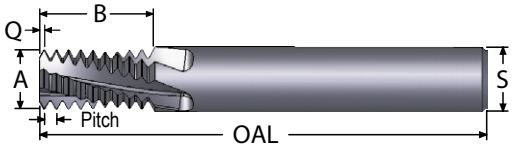
THREAD MILL - METRIC - 15° HELICAL FLUTE - CARBIDE

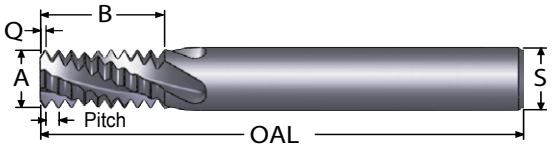


- Helical flute for reduced side cutting pressure
- ALTiN+ coating extends tool life

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			
M5.0-.8	0.118	0.328	0.014	6mm	58mm	3	TMI5.0-0.80-H	TMI5.0-0.80-HA	100704	100722
M6.0-1	0.169	0.488	0.018	6mm	58mm	3	TMI6.0-1.00-H	TMI6.0-1.00-HA	100707	100725
M8.0-.75	0.234	0.632	0.013	6mm	58mm	3	TMI8.0-0.75-H	TMI8.0-0.75-HA	100710	100728
M8.0-1	0.234	0.646	0.018	6mm	58mm	3	TMI8.0-1.00-H	TMI8.0-1.00-HA	100713	100731
M8.0-1.25	0.234	0.659	0.022	6mm	58mm	3	TMI8.0-1.25-H	TMI8.0-1.25-HA	100716	100734
M10-1.5	0.300	0.790	0.027	8mm	75mm	4	TMI10-1.50-H	TMI10-1.50-HA	103153	103156
M12-1	0.360	0.881	0.018	10mm	100mm	4	TMI12-1.00-H	TMI12-1.00-HA	103117	103126
M12-1.75	0.360	0.923	0.031	10mm	100mm	4	TMI12-1.75-H	TMI12-1.75-HA	103120	103129
M14-1.5	0.370	0.909	0.027	10mm	100mm	4	TMI14-1.50-H	TMI14-1.50-HA	103123	103132
M16-2	0.470	1.290	0.035	12mm	100mm	4	TMI16-2.00-H	TMI16-2.00-HA	103135	103144
M18-1.5	0.470	1.263	0.027	12mm	100mm	4	TMI18-1.50-H	TMI18-1.50-HA	103138	103147
M20-2.5	0.470	1.318	0.044	12mm	100mm	4	TMI20-2.50-H	TMI20-2.50-HA	103141	103150

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

THREAD MILL - METRIC - 30° HELICAL FLUTE - CARBIDE



- Optional short length-of-cut for ideal length-to-diameter ratio
- Internal and external threads

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			
M6-.5	0.170	0.520	0.009	0.250	2.50	3	TM6-.5MM-H	TM6-.5MM-HA	100751	100817
M6-.5	0.170	0.382	0.009	0.250	2.50	3	TM6-.5MM-SH	TM6-.5MM-SHA	100754	100820
M6-.75	0.170	0.543	0.013	0.250	2.50	3	TM6-.75MM-H	TM6-.75MM-HA	100757	100823
M6-.75	0.170	0.366	0.013	0.250	2.50	3	TM6-.75MM-SH	TM6-.75MM-SHA	100760	100826
M6-1	0.170	0.528	0.018	0.250	2.50	3	TM6-1MM-H	TM6-1MM-HA	100769	100835
M6-1	0.170	0.370	0.018	0.250	2.50	3	TM6-1MM-SH	TM6-1MM-SHA	100772	100838
M6-1.25	0.170	0.561	0.022	0.250	2.50	3	TM6-1.25MM-H	TM6-1.25MM-HA	100763	100829
M6-1.25	0.170	0.364	0.022	0.250	2.50	3	TM6-1.25MM-SH	TM6-1.25MM-SHA	100766	100832
M8-.75	0.235	0.662	0.013	0.250	2.50	3	TM8-.75MM-H	TM8-.75MM-HA	100775	100841
M8-1	0.235	0.685	0.018	0.250	2.50	3	TM8-1MM-H	TM8-1MM-HA	100781	100847
M8-1.25	0.235	0.660	0.022	0.250	2.50	3	TM8-1.25MM-H	TM8-1.25MM-HA	100778	100844
M10-1	0.290	0.803	0.018	0.3125	3.50	4	TM10-1MM-H	TM10-1MM-HA	100853	100865
M10-1.5	0.290	0.792	0.027	0.3125	3.50	4	TM10-1.5MM-H	TM10-1.5MM-HA	100850	100862
M12-1.25	0.345	0.807	0.022	0.375	3.50	4	TM12-1.25MM-H	TM12-1.25MM-HA	100868	100886
M12-1.5	0.345	0.792	0.027	0.375	3.50	4	TM12-1.5MM-H	TM12-1.5MM-HA	100871	100889
M12-1.75	0.345	0.787	0.031	0.375	3.50	4	TM12-1.75MM-H	TM12-1.75MM-HA	100874	100892
M12-1	0.400	1.079	0.018	0.500	3.50	4	TM12-1MM-H	TM12-1MM-HA	100895	100931
M14-1.25	0.450	1.103	0.022	0.500	3.50	4	TM14-1.25MM-H	TM14-1.25MM-HA	100898	100934
M14-1.5	0.450	1.087	0.027	0.500	3.50	4	TM14-1.5MM-H	TM14-1.5MM-HA	100901	100937
M14-1.75	0.450	1.134	0.031	0.500	3.50	4	TM14-1.75MM-H	TM14-1.75MM-HA	100904	100940
M14-2	0.450	1.134	0.035	0.500	3.50	4	TM14-2MM-H	TM14-2MM-HA	100907	100943
M16-2.5	0.450	1.122	0.044	0.500	3.50	4	TM16-2.5MM-H	TM16-2.5MM-HA	100910	100946

***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