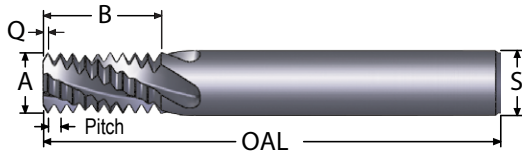


UN THREAD MILLS

30° HELICAL FLUTE SOLID CARBIDE



- Helical flute for reduced side cutting pressure
- 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	"Q" LENGTH	"S" SHANK DIA.	OAL	FLUTES	ORDER #		EDP #	
							UNCOATED	ALTiN+	UNCOATED	ALTiN+
<i>INTERNAL OR EXTERNAL THREADS</i>										
10-24	0.125	0.350	0.019	0.250	2.50	3	TM125-24H	TM125-24HA	103204	103348
10-24	0.140	0.392	0.019	0.250	2.50	3	TM140-24H	TM140-24HA	103210	103354
10-28	0.140	0.407	0.016	0.250	2.50	3	TM140-28H	TM140-28HA	103213	103357
10-32	0.140	0.388	0.014	0.250	2.50	3	TM140-32H	TM140-32HA	103216	103360
10-48	0.140	0.383	0.009	0.250	2.50	3	TM140-48H	TM140-48HA	103219	103363
1/4-20	0.170	0.520	0.023	0.250	2.50	3	TM170-20H	TM170-20HA	103222	103366
1/4-24	0.170	0.517	0.019	0.250	2.50	3	TM170-24H	TM170-24HA	103225	103369
1/4-28	0.170	0.514	0.016	0.250	2.50	3	TM170-28H	TM170-28HA	103228	103372
1/4-32	0.170	0.513	0.014	0.250	2.50	3	TM170-32H	TM170-32HA	103231	103375
1/4-36	0.170	0.511	0.013	0.250	2.50	3	TM170-36H	TM170-36HA	103234	103378
1/4-20	0.187	0.520	0.023	0.250	2.50	3	TM187-20H	TM187-20HA	103237	103381
1/4-24	0.187	0.517	0.019	0.250	2.50	3	TM187-24H	TM187-24HA	103240	103384
1/4-28	0.187	0.514	0.016	0.250	2.50	3	TM187-28H	TM187-28HA	103243	103387
1/4-32	0.187	0.513	0.014	0.250	2.50	3	TM187-32H	TM187-32HA	103246	103390
1/4-36	0.187	0.511	0.013	0.250	2.50	3	TM187-36H	TM187-36HA	103249	103393
1/4-40	0.187	0.511	0.011	0.250	2.50	3	TM187-40H	TM187-40HA	103252	103396
5/16-18	0.235	0.689	0.025	0.250	2.50	3	TM235-18H	TM235-18HA	103255	103399
5/16-20	0.235	0.670	0.023	0.250	2.50	3	TM235-20H	TM235-20HA	103258	103402
5/16-24	0.235	0.684	0.019	0.250	2.50	3	TM235-24H	TM235-24HA	103261	103405
5/16-28	0.235	0.657	0.016	0.250	2.50	3	TM235-28H	TM235-28HA	103264	103408
5/16-32	0.235	0.669	0.014	0.250	2.50	3	TM235-32H	TM235-32HA	103267	103411
5/16-40	0.235	0.660	0.011	0.250	2.50	3	TM235-40H	TM235-40HA	103270	103414
3/8-16	0.290	0.775	0.028	0.3125	3.50	4	TM290-16H	TM290-16HA	103417	103447
3/8-20	0.290	0.820	0.023	0.3125	3.50	4	TM290-20H	TM290-20HA	103420	103450
3/8-24	0.290	0.808	0.019	0.3125	3.50	4	TM290-24H	TM290-24HA	103423	103453
3/8-27	0.290	0.793	0.017	0.3125	3.50	4	TM290-27H	TM290-27HA	103426	103456
3/8-32	0.290	0.794	0.014	0.3125	3.50	4	TM290-32H	TM290-32HA	103429	103459
7/16-14	0.345	0.814	0.032	0.375	3.50	4	TM345-14H	TM345-14HA	103462	103492
7/16-18	0.345	0.800	0.025	0.375	3.50	4	TM345-18H	TM345-18HA	103465	103495
7/16-20	0.345	0.820	0.023	0.375	3.50	4	TM345-20H	TM345-20HA	103468	103498
7/16-24	0.345	0.808	0.019	0.375	3.50	4	TM345-24H	TM345-24HA	103471	103501
7/16-28	0.345	0.800	0.016	0.375	3.50	4	TM345-28H	TM345-28HA	103474	103504
9/16-12	0.400	1.117	0.038	0.500	3.50	4	TM400-12H	TM400-12HA	103507	103585
1/2-13	0.400	1.108	0.035	0.500	3.50	4	TM400-13H	TM400-13HA	103510	103588
1/2-16	0.400	1.088	0.028	0.500	3.50	4	TM400-16H	TM400-16HA	103513	103591
1/2-20	0.400	1.120	0.023	0.500	3.50	4	TM400-20H	TM400-20HA	103516	103594
1/2-24	0.400	1.100	0.019	0.500	3.50	4	TM400-24H	TM400-24HA	103519	103597
1/2-28	0.400	1.086	0.016	0.500	3.50	4	TM400-28H	TM400-28HA	103522	103600
1/2-32	0.400	1.106	0.014	0.500	3.50	4	TM400-32H	TM400-32HA	103525	103603
3/4-10	0.450	1.140	0.045	0.500	3.50	4	TM450-10H	TM450-10HA	103528	103606
5/8-11	0.450	1.127	0.041	0.500	3.50	4	TM450-11H	TM450-11HA	103531	103609
5/8-12	0.450	1.117	0.038	0.500	3.50	4	TM450-12H	TM450-12HA	103534	103612
9/16-16	0.450	1.088	0.028	0.500	3.50	4	TM450-16H	TM450-16HA	103537	103615
9/16-18	0.450	1.078	0.025	0.500	3.50	4	TM450-18H	TM450-18HA	103540	103618
9/16-20	0.450	1.120	0.023	0.500	3.50	4	TM450-20H	TM450-20HA	103543	103621

*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-.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-.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 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: **$(150 \times 3.82) / 0.290$** , 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: **$1975 \times 0.001 \times 4 = 7.9$ 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 **$AFR = NAFR \times (D-d) \div D$**

In this example: **$AFR = 7.9 \times (0.375 - 0.290) \div 0.375$**

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.