#### THREAD MILLS - EXJ - SOLID CARBIDE (EXTERNAL UNJ THREAD) FULL PROFILE



- ALTiN+ coating extends tool life
- > Precision ground for maximum concentricity
- > Made with premium submicron carbide

EXT.	"A"	"B"	"R"	"Q"	"S"			ORDER #		
THREAD	TOOL	LENGTH	ROOT	LENGTH	SHANK	OAL	FLUTES	UNCOATED	ALTiN+	
/ РІТСН	DIA.	OF CUT	RADIUS		DIA.			EXTERNAL 1	HREADS ONLY	
UNJ-32	0.245	0.668	0.0051	0.016	0.250	2.50	4	TM245-32EXJ	TM245-32EXJ-A	
UNJ-28	0.245	0.657	0.0059	0.018	0.250	2.50	4	TM245-28EXJ	TM245-28EXJ-A	
UNJ-24	0.245	0.683	0.0069	0.020	0.250	2.50	4	TM245-24EXJ	TM245-24EXJ-A	
UNJ-20	0.370	0.773	0.0082	0.024	0.375	3.50	5	TM370-20EXJ	TM370-20EXJ-A	
UNJ-18	0.370	0.800	0.0091	0.026	0.375	3.50	5	TM370-18EXJ	TM370-18EXJ-A	
UNJ-16	0.370	0.774	0.0103	0.029	0.375	3.50	5	TM370-16EXJ	TM370-16EXJ-A	
UNJ-14	0.495	1.027	0.0118	0.033	0.500	3.50	6	TM495-14EXJ	TM495-14EXJ-A	
UNJ-12	0.495	1.031	0.0137	0.038	0.500	3.50	6	TM495-12EXJ	TM495-12EXJ-A	
UNJ-10	0.495	1.037	0.0165	0.046	0.500	3.50	6	TM495-10EXJ	TM495-10EXJ-A	
UNJ-8	0.495	1.046	0.0207	0.057	0.500	3.50	6	TM495-8EXJ	TM495-8EXJ-A	

åãa dãa čo^å Áa ^ Á, , È; ^, { aa)q[[|● È&[{ Á‰ko^|Á∓ËI €€ËEÎ Î Î ËFHÌ | Á‱‰k[[|● O } ^, { aa)q[[|● È&[{



16

## THREAD MILL FEED AND SPEED CHART

	HB/Rc	SPEED SFM* UNCOATED	SPEED SFM ALTIN+	FEED ( INCHES PER TOOTH)						
MATERIAL				TOOL DIAMETER						
				.032056	.059090	.100190	.200350	.370595	.600+	
CAST IRON	160 HB	100-220	200-425	.0004001	.00040008	.00040014	.0004002	.00040035	.0004006	
CARBON STEEL	18 Rc	100-200	190-425	.0003001	.00030008	.00030014	.0003002	.0003005	.0003006	
ALLOY STEEL	20 Rc	80-200	200-375	.0003001 2 Passes	.00030008 3 Passes	.00030014	.00030024	.0003005	.0003006	
TOOL STEEL	20 Rc	80-175	175-250	.00030004 2 Passes	.0003-0.0005 3 Passes	.00030005	.00030009	.00030026	.0003004	
300 STAINLESS STEEL	150 HB	90-120	120-255	.00030005 2 Passes	.0003-0.0006 3 Passes	.00030007	.0003002	.00030035	.00030045	
400 STAINLESS STEEL	195 HB	90-150	140-375	.00030005 2 Passes	.00030006 3 Passes	.00030007	.0003002	.00030026	.00030045	
HIGH TEMP ALLOY (Ni & Co BASE)	20 Rc	50-125	100-125	.00030004 3 Passes	.000300045 3 Passes	.00030005 2 Passes	.00030009	.00030026	.0003004	
TITANIUM	25 Rc	50-130	100-170	.00030004 3 Passes	.000300045 3 Passes	.0003001 2 Passes	.00030009	.00030015	.0003003	
HEAT TREATED ALLOYS (38-45Rc)	40 Rc	50-90	90-150	.00030004 3 Passes	.000300045 3 Passes	.00030005 2 Passes	.00030008	.0003001	.00030025	
ALUMINUM	100 HB	100-800	100-1200	.00050015	.0005002	.00050025	.0005003	.0005006	.0005009	
BRASS, ZINC	80 HB	200-350	200-750	.00050015	.0005002	.00050025	.0005003	.0005006	.0005009	

\*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.

åãa dãa čo^åÁa ^ Á, , È; ^, { aa) q[[|● È&[{ Á% ko^|Á∓ËI €€ËEÎ Î Î Ë= HÌ I Á‱ ka[[|● O } ^, { aa) q[[|● È&[{



# TECH INFO

### 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 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.

åãrdâāčơ∿åÁà^Á,,È;^{a)q[[|•È&[{ ¼¼x°|ÁFÈ:€€ËİÎÍËFHÌ| ¼₩₩q[[|•O}}^,{ a)q[[|•È&[{



### 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 pitch  $\div$  4 since 90° is 360°  $\div$  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.

åãa dãa čơ∿å Áà ^ Á, , È; ^, { æ) q[[|• È&[{ ¼¼ơ | Á∓Ëi €€Ëi Î Í ËFHÌ | ¼₩₩4ξ[[|• O } ^, { æ) q[[|• È&[{

