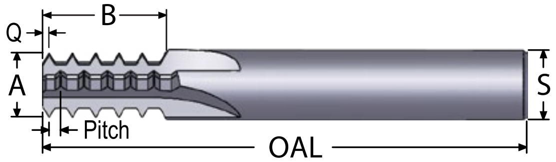
UN THREAD MILLS - STAGGERED TOOTH

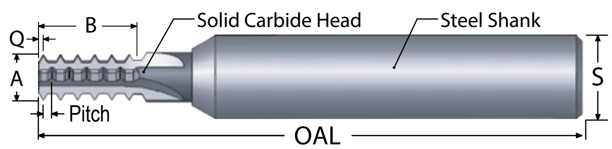


- ➢ Staggered tooth design reduces tool pressure
- ➢ Non-crest cutting for max thread size adjustability

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 # UNCOATED	ORDER # ALTiN+
							INTERNAL OR EXTERNAL THREADS	
3/8-20	0.250	0.675	0.027	0.250	2.50	4	TM250-20	TM250-20A
3/8-24	0.250	0.687	0.024	0.250	2.50	4	TM250-24	TM250-24A
3/8-28	0.250	0.661	0.020	0.250	2.50	4	TM250-28	TM250-28A
3/8-32	0.250	0.672	0.017	0.250	2.50	4	TM250-32	TM250-32A
3/8-36	0.250	0.682	0.016	0.250	2.50	4	TM250-36	TM250-36A
3/8-40	0.250	0.662	0.014	0.250	2.50	4	TM250-40	TM250-40A

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

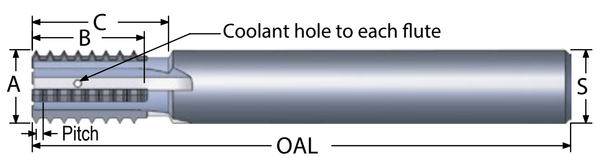


- ➢ Cuts UNC, UNF, UNEF, and UNS threads
- ➢ Cuts UNJ threads (internal only)
- ➢ Non-crest cutting design cuts internal and external threads

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 # UNCOATED	ORDER # ALTiN+
							INTERNAL OR EXTERNAL THREADS	
7/16-16	0.350	0.783	0.036	0.750	6.00	4	TM350-16	TM350-16A
7/16-18	0.350	0.807	0.032	0.750	6.00	4	TM350-18	TM350-18A
7/16-20	0.350	0.823	0.027	0.750	6.00	4	TM350-20	TM350-20A
7/16-24	0.350	0.856	0.024	0.750	6.00	4	TM350-24	TM350-24A
5/8-12	0.500	1.042	0.046	0.750	6.00	4	TM500-12	TM500-12A
5/8-14	0.500	1.037	0.040	0.750	6.00	4	TM500-14	TM500-14A
5/8-16	0.500	1.033	0.036	0.750	6.00	4	TM500-16	TM500-16A

***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 # UNCOATED	ORDER # ALTiN+
								INTERNAL OR EXTERNAL THREADS	
1-12	0.750	1.176	1.370	0.048	0.750	6.00	4	TMC750-12	TMC750-12A
1-14	0.750	1.152	1.370	0.042	0.750	6.00	4	TMC750-14	TMC750-14A
1-18	0.750	1.117	1.370	0.032	0.750	6.00	4	TMC750-18	TMC750-18A
1-20	0.750	1.106	1.370	0.029	0.750	6.00	4	TMC750-20	TMC750-20A
1½-12	1.000	1.176	2.000	0.047	1.000	6.00	6	TMC1000-12	TMC1000-12A
1½-16	1.000	1.196	2.000	0.037	1.000	6.00	6	TMC1000-16	TMC1000-16A

***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 (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.