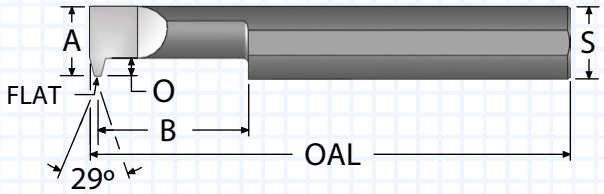
ACME THREADING TOOLS - SOLID CARBIDE



- ALTiN+ coating provides better surface finish
- Elliptically ground neck provides maximum strength
- Polished flute face for optimum performance

MIN THREAD SIZE*	"A" MIN HOLE	"B" MAX DEPTH	FLAT WIDTH	"O" OFF SET	"S" SHANK DIA.	OAL	ORDER #		EDP #	
							UNCOATED	ALTiN+	UNCOATED	ALTiN+
1/4-16	0.180	0.350	0.021	0.045	0.250	2.50	FAT180350-16	FAT180350-16A	230904	230928
1/4-16	0.180	0.500	0.021	0.045	0.250	2.50	FAT180500-16	FAT180500-16A	230907	230931
1/4-16	0.180	0.750	0.021	0.045	0.250	2.50	FAT180750-16	FAT180750-16A	230910	230934
1/4-16	0.180	1.000	0.021	0.045	0.250	2.50	FAT1801000-16	FAT1801000-16A	230901	230925
5/16-14	0.230	0.400	0.024	0.055	0.3125	2.50	FAT230400-14	FAT230400-14A	230943	231003
5/16-14	0.230	0.600	0.024	0.055	0.3125	2.50	FAT230600-14	FAT230600-14A	230946	231006
5/16-14	0.230	0.750	0.024	0.055	0.3125	2.50	FAT230750-14	FAT230750-14A	230949	231009
5/16-14	0.230	1.000	0.024	0.055	0.3125	2.50	FAT2301000-14	FAT2301000-14A	230937	230997
5/16-14	0.230	1.250	0.024	0.055	0.3125	2.50	FAT2301250-14	FAT2301250-14A	230940	231000
3/8-12	0.290	0.400	0.028	0.070	0.3125	2.50	FAT290400-12	FAT290400-12A	230958	231018
3/8-12	0.290	0.600	0.028	0.070	0.3125	2.50	FAT290600-12	FAT290600-12A	230961	231021
3/8-12	0.290	0.750	0.028	0.070	0.3125	2.50	FAT290750-12	FAT290750-12A	230964	231024
3/8-12	0.290	1.000	0.028	0.070	0.3125	2.50	FAT2901000-12	FAT2901000-12A	230952	231012
3/8-12	0.290	1.250	0.028	0.070	0.3125	2.50	FAT2901250-12	FAT2901250-12A	230955	231015
1/2-10	0.360	0.500	0.032	0.085	0.375	2.50	FAT360500-10	FAT360500-10A	231036	231066
1/2-10	0.360	0.750	0.032	0.085	0.375	2.50	FAT360750-10	FAT360750-10A	231039	231069
1/2-10	0.360	1.000	0.032	0.085	0.375	2.50	FAT3601000-10	FAT3601000-10A	231027	231057
1/2-10	0.360	1.250	0.032	0.085	0.375	2.50	FAT3601250-10	FAT3601250-10A	231030	231060
1/2-10	0.360	1.500	0.032	0.085	0.375	2.50	FAT3601500-10	FAT3601500-10A	231033	231063
5/8-8	0.490	0.750	0.041	0.120	0.500	3.00	FAT490750-8	FAT490750-8A	231096	231150
5/8-8	0.490	1.000	0.041	0.120	0.500	3.00	FAT4901000-8	FAT4901000-8A	231078	231132
5/8-8	0.490	2.000	0.041	0.120	0.500	3.00	FAT4902000-8	FAT4902000-8A	231087	231141
3/4-6	0.490	0.750	0.057	0.120	0.500	3.00	FAT490750-6	FAT490750-6A	231093	231147
3/4-6	0.490	1.000	0.057	0.120	0.500	3.00	FAT4901000-6	FAT4901000-6A	231075	231129
3/4-6	0.490	2.000	0.057	0.120	0.500	3.00	FAT4902000-6	FAT4902000-6A	231084	231138
1.0-5	0.490	0.750	0.069	0.130	0.500	3.00	FAT490750-5	FAT490750-5A	231090	231144
1.0-5	0.490	1.000	0.069	0.130	0.500	3.00	FAT4901000-5	FAT4901000-5A	231072	231126
1.0-5	0.490	2.000	0.069	0.130	0.500	3.00	FAT4902000-5	FAT4902000-5A	231081	231135

SOLID CARBIDE BORING BAR FEED AND SPEED CHART

MATERIAL	HB/Rc	SPEED (SFM)		CUTTING CONDITIONS						
				FEED IPR	TOOL DIAMETER					
		UNCOATED	ALTiN+		.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.

SFM = .262 × DIAMETER × RPM
RPM = 3.82 × SFM ÷ DIAMETER
IPM = FPT × Number of Teeth × RPM

Meters/Min = SFM × .3048
Millimeters/Rev = IPR x 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.

Scientific Cutting Tools, Inc.
www.sct-usa.com sales@sct-usa.com
(805) 584-9495 Fax: (805) 584-9629
(800) 383-2244 Fax: (888) 728-3295

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					
				TOOL DIAMETER					
		UNCOATED	ALTiN+	.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

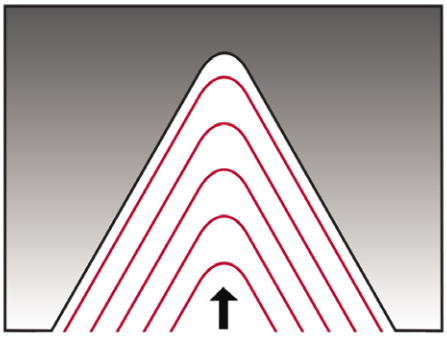
Make sure feed rate calculation does not exceed the maximum feed rate of the machine

Scientific Cutting Tools, Inc.
www.sct-usa.com sales@sct-usa.com
(805) 584-9495 Fax: (805) 584-9629
(800) 383-2244 Fax: (888) 728-3295

SINGLE POINT THREADING TROUBLESHOOTING

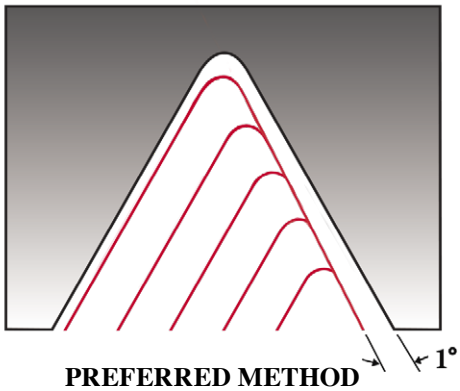
PROBLEM	CAUSE	SOLUTION
RAPID FLANK WEAR	CUTTING CONDITIONS	Check for excessive speed - see chart.
	PART	Make sure prior operation did not work harden the material.
	TOOL	Select a coated tool.
BUILT-UP EDGE	TOOL	Select a coated tool.
	CUTTING FORCE	Increase the number of passes.
	HEAT	Use the SCT coolant 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 INFEED



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.

MATERIAL	BHN/Rc	SPEED RANGE (SFM)	FEED IPR	SINGLE POINT PCD TIPPED BARS			
				TOOL DIAMETER			
				.120-160	.180-.230	.290-.320	.360+
				MAX DOC	MAX DOC	MAX DOC	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

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

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.

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	.180-.230	290-.320	.360+
				MAX DOC	MAX DOC	MAX DOC	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

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

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.

Scientific Cutting Tools, Inc.
www.sct-usa.com sales@sct-usa.com
(805) 584-9495 Fax: (805) 584-9629
(800) 383-2244 Fax: (888) 728-3295