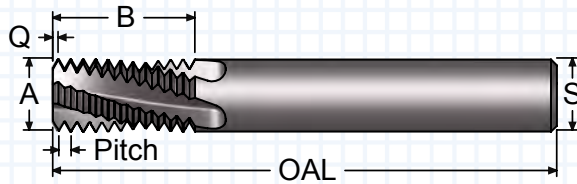


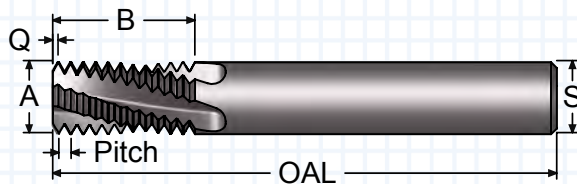
THREAD MILLS - NPS - HELICAL - SOLID CARBIDE (NATIONAL PIPE STRAIGHT)



- Cuts NPSC and NPSM straight pipe threads
- Cuts NPSL and NPSH straight pipe threads
- ALTiN+ for longevity and higher Surface Feet per Minute

THREAD DIA/PITCH	"A" TOOL DIA.	"B" LENGTH OF CUT	"Q" LENGTH OF CUT	"S" SHANK DIA.	OAL	FLUTES	ORDER #		EDP #	
							UNCOATED	ALTiN+	UNCOATED	ALTiN+
INTERNAL THREADS ONLY										
1/8-27	0.280	0.507	0.018	0.3125	3.00	4	TM280-27NPS-H	TM280-27NPS-HA	121000	121001
1/4, 3/8-18	0.370	0.693	0.027	0.3750	3.50	4	TM370-18NPS-H	TM370-18NPS-HA	121004	121005
1/2, 3/4-14	0.490	0.893	0.035	0.5000	3.50	4	TM490-14NPS-H	TM490-14NPS-HA	121008	121009
1 to 2 - 11.5	0.620	1.172	0.043	0.6250	4.00	4	TM620-11.5NPS-H	TM620-11.5NPS-HA	121012	121013
2-1/2 up - 8	0.740	1.561	0.062	0.7500	4.00	4	TM740-8NPS-H	TM740-8NPS-HA	121016	121017

THREAD MILLS - NPSF - HELICAL - SOLID CARBIDE (NATIONAL PIPE STRAIGHT DRYSEAL)



- Cuts NPSF and NPSI straight pipe dryseal threads
- Helical flute reduces side cutting pressure
- Precision ground for maximum concentricity

THREAD DIA/PITCH	"A" TOOL DIA.	"B" LENGTH OF CUT	"Q" LENGTH OF CUT	"S" SHANK DIA.	OAL	FLUTES	ORDER #		EDP #	
							UNCOATED	ALTiN+	UNCOATED	ALTiN+
INTERNAL THREADS ONLY										
1/8-27	0.280	0.507	0.018	0.3125	3.00	4	TM280-27NPSF-H	TM280-27NPSF-HA	121002	121003
1/4, 3/8-18	0.370	0.693	0.027	0.3750	3.50	4	TM370-18NPSF-H	TM370-18NPSF-HA	121006	121007
1/2, 3/4-14	0.490	0.893	0.035	0.5000	3.50	4	TM490-14NPSF-H	TM490-14NPSF-HA	121010	121011
1 to 2 - 11.5	0.620	1.172	0.043	0.6250	4.00	4	TM620-11.5NPSF-H	TM620-11.5NPSF-HA	121014	121015
2-1/2 up - 8	0.740	1.561	0.062	0.7500	4.00	4	TM740-8NPSF-H	TM740-8NPSF-HA	121018	121019

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.