3/8	2"	4"	3/8	39754		11761	11775	11788							\$43.10	\$54.20
1/2	3/4	3"	1/2	39755		11762	11776	11789							\$42.60	\$53.00
1/2	1-1/4	3"	1/2	39712		11763	11777	11790					11813		\$44.80	\$55.30
1/2	2"	4"	1/2	39756		11764	11778	11791							\$66.60	\$80.80
5/8	1-3/4	3-1/2	5/8	39713			11779	11792	11798				11814		\$84.30	\$101.00
5/8	3"	6"	5/8	39757			11780	11793	11799						\$130.40	\$156.30
3/4	1-3/4	4"	3/4	39714				11794	11800	11804	27219	27223	11815		\$112.80	\$131.30
3/4	3"	6"	3/4	39758				11795	11801	11805	27220	27224			\$182.80	\$211.10
1"	1-7/8	4"	1"	39715				11796	11802	11806	27221	27225	11816		\$188.30	\$216.10
1"	3"	6"	1"	39759				11797	11803	11807	27222	27226			\$291.80	\$331.90

new!



Ultra-Tool offers 2 world class coatings as standard for our aluminum tools!

Simply add a "TC" or "A1" suffix to the EDP#. For UnCoated, either use no suffix or add "UC."

SmoothEdge is included!

Series 333AL
Ultra-Grain® End Mill
Three Flute • 45 Degree RH Spiral

The Ultra-Tool® Series 333AL End Mill is designed specifically for the highest possible efficiency in the milling of aluminum. The 333AL will allow full machine capacity roughing and close tolerance finishing of aluminum and other non-ferrous materials. SmoothEdge now included with TC coating!



SFR 333AL Series End Mill Specs:
Cutting Diam +.0000/-0.0003
Shank Diam +.0000/-0.0003



Clearance neck available via Express Service

ULTRATOOL PERFORMANCE S E R V I C E S



Diam	LOC	OAL	Shank	Square EDP#	.010 Rad EDP#	.020 Rad EDP#	.030 Rad EDP#	.060 Rad EDP#	.090 Rad EDP#	.125 Rad EDP#	.190 Rad EDP#	.250 Rad EDP#	Ball EDP#	Coating		
														UnCoated	TC or A1	
1/8	1/4	1-1/2	1/8	11544	11640										\$10.20	\$12.50
1/8	1/2	1-1/2	1/8	11545	11641								27235		\$10.30	\$12.60
3/16	3/8	2"	3/16	11546	11642										\$14.00	\$17.10
3/16	5/8	2"	3/16	11547	11643								27236		\$14.30	\$17.40
1/4	3/8	2-1/2	1/4	11548		11644	11658	11674							\$19.80	\$25.40
1/4	1/2	2"	1/4	11549		11645	11659	11675							\$18.90	\$24.40
1/4	1"	2-1/2	1/4	11550		11646	11660	11676					27237		\$23.70	\$29.30
1/4	2"	4"	1/4	11551		11647	11661	11677							\$39.90	\$47.30
5/16	1/2	2-1/2	5/16	11552		11648	11662								\$25.80	\$33.20
5/16	3/4	2-1/2	5/16	11553		11649	11663						27238		\$26.30	\$33.80
5/16	1-3/8	3"	5/16	11554		11650	11664								\$41.10	\$48.50
3/8	5/8	2-1/2	3/8	11555		11651	11665	11678							\$27.90	\$35.30
3/8	3/4	2-1/2	3/8	11556		11652	11666	11679					27239		\$28.80	\$36.20
3/8	1-1/8	2-1/2	3/8	11557		11653	11667	11680							\$32.60	\$40.00
3/8	2"	4"	3/8	11558		11654	11668	11681							\$47.50	\$58.50
1/2	3/4	3"	1/2	11559		11655	11669	11682							\$46.70	\$57.20
1/2	1-1/4	3"	1/2	11560		11656	11670	11683					27240		\$49.30	\$59.80
1/2	2"	4"	1/2	11561		11657	11671	11684							\$73.10	\$87.30
5/8	1-3/4	3-1/2	5/8	11562			11672	11685	11691				27241		\$92.90	\$109.50
5/8	3"	6"	5/8	11563			11673	11686	11692						\$143.40	\$169.40
3/4	1-3/4	4"	3/4	11564				11687	11693	11697	27227	27231	27242		\$123.90	\$142.40
3/4	3"	6"	3/4	11565				11688	11694	11698	27228	27232			\$201.20	\$229.50
1"	1-7/8	4"	1"	11566				11689	11695	11699	27229	27233	27243		\$207.20	\$235.00
1"	3"	6"	1"	11567				11690	11696	11700	27230	27234			\$321.20	\$361.40



Application Data for High Performance Series 330AL and 333AL ULTRATOOL End Mills

The milling data presented below is for the 330AL and 333AL Series of Ultra end mills. When using SmoothCoat & SmoothEdge surface treatments, Surface Feet or Meters Per Minute may be increased from the stated levels. Do not use a radial DOC exceeding more than 50% of diameter.



Peripheral Milling data based on axial depth \leq 150% of tool diameter & radial depth of \leq 50% of tool diameter.



Slot Milling data based on axial depth of cut \leq 100% of tool diameter.

End Mill Specifications:

Diameter: $+.0000 / -.0003$
Shank Diameter: $+.0000 / -.0003$
LOC: $+.060 / -.000$
OAL: $\pm .060$
Helix: $\pm 1^\circ$

Milling;
Fractional

Material	Type	SFPM	1/8"	3/16"	1/4"	5/16"	3/8"	7/16"	1/2"	5/8"	3/4"	1"
Wrought Aluminum 6061 / 7075 / 2024	Peripheral	1200 to 2000	.0015	.0025	.0030	.0035	.0050	.0055	.0060	.0065	.0070	.0090
	Slotting	1200 to 2000	.0010	.0018	.0025	.0025	.0035	.0040	.0045	.0050	.0060	.0080
Cast Aluminum 319 / 355 / 390	Peripheral	750 to 1200	.0015	.0025	.0030	.0035	.0050	.0055	.0060	.0065	.0070	.0090
	Slotting	750 to 1200	.0010	.0018	.0025	.0025	.0035	.0040	.0045	.0050	.0060	.0080
Copper Alloys Bronze / Brass	Peripheral	500 to 1200	.0009	.0015	.0020	.0025	.0035	.0035	.0040	.0050	.0050	.0060
	Slotting	500 to 1200	.0008	.0012	.0020	.0020	.0030	.0040	.0040	.0050	.0050	.0060

330AL and 333AL Tech Tip: Start with these recommended values for standard length tools. For stub LOC's, speeds and feeds may be increased. Longer LOC's and OAL's may require reductions in FPT and DOC. Our Monolith Series operating parameters should be adjusted to maintain a stable cut and reduced chatter (due to extended reach lengths).

There are lots of reasons...

ULTRAGrain® + SmoothGrind® + SmoothConcricity® + SmoothEdge® + SmoothCoat® + SmoothFlute

why ours work better.



TC

UC

A1

Ultra-Tool offers 2 world class coatings as standard for our aluminum tools!

Simply add a "TC" or "A1" suffix to the EDP#. For UnCoated, either use no suffix or add "UC."

SmoothEdge 8 is included!



The newest SmoothEdge® for our Aluminum end mills contributes to amazing surface finishes on milled parts.

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



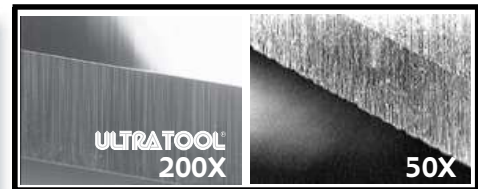
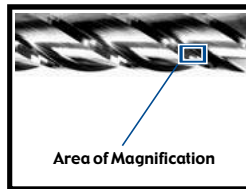
Ultra-Grain® 1
Cobalt Percentage: 10%
Grain Size (µm): ≤ 0.7
Hardness: 92.7 HRA
Fracture Toughness (K1c): 7.9
TRS (GPa): 4.1
Density (gm/cc): 14.30



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®

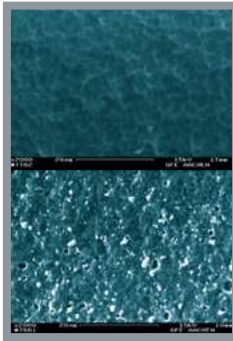


.0001 SmoothEdge atop cylindrical margin atop primary relief.



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.

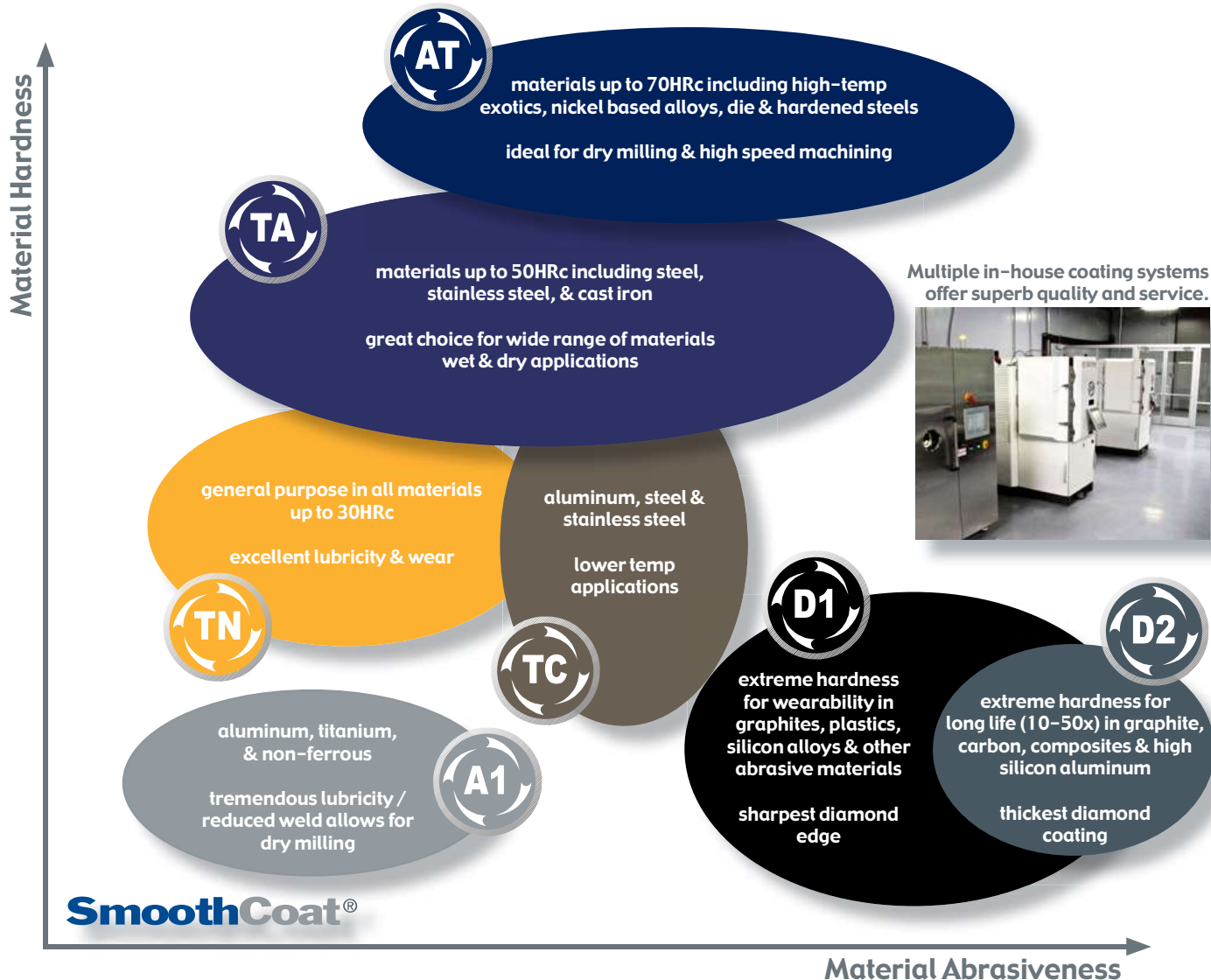


Our coating @ 2,000X (top).
Everybody else's (bottom).

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®

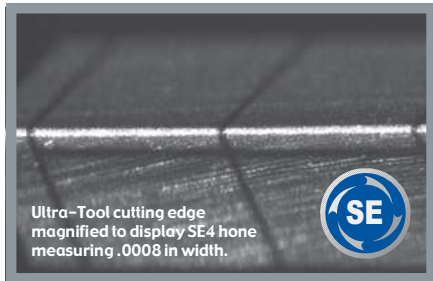
Material Abrasiveness

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.

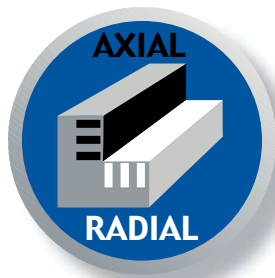
ULTRATOOL Technical Data

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.



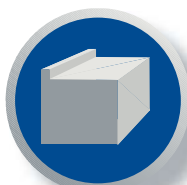
Commonly Used Formulas:

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

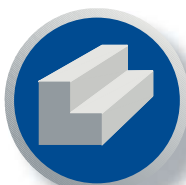
Tech Help Call, eMail us at technical@toolalliance.com, or copy / fax us this page for detailed assistance beyond what printed materials can provide. Please have the following information available to assure we can promptly process a response.

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



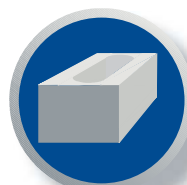
Face Milling



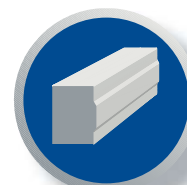
End Milling



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