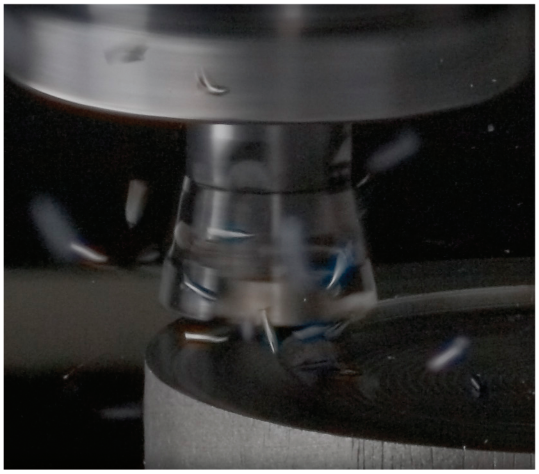
Greenleaf®

metalcutting tools and systems

CERAMIC PRODUCTIVITY MANUAL



ADD TO CART

www.greenleafglobalsupport.com

Greenleaf®
800-458-1850

Index

Illustrations

Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

Illustrations *continued*

What is a Whisker-Reinforced Ceramic?

Greenleaf WG-300®, developed by Greenleaf Corporation, is the first commercially available ceramic composite using the technology of whisker reinforcement. It can operate up to 10 times the speed used for uncoated carbide tools.

Greenleaf WG-600® is the first commercially available coated, whisker-reinforced ceramic composite. Greenleaf WG-600 offers up to 30% speed improvement and up to 3 times tool life over uncoated ceramics.

Greenleaf WG-700™ is the newest whisker-reinforced ceramic substrate. Featuring improved toughness and a unique high-speed coating, WG-700 is ideal for machining nickel- and cobalt-based super alloys and other difficult-to-cut materials. WG-700 offers high metal removal rates with exceptional tool life.

The basic concept involves reinforcing a hard ceramic matrix with extremely strong, stiff, silicon-carbide crystals, commonly called whiskers.

These whiskers are grown under carefully controlled conditions and, due to their high purity and lack of grain boundaries, approach the theoretical maximum strength obtainable. This strength is calculated to be in the order of 1 million psi (6,900 MPa) tensile!

The super-strong whiskers are dispersed into a matrix of fine-grained aluminum oxide where they act much like glass filaments do in fiberglass, for example, by adding tensile strength and improving the fracture toughness of the brittle matrix.

The increase in the fracture toughness of the material is such that inserts are now offered without hones as a standard, making them suitable for finish cuts on most forged nickel-based alloys without "smearing."

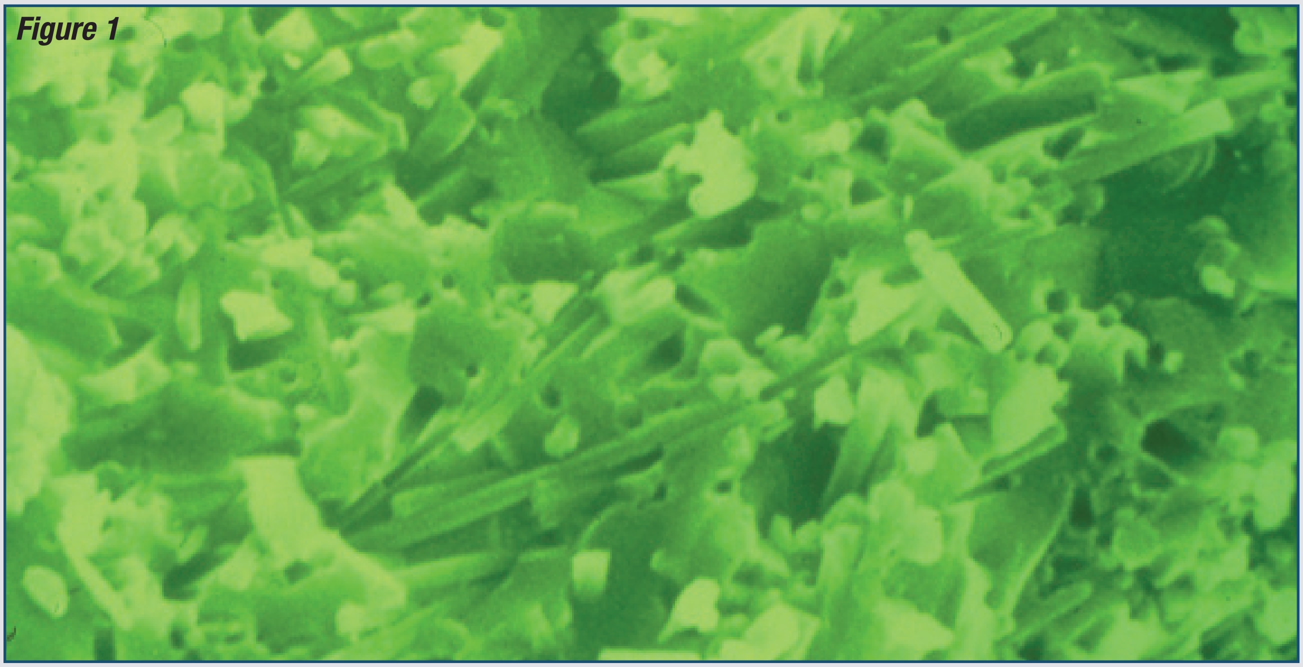
A properly manufactured whisker-reinforced ceramic has outstanding thermal and mechanical shock resistance. It can withstand intermittent cut applications, such as in milling, without breakage.

WG-300® Fracture Surface

The fracture toughness of a whisker-reinforced ceramic is enhanced by the phenomenon of whisker "pull-out." A close examination of the fracture surface at 3000x will reveal not only a clear indication of the whiskers randomly dispersed throughout the matrix, but also the obvious hexagonal holes where whiskers have actually been pulled out in the fracture process. A large amount of energy is required to pull the whiskers out. This greatly enhances the fracture toughness and the high predictability of the inserts.

Greenleaf WG -300® will not fail by catastrophic breakage unless grossly misapplied, but will be gradually consumed in a predictable wear pattern. This wear pattern will be unlike the wear modes of carbide tools. It is the subject of a later paragraph in this section and should be studied and clearly understood for successful results.

Figure 1



SEM Photomicrograph 3000x

Physical Properties

Physical properties are only a rough indicator of cutting tool performance.Ceramic cutting tools should always be evaluated in actual service where the synergisms of material properties and cutting-tool engineering can be clearly seen.

Particular note should be made that “Modulus of Rigidity” or “Transverse Rupture Strength” is expressed according to the accepted laboratory procedures for ceramics.

Here, a 2" *(50,8 mm)* long sample is broken by a four-point bend test. It is more common to refer to the 9/16" *(14,3 mm)* long three-point test in the case of carbide, and some manufacturers use this test also for ceramics. Naturally, the T.R.S. values for ceramic on the 2" *(50,8 mm)* sample are appreciably lower than they would be on a 9/16" *(14,3 mm)* test bar.

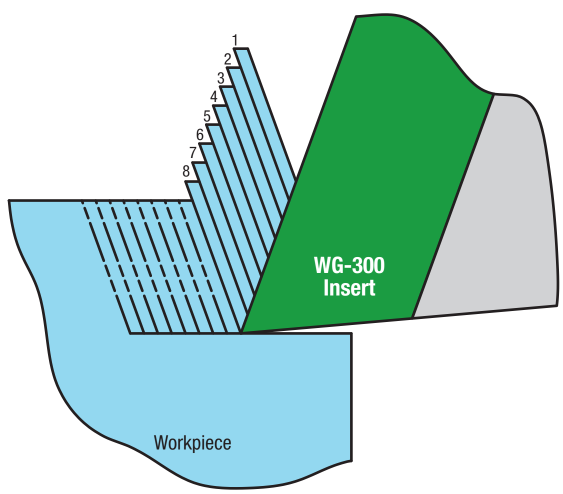
Figure 2 – Physical Properties

Microstructure	2 Phase Polycrystalline > 50% Alumina < 50% Silicon Carbide Whiskers
Density	= 3.74 g/cc
Melting Point	2040°C (3,700° F)
Hardness	≥ 94.4 RA
Modulus of Rigidity (E) **(4-Point Bend)**	TRS = 100,000 P.S.I.± 6,000 690 MPa ± 41
Young’s Modulus (E) **Modulus of Rigidity(G)**	$= 57 \times 10^6$ P.S.I. $= 23 \times 10^6$ P.S.I.
Poisson’s Ratio (M) $M = \frac{E}{2G} - 1$	= .23
Fracture Toughness *(Measures resistance of crack growth from stress)*	
Cemented Carbide	= 13.0
Hot Pressed Composite	= 3.8
WG Ceramics	= 10.0

How to Use the Properties of Greenleaf Advanced Ceramics

During the metal-removal operation, material is displaced ahead of the tool by being forced through a "shear zone" and subsequently sliding over the rake face of the tool as a chip. This action has been studied by numerous researchers including "Piispanen and Merchant," who demonstrated the mechanism of chip formation, likening it to the sideways slide of a deck of cards, caused by the rake face of the tool. *(Figure 3)*

Figure 3 – Deck-of-Cards Principle

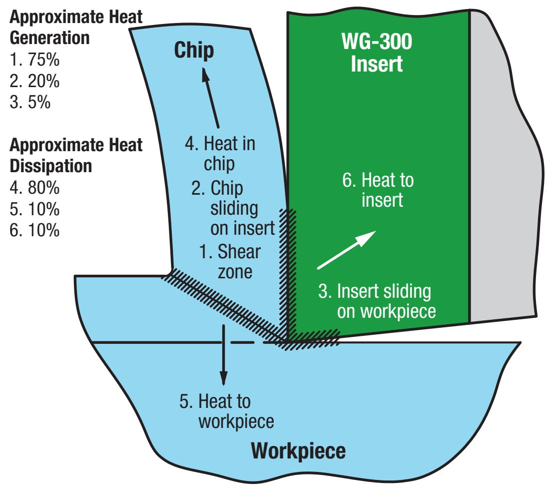


The chip is formed first by grain boundary distortion in front and below the shear plane, followed by grain boundary dislocation. This results in a chip that is always thicker than the layer of material being removed.

A large amount of shear stress is required to cause plastic deformation and shear to occur in the "shear zone," and this results in the generation of significant quantities of heat. In fact, as much as 75% of the heat generated during cutting is produced in this way. The other 25% comes from the sliding of the chip over the tool rake face and the contact of the flank of the tool with the workpiece. *(Figure 4)*

Most of the heat generated during metal cutting is dissipated by the chip carrying it away. As cutting speeds increase, the metal-cutting process becomes more adiabatic. In other words, the heat generated in the "shear zone" cannot be conducted away during the very short time in which the metal passes through this zone. We can benefit from the heat generation, temperature rise and softening effects in the "shear zone."

Figure 4 – Heat Dissipation in Ceramic Machining



The heat generated in the "shear zone" has been traditionally thought of as a negative factor since it is also associated with heat-related failure of cemented carbide cutting tools. This often leads to the need to slow down the cutting operation to a point where carbide inserts will give acceptable life.

Whisker-reinforced ceramics are able to withstand high temperatures while maintaining strength and hardness, and it has been shown that contrary to traditional methods of machining, we can, in fact, use the heat generated in the shear zone ahead of the tool to our advantage. There is an optimum speed outside the range of carbide tools where the heat generated lessens the cutting forces by softening the metal and aiding in the grain boundary dislocation.

This advantage can be very dramatic, sometimes moving the possible metal-cutting speeds from a few hundred feet per minute to thousands of feet per minute!

Such is the case with Greenleaf's whisker-reinforced ceramics when applied to most forged nickel-based alloys. Optimum speeds can be achieved with temperatures exceeding 1000° Celsius.

The excellent thermal shock resistance of whisker-reinforced ceramics results in a cutting material which can be used either dry, wet or even intermittently cooled without fear of catastrophic tool failure from thermal cracking.

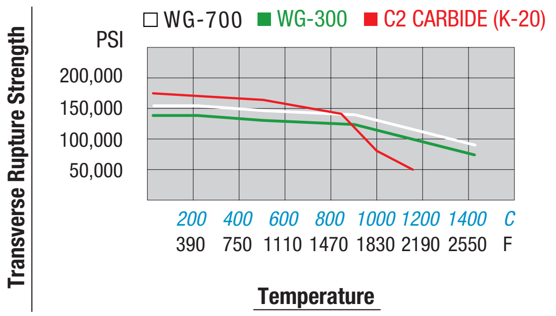
The outstanding hardness of Greenleaf whisker-reinforced ceramic inserts, combined with the high strength imparted by the reinforcing silicon-carbide whiskers, makes possible the machining of many materials previously workable only by grinding. Heat-treated alloy steels, die steels, weld overlays, and hard irons with interrupted cuts are just a few of the successful applications completed on a daily basis.

If your job is in the 45Rc to 65Rc range, chances are that Greenleaf's whisker-reinforced ceramic inserts can increase productivity and cut machining costs substantially.

Relative Strength at Elevated Temperatures

It is important to recognize that laboratory hardness and strength tests are conducted at room temperatures. Under actual cutting conditions where temperature at the tool/chip interface may reach over 1000° C, Greenleaf whisker-reinforced ceramics will retain high strength and hardness well beyond the point at which a tungsten-carbide material has softened, deformed or failed completely. Productivity advantages multiply quickly in this range of application.

Figure 5 – Relative Strength at Elevated Temperatures



Ceramic Application Guidelines

Rethink the process

The correct application of ceramic tooling on a CNC machine necessitates reprogramming of the part. Since we are doing this, we might just as well re-examine the entire process. Are we using the best geometry, largest radius, thickest insert, best tool path, etc.? When you have studied this application guide, you will be more aware of the variables and best approaches to the job using ceramic cutting tools.

Integrate the following tested methods into your programs:

Figure 6 – Ceramic Application Guidelines

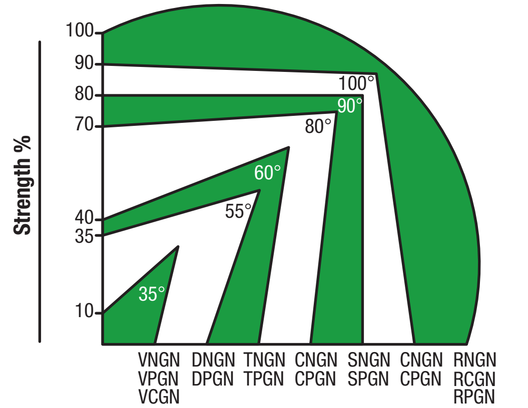
1. Use a toolholder system designed for ceramic inserts.
2. Use the strongest insert shape possible.
3. Use the largest corner radius possible.
4. Use the correct edge preparation for the application.
5. Use the thickest inserts available for roughing.
6. Use a toolholder or boring bar with the largest possible cross section.
7. Consider heavy metal or carbide bars for boring applications.
8. Prechamfer on entry and exit whenever possible.
9. Keep toolholder overhang to a minimum.
10. ***Rethink the process***

Strength Comparison of Ceramic Inserts

Use the strongest insert shape

In declining order of corner strength, the strongest inserts are: Round, 100° Diamond, Square, 80° Diamond, Triangle, 55° Diamond, and 35° Diamond. Always use the strongest possible shape to maximize corner strength and metal-removal capability.

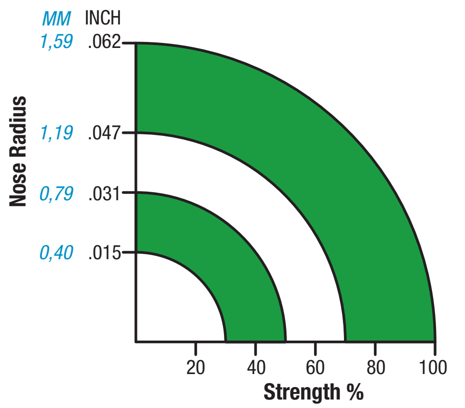
Figure 7 – Insert Shapes and Strengths



Use the largest corner radius possible

The larger the corner radius, the stronger the corner. Do not attempt to do all roughing operations with a small corner radius just because the finished fillet calls for a small radius. Use a round insert or large radius insert for roughing and change the tool for the final cuts.

Figure 8 – Relative Strength for Various Insert Radii

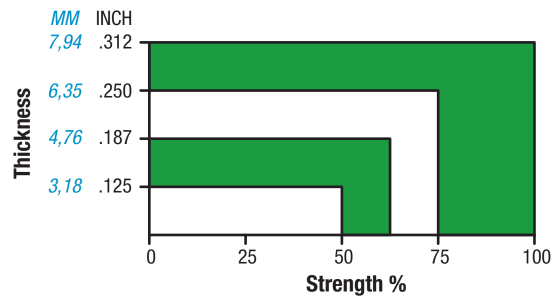


Use thick inserts for roughing

Increased insert thickness results in far better impact resistance, better heat dispersion, and longer tool life.

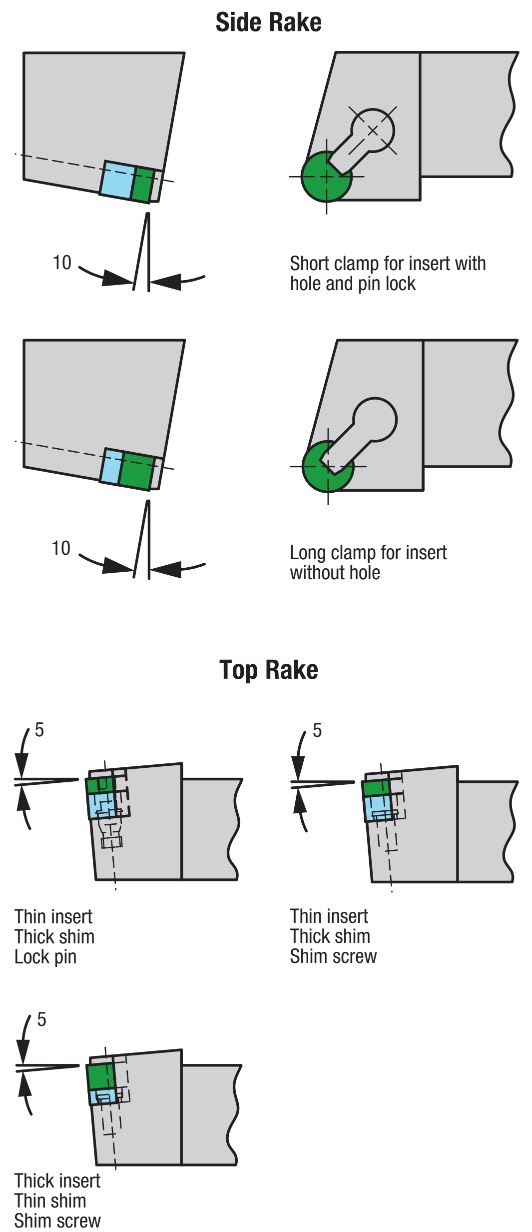
This adds to greater predictability of performance and less downtime.

Figure 9 – Relative Strength for Various Insert Thicknesses



Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

Figure 10 – Toolholder System



Use the Greenleaf toolholder system

The use of Greenleaf toolholders and accessories permits:

a) Either standard or thick inserts to be used in the same toolholder by changing shim seats.

b) The toolholders can be used for pinlock-style inserts by exchanging the shim screw for a tilt pin.

c) Alternative lengths of clamps are available with the holder being supplied standard with the large clamp to ensure good retention of ceramic inserts without holes.

d) In the case of negative-rake tooling, we have found that the normal carbide tool geometry of -5° top and side rake may be changed very advantageously to -5° top rake, and -10° side rake for materials under 45Rc hardness. Greenleaf tools for use with whisker-reinforced ceramics are illustrated in the Ceramic Toolholders in the Turning section of this catalog and have been designed to take advantage of this increased negative rake which will give longer tool life. The increased pressure associated with a greater negative rake is insignificant and not evident at the high velocity and temperatures at which these tools are used.

NOTE:
Greenleaf toolholders for v-bottom inserts are designed to take 7° side-clearance inserts as well as 11°.

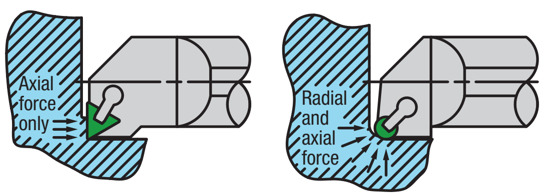
Use toolholder or boring bar with greatest cross sectional area

Stability of the tool and freedom from deflection are paramount to consistent performance. If the tool post will accept 1-1/4" *(32 mm)* shanks, do not be satisfied to take a 1" *(25 mm)* shank and shim it to suit. This is false economy.

Straight-edged inserts versus rounds

Long overhangs for tools are necessary when working with turrets in order to clear other tools. In these cases, straight-edged inserts should be applied to eliminate radial tool forces and avoid chatter.

Figure 11 – Straight-Edged Inserts vs. Round



Keep overhang to a minimum

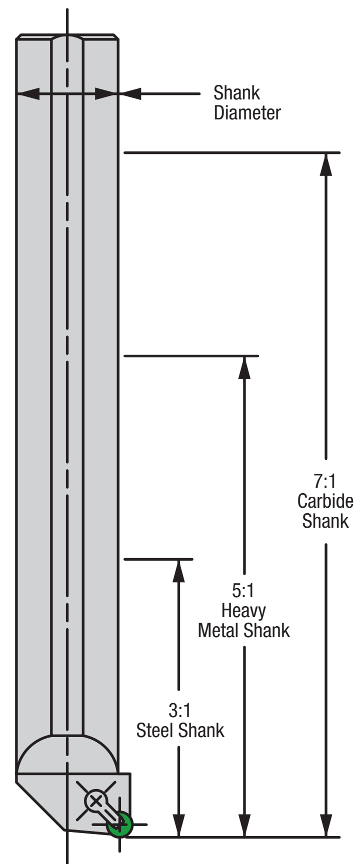
Any deflection will lead to vibrations, which are particularly damaging to ceramic tools. Unnecessary tool overhang is the principle cause of vibration. It should be noted, the force required to produce a particular deflection decreases by the cube of the overhang! That means that doubling the overhang will increase deflection eight (8) times if all other conditions are constant.

Boring bars, in particular, usually operate with much greater length-to-diameter ratios than turning tools. In this case, "heavy" metal or solid-carbide bars are often easily justified.

Solid-carbide boring bars have three (3) times the modulus of elasticity of a steel bar. This means that a carbide bar will only deflect 1/3 as much as a comparable steel bar under identical circumstances.

As a general rule, when machining nickel-based alloys, steel boring bars will give adequate performance at overhang-to-bar diameter ratios of up to 3:1. Special boring bars manufactured from "heavy" metals give an advantage over steel bars and can be used at ratios up to 5:1. Carbide boring bars extend this range to ratios up to 7:1.

Figure 12 – Shank Diameter-to-Bar-Length Ratio for Ceramic Inserted Boring Bars



Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

NOTES:

The Application of Greenleaf Advanced Ceramics

Feed and speed recommendations are expressed in the graph below *(Figure 13)*. This graph is based on empirical data gathered during extensive testing under shop conditions.

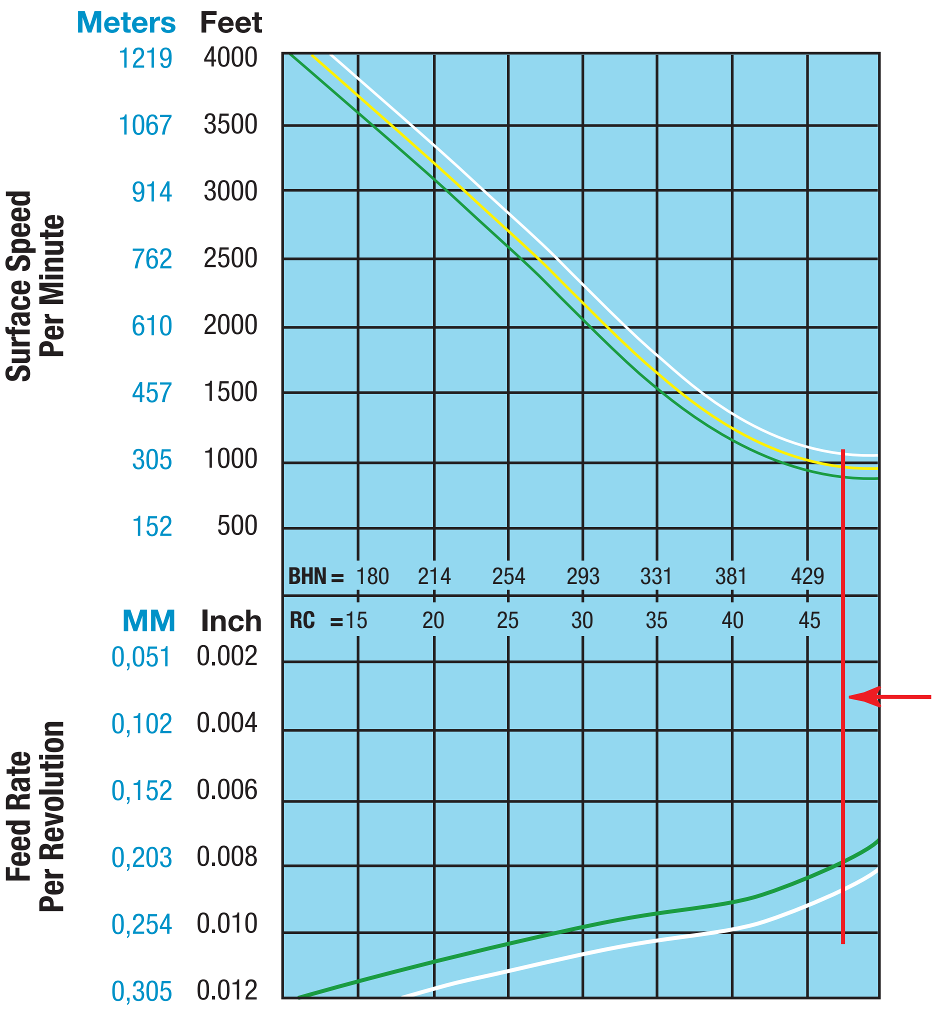
The most significant factors are the hardness of the material and the surface condition. It is on the basis of these parameters that the data are presented.

To achieve optimum cutting conditions, it is necessary to regulate not only the speed but also the feed. There must be a carefully balanced relationship between the speed and the feed. The higher temperatures required can be generated by a slower speed than the optimum, provided that the chip is thinned by reducing the feed.

Any reduction of speed from the recommended starting points without a corresponding decrease in feed results in a thicker, cooler chip and an increase in cutting forces. This may result in shortened tool life or failure by chipping and breakage.

It must be noted that thick chips provide a larger heat sink and tend to be cooler and stiffer, and that thin chips do not have sufficient heat absorbing capacity and tend to be too hot. For each metal hardness and surface condition, there is a speed and a feed at which the best temperature balance is obtained.

Figure 13 – Greenleaf Advanced Ceramics Machining Recommendations



Starting point based
on Insert
RNGN-45
(RNGN-120700)
D.O.C. = .125" (3,175 mm)

When working in Forging Scale Condition reduce speed by 15%

Example:
718 Inconel 47RC
700 SFM (213 m/min.)
.0075 IPR (0,191 mm/rev)

*Example data is for WG-300®

NOTE:
Feed for WG-300 is the same as WG-600.

Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

How to use Figure 13 graph:

1) Feed and speed are based on RNGN-45 (12 07 00) round inserts. When using inserts with weaker shapes such as triangles, etc., some reduction of feed will be required. *(Figure 21)*

2) **You must know the physical hardness of the material.**

3) The recommendations are based on an average depth of cut of .125" *(3,175 mm)*. Deeper depths will require some reduction of speed and feed, and shallower depths can be cut at elevated speeds and feeds. See *Figures 15 & 16.*

4) From the material hardness, move vertically downward to the curve and then horizontally to the left to read the recommended feed rate per revolution.

5) Whenever the recommended speed is not achievable on the machine tool, then the recommended feed must be reduced by the same percentage (%), i.e.
   Speed recommended – 2000 SFM (610 m/min.)
   Feed recommended – .010" I.P.R. *(0,254 mm)*
   Top speed on machine –
      1000 SFM (305 m/min.) = 50% of
      recommended speed then use feed of
      .005" I.P.R. *(0,127 mm)*

   The feed and speed are based upon the ability of the ceramic insert to withstand high temperatures and to run with a chip thickness which results in heat being concentrated in the shear zone ahead of the tool. This will reduce cutting pressure and minimize wear. If the speed is reduced without a corresponding reduction in feed, this effect will be lost and performance will fall off due to chipping of the cutting edge from a colder chip.

   General starting speeds should be eight times the uncoated carbide speeds and four times coated carbide speeds.

   Compared to sialon materials, speed and then feed should be increased 25% to 50%.

Rule of thumb when cutting nickel-based cast material rather than forged material:

1. Increase speed from graph recommended by factor of 2x.
2. Decrease recommended feed to one half of value.
3. Maintain a depth of cut of less than .060" *(1,5 mm)* for an RNGN-45 (12 07 00) insert.
4. Use plentiful supply of coolant.

Aged and solution treated nickel-based cast alloys – use same parameters as forged materials, except less than .075 *(2 mm)* DOC for an RNGN-45 (12 07 00) insert.

Anticipated Tool Life

For programming purposes, it is useful to have a starting guideline for anticipated tool life. We present here some approximate values which are based upon actual experience at the maximum recommended depth of cut (1/4 of insert diameter) and at the speed given in the graph. It should be noted that these speeds are up to eight times those used with uncoated carbide tools. Even at the conservative starting values for tool life per corner given, the actual volume of metal removed per index also will be eight times that produced in the same period of time with carbide tools.

Another way of stating this is – five minutes of tool life with a Greenleaf advanced ceramic is equivalent, in work produced, to 40 minutes of life with a carbide tool! *In fact, a carbide tool will never last 40 minutes.*

Figure 14 – Anticipated Tool Life

Starting Points for Time in Cut

Round Insert	Life per Index
.250" *(6,3 mm)*	3 min.
.375" *(9,5 mm)*	4 min.
.500" *(12,7 mm)*	5 min.

Speed and Feed vs. Depth of Cut for Round Inserts

The round insert behaves differently from a straight-sided insert as depth of cut is changed.

Because the chip produced by a round insert is crescent shaped and reduces in thickness toward the finished surface, as the depth is reduced, the thinning chip, combined with increased lead angle, gives a significant drop in pressure at the workpiece surface/tool interface. This means that both speed and feed can be increased without detriment to tool life as depth reduces.

Speed and feed should be increased by a like amount percentage (%) to achieve the best result.

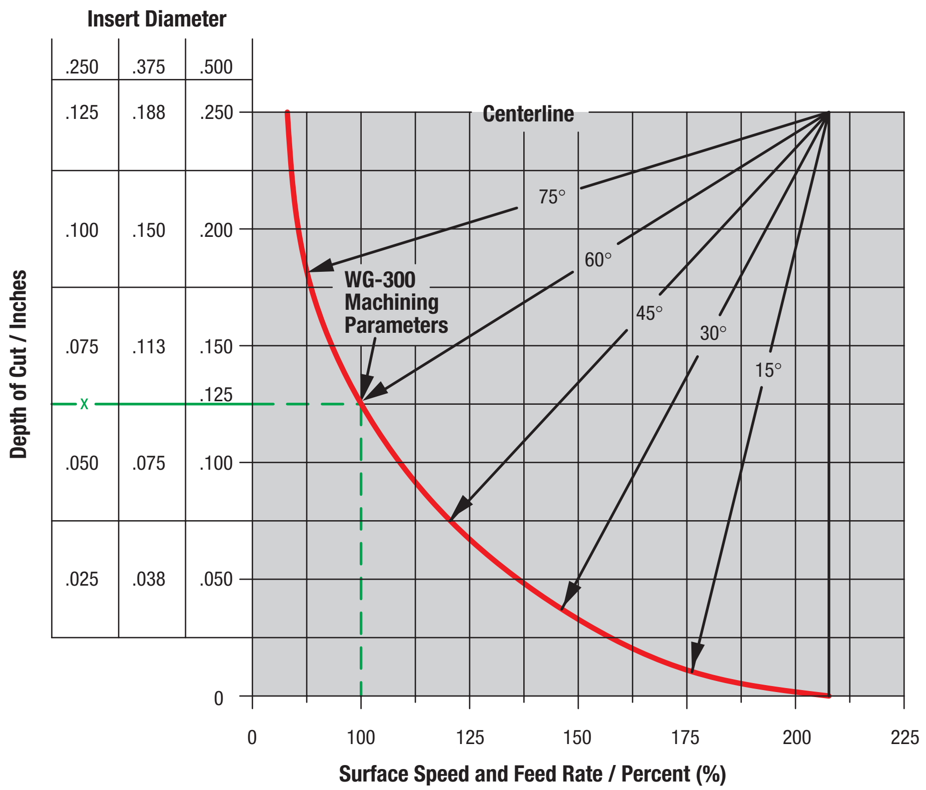
We can now refer to the graph ***Surface Speed and Feed Rate/Percent (%) Versus Depth-of-Cut of Radii***. *(Figure 15)*

1) It will be seen from reference to this chart that the speed/feed graph is set up on .125" *(3,18 mm)* depth of cut, using a .250" *(6,35 mm)* radius tool or .500" *(12,7 mm)* round insert. This results in a depth equal to the 60° mark (90° being half the diameter or radius) and gives a reasonably conservative starting point for most Inconel 718 applications. A slightly shallower depth at around the 45° line will usually give the best tool life in exchange for a small decrease in metal removal rates.

2) Finishing cuts are usually taken at depths of cut less than those set up as reference points on the graph. *(Figure 13)*

When using round inserts, it will be possible to make substantial increases in both feed and speed beyond the values given in the graph if the depth of cut is less than "**X**" value.

Figure 15 – Surface Speed and Feed Rate/Percent (%) vs. Depth of Cut of Radii (IMPERIAL)



Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

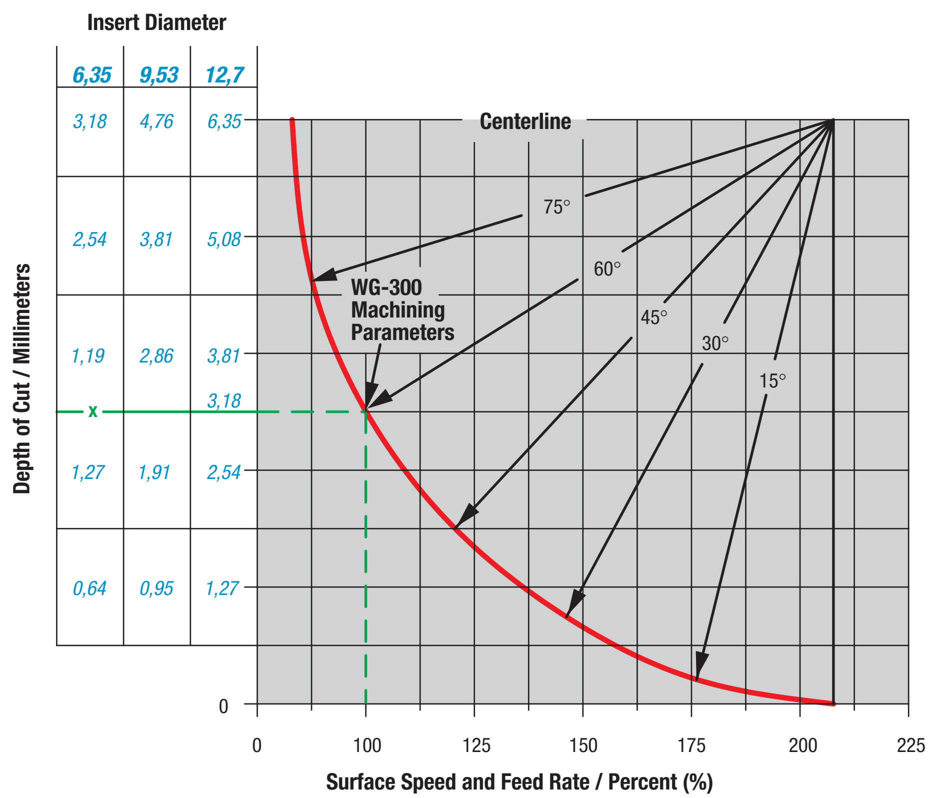
As always, the rule should be applied that feed and speed will be increased together and by the same percentage (%). Failure to regard this rule will result in cooler or hotter interface temperatures with corresponding drop-off of tool performance.

When taking a depth of cut that is less than the graph value for “X,” refer to *Figures 15 and 16 (Imperial and Metric)*. Select the insert diameter that best suits the application and provides the depth-of-cut capability that is closest to the graph value of “X,” then adjust the speed/feed according to the chart.

For example:
Column 1 is for .250" *(6,35 mm)* round insert or .125" *(3,18 mm)* radius.
Column 2 is for .375" *(9,53 mm)* round insert or .188" *(4,76 mm)* radius.
Column 3 is for .500" *(12,7 mm)* round insert or .250" *(6,35 mm)* radius.

1. Select approximate desired depth of cut *Example:* .500" *(12,7 mm)* diameter round at .050" *(1,27 mm)* depth of cut is bottom box of column 3.
2. Follow the line to the right until it intersects the heavy curved line.
3. Follow the line vertically downward to the bottom scale and read value of 137% (midway between 125% and 150%).
4. You may increase the speed and feed values in the graph *(Figure 13)* by 37% for this cut.

Figure 16 – Surface Speed and Feed Rate/Percent (%) vs. Depth of Cut of Radii (METRIC)



Lead-Angle Effect on Round Versus Straight-Edged Inserts

To emphasize the advantage of using round inserts, let us look at a comparison between the chip-thinning effect obtained at various depths on a round insert compared to the lead angle needed to get the same effect with a straight-edged insert.

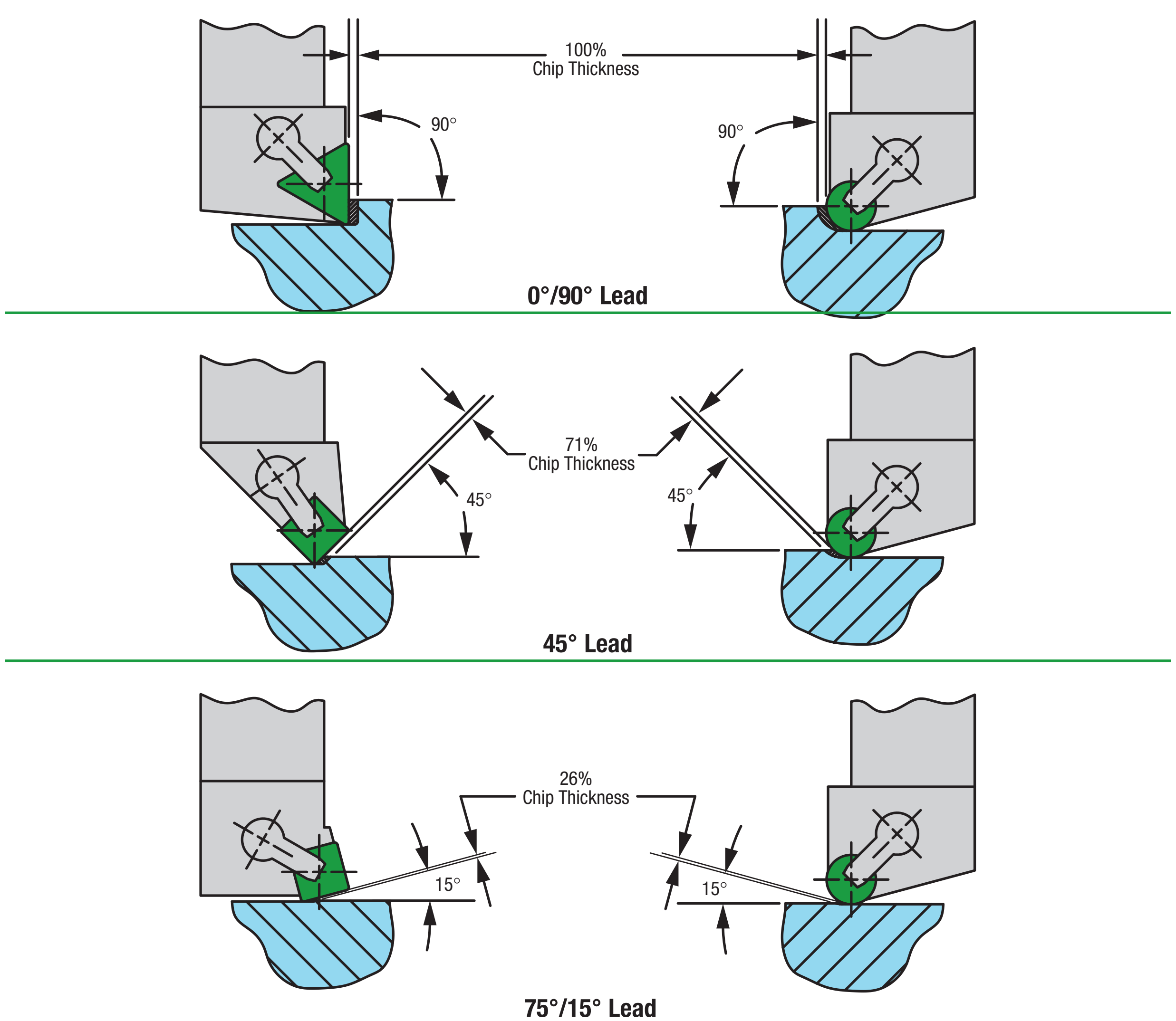
If we assume operations with lighter depths of cut, then a round insert engaged up to 45° or halfway along the available cutting edge and advancing at .010" *(0,25 mm)* per revolution will produce an actual maximum chip thickness of 71% or .007" *(0,18 mm)*.

The chip actually thins from this point gradually towards the finished surface. To thin the chip to .007" *(0,18 mm)* with a straight-edged insert requires a lead angle of 45°, which is about the maximum lead angle practical.

Beyond this point, the round insert can be used very easily at 30° or less. To get the same chip-thinning effect from a straight-edged insert requires 60° or more of lead angle which is just not practical.

In summary, the high lead-angle effect with corresponding reduction of pressure, especially at the depth-of-cut line, is more practical with round inserts.

Figure 17 – Lead-Angle Effect on Round vs. Straight-Edged Inserts and the Theoretical Chip Thickness



Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

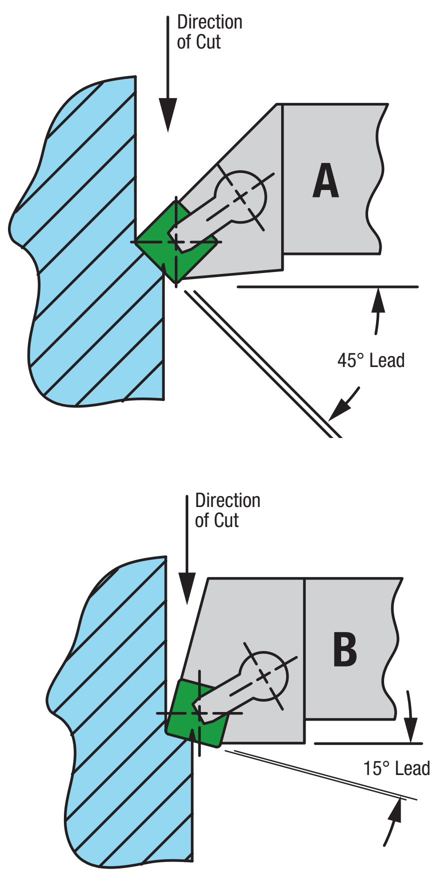
Lead-Angle Effect with Other Than Round Inserts

In the cutting of nickel-based alloys, the lead angle employed is of significance. Larger lead angles reduce chip thickness, improving tool life and surface finish.

Figure 18 shows the change in lead-angle effect. It may be necessary to design tooling which does not stand on traditional carbide values to get optimum performance.

It should be noted that example (A) will produce more pressure on the part piece and may not be feasible on thin sections.

Figure 18 – Lead-Angle Effect



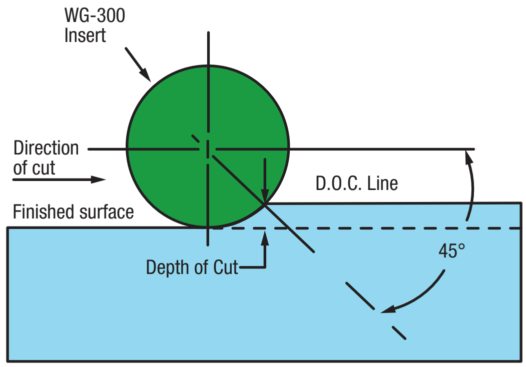
Recommended Depth of Cut for Round Inserts

For best results, there must be a planned relationship between the insert radius and the proposed depth of cut, if the notching effect at the depth-of-cut line is to be minimized.

It will be seen by reference to the illustration, *(Figure 19)* that there will be a sudden decrease in lead-angle effect beyond a given depth on a given insert radius. This point lies at the intersection of a line drawn at 45° from the center of the insert. The effect of the decreasing lead angle is increased cutting pressures. The deeper the cut with a round insert beyond this point, the greater the depth-of-cut notching.

It is often a clear advantage in nickel-based alloys to make light cuts with relatively large-diameter round inserts. Here are the depths of cut that produce the optimum relationship on given insert sizes.

Figure 19 – Recommended Depth of Cut for Round Inserts



Insert Radius		Optimum Depth of Cut	
Inches	***Millimeter***	**Inches**	***Millimeter***
.125	*3,18*	.037	*0,93*
.187	*4,76*	.052	*1,40*
.250	*6,35*	.073	*1,86*
.312	*7,94*	.092	*2,33*
.375	*9,53*	.110	*2,79*
.437	*11,11*	.128	*3,26*
.500	*12,70*	.147	*3,72*

Of course, depths lighter than those given will increase tool life at some penalty of metal removal rate. Refer back to *Figures 15 and 16* where we discuss the use of lighter depths of cut combined with increased feeds and speeds.

Recommended Depth of Cut for Insert Nose Radii

It is very important in roughing operations with round inserts to leave the recommended amount of stock for finishing with straight inserts.

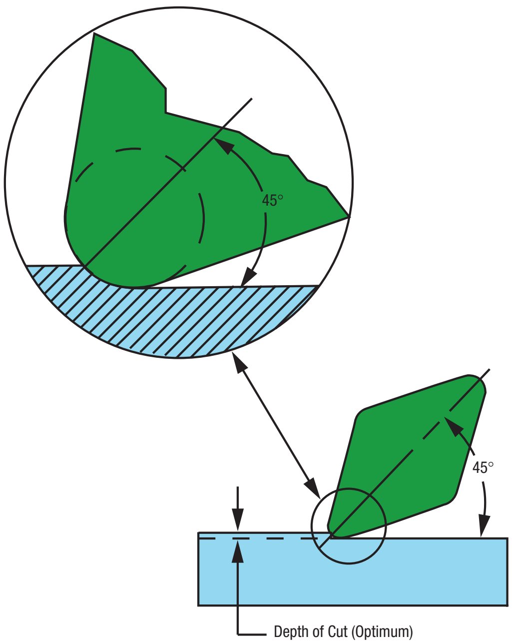
For maximum tool life when using straight-edged inserts with corner radii as opposed to round inserts, a similar effect as described with round inserts is obtained. In this case, the allowable depths of cut are related to the radius and not the insert size, assuming that the depth of cut being attempted is relatively light, such as in finishing operations.

The table to the right shows the *optimum* depth of cut at which maximum tool life (minimum notching) should start.

The accuracy for roughing therefore becomes more important, and the depth of recommended passes for finishing must be as illustrated in *Figure 20*:

A large radius, while having reduced notching tendency, will sometimes be impractical because of radius requirements on the workpiece. Larger insert radii may also cause the deflection of thin sections as a consequence of larger radial forces acting between the tool and the workpiece. A compromise between notching and these factors must often be made. However, it should be remembered that regardless of geometry, cutting force will be lower when using WG-300 high-speed techniques to plasticize the material.

Figure 20 – Recommended Depth of Cut for Insert Nose Radii



Insert Radius		Optimum Depth of Cut	
Inches	***Millimeter***	**Inches**	***Millimeter***
.015	*0,38*	.0046	*0,12*
.031	*0,80*	.0092	*0,23*
.048	*1,21*	.0139	*0,35*
.063	*1,59*	.0183	*0,47*
.094	*2,38*	.0275	*0,70*
.125	*3,18*	.0370	*0,93*

Recommended Percentage (%) Reduction of Feed Per Revolution for Other Than Round Inserts (Except Grooving)

When applying inserts other than rounds, there are a number of variables present which have a direct effect on the ability of the insert to tolerate the cutting loads. These variables include the specific corner or nose radius of the insert, the included angle of the corner (i.e. Triangle, Square or Diamond shape), the lead angle of the cutting tool to be used, the depth of cut selected, and the part piece being machined due to entry and exit angles.

It is possible to look at all of the variables and then apply percentage (%) reduction factors to the recommended feed of the graph *(Figure 13)* to compensate for them.

In all cases the speed should be maintained as recommended.

Figure 21 – Feed Adjustment for Straight-Sided Inserts

Nose radius	1/64	1/32	3/64	1/16	3/32	1/8
ANSI designation	1	2	3	4	6	8
ISO designation	04	08	12	16	24	32
Inches	.015	.031	.047	.062	.094	.125
mm	*0,4*	*0,8*	*1,19*	*1,59*	*2,38*	*3,18*
Reduction percentage	**19%**	**16%**	**13%**	**10%**	**5%**	**2%**
DOC/inches	0-.050	.125	.250	.375	.500	.750
DOC/mm	*0-1,27*	*3,18*	*6,35*	*9,53*	*12,7*	*19,05*
Reduction percentage	**5%**	**8%**	**13%**	**16%**	**18%**	**20%**
Lead angle	0° & -5°	15°	30°	45°	60°	75°
Reduction percentage	**18%**	**17%**	**15%**	**12%**	**8%**	**5%**
Included angle	35°	55°	60°	80°	90°	100°
Reduction percentage	**17%**	**13%**	**10%**	**6%**	**4%**	**2%**
Part diameter/inches	0-5	10	20	30	40	50
Part diameter/mm	*0-127*	*254*	*508*	*762*	*1016*	*1270*
Reduction percentage	**18%**	**14%**	**10%**	**6%**	**2%**	**0%**

Select and add five reduction percentage (%) factors and subtract from feed rate of graph in *Figure 13*.

Example: CNGN 432 (120408)

Nose Radius 0.031 *(0,8 mm)*	=	16%
Assumed DOC .125 *(3,18 mm)*	=	8%
Lead -5°	=	18%
Included angle (80°)	=	6%
Assumed part dia. 20" *(508 mm)*	=	10%
		58%

58% reduction of feed rate recommended from graph *(Figure 13)*.

Greenleaf®
800-458-1850

Theoretical Surface Roughness vs. Feed and Insert Radius

Rethink the process

The quality of the finished surface is affected directly by the radius of the tool and the feed rate at which the tool is advanced. The larger the radius, the faster the tool may be fed for a given degree of surface finish. The finish is usually stated as surface roughness in micro inches or micro meters.

The chart can be used to determine the combination of tool radius and feed rate for various roughness measurements from 8 micro inches (0,2 micro meters) to 250 micro inches (6,3 micro meters). *Using less feed rate than necessary results in premature tool wear causing bad finish, taper, and size problems.*

Figure 22 – Theoretical Surface Roughness

Roughness average Micro inches (Ra) *Micro meter (µm)*		8 *0,2*	16 *0,4*	32 *0,8*	63 *1,6*	80 *2,0*	100 *2,5*	125 *3,1*	150 *3,8*	200 *5,0*	250 *6,3*
	Nose radius	Feed rate per revolution									
Inches	.0156	.002	.0025	.004	.0055	.0065	.007	.0075	.008	.010	.011
mm	*0,40*	*0,05*	*0,06*	*0,10*	*0,14*	*0,17*	*0,18*	*0,19*	*0,20*	*0,25*	*0,23*
Inches	.0313	.003	.004	.0055	.008	.009	.010	.011	.012	.014	.016
mm	*0,79*	*0,08*	*0,10*	*0,14*	*0,20*	*0,23*	*0,25*	*0,28*	*0,30*	*0,35*	*0,41*
Inches	.0469	.0035	.005	.007	.0095	.0105	.012	.013	.015	.017	.019
mm	*1,19*	*0,09*	*0,13*	*0,18*	*0,24*	*0,27*	*0,30*	*0,33*	*0,38*	*0,43*	*0,42*
Inches	.0625	.004	.0055	.008	.011	.0125	.014	.015	.017	.020	.022
mm	*1,59*	*0,10*	*0,14*	*0,20*	*0,28*	*0,32*	*0,35*	*0,38*	*0,43*	*0,50*	*0,56*
Inches	.0938	.0045	.007	.009	.013	.015	.017	.019	.021	.023	.026
mm	*2,38*	*0,11*	*0,18*	*0,23*	*0,33*	*0,33*	*0,43*	*0,43*	*0,53*	*0,58*	*0,66*
Inches	.125	.0055	.008	.011	.016	.018	.020	.022	.024	.027	.031
mm	*3,13*	*0,14*	*0,20*	*0,23*	*0,41*	*0,45*	*0,50*	*0,56*	*0,60*	*0,69*	*0,79*
Inches	.1875	.007	.0095	.0135	.017	.021	.025	.027	.030	.034	.040
mm	*4,76*	*0,18*	*0,24*	*0,34*	*0,43*	*0,53*	*0,64*	*0,69*	*0,76*	*0,86*	*1,02*
Inches	.250	.008	.011	.016	.022	.025	.027	.031	.034	.040	.044
mm	*6,35*	*0,20*	*0,28*	*0,41*	*0,56*	*0,65*	*0,69*	*0,79*	*0,86*	*1,02*	*1,12*

Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

The Effect of Increased Clearance on Tool Life

Under normal tool wear circumstances, a tool is said to be "worn out" when the flank wear has developed to the point that surface finish has deteriorated outside of acceptable limits. This is determined when the width of the wear land has decreased clearance and increased heat and pressures in the tool workpiece interface area to the point that further use will lead to complete failure of the tool by severe chipping or catastrophic breakage. On nickel-based alloys, the depth-of-cut notch may become too severe before this flank wear has progressed to that limit.

However, assuming that notching is under reasonable control, tool life, as judged by wear land development, can be prolonged by increasing the tool side clearance. With "normal" hot-pressed or cold-pressed ceramic and composites, this clearance is usually limited to about 7° since the materials are too brittle and friable to permit a larger angle. Greenleaf advanced whisker-reinfored ceramics do not suffer from this problem, and large clearances can be used because of the greater edge strength. To view the difference that, for example, an 11° clearance makes compared to a 7° clearance, refer to the illustration. *(Figure 23)*

It will be seen that with a 7° clearance angle, .003" *(0,07 mm)* of material will be worn from the insert to produce a .025" *(0,64 mm)* wear land, whereas .005" *(0,12 mm)* of material must be worn from an 11° clearance insert to produce the same amount of wear land. This will then equate to increased tool life between indexes.

It is recommended that tooling be carefully evaluated on all operations relative to using clearance angle inserts. In most cases, investments in new tools can be justified.

Remember, a Greenleaf tool is designed to take 7° side-clearance inserts as well as 11°.

Figure 23 – The Effect of Increased Clearance on Tool Life

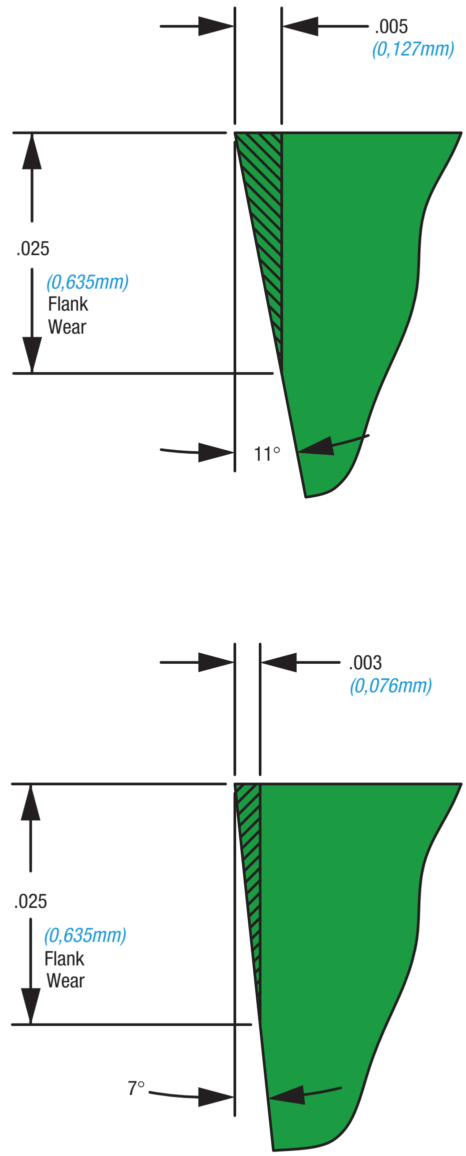
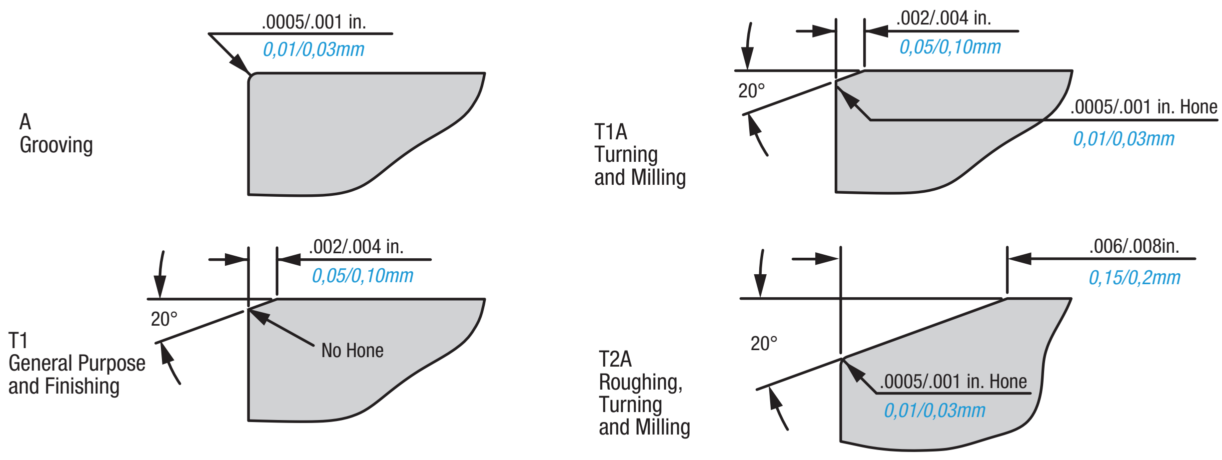


Figure 24 – Standard Edge Preparations



Edge Preparation For Nickel-Based Alloys

Greenleaf advanced whisker-reinforced ceramics have inherently strong cutting edges, and it is recommended to use them without hones, except when making heavy roughing cuts or in scale conditions. A sharp cutting edge is a clear advantage in finishing operations to avoid "burnishing" and "smearing."

For most nickel-alloy operations involving light roughing and finishing in clean material, the T1 edge preparation should be standard. *No hone is used.*

In ceramic tool applications, edge preparation is critical to tool life and surface integrity. Edge preparations are used to change the shear forces at the edge to compressive forces, thereby guarding against chipping and breakage.

If we use a wide negative land on a tool and use a very light feed rate, we have in fact changed the geometry of the tool by doing all of the cutting on the land itself. This is incorrect use of a negative land giving rise to a new set of problems. We highly recommend using a T1 edge preparation (.002"-.004" x 20°) *(0,05 mm-0,10 mm x 20°)* for increased strength with minimum "smearing" effect during finishing operations. (See *Figure 24* for more details.) It may be necessary to add a hone (T1A) when light scale conditions or minor interruptions are present.

For milling and roughing other than very heavy-duty machining, a T2A edge prep should be used (.006"-.008"x 20°+.0005" hone)
(0,15 mm-0,2 mm x 20°+ 0,013 mm hone).
For most aircraft-type work, T1 and T2A should be the only edge conditions required.

Remember; *never* use a honed edge unless it has been shown conclusively that a hone is required. This should be in very few applications. *Always* start by testing the T1 edge preparation.

The exceptions to the general stated rule would be:

Grooving

Because a grooving tool moves constantly forward into clean material, there is no notching problem in normal usage. Also, a grooving tool is usually a relatively fragile tool especially in the narrow-width grooves found in jet engine work.

For these reasons, we highly recommend that grooving tools do not have a negative land. This will keep cutting forces to a minimum.

For grooving use an "A" hone only.

Heavy Interruptions

In severely interrupted cuts, we need to keep the cutting edge in compression to avoid shear forces. This will reduce chipping and breakage. The chip width is smaller than the negative land width.

Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

Coolant

The recommendation to use flood coolant on whisker-reinforced ceramics may appear to be something of a paradox. "How to Use the Properties of Greenleaf Advanced Whisker-Reinforced Cermics", page 6, introduces the concept that heat produced by the application of WG-300 plasticizes the metal in front of the cutting edge. This action is desirable and heat is used advantageously. However, it is also desirable to cool the whole operation by flooding it with coolants—thus, the paradox.

Whisker-reinforced ceramics are a good heat conductor compared to ordinary ceramic materials. The heat is pulled away from the tool/workpiece interface into the body of the insert where the coolant can help to maintain a lower tool temperature. We can lower the temperature of the chip with coolant after it has formed and make it more manageable.

Coolant will also help keep the part piece temperature stable to aid in size control and reduce distortion. Use coolant liberally at all times. Unlike ordinary ceramics, the whisker-reinforced grades will not suffer breakage or cracking from intermittent coolant use.

The coolant does not decrease the temperature in the interface area; however, coolant often doubles tool life.

It is important to use clean coolant. This is not a problem when a central coolant system is used. With a stand-alone machine, the coolant must be checked very closely. Water evaporates faster than oil at these high temperatures. Adding more coolant will increase the soluble oil content, which leads to smoking, less cooling effect and shorter tool life. Contamination of the coolant from any material such as cigarette butts, coffee, etc., has proven very detrimental and should be monitored.

High coolant pressure on nickel-based alloys is not as important as volume. A minimum of a 3/8" *(10 mm)* inside diameter pipe is recommended.

The coolant must be directed exactly on the cutting area without any interference from clamps, screws, or otherwise. Oil-based, water-soluble, emulsion-type coolants have proven to be the best.

The use of straight oils is to be avoided since the hazards of oil smoke and fire exist.

***Figure 25* – Coolant**

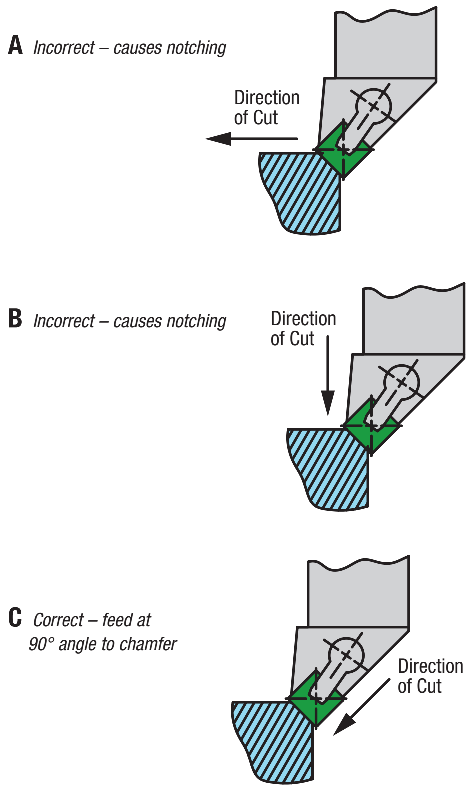
Notching and Correct Tool Path

Of all the precautions that can be taken to reduce or eliminate notching, none are as important as programming the most desirable tool path. It is very important that machine programmers, along with operators and tool engineers, are aware of the programming options. We will examine a series of circumstances that represent standard procedures for carbide tools but produce rapid failure in a notch-sensitive, ceramic material.

Pre-Chamfer Parts Whenever Possible

Pre-chamfering trues the part and ensures a progressive entry onto a true running surface. It also provides a regressive exit from the part and in both cases protects the cutting edge from damage. When using separate pre-chamfer operations as illustrated (Figure 26), the direction of feed is important to eliminate notching. Moving on a single axis, as in examples **A** and **B**, will cause notching. The direction should be at 90° to the chamfer as shown in example **C** to eliminate notching and increase tool life.

Figure 26 – Chamfering Techniques



The Chamfer Ramp Approach

In the illustration *(Figure 27)* which shows an actual operation on a jet engine rotor, we can see that feeding straight in will produce rapid notch wear. This notch will become a stress-concentration point leading to early failure.

A simple change in the programming of the part *(Figure 28)* can accomplish chamfering and facing of the part effectively in one continuous operation without any measurable difference in cutting time. This eliminates the separate pre-chamfer operation.

It is important that the program provide a "continuous move" around the part-piece edge. This will keep the material ahead of the cutting edge in a plasticized state, which is desirable for ceramic cutting methods. Another benefit is the elimination of the burr normally created with two operations, i.e. chamfer after or before the turning or facing operation.

Figure 27 – Straight Facing

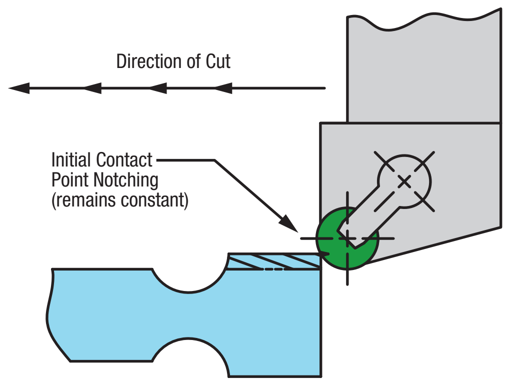
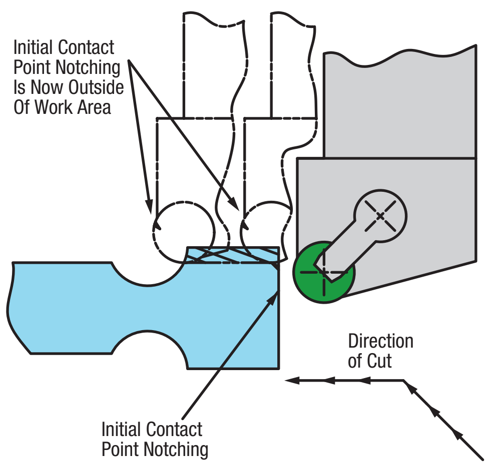


Figure 28 – Chamfering and Facing



Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

continued

The technique of generating a chamfer with the same tool used for the turning operation is valid and equally effective in terms of enhanced tool life with any shape of insert, any lead-angle tool and any given insert-corner radius.

In *Figure 29* we show a roughing operation using a square insert. Here we have programmed a 45° move to pre-chamfer the corner prior to the turning operation. This is done in one continuous motion with the 45° move transitioning into the straight turning. In this way, the section of insert in initial contact with the junction of two work-hardened surfaces is now outside the cut path. This will greatly reduce further notching tendencies.

The second example *(Figure 30)* shows a light finishing cut working on the radius of a tool. Once again, the 45° approach to the finish turned surface will reduce any notching effect initiated by the first contact. This programming approach can be used to leave a chamfer on the corner of the workpiece as well as either a radius or a sharp corner. *(See Figure 31)*

Chamfer advantages:

1. Increased tool life
2. No deburr time
3. Chips cannot hang up
4. Higher safety factor

Figure 29 – Producing a Sharp Corner

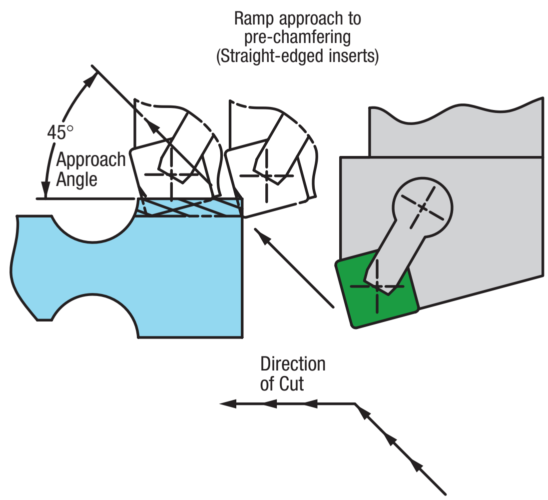


Figure 30 – Producing a Sharp Corner

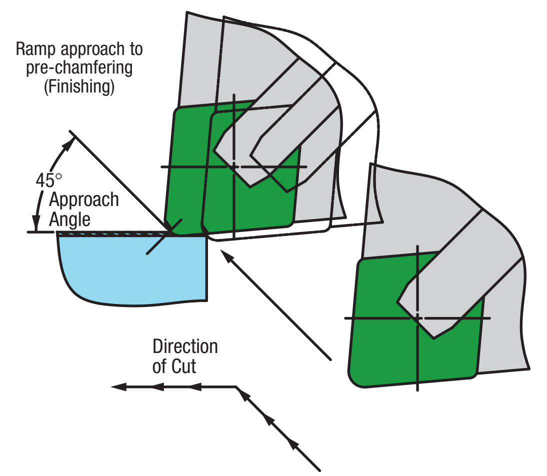
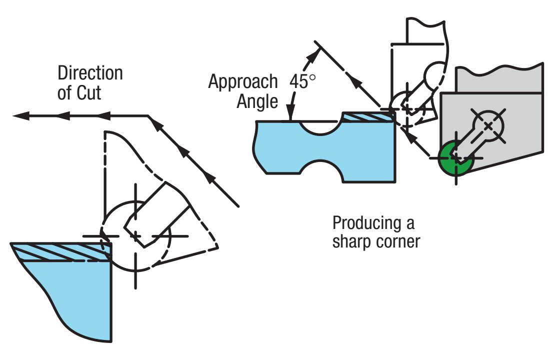


Figure 31 – Ramp Approach for Entry of Nickel-Based Alloys (Round Inserts)



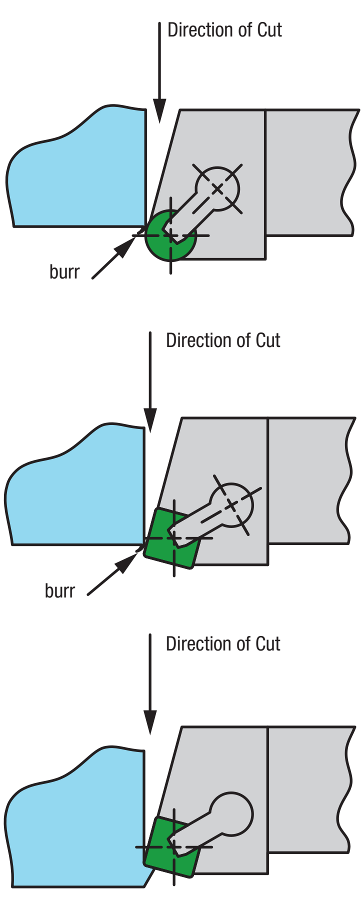
To Exit a Cut

Potential problems exist on exiting the cut if a chamfer preparation has not been made. If the exit is made on a sharp corner, on high-nickel materials in particular, a burr will result.

The burr will tend to constantly deflect or roll over and cause chipping or breakage of the cutting edge upon exit. In addition, the burr needs to be removed by a secondary operation.

The problem described tends to be more pronounced when cutting at high speeds since high heat is maintained ahead of the tool. This will mean that the material is in a more plastic condition and the rollover tendency is greater. Pre-chamfering helps correct this problem as shown. *(Figure 32)*

Figure 32 – Pre-Chamfer to Eliminate Burrs



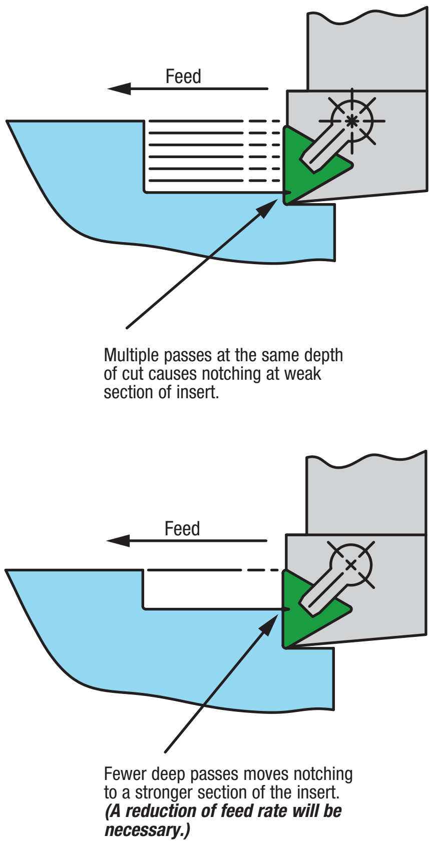
Programming Alternatives for Roughing Operations

Avoid or reduce multiple passes by taking deeper depths of cut

The strength of Greenleaf whisker-reinforced ceramics will enable much greater depths of cuts than other ceramic materials. For example, when turning with a Triangle or Diamond insert, take the greatest depth possible, even to the extent of 1/2 of the cutting edge. This not only reduces the number of passes required, it also will place any notch formed in a stronger section of the insert, leaving the tool radius area often unscathed and available for subsequent finish operations. *(Figure 33)*

A reduction of feed rate will be necessary in this case, and the feed rate recommendation chart should be used *(Figure 21)*.

Figure 33 – Rethink Depth of Cut



Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

Preserving Tool Life

When a large amount of stock has to be removed, it is often done by taking multiple passes at the same depth of cut. *(Figure 34)* This is not a good practice. A very rapid development of severe notching will result since the same point on the cutting edge is subjected to the depth-of-cut line. Consequently, many indexes are required, escalating costs due to downtime and tool costs.

Vary the depth-of-cut contact point at the workpiece/ insert interface. This can be best accomplished by two techniques:

Variation in the depth of cut from pass to pass

Gradually decrease the depth of cut per pass. This may take a very small amount of time but will be more than compensated for by increased tool life, less indexing of the insert, and less downtime. *(Figure 35)*

Ramping

Of all the techniques readily available on a CNC machine, “ramping” has proven to be the most important. By gradually feeding out while traversing the work, depth-of-cut notching can be, for all practical purposes, eliminated. The next cut is then programmed at a constant depth since the surface itself is now ramped. A similar effect is achieved. *(Figure 36)*

Figure 34 – Multiple Passes at the Same Depth of Cut

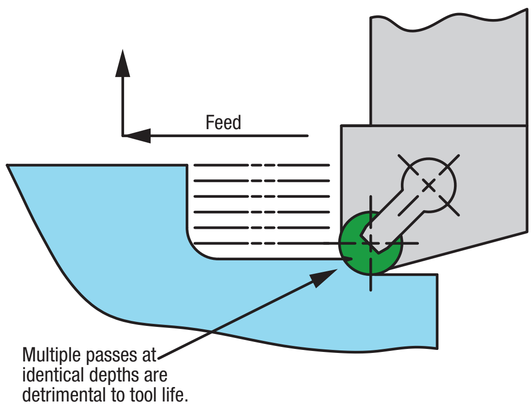


Figure 35 – Multiple Passes at Varying Depths of Cut

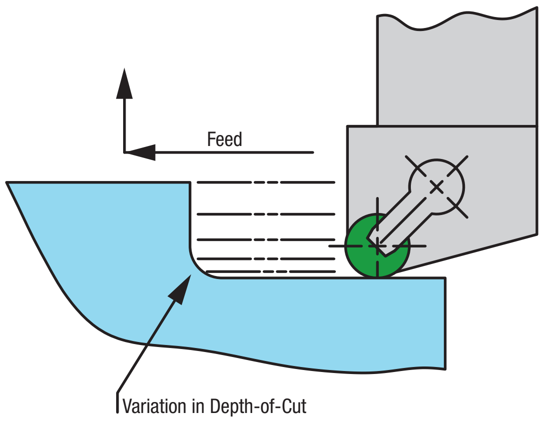
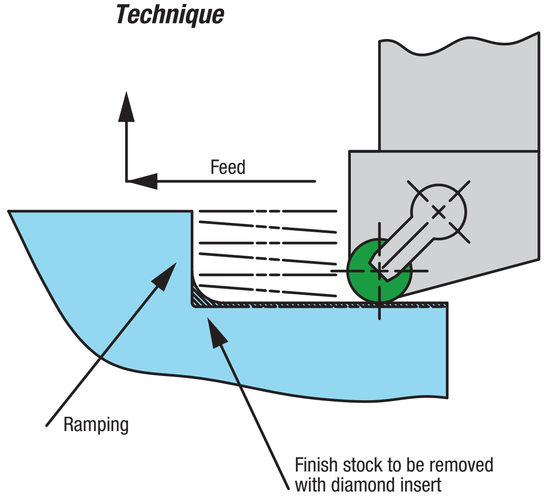


Figure 36 – Multiple Passes Using Ramping Technique

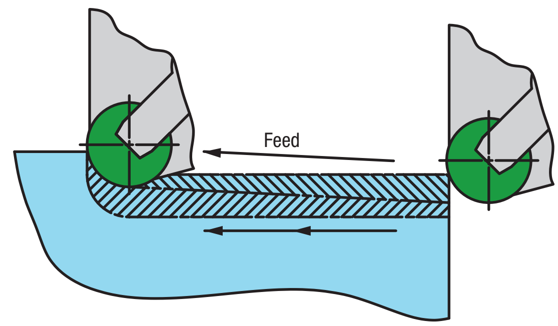


Ramping with negative round inserts

The ramp **must** start out with a deep cut, then the depth of cut must diminish. This constantly lifts the insert higher and more out of the cut, creating a ramp. The second cut is programmed straight and in the same direction, effectively removing the ramped surface left by the first cut. *(Figure 37)*

Tool life on the first cut is longer than on the second since the damaged cutting edge from the work-hardened surface is lifted out of the cut. Tool life on the second cut is shorter since the damaged cutting edge at the depth-of-cut line is buried more and more as it continues cutting straight and the ramp gets higher. However, tool life in both described ramped cuts is longer than in straight cuts.

Figure 37 – Ramping/Negative Inserts/RNGN

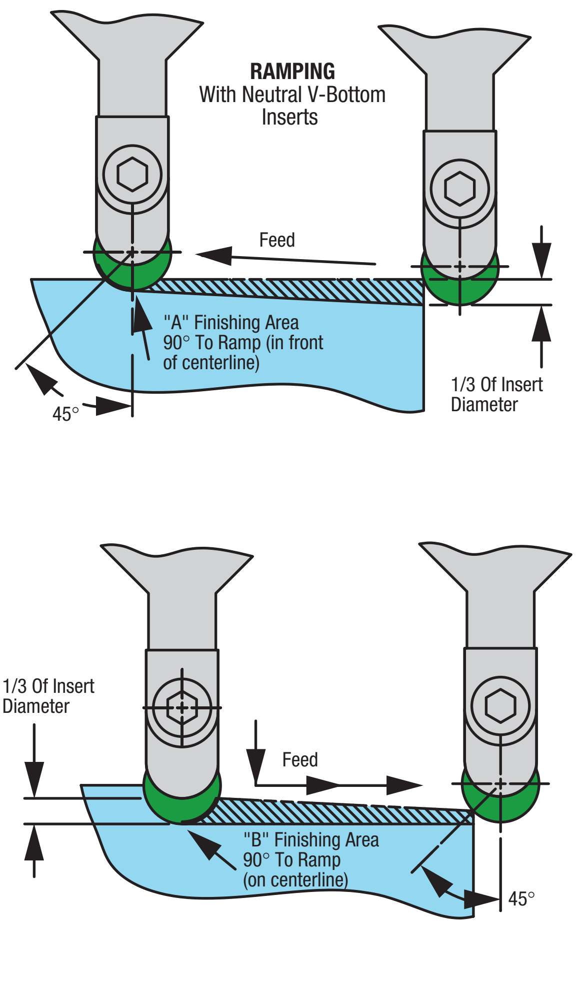


Ramping with positive round inserts

When using RPGN or RCGN inserts, ramping can be done in both directions without indexing *(Figure 38)*. Area “B,” which is the bottom of the inserts, is constantly lifted out of the cut on the first pass, and the insert finishes with area “A”. The second pass in the opposite direction will then use area “B” for finishing.

The above is not possible if the ramping is started from the lesser depth of cut then moves to the deeper depth of cut. Ramping is always better from a deep to a shallow depth.

Figure 38 – Ramping/Positive Inserts/RPGN-RCGN

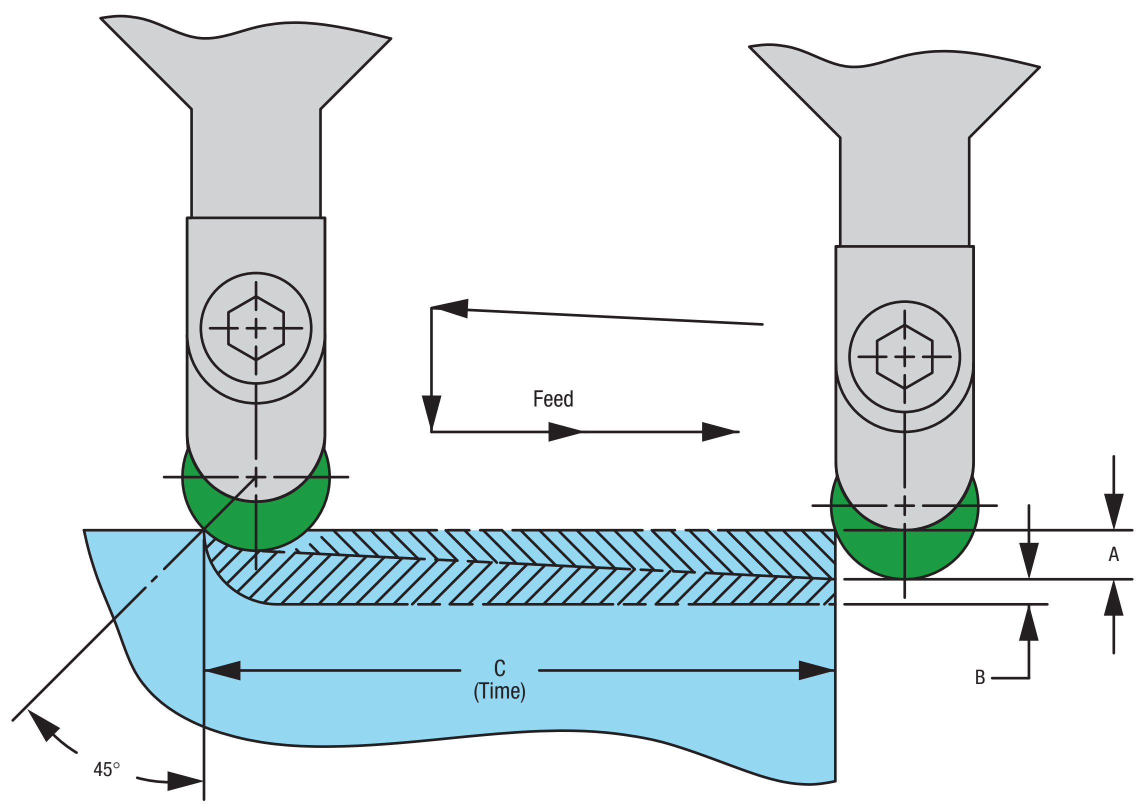


Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

To Optimize Tool Life in a Ramping Mode

The time "C" is a maximum value in minutes. The actual length of the cut represented by "C" will vary with part piece diameter. The smaller the diameter of the workpiece, the longer the length of cut. We suggest that time be limited to approximately five minutes for a .500" diameter *(12,7 mm)* insert. *(Figure 39)*

Figure 39 – Optimized Ramping Technique



DIAMETER		"A"		"B"		"C"
inches	*mm*	inches	*mm*	inches	*mm*	minutes
.250	*6,3*	.080	*2,0*	.040	*1,0*	3
.375	*9,5*	.120	*3,0*	.060	*1,5*	4
.500	*12,7*	.160	*4,0*	.080	*2,0*	5

To Optimize Ramping Technique *(Figure 40)*

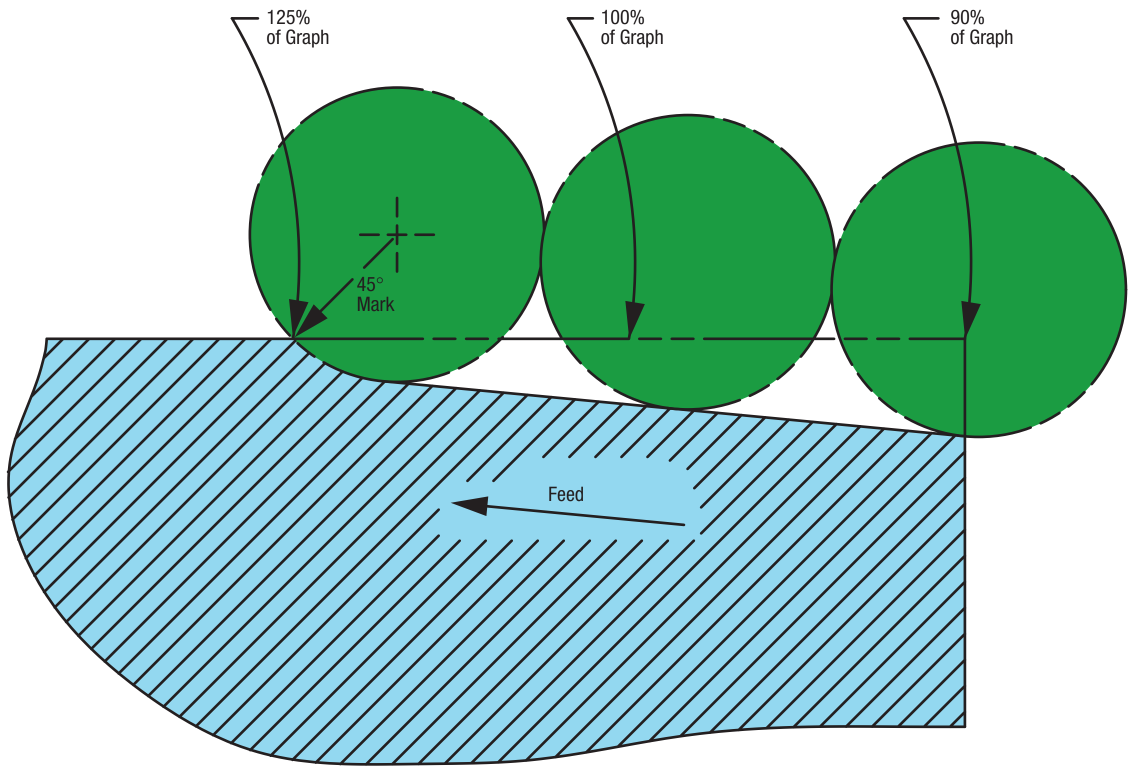
a) Using a .500 diameter *(12,7 mm)* round insert, select recommended feed and speed from graph *(Figure 13)*. This will equal 100% of feed, speed and a .125" *(3,18 mm)* depth of cut at the mid-point of the ramp.

b) Start the ramping cut at a depth of approximately 1/3 the diameter (.160") *(4,0 mm)* and select the appropriate speed and feed percentage (%). *(Figures 15 and 16)*

c) Proceed with ramping cut until the depth of cut is approximately .080" *(2,0 mm)*. This is the 45° mark on a .500" *(12,7 mm)* diameter round insert. During the cut, the feed and speed should be incrementally or continuously increased. At the conclusion of the cut, the parameters should be at the appropriate speed and feed percentage (%). *(Figures 15 and 16)*.

d) Cutting distance is measured in minutes and can be programmed for five minutes with a .500" diameter *(12,7 mm)* round insert. *(Figure 14)*

Figure 40 – Optimization of Ramping Technique with 1/2" (12,7 mm) Round Inserts



Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

To achieve the desired effect of a constantly changing depth of cut to eliminate notching, it is not necessary to think of ramping in terms of a straight line. For example, a wavy line achieves the same objective, perhaps more efficiently, by moving the hardened surface back and forth on the cutting edge. On both the first and second cuts, the material is gradually increased and then gradually decreased. *(Figure 41)*

Also illustrated are examples of plunging in and then ramping using a positive round insert or producing a ramp with a lead-angle tool using a straight-edged insert.

Figure 41 – Various Ramping Methods

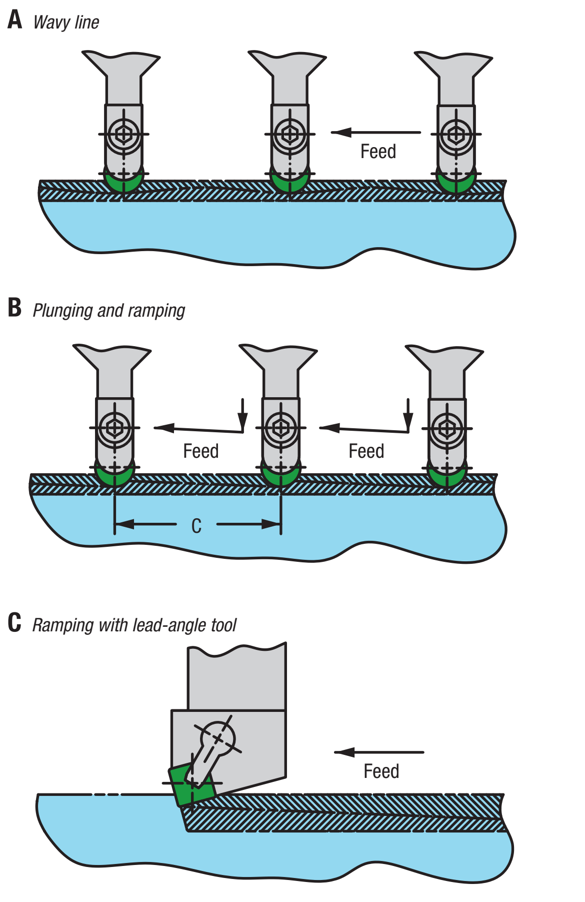
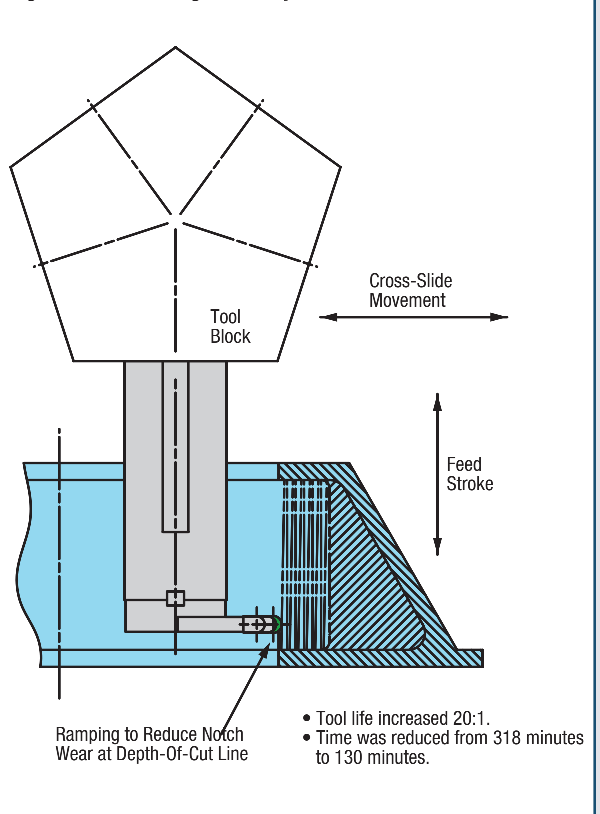


Figure 42 shows an operation boring a cavity on an Inconel 718 part on a Vertical Turret Lathe. As originally programmed, this operation required five tool indexes to bore out all of the material. Five times the tool returned to "home" position and was out of the cut.

By changing to a new program and converting from carbide to Greenleaf WG-300 with "ramping," the entire cavity was machined without a tool change. Productivity increased three (3) times and tool life increased by a factor of 20 to 1. Actual machine time was reduced from 318 minutes to 130 minutes.

Our files contain numerous cases of productivity gains of this magnitude by "ramping".

Figure 42 – Boring a Cavity with Ceramics



Turning to a Shoulder

When rough or finish turning into a shoulder with high velocity techniques, it is most important to observe some basic rules.

With negative inserts in particular, it should be remembered that the chips are being pushed forward. As the shoulder is approached, the chips will be trapped, giving rise to an increase in tool pressure. *(Figure 43)* Also tool pressures increase as insert engagement increases near shoulders. This may result in tool breakage. *It is strongly recommended that the feed rate be reduced by about 50% when the tool is within .125" (3 mm) of the shoulder.* Reduction of the feed will tend to straighten out the chip as the chip temperature increases, reducing pressure on the insert cutting edge. ***This applies to any shape of insert.***

When the tool stops at the shoulder and then withdraws, a hard crystallized layer of material is left. This may produce a series of steps when a square insert is used or scallops when using a round insert. *(Figure 44)*

Very poor tool life will be experienced in any subsequent operation to finish machine these stepped or scalloped surfaces. The solution is to program the tool to continue moving up the shoulder face upon completion of each pass. This will remove the scallops or steps while the material is still hot ahead of the cutting edge and leave a more readily machinable surface for finishing.

Figure 43 – Chip Being Trapped Against Shoulder (increased engagement increases tool pressure)

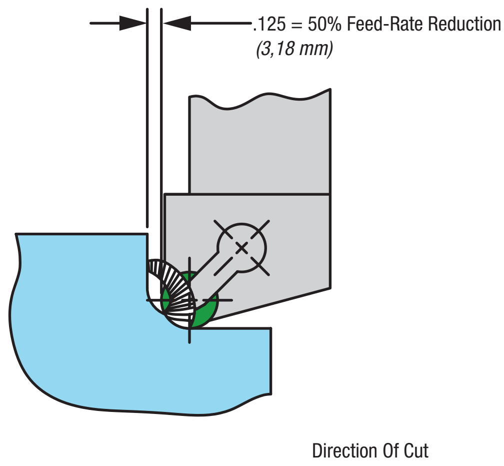


Figure 44 – Avoid Leaving Scallops at Shoulder

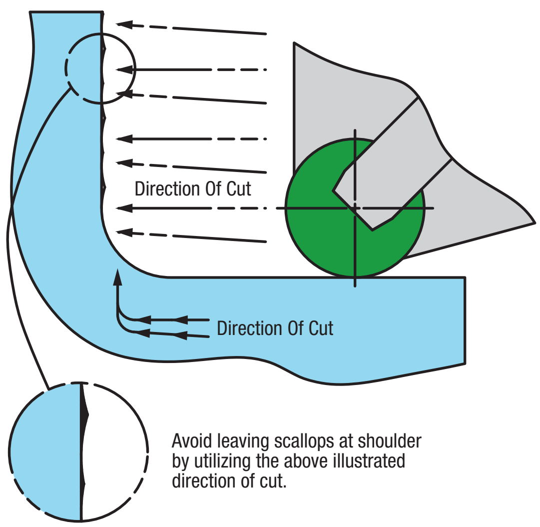
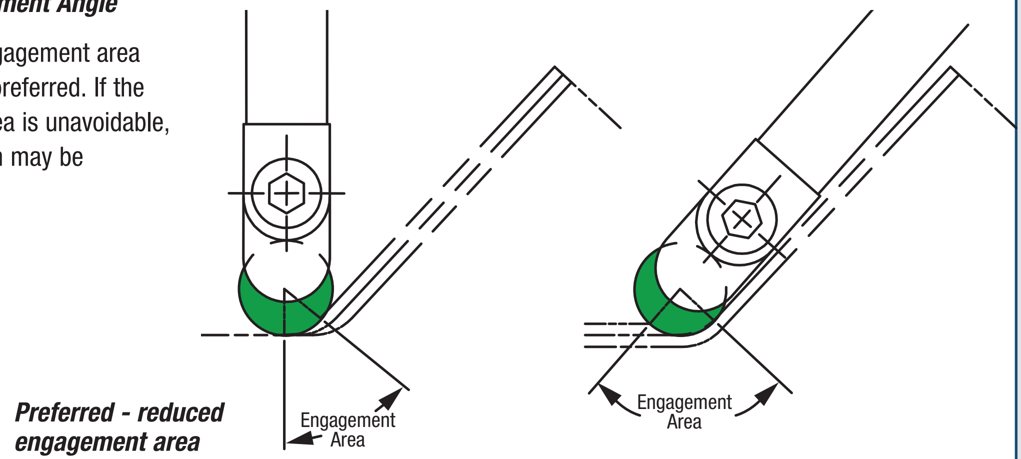


Figure 45 – Tool Engagement Angle

Maintaining a reduced engagement area as shown in *Figure 45* is preferred. If the increased engagement area is unavoidable, then a 50% feed reduction may be necessary.



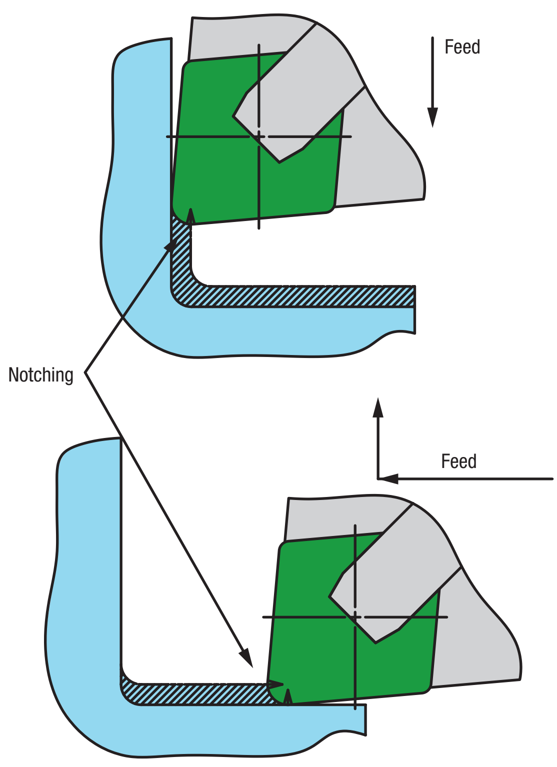
Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

Double Notching – Not Recommended for Notch-Sensitive Situations

It is quite common to program a cut in both directions in CNC machining using carbide inserts. This may have been a matter of convenience to avoid a tool change. It should be noted however, that this is a very undesirable method in notch-sensitive situations such as in high-velocity machining with ceramics.

Here, the tool was fed first from the top and then along the bottom and a blend was made in the radius area. The problem is obvious. Notching has occurred on both sides of the insert. During the second cut, material jams into the first notch causing chipping. Stress may generate a failure crack from notch to notch and break off the corner. *(Figure 46)*

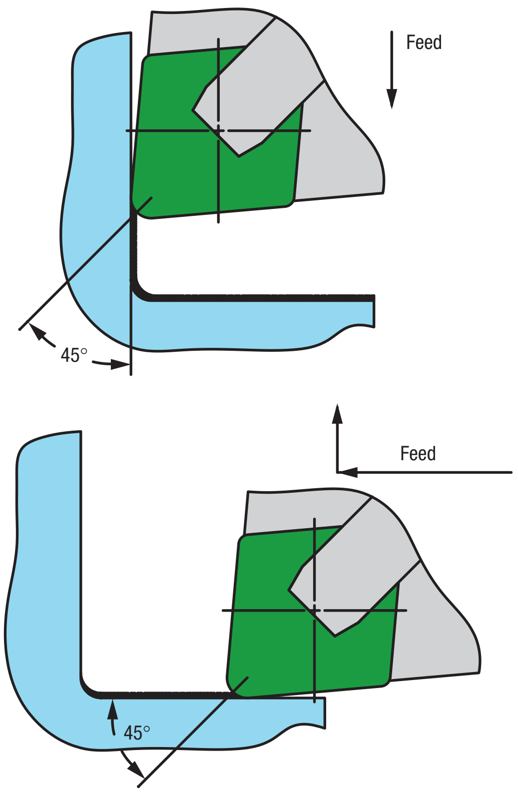
Figure 46 – Carbide Method Beware



Rethink the process

The correct procedure is to take more material off during the previous roughing application, then remove the amount of stock suitable for the nose radius of the insert *(Figure 20)* by staying below the 45° mark of the corner radius. This will minimize notching and allow a cut from both directions. *(Figure 47)*

Figure 47 – Ceramic Method



Finishing a Fillet

Sound design criteria, especially on highly stressed parts such as jet engine components, calls for fillets or specific radii at the junction of most angles.

Problems encountered in finish machining of fillets can often be traced to the approach made in the rough-machine operations.

The amount of stock left for finishing and the shape and condition at the surface of this stock are affected greatly by the tool path and insert configuration used in roughing.

It is not uncommon for a programmer to call for a tool having the specific radius of the fillet and do the entire operation with this tool. This radius is usually small, therefore the tool is weak and must typically be indexed or changed to complete the operation.

There are a number of effective methods available to accomplish these corners, all of them superior to the common method of multiple passes with the weak radius tool.

Method 1 *(Figure 48)*

#1 – The material is roughed using a .500" *(12,7 mm)* diameter round insert. This leaves a .250" *(6,35 mm)* corner radius. In addition, stock for finishing has been left on both walls.

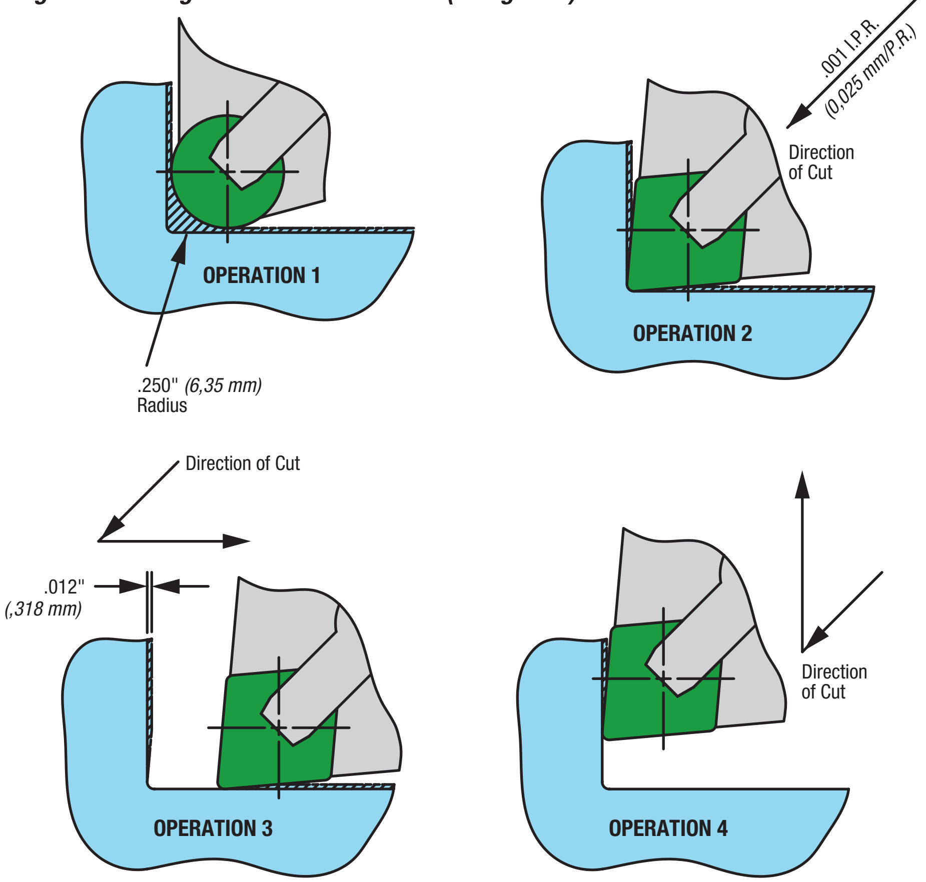
#2 – The finished radius is now generated by plunging the 80° Diamond-shaped insert finishing tool at 45° into the corner. This plunging operation spreads the effect of the work-hardened surface across the nose of the tool without notching it. In addition, the tool is supported by equalized forces on both sides. A clean, accurate radius is also produced.

#3 – The tool is then drawn across one of the faces to produce the finished surface. The long 5° reverse lead angle inherent in the 80° Diamond insert will produce a good finish without damage to the cutting edge.

#4 – The second wall is finished by turning the tool to the corner and feeding out in the other direction, again working on the long lead angle.

Method 1

Figure 48 – Finishing a Fillet Using an 80° Diamond Insert (Plunge Cut)

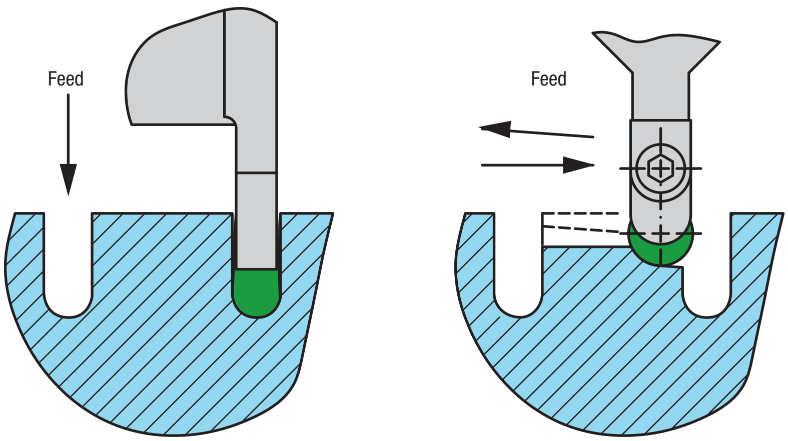


Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

Method 2

Figure 49 – Finishing a Fillet Using a Grooving Tool and a Round Insert

Very small radius fillets on parts are often produced with the fewest problems by using a grooving tool on the first operation. A grooving tool is self-stabilizing and always moving forward into clean material. This results in efficient machining without tool notching and produces an accurate corner radius. The remaining material is then removed by ramping cuts with a round insert.

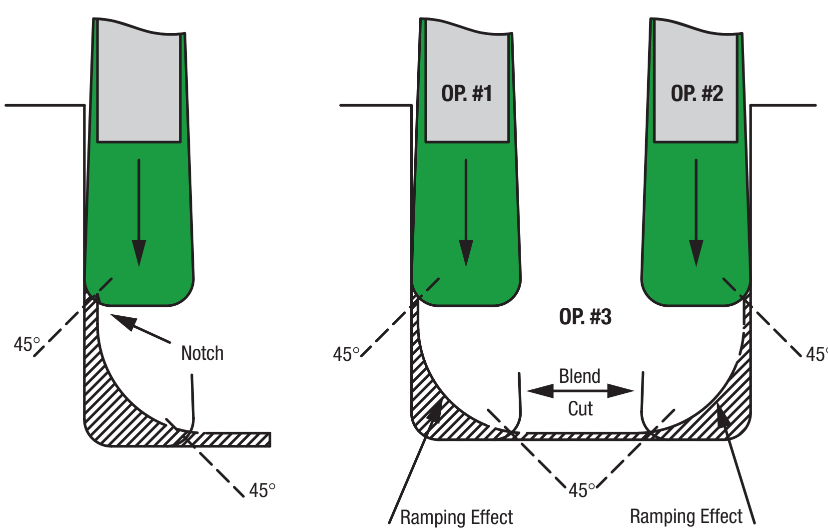


Method 3

Figure 50 – Turning to a Shoulder in Cavities with V-Bottom Grooving Inserts

This example shows the profiling of the groove or cavity using a V-bottom grooving insert. It is important to keep the finish stock very light on the sides so that the cut is below the 45° mark on the insert radius. This will vary with the radius needed. The larger the radius, the greater the stock can be. *(See Figure 20)*

In the corner itself, we use the “ramp” inherent in the radius left by the round insert used for roughing to reduce or eliminate “notching” of the tool. This is a further benefit of roughing with round inserts or profiling the corner in the program.

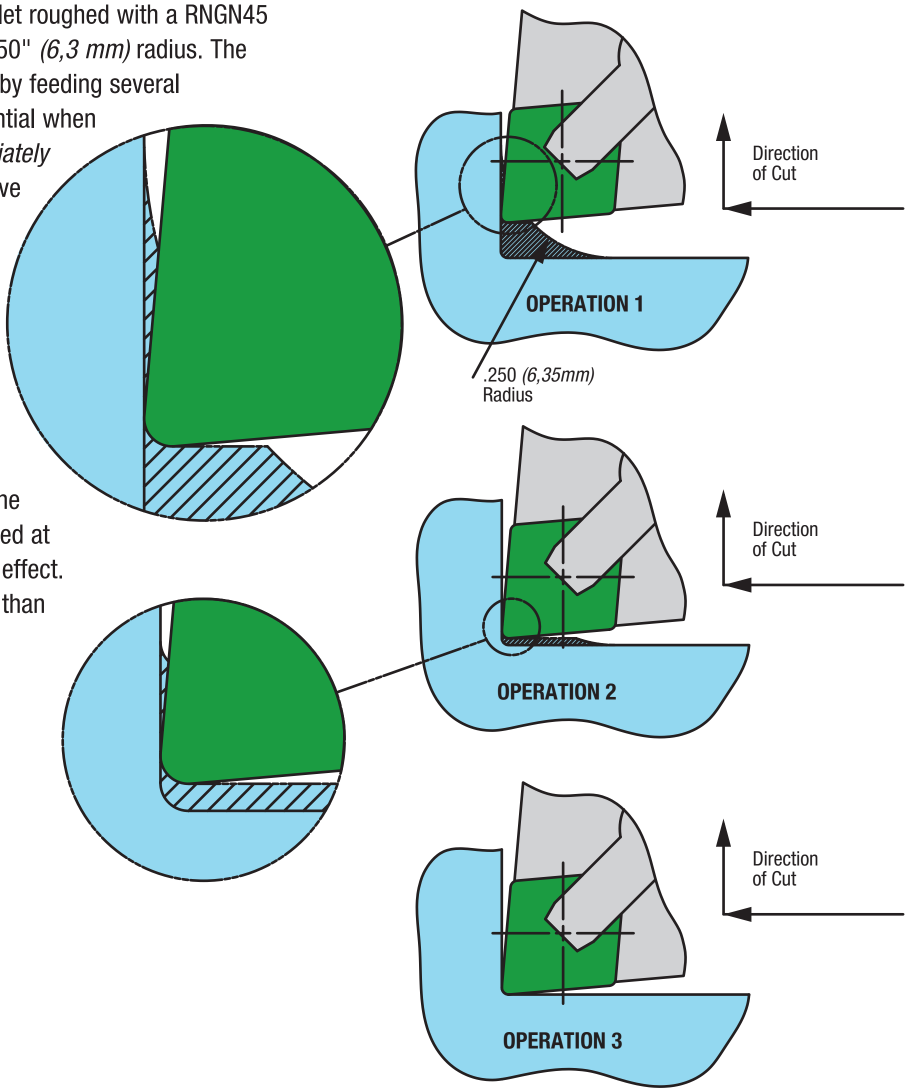


Watch the depth-of-cut line!

Method 4

Figure 51– Ramping Effect on Shoulder Cuts

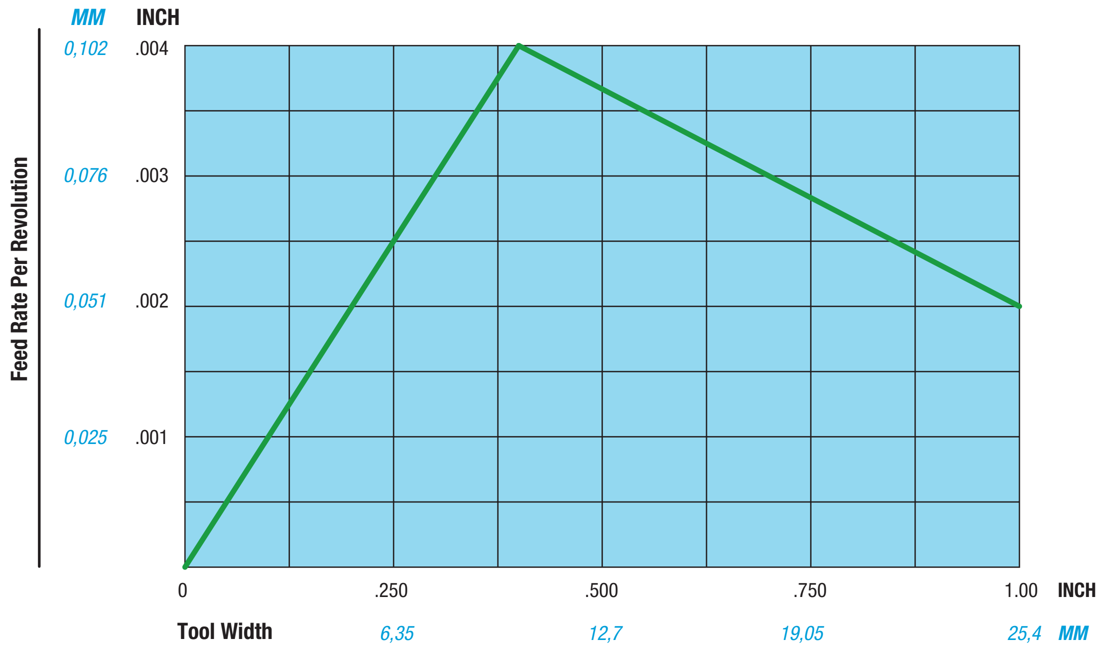
In this method, a CNGN452 (12 07 08) insert is shown in the finish operation on a fillet roughed with a RNGN45 (12 07 00) insert leaving a .250" *(6,3 mm)* radius. The finish operation is performed by feeding several times into the fillet. It is essential when the wall is reached to *immediately* raise the tool vertical to remove the scallop which would otherwise be left on the wall. This material will tend to cool and present a hardened, irregular surface needing a subsequent operation *(Figure 43)*. The finish passes described will tend to notch the tool and should be programmed at various depths to reduce this effect. The final pass should be less than the 45° line of the tool nose radius *(Figure 20)*.



Grooving

Feed and speed for grooving

Figure 52 – Grooving Feeds vs. Tool Width



Select the correct speed from the graph using material hardness as the basis. (*Figure 13*)

A good starting point for feed rates in grooving has been shown to be one percent (1%) of the tool width for widths up to .400" *(10 mm)*. For widths over .400" *(10 mm)* some reduction of this feed will be required. (*Figure 52*)

Case history

Material Rene 95
Part 19" *(483 mm)* diameter
Application .375" *(9,53 mm)* wide O.D. Grove
.500" *(12,7 mm)* deep
Surface speed per minute 400 feet *(122 meters)*
Feed rate per revolution 0.0035" *(0,09 mm)*

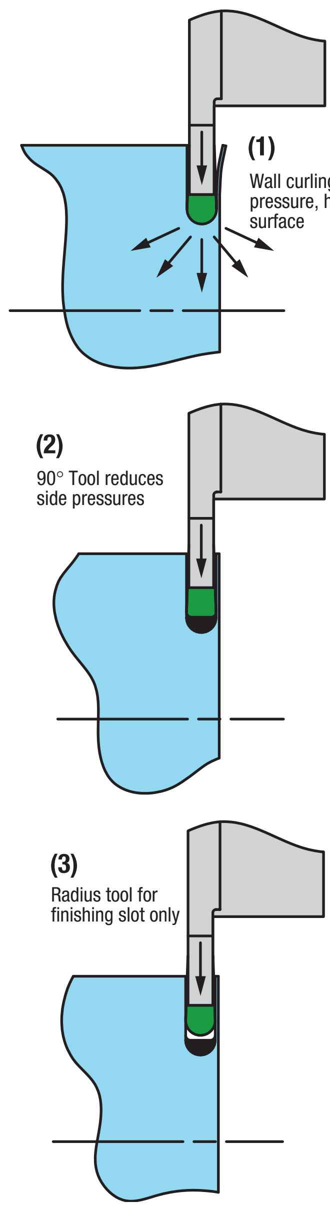
	Carbide	WG-300®
	Cost $	Cost $
Cycle time	35 min.	3.5 min.
Burden rate	@ $60/hr=$35.00	@ $60/hr.=$3.50
Insert cost	$15.00	$50.00
Sub total	$50.00	$53.50
Time saved	0	31.5 min.
	$= 0	@ $60/hr.= $31.50
Total cost	***$50.00***	***$22.00***

Grooving Thin-Wall Sections

A problem may occur when attempting to machine deep grooves, leaving a thin wall standing. If the groove form calls for a radius in the root, then the heat and pressure generated by cutting the entire groove with the radius tool will cause the wall to curve away from the tool. *(Figure 53,1)* This is a combined reaction of the actual pressure and heat of the cut plus the formation of a stressed surface layer.

Good practice dictates roughing the groove with a straight-edged tool ***(2)*** and finishing the radius area only with the radius tool. ***(3)*** *(Figures 53, **2** and **3**)*

Figure 53 – Thin-Wall Grooving



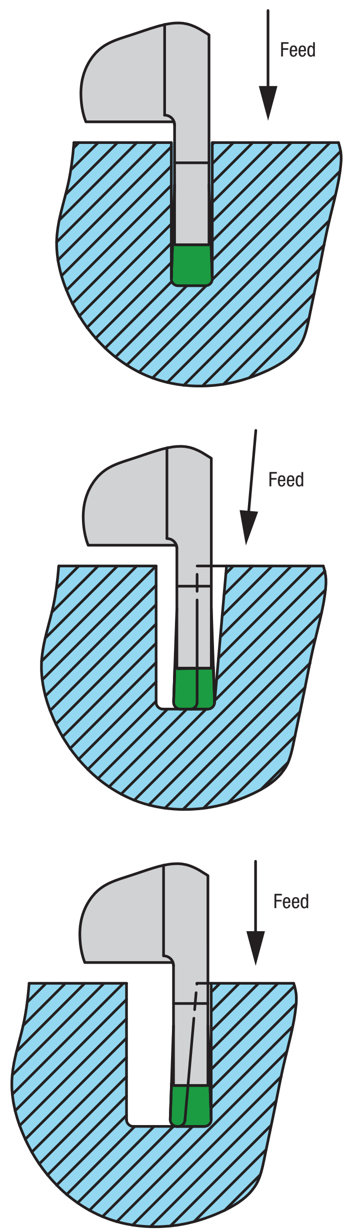
Machining Cavities with Grooving Tools

There are several proven approaches to the machining of cavities using grooving tools. All of the methods shown are satisfactory, however, Methods B and D are the most effective.

Method A *(Figure 54)*

The grooving tool is used to produce a groove in the normal manner by plunging straight into the work. The groove is then widened by using a ramping technique.

Figure 54 – Widening Cavity Techniques



Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

Method B *(Figure 55)*

Two grooves have been produced by plunging straight in. This allows both finished sidewalls and corner radii to be generated. The material between the grooves is removed with a round insert making a straight plunge cut to finish the cavity.

Method C *(Figure 56)*

The cavity is produced by a series of plunge cuts. In this case it is very important to keep the work-hardened surface of the previous cut working on the radius at or beyond the 45° mark to reduce notching. If this is disregarded, rapid notch wear will develop leading to the fracture of the insert corner.

Figure 55 – Additional Cavity Techniques

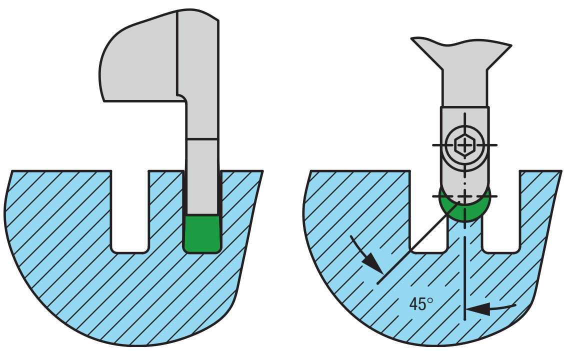
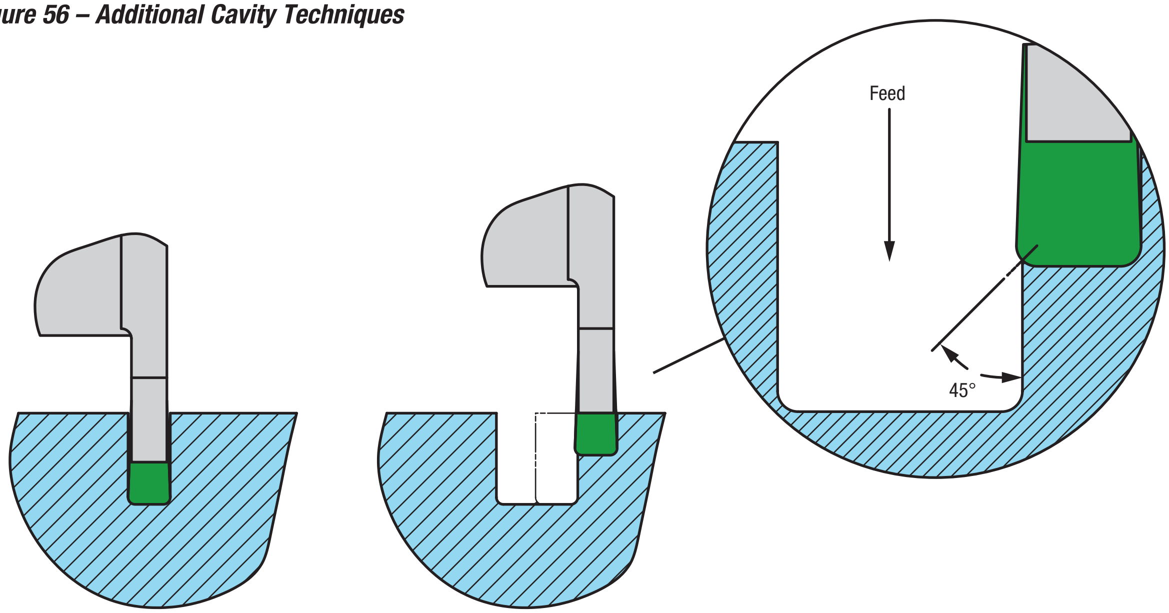


Figure 56 – Additional Cavity Techniques



Method D *(Figure 57)*

The example shown in *Figure 57* is similar to *Figure 55*, except the cavity is wider and the material between the two grooves may be removed by a ramping operation with round inserts. This is an effective method of approaching a wide-cavity application.

Figure 57– Ramping in Cavities

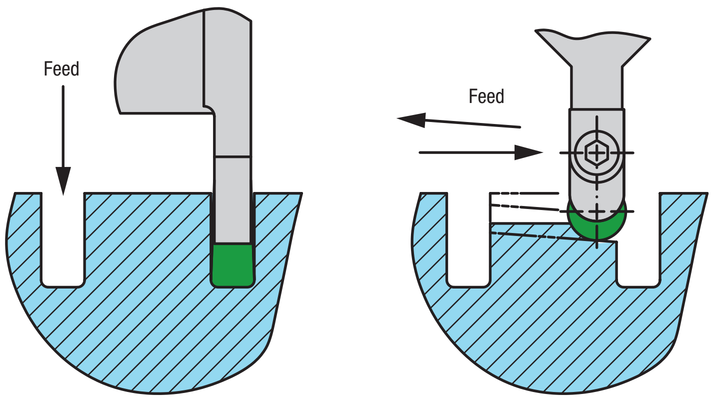
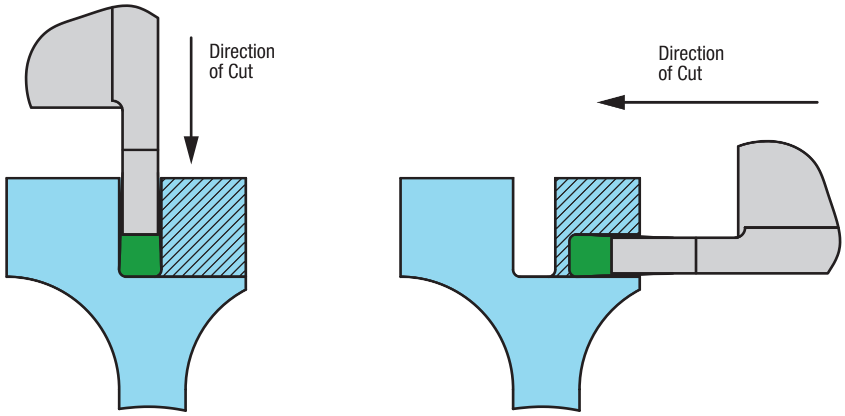


Figure 58 – Producing a Test Sample



Grooving Tools for Shoulder Cuts

It is possible to make shoulder cuts with grooving tools involving the removal of large amounts of material by producing a complete ring.

This technique is being applied in the production of large jet engine discs very effectively but requires special set-up. The method is illustrated in *Figure 58*.

In effect, two 90° opposing grooves are plunged into the part using a V-bottom grooving tool. This generates two clean walls and the required corner radius.

When the second groove breaks into the first one, a complete ring is produced which may be used for some other component. A fixture must be used to hold the ring as it parts from the main forging. It is worth constructing a special clamping fixture for such cases since the method itself is so economical.

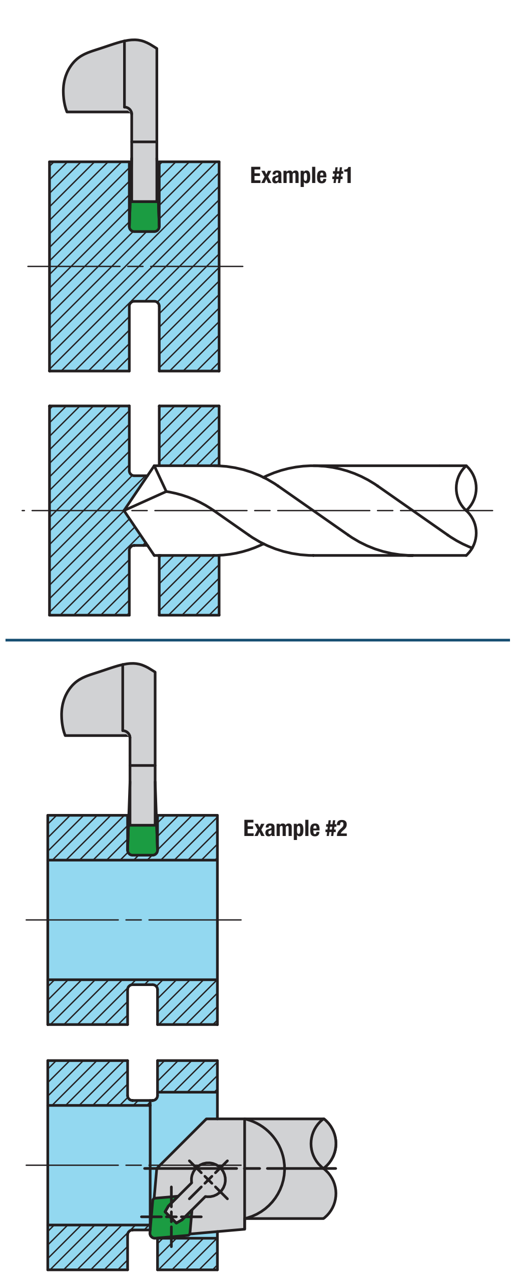
Rethink the process

Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

Cut-Off Operation with Ceramic Inserts

Using a whisker-reinforced ceramic grooving tool and then completing the cut-off with a drill or boring tool in a secondary operation is illustrated in *Figure 59*. This will eliminate tool breakage which would occur if attempting to totally cut off with a ceramic tool. This technique works best with smaller components where the cut-off piece can be captured on the drill or boring tool. There are other variations of this method.

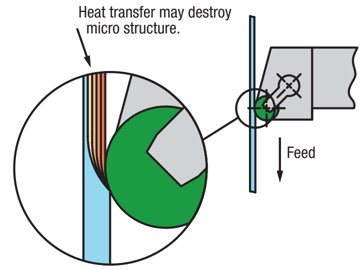
Figure 59 – Ceramic Inserts Used in Cut-Off Operations



Thin-Wall Applications

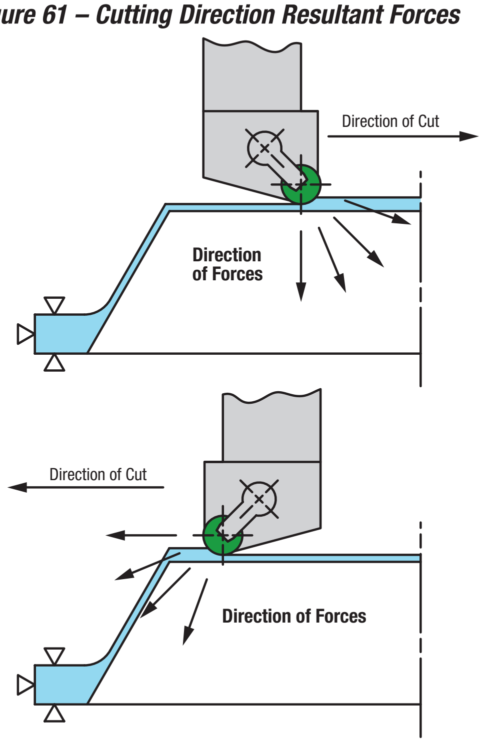
Many turbine components have very thin sections. Distortion of thin-wall parts can become a significant problem. Too much heat in the part due to excessive tool pressure and stress in the material surface due to deformed metal can be the cause of this distortion. In very thin walls, heat may penetrate an entire section causing microstructural damage in the material *(Figure 60)*. In these cases, the speed reduction necessary to limit heat penetration may dictate the use of carbide insert technology.

Figure 60 – Thin-Wall Heat Penetration



In certain situations, the direction of the cut may be of extreme importance. For example, a severe chatter/deflection problem was eliminated in the illustrated aircraft part by a change to facing from the center out. Facing in this way concentrated the resultant forces into a supported section of the part. *(Figure 61)*

Figure 61 – Cutting Direction Resultant Forces



Greenleaf®
800-458-1850

Thin-Wall Applications cont'd

It is important to note the following rules of good practice for thin-walled parts:

1. Reduce the tool nose radius while maintaining the largest radius for best tool life that does not cause distortion.
2. Reduce the lead angle so that the resultant force is directed into a strong or supported section of the part piece.
3. Reduce depth of cut.
4. Do not cause the tool to dwell excessively.
5. Reduce speed.
6. If necessary, change back to carbide for lower surface speed resulting in less deflection, less surface material distortion and less heat.

Rethink the process

Interrupted Cuts

Whisker-reinforced ceramics are inherently very strong and able to withstand interruptions provided the recommended speeds *(Figure 13)* are increased. Speed is all-important in the successful cutting of parts with interruptions.

Do not give in to the temptation to reduce speed.

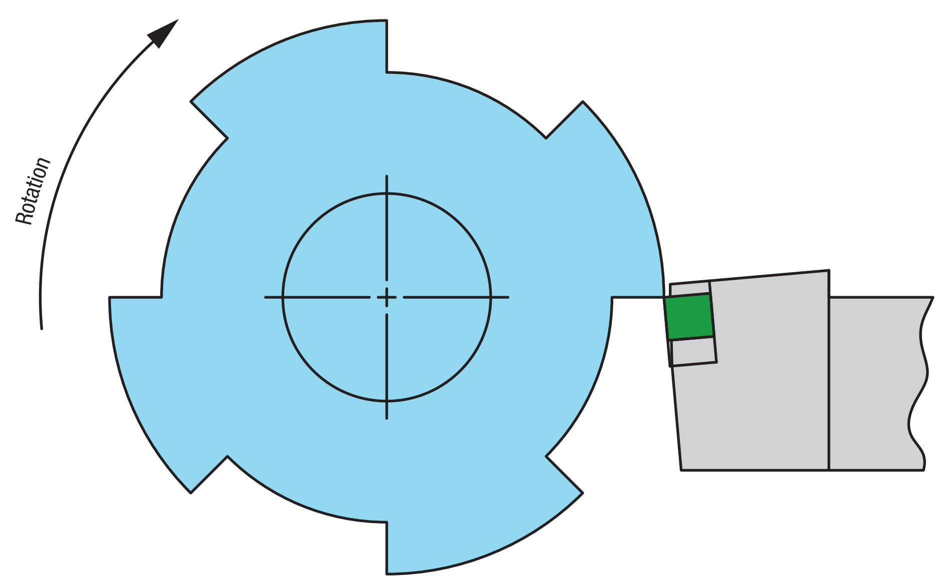
The amount of increase in the recommended speed for severely interrupted cuts can usually be calculated. It is necessary to increase the speed to get back into a temperature zone where the interruptions have lowered by virtue of the intermittent contact between tool and workpiece. First, calculate the circumference of the part and then subtract the sum total of the interruptions. This will give a smaller diameter value. Then increase the RPM so the smaller diameter value returns us to the originally recommended surface speed.

As a simple example *(Figure 62)*, if 50% of the material is taken away by voids or interruptions at the surface, 50% of the surface remains in contact with the tool compared to an uninterrupted part. In this case, double the surface speed to compensate.

A simple estimate will often suffice. Look at the part. Estimate the percentage of surface missing due to interruptions and then increase the speed by at least that amount.

Rethink the process

Figure 62 – Interrupted Cuts



Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

Edge Preparations for Interrupted Cuts

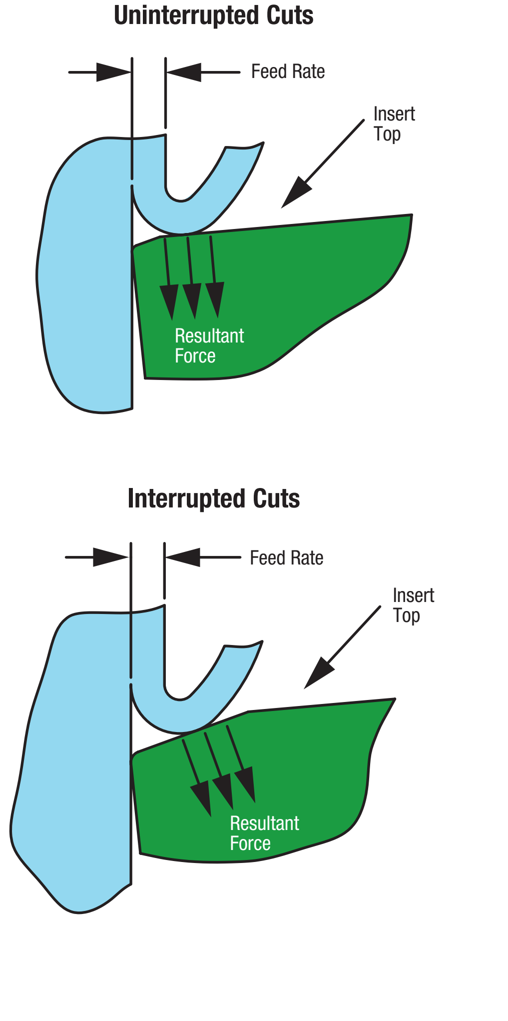
It is advantageous in interrupted cutting to ensure that the feed rate is less than the width of the negative edge preparation on the insert. This assures that the insert cutting edge is in compression at all times and not in shear, as would be the case if the feed exceeds the width of the land. For this reason the T2A or T7A edge preparation should be used.

Feed rate must be reduced on severe interruptions to get more heat into a thinner chip. This will reduce the cutting pressures. If these rules are followed, few problems will be encountered on interrupted cuts. *(Figure 63)*

For interrupted cuts, the rules are:

1. Select a larger edge preparation
2. Reduce feed rate
3. Increase recommended speed

Figure 63 – Edge Preparations



Surface Hardening

Incorrect tooling practices, worn tools, tools with too much hone, etc., can cause excessive surface hardening effects during the machining of nickel-based alloys, particularly in finishing.

It has been shown that cutting with the higher speeds and feeds will decrease (not eliminate) the work-hardening effect and will be an eventual factor in tool life due to notching of the tool at the depth-of-cut line.

If a tool is allowed to dwell without feed, the workpiece will be burnished or glazed and thereby work-hardened. Sharp tools are needed for light operations to avoid burnishing.

Greenleaf whisker-reinforced ceramics have the advantage of being available without hones to accomplish finishing cuts and has the edge strength to make this possible.

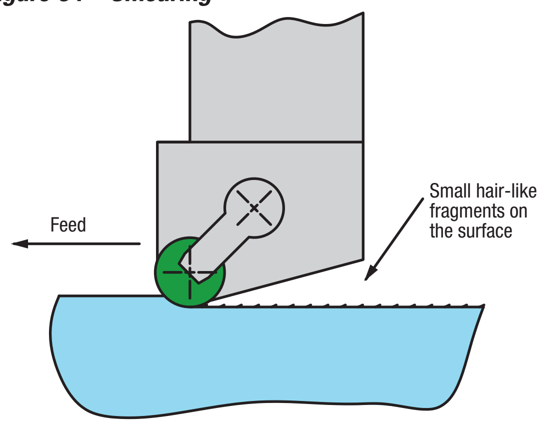
Smearing

Smearing can often be identified as small hair-like particles embedded into the finished surface. *(Figure 64)* This is caused by the nickel, being very gummy in nature, which is built up on the flank of the tool and then swept past a worn, chipped, or honed area of the insert under great pressure and is embedded or pressure-welded in small fragments into the finished surface.

Greenleaf advanced whisker-reinforced ceramics are strong enough that inserts are recommended and produced as standard without a hone. A clean, sharp edge is then presented to the part piece, reducing stress and eliminating the tendency to smear the material in finishing cuts.

Smearing will occur even with whisker-reinforced ceramics if the tool is allowed to wear excessively before indexing or if it chips or flakes due to side pressures caused by flank wear.

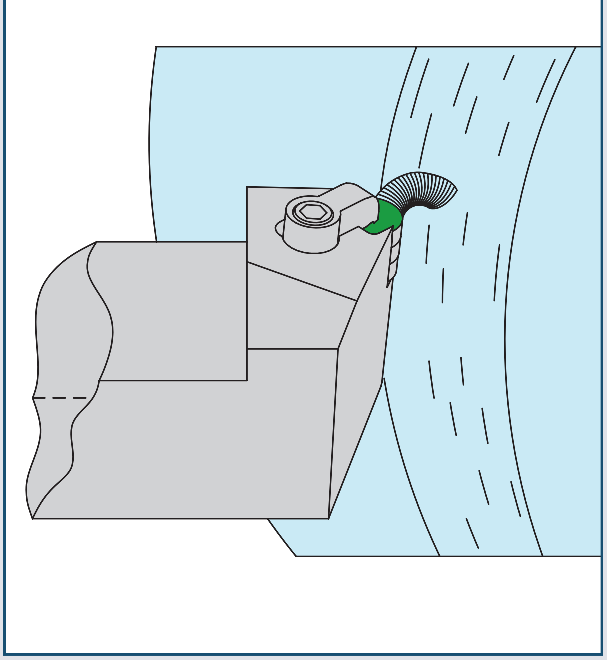
Figure 64 – Smearing



Impingement

Conditions can arise where the chip is curled onto the finish surface immediately behind the tool. Under these circumstances, fragments of the hot, plasticized chip may adhere to the finished surface. *(Figure 65)* Every effort must be made to avoid this condition when cutting at ceramic speeds. Usually a change in tool geometry lead angle, tool radius, depth of cut, feed rate or some combination of these will redirect the chip away from the finished surface.

Figure 65 – Impingement



Boring Holes

With a quill feed machine, boring into a hole increases the spindle extension and as the tool becomes dull, the cutting forces increase *(Figure 66)*. The cutting conditions will deteriorate as the quill becomes progressively less rigid. The results are bores that are tapered and not concentric.

Additionally, chips may accumulate in the bottom of the bore and will eventually be re-cut, further worsening conditions.

It is often advantageous to back bore a hole *(Figure 67)*. This improves the spindle's stability as the insert wears while giving better size, finish and roundness to the bore with less chance of insert breakage and no chance of chip clogging.

Rethink the process

Figure 66 – Spindle Overhang Increasing

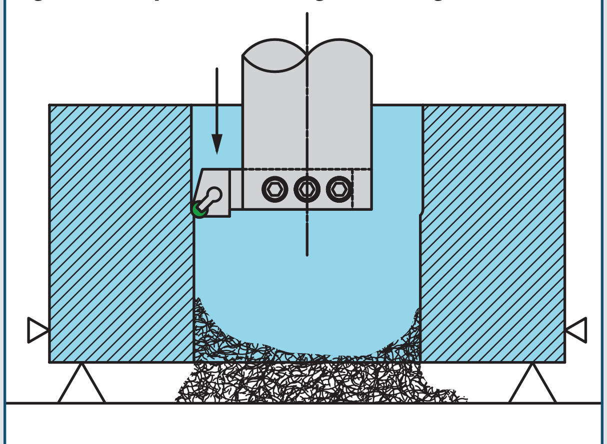
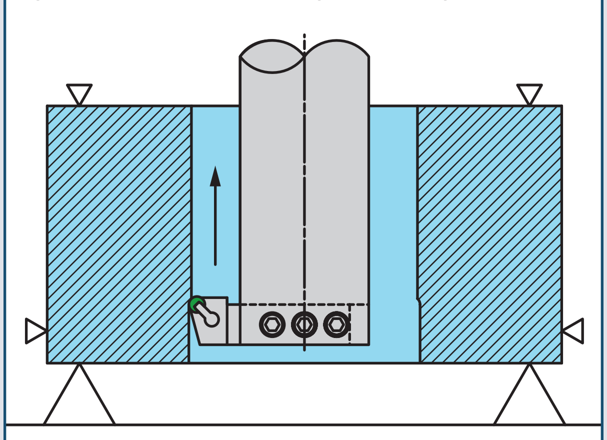


Figure 67 – Spindle Overhang Decreasing



Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

Wear Mechanisms

The normal wear patterns on Greenleaf advanced whisker-reinforced ceramic inserts are unlike the familiar wear patterns on carbide tools. Any attempts to analyze wear or failure mechanisms based upon that prior knowledge will result in ineffective utilization of the WG Ceramics material.

Flaking off of small pieces around the top periphery of the insert is the result of pressure caused by the development of flank wear. In **roughing operations** where surface finish is not of primary concern, this type of tool wear *(Figure 68)* is not usually detrimental to tool performance. In fact, as the tool flakes, a new sharp edge is produced and the tool may go on cutting for long periods in this flaked condition with satisfactory results. In finishing cuts, the flaking will be detrimental to the finish and may also lead to smearing.

At the moment of flaking, sparking can be seen mainly in an upward direction from the insert top surface. The high-temperature material now flowing over a rough top surface of the insert creates sparks. This is not a matter of concern for tool failure. We recommend that the feed rate be turned back 50% to finish the cutting operation.

Before the next operation, a tool inspection should determine whether or not the edge needs to be indexed. It is very important to use the insert to the maximum flaked condition in roughing before indexing or discarding the inserts. Do not make hasty judgment of the tool's ability to continue based upon this flaked appearance. Greenleaf whisker-reinforced ceramics are a completely different material. Continue to use the insert until some experience has been gained on where the limits actually lie in your operation.

***Caution!** When sparks are visibly being carried along the cutting surface, then the insert cutting edge is chipped or broken severely enough that it is not able to cut anymore. This may cause catastrophic failure. Quick action to withdraw the cutting tool is recommended.*

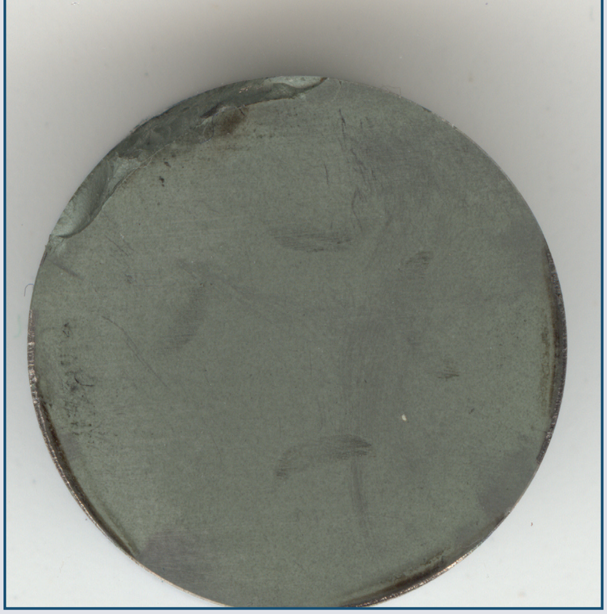
Unlike traditional ceramics, WG Ceramics does not fail by catastrophic breakage except under conditions of severe misuse. The most commonly observed wear/failure modes are chipping of the edge, flank wear, notching and flaking.

Flank wear is a normal progressive wear phenomenon present in all cutting tools. The magnitude of this wear and the speed at which it develops are the values by which tool life should be judged.

In nickel-based alloys, notching will occur at the depth-of-cut line under almost all circumstances. The ideal tool application would be one in which the notch wear was at an acceptable maximum at exactly the same time as flank wear had developed to an acceptable maximum. However, one wear phenomenon usually develops ahead of the other.

Notch wear should not extend past 1/3 of the thickness of the insert. Rapid notch wear or chipping of the insert is often the result of insufficient heat in the shear zone ahead of the tool. Increasing the speed or decreasing the feed or a combination of both can remedy this.

Figure 68



Indexing of Inserts

If the following indexing practices are observed in the application of Greenleaf whisker-reinforced ceramics, the result may be two to three times more part production per insert than when following indexing practices for traditional ceramic or carbide tools. Tooling costs may be cut in half…or better.

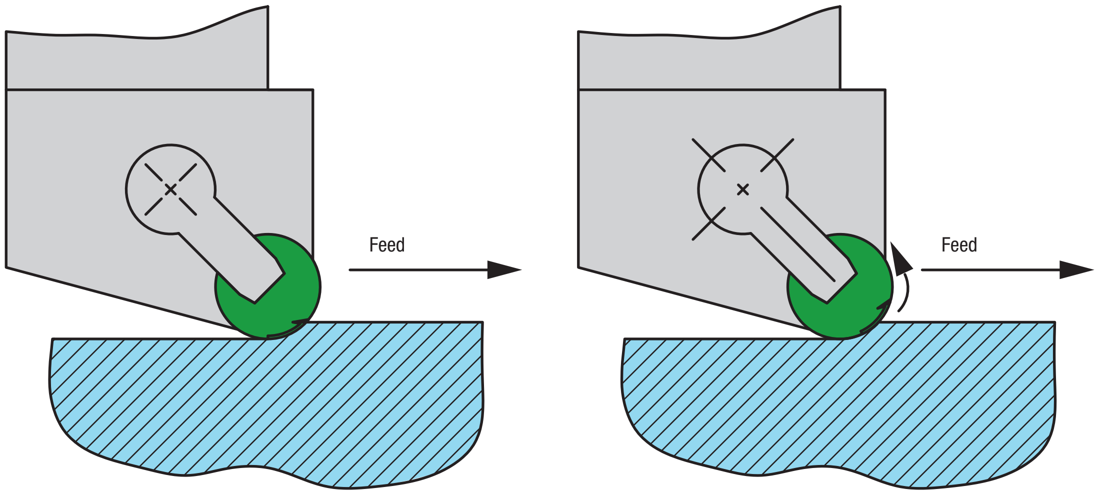
For Optimum Tool Life:

Method 1 *(Figure 69)*
When notching has reached the maximum depth of 1/3 the thickness of the insert, but the flank wear land is not at the maximum; index the inserts as illustrated so that the next notch is developed on an area where the wear land is now present. This is done by turning the insert away from the finished surface so that the notch is clear of the hardened surface layer, but the wear land is still inside the next cutting zone.

Method 2 *(Figure 70)*
When both notching and flank wear land have developed at an equal pace and both are at a determined maximum, index the insert's notch towards the finished surface so that the notch is just clear of the finished surface and adjacent to the start of the toolholder pocket.

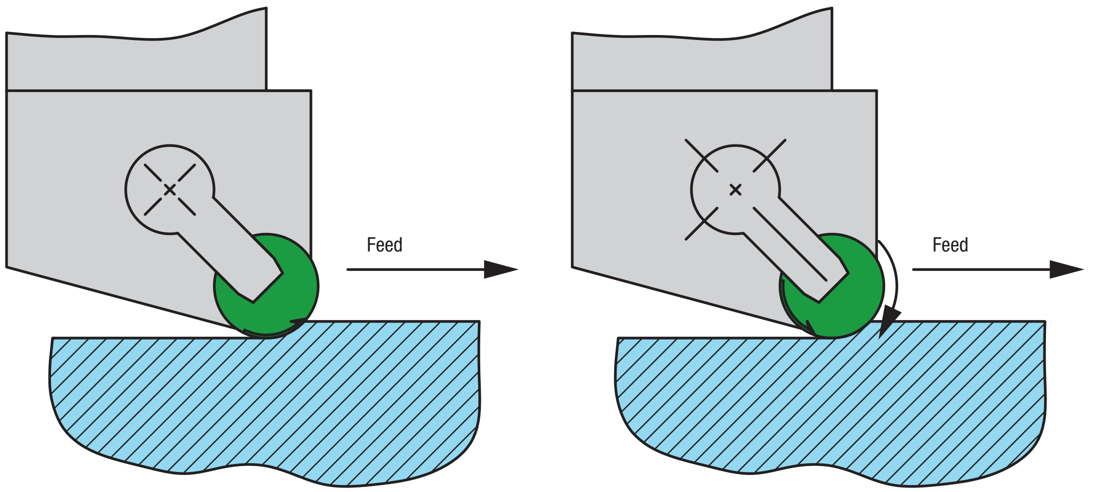
Method 1

Figure 69 – Indexing of Round Inserts (Due to Notching)



Method 2

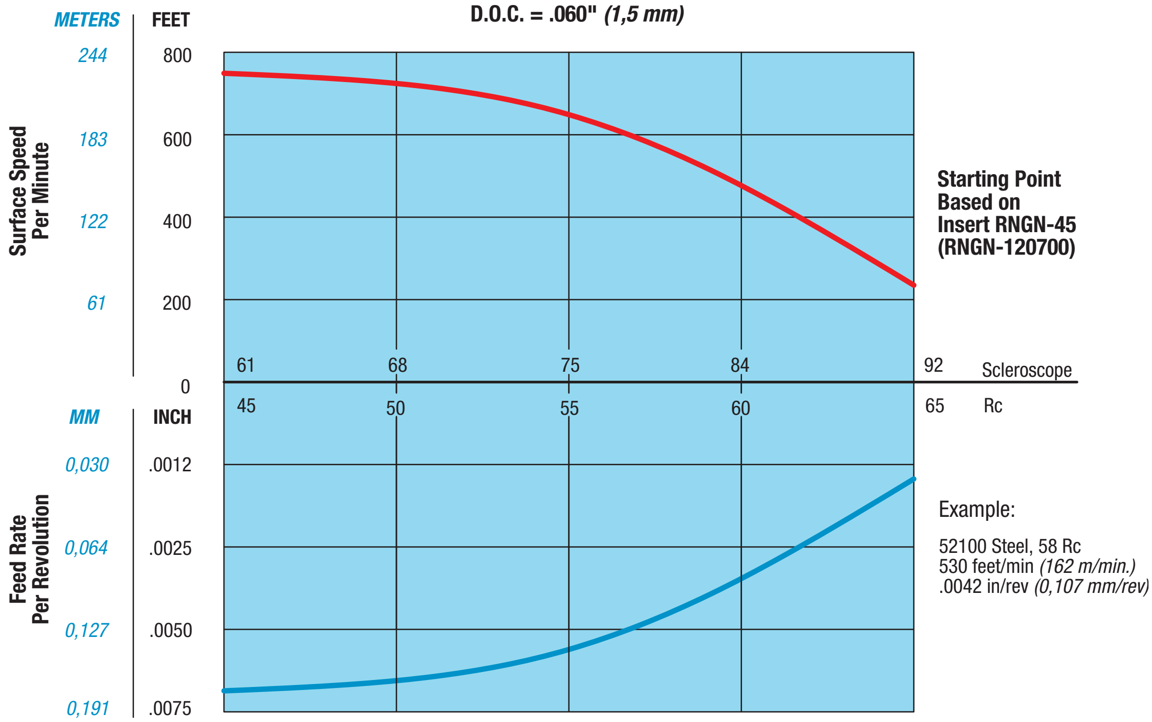
Figure 70 – Indexing of Round Inserts (Due to Notching and Wear)



Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

Turning Hard Materials 45-65 Rc

Figure 71 – Machining Recommendations for Hardened Materials



Greenleaf advanced whisker-reinforced ceramics are successfully used for the turning of hard materials other than nickel-based alloys in the range of 45-65 Rc. The outstanding hardness, combined with the high strength imparted by the reinforcing silicon carbide whiskers, makes possible the machining of many materials previously workable only by grinding. Some areas where the greatest savings have been shown are in the heat-treated alloy steels, die steel, weld overlays, hard facings and hard irons.

As in nickel-based alloys, speeds can be increased up to 8x those used for uncoated tungsten carbide tools and 4x those of coated carbide tools.

The above graph *(Figure 71)* gives starting points for speeds and feeds based upon material hardness. In hard turning the use of a light hone on the insert such as edge preparation T1A may help reduce chipping. ***Coolant should not be used.*** In this application we also recommend the use of an "ANSI" toolholder system which inherently has a five-degree double-negative rake.

If your job is in the 45 to 65 Rc range, chances are that Greenleaf whisker-reinforced ceramics can increase productivity and cut machining costs substantially.

Milling of Nickel-Based Alloys

Milling can be compared to interrupted machining in turning. Since each insert is in and out of the cut during each revolution, the desirable temperature ahead of the tool is not easily achieved and calls for increased surface speed, reduced feed per tooth or a combination of both. It can be surprising how much extra speed is needed in some operations to get the heat back compared to machining the same material continuously as in turning. The increase can be many times the turning speed.

If a cutter designed for carbide is employed, new problems can arise. Often carbide insert milling cutter designs do not incorporate safety features to prevent components from dislodging at high speeds.

The use of coolants is not recommended.

With milling, unlike turning, the chip can be generated from thin to thick as in conventional or "up" milling or thick to thin as in "climb" or "down" milling. It is highly recommended to use the climb milling technique to avoid high heat in a thin section of the chip which encourages chip welding and re-cutting of the chip, which in turn reduces tool life.

To summarize, when milling with Greenleaf whisker-reinforced ceramics:

1. Increase the speed from the turning recommendations in *Figure 13* according to the width of cut.
2. Reduce the feed rate recommended for turning in *Figure 13* by about 50%. **Remember, this is feed per tooth, not per revolution of the milling cutter.**
3. Be sure to use a Greenleaf high-velocity milling cutter or a cutter designed specifically for use with ceramics at high surface speeds.

Recommended Speed Increase for Milling with Various Declining Widths of Cut

In a milling operation the width of cut has a direct bearing upon the temperature generated ahead of the inserts. As the width is decreased, so is the temperature since each insert now passes through air for a longer time than it actually cuts material.

Figure 72 shows the percentage of increase to the speeds given in the graph *(Figure 13)* for various declining widths of cut. The widths are also expressed as percentages of the cutter diameter so the chart can be applied to all cutter sizes.

At the very best, a milling insert can only be cutting 50% of each revolution if the path of cut is equal to the cutter diameter. For this reason, it will always be necessary to increase speed and reduce feed compared to the turning recommendations in *Figure 13* to achieve the same temperature range.

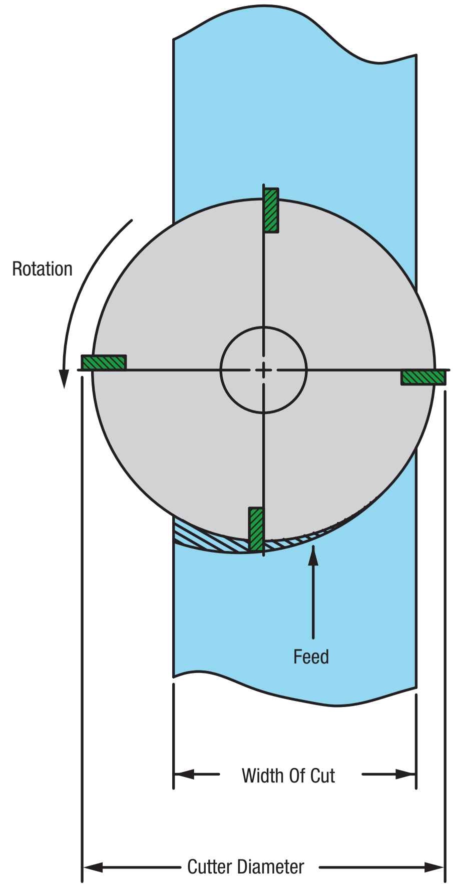
Example of a Ceramic Milling Application

The following data indicates outstanding success with a ceramic milling application:

Material.......................... *Waspaloy*
Condition........................ *Forging*
Hardness........................ *41 Rc*
Operation .... *Rough and Finish Milling*
Cutter Diameter............... *3" (76 mm) WSRN-60003*
Number of Inserts *4*
Depth of cut (rough)........ *0.050" (1,27 mm)*
Depth of cut (finish)......... *0.025" (0,64 mm)*
Insert .... *RNGN 45 T2 (120700)*
Grade............................. *WG-300®*
Speed .... *3144 SFM (958 m/min)*
Feed............................... *64 ipm (1,6 m/min)*
Feed per tooth................. *0.004" (0,1 mm)*

This application resulted in an 80-to-1 reduction in the cutting time cycle over carbide.

Figure 72 – Recommended Speed Increase for Milling with Various Declining Widths of Cut



Width of Cut in Percentage of Cutter Diameter Engaged in Workpiece	Surface Speed in Percentage of Graph (Figure 13)
100%	125%
90%	150%
80%	220%
70%	280%
60%	340%
50%	400%
40%	460%

Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

Targeted Application Areas for Greenleaf Advanced Whisker-Reinforced Ceramics

The potential for applying whisker-reinforced ceramics extends considerably outside of the aircraft industry and involves categories of materials where little or no work has been done to date.

In order to provide you with a starting point for cost justification calculations, a rating list follows.

This list is extrapolated from carbide performance data published by leading users of the alloys listed. A sampling of the data gives us every reason to believe that it will work very well for WG-300® as a starting point. Here's how it works.

a.) In the Ceramic Productivity Manual, the graph *(Figure 13)* gives speeds and feeds for a given hardness value assuming the use of RNGN 45 (120700) inserts. The value represents 100% in that system.

b.) The following list gives a percentage rating for a number of materials where there is existing data. ***Note:*** *This is for forged (wrought) materials. Only the speed is to be considered at these new machinability values as a starting point with WG-300®. For values below 100%, speed, feed, depth of cut and time in cut must be reduced to the suggested rating.*

Starting Point from *Figure 13* for Various Materials

Alloy	#AMS	UNS#	% Rating
A-286	5726	S66286	115
A-286	5731	S66286	115
A-286	5732	S66286	130
A-286	5734	S66286	115
Astroloy	5882	N1307	120
Custom 450	5863	S45000	180
Custom 455	5617	545500	140
Greek Ascoloy	5616	S41800	250
Hastelloy B		N10001	
Hastelloy C	5750	N10002	180
Hastelloy D			
Hastelloy G		N06007	
Hastelloy N	5771	N10003	150
Hastelloy S	5711	N06635	180
Hastelloy W	5755	N10004	130
Hastelloy X	5754	N06002	130
Haynes 25	5759	R30605	85
Haynes 188	5772	R30188	85
Haynes 263		N07263	50
IN-100	5397	N13100	60
Incoloy 804		N06804	
Incoloy 825		N08825	
Incoloy 901	5660	N09901	130
Incoloy 901 Mod.	5661	N09901	115
Incoloy 903		N19903	120
Incoloy 925			100
Inconel 600	5665	N06600	140
Inconel 601	5715	N06601	140
Inconel 617	5887	N06617	100
Inconel 625	5666	N06625	115
Inconel 700			
Inconel 702		N07702	
Inconel 706	5702	N09706	115
Inconel 718	5662	N07718	100
Inconel 718	5663	N07718	100
Inconel 718	5664	N07718	140
Inconel 721		N07721	
Inconel 722	5717	N07722	115
Inconel X-750	5667	N07750	115
Inconel 751		N07751	
MP-35-N	5758	R30035	115
Monel 400		N04400	
Monel 401		N04401	
Monel 404		N04404	
Monel 502		N05502	
Monel K500		N05500	
Monel R405		N04405	
Nicocraly			
Nickel 200		N02200	
Nickel 201	5553	N02201	200
Nickel 205	5555	N02205	220
Nickel 211		N02211	

Alloy	#AMS	UNS#	% Rating
Nickel 220		N02220	
Nimonic 75		N06075	
Nimonic 80		N07080	
Nimonic 90		N07090	
Nimonic 95			
Nimonic C-263	5886	N07263	30
Nitralloy 125			
Nitralloy 135			
Nitralloy 135 Mod.			120
Nitralloy 225			
Nitralloy 230			
Nitralloy EZ			
Nitralloy N			
Permanickel 300		N03300	
Rene 41	5712	N07041	80
Rene 41	5713	N07041	80
Rene 63			
Rene 77			
Rene 88			80
Rene 95			60
Stainless Steel 15-5 PH	5659	S15500	115
Stainless Steel 17-4 PH	5622	S17400	115
Stainless Steel 17-4 PH	5643	S17400	115
Stainless Steel 410	5618		85
Stainless Steel 430	5627	S43000	400
Tool Steel D2		T30402	125
Tool Steel D3		T30403	

Alloy	#AMS	UNS#	% Rating
Tool Steel D4		T30404	
Tool Steel D5		T30405	
Tool Steel D6			
Tool Steel D7		T30407	
Tool Steel H-10		T20810	
Tool Steel H-11		T20811	
Tool Steel H-12		T20812	
Tool Steel H-13		T20813	125
Tool Steel H-14		T20814	
Tool Steel H-19		T20819	
Tool Steel H-21		T20821	
Tool Steel H-23		T20823	
Tool Steel H-24		T20824	
Tool Steel H-25		T20825	
Tool Steel H-26		T20826	
Tool Steel H-42		T20842	
Udimet 500	5751	N07500	100
Udimet 500	5384	N07500	85
Udimet 630			
Udimet 700			
Udimet 710			
Waspaloy	5704	N07001	115
Waspaloy	5706	N07001	100
Waspaloy	5707	N07001	100
Waspaloy	5708	N07001	100
Waspaloy	5709	N07001	100

*AMS # = Aerospace Material Specification Number

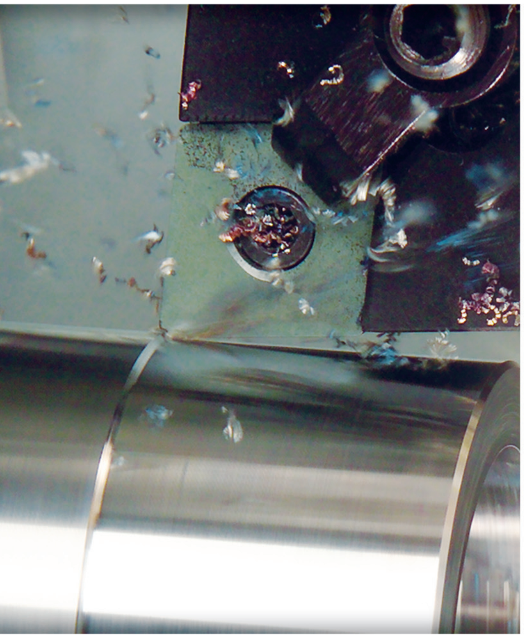
Greenleaf Sales
Phone: 814-763-2915 • 800-458-1850 • Fax: 814-763-4423
sales@greenleafcorporation.com • www.greenleafcorporation.com

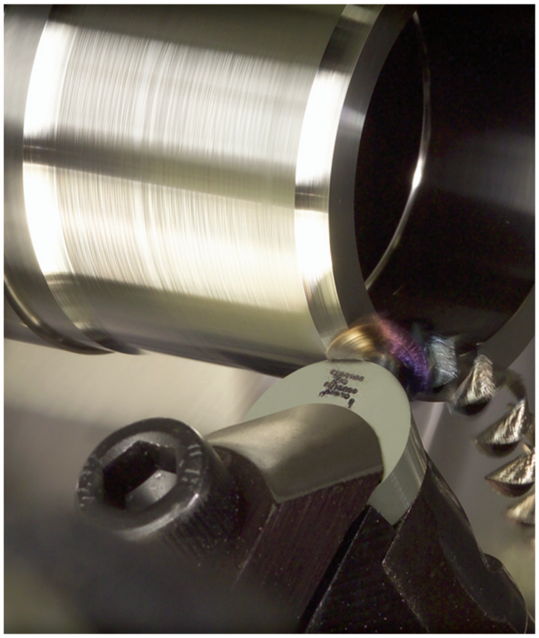
NOTES:

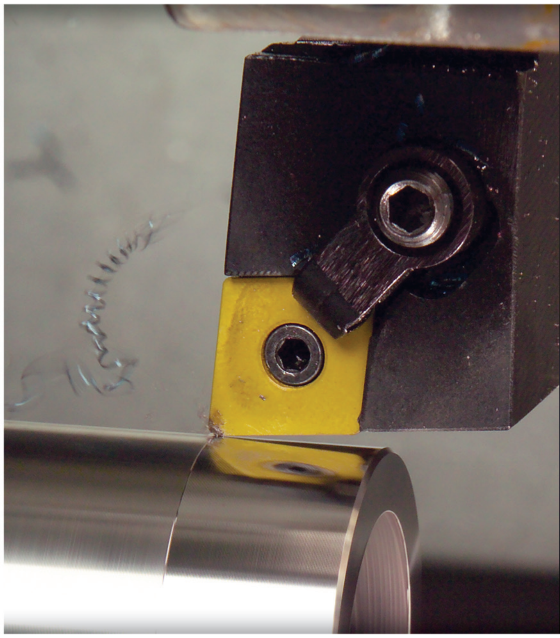
ADD TO CART

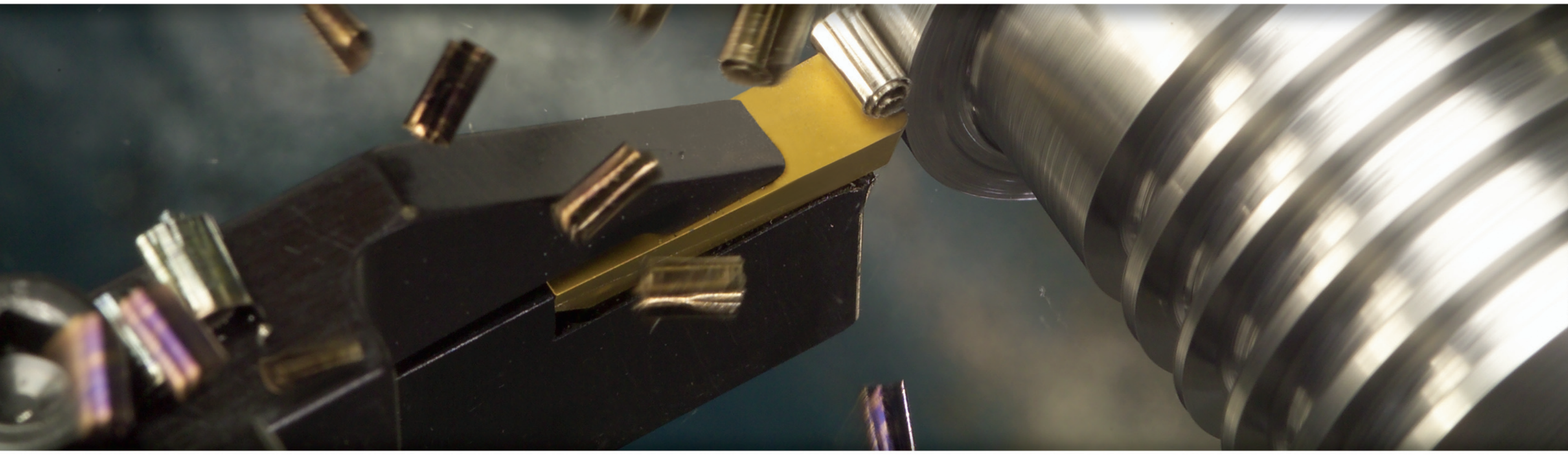
www.greenleafglobalsupport.com

Greenleaf®









MADE IN THE USA

Greenleaf Corporation is ISO 9001 Certified.

GREENLEAF CORPORATION
18695 Greenleaf Drive
PO Box 1040
Saegertown, PA 16433 USA
US Phone: +1-814-763-2915
sales@greenleafcorporation.com

GREENLEAF EUROPE BV
De Steeg 2
6333 AP Schimmert
The Netherlands
EU Phone: +31-45-404-1774
eurooffice@greenleafcorporation.com

GREENLEAF (HUNAN) HIGH-TECH MATERIALS CO., LTD.
No. 3402-A Xiangshang Century Xincheng
No. 17 Kaiyuan Road
Economical and Technical Development Zone
Changsha, Hunan 410100
China
CN Phone: +86-731-84658507
info@greenleafcorporation.com.cn