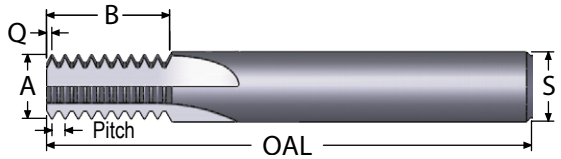
THREAD MILLS - METRIC
STRAIGHT FLUTE - CARBIDE
FULL PROFILE



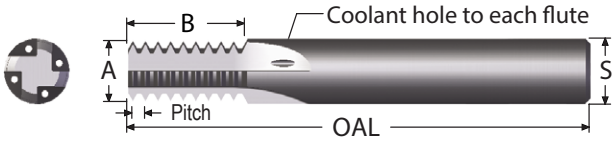
- Short length-of-cut for ideal length-to-diameter ratio
- Polished flute face for optimum performance
- Made with premium submicron grade carbide
- Internal crest cutting design for strongest possible tool

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			
M3-.5	0.090	0.264	0.009	0.250	2.50	3	TM3-.5MM	TM3-.5MM-A	100104	100188
M3-.5	0.090	0.185	0.009	0.250	2.50	3	TM3-.5MM-S	TM3-.5MM-SA	100554	100620
M3.5-.6	0.090	0.269	0.011	0.250	2.50	3	TM3.5-.6MM	TM3.5-.6MM-A	100101	100185
M3.5-.6	0.090	0.175	0.011	0.250	2.50	3	TM3.5-.6MM-S	TM3.5-.6MM-SA	100551	100617
M4-.5	0.110	0.323	0.009	0.250	2.50	3	TM4-.5MM	TM4-.5MM-A	100110	100194
M4-.5	0.110	0.224	0.009	0.250	2.50	3	TM4-.5MM-S	TM4-.5MM-SA	100560	100626
M4-.7	0.110	0.342	0.012	0.250	2.50	3	TM4-.7MM	TM4-.7MM-A	100113	100197
M4-.7	0.110	0.231	0.012	0.250	2.50	3	TM4-.7MM-S	TM4-.7MM-SA	100563	100629
M4.5-.75	0.125	0.337	0.013	0.250	2.50	3	TM4.5-.75MM	TM4.5-.75MM-A	100107	100191
M4.5-.75	0.125	0.219	0.013	0.250	2.50	3	TM4.5-.75MM-S	TM4.5-.75MM-SA	100557	100623
M5-.7	0.140	0.397	0.012	0.250	2.50	3	TM5-.7MM	TM5-.7MM-A	100116	100200
M5-.7	0.140	0.259	0.012	0.250	2.50	3	TM5-.7MM-S	TM5-.7MM-SA	100566	100632
M5-.8	0.140	0.391	0.014	0.250	2.50	3	TM5-.8MM	TM5-.8MM-A	100119	100203
M5-.8	0.140	0.265	0.014	0.250	2.50	3	TM5-.8MM-S	TM5-.8MM-SA	100569	100635
M6-.5	0.170	0.520	0.009	0.250	2.50	3	TM6-.5MM	TM6-.5MM-A	100122	100206
M6-.5	0.170	0.382	0.009	0.250	2.50	3	TM6-.5MM-S	TM6-.5MM-SA	100572	100638
M6-.75	0.170	0.543	0.013	0.250	2.50	3	TM6-.75MM	TM6-.75MM-A	100125	100209
M6-.75	0.170	0.366	0.013	0.250	2.50	3	TM6-.75MM-S	TM6-.75MM-SA	100575	100641
M6-1	0.170	0.528	0.018	0.250	2.50	3	TM6-1MM	TM6-1MM-A	100131	100215
M6-1	0.170	0.370	0.018	0.250	2.50	3	TM6-1MM-S	TM6-1MM-SA	100581	100647
M6-1.25	0.170	0.561	0.022	0.250	2.50	3	TM6-1.25MM	TM6-1.25MM-A	100128	100212
M6-1.25	0.170	0.364	0.022	0.250	2.50	3	TM6-1.25MM-S	TM6-1.25MM-SA	100578	100644
M8-.75	0.235	0.662	0.013	0.250	2.50	3	TM8-.75MM	TM8-.75MM-A	100134	100218
M8-1	0.235	0.685	0.018	0.250	2.50	3	TM8-1MM	TM8-1MM-A	100140	100224
M8-1.25	0.235	0.660	0.022	0.250	2.50	3	TM8-1.25MM	TM8-1.25MM-A	100137	100221
M10-1	0.290	0.803	0.018	0.3125	3.50	4	TM10-1MM	TM10-1MM-A	100230	100242
M10-1.5	0.290	0.792	0.027	0.3125	3.50	4	TM10-1.5MM	TM10-1.5MM-A	100227	100239
M12-1.25	0.345	0.807	0.022	0.375	3.50	4	TM12-1.25MM	TM12-1.25MM-A	100245	100263
M12-1.5	0.345	0.792	0.027	0.375	3.50	4	TM12-1.5MM	TM12-1.5MM-A	100248	100266
M12-1.75	0.345	0.787	0.031	0.375	3.50	4	TM12-1.75MM	TM12-1.75MM-A	100251	100269
M12-1	0.400	1.079	0.018	0.500	3.50	4	TM12-1MM	TM12-1MM-A	100272	100308
M14-1.25	0.450	1.103	0.022	0.500	3.50	4	TM14-1.25MM	TM14-1.25MM-A	100275	100311
M14-1.5	0.450	1.087	0.027	0.500	3.50	4	TM14-1.5MM	TM14-1.5MM-A	100278	100314
M14-1.75	0.450	1.134	0.031	0.500	3.50	4	TM14-1.75MM	TM14-1.75MM-A	100281	100317
M14-2	0.450	1.134	0.035	0.500	3.50	4	TM14-2MM	TM14-2MM-A	100284	100320
M16-2.5	0.450	1.122	0.044	0.500	3.50	4	TM16-2.5MM	TM16-2.5MM-A	100287	100323

***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

METRIC THREAD MILLS
COOLANT THROUGH - SOLID CARBIDE - FULL PROFILE

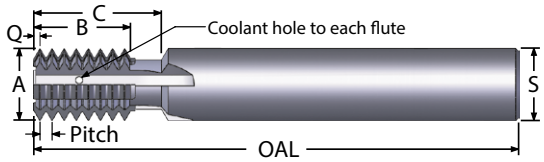


- ALTiN+ coating for higher cutting speed
- Coolant to each flute
- Made with premium submicron grade carbide
- Internal Threads Only

MIN IN THREAD/ PITCH*	"A" TOOL DIA.	"B" LENGTH OF CUT	"Q" LENGTH	"S" SHANK DIA.	OAL	FLUTE	ORDER #		EDP #	
							UNCOATED	AlTiN	UNCOATED	AlTiN
							INTERNAL THREADS ONLY			
M3-.5	0.090	0.264	0.009	0.250	2.50	3	TMC3-.5MM	TMC3-.5MM-A	100401	100428
M4-.5	0.110	0.323	0.009	0.250	2.50	3	TMC4-.5MM	TMC4-.5MM-A	100407	100434
M4-.7	0.110	0.342	0.012	0.250	2.50	3	TMC4-.7MM	TMC4-.7MM-A	100410	100437
M4.5-.75	0.125	0.337	0.013	0.250	2.50	3	TMC4.5-.75MM	TMC4.5-.75MM-A	100404	100431
M5-.8	0.140	0.391	0.014	0.250	2.50	3	TMC5-.8MM	TMC5-.8MM-A	100413	100440
M6-.5	0.170	0.520	0.009	0.250	2.50	3	TMC6-.5MM	TMC6-.5MM-A	100416	100443
M6-1	0.170	0.528	0.018	0.250	2.50	3	TMC6-1MM	TMC6-1MM-A	100419	100446
M8-1	0.235	0.685	0.018	0.250	2.50	3	TMC8-1MM	TMC8-1MM-A	100425	100452
M8-1.25	0.235	0.660	0.022	0.250	2.50	3	TMC8-1.25MM	TMC8-1.25MM-A	100422	100449
M10-1	0.290	0.803	0.018	0.3125	3.50	4	TMC10-1MM	TMC10-1MM-A	100458	100464
M10-1.5	0.290	0.792	0.027	0.3125	3.50	4	TMC10-1.5MM	TMC10-1.5MM-A	100455	100461
M12-1.25	0.345	0.807	0.022	0.375	3.50	4	TMC12-1.25MM	TMC12-1.25MM-A	100467	100470
M14-1.5	0.450	1.087	0.027	0.500	3.50	4	TMC14-1.5MM	TMC14-1.5MM-A	100473	100479
M14-2	0.450	1.134	0.035	0.500	3.50	4	TMC14-2MM	TMC14-2MM-A	100476	100482

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

METRIC THREAD MILL
COOLANT THROUGH - CARBIDE TIPPED



- Non-crest cutting on the internal thread allows maximum flexibility for plated and non-standard threads

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			
M24-1.5	0.740	1.058	1.370	0.027	0.750	6.00	4	TMC24-1.5MM	TMC24-1.5MM-A	100485	100515
M24-2	0.740	1.100	1.370	0.036	0.750	6.00	4	TMC24-2MM	TMC24-2MM-A	100494	100524
M24-2.5	0.740	1.076	1.370	0.045	0.750	6.00	4	TMC24-2.5MM	TMC24-2.5MM-A	100491	100521
M24-3	0.740	1.058	1.370	0.054	0.750	6.00	4	TMC24-3MM	TMC24-3MM-A	100497	100527
M36-4	0.990	1.095	2.000	0.071	1.000	6.00	6	TMC36-4MM	TMC36-4MM-A	100530	100536

***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