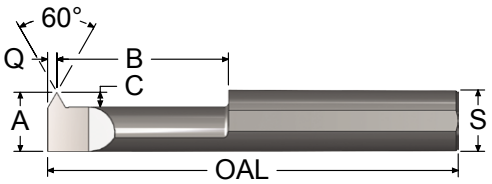
LEFT HAND THREADING TOOLS - SOLID CARBIDE



- 60° thread form for cutting UN, ISO, and NPT threads
- ALTiN+ coating extends tool life
- Precision ground shank flat guarantees tool orientation

"A" MIN BORE	"B" MAX DEPTH	"O" MIN OFFSET	RECOM-MENDED TPI*	"Q" LENGTH	"S" SHANK DIA.	OAL	ORDER #		EDP #	
							UNCOATED	ALTiN+	UNCOATED	ALTiN+
0.040	0.080	0.013	56 to 80	0.009	0.125	1.50	LHTT040080	LHTT040080A	231501	231603
0.040	0.100	0.013	56 to 80	0.009	0.125	1.50	LHTT040100	LHTT040100A	231504	231606
0.040	0.130	0.013	56 to 80	0.009	0.125	1.50	LHTT040130	LHTT040130A	231507	231609
0.050	0.100	0.017	48 to 80	0.012	0.125	1.50	LHTT050100	LHTT050100A	231510	231612
0.050	0.150	0.017	48 to 80	0.012	0.125	1.50	LHTT050150	LHTT050150A	231513	231615
0.050	0.200	0.017	48 to 80	0.012	0.125	1.50	LHTT050200	LHTT050200A	231516	231618
0.060	0.150	0.020	40 to 80	0.014	0.125	1.50	LHTT060150	LHTT060150A	231519	231621
0.060	0.200	0.020	40 to 80	0.014	0.125	1.50	LHTT060200	LHTT060200A	231522	231624
0.060	0.250	0.020	40 to 80	0.014	0.125	1.50	LHTT060250	LHTT060250A	231525	231627
0.060	0.300	0.020	40 to 80	0.014	0.125	1.50	LHTT060300	LHTT060300A	231528	231630
0.075	0.200	0.020	36 to 72	0.014	0.125	1.50	LHTT075200	LHTT075200A	231531	231633
0.075	0.300	0.020	36 to 72	0.014	0.125	1.50	LHTT075300	LHTT075300A	231534	231636
0.075	0.400	0.020	36 to 72	0.014	0.125	1.50	LHTT075400	LHTT075400A	231537	231639
0.090	0.200	0.025	32 to 64	0.017	0.125	1.50	LHTT090200	LHTT090200A	231540	231642
0.090	0.300	0.025	32 to 64	0.017	0.125	1.50	LHTT090300	LHTT090300A	231543	231645
0.090	0.400	0.025	32 to 64	0.017	0.125	1.50	LHTT090400	LHTT090400A	231546	231648
0.090	0.500	0.025	32 to 64	0.017	0.125	1.50	LHTT090500	LHTT090500A	231549	231651
0.120	0.250	0.030	24 to 56	0.021	0.1875	2.00	LHTT120250	LHTT120250A	231654	231696
0.120	0.400	0.030	24 to 56	0.021	0.1875	2.00	LHTT120400	LHTT120400A	231657	231699
0.120	0.600	0.030	24 to 56	0.021	0.1875	2.00	LHTT120600	LHTT120600A	231660	231702
0.120	0.750	0.030	24 to 56	0.021	0.1875	2.00	LHTT120750	LHTT120750A	231663	231705
0.150	0.350	0.035	20 to 56	0.023	0.1875	2.00	LHTT150350	LHTT150350A	231666	231708
0.150	0.500	0.035	20 to 56	0.023	0.1875	2.00	LHTT150500	LHTT150500A	231669	231711
0.150	0.750	0.035	20 to 56	0.023	0.1875	2.00	LHTT150750	LHTT150750A	231672	231714
0.180	0.350	0.040	18 to 56	0.027	0.250	2.50	LHTT180350	LHTT180350A	231720	231768
0.180	0.500	0.040	18 to 56	0.027	0.250	2.50	LHTT180500	LHTT180500A	231723	231771
0.180	0.750	0.040	18 to 56	0.027	0.250	2.50	LHTT180750	LHTT180750A	231726	231774
0.180	1.000	0.040	18 to 56	0.027	0.250	2.50	LHTT1801000	LHTT1801000A	231717	231765
0.200	0.400	0.045	16 to 40	0.029	0.250	2.50	LHTT200400	LHTT200400A	231732	231780
0.200	0.600	0.045	16 to 40	0.029	0.250	2.50	LHTT200600	LHTT200600A	231735	231783
0.200	0.800	0.045	16 to 40	0.029	0.250	2.50	LHTT200800	LHTT200800A	231738	231786
0.200	1.000	0.045	16 to 40	0.029	0.250	2.50	LHTT2001000	LHTT2001000A	231729	231777
0.230	0.400	0.055	14 to 40	0.038	0.3125	2.50	LHTT230400	LHTT230400A	231795	231855
0.230	0.600	0.055	14 to 40	0.038	0.3125	2.50	LHTT230600	LHTT230600A	231798	231858
0.230	0.750	0.055	14 to 40	0.038	0.3125	2.50	LHTT230750	LHTT230750A	231801	231861
0.230	1.000	0.055	14 to 40	0.038	0.3125	2.50	LHTT2301000	LHTT2301000A	231789	231849
0.230	1.250	0.055	14 to 40	0.038	0.3125	2.50	LHTT2301250	LHTT2301250A	231792	231852
0.290	0.500	0.070	12 to 40	0.047	0.3125	2.50	LHTT290500	LHTT290500A	231813	231873
0.290	0.750	0.070	12 to 40	0.047	0.3125	2.50	LHTT290750	LHTT290750A	231816	231876
0.290	1.000	0.070	12 to 40	0.047	0.3125	2.50	LHTT2901000	LHTT2901000A	231804	231864
0.290	1.250	0.070	12 to 40	0.047	0.3125	2.50	LHTT2901250	LHTT2901250A	231807	231867
0.290	1.500	0.070	12 to 40	0.047	0.3125	2.50	LHTT2901500	LHTT2901500A	231810	231870
0.320	0.500	0.075	10 to 32	0.049	0.375	2.50	LHTT320500	LHTT320500A	231888	231942
0.320	0.750	0.075	10 to 32	0.049	0.375	2.50	LHTT320750	LHTT320750A	231891	231945
0.320	1.000	0.075	10 to 32	0.049	0.375	2.50	LHTT3201000	LHTT3201000A	231879	231933
0.320	1.250	0.075	10 to 32	0.049	0.375	2.50	LHTT3201250	LHTT3201250A	231882	231936
0.320	1.500	0.075	10 to 32	0.049	0.375	2.50	LHTT3201500	LHTT3201500A	231885	231939
0.360	0.500	0.080	8 to 32	0.057	0.375	2.50	LHTT360500	LHTT360500A	231900	231954
0.360	0.750	0.080	8 to 32	0.057	0.375	2.50	LHTT360750	LHTT360750A	231903	231957
0.360	1.000	0.080	8 to 32	0.057	0.375	2.50	LHTT3601000	LHTT3601000A	231894	231948
0.360	1.250	0.080	8 to 32	0.057	0.375	2.50	LHTT3601250	LHTT3601250A	231897	231951
0.360	1.500	0.080	8 to 32	0.057	0.375	2.50	LHTT3601500	LHTT3601500A	231899	231953
0.360	1.800	0.080	8 to 32	0.057	0.375	2.50	LHTT3601800	LHTT3601800A	231960	231966
0.490	0.750	0.120	8 to 32	0.077	0.500	3.00	LHTT490750	LHTT490750A	231975	231993
0.490	1.500	0.120	8 to 32	0.077	0.500	3.00	LHTT4901500	LHTT4901500A	231969	231987
0.490	2.000	0.120	8 to 32	0.077	0.500	3.00	LHTT4902000	LHTT4902000A	231972	231990

*TPI = Threads Per Inch

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

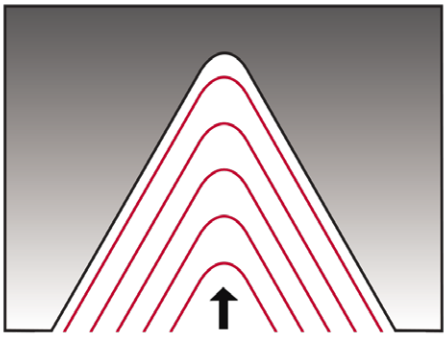
Make sure feed rate calculation does not exceed the maximum feed rate of the machine

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

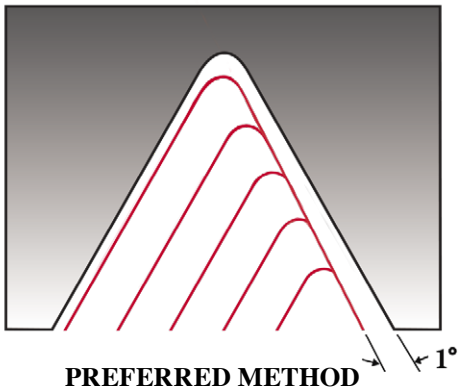
PROBLEM	CAUSE	SOLUTION
RAPID FLANK WEAR	CUTTING CONDITIONS	Check for excessive speed - see chart.
	PART	Make sure prior operation did not work harden the material.
	TOOL	Select a coated tool.
BUILT-UP EDGE	TOOL	Select a coated tool.
	CUTTING FORCE	Increase the number of passes.
	HEAT	Use the SCT coolant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