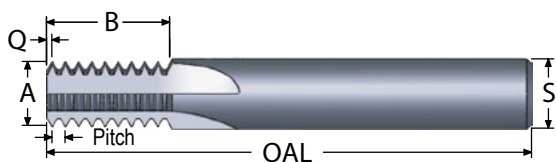


THREAD MILLS - METRIC

STRAIGHT FLUTE - CARBIDE

FULL PROFILE



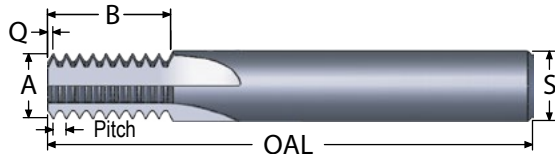
- Short length-of-cut for ideal length-to-diameter ratio
- Polished flute face for optimum performance
- Made with premium submicron grade carbide
- Internal crest cutting design for strongest possible tool

3 FLUTE

MIN ID THREAD/ PITCH*	"A" TOOL DIA.	"B" LENGTH OF CUT	"Q" LENGTH	"S" SHANK DIA.	OAL	ORDER #	
						UNCOATED	ALTiN+
						INTERNAL THREADS ONLY	
M3-.5	0.090	0.264	0.009	0.250	2.50	TM3-.5MM	TM3-.5MM-A
M3-.5	0.090	0.185	0.009	0.250	2.50	TM3-.5MM-S	TM3-.5MM-SA
M3.5-.6	0.090	0.269	0.011	0.250	2.50	TM3.5-.6MM	TM3.5-.6MM-A
M3.5-.6	0.090	0.175	0.011	0.250	2.50	TM3.5-.6MM-S	TM3.5-.6MM-SA
M4-.5	0.110	0.323	0.009	0.250	2.50	TM4-.5MM	TM4-.5MM-A
M4-.5	0.110	0.224	0.009	0.250	2.50	TM4-.5MM-S	TM4-.5MM-SA
M4-.7	0.110	0.342	0.012	0.250	2.50	TM4-.7MM	TM4-.7MM-A
M4-.7	0.110	0.231	0.012	0.250	2.50	TM4-.7MM-S	TM4-.7MM-SA
M4.5-.75	0.125	0.337	0.013	0.250	2.50	TM4.5-.75MM	TM4.5-.75MM-A
M4.5-.75	0.125	0.219	0.013	0.250	2.50	TM4.5-.75MM-S	TM4.5-.75MM-SA
M5-.7	0.140	0.397	0.012	0.250	2.50	TM5-.7MM	TM5-.7MM-A
M5-.7	0.140	0.259	0.012	0.250	2.50	TM5-.7MM-S	TM5-.7MM-SA
M5-.8	0.140	0.391	0.014	0.250	2.50	TM5-.8MM	TM5-.8MM-A
M5-.8	0.140	0.265	0.014	0.250	2.50	TM5-.8MM-S	TM5-.8MM-SA
M6-.5	0.170	0.520	0.009	0.250	2.50	TM6-.5MM	TM6-.5MM-A
M6-.5	0.170	0.382	0.009	0.250	2.50	TM6-.5MM-S	TM6-.5MM-SA
M6-.75	0.170	0.543	0.013	0.250	2.50	TM6-.75MM	TM6-.75MM-A
M6-.75	0.170	0.366	0.013	0.250	2.50	TM6-.75MM-S	TM6-.75MM-SA
M6-1	0.170	0.528	0.018	0.250	2.50	TM6-1MM	TM6-1MM-A
M6-1	0.170	0.370	0.018	0.250	2.50	TM6-1MM-S	TM6-1MM-SA
M6-1.25	0.170	0.561	0.022	0.250	2.50	TM6-1.25MM	TM6-1.25MM-A
M6-1.25	0.170	0.364	0.022	0.250	2.50	TM6-1.25MM-S	TM6-1.25MM-SA
M8-.75	0.235	0.662	0.013	0.250	2.50	TM8-.75MM	TM8-.75MM-A
M8-1	0.235	0.685	0.018	0.250	2.50	TM8-1MM	TM8-1MM-A
M8-1.25	0.235	0.660	0.022	0.250	2.50	TM8-1.25MM	TM8-1.25MM-A

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

THREAD MILLS - METRIC STRAIGHT FLUTE - CARBIDE FULL PROFILE



- Polished flute face for optimum performance
- Made with premium submicron grade carbide
- Internal crest cutting design for strongest possible tool

4 FLUTE

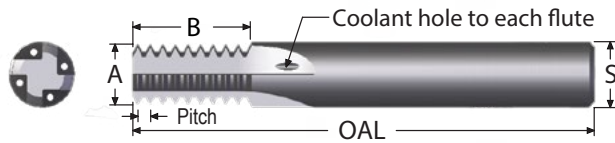
MIN ID THREAD / PITCH*	"A" TOOL DIA.	"B" LENGTH OF CUT	"Q" LENGTH	"S" SHANK DIA.	OAL	ORDER #	
						UNCOATED	ALTiN+
						INTERNAL THREADS ONLY	
M10-1	0.290	0.803	0.018	0.3125	3.50	TM10-1MM	TM10-1MM-A
M10-1.5	0.290	0.792	0.027	0.3125	3.50	TM10-1.5MM	TM10-1.5MM-A
M12-1.25	0.345	0.807	0.022	0.375	3.50	TM12-1.25MM	TM12-1.25MM-A
M12-1.5	0.345	0.792	0.027	0.375	3.50	TM12-1.5MM	TM12-1.5MM-A
M12-1.75	0.345	0.787	0.031	0.375	3.50	TM12-1.75MM	TM12-1.75MM-A
M12-1	0.400	1.079	0.018	0.500	3.50	TM12-1MM	TM12-1MM-A
M14-1.25	0.450	1.103	0.022	0.500	3.50	TM14-1.25MM	TM14-1.25MM-A
M14-1.5	0.450	1.087	0.027	0.500	3.50	TM14-1.5MM	TM14-1.5MM-A
M14-1.75	0.450	1.134	0.031	0.500	3.50	TM14-1.75MM	TM14-1.75MM-A
M14-2	0.450	1.134	0.035	0.500	3.50	TM14-2MM	TM14-2MM-A
M16-2.5	0.450	1.122	0.044	0.500	3.50	TM16-2.5MM	TM16-2.5MM-A

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

METRIC THREAD MILLS

COOLANT THROUGH - SOLID CARBIDE

FULL PROFILE



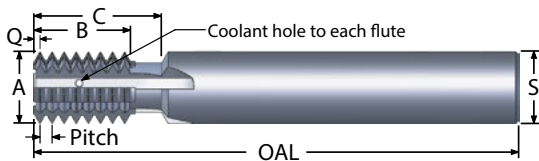
- ALTiN+ coating for higher cutting speed
- Coolant to each flute
- Made with premium submicron grade carbide
- Internal Threads Only

MIN IN THREAD/ PITCH*	"A" TOOL DIA.	"B" LENGTH OF CUT	"Q" LENGTH	"S" SHANK DIA.	OAL	FLUTE	ORDER #	
							UNCOATED	COATED
							INTERNAL THREADS ONLY	
M3-.5	0.090	0.264	0.009	0.250	2.50	3	TMC3-.5MM	TMC3-.5MM-A
M4-.5	0.110	0.323	0.009	0.250	2.50	3	TMC4-.5MM	TMC4-.5MM-A
M4-.7	0.110	0.342	0.012	0.250	2.50	3	TMC4-.7MM	TMC4-.7MM-A
M4.5-.75	0.125	0.337	0.013	0.250	2.50	3	TMC4.5-.75MM	TMC4.5-.75MM-A
M5-.8	0.140	0.391	0.014	0.250	2.50	3	TMC5-.8MM	TMC5-.8MM-A
M6-.5	0.170	0.520	0.009	0.250	2.50	3	TMC6-.5MM	TMC6-.5MM-A
M6-1	0.170	0.528	0.018	0.250	2.50	3	TMC6-1MM	TMC6-1MM-A
M8-1	0.235	0.685	0.018	0.250	2.50	3	TMC8-1MM	TMC8-1MM-A
M8-1.25	0.235	0.660	0.022	0.250	2.50	3	TMC8-1.25MM	TMC8-1.25MM-A
M10-1	0.290	0.803	0.018	0.3125	3.50	4	TMC10-1MM	TMC10-1MM-A
M10-1.5	0.290	0.792	0.027	0.3125	3.50	4	TMC10-1.5MM	TMC10-1.5MM-A
M12-1.25	0.345	0.807	0.022	0.375	3.50	4	TMC12-1.25MM	TMC12-1.25MM-A
M14-1.5	0.450	1.087	0.027	0.500	3.50	4	TMC14-1.5MM	TMC14-1.5MM-A
M14-2	0.450	1.134	0.035	0.500	3.50	4	TMC14-2MM	TMC14-2MM-A

*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 #	
								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
M24-2	0.740	1.100	1.370	0.036	0.750	6.00	4	TMC24-2MM	TMC24-2MM-A
M24-2.5	0.740	1.076	1.370	0.045	0.750	6.00	4	TMC24-2.5MM	TMC24-2.5MM-A
M24-3	0.740	1.058	1.370	0.054	0.750	6.00	4	TMC24-3MM	TMC24-3MM-A
M36-4	0.990	1.095	2.000	0.071	1.000	6.00	6	TMC36-4MM	TMC36-4MM-A

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

ã ä å ç è é ê ë Æ Å Á Â Ã Ä Å Æ Ç È É Ê Ë Ì Í Î Ï Ñ Ò Ó Ô Õ Ö × Ø Ù Ú Û Ü Ý Þ ß à á â ã



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 is 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 $\text{pitch} \div 4$ since 90° is $360^\circ \div 4$.

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.

