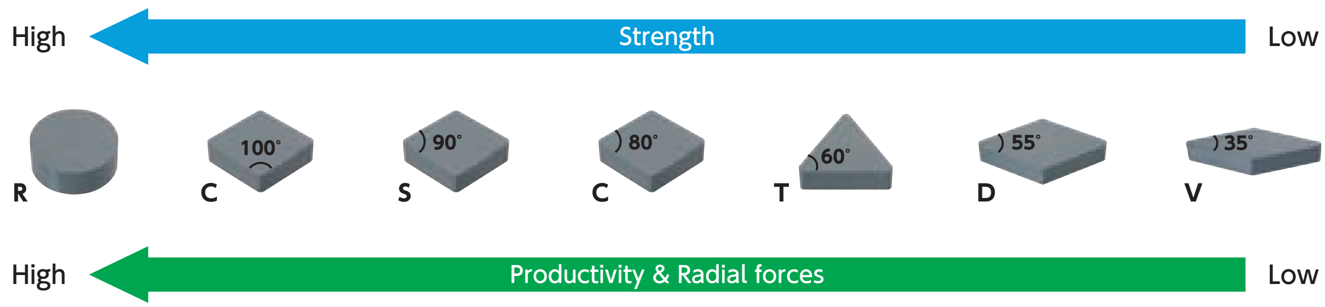
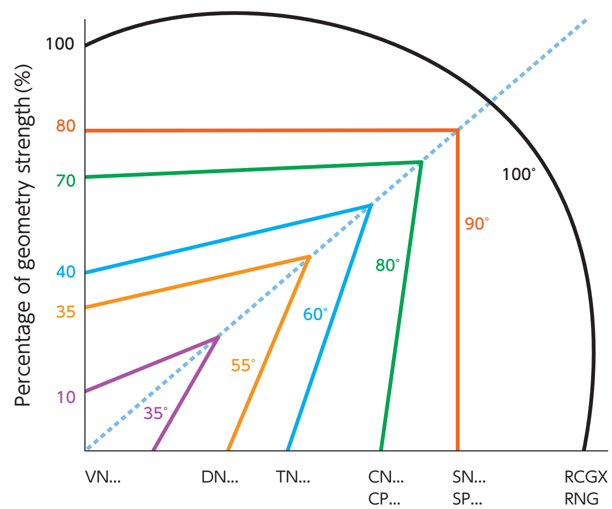
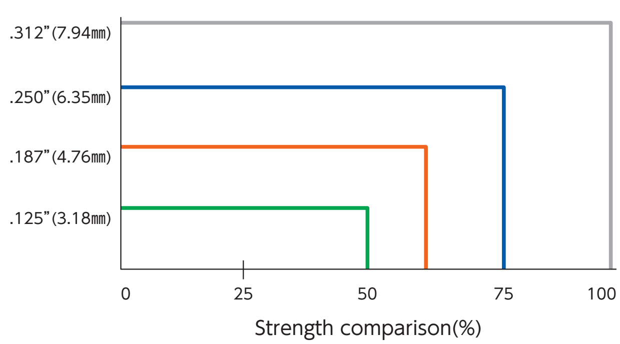
Guidelines for Machining HRSA Materials

Guidelines for Insert Selection

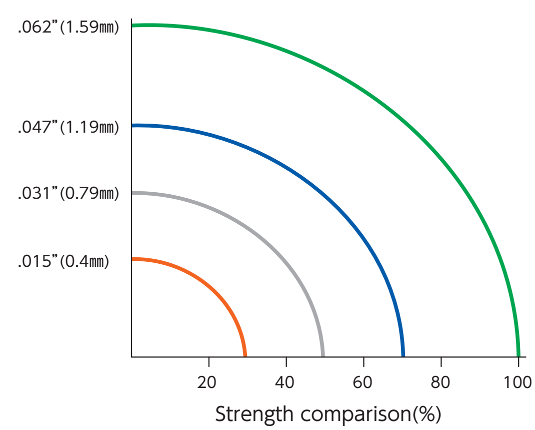




Insert Thickness



Insert Nose Radius Inches



For the best performance always use the strongest possible insert shape to maximize corner strength and productivity. If the operation allows, it is best to use round inserts or square inserts with a large nose radius and a small entering angle.
Use the largest nose radius possible for the operation, so you increase the strength of the insert which will result in better tool life but remember that this will result in increased tool pressure.
Larger insert thickness gives added strength and integrity during machining offering far better impact resistance, heat dispersion, and longer tool life. This results in higher productivity.

Edge Conditions are a Key to Success

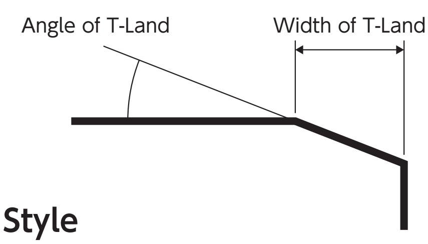
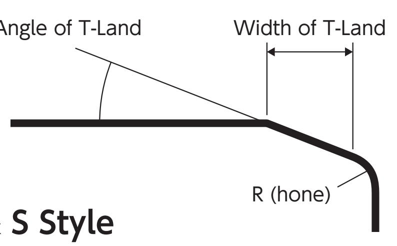
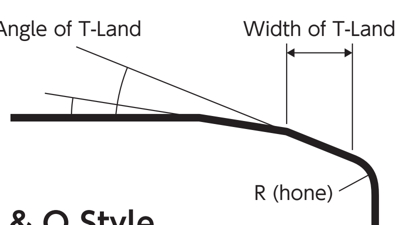
An important factor for achieving success when machining with ceramic inserts is to use the correct edge preparation. Ceramic is a hard material therefore the insert needs some edge work in order to withstand cutting forces and optimize the cutting tool performance. The edge preparation must correspond to the ceramic grade selected, the type of HRSA material being machined and the machining operation being performed. The majority of ceramic applications can be handled with NTK's standard edge preparations.

In unique circumstances that may arise, an edge preparation may need to be specialized to meet the conditions.

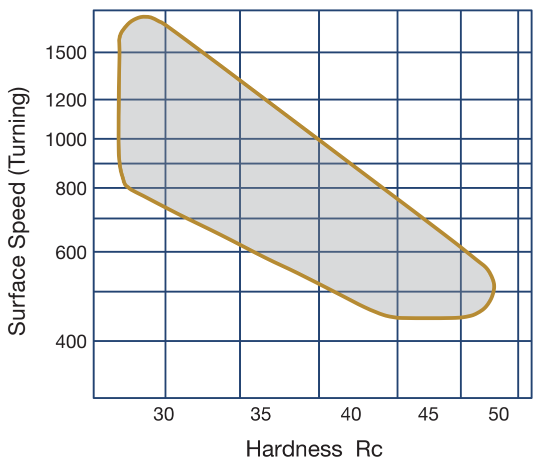
The chart below describes standard edge preps.

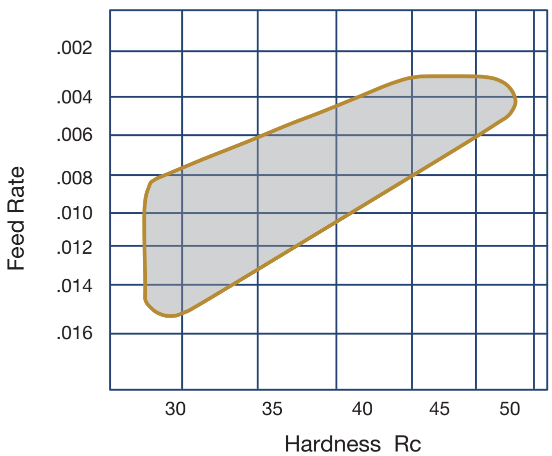
Description of Insert Edge Preparations

Edge Strength Increases

 **FNX Style**	Up sharp edges are not recommended for ceramics.
 **E Style**	Hones help protect the edge of ceramics from chipping or fracturing. Feed rates must be greater than the hone size to prevent a rubbing rather than a cutting action. Excessive honing reduces tool life.
 **T Style**	This geometry is typically the most common ceramic edge preparation. The cutting forces are distributed over a concentrated area of the ceramic edge.
 **Z & S Style**	A hone added to a T-land provides a stronger edge to prevent chipping. Usually this type of geometry works best on interrupted cuts or turning hardened steels.
 **J, P & Q Style**	Double T-lands and hones are generally used in heavy roughing cuts or hardened materials. This edge is extremely shock resistant but also generates large cutting forces.

Know the Workpiece Material to Determine Parameters

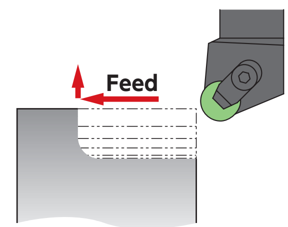




To effectively machine with ceramic inserts it is important to factor in the physical hardness of the material and the surface condition to determine the starting speed and feed. The chart information is based on using an RNG 45 insert with a depth of cut from .125 inches or less. In rough / scale conditions, use the lower side of the speed range for the hardness of the material. If machining clean HRSA material use the higher speed range based on the hardness. When using weaker geometry inserts such as triangles it is important to reduce feeds.

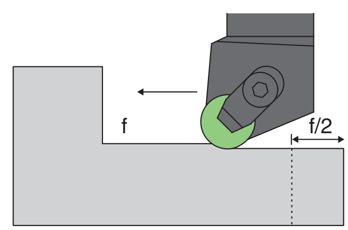
The formation of the chip is a good indicator of the material hardness, the chips will break easily. The cutting temperatures will be higher with hardened materials resulting in more notch wear on the insert edge.

Softer HRSA materials machine similar to stainless steels. Insert grades with greater toughness and reduced hot hardness resistance are ideal in these conditions due to reduced machining temperatures and an increase of the chip breaking against the insert.



Parts that have a forged scale work surface, typically machined in soft state around 26 Rc, require a 25% speed reduction and an increase of the feed until the scale is gone. When cutting cast HRSA material speeds can be increased from those indicated on the graph and it is recommended to reduce feeds to one half of the value indicated on the chart. The maximum depth of cut should be around .060" (for an RNG45) and use flood coolant conditions where applicable. Bar stock is the easiest to machine allowing the use of harder more wear resistant insert grades than when machining forgings.

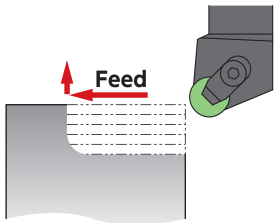
Material	Components	Advantage	Machinability
Forging	Large	High strength	Medium
Casting	Complex shapes	Low strength	Poor
Bar Stock	Less than 7.5" dia.	Availability/strength	good



Guidelines for Machining Heat Resistant Alloy

Rough

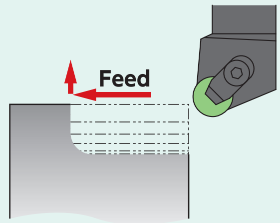
Same Depth of Cut



Note)
Notch wear on the insert cutting edge as shown in is the result of multiple passes being taken at the same depth of cut. This type of wear will minimize tool life. The following programming examples will help to minimize this mode of failure.

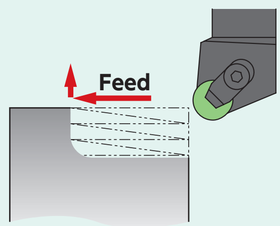
change to

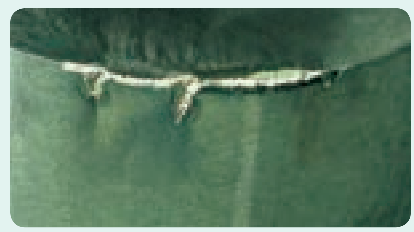
Varying Depth of Cut



Note) Another programming change that may help to reduce notching is by varying the depth of cut. Again, the same principle applies, notching takes place at various points on the cutting edge rather than concentrated at one point.

Ramping





Note) Programming " Ramping " cuts in the same cutting direction is one of the best procedures to use to minimize notching. By varying the DOC, wear is distributed over the entire cutting edge not on one point.

Repeated passes with same depth of cut

This is not a beneficial practice because the insert will develop severe notching at the point of the repetition of DOC. This will result in indexing the insert often. For this reason it is best to vary the depth of cut point by utilizing one of two techniques.

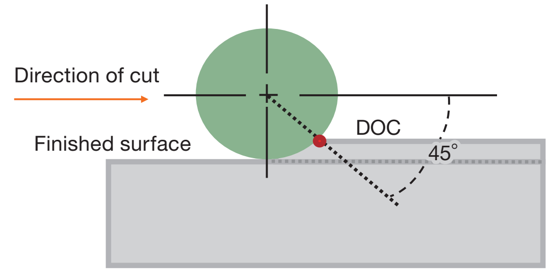
Vary the Depth of cut through multiple passes.

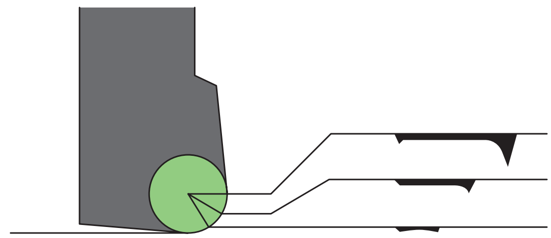
Gradually reduce the depth of cut with every pass. This may increase the operation time slightly but will result in longer tool life for the insert and less indexing of the insert.

Multiple passes using ramping programming

This technique has a proven benefit to roughing operations. Gradually feed out while traversing the part will result in significant reductions in notching. The subsequent pass is programmed at a constant cut since the surface is now ramped.

Depth of Cut and Tool Life

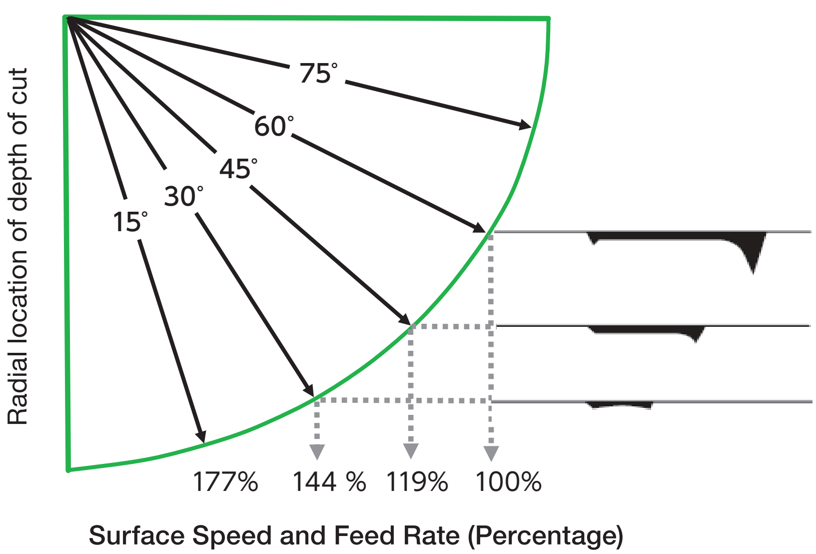




Insert failure due to depth of cut notching is a typical result when machining heat resistant alloys and must be controlled to prevent a catastrophic failure of the cutting edge. The depth of cut is a key consideration during the machining operation to maximize tool life and minimize notch wear on the cutting edge. A decrease in the lead angle results in increased cutting forces on the insert edge. As the DOC exceeds beyond the point on the insert edge where a 45 degree line from the center of the insert intersects the cutting edge the greater the notch wear and the increased risk for failure of the insert edge. There is a direct relationship between the insert radius size and the maximum depth of cut (at around 60 deg. mark) which should be taken. See the chart below for recommendations.

Any increase in DOC requires a reduction of the speed and feed rates. Parameters are based on the ceramic insert's ability to withstand high temperatures and run with a chip thickness that allows the heat to be concentrated in the zone ahead of the insert resulting in low cutting pressure and minimal wear. If the speed is reduced without a corresponding reduction in feed, this effect will be lost and the performance will fall off due to chipping of the insert edge from a cooler chip.

Speed and Feed Rate (%) vs. Depth of Cut on the Radius



Insert Diameter		
.250"	.375"	.50"
Depth of Cut (Inches)		
.063"	.094"	.125"
.038"	.056"	.075"
.018"	.028"	.035"

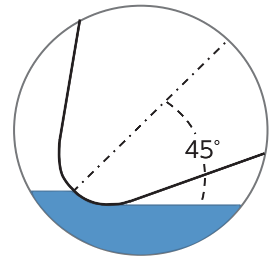
Insert Radius Inches (mm)	Ideal DOC Inches (mm)	Insert Descriptions
.125 (3.18)	.038 (0.93)	RCGX / RPGX23; RPG21..
.187 (4.76)	.056 (1.40)	RCGX / RPGX35; RNG / RPG32
.250 (6.35)	.075 (1.86)	RCGX / RPGX45; RNG43 / 45; RPG 43
.312 (7.94)	.092 (2.33)	RNG55
.375 (9.53)	.110 (2.79)	RNG64 / 65; RPG65
.50 (12.70)	.147 (3.72)	RNG85 / 86

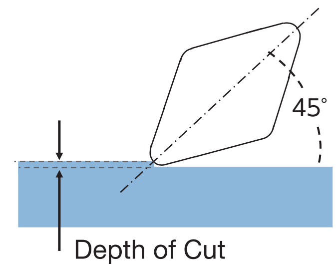
Guidelines for Machining Heat Resistant Alloy

Depth of Cut Recommendation based on Insert Corner Radius

To maximize tool life when using straight-edged inserts (C, D, or S) with corner radii , as opposed to a round insert, the allowable depths of cut are related to the radius and not the insert size. To minimize notching and allow a cut from both directions, the effective machining procedure is to take more material off during the roughing operation, with a round insert. Then the material removal amount for the finishing operation, with a straight edge insert, should be suitable for the nose radius of the insert.

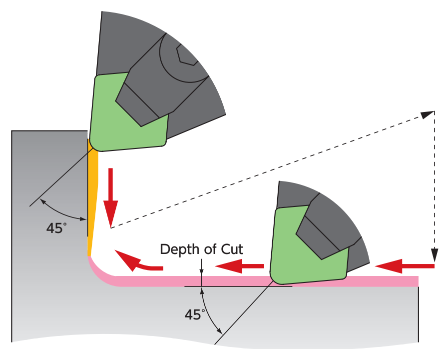
It is important to choose the insert with the appropriate corner radius to complete the finishing operation's depth of cut. If the part has a required radius feature called out, then do not leave more than the amount of material called out for the required insert radius to finish the part and feature. A large corner radius may deflect a part with thin walls because of radial forces generated between the workpiece and insert.



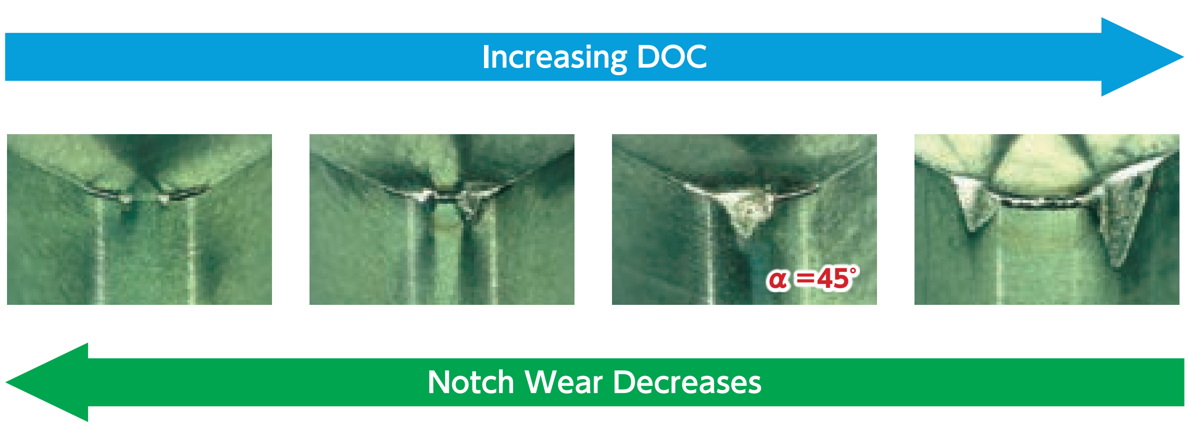


Insert Corner Radius Inch (mm)	Ideal Depth of Cut Inch (mm)	Corner Radius Designation Inch (mm)
.015 (0.38)	.0046 (0.12)	1 (04)
.031 (0.80)	.0092 (0.23)	2 (08)
.048 (1.21)	.0139 (0.35)	3 (12)
.063 (1.59)	.0183 (0.47)	4 (16)
.094 (2.38)	.0275 (0.70)	6 (24)
.125 (3.18)	.0370 (0.93)	8 (32)

Optimum DOC is 5-15% of insert diameter (based on 0 deg. Lead angle)

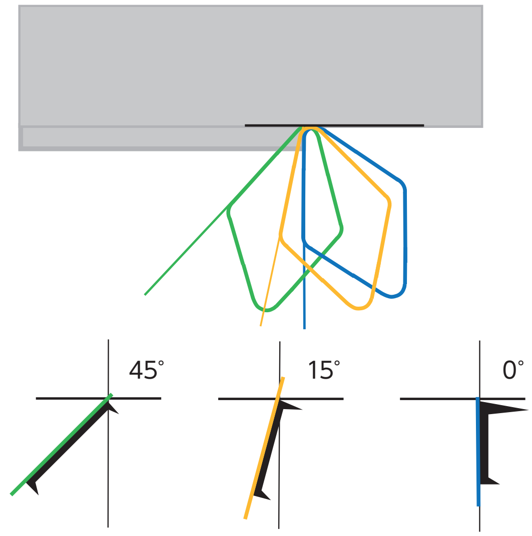


As seen in these photos, by removing the appropriate amount of stock for the nose radius of the insert and staying below the 45° mark of the corner radius notching is minimized allowing a cutting operation to be programmed from both directions on the insert.



Lead Angles

When cutting heat resistant alloys consideration should be given to using the largest lead angle possible. A large lead angle allows the cutting forces to be spread over a larger surface area of the insert. This will also improve tool life and surface finish while reducing notching. As the lead angle increases the chip will flow more easily.



Typical insert wear pattern showing the effect of various lead angle changes and the resulting increase of depth of cut notching

Feed

Surface finish is directly related to the insert nose radius and the feed rate programmed. The larger the radius on the insert the faster the tool can be fed to achieve the appropriate finish. When machining HRSA materials with SiAlON ceramics utilize their superior strength by increasing the feed rate which will minimize wear and cutting time.

Milling Operations on Heat Resistant Alloys (High Nickel)

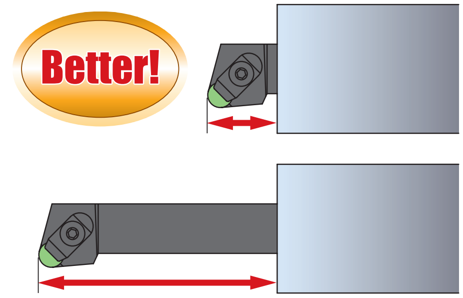
Button inserts in a milling cutter rotate in and out of the cut during a revolution this reaction on the insert edge is comparable to machining an interrupted cut on a turning operation. This rotation in and out of the material also can hinder achieving the desired temperature ahead of the tool. So, an increase in speed, reduced feed/tooth in order or a combination will help generate the heat. It is recommended to use climb milling techniques to avoid elevated temperatures in a thin area of the chip which could create chip welding and re-cutting of the chip which reduces tool life.

Increase speeds from turning recommendations in chart according to width of cut. Reduce the feed rate recommendations for turning in chart by about 50% (This is feed per tooth, not per revolution of the cutter)
The width of the cut has a direct relationship to the temperature generated ahead of the inserts. As the width is decreased the temperature decreases because the insert is out of the cut more than in the cut. The chart below shows the percentage to increase speeds given in the previous chart for various widths of cut. The widths are also expressed as percentages of the cutter diameter (so all cutter sizes apply)
A milling insert can only be cutting 50% of each revolution if the path of cut is equal to the cutter diameter. So, it will always be necessary to increase speed and reduce feed compared to the turning recommendations to achieve the temperatures needed.

Width of cut in % of cutter diameter engaged	Surface speed in % of Graph
100%	125%
90%	150%
80%	220%
70%	280%
60%	340%
50%	400%
40%	460%

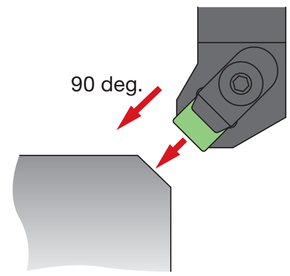
Minimize overhang

Too much overhang causes the holder to deflect resulting in vibration and chatter which is damaging to ceramic inserts and can lead to insert breakage. When working with turret style machines, straight edged inserts should be considered in place of round inserts. The straight edge eliminates radial tool forces and chatter issues.



Pre-chamferring

Pre-chamferring the part reduces the potential for insert chipping or breaking upon the entry or exit point of work material. To effectively complete a pre-chamfer operation it is important to program the feed at a 90 degree angle to the chamfer in order to prevent notching and increase insert tool life.



No dwelling

Inserts wear out when rubbing the part instead of cutting

Coolant

When turning with BIDEMICS, SiAlON and Whisker a flood coolant ondition should be used
In some cases where a high interruption is encountered it may be best to shut off the coolant
No coolant should be used while milling with SX3, SX7 and SX9

Edge preparations

Typical HRSA machining requires the insert cutting edge to be sharp. Using a slight T-land or honed edge is also effective to reduce notching, flaking and built up edge

Guidelines for Machining HRSA Materials

Troubleshooting

Cutting Conditions & Parameters Adjustment

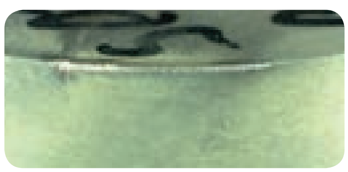
		Cutting speed (SFM)		Feed rate (IPR)		Grade attribute		
		SiAlON	BIDEMICS	SiAlON	BIDEMICS	BIDEMICS	SiAlON	Whisker
	Notching		↗ [a]	↗ [b]		🔴	🔴	
	Flank wear	↘ [c]		↗ [d]		🔴	🔵 SX3 SX7	🔴
	Breakage			↘	↘	🔵	🔴	
Heat		↘	↘	↘	↘	—	—	—
Chatter		↗	↗	↘	↘	—	—	—

🔴 1st Choice 🔵 2nd Choice

Test Results

[a] WA1 : Increase cutting speed

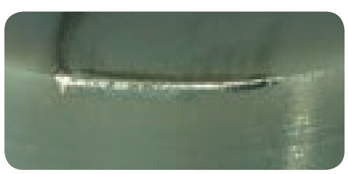
500 SFM



1800 SFM

[b] SX7 • SX3 • SX9 • SX5 : Increase feed rate

.008 IPR



.016 IPR

[c] SX7 • SX3 • SX9 • SX5 : Decrease cutting speed

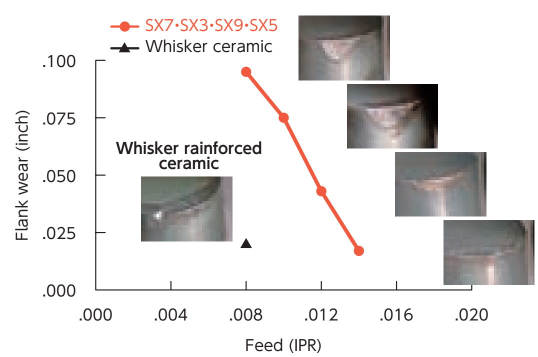
1300 SFM

500 SFM

Note : Speed and feed rates shown are recorded test data and should not be thought of as recommended cutting conditions.

[d] SX7 • SX3 • SX9 • SX5 : Increase feed rate

Feed rate increased decreases wear amount of SiAlON



Cutting condition
Work material : Inco718
Insert shape : RNG45
Cutting Speed : 800 SFM
Depth of Cut : .080"
WET

In some cases, in order to increase the wear resistance of SX7 & SX3 & SX9 & SX5, the feed must be increased. By increasing the feed and utilizing the toughness of SX7 & SX3 & SX9 & SX5, the inserts are off the part sooner causing less wear. Increasing the feed also decreases cycle time and improves productivity and profitability.

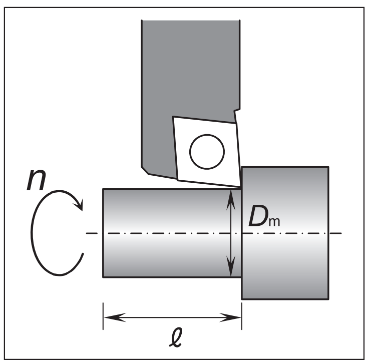
Note : Be careful to reduce the feed rate by 25%, when going into a corner.

Grade Recommendations based on Material's Machinability and Application

Work Material	Rough Turning w/ Scale		Rough no scale & Semifinishing			Grooving		Milling	
	1st	2nd	1st	2nd	High RPM Potential use	1st	2nd	1st	2nd
Hastelloy C	SX5	SX9	SX7	SX3	JX1	SX7	SX3	SX7	WA1
Inconel 625	SX5	SX7	SX7	SX3	JX1	SX7	SX3	SX7	SX9
Inconel 718	SX5	SX9	SX9	SX7	JX1	SX7	SX9	SX9	SX7
Mar M247	SX5	SX3	SX3	SX7	JX1	SX7	WA1	SX7	WA1
Udimet 720	SX5	SX9	SX7	WA1	JX3	SX7	SX3	SX7	WA1
Waspaloy	SX5	SX9	SX7	WA1	JX3	SX5	SX3	SX7	WA1
Rene	SX3	SX7	SX3	SX7	JX1	SX7	SX3	SX3	SX7
Stellite 6	SX5	SX9	SX9	WA1	JX3	SX7	SX3	SX9	SX7
MP35N	SX5	SX3	SX3	WA1	JX3	SX7	WA1	SX7	WA1
Monel	SX3	WA1	SX7	SX3	JX1	SX7	WA1	SX7	SX3
Haynes	SX9	WA1	SX9	SX3	JX1	SX7	WA1	SX7	SX3
Inconel 903	SX3	SX9	SX7	SX3	JX1	SX7	WA1	SX7	SX3
Invar	SX5	SX9	SX9	SX3	JX1	SX7	WA1	SX9	SX3

■ Formula for Turning

● Calculating the cutting speed



Calculating the cutting speed from the rotation speed

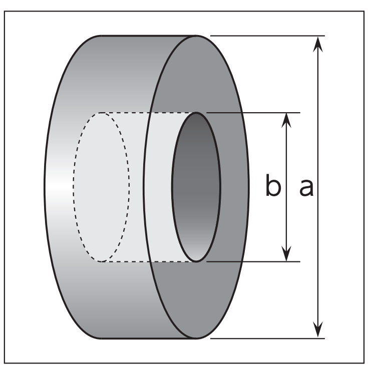
$$v_c \text{ (SFM)} = \frac{\pi \times D_m \times n}{12}$$

Calculating the revolution speed from the cutting speed

$$n \text{ (rpm)} = \frac{12 \times v_c}{\pi \times D_m}$$

v_c : Cutting speed (SFM)
D_m : Machining diameter (inch)
n : Spindle speed (rpm)
π : Pi (3.14)

● Calculating the cutting time



Calculating the cutting time for OD (ID) machining

$$T \text{ (min)} = \frac{\ell}{f \times n}$$

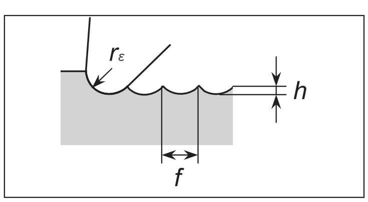
T : Cutting time (min)
ℓ : Cutting length (inch)
f : Feed rate (IPR)
n : Spindle speed (rpm)

Calculating the cutting time for facing

$$T \text{ (min)} = \frac{\pi \times (a^2 - b^2)}{4000 \times v_c \times f}$$

T : Cutting time (min)
v_c : Cutting speed (m/min)
f : Feed amount (mm/rev)
π : Pi (3.14)

● Calculating the theoretical surface roughness



$$h \text{ (μinch)} = \frac{f^2}{8 r_\varepsilon} \times 1000^2$$

h : Theoretical surface roughness (μinch)
f : Feed amount (IPR)
r_ε : Corner radius (inch)

【Guidelines for actually finished surface roughness】
Steel type work: Theoretical surface roughness × 1.5 to 3
Cast iron type work: Theoretical surface roughness × 3 to 5