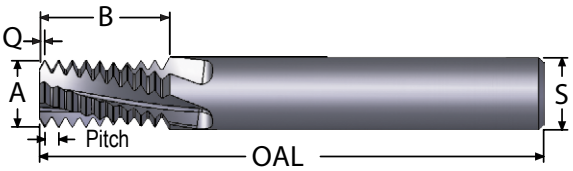
UN THREAD MILLS
15° HELICAL FLUTE SOLID CARBIDE



- Cuts UNC, UNF, UNEF, UNS and UNJ (internal only)
- Non-crest cutting allows maximum flexibility for plated and non-standard threads
- Long length-of-cut

MIN ID THREAD / PITCH*	"A" TOOL DIA.	"B" LENGTH OF CUT	"Q" LENGTH	"S" SHANK DIA.	OAL	FLUTES	ORDER #		EDP #	
							UNCOATED	ALTiN+	UNCOATED	ALTiN+
							INTERNAL THREADS ONLY			
4-40	0.079	0.185	0.011	0.250	2.50	2	TMI079-40H	TMI079-40HA	102901	102937
6-32	0.100	0.263	0.014	0.250	2.50	3	TMI100-32H	TMI100-32HA	102904	102940
8-32	0.115	0.263	0.014	0.250	2.50	3	TMI115-32H	TMI115-32HA	102907	102943
10-24	0.120	0.351	0.019	0.250	2.50	3	TMI120-24H	TMI120-24HA	102910	102946
10-28	0.120	0.336	0.016	0.250	2.50	3	TMI120-28H	TMI120-28HA	102913	102949
10-32	0.120	0.326	0.014	0.250	2.50	3	TMI120-32H	TMI120-32HA	102916	102952
1/4-20	0.180	0.521	0.023	0.250	2.50	3	TMI180-20H	TMI180-20HA	102919	102955
1/4-28	0.180	0.515	0.016	0.250	2.50	3	TMI180-28H	TMI180-28HA	102922	102958
5/16-18	0.234	0.632	0.025	0.250	2.50	3	TMI234-18H	TMI234-18HA	102925	102961
5/16-24	0.234	0.641	0.019	0.250	2.50	3	TMI234-24H	TMI234-24HA	102928	102964
5/16-32	0.234	0.638	0.014	0.250	2.50	3	TMI234-32H	TMI234-32HA	102931	102967
5/16-40	0.234	0.635	0.011	0.250	2.50	3	TMI234-40H	TMI234-40HA	102934	102970
3/8-16	0.285	0.775	0.028	0.3125	3.00	4	TMI285-16H	TMI285-16HA	102973	102988
3/8-20	0.285	0.770	0.023	0.3125	3.00	4	TMI285-20H	TMI285-20HA	102976	102991
3/8-24	0.285	0.766	0.019	0.3125	3.00	4	TMI285-24H	TMI285-24HA	102979	102994
3/8-32	0.285	0.763	0.014	0.3125	3.00	4	TMI285-32H	TMI285-32HA	102982	102997
7/16-14	0.305	0.886	0.032	0.3125	3.00	4	TMI305-14H	TMI305-14HA	102985	103000
7/16-18	0.335	0.888	0.025	0.375	3.00	4	TMI335-18H	TMI335-18HA	103003	103021
7/16-20	0.335	0.870	0.023	0.375	3.00	4	TMI335-20H	TMI335-20HA	103006	103024
1/2-13	0.350	0.877	0.035	0.375	3.00	4	TMI350-13H	TMI350-13HA	103009	103027
9/16-12	0.370	0.867	0.038	0.375	3.00	4	TMI370-12H	TMI370-12HA	103012	103030
9/16-18	0.370	0.911	0.025	0.375	3.00	4	TMI370-18H	TMI370-18HA	103015	103033
9/16-32	0.370	0.888	0.014	0.375	3.00	4	TMI370-32H	TMI370-32HA	103018	103036
5/8-11	0.470	1.309	0.041	0.500	4.00	4	TMI470-11H	TMI470-11HA	103039	103063
3/4-10	0.495	1.340	0.045	0.500	4.00	4	TMI495-10H	TMI495-10HA	103042	103066
3/4-12	0.495	1.283	0.038	0.500	4.00	4	TMI495-12H	TMI495-12HA	103045	103069
3/4-14	0.495	1.314	0.032	0.500	4.00	4	TMI495-14H	TMI495-14HA	103048	103072
3/4-16	0.495	1.338	0.028	0.500	4.00	4	TMI495-16H	TMI495-16HA	103051	103075
3/4-18	0.495	1.300	0.025	0.500	4.00	4	TMI495-18H	TMI495-18HA	103054	103078
3/4-20	0.495	1.320	0.023	0.500	4.00	4	TMI495-20H	TMI495-20HA	103057	103081
3/4-32	0.495	1.325	0.014	0.500	4.00	4	TMI495-32H	TMI495-32HA	103060	103084
7/8-9	0.620	1.489	0.049	0.625	4.00	5	TMI620-9H	TMI620-9HA	103099	103114
1.0-8	0.620	1.550	0.056	0.625	4.00	5	TMI620-8H	TMI620-8HA	103096	103111
1.0-12	0.620	1.534	0.038	0.625	4.00	5	TMI620-12H	TMI620-12HA	103087	103102
1.0-14	0.620	1.529	0.032	0.625	4.00	5	TMI620-14H	TMI620-14HA	103090	103105
1.0-16	0.620	1.525	0.028	0.625	4.00	5	TMI620-16H	TMI620-16HA	103093	103108

***Thread mills can cut any larger size internal thread of the same pitch**

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 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.

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