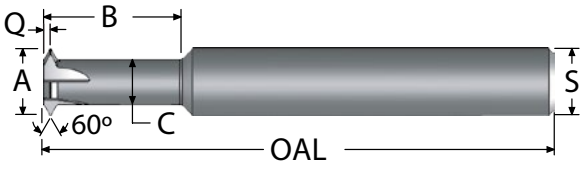
UN THREAD MILLS
SINGLE PROFILE (SPTM) - SOLID CARBIDE



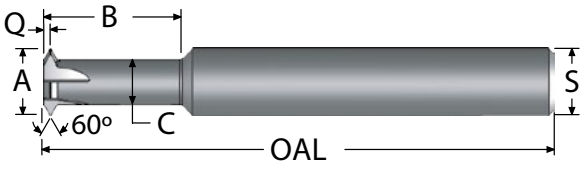
Fine and coarse threads ranging from #00 to 1¼ + can be milled using the 19 varieties of these single profile thread mills.

Min ID THREAD*	"A" TOOL DIA.	"B" LENGTH OF CUT	"C" NECK DIA.	"Q" LENGTH	"S" SHANK DIA.	OAL	RECOM-MENDED TPI	FLUTES	ORDER #	
									UNCOATED	ALTiN+
									INTERNAL OR EXTERNAL THREADS	
#00	0.032	0.060	0.018	0.005	0.125	1.50	85 to 120	2	SPTM032	SPTM032A
#00	0.032	0.100	0.018	0.005	0.125	1.50	85 to 120	2	SPTM032L	SPTM032LA
#0	0.040	0.090	0.022	0.006	0.125	1.50	72 to 90	2	SPTM040	SPTM040A
#0	0.040	0.109	0.022	0.006	0.125	1.50	72 to 90	2	SPTM040ML	SPTM040MLA
#0	0.040	0.125	0.022	0.006	0.125	1.50	72 to 90	2	SPTM040L	SPTM040LA
#1	0.050	0.100	0.028	0.007	0.125	1.50	64 to 80	3	SPTM050	SPTM050A
#1	0.050	0.125	0.028	0.007	0.125	1.50	64 to 80	3	SPTM050ML	SPTM050MLA
#1	0.050	0.150	0.028	0.007	0.125	1.50	64 to 80	3	SPTM050L	SPTM050LA
#2	0.059	0.125	0.034	0.008	0.125	1.50	56 to 80	3	SPTM059	SPTM059A
#2	0.059	0.165	0.034	0.008	0.125	1.50	56 to 80	3	SPTM059ML	SPTM059MLA
#2	0.059	0.200	0.034	0.008	0.125	1.50	56 to 80	3	SPTM059L	SPTM059LA
#2	0.060	0.125	0.034	0.009	0.1875	2.00	56 to 80	3	SPTM060	SPTM060A
#2	0.060	0.165	0.034	0.009	0.1875	2.00	56 to 80	3	SPTM060ML	SPTM060MLA
#2	0.060	0.200	0.034	0.009	0.1875	2.00	56 to 80	3	SPTM060L	SPTM060LA
#3	0.072	0.150	0.040	0.010	0.1875	2.00	48 to 72	3	SPTM072	SPTM072A
#3	0.072	0.250	0.040	0.010	0.1875	2.00	48 to 72	3	SPTM072L	SPTM072LA
#4	0.080	0.190	0.045	0.011	0.1875	2.00	40 to 64	3	SPTM080	SPTM080A
#4	0.080	0.250	0.045	0.011	0.1875	2.00	40 to 64	3	SPTM080ML	SPTM080MLA
#4	0.080	0.300	0.045	0.011	0.1875	2.00	40 to 64	3	SPTM080L	SPTM080LA
#6	0.098	0.250	0.049	0.015	0.1875	2.00	32 to 64	3	SPTM098	SPTM098A
#6	0.098	0.330	0.049	0.015	0.1875	2.00	32 to 64	3	SPTM098ML	SPTM098MLA
#6	0.098	0.400	0.049	0.015	0.1875	2.00	32 to 64	3	SPTM098L	SPTM098LA
#8	0.120	0.300	0.070	0.016	0.1875	2.00	32 to 56	3	SPTM120	SPTM120A
#8	0.120	0.400	0.070	0.016	0.1875	2.00	32 to 56	3	SPTM120ML	SPTM120MLA
#8	0.120	0.500	0.070	0.016	0.1875	2.00	32 to 56	3	SPTM120L	SPTM120LA
#10	0.138	0.400	0.075	0.020	0.1875	2.00	24 to 56	3	SPTM138	SPTM138A
#10	0.138	0.500	0.075	0.020	0.1875	2.00	24 to 56	3	SPTM138ML	SPTM138MLA
#10	0.138	0.600	0.075	0.020	0.1875	2.00	24 to 56	3	SPTM138L	SPTM138LA
#12	0.160	0.400	0.080	0.025	0.1875	2.00	24 to 56	3	SPTM160	SPTM160A
#12	0.160	0.650	0.080	0.025	0.1875	2.00	24 to 56	3	SPTM160L	SPTM160LA

TPI = Threads Per Inch

***Single profile thread mills can cut any larger size internal thread within the recommended TPI**

UN THREAD MILLS
SINGLE PROFILE (SPTM) - SOLID CARBIDE



- Solid carbide provides maximum tool rigidity
- Long reach tools are available from stock
- Cuts UNC, UNF, UNEF, and UNS threads
- Cuts UNJ threads (internal only)

Min ID THREAD*	"A" TOOL DIA.	"B" LENGTH OF CUT	"C" NECK DIA.	"Q" LENGTH	"S" SHANK DIA.	OAL	RECOM-MENDED TPI	FLUTES	ORDER #	
									UNCOATED	ALTiN+
									INTERNAL OR EXTERNAL THREADS	
1/4	0.182	0.400	0.104	0.025	0.250	2.50	18 to 56	4	SPTM182	SPTM182A
1/4	0.182	0.530	0.104	0.025	0.250	2.50	18 to 56	4	SPTM182ML	SPTM182MLA
1/4	0.182	0.650	0.104	0.025	0.250	2.50	18 to 56	4	SPTM182L	SPTM182LA
5/16	0.240	0.500	0.153	0.028	0.250	2.50	16 to 48	4	SPTM240	SPTM240A
5/16	0.240	0.800	0.153	0.028	0.250	2.50	16 to 48	4	SPTM240L	SPTM240LA
3/8	0.290	0.600	0.192	0.031	0.375	3.00	14 to 40	4	SPTM290	SPTM290A
3/8	0.290	1.000	0.192	0.031	0.375	3.00	14 to 40	4	SPTM290L	SPTM290LA
1/2	0.372	0.750	0.240	0.041	0.375	3.00	12 to 32	4	SPTM372	SPTM372A
1/2	0.372	1.200	0.240	0.041	0.375	3.00	12 to 32	4	SPTM372L	SPTM372LA
5/8	0.488	0.850	0.340	0.046	0.500	3.50	11 to 32	5	SPTM488	SPTM488A
5/8	0.488	1.350	0.340	0.046	0.500	3.50	11 to 32	5	SPTM488L	SPTM488LA
3/4	0.595	1.250	0.430	0.051	0.625	4.00	10 to 32	6	SPTM595	SPTM595A
3/4	0.595	2.000	0.430	0.051	0.625	4.00	10 to 32	6	SPTM595L	SPTM595LA
7/8	0.695	1.500	0.490	0.063	0.750	5.00	8 to 24	6	SPTM695	SPTM695A
7/8	0.695	2.500	0.490	0.063	0.750	5.00	8 to 24	6	SPTM695L	SPTM695LA
1¼	0.745	1.500	0.400	0.107	0.750	5.00	4 to 8	6	SPTM745	SPTM745A
1¼	0.745	2.500	0.400	0.107	0.750	5.00	4 to 8	6	SPTM745L	SPTM745LA

TPI = Threads Per Inch

***Single profile thread mills can cut any larger size internal thread within the recommended TPI**

THREAD MILL FEED AND SPEED CHART

MATERIAL	HB/Rc	SPEED SFM* UNCOATED	SPEED SFM ALTiN+	FEED (INCHES PER TOOTH)					
				TOOL DIAMETER					
				.032 - .056	.059 - .090	.100 - .190	.200 - .350	.370 - .595	.600+
CAST IRON	160 HB	100-220	200-425	.0004-.001	.0004-.0008	.0004-.0014	.0004-.002	.0004-.0035	.0004-.006
CARBON STEEL	18 Rc	100-200	190-425	.0003-.001	.0003-.0008	.0003-.0014	.0003-.002	.0003-.005	.0003-.006
ALLOY STEEL	20 Rc	80-200	200-375	.0003-.001 2 Passes	.0003-.0008 3 Passes	.0003-.0014	.0003-.0024	.0003-.005	.0003-.006
TOOL STEEL	20 Rc	80-175	175-250	.0003-.0004 2 Passes	.0003-0.0005 3 Passes	.0003-.0005	.0003-.0009	.0003-.0026	.0003-.004
300 STAINLESS STEEL	150 HB	90-120	120-255	.0003-.0005 2 Passes	.0003-0.0006 3 Passes	.0003-.0007	.0003-.002	.0003-.0035	.0003-.0045
400 STAINLESS STEEL	195 HB	90-150	140-375	.0003-.0005 2 Passes	.0003-.0006 3 Passes	.0003-.0007	.0003-.002	.0003-.0026	.0003-.0045
HIGH TEMP ALLOY (Ni & Co BASE)	20 Rc	50-125	100-125	.0003-.0004 3 Passes	.0003-.00045 3 Passes	.0003-.0005 2 Passes	.0003-.0009	.0003-.0026	.0003-.004
TITANIUM	25 Rc	50-130	100-170	.0003-.0004 3 Passes	.0003-.00045 3 Passes	.0003-.001 2 Passes	.0003-.0009	.0003-.0015	.0003-.003
HEAT TREATED ALLOYS (38-45Rc)	40 Rc	50-90	90-150	.0003-.0004 3 Passes	.0003-.00045 3 Passes	.0003-.0005 2 Passes	.0003-.0008	.0003-.001	.0003-.0025
ALUMINUM	100 HB	100-800	100-1200	.0005-.0015	.0005-.002	.0005-.0025	.0005-.003	.0005-.006	.0005-.009
BRASS, ZINC	80 HB	200-350	200-750	.0005-.0015	.0005-.002	.0005-.0025	.0005-.003	.0005-.006	.0005-.009

*SFM = Surface Feet per Minute

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

THREAD MILL FEED AND SPEED APPLICATION

It may be necessary to use more radial depth passes than shown on the chart (p.40) when cutting an unfavorable length-to-diameter ratio, coarse pitches, or hard materials. When cutting a thread with two passes, cut approximately **65% of the thread on the first pass and 35 percent on the finish pass.** For three passes, use a **50/30/20** ratio. For four passes, use a **40/27/20/13** ratio. The idea is to equalize the side cutting pressure.

Thread mills can sometimes be used to cut multiple start threads. Call engineering for assistance.
Thread mills can be cut off for shorter thread depths or necked back for deeper thread depths. Call for price and delivery.

In order to apply the Feed and Speed chart appropriately, it is necessary to understand that machining centers will apply the feed rate at the centerline of the spindle. It is correct to use a normal calculation and the following Feed & Speed Chart when cutting in a straight line; however, it is incorrect when cutting an internal thread. Therefore, the feed rate must be recalculated.

The following is an example of how to apply the feed rate correctly:
The tool is a TM290-24A cutting a 3/8-24 thread in stainless steel.
The outside diameter of the tool is 0.290.
The surface foot per minute (SFM) is 150.
The chip per tooth ia 0.001. The tool has four flutes.
The revolutions per minute (RPM) equal the SFM x 3.82 divided by the outside diameter of the tool.

In this example: **(150 x 3.82) / 0.290**, which equals 1975 RPM.
The RPM x feed (chip per tooth) x the number of flutes equals the Non-Adjusted Feed Rate or NAFR.
In this example: **1975 x 0.001 x 4 = 7.9 NAFR**
The major diameter of the thread is 0.375. We will call this D.
The outside diameter of the tool is 0.290. We will call this d.
We will call the Adjusted Feed Rate the AFR.
The formula for the AFR for internal interpolation is **AFR = NAFR x (D-d) ÷ D**

In this example: **AFR = 7.9 x (0.375 - 0.290) ÷ 0.375**

Therefore, the Adjusted Feed Rate equals 1.79. This is the feed rate that will equal 0.001 chip per tooth in the above example. This is the feed rate that must be used in the CNC program.

THREAD MILL TROUBLESHOOTING

PROBLEM	CAUSE	SOLUTION
TAPERED THREADED HOLE	TOOL PRESSURE	Reduce the chip load and/or make more radial passes.
NO-GO GAGE GOES & GO GAGE DOES NOT GO	THREAD OVERCUTTING	Use a tool of smaller diameter with correct pitch. Make sure helical "ramp in" is used.
TEETH ARE CHIPPING	TOOL PRESSURE	Reduce feed rate per tooth.
	BUILT-UP EDGE	Use a coated tool to help reduce built-up edge.
RAPID WEAR	TOOL RUBBING NOT CUTTING	Increase chip load per tooth.
TEETH ARE BURNING	TOO MUCH HEAT	Reduce speed. Use a coated tool. Increase coolant.
TOOL BREAKS	TOO MUCH TOOL PRESSURE	Helical "arc in" must be used. Reduce feed rate and/or use more radial passes. Adjusted Feed Rate (AFR) must be used. (See Thread Mill Feed and Speed Chart)

Thread milling tools form a thread using a motion referred to as “helical interpolation.” This process involves the machine simultaneously moving all three axes. The resulting motions are circular and axial. The “X” and “Y” axes move in a circular manner and the “Z” axis in an axial direction per 360° at a distance equal to the pitch of the thread being machined. The tool should "ramp in" over 90° in order to avoid breakage. This must be a helical move. Move "Z" axially by pitch ÷ 4 since 90° is 360° ÷ 4.

Bottom-to-top climb cutting machining is recommended when machining a right-hand thread. This will avoid re-cutting any chips. For left hand threading, a top-to-bottom machining with a right-hand helical tool is the preferred method. Refer to troubleshooting chart above for solutions to potential thread milling problems.