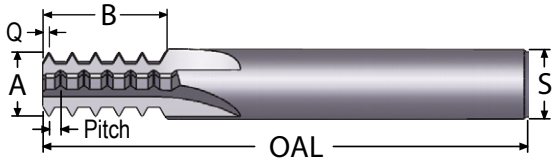
UN THREAD MILLS - STAGGERED TOOTH

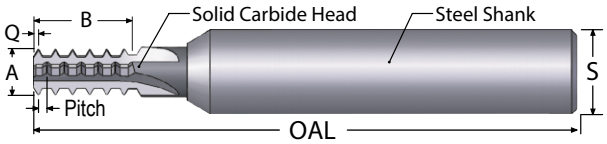


- Staggered tooth design reduces tool pressure
- Non-crest cutting for max thread size adjustability

STRAIGHT FLUTE - STAGGERED TOOTH - SOLID CARBIDE

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 OR EXTERNAL THREADS			
3/8-20	0.250	0.675	0.027	0.250	2.50	4	TM250-20	TM250-20A	102651	102687
3/8-24	0.250	0.687	0.024	0.250	2.50	4	TM250-24	TM250-24A	102654	102690
3/8-28	0.250	0.661	0.020	0.250	2.50	4	TM250-28	TM250-28A	102657	102693
3/8-32	0.250	0.672	0.017	0.250	2.50	4	TM250-32	TM250-32A	102660	102696
3/8-36	0.250	0.682	0.016	0.250	2.50	4	TM250-36	TM250-36A	102663	102699
3/8-40	0.250	0.662	0.014	0.250	2.50	4	TM250-40	TM250-40A	102666	102702

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

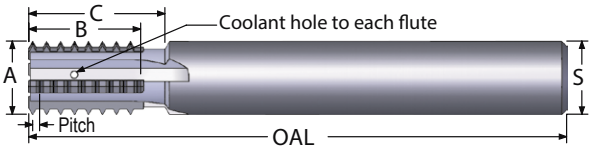


- Cuts UNC, UNF, UNEF, and UNS threads
- Cuts UNJ threads (internal only)
- Non-crest cutting design cuts internal and external threads

STRAIGHT FLUTE - STAGGERED TOOTH - CARBIDE HEAD

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 OR EXTERNAL THREADS			
7/16-16	0.350	0.783	0.036	0.750	6.00	4	TM350-16	TM350-16A	102705	102747
7/16-18	0.350	0.807	0.032	0.750	6.00	4	TM350-18	TM350-18A	102708	102750
7/16-20	0.350	0.823	0.027	0.750	6.00	4	TM350-20	TM350-20A	102711	102753
7/16-24	0.350	0.856	0.024	0.750	6.00	4	TM350-24	TM350-24A	102714	102756
5/8-12	0.500	1.042	0.046	0.750	6.00	4	TM500-12	TM500-12A	102717	102759
5/8-14	0.500	1.037	0.040	0.750	6.00	4	TM500-14	TM500-14A	102720	102762
5/8-16	0.500	1.033	0.036	0.750	6.00	4	TM500-16	TM500-16A	102723	102765

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



- ALTiN+ coating extends tool life
- Ideal for plated thread applications

COOLANT THROUGH THREAD MILLS
STRAIGHT FLUTE - STAGGERED TOOTH - CARBIDE TIPPED

MIN ID THREAD / PITCH*	"A" TOOL DIA.	"B" LENGTH OF CUT	"C" TOOL REACH	"Q" LENGTH	"S" SHANK DIA.	OAL	FLUTES	ORDER #		EDP #	
								UNCOATED	ALTiN+	UNCOATED	ALTiN+
								INTERNAL OR EXTERNAL THREADS			
1-12	0.750	1.176	1.370	0.048	0.750	6.00	4	TMC750-12	TMC750-12A	102801	102825
1-14	0.750	1.152	1.370	0.042	0.750	6.00	4	TMC750-14	TMC750-14A	102804	102828
1-18	0.750	1.117	1.370	0.032	0.750	6.00	4	TMC750-18	TMC750-18A	102807	102831
1-20	0.750	1.106	1.370	0.029	0.750	6.00	4	TMC750-20	TMC750-20A	102810	102834
1½-12	1.000	1.176	2.000	0.047	1.000	6.00	6	TMC1000-12	TMC1000-12A	102837	102849
1½-16	1.000	1.196	2.000	0.037	1.000	6.00	6	TMC1000-16	TMC1000-16A	102840	102852

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

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

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