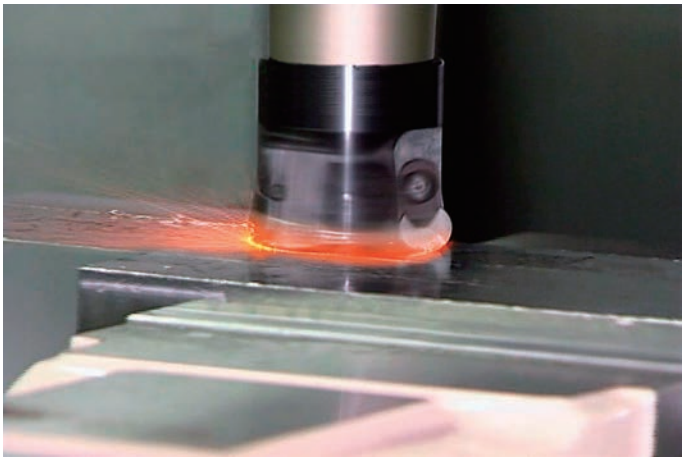


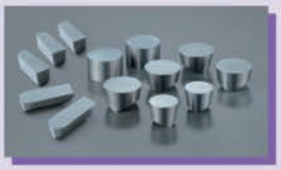
Solution

Heat-resistant alloy machining

BIDEMICS - Game Changer

- 480m/min **Speed** Capability
- Double tool life at whisker's speed range

JX1



■Features
- **Up to 480m/min speed capability**
- Much longer tool life at Whisker ceramics' speed range
- Superior surface finish vs. Whisker ceramics

■Work Materials
- Inco 718 • 718 Plus
- Powdered metal
- Inco 625 • Rene

➡C11

JP2/120

■Features
- 10 to 15x speed capability vs. carbide
- Better wear resistance and notching resistance than CBNs
- Superior surface finish to Carbide or CBN

■Work Materials
- Inco 718 • 718 Plus
- Powdered metal • Inco 625 • Rene

➡C12

SX7

■Features
- Can run at same cutting condition as whisker ceramics
- Best grade for high-speed milling

■Work Materials
- Inco 718 • Inco 625
- Waspaloy • Udimet 720

➡C27

SX3

■Features
- Excellent wear resistance and toughness. Wide range of HRSA machining applications: Roughing with scale - semi finishing turning.
- Able to machine even the newest generation of HRSA work materials (like Rene) as well as most common HRSA materials; such as Inconel 718.

■Work Materials
- Inco 718 • 718 Plus
- Powdered metal • Inco 625
- Rene

➡C25

SiAlON - Workhorse

- Durable for scale to semi-finish machining

Whisker - Versatile Player

● Productivity and reliability

JX3

■Features
- Added toughness in BIDEMICS
- Same speed capability as JX1

■Work Materials
- Inco 718
- 718 Plus
- Powdered metal
- Inco 625
- Rene

➡C11

WA5 / WA1

■Features
- Better flank wear resistance compared to SiAlON ceramics
- Better notching resistance compared to competitor's whisker ceramics

■Work Materials
- Inco 718
- Inco 625

➡C29

SX5

■Features
- Best grade for scale and interruptions
- Best grade for machining high-cobalt alloys

■Work Materials
- Waspaloy
- Udimet 720
- 718 Plus
- Rene 41

※ Production by order.

➡C26

SX9

■Features
- Extreme toughness makes higher feed and heavier DOC machining possible
- Best grade for machining Inco 718 with scale

➡C28

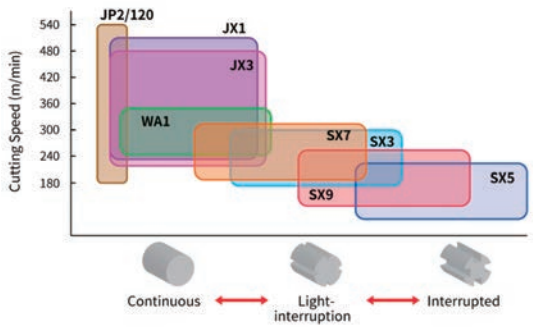
■Work Materials
- Inco 718
- Inco 706
- Inco 713
- Rene

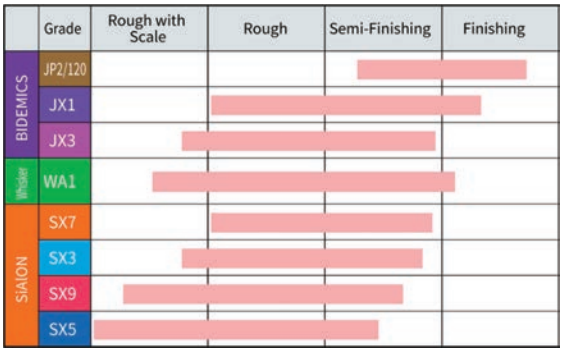
Heat-resistant alloy machining

Insert grades

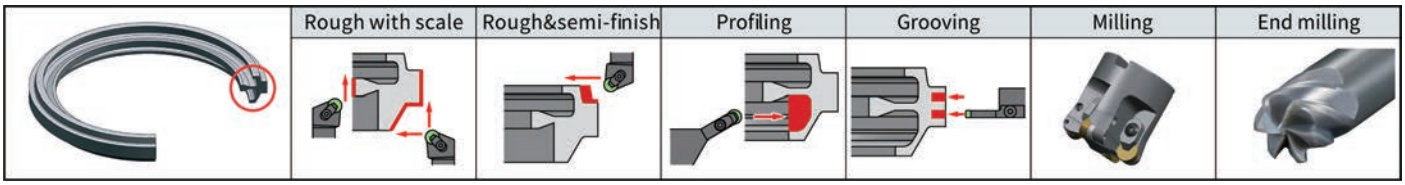
Category	Grade	Attributes	Applications						
			Scale	No scale	Profiling	Finishing	Grooving	Milling	End milling
									
BIDEMICS	JX1	Special grade with higher speed and longer tool life potential		●	●	○	●		
BIDEMICS	JP2/120	Special grade for finish turning				●			
BIDEMICS	JX3	Added toughness in BIDEMICS		●	●	○	●		
Whisker	WA1	General versatile grade for turning		○	○		○		
SiAlON	SX3	Best balance of toughness and hardness	○	●	●		●	●	
SiAlON	SX5	Best grade for Waspaloy with scale	●				○		
SiAlON	SX7	Versatile grade for turning and milling	○	○	○		●	●	
SiAlON	SX9	Best grade for scale of Inco718	●	○	○			○	●

● 1st Choice ○ 2nd Choice





Application



Cutting conditions

Application	Grade	Work material	Cutting speed (m/min) 180 240 300 360 420 480	Feed (mm/rev) 0.1 0.2 0.3 0.4 0.5	Depth of cut (mm) 0.5 1.0 1.5 2.0 2.5	Coolant
Rough with Scale	SX5	Waspaloy	200(180-240)	0.3(0.2-0.35)	2.0(1.0-5.0)	WET
	SX9	Inco718	200(180-240)	0.3(0.2-0.35)	2.0(1.0-5.0)	
	SX3	Overall	240(180-270)	0.2(0.1-0.22)	2.0(1.0-5.0)	
Rough no Scale	JX1 JX3	Overall	210-390(180-480)	0.2(0.13-0.28)	1.7(1.0-2.5)	WET
	SX9 SX3 SX7	Overall	210(180-270)	0.2(0.15-0.3)	2.0(1.0-2.5)	
	WA1	Overall	240(180-300)	0.2(0.12-0.25)	1.7(1.0-2.5)	
Profiling & Semi-Finish	JX1 JX3	Overall	210-450(180-480)	0.2(0.1-0.25)	1.5(1.0-2.0)	WET
	SX3 SX7	Overall	240(180-270)	0.2(0.12-0.25)	1.5(1.0-2.0)	
	WA1	Overall	240(180-330)	0.2(0.1-0.25)	1.5(1.0-2.0)	
Finishing	JP2/120	Overall	210-480(180-510)	0.1(0.05-0.18)	0.25(0.13-0.76)	WET
Grooving	JX1 JX3	Overall	360(180-480)	0.07(0.05-0.1)	When using SX7 / SX3/ SX5, increase feed rates 100% vs. Whisker Ceramics	WET
	SX5	Waspaloy	210(180-240)	0.15(0.07-0.17)		
	SX3 SX7	Overall	230(180-270)	1.1(0.07-0.15)		
	WA1	Overall	240(180-330)	0.07(0.05-0.1)		

Application	Grade	Work material	Cutting speed (m/min) 450 600 750 900 1000 1200	Feed (mm/t) 0.05 0.07 0.1 0.12 0.15	Depth of cut (mm) 0.5 1.0 1.5 2.0 2.5	Coolant
Milling	SX3 SX7	Overall	810(600-1200)	0.1(0.07-0.12)	1.7(1.0-2.5)	DRY
	SX9	Overall	750(450-1000)	0.12(0.1-0.15)	2.0(1.0-2.5)	
End milling	SX9	Overall	600(300-1000)	0.02-0.03		DRY

Heat-resistant alloy machining

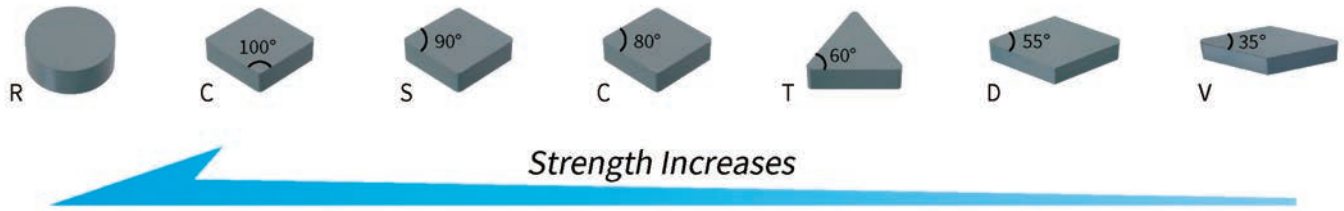
Key Points in Machining

The key to successful heat-resistant alloy machining is the use of "BIDEMICS" and "ceramics".

- BIDEMICS and ceramic materials improve productivity in machining heat-resistant alloys.
- BIDEMICS has excellent VB wear resistance and SiAlON ceramics has excellent wear resistance on the infeed side
- BIDEMICS provides high-speed machining and superior surface finishes not possible with conventional ceramic materials.
- Optimizing cutting conditions and tool grade enables more stable machining.

Selection of insert shape with toughness

Select inserts with higher strength cutting edges if at all possible.

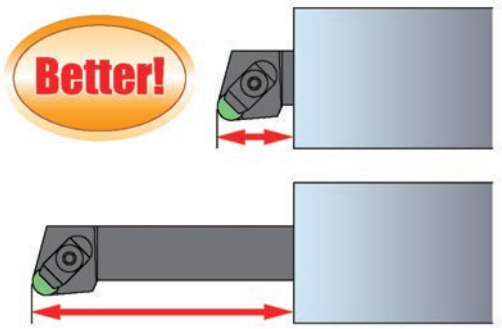


Selection of corner R with superior strength

The larger the insert's corner radius, the stronger the insert's cutting edge and the longer its life. However, please note that the larger the corner radius, the higher the cutting resistance. In general, RNGN1207 inserts are used for rough machining and CNGN1204 inserts are used for finish machining of heat resistant alloys.

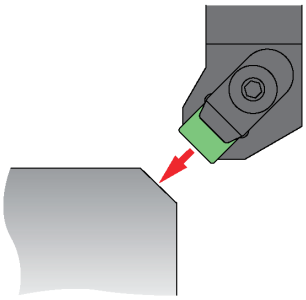
Minimize the amount of overhang

If the overhang is too long, chattering or chip loss will occur.



Insert chipping protection

Before machining, be sure to chamfer the corners of the workpiece. Machining sharp corners of workpieces without chamfering will result in chipping or defects of the inserts.



No dwell allowed.

Please note that if the insert is in contact with the workpiece at zero feed, wear will progress significantly.

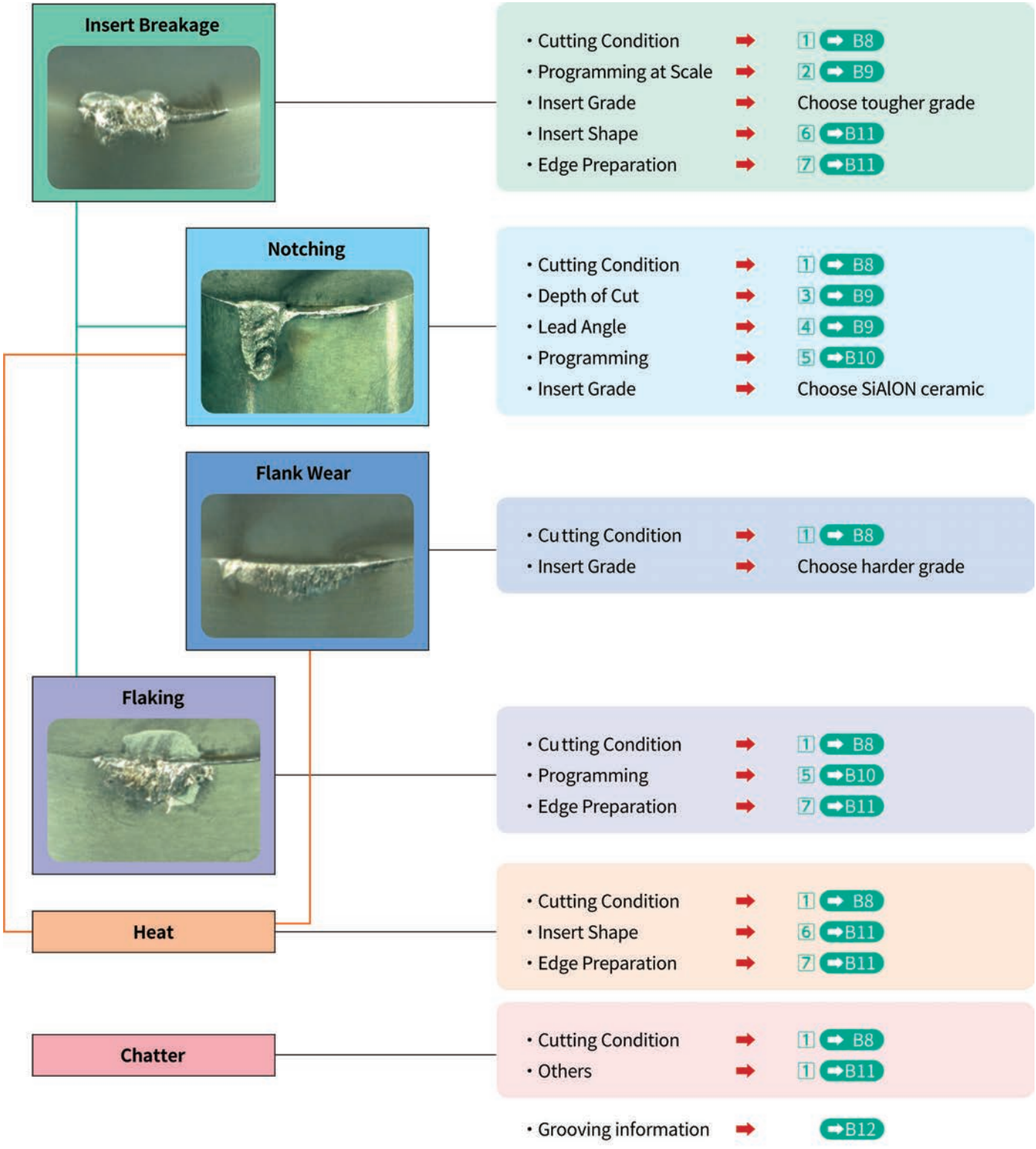
Coolant

WET machining is recommended when using BIDEMICS, SiAlON-based ceramics, or whisker ceramics in turning.
However, DRY machining may be more effective in case of strong interrupted machining.
When using SiAlON ceramics (SX3, SX7, SX9) in milling, be sure to use DRY machining.

Cutting Edge Treatment

Sharp edge preparation is required when machining heat-resistant alloys, but in the case of ceramic inserts, minute angle chamfering or round honing is better for wear resistance, especially border wear resistance.

Troubleshooting



Heat-resistant alloy machining

Troubleshooting

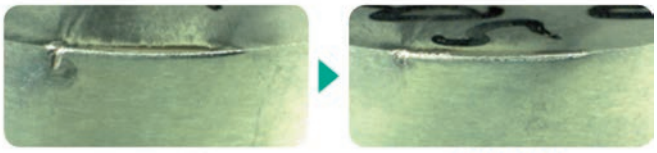
Adjustment of cutting conditions

		Cutting speed (m/min)		Feed rate (mm/rev)		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

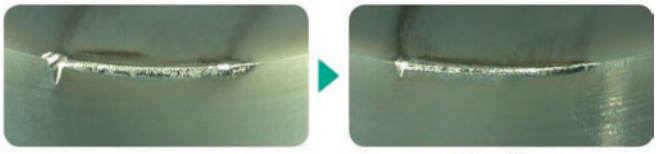
Result

「a」 WA1 : Increase cutting speed



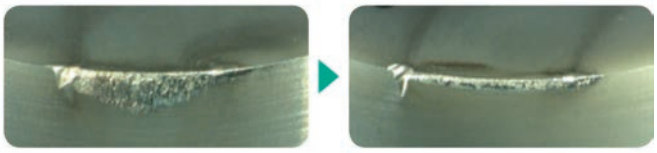
150m/ min → 550m/ min

「b」 SX7 • SX3 • SX9 • SX5 : Increase feed rate



0.2mm/rev → 0.4mm/rev

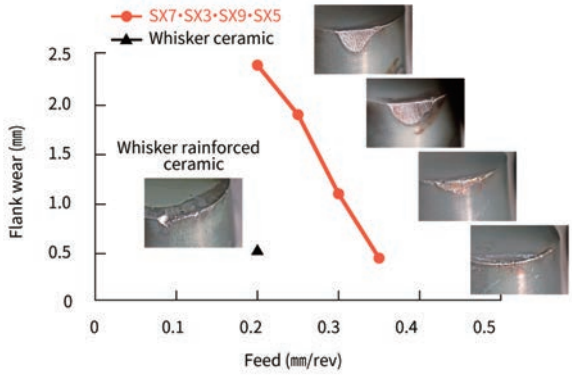
「c」 SX7 • SX3 • SX9 • SX5 : Decrease cutting speed



400m/min → 150m/ min

「d」 SX7 • SX3 • SX9 • SX5 : Increase feed rate

Feed rate increased decreases wear amount of SiAlON



Cutting condition
Work material : Inco718
Insert shape : RNGN120700
Cutting Speed : 250m/min
Depth of Cut : 2.0mm
WET

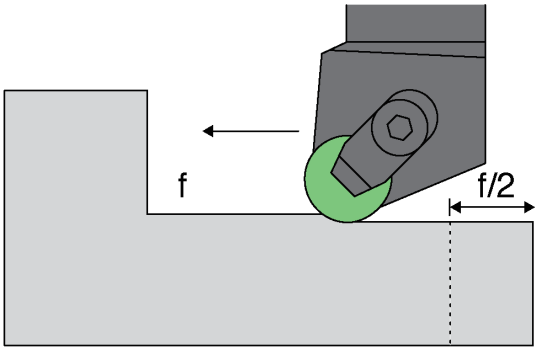
When using the SX7 & SX3 & SX9 & SX5, increased feed is necessary to increase the wear resistance of the tool. By increasing the feed rate and taking advantage of the high tool material strength of the SX7 & SX3 & SX9 & SX5, the number of times the tool and workpiece material rub against each other can be reduced, thereby reducing wear. In addition, higher feed rates shorten cycle time, increasing productivity and profitability.

Note: When machining corner R, reduce feed rate by 25% to prevent insert defects.

Machining with scale

If inserts break in the early stages of scale machining, high cutting speeds and feed rates may be the cause.

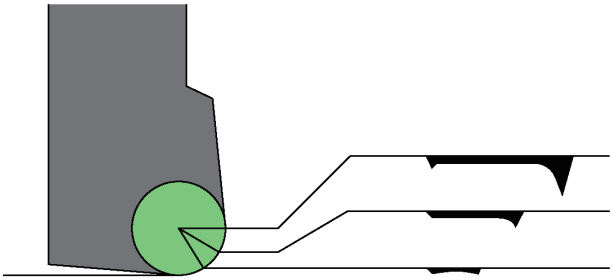
Understanding the hardness of the work material is the key to a successful cutting process. Many machining operators do not know the hardness of their work material. This causes them to spend a lot of time finding the optimum cutting conditions for test machining. The higher the hardness of the work material, the lower the cutting speed should be. Also, where there is scale on the workpiece surface, the cutting speed and feed rate must be reduced by 25%. By changing the machining program in this way, excessive tool damage can be reduced.



Depth of cut

As shown in the figure on the right, as the depth of cut increases, the amount of wear, especially border wear, increases. In order to reduce border wear and extend tool life, it is necessary to control the depth of cut.

The table below shows the maximum depth of cut for RN inserts and the maximum depth of cut by corner R size. Please refer to these values to determine the depth of cut.



Recommended depth of cut

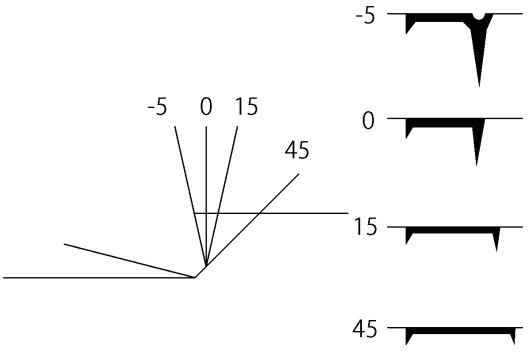
IC size of RN insert	Max. depth of cut	*Corner R size	Max. depth of cut
φ6.35mm	~1.5mm	0.8	0.2mm
φ9.525mm	~2.3mm	1.2	0.3mm
φ12.7mm	~3.2mm	1.6	0.4mm
φ25.4mm	~6.4mm	2.4	0.6mm

The optimum depth of cut is 5-15% of the insert diameter.
*In case of lead angle: 0°

Lead angle

In the machining of heat-resistant alloys, the larger the lead angle, the less wear is likely to occur. Also, the larger the lead angle, the more cutting resistance is distributed over a wider area of the insert, which reduces border wear and at the same time improves tool life and workpiece surface roughness.
In addition, the larger the lead angle, the better the chip control. In the case of the SX9 inserts, which have excellent chip resistance, higher feed rates reduce wear and machining time.

Effect of lead angle on wear pattern



Heat-resistant alloy machining

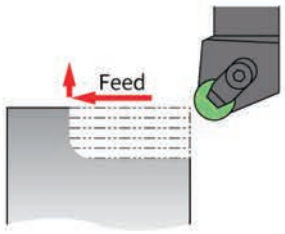
Troubleshooting

Continued

Programming

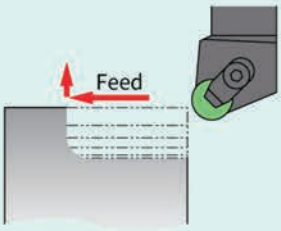
● Rough

Same Depth of Cut



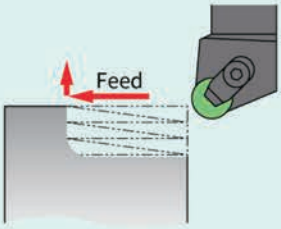
Note)
Notch wear on the insert cutting edge as shown is the result of multiple passes being taken at the same depth of cut. This type of wear will minimize tool life. The following programming examples will help to minimize this mode of failure.

Varying Depth of Cut



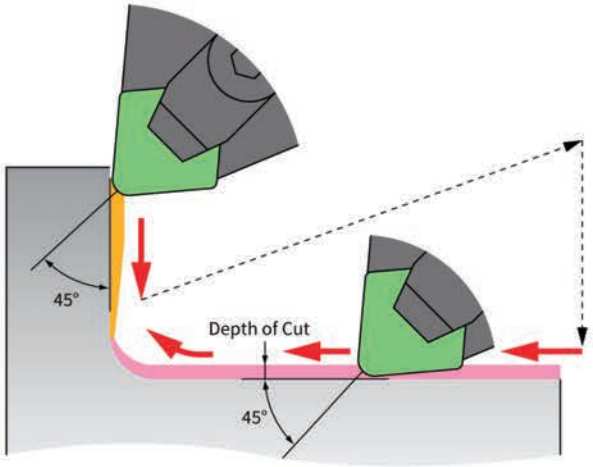
Note) Roughing while varying the depth of cut reduces notch wear because the point at which notch wear occurs changes each time.

Ramping



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

● Finish

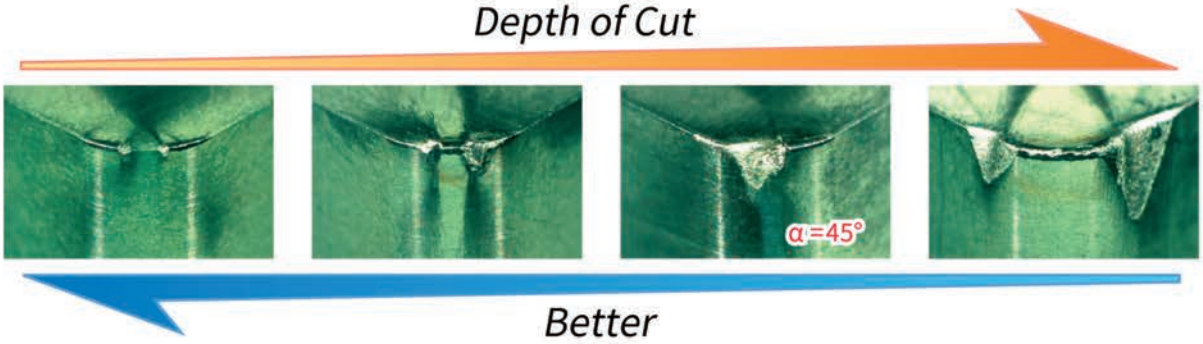


・α= 45°

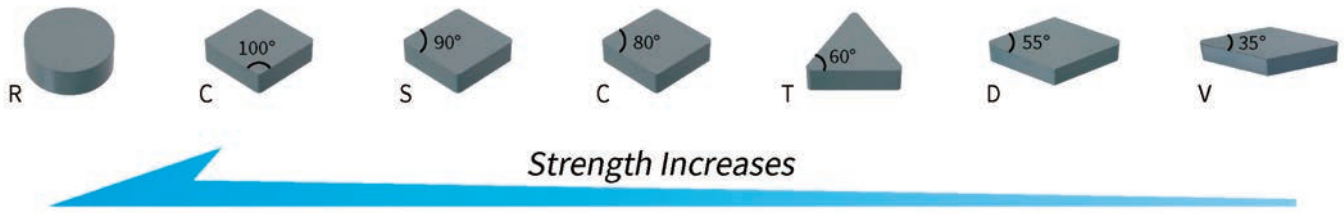
(mm)

Insert radius	DOC
0.4	0.12
0.8	0.23
1.2	0.35
1.6	0.47
2.4	0.70
3.2	0.94

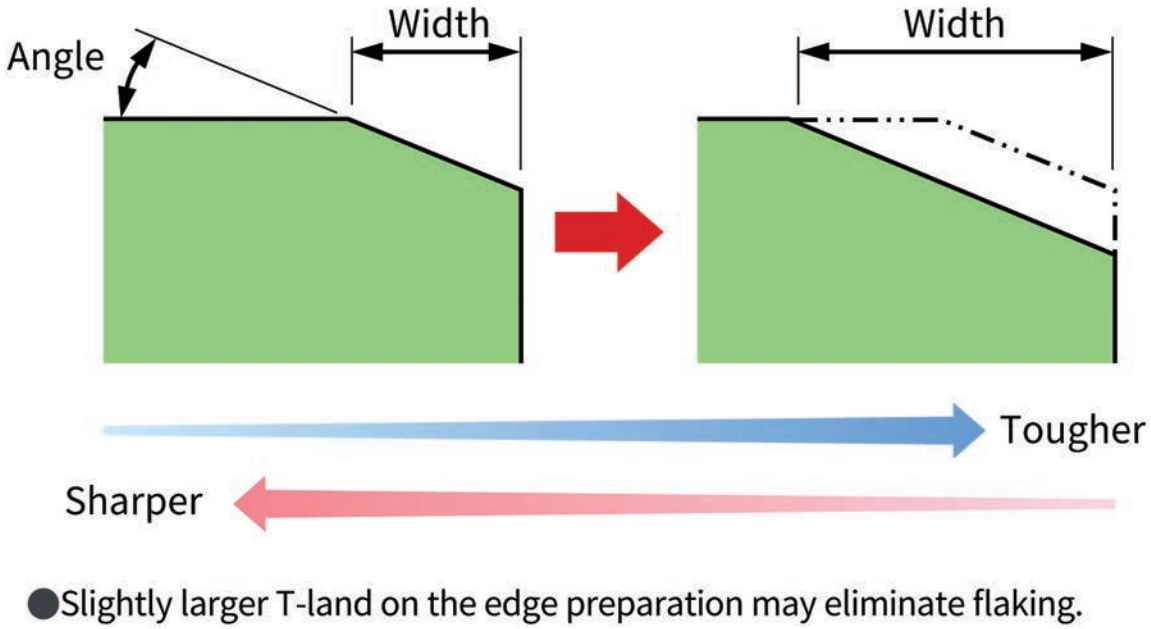
Note) The correct procedure is to take more material off during the previous roughing application.
Then remove the amount of stock suitable for the nose radius of the insert by staying
below the 45° mark of the corner radius.
This will minimize notching and allow a cut from both directions.



Insert shape



Edge treatment



●Slightly larger T-land on the edge preparation may eliminate flaking.

Prevention of chattering

When machining nickel-based heat-resistant alloys, chattering often occurs due to increased cutting resistance. Chatter is especially likely to occur when using a holder with a large overhang for copying or grooving, when machining thin-walled workpieces, or when using a machine with low rigidity, resulting in abnormal insert wear or sudden loss of inserts. Generally, increasing the cutting speed and decreasing the feed rate will reduce or eliminate chattering. In addition, the following methods are also effective.

- Increase cutting speed and reduce feed rate.
- Change to an insert grade with higher hardness.
- Change to an insert with a smaller inscribed circle or smaller corner radius.
- Change the cutting edge treatment to the sharpest possible shape.
- Change to a positive insert.
- Reduce the lead angle.
- Minimum overhang.
- Change the holder material to anti-vibration material.

Heat-resistant alloy machining

Key points for grooving

BIDEMICS and ceramic inserts enable high-speed, high-efficiency grooving. NTK offers a wide range of materials, including whisker ceramics, BIDEMICS, and SiAlON materials, which contribute to further productivity improvement and stable machining.

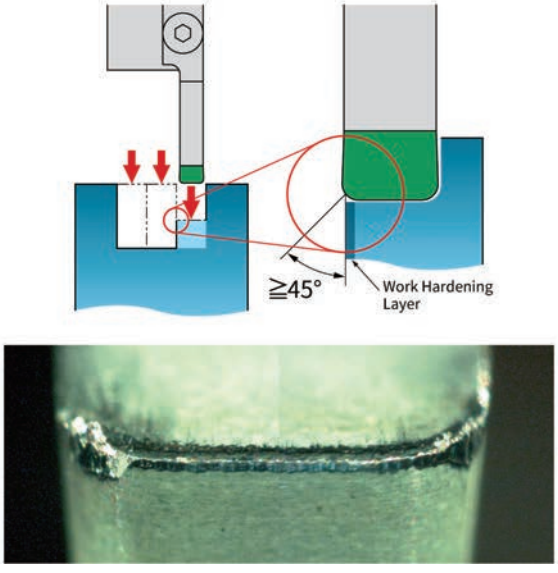
	JX1 / JX3	SX3 / SX7	SX5	WA1/WA5
				
Speed	●	●	●	●
Feed		●	●	
Versatility	●	●		●
Toughness		●	●	
	Can run at up to 1500 SFM. Double the speed of whisker	Double the feed of whisker	Best for scale and interruption	Versatile grade

● : 1st choice ● : 2nd choice

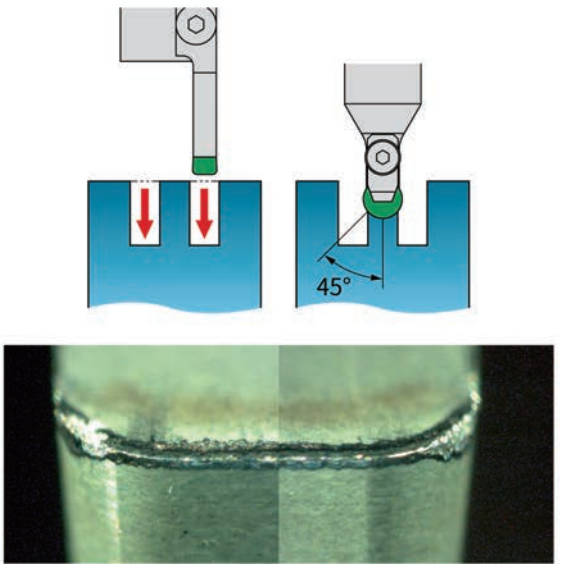
Application	Grade	Work material	Cutting speed (m/min) 180 240 300 360 420 480	Feed (mm/rev) 0.1 0.2 0.3 0.4 0.5	Depth of cut (mm) 0.5 1.0 1.5 2.0 2.5	Coolant
Grooving 	JX1 JX3	Overall	360(180-480)	0.07(0.05-0.1)	When using SX7 / SX5, increase feed rates 100% vs. Whisker Ceramics	WET
	SX5	Waspaloy	210(180-240)	0.15(0.07-0.17)		
	SX3 SX7	Overall	230(180-270)	1.1(0.07-0.15)		
	WA1	Overall	240(180-330)	0.07(0.05-0.1)		

*When applying JX1/JX3, increase speed to over **300** m/min*
*When applying SX3/SX7/SX5, increase feed rates **100%** vs. Whisker Ceramics*

Application



Change to



When grooving in multiple passes:
When grooving a final pass, the cutting edge contacts the work-hardened area.
This causes chipping of the corner radius and border wear.

The grooving is performed on both sides, leaving the center machining point.
Finally, grooving is performed with a strong insert shape such as RCGX type.

SX6 Silicon Nitride Ceramic ➡C20

Features

- 1st choice for roughing gray cast iron
- Applicable for wet cutting
- Excellent thermal shock resistance makes high speed milling possible

	SX6
Notching	◎
Flank Wear	
Toughness	○
Heat Shock	◎

Recommended Applications

- Gray cast iron — Rough — Turning and milling

Recommended Cutting Conditions

Work material	Purpose	Grade	Cutting speed (m/min)	Feed	Depth of cut (mm)	DRY	WET
Gray cast iron	Turning	SX6	500-1000	0.3-0.6(mm/rev)	0.5-3.5	●	●
	Milling	SX6	450-1200	0.07-0.25(mm/t)	0.5-3.5	●	○

HC1, HW2 Alumina Oxide Ceramic ➡C17・C18

Features

- 1st choice for finishing gray cast iron with no coolant
- Excellent wear resistance makes high speed finishing possible

Recommended Applications

- Gray cast iron — Finish — Turning
- Chilled liners — Rough / Finish — Turning（HW2）

Recommended Cutting Conditions

Work material	Purpose	Grade	Cutting speed (m/min)	Feed (mm/rev)	Depth of cut (mm)	DRY	WET
Gray cast iron	Turning	HC1	300-600	0.1-0.4	0.5-2.0	●	
		HW2	300-600	0.1-0.4	0.5-2.0	●	
Chilled liners	Turning	HW2	250-350	0.1-0.3	0.5-2.0	●	

SP9 CVD Coated Silicon Nitride Ceramic ➡C21

SP9
◎
◎

Features

- Extremely tough – Tough enough to rough cast iron with T01020 (0.1 × 20°) edge preparation
- Small edge preparation – Low tool pressure for stable precision machining
- SP9' s toughness makes higher feed rates possible
- Dramatically reduced flank wear due to CVD coating

Recommended Applications

- Gray cast iron — Rough — Turning and milling
- Ductile cast iron — Rough — Turning and milling

Recommended Cutting Conditions

Work material	Purpose	Grade	Cutting speed (m/min)	Feed (mm/rev, mm/t)	Depth of cut (mm)	DRY	WET
Gray cast iron	Turning	SP9	360-800	0.3-0.6	~3.5	●	○
	Milling		360-750	0.08-0.25	-6.0	●	○
Ductile cast iron	Turning	SP9	240-600	0.3-0.6	~3.5	○	●
	Milling		630-900	0.05-0.25	-6.0	●	○

HC2, HC6 TiC Ceramic ➡C22 ・C19 WA1 Whisker Reinforced Ceramic ➡C29

Features

- All grades make high speed finishing of cast iron possible
- Applicable for wet cutting conditions
- HC6 – Optimized for finishing ductile cast iron

Recommended Applications

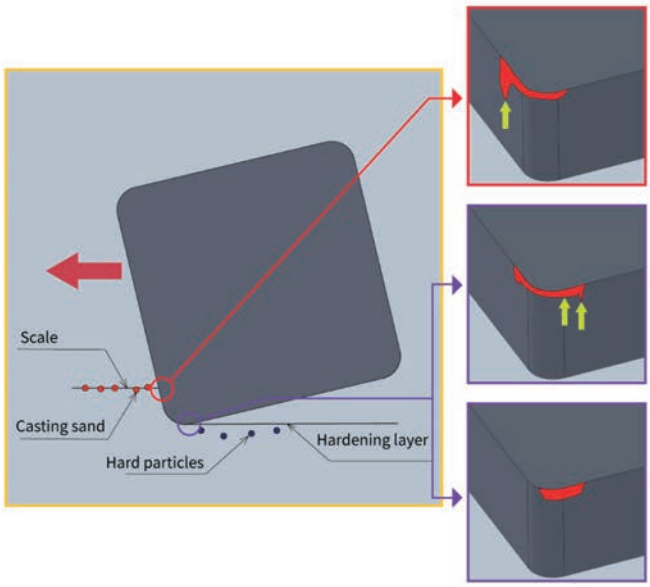
- Gray cast iron — Finish — Turning (HC2 ・ HC6 ・ WA1)
- Ductile cast iron — Finish — Turning (HC6)

Recommended Cutting Conditions

Work material	Purpose	Grade	Cutting speed (m/min)	Feed (mm/rev, mm/t)	Depth of cut (mm)	DRY	WET
Gray cast iron	Turning	HC2/HC6	360-630	0.1-0.4	-1.5	●	●
		WA1	360-630	0.1-0.4	-3.0	●	●
Ductile cast iron	Turning	HC6	180-450	0.1-0.3	-0.2	○	●

Cast iron / ductile cast iron machining

Recommended grade from cutting edge damage



Recommended when wear on the infeed side wear has progressed due to scale or casting sand, resulting in chipping of the cutting edge.

Roughing **SX6**

Recommended when VB wear has progressed due to the surface machining layer and hard particles, and the machined surface has deteriorated.

Roughing **SP9** *DRY machining

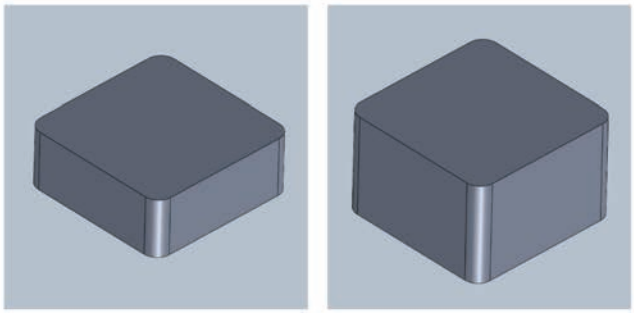
Finishing **HC2 HC6** *HC6 WET machining recommended

Recommended when the heat generated during DRY machining causes progressive wear, resulting in worsening of the machined surface and deterioration of dimensional accuracy.

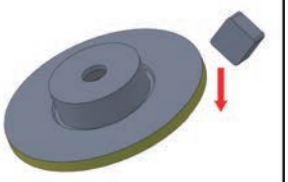
Finishing **HC1 HW2** *DRY machining

Effect of insert thickness

In machining with high cutting loads, such as roughing, a thicker insert thickness effectively reduces damage such as chipping, thus extending tool life.



Brake disc	
Work material	: FC250
Cutting speed(m/min)	: 550
Feed(mm/rev)	: 0.45
Depth of cut(mm)	: 2.5
Coolant	: DRY

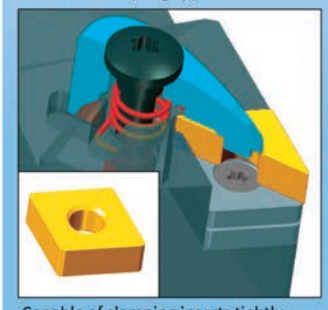


SNGN1207 type	100 pcs/corner stable
SNGN1204 type	50-70 pcs/corner unstable

Recommended clamping type when using ceramic inserts

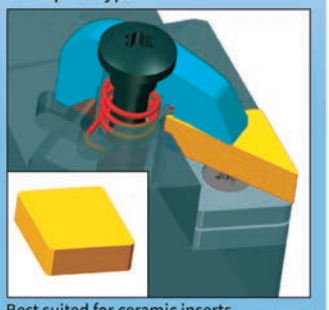
To maximize the performance of ceramic inserts, clamping rigidity of the inserts is important.
Select the clamping method best suited for your machining method.

Double clamping type



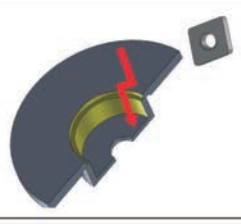
Capable of clamping inserts tightly. Supports cutting loads from any direction.

Clamp on type



Best suited for ceramic inserts. Not suitable where cutting loads are applied from each direction.

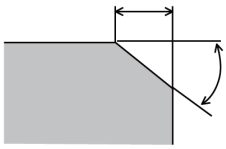
Brake disc	
Work material	: FC250
Cutting speed(m/min)	: 750
Feed (mm/rev)	: 0.35
Depth of cut(mm)	: 2.0
Coolant	: DRY



NTK double clamping type	100 pcs /corner
Lever lock type	45 pcs /corner

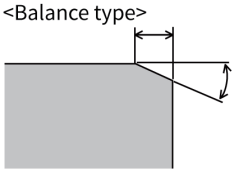
Use of different edge treatments

<Tougher type>



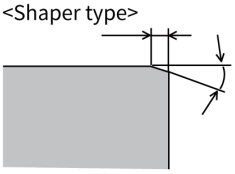
Width: Bigger
Angle: Bigger
Ex. width 0.3 × angle 35°

<Balance type>



Width: Medium
Angle: Medium
Ex. width 0.2 × angle 20°

<Shaper type>



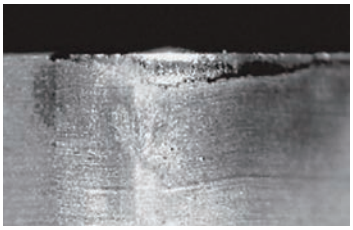
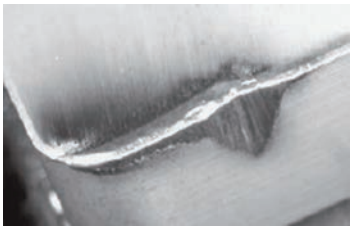
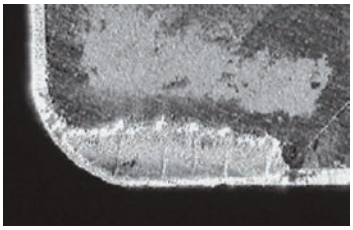
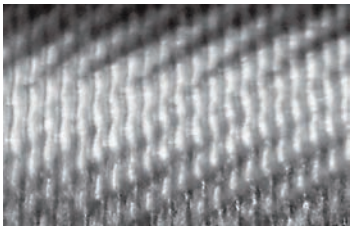
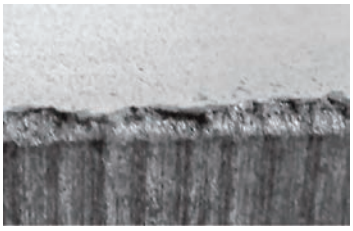
Width: Smaller
Angle: Smaller
Ex. width 0.1 × angle 15°

Case1.
Excessive notch wear or chipping in the early stages of machining.
→Tougher type is recommended.

Case2.
Unstable machining dimensions and excessive VB wear
→Sharper type is recommended.

Case3.
Micro chipping occured.
→Add round honing on the cutting edge is recommended.

Troubleshooting

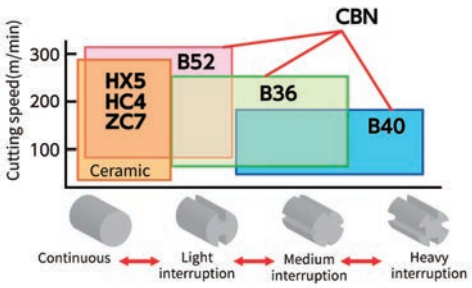
	Case study		Cause	Measure
Insert	VB wear		• Cutting speed too high • Feed rate too low • Insert shape incorrect • Incorrect insert grade	• Reduce cutting speed • Increase feed rate • Increase corner radius • Change to a grade with superior wear resistance
	Notch wear		• Incorrect insert grade • Cutter geometry incorrect • Insert shape incorrect	• Change to a grade with better wear resistance • Increase the lead angle • Change the geometry of inserts
	Thermal crack		• Cutting conditions incorrect • Incorrect insert grade	• Reduce cutting speed • Change from WET to DRY machining • Change to a grade with superior thermal shock resistance
	Fracture		• Cutting condition incorrect • Edge treatment incorrect • Use coolant	• Lower feed • Increase cutting edge treatment • Apply honing • Change from WET to DRY machining
	Cracked		• Insert clamped with incorrect seating	• Clean the mounting area and install according to the correct procedure • Tighten to the correct torque.
Work material	Chatter		• Low cutting resistance • Workpiece/tool less rigidity • Low cutting speed	• Lower feed • Smaller cutting edge treatment • Increase the clearance angle of the insert • Shorten tool overhang • Increase cutting speed
	Edge chipped		• High feed rate • Small cutting edge corner radius • Insert wear	• Lower feed • Increase the corner radius of insert • Use wiper inserts • Reduce cutting speed

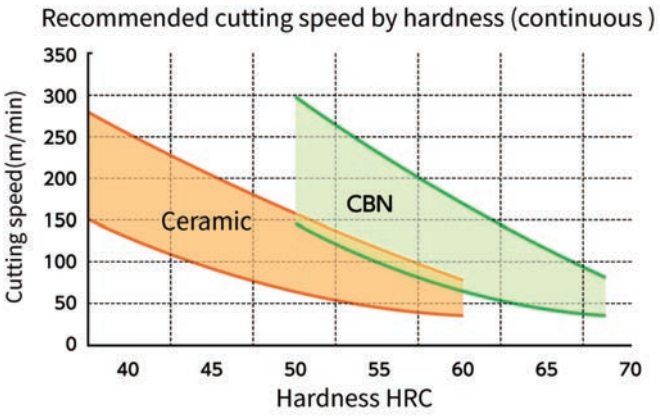
Hardened material machining

Features

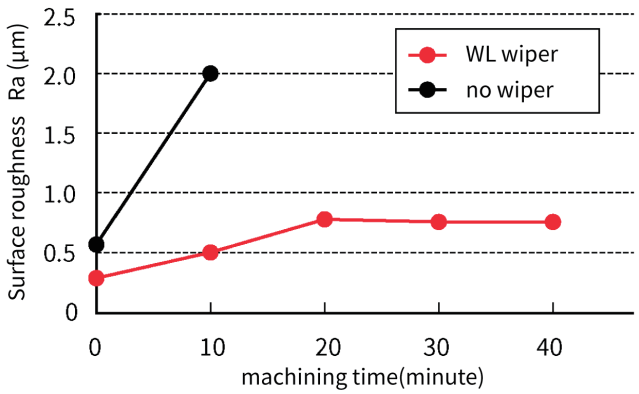
- ZC7 and HC4 ceramic materials have high wear resistance in machining of hardened materials.
- ZC7 is suitable for a wide range of machining applications from carburized hardened steel to induction hardened steel.
- HC4 offers high cutting performance in workpiece hardness in the HRc 55-70 range.
- Wiper inserts and inserts with breakers (AG) are available to improve machining efficiency.

Recommended insert grades and cutting speed





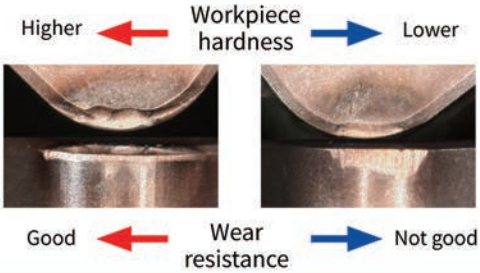
Advantages of Wiper Inserts



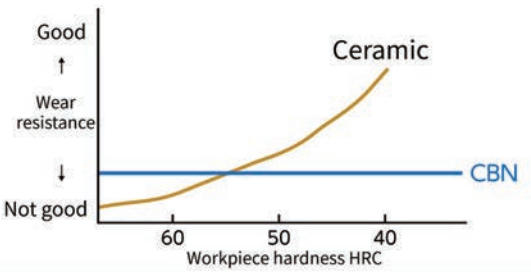
Recommended depth of cut and feed

Corner R	Depth of cut(mm)	Feed(mm /rev)
R0.4	0.15	0.05 ~ 0.08
R0.8	0.3	0.08 ~ 0.10
R1.2	0.4	0.10 ~ 0.13
R1.6	0.5	0.13 ~ 0.16
R6.35	2.0	0.16 ~ 0.25

Advantages of Ceramic Inserts

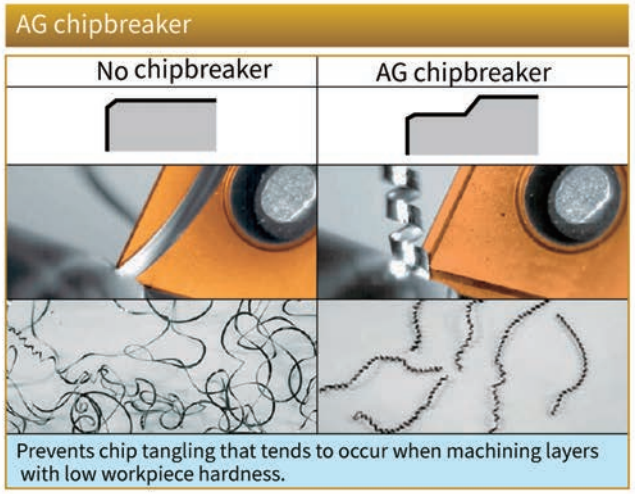


Lower workpiece hardness makes CBN tools wear out easier.

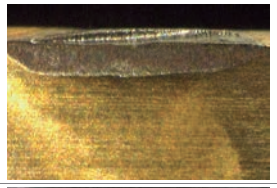
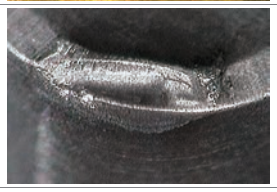
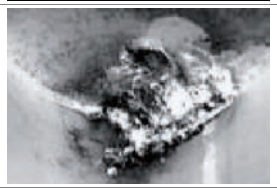
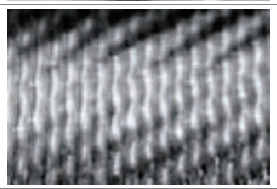


Ceramic has superior wear resistance to CBN in machining workpieces with hardnesses of HRC55 or less.

Key points for using different edge treatments

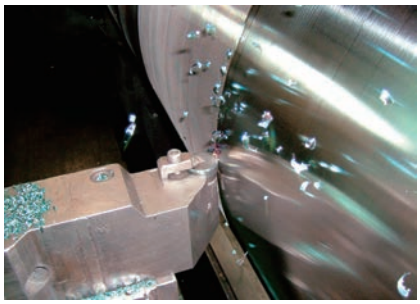


Troubleshooting

	Case		Cause	Measure
Insert	VB wear		●Cutting speed too high ●Feed too low ●Insert geometry unsuitable	●Reduce cutting speed ●Increase feed ●Increase the corner R
	Crater wear		●Unsuitable cutting conditions ●Insert geometry unsuitable	●Reduce cutting speed ●Smaller cutting edge treatment angle
	Flaking		●Insert geometry unsuitable	●Smaller cutting edge treatment angle ●Eliminate honing ●Lower feed ●Increase cutting speed
	Fracture		●Insert geometry unsuitable ●Unsuitable edge treatment ●Using coolant	●Lower feed ●Larger cutting edge treatment ●Add honing ●Change from WET to DRY machining
Work piece	Chatter		●Higher cutting resistance ●Not rigid enough for workpiece and tool ●Low cutting speed	●Lower feed ●Smaller cutting edge treatment angle ●Larger insert relief angle ●Shorten tool overhang ●Increase cutting speed
	Surface		●Higher feed rate ●Smaller insert corner R ●Insert wear	●Lower feed ●Larger insert corner R ●Using wiper inserts ●Reduce cutting speed

Mill roll machining

Features

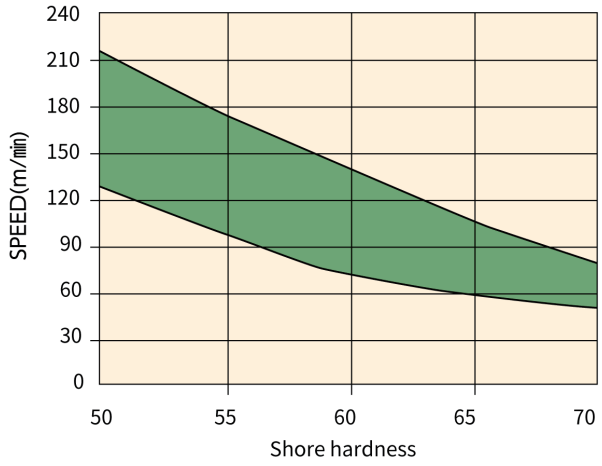


- In addition to the "HC2" ceramic material, which can be used for various types of mill roll machining, NTK offers the "HC5" and "HC7" ceramic materials, which improve machining efficiency even further.
- "WA1" has excellent wear resistance and is best suited for rough machining of carbide and high-hardness mill rolls.
- "ZC7" is applicable for a wide range of applications including carburized and induction hardened steels.
- "ZC4" has the best performance in hardened mill roll machining in the Shore hardness range of 74 to 97.

Recommended cutting conditions

Mill roll	Insert grade		Cutting speed(m/min)	Shore hardness Hs			Feed (mm/rev)	Depth of cut (mm)	DRY	WET
				55-65	65-72	72-				
Steel	ceramic	**HC7**	130-180	100%	80%	60%	0.1-0.3	0.5-2.0	●	
	ceramic	**HC5**	130-180	100%	80%	60%	0.1-0.3	0.5-2.0	●	
	ceramic	**HC2**	100-130	100%	80%	60%	0.1-0.3	0.5-2.0	●	
Chilled cast iron	ceramic	**HC7**	130-180	100%	80%	60%	0.1-0.3	0.5-2.0	●	
	ceramic	**HC5**	130-180	100%	80%	60%	0.1-0.3	0.5-2.0	●	
	ceramic	**HC2**	100-130	100%	80%	60%	0.1-0.3	0.5-2.0	●	
Ductile cast iron	ceramic	**HC7**	90-180	100%	80%	60%	0.1-0.3	0.5-2.0	●	
	ceramic	**HC5**	90-180	100%	80%	60%	0.1-0.3	0.5-2.0	●	
	ceramic	**HC2**	70-130				0.1-0.3	0.5-2.0	●	
Carbide	CBN	**B30**	30-60				0.1-0.3	0.2	●	
	Whisker ceramic	**WA1**	40-150				0.1-0.3	0.25-2.0	●	
CPM	ceramic	**ZC4**	120-150				0.1-0.3	0.6-2.0	●	
	ceramic	**HC5**	120-150				0.1-0.3	0.6-2.0	●	
	ceramic	**HC7**	120-150				0.1-0.3	0.6-2.0	●	
Hardness Hs46-86 (continuous machining)	ceramic	**ZC7**	40-200	finish	finish	finish	0.07-0.2	0.1-0.7	●	●
Hardness Hs74-97 (continuous machining)	ceramic	**ZC4**	40-200	finish	finish	finish	0.07-0.2	0.1-0.7	●	●

Recommended cutting speed



Recommended feed rate

Corner R	Depth of cut (mm)	Feed (mm/rev)	
		Ra 0.8μm	Ra 1.6μm
0.4	-0.18	0.05-0.07	0.07-0.1
0.8	-0.4	0.07-0.1	0.1-0.13
1.2	-0.5	0.1-0.13	0.13-0.16
1.6	-0.8	0.1-0.14	0.15-0.2
6.35	-2.0	0.17-0.25	0.25-0.35

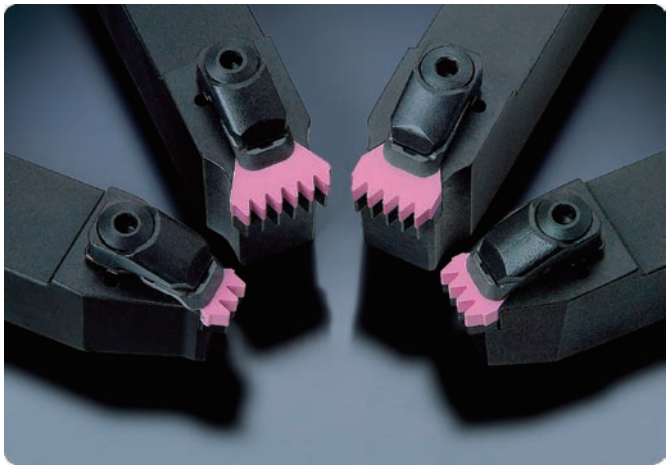
Key Points of Mill roll machining

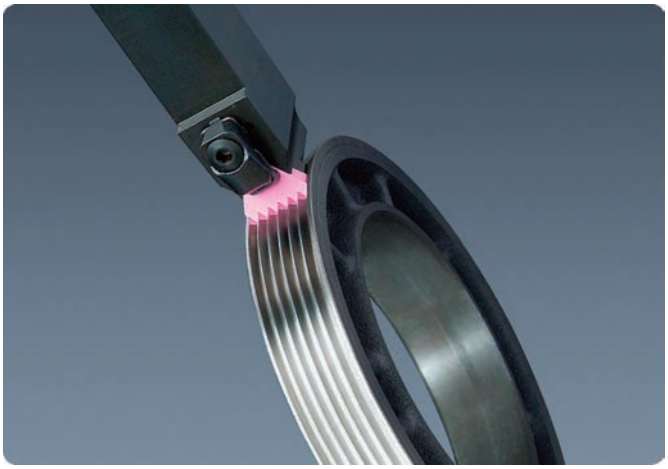
- Mill roll hardness is an important factor. The higher the hardness, the lower the cutting speed should be.
- RCGX type inserts are effective for high rigidity and tool cost reduction.
- When machining multiple passes in a single corner, the wear position can be distributed and notch wear can be reduced by varying the depth of cut.
- If chattering occurs, increase the feed rate. Chattering may be suppressed by adjusting the cutting speed.
- If chattering is severe, the machining point and tool cutting edge may not be centered.
- Chilled cast iron mill rolls and ductile cast iron mill rolls are generally low hardness, high strength materials. Even after use in rolling mills, the hardness of roll material rarely exceeds Hs67.
- HSS and CPM rolls typically have a hardness of Hs100 or higher. They are considered strong materials with high chromium and cobalt content. Considering the type and hardness of the roll material, machining at low speeds is required.

Types, applications, and features of mill rolls

Mill rolls	Applications	Features
Forged Rolls • Cr-Mo-based • High-speed-steel-based • Carbide-based	Bloom-milling at heavy rolling load. Work rolls for rough cold rolling, and rolls for reinforcement.	Strong and relatively high in heat resistance.
Cast Iron Rolls • Carbide-based	Semi-rolling or finishing that requires a very heavy load.	More wear-resistant and high-heat-resistant than steel in between ordinary steel and cast-iron-based steel.
Cast Steel rolls • Adamite roll for deep profile • Chilled roll for boards and wire steel process • Grain roll for steel finishing process boards (Resistant to thermal crack) • Ductile roll for boards, profile steel, and bar wire steel process (Rolls for roughing and finishing use) • Special cast iron roll	Wide range of applications from bloom-milling and semi-rolling to finishing.	Suitable for the applications that require heat resistance and strength. Suitable for the applications that require wear resistance.
Carbide rolls	• Pinch mills • Wire rod • Wire flattering or forming • ERWtube mills • Turks heads • Hot & Cold rolls • Work reducing rolls	Preferred in abrasive operations. High wear capabilities.

V-pulley machining



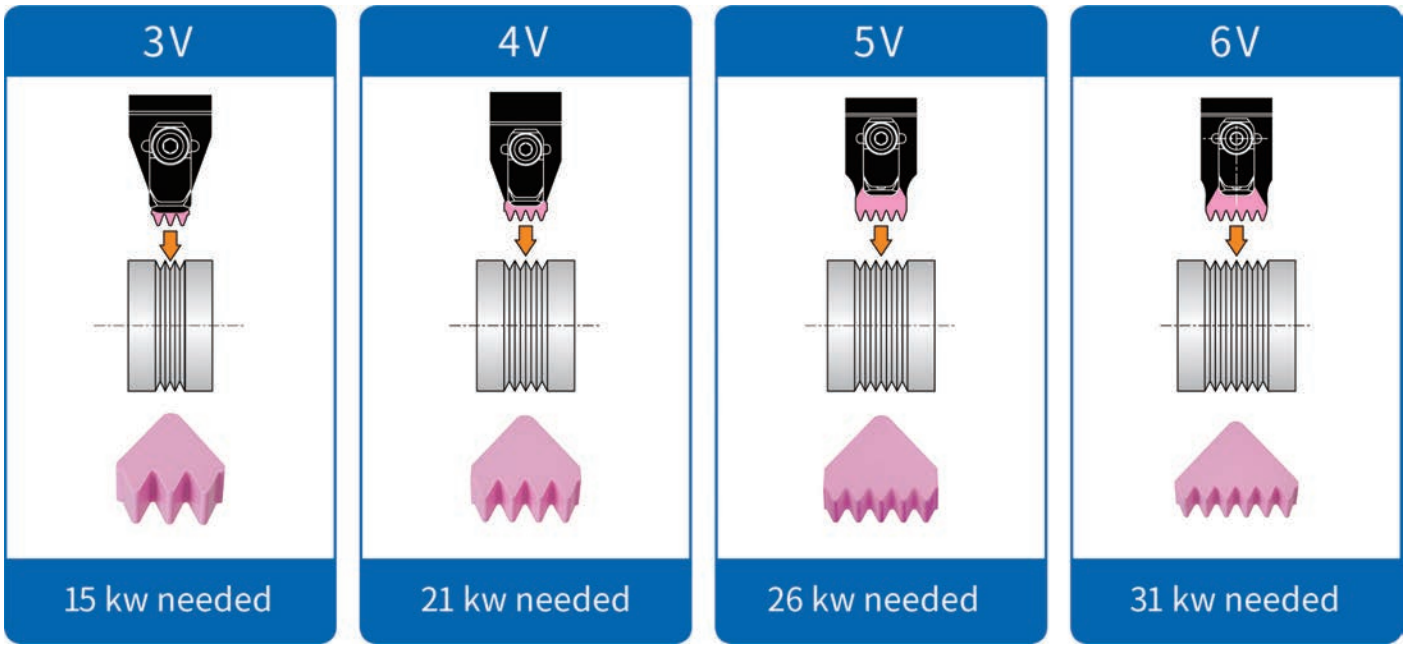


Features

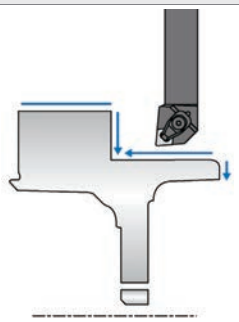
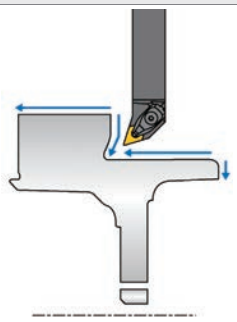
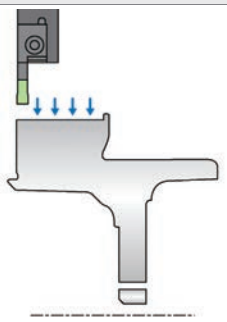
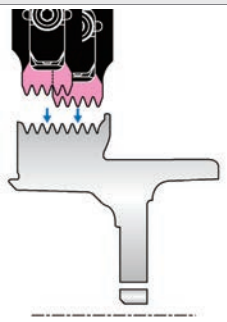
- **High-speed machining of poly-V pulleys**
- **Up to 6 grooves can be machined in a single pass**
- **High-precision inserts can be produced by profile machining**

Recommended Cutting Conditions

Material	Insert grade	Cutting speed (m/min)	Feed (mm/rev)	DRY	WET
cast iron	**HW2**	300-600	0.05-0.15	●	



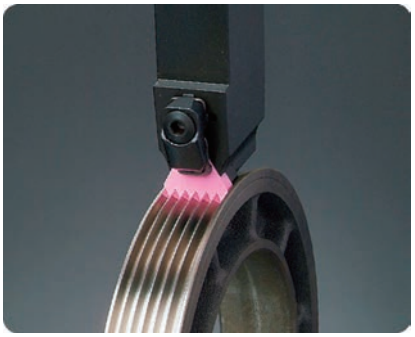
High-speed machining of poly V with NTK ceramic inserts

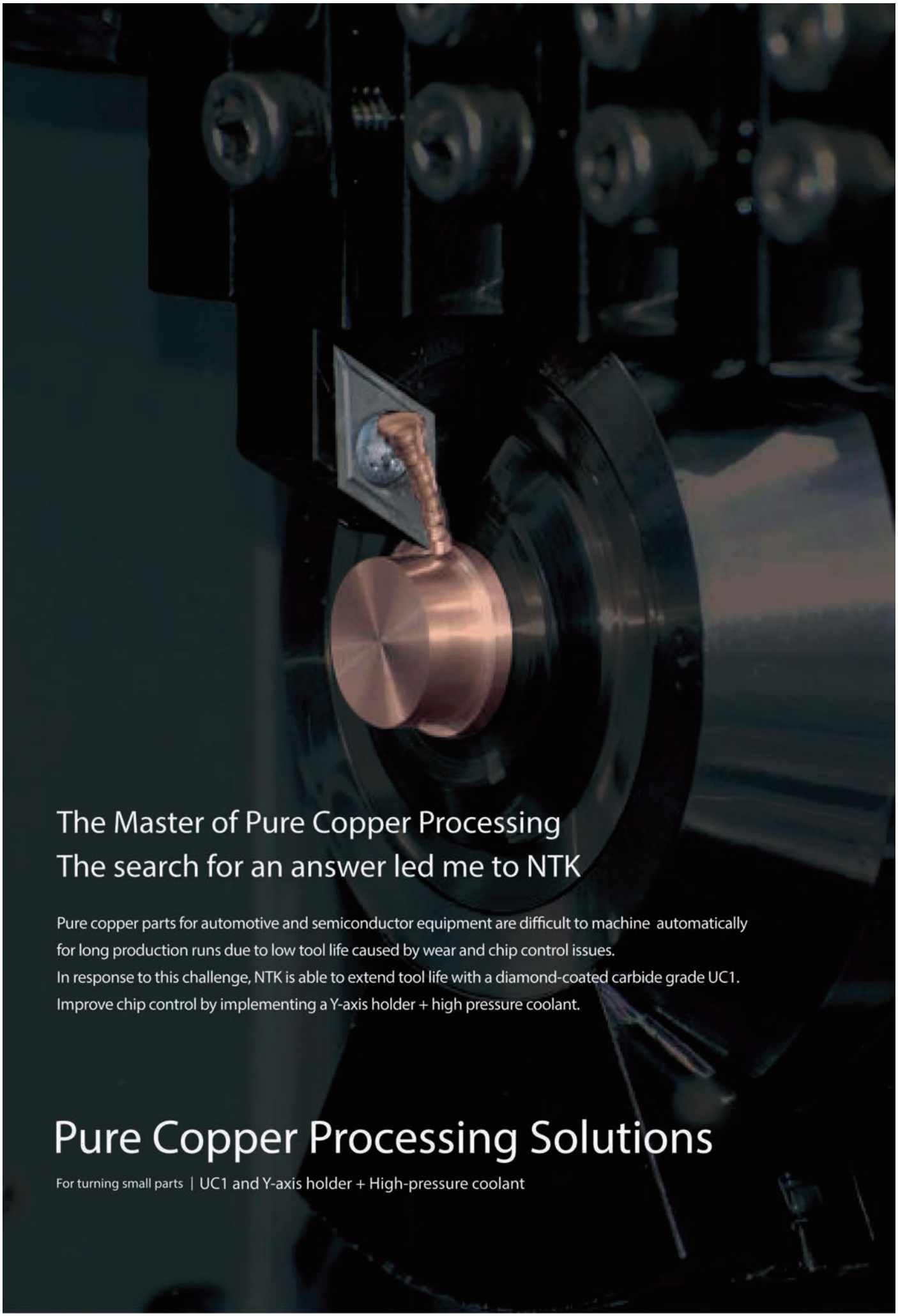
	Process #1	Process #2	Process #3	Process #4
	OD and Profile Roughing	OD and Profile Finishing	Plunge Grooving	Poly-V Grooving
Tooling				
Insert	SX6 CNGX120712T02020	HC6 DNGA150408T01225 SP9 DNGA150408T01020	WA1 VGW6250-2EX0001	HW2 PTM 53 K50504 ENB* 360-450 (420 m/min recommend)
Cutting speed	600-840	450-600(HC6) 540-720 (SP9)	300-420	(420 m/min recommend)
Feed	0.45-0.6	0.3-0.45 (HC6) 0.45-0.6 (SP9)	0.2-0.25	0.05-0.15
Depth of cut (mm)	2.0-3.0	0.5	-	-
Coolant	DRY (WET)	DRY (WET)	DRY (WET)	DRY
Pcs / corner	-300 pcs	-300 pcs	-300 pcs	-300 pcs

*Check the machine required power.

	3V	4V	5V	6V
Required HP kw	16kw	21kw	26kw	31kw

NTK's Ceramic Inserts ensure higher productivity and stable tool life for Damper-Pulley machining.





The Master of Pure Copper Processing
The search for an answer led me to NTK

Pure copper parts for automotive and semiconductor equipment are difficult to machine automatically for long production runs due to low tool life caused by wear and chip control issues.
In response to this challenge, NTK is able to extend tool life with a diamond-coated carbide grade UC1.
Improve chip control by implementing a Y-axis holder + high pressure coolant.

Pure Copper Processing Solutions

For turning small parts | UC1 and Y-axis holder + High-pressure coolant

NTK's Work Changing Concept

Diamond-coated carbide : Longer tool life with UC1
Improved chip control with Y-axis tool holder + high-pressure coolant

Performance

- **High purity, high hardness diamond coating, and excellent adhesion performance enable long-term stable machining.**
- **Improved chip control by applying Y-axis direction machining with high-pressure coolant.**

Applications

Pure copper (C1020/C1100) machined using Sliding head automatic lathes or CNC-lathes.

〔 Machining Property of C1100 Tough Pitch Copper 〕

Diamond-coated carbide: UC1 with excellent wear resistance & welding resistance is recommended because tooling tends to wear and the machined surface deteriorates due to welding, resulting in short tool life.

A good machined surface can be obtained by applying the appropriate cutting condition (low cutting depth/low feed) which reduces the chip thickness.

Case Study

Battery connector : C1100 ϕ10 - 20

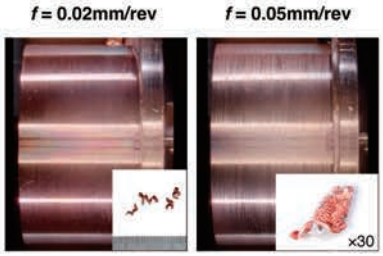
	NTK	Competitor
Tool	UC1 DCMT11T302 FNAM3	PVD super coat DCGT11T302 molded chipbreaker
Speed (m/min)	55 - 110	
Feed (mm/rev)	0.03	
DOC (mm)	0.2	
Coolant	WET	
Tool life	1000 pcs.	50 pcs.

Electrode part : C1100 ϕ12

	NTK	Competitor
Tool	UC1 DCMT11T301 FNAM3	PVD super coat DCGT11T301 molded chipbreaker
Speed (m/min)	80	
Feed (mm/rev)	0.05	
DOC (mm)	1.0	
Coolant	WET	
Tool life	2000 pcs.	100 pcs.

Machined Surface Comparison

Part material : C1100 v_c = 80m/min a_p = 1.0mm WET
Tool : DCMT11T302FNAM3 UC1



If the cutting conditions cause the chips to become thick (high cutting depth or high feed) then the machined surface will deteriorate due to chip clogging.

Cutting conditions

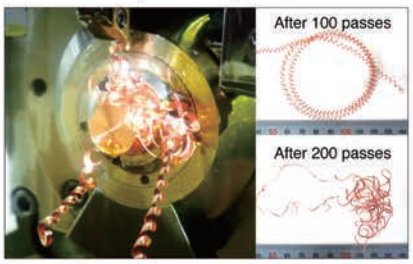
Grade	Material	Operation	Machining	Speed (m/min)	Feed (mm/rev)	DOC (mm)	WET
UC1	Tough pitch copper C1100	Turning	Rough - Finish	50 - 150	0.02 - 0.05	0.2 - 2.0	●

Using the cutting conditions shown, chips can be segmented and controlled with AM3 chipbreaker.
When machining at large depths of cut and high feed rate, select CL or ZP chipbreakers to suppress chip clogging.

〔 Machining Properties of C1020 Oxygen-free Copper 〕

Stable chip generation may be difficult with a chipbreaker alone, it is recommended to add a Y-axis holder + high pressure coolant

Example of Chip Issues

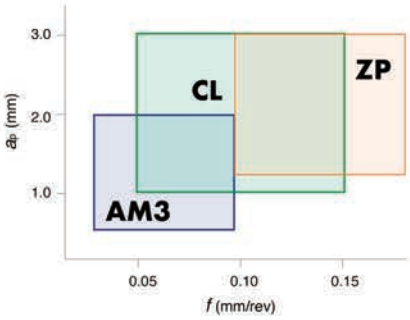


Even if good chip control is obtained at initial stage of machining, sudden chip entanglement can occur leading to tool wear.

Y-axis holder + high-pressure coolant

Chip tangling can be suppressed by Y-axis machining with high-pressure coolant.

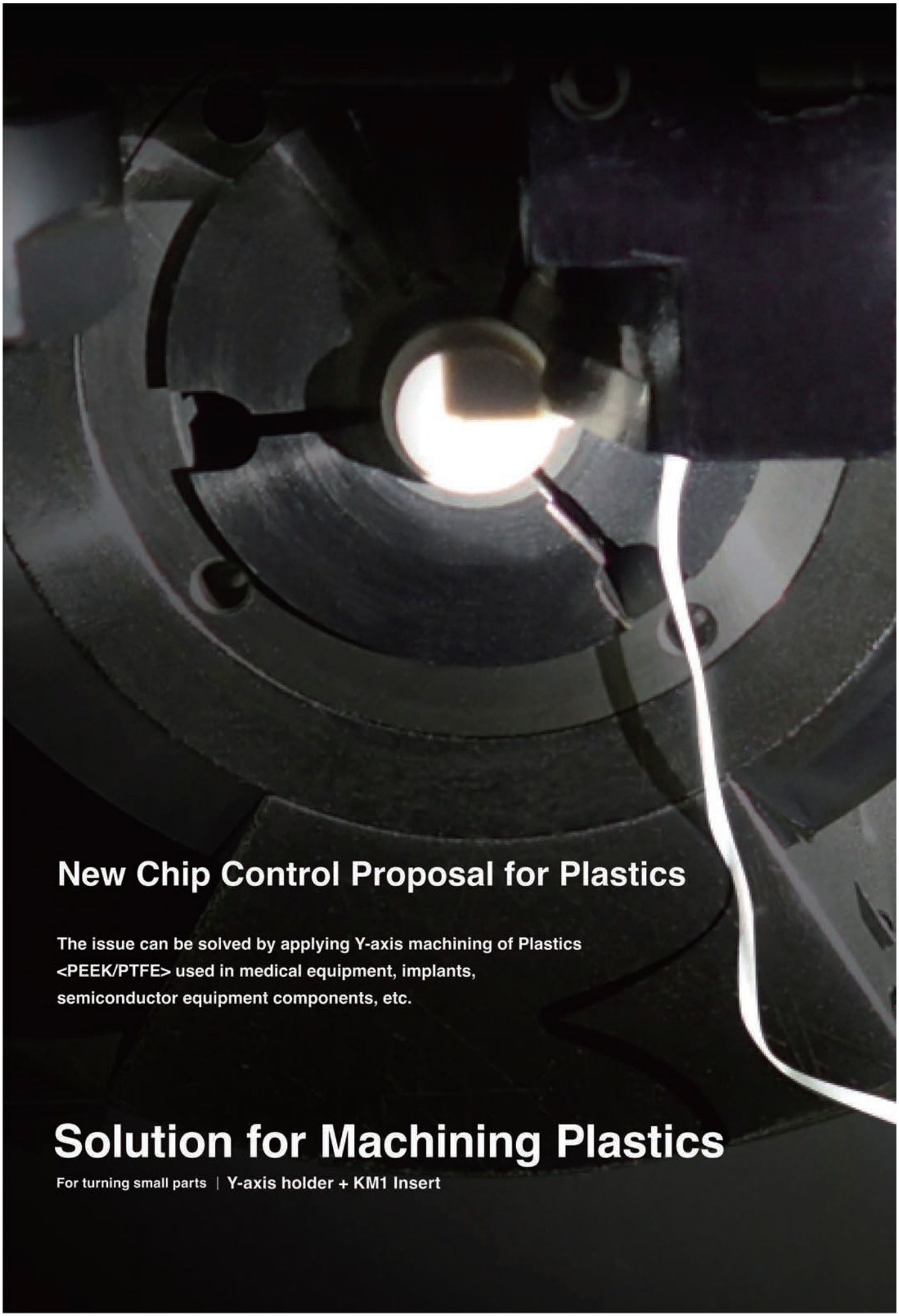
Chipbreaker machining range



Cutting Conditions

Grade	Material	Operation	Machining	Speed (m/min)	Feed (mm/rev)	DOC (mm)	WET
UC1 TM4	Oxygen-free copper C1020	Turning	Rough - Finish	50 - 150	0.02 - 0.20	0.5 - 3.0	●

Refer to content above to select cutting conditions and chipbreaker to obtain good chip control.
If you want longer tool life than PVD Carbide: TM4, use Diamond-Coated Carbide: UC1.



New Chip Control Proposal for Plastics

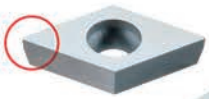
The issue can be solved by applying Y-axis machining of Plastics <PEEK/PTFE> used in medical equipment, implants, semiconductor equipment components, etc.

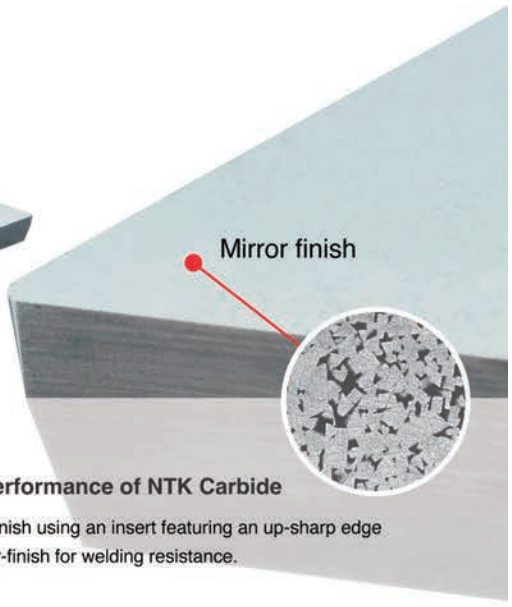
Solution for Machining Plastics

For turning small parts | Y-axis holder + KM1 Insert

Solution for Machining Plastics
The Realization of Stable Machining

Improved chip control with a Y-axis holder
High quality surface finish with KM1





Performance

- **Applying Y-axis machining eliminates chip control issues.**
- **Mirror-finish polished fine grain cemented carbide ensures an excellent surface finish**

Application Area

Automatic lathe (Gang type) machining plastic materials (PEEK/PTFE, etc.)

Machining Performance of NTK Carbide

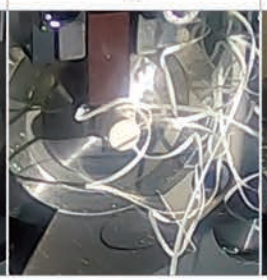
Excellent surface finish using an insert featuring an up-sharp edge and polished mirror-finish for welding resistance.

Recommended Cutting Conditions

Grade	Material	Operation	Machining	Cutting speed (m/min)	Feed (mm/rev)	DOC (mm)	DRY	AIR
KM1	Plastic (PEEK,PTFE,etc.)	Turning	Roughing - Finishing	50 - 150	0.05 - 0.10	0.5 - 3.0	●	●

Chip Control Performance

Material : PEEK(φ10) Cutting conditions : v_c=80m/min f=0.05mm/rev a_p=1.00mm

Machining approach	Standard machining		Y-axis machining	
Chipbreaker	Yes	No	Yes	No
Machining image				

Case Study

Medical implant : PEEK

	NTK	Competitor
Tool	KM1 VCGT11T302H No chipbreaker	Carbide VCGT11T302 Molded chipbreaker
Cutting speed (m/min)	100	
Feed (mm/rev)	0.06	
DOC (mm)	2.50	
Coolant	AIR	DRY
Tool life	**80 pcs.**	**40 pcs.**

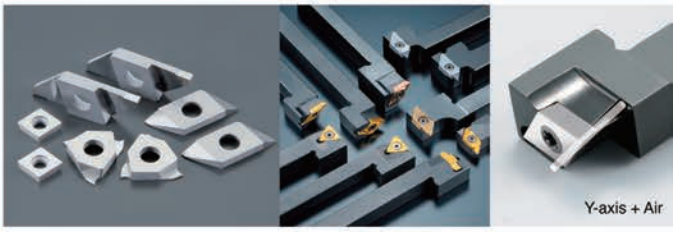
Automotive component : PEEK (with glass fiber)

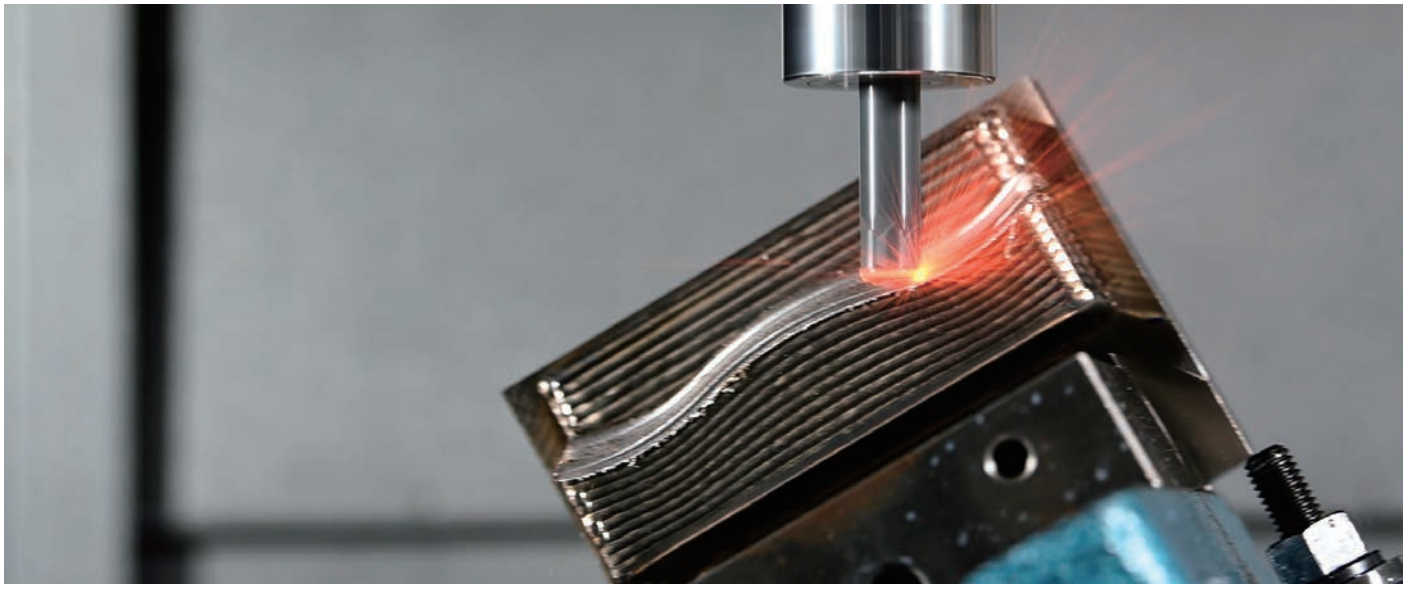
	NTK	Competitor
Tool	KM1 DCGT11T302H No chipbreaker	PVD Carbide VNMG160408 Molded chipbreaker
Cutting speed (m/min)	120	40
Feed (mm/rev)	0.08	0.05
DOC (mm)	0.25	
Coolant	AIR	DRY
Tool life	**3 pcs.**	**1 pc.**

Lineup

Application types : Front turning (ISO) / Back turning / Grooving / Cut-off / Threading / Boring

Standard holder	☐ 7 / 8 / 10 / 12 / 16 / 20 * ☐10~Coolant through available
Y-axis coolant through holder	☐ 12 / 16
Boring bar	Minimum machining diameter: Standard holder - from φ1mm * Coolant through holder - from φ2.2mm available





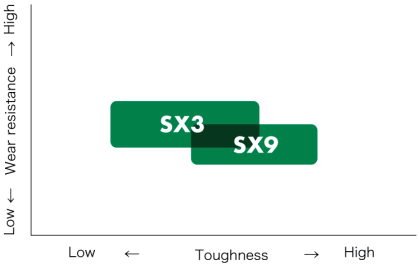
Additive manufacturing workpieces for rough machining | SiAlON ceramics

Additive manufacturing machining

Speedy rough machining of additive manufacturing workpieces (nickel-based alloys)
High-speed machining at about 10 times faster than with carbide tools.

Performance

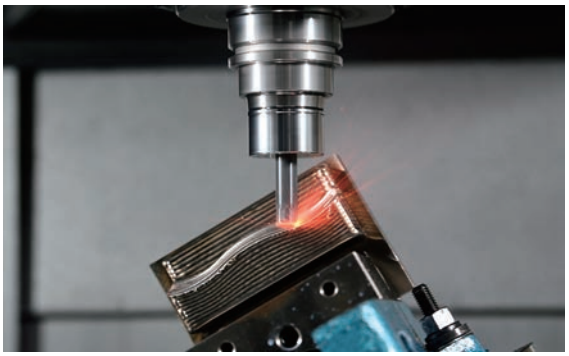
- Highly efficient machining of additionally built nickel-based alloys
- Ceramic grade with excellent toughness
- High-speed and stable machining
- Lineup of milling and end milling tools



Applications

- Workpiece built up by Nickel-based alloys
- Milling/End milling for rough machining

End milling



Helical milling