phone 714.898.9224 / 800.854.2431 / fax 714.891.7816
5451 McFadden Avenue • Huntington Beach, CA 92649
ultra-tool.com

Patented SmoothFlute Series 365 Variable Helix End Mills by ULTRATOOL®

33° RHS VARIABLE HELIX

Series 365 Patented SmoothFlute™ Six Flute Variable Helix End Mill Square, Corner Radius & Ball Stub, Standard or Long Length

High speed finishing: The Ultra-Tool® Series 365 is a revolutionary solid carbide end mill featuring our patented SmoothFlute™ geometric design that can finish and rough at incredible speeds and feeds. SmoothFlute + SmoothEdge technology provides extended tool life and excellent work piece finishes by increasing stability and edge integrity. Series 365 is precision ground from Ultra-Grain®1, a premium carbide substrate that couples high hardness with excellent chipping resistance. Choose from Square, Ball, or 7 different standard radii for your roughing or finishing requirements, plus standard AT (AlTiN) coating.

ULTRATOOL® PERFORMANCE SERIES

Premium Series EM Specs:
Cutting Diam +.000/-.002
Shank Diam -.0000/-.00025
Radius ±.0005

SmoothEdge® ULTRA-Grain® SmoothFlute

AT + SE

Diam	LOC	OAL	Shank	Square EDP#	.015R EDP#	.030R EDP#	.060R EDP#	.090R EDP# (new!)	.125R EDP#	.190R EDP# (new!)	.250R EDP# (new!)	Ball EDP#	AT Coated
1/4	3/8	2"	1/4	31680AT	31681AT	31682AT	31683AT					31685AT	$22.00
1/4	3/4	2-1/2	1/4	31690AT	31691AT	31692AT	31693AT					31695AT	$25.70
1/4	1"	4"	1/4	31700AT	31701AT	31702AT	31703AT					31705AT	$33.50
5/16	13/16	2-1/2	5/16	31710AT	31711AT	31712AT	31713AT					31715AT	$30.40
3/8	1/2	2-1/2	3/8	31720AT	31721AT	31722AT	31723AT	27187AT	31724AT			31725AT	$31.90
3/8	1"	2-1/2	3/8	31730AT	31731AT	31732AT	31733AT	27188AT	31734AT			31735AT	$36.50
3/8	1-1/2	4"	3/8	31740AT	31741AT	31742AT	31743AT	27189AT	31744AT			31745AT	$54.60
1/2	5/8	2-1/2	1/2	31750AT	31751AT	31752AT	31753AT	27190AT	31754AT			31755AT	$51.70
1/2	1"	3"	1/2	31760AT	31761AT	31762AT	31763AT	27191AT	31764AT			31765AT	$57.80
1/2	1-1/4	3"	1/2	31920AT	31921AT	31922AT	31923AT	27192AT	31924AT			31925AT	$62.70
1/2	1-3/4	4"	1/2	31770AT	31771AT	31772AT	31773AT	27193AT	31774AT			31775AT	$82.30
5/8	3/4	3-1/2	5/8	31780AT	31781AT	31782AT	31783AT	27194AT	31784AT			31785AT	$96.40
5/8	1-1/4	3-1/2	5/8	31790AT	31791AT	31792AT	31793AT	27195AT	31794AT			31795AT	$108.30
5/8	1-5/8	3-1/2	5/8	31650AT	31651AT	31652AT	31653AT	27196AT	31654AT			31655AT	$112.90
5/8	2-1/4	5"	5/8	31800AT	31801AT	31802AT	31803AT	27197AT	31804AT			31805AT	$149.00
3/4	7/8	4"	3/4	31810AT	31811AT	31812AT	31813AT	27198AT	31814AT	27205AT	27212AT	31815AT	$138.90
3/4	1-1/2	4"	3/4	31820AT	31821AT	31822AT	31823AT	27199AT	31824AT	27206AT	27213AT	31825AT	$150.00
3/4	2-1/2	6"	3/4	31830AT	31831AT	31832AT	31833AT	27200AT	31834AT	27207AT	27214AT	31835AT	$224.70
3/4	3-1/4	6"	3/4	31660AT	31661AT	31662AT	31663AT	27201AT	31664AT	27208AT	27215AT	31665AT	$241.30
1"	1-1/2	4"	1"	31840AT	31841AT	31842AT	31843AT	27202AT	31844AT	27209AT	27216AT	31845AT	$224.50
1"	2-1/2	6"	1"	31850AT	31851AT	31852AT	31853AT	27203AT	31854AT	27210AT	27217AT	31855AT	$334.20
1"	3-1/4	6"	1"	31670AT	31671AT	31672AT	31673AT	27204AT	31674AT	27211AT	27218AT	31675AT	$365.30

new! Standard Radius sizes throughout the range!

Metric Sizes: The Ultra-Tool® Series 365 is also available in a limited range of metric sizes, each provided with a popular sized radius. Please see catalog page #16.

The patented 365 Monolith™ end mill, Flagship of the industry.

100% American Designed American Made American Owned

High Efficiency Milling (HEM) ratio

phone 714.898.9224 / 800.854.2431 / fax 714.891.7816
5451 McFadden Avenue • Huntington Beach, CA 92649
ultra-tool.com

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

phone 714.898.9224 / 800.854.2431 / fax 714.891.7816
5451 McFadden Avenue • Huntington Beach, CA 92649
ultra-tool.com

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 (K_{1c}): 6.6
TRS (GPa): 3.8
Density (gm/cc): 14.90

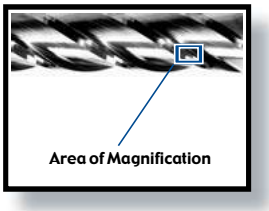
ULTRA-Carb®

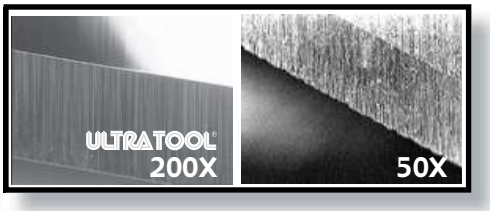
Ultra-Grain® 1
Cobalt Percentage: 10%
Grain Size (µm): ≤ 0.7
Hardness: 92.7 HRa
Fracture Toughness (K_{1c}): 7.9
TRS (GPa): 4.1
Density (gm/cc): 14.30

ULTRA-Grain®

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.

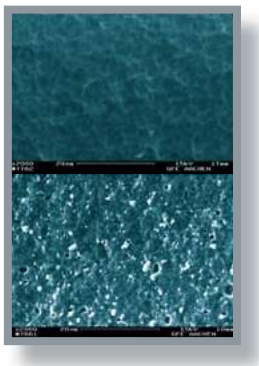
SFR
Shrink Fit Ready

4 SmoothEdge®

.0001 SmoothEdge atop cylindrical margin atop primary relief.

SE

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

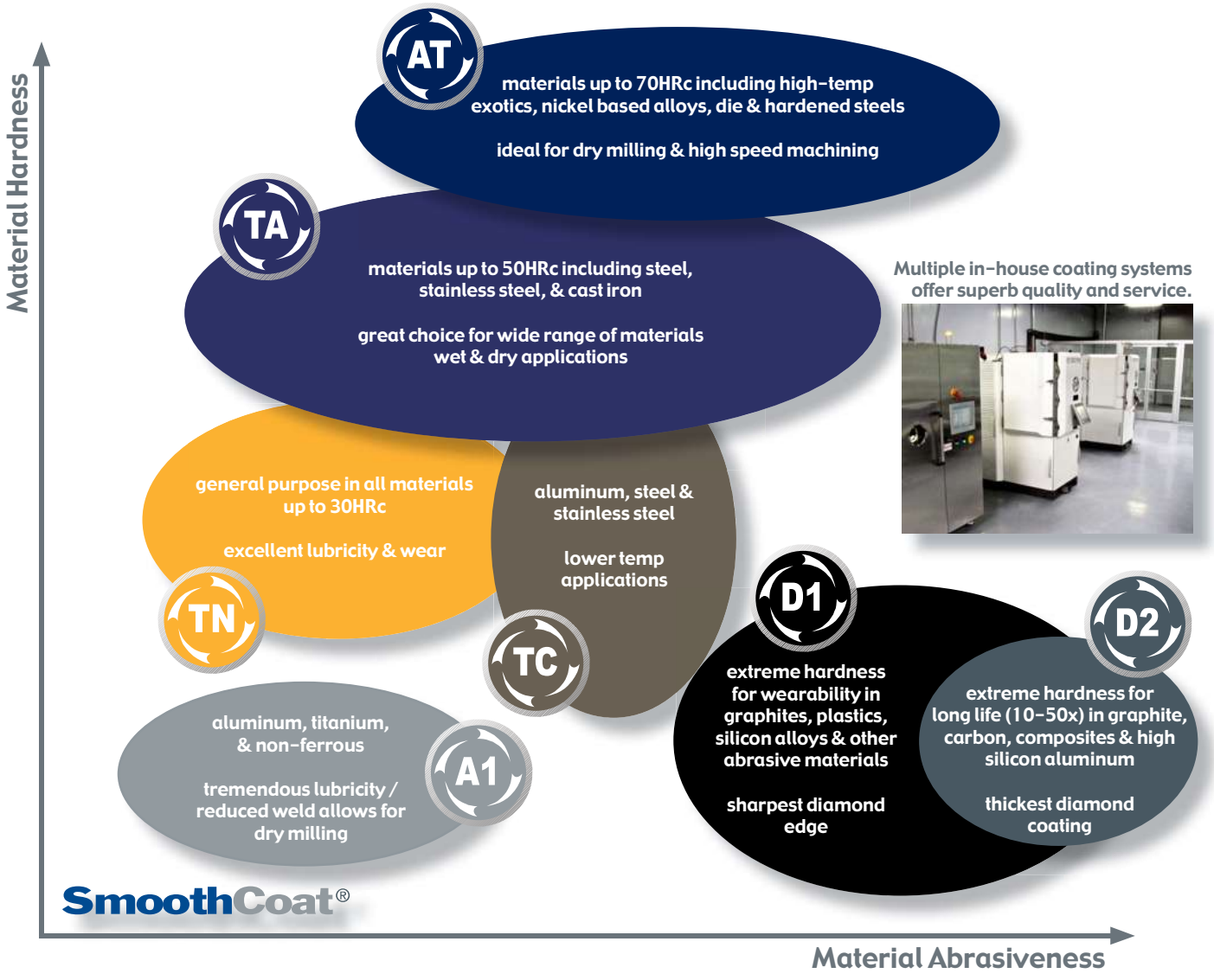
COMPONENT #5: The Coating Process The challenge of finding a coating method to leverage 100% of the inherent assets of our carbide grade and grinding technologies was difficult. What we finally discovered was such a perfect fit and so logical for our product lines that we invested heavily into the process 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 #.

UC	TA	TN	AT	TC	A1	D1	D2
UnCoated	TiAlN	TiN	AlTiN HSN^2	TiCN	TiB2	PVD Diamond	CVD Diamond

Standard Coatings available at "Coated" List Price

Premium Coatings available



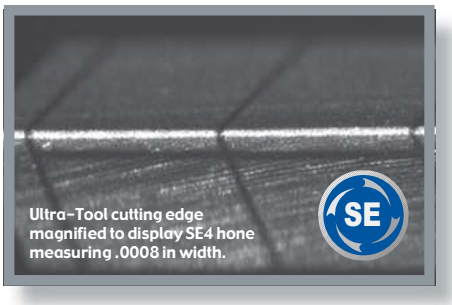
Multiple in-house coating systems offer superb quality and service.

SmoothEdge®

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!

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.



Ultra-Tool cutting edge magnified to display SE4 hone measuring .0008 in width.

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

Primary SmoothCoat recommendations:

A1 for SE2 TA for SE4 AT for SE5

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.

phone 714.898.9224 / 800.854.2431 / fax 714.891.7816
5451 McFadden Avenue • Huntington Beach, CA 92649
ultra-tool.com

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.

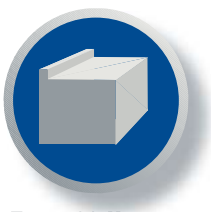
WARNING: Grinding or other use of this tool may produce hazardous dust and fumes which may endanger health. Grinding or modification should be done by professionals only. To avoid adverse health effects, read the material safety data sheet for this product. Utilize adequate ventilation and appropriate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

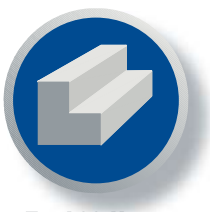
Commonly Used Formulas:

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

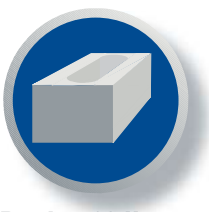
Tech Help Call, eMail us at **technical@toolalliance.com**, or copy / fax us this page for detailed assistance beyond what printed materials can provide. Please have the following information available to assure we can promptly process a response.

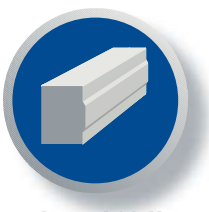
Checklist:
- Tool Description
- Application Description
- Work Piece Material
- Hardness (HRc)
- Current Speed (RPM or SFPM)
- Current Feed (CPT or IPM or FPR)
- Axial DOC
- Radial DOC
- Hole Depth (drilling)
- Machine Tool


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	• Feed rate too high • Up milling (conventional) • Cutting edge too sharp • Chattering • Loose tool • Workpiece rigidity • Tool rigidity • Low cutting speed • Loose toolholder	• 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	• High cutting speed • Low feed rate • Up milling (conventional) • Hard material • Poor chip evacuation • Improper cutter helix • Poor coolant	• 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	• Feed rate too high • Depth of cut too large • Poor tool rigidity • Tool wear • Poor chip evacuation	• Reduce feed rate • Reduce depth of cut • Shorten LOC, place shank further up holder • Replace/regrind sooner • Reposition coolant lines, use air blasting
Chattering	• 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	• 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	• Cutter/workpiece friction • Hard material • Poor material condition • Improper cutter angle • Poor coolant	• Use coated tool • Use coated tool • Use coated tool, clean material surface • Regrind with proper primary relief angle • Replace coolant or correct mixture
Chip Packing	• Feed rate too high • Low cutting speed • Insufficient chip room • Insufficient coolant	• 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	• Feed rate too high • Low cutting speed • Tool wear • Edge build up • Depth of cut too large • Chip welding	• 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	• Tool wear • Improper helix angle • Feed rate too high • Depth of cut too large	• Replace or regrind tool • Change to recommended helix angle • Reduce feed rate • Reduce depth of cut
Workpiece Inaccuracy	• Loose/worn toolholder • Poor toolholder rigidity • Poor spindle rigidity • Insufficient number of flutes • Tool deflection	• 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	• Insufficient coolant • Incorrect speed & feed	• 5, 6 • 1, 2, 8
Chipping at Outer Cutting Edge	• Loose tool, tool movement • Workpiece movement • Poor coolant conditions • Incorrect speed & feed	• 8, 10, 11, 12, 14, 16, 17, 21 • 8, 12, 13, 21 • 5, 6 • 1, 2, 3, 4
Drill Point Chipping	• Loose tool, tool movement • Incorrect speed & feed • Drill centering	• 10, 11, 12, 14 • 1, 2, 3, 4 • 8, 10, 11, 12, 21
Margin Wear	• Drill margin rubbing wall • Poor chip evacuation • Poor coolant conditions • Workpiece movement	• 20 (check drill for backtaper) • 5, 6, 8, 20 • 5, 6 • 8, 13, 21
Tool Breakage	• Loose tool, tool movement • Workpiece movement • Wrong drill type • Poor coolant conditions • Incorrect speed & feed	• 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	• Incorrect speed & feed • Poor coolant conditions • Wrong drill point	• 1, 2, 3, 4 • 5, 6 • 8, 21
Drill Walk	• Incorrect speed & feed • Tool wear • Wrong drill point • Material condition	• 1, 2 • 7, 8, 21 • 8, 10, 11, 21 • 11, 12, 15, 16, 17
Chip Welding	• Poor coolant conditions • Wrong drill type	• 5, 6 • 19, 20
Hole Size Inaccuracy	• Incorrect speed & feed • Poor coolant conditions • Loose tool • Wrong drill type	• 1, 2, 3, 4 • 5, 6 • 14 • 9, 18
Non-Cylindrical Hole	• Loose tool, tool movement • Workpiece movement • Incorrect speed & feed • Wrong drill type	• 8, 10, 11, 12, 14, 16, 17 • 13 • 1, 2 • 18, 21
Heavy Burr	• Incorrect speed & feed • Incorrect drill point	• 1, 2 • 8, 21
Blue Chips	• Poor coolant conditions • Tool wear	• 5, 6 • 7, 8
Long Chips	• Poor point grind • Incorrect speed & feed	• 8 • 1, 2

Solutions Key for Drills

1) Reduce RPM	8) Repoint drill	15) Use straight flute
2) Increase feed	9) Correct drill type/size	16) Use stub length
3) Increase RPM	10) Use self-centering drill	17) Place further up holder
4) Reduce feed	11) Spot/center drill	18) Use three-flute
5) Increase coolant	12) Clean surface	19) Use slower helix
6) Increase mixture	13) Improve rigidity/clamp	20) Use parabolic design
7) Add negative hone	14) Tighten holder	21) Change point style

Trouble Shooting for Ultra-Tool® Carbide Reamers

Problem	Cause	Solution
Chatter	• High cutting speed • Feed rate too low • Workpiece movement • Toolholder rigidity • Tool rigidity	• 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	• Incorrect feed rate • Incorrect speed • Poor hole condition • Abrasive material • Poor chip evacuation • Poor coolant • Insufficient coolant • Workpiece alignment • Excessive stock removal	• 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	• Incorrect feed rate • Incorrect speed • Tool wear • Bottoming of hole • Coolant conditions • Insufficient stock removal • Poor set up • Excessive stock removal	• 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	• Feed rate too low • Insufficient stock removal • Poor hole condition • Poor coolant • Insufficient coolant	• Increase feed rate • Use smaller diameter starter drill • Work-hardened hole; change drilling type • Replace/correct coolant mixture • Increase coolant volume
Hole Tolerance	• Workpiece alignment • Incorrect tool size • Material shrinkage • Tool wear • Toolholder runout	• Use bushing, floating toolholder • Check diameter of tool • Adjust diameter for shrinkage; more coolant • Sharpen or replace tool • Adjust or replace toolholder