

REGAL CUTTING TOOLS

THE CUTTING EDGE OF PERFORMANCE



Regal Premium

Vortex

Thread mills

Thread Mills

Spiral Fluted Solid Carbide



Regal Cutting Tools Premium Vortex Thread Mills are a viable alternative to the traditional methods of thread cutting or thread forming. They offer a wide variety of features and benefits.

- Made from sub-micrograin carbide with high transverse rupture strength and excellent wear resistant properties
- Tools are manufactured on state-of-the-art CNC equipment for consistent quality
- Full product offering of UN, Metric, NPT and NPTF
- Stocked with TiALN coating, other coatings upon request
- One thread mill can produce a variety of thread diameters of the same pitch
- One tool can be used for both right hand and left hand threads
- Control the pitch diameter to within .0001
- Produce threads to within one pitch of a shoulder or counterbore
- Produce 100% thread height to UN specifications
- 20° helical flute design reduces chatter and guarantees two flutes always in the cut
- Produces threads with excellent form, finish and accuracy
- Helical flute reduces side-cutting pressure allowing for higher feed rates
- Makes threading difficult materials easy
- Produces small controllable chips
- No need to reverse the spindle
- Eliminates the need of removing broken taps
- Will reduce cycle times compared to indexable thread mills
- Smaller machines can produce larger threads due to less spindle torque
- Eliminates the need for many special dimensional taps

UN Thread Forms

RCT EDP	Min. Thread Size & Pitch	Pitch	Flutes	Cutter Diameter	Shank Diameter	Length of Cut	Overall Length
085900TM	#8-32	32	3	0.115	0.125	0.250	2.000
085901TM	#10-32	32	3	0.120	0.125	0.312	2.000
085902TM	#10-28	28	3	0.120	0.125	0.312	2.000
085903TM	1/4-28	28	3	0.180	0.187	0.500	2.500
085904TM	#10-24	24	3	0.120	0.125	0.312	2.000
085905TM	5/16-24	24	3	0.235	0.250	0.625	2.500
085906TM	3/8-24	24	4	0.285	0.312	0.750	3.000
085907TM	1/4-20	20	3	0.180	0.187	0.500	2.500
085908TM	7/16-20	20	4	0.335	0.375	0.875	3.500
085909TM	5/16-18	18	3	0.235	0.250	0.625	2.500
085910TM	9/16-18	18	4	0.370	0.375	0.875	3.500
085911TM	3/8-16	16	4	0.285	0.312	0.750	3.000
085912TM	3/4-16	16	4	0.490	0.500	1.250	3.500
085913TM	7/16-14	14	4	0.305	0.312	0.750	3.000
085914TM	7/8-14	14	4	0.490	0.500	1.250	3.500
085915TM	1/2-13	13	4	0.350	0.375	0.875	3.500
085916TM	9/16-12	12	4	0.370	0.375	0.875	3.500
085917TM	3/4-12	12	4	0.495	0.500	1.250	3.500
085918TM	5/8-11	11	4	0.470	0.500	1.250	3.500
085919TM	3/4-10	10	4	0.495	0.500	1.250	3.500
085920TM	7/8-9	9	4	0.620	0.625	1.375	4.000
085921TM	1"-8	8	4	0.620	0.625	1.375	4.000

Thread Mills

Spiral Fluted Solid Carbide

THREAD MILLS

Metric Thread Mills



RCT EDP No.	Min. Thread Size & Pitch	Pitch	Flutes	Cutter Diameter	Shank Diameter	Length of Cut	Overall Length
085950TM	M4.5	0.75	3	0.120	0.125	0.250	2.000
085951TM	M8	0.75	3	0.235	0.250	0.625	2.500
085952TM	M5	0.80	3	0.120	0.125	0.312	2.000
085953TM	M6	1	3	0.170	0.187	0.500	2.500
085954TM	M12	1	4	0.360	0.375	0.875	3.500
085955TM	M8	1.25	3	0.235	0.250	0.625	2.500
085956TM	M10	1.5	4	0.300	0.312	0.750	3.000
085957TM	M14	1.5	4	0.370	0.375	0.875	3.500
085958TM	M18	1.5	4	0.490	0.500	1.250	3.500
085959TM	M16	2	4	0.470	0.500	1.250	3.500
085960TM	M20	2.5	4	0.495	0.500	1.250	3.500
085961TM	M24	3	4	0.620	0.625	1.375	4.000

NPT & NPTF Thread Mills



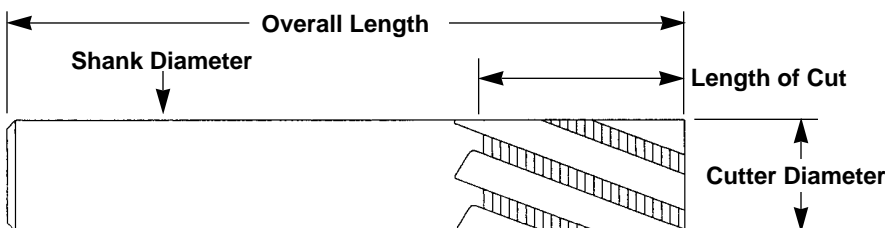
RCT EDP No.	Thread Size & Pitch	Pitch	Flutes	Large End Cutter Diameter	Shank Diameter	Length of Cut	Overall Length
085970TM	1/16 & 1/8 NPT	27	3	0.245	0.250	0.437	2.500
085971TM	1/4 & 3/8 NPT	18	4	0.305	0.312	0.625	3.000
085972TM	1/2 & 3/4 NPT	14	4	0.495	0.500	0.875	3.500
085973TM	1" & 2" NPT	11-1/2	4	0.620	0.625	1.125	4.000
085980TM	1/16 & 1/8 NPTF	27	3	0.245	0.250	0.437	2.500
085981TM	1/4 & 3/8 NPTF	18	4	0.305	0.312	0.625	3.000
085982TM	1/2 & 3/4 NPTF	14	4	0.495	0.500	0.875	3.500
085983TM	1" & 2" NPTF	11-1/2	4	0.620	0.625	1.125	4.000

Heavy Duty Thread Mills



- Designed for tough to machine aerospace alloys
- Special geometry for hard materials (40 + RC)

RCT EDP No.	Min. Thread Size & Pitch	Pitch	Flutes	Cutter Diameter	Shank Diameter	Length of Cut	Overall Length
085940TM	#10-32	32	3	0.150	0.187	0.312	2.500
085941TM	#10-24	24	3	0.145	0.187	0.312	2.500
085942TM	1/4-20	20	3	0.195	0.250	0.500	2.500
085943TM	5/16-18	18	3	0.245	0.312	0.625	3.000
085944TM	3/8-16	16	4	0.300	0.375	0.750	3.500
085945TM	1/2-13	13	4	0.400	0.500	0.875	3.500



Special Thread Mills are available upon request. Two weeks delivery on most specials; add additional week for coatings.

Programming Request & Support

REQUEST THE NO CHARGE SPREADSHEET PROGRAM!

A programming spread sheet is available upon request at no charge to our customers.

(The program may not be suitable for all machines or circumstances)

Regal Cutting Tools' technical support staff is available to aid or provide programming for our thread mills. Complete and accurate job details need to be supplied by fax. Phoned information will not be acceptable.

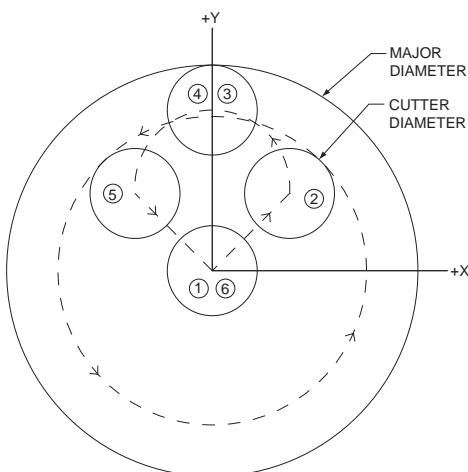
Most CNC machines produced in the last 10 years are suitable for helical interpolation. Some machines prior to that time supplied it as an option which can be enacted with a parameter. The best way to determine if your machine is capable is to MDI the part program to see if it will make the moves. It is generally recommended to cut fine threads in one pass and coarse threads in two passes. If you need to use two passes, just repeat the program below and use a different D word to reduce the depth of the first pass. If cutting a 3/8-16 thread, increase the offset for the first D worked by about 35% of the pitch or $(.35 \times .0625) = .0218$.

Sample Program for a 3/8-16 x 3/4 Deep Thread with a .235 Diameter Thread Mill

N05	M3 S3521			Z-7578			
N10	G01G91			D (Offset #)			F30.00
N15	G41	X .0350	Y .0350	Z .0078	I -0.350	J 0	
N20	G03	X -.0350	Y -.0350	Z .0625	I 0	J -.070	F4.85
N25	G03	X 0	Y 0	Z .0078	I 0	J -.035	F4.85
N30	G03	X -.0350	Y -.0350				F9.70
N35	G01G40	X .0350	Y .0350	Z .6797			F30.00
N40	G00						
N45	G90						

Thread Mills cut by feeding into a predrilled hole, feed out to the major diameter and then in one revolution generate the entire length of thread. In producing a right hand thread, the "Z" axis will feed the tool **out** one pitch as the tool rotates clockwise while the spindle travels 360° counterclockwise. To produce a left hand thread, the tool is fed one pitch **in** of the part; all else remains equal.

Thread Mill Programming Help Sheet for UN Internal Threads



When programming thread mills, the start position in X and Y should be at the center of the drilled hole. The Z start position will be at the top of the part. All the programming is done in incremental mode since the absolute hole positions are not known. All the cutting will be climb milling.

The machine will go thru the following sequences when programming:

- (0-1) Feed Z in at 30 IPM to the full thread depth plus 1/8 of a pitch
- (1-2) Feed X and Y to position 2 (clear diameter) and activate the cutter compensation
- (2-3) Feed CCW thru 45 degrees at the programmed feed rate in X, Y, and Z to ramp out to the major diameter. The Z move will be out of the part and equal to 1/8 of the pitch (45/360 degrees = 1/8)
- (3-4) Feed CCW around the entire circle to cut the threads in on revolution. The Z move will be out of part and be equal to the pitch (ex: 16 pitch = .0625)
- (4-5) Feed CCW thru 45 degrees to ramp out of the part to the clear diameter. The Z move will be out of the part and equal 1/8 of the pitch.
- (5-6) Move the cutter back to centerline at 30 IPM and delete the cutter compensation.

The last move is to rapid Z out to the top of the part and return to "absolute" mode.

Programming Request & Support

Work Piece Material	Speed SFM	Feed Rate (Inches/Tooth)													
		Tool Cutting Diameter													
		1/8		3/16		1/4		5/16		3/8		1/2 5/8			
Aluminum	800 1400	.0005	.001	.001	.0015	.0015	.0025	.002	.003	.003	.0045	.0035	.0055	.005	.007
Magnesium	800 1400	.0005	.001	.001	.0015	.0015	.0025	.002	.003	.003	.0045	.0035	.0055	.005	.007
Brass	600 800	.0005	.001	.001	.0015	.0015	.0025	.002	.003	.003	.0045	.0035	.0045	.005	.006
Bronze	500 600	.0005	.001	.001	.0015	.0015	.0025	.002	.003	.003	.0045	.0035	.0045	.005	.006
Hard Bronze	200 300	.0004	.0008	.0007	.0012	.001	.002	.001	.002	.0015	.0025	.002	.003	.003	.004
Low Alloy Steels <25Rc	350 500	.0005	.001	.001	.0015	.0015	.0025	.002	.003	.0025	.0035	.003	.004	.004	.005
High Alloy Steels >25Rc	250 400	.0003	.0006	.0005	.001	.0008	.0015	.001	.002	.0015	.0025	.002	.003	.003	.004
Stainless Steel	150 250	.0004	.0008	.0006	.001	.001	.0015	.0015	.002	.0015	.0030	.002	.0035	.003	.004
Cast Iron Soft	250 350	.0004	.0008	.0007	.0013	.0007	.0013	.0015	.002	.002	.003	.002	.004	.003	.005
Cast Iron Hard	200 300	.0003	.0006	.0005	.001	.0008	.0015	.001	.002	.0015	.0025	.002	.003	.003	.004
Titanium	80 150	.0003	.0006	.0005	.001	.0008	.0015	.001	.002	.0015	.0025	.0015	.0025	.0025	.0035
Inconel	60 100	.0003	.0006	.0005	.001	.0008	.0015	.001	.002	.0015	.0025	.0015	.0025	.002	.003

When cutting an internal thread, the linear feed rate has to be reduced to compensate for the ratio of the tool's cutting diameter to the major diameter being cut. If not compensated, the feed rate that the cutting edge sees will be much greater and tool failure will occur.

Example #1:

Thread Diameter to be Cut (D2): 3/8
Thread Mill Diameter (D1): .285

If the linear feed rate calculated from the feeds and speeds above are 10 IPM, the programmed feed rate for the internal thread milling cut would be $((D2-D1)/D2) \times \text{linear feed rate}$ or $((.375 - .285)/.375) \times 10$ or 2.4 inches per minute.

Example #2:

Application: To cut a 1/2-13 thread in low alloy steel, less than 25 Rc hardness.

Thread Mill: EDP 85915 .350 diameter, 4 flutes

SFM: 350 to 500 (use 400 in this example)

Feed/Tooth: .0025 .0035 (use .003"/tooth in the example)

RPM: Take 3.8 divided by the thread mill diameter and multiply by the SFM

Example: $((3.8/.350) \times 40\text{SFM}) = 4343 \text{ RPM}$

Liner Inches: Take the (inch/tooth) \times (# flutes) \times RPM

Example: $((.003" \times 4 \text{ flutes}) \times 4343 \text{ RPM}) = 52 \text{ linear inches per minute}$

Answer: $((D2-D1)/D2) \times \text{linear feed rate}$ or $((.500 - .350)/.500) \times 52 \text{ IPM} = 15.5 \text{ IPM}$

Tool Holder Requirements

Regal Cutting Tools Solid Carbide Thread Mills are capable of running at very high RPM's and therefore must be held securely and concentrically. For the best tool performance and thread quality, an end mill type holder with screw tightening is recommended. Only use precision ground holders or one that has been ground eccentric to minimize runout due to the side clamping screw. Collet chucks are not recommended as they tend to runout, which may lead to premature tool chipping or catastrophic failure.

It is equally important to keep the tool length projecting out of the holder to a minimum. Minimizing the overhand will reduce deflection, aid in extending tool life and enhance the quality of threads being produced. A tool that extends out of the holder twice as far will have eight times more deflection given the same side forces. The tools should be inserted into the holder as close to the flute section as possible.