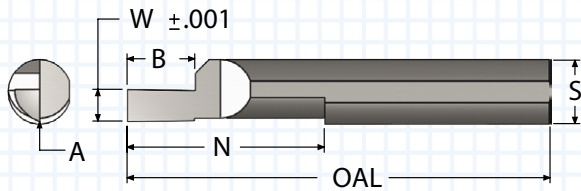


# FACE GROOVE TOOLS - SOLID CARBIDE



- ALTiN+ coating extends tool life
- Polished flute face for maximum performance
- Precision ground flat for guaranteed tool orientation
- Made with premium submicron grade carbide

"A" MIN. DIA.	"W" GROOVE WIDTH	"B" GROOVE DEPTH	"N" NECK RELIEF	"S" SHANK DIA.	OAL	ORDER #		EDP #	
						UNCOATED	ALTiN+	UNCOATED	ALTiN+
0.135	0.015	0.040	0.400	0.125	1.50	FG125-015	FG125-015A	220001	220013
0.135	0.020	0.050	0.400	0.125	1.50	FG125-020	FG125-020A	220004	220016
0.135	0.025	0.050	0.400	0.125	1.50	FG125-025	FG125-025A	220007	220019
0.135	0.030	0.060	0.400	0.125	1.50	FG125-030	FG125-030A	220010	220022
0.195	0.035	0.070	0.500	0.1875	2.00	FG187-035	FG187-035A	220025	220037
0.195	0.040	0.080	0.500	0.1875	2.00	FG187-040	FG187-040A	220028	220040
0.195	0.045	0.090	0.500	0.1875	2.00	FG187-045	FG187-045A	220031	220043
0.195	0.050	0.100	0.500	0.1875	2.00	FG187-050	FG187-050A	220034	220046
0.260	0.021	0.050	0.750	0.250	2.50	FG250-020	FG250-020A	220049	220061
0.260	0.031	0.060	0.750	0.250	2.50	FG250-030	FG250-030A	220052	220064
0.260	0.041	0.080	0.750	0.250	2.50	FG250-040	FG250-040A	220055	220067
0.260	0.051	0.100	0.750	0.250	2.50	FG250-050	FG250-050A	220058	220070
0.320	0.031	0.060	1.000	0.3125	2.50	FG312-030	FG312-030A	220073	220085
0.320	0.041	0.080	1.000	0.3125	2.50	FG312-040	FG312-040A	220076	220088
0.320	0.051	0.100	1.000	0.3125	2.50	FG312-050	FG312-050A	220079	220091
0.320	0.063	0.130	1.000	0.3125	2.50	FG312-062	FG312-062A	220082	220094
0.385	0.031	0.060	1.125	0.375	2.50	FG375-030	FG375-030A	220097	220109
0.385	0.063	0.130	1.125	0.375	2.50	FG375-062	FG375-062A	220100	220112
0.385	0.094	0.190	1.125	0.375	2.50	FG375-093	FG375-093A	220103	220115
0.385	0.126	0.250	1.125	0.375	2.50	FG375-125	FG375-125A	220106	220118
0.510	0.063	0.130	1.250	0.500	3.00	FG500-062	FG500-062A	220121	220133
0.510	0.094	0.190	1.250	0.500	3.00	FG500-093	FG500-093A	220124	220136
0.510	0.125	0.250	1.250	0.500	3.00	FG500-125	FG500-125A	220127	220139
0.510	0.157	0.310	1.250	0.500	3.00	FG500-156	FG500-156A	220130	220142
0.635	0.063	0.130	1.500	0.625	3.50	FG625-062	FG625-062A	220145	220160
0.635	0.094	0.190	1.500	0.625	3.50	FG625-093	FG625-093A	220148	220163
0.635	0.126	0.250	1.500	0.625	3.50	FG625-125	FG625-125A	220151	220166
0.635	0.157	0.310	1.500	0.625	3.50	FG625-156	FG625-156A	220154	220169
0.635	0.188	0.375	1.500	0.625	3.50	FG625-187	FG625-187A	220157	220172
0.760	0.063	0.130	1.750	0.750	4.00	FG750-062	FG750-062A	220175	220193
0.760	0.094	0.190	1.750	0.750	4.00	FG750-093	FG750-093A	220178	220196
0.760	0.126	0.250	1.750	0.750	4.00	FG750-125	FG750-125A	220181	220199
0.760	0.157	0.310	1.750	0.750	4.00	FG750-156	FG750-156A	220184	220202
0.760	0.188	0.375	1.750	0.750	4.00	FG750-187	FG750-187A	220187	220205
0.760	0.251	0.500	1.750	0.750	4.00	FG750-250	FG750-250A	220190	220208

THREAD MILLS

SINGLE POINT TOOLS  
GROOVING

INDEXABLE TOOLS

PORT - CAVITY

SPECIALTY

# SOLID CARBIDE BORING BAR FEED AND SPEED CHART

MATERIAL	HB/Rc	SPEED (SFM)		FEED IPR	CUTTING CONDITIONS					
		UNCOATED	ALTiN+		TOOL DIAMETER					
					.015-.045	.050-.100	.110-.160	.180-.230	.290-.320	.360+
					MAX DOC	MAX DOC	MAX DOC	MAX DOC	MAX DOC	MAX DOC
CAST IRON	160 HB	75-200	200-550	.0005-.010	0.006	0.008	0.010	0.014	0.020	0.031
CARBON STEEL	18 Rc	75-200	200-450	.0005-.007	0.003	0.005	0.006	0.008	0.012	0.017
ALLOY STEEL	20 Rc	75-200	200-425	.0005-.007	0.003	0.004	0.005	0.007	0.010	0.015
TOOL STEEL	25 Rc	75-175	175-300	.0005-.005	0.002	0.003	0.004	0.006	0.008	0.012
300 STAINLESS STEEL	150 HB	75-175	175-350	.0005-.005	0.003	0.003	0.004	0.006	0.008	0.013
400 STAINLESS STEEL	195 HB	75-210	130-420	.0005-.005	0.002	0.003	0.004	0.006	0.008	0.012
HIGH TEMP ALLOY (Ni & Co BASE)	20 Rc	50-130	130-300	.0005-.004	0.002	0.003	0.003	0.005	0.007	0.010
TITANIUM	25 Rc	50-120	120-275	.0005-.005	0.003	0.004	0.005	0.006	0.009	0.014
HEAT TREATED ALLOYS (38-45Rc)	40 Rc	50-100	100-200	.0005-.005	0.002	0.002	0.003	0.004	0.006	0.009
ALUMINUM	100 HB	75-250	250-750	.0005-.015	0.011	0.015	0.019	0.026	0.038	0.056
BRASS, ZINC	80 HB	75-300	250-650	.001-.010	0.009	0.012	0.015	0.021	0.030	0.045

SFM = Surface Feet Per Minute    DOC = Depth of Cut    IPR = Inches Per Revolution

Starting parameters only. Length-to-diameter ratios, setup, and machine rigidity may affect performance.

$$\text{SFM} = .262 \times \text{DIAMETER} \times \text{RPM}$$

$$\text{RPM} = 3.82 \times \text{SFM} \div \text{DIAMETER}$$

$$\text{IPM} = \text{FPT} \times \text{Number of Teeth} \times \text{RPM}$$

$$\text{Meters/Min} = \text{SFM} \times .3048$$

$$\text{Millimeters/Rev} = \text{IPR} \times 25.40$$

# SOLID CARBIDE BORING TROUBLESHOOTING

PROBLEM	CAUSE	SOLUTION
RAPID FLANK WEAR	CUTTING CONDITIONS	Check for excessive speed and feed - See chart.
	TOOL	Select a coated tool.
	PART	Make sure prior operation did not work harden the metal.
BUILT-UP EDGE	TOOL	Select a coated tool.
	CUTTING FORCE	Check for excessive feed rate (IPR) - See chart.
	HEAT	Use the SCT coolant holder. If coolant is not available, use shop air and a coated tool.
CORNER BREAKAGE	CUTTING CONDITIONS	Check for excessive feed and speed and depth of cut - see chart.
	TOOL	Select a tool with a radius. A radius is stronger than a sharp corner.
	PART	Check the drilled hole.
SURFACE TOO ROUGH	CUTTING CONDITIONS	Check for excessive feed rate (IPR) - See chart.
	BUILT-UP EDGE	See above (Built-Up Edge).
CHATTER	SET UP	Set tool above center. Reduce the overhang ratio. Clamping length should be at least 3x the boring bar diameter. Change the speed of the machine. Speed change may break up harmonics and reduce chatter.
	BORING BAR	Select the largest diameter boring bar that will bore the required diameter.
TAPER SMALLER IN BACK	CHIP PACKING	If the boring bar is too large to allow chips to evacuate, then the chips may pack on the tool and cause the bar to deflect away from the bore.
	PROGRAM	If the taper is consistent, then the program can be altered to bore a taper in opposite direction resulting in a straight hole.
TAPER BIGGER IN BACK	CUTTING FORCES	Reduce forces. Deflecting bar below center causes hole to become larger.
	BUILT-UP EDGE	Built-up edge will cause the hole to become larger until the built edge breaks off, then the hole becomes smaller.
	PROGRAM	If taper is consistent, then the program can be altered to bore a taper in the opposite direction resulting in a straight hole.

# GROOVING TOOL FEED AND SPEED CHART

MATERIAL	HB/Rc	SPEED (SFM)		CUTTING CONDITIONS				
				TOOL DIAMETER				
		UNCOATED	ALTiN+	.060 -0.080	.090 -.120	.187	.250-.312	.375+
				MAX FPR	MAX FPR	MAX FPR	MAX FPR	MAX FPR
CAST IRON	160 HB	75-200	200-550	0.0010	0.0012	0.0017	0.0031	0.0044
CARBON STEEL	18 Rc	75-200	200-450	0.0007	0.0008	0.0011	0.0022	0.0030
ALLOY STEEL	20 Rc	75-200	200-425	0.0006	0.0007	0.0010	0.0019	0.0026
TOOL STEEL	25 Rc	75-175	175-300	0.0005	0.0006	0.0008	0.0015	0.0022
300 STAINLESS STEEL	150 HB	75-175	75-350	0.0006	0.0007	0.0010	0.0019	0.0026
400 STAINLESS STEEL	195 HB	75-210	130-420	0.0005	0.0006	0.0008	0.0016	0.0023
HIGH TEMP ALLOY (NICKEL & COBALT BASE)	20 Rc	50-130	130-300	0.0004	0.0005	0.0007	0.0013	0.0017
TITANIUM	25 Rc	50-120	120-275	0.0005	0.0006	0.0008	0.0016	0.0022
HEAT TREATED ALLOYS (38-45Rc)	40 Rc	50-100	100-200	0.0004	0.0004	0.0006	0.0011	0.0016
ALUMINUM	100 HB	75-250	250-750	0.0022	0.0026	0.0037	0.0065	0.0085
BRASS, ZINC	80 HB	250-300	250-650	0.0018	0.0021	0.0030	0.0053	0.0079

SFM = Surface Feet Per Minute

FPR = Feed Per Revolution

Starting parameters only. Length-to-diameter ratios, setup, and machine rigidity may affect performance.

## GROOVING TOOL TROUBLESHOOTING

PROBLEM	CAUSE	SOLUTION
RAPID FLANK WEAR	CUTTING CONDITIONS	Check for excessive speed - see chart.
	TOOL	Select a coated tool.
	PART	Make sure prior operation did not work harden the material.
BUILT-UP EDGE	TOOL	Select a coated tool.
	CUTTING FORCE	Check for excessive speed rate (IPR) - see chart.
	HEAT	Use the SCT coolant holder. If coolant is not available, use shop air and a coated tool.
CHATTER	CUTTING CONDITIONS	Reduce RPM and increase feed rate within the feed and speed chart parameters.
	CLAMPING	Clamping length should be a minimum of 3x the shank diameter in the tool holder. Check tool holding rigidity.
	TOOL	Hone cutting edge. A light hone (0.0001-0.0003 inch) will help keep force constant.
TOOL BREAKAGE	CUTTING CONDITIONS	Check for excessive feed rate (IPR) - see chart.
	CHIP PACKING	Stagger - Peck grooving.

# SINGLE POINT THREADING TECHNICAL CHART

MATERIAL	HB/Rc	SPEED (SFM)		FIRST PASS DEPTH					
		UNCOATED	ALTiN+	TOOL DIAMETER					
				.040-.050	.060-.092	.120-.152	.180-.232	.290-.362	.373+
CAST IRON	160 HB	75-200	200-550	0.003	0.004	0.005	0.007	0.008	0.009
CARBON STEEL	18 Rc	75-200	200-450	0.003	0.005	0.006	0.007	0.008	0.009
ALLOY STEEL	20 Rc	75-200	200-425	0.003	0.004	0.005	0.006	0.007	0.008
TOOL STEEL	25 Rc	75-175	175-300	0.002	0.003	0.004	0.005	0.006	0.007
300 STAINLESS STEEL	150 HB	75-175	175-350	0.003	0.003	0.004	0.005	0.006	0.007
400 STAINLESS STEEL	195 HB	75-210	130-420	0.003	0.004	0.005	0.006	0.006	0.007
HIGH TEMP ALLOY (NICKEL & COBALT BASE)	20 Rc	50-130	130-300	0.002	0.003	0.003	0.004	0.005	0.005
TITANIUM	25 Rc	50-100	120-275	0.003	0.003	0.004	0.005	0.006	0.007
HEAT TREATED ALLOYS (38-45Rc)	40 Rc	50-100	100-200	0.002	0.002	0.003	0.004	0.004	0.005
ALUMINUM	100 HB	75-250	200-750	0.004	0.005	0.007	0.008	0.010	0.011
BRASS, ZINC	80 HB	75-300	250-650	0.003	0.005	0.006	0.007	0.008	0.009

Parameters are a starting point based on machinability rating at hardness listed.  
Check machinability rating of the material to be machined and adjust First Pass Depth.

## Helpful Formulas and Information

$$\text{PITCH} = \frac{1}{\text{TPI}}$$

TPI = Threads Per Inch

ACME Thread Depth = Pitch × 0.5

Stub ACME Thread Depth = Pitch × 0.3

NPT Pipe Thread Depth = Pitch × 0.76

Internal 60° Thread Depth = Pitch × 0.54

Feed Rate = Pitch × Number of Thread Starts

Minimum Depth per Pass should not be less than 0.0003

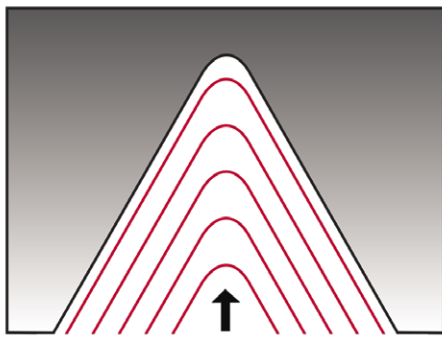
Threads not ending in a relief need at least one thread pitch length of pullout

Make sure feed rate calculation does not exceed the maximum feed rate of the machine

# SINGLE POINT THREADING TROUBLESHOOTING

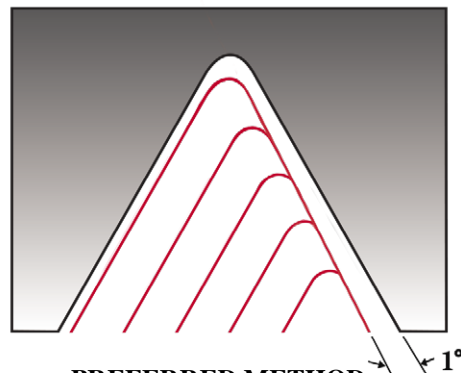
PROBLEM	CAUSE	SOLUTION
RAPID FLANK WEAR	CUTTING CONDITIONS	Check for excessive speed - see chart.
	PART	Make sure prior operation did not work harden the material.
	TOOL	Select a coated tool.
BUILT-UP EDGE	TOOL	Select a coated tool.
	CUTTING FORCE	Increase the number of passes.
	HEAT	Use the SCT coolant holder. If coolant is not available, use shop air and a coated tool.
CORNER BREAKAGE	CUTTING CONDITIONS	Reduce the depth-of-cut on the first pass.
	PROGRAM	If there is no thread relief, withdraw the tool on an angle.
	PART	End in thread relief.
CHIPS WRAPPING AROUND TOOL	TOOL	Use a tool that is at least 30% smaller than the hole diameter.

## RADIAL INFEEED



NOT RECOMMENDED

## MODIFIED FLANK



PREFERRED METHOD

Radial Infeed is not recommended. Modified flank at 1° is recommended.

For unfavorable length-to-diameter ratios or difficult-to-machine materials, the number of passes will need to be increased up to 40% more.

Depth of cut per pass should not be less than 0.0003 inch.

# SINGLE POINT CBN & PCD TECHNICAL & APPLICATION

## PCD TIPPED TOOL INFORMATION

SCT PCD tools and inserts are excellent for continuous cutting of a wide range of non-ferrous and non-metal materials. The products are precision ground for machining to sub-micron finishes with maximum tool life. PCD allows for higher cutting speeds with longer tool life.

SINGLE POINT TOOLS  
TECH INFO

MATERIAL	BHN/Rc	SPEED RANGE (SFM)	FEED IPR	SINGLE POINT PCD TIPPED BARS			
				TOOL DIAMETER			
				.120-160 MAX DOC	.180-.230 MAX DOC	.290-.320 MAX DOC	.360+ MAX DOC
LOW SILICON ALUMINUM	225-350 BHN	1000-5000	.001-.007	0.015	0.021	0.03	0.045
HIGH SILICON ALUMINUM	270-425 BHN	600-3000	.001-.007	0.015	0.021	0.03	0.045
METAL MATRIX COMPOSITIES	N/A	500-2000	.001-.007	0.008	0.012	0.02	0.03
COPPER ALLOYS, BRASS, BRONZE	80-120 BHN	750-3500	.001-.007	0.015	0.021	0.03	0.045
PRESINTERED TUNGSTEN CARBIDE	140-300 BHN	100-350	.001-.005	0.003	0.005	0.007	0.012
ACRYLICS	N/A	700-1500	.001-.007	0.015	0.021	0.03	0.045
FIBERGLASS	N/A	600-1000	.001-.007	0.012	0.02	0.03	0.045
GRAPHITES	N/A	600-1000	.001-.007	0.015	0.021	0.03	0.045
NYLON, PLASTIC	N/A	700-1500	.001-.007	0.015	0.021	0.03	0.045
HARD RUBBER	N/A	500-2500	.001-.007	0.015	0.021	0.03	0.045

APPLICATION GUIDELINES
Make sure the machine and setup is rigid and solid. Chatter will cause chipping.
Tool height when boring should be slightly above center. Tool deflection will put the tool on center.
Do not stop the machine with the tool in cut. This will result in tool breakage.
Use of coolant will reduce heat and improve surface finish.
Do not contact the tool to a hard surface prior to the machining process- this will cause chipping.
Higher speeds minimize tool buildup.
Depth of cut should not exceed 70% of PCD tip length.

As the DOC decreases the feed rate can increase DOC = Depth of Cut SFM = Surface Feet per Minute

## CBN TIPPED TOOL INFORMATION

SCT CBN tools and inserts are excellent for continuous cutting of a wide range of hardened steels, powdered metals, cast irons and super alloys. The products are precision ground with hones for machining to sub-micron finishes with maximum tool life. CBN tipped tools and inserts can take the place of grinding.

MATERIAL	BHN/Rc	SPEED RANGE (SFM)	FEED IPR	SINGLE POINT CBN TIPPED BARS			
				TOOL DIAMETER			
				.120-160 MAX DOC	.180-.230 MAX DOC	.290-.320 MAX DOC	.360+ MAX DOC
HEAT TREATED ALLOY	45-60Rc	200-600	.001-.005	0.003	0.004	0.006	0.009
TOOL STEEL	45-60Rc	200-600	.001-.005	0.003	0.004	0.006	0.009
NODULAR IRON	N/A	600-1500	.001-.005	0.006	0.01	0.02	0.03
PEARLITIC IRON	220-240BHN	600-2500	.001-.007	0.006	0.01	0.02	0.03
WHITE/CHILLED IRON	54-60Rc	200-500	.001-.005	0.005	0.008	0.012	0.015
SUPER ALLOY Ni BASE	240-475 BHN	200-800	.001-.005	0.003	0.004	0.006	0.025
COBOLT BASED ALLOY, STELLITE	45-55Rc	200-500	.001-.005	0.003	0.004	0.006	0.009
INCONELS	45-55Rc	200-500	.001-.005	0.003	0.004	0.006	0.009

APPLICATION GUIDELINES
Make sure the machine and setup is rigid and solid. Chatter will cause chipping
Tool height when boring should be slightly above center. Tool deflection will put the tool on center.
Do not stop the machine with the tool in cut. This will result in tool breakage.
Coolant use is not advised as it could cause thermal cracking.
Do not contact the tool to a hard surface prior to the machining process. This will cause chipping.
Depth of cut should not exceed 30% of CBN tip length.

As the DOC decreases the feed rate can increase DOC = Depth of Cut SFM = Surface Feet per Minute