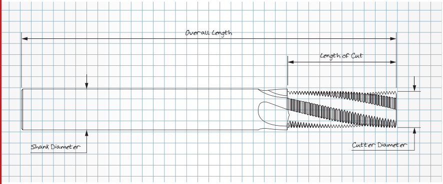
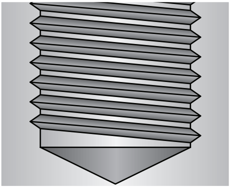
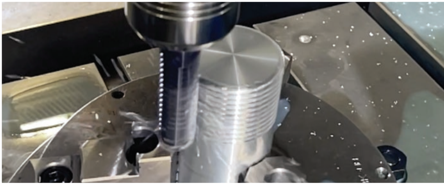
THREAD MILLING

A Quick Reference Pocket Guide







ALLIED MACHINE & ENGINEERING®

Holemaking Solutions for Today's Manufacturing

www.**alliedmachine**.com

No matter the type of holemaking, Allied is here to help you.

Whether you're a production facility producing thousands of parts for one customer or a job shop making a handful of parts for a thousand customers, we're here to make sure the job gets done. Our precision holemaking and finishing solutions are backed by our experienced staff of knowledgeable engineers who are standing by.

Don't hesitate to call us. Let us know what problems you're having, and we'll find the solution. Machining is what we do, and we don't mind showing off what we know.

After all, we are the holemaking experts. All you have to do is ask.

ALLIED MACHINE & ENGINEERING®
Holemaking Solutions for Today's Manufacturing

Contents

What We Offer

AccuThread® 856
Solid Carbide

ThreadMills USA™
Solid Carbide

AccuThread® T3
Solid Carbide

AccuThread® 856
Bolt-in Style Indexable Inserts

AccuThread® 856
Pin Style Indexable Inserts

Insta-Code®

Find your thread mill.
Create your program.

The all new software lets you choose the best thread mill product for your application and create the program code for your machine. Insta-Code is available as a PC download app (that can be used offline) and an online web app available 24/7 at ***www.alliedmachine.com/InstaCode***.

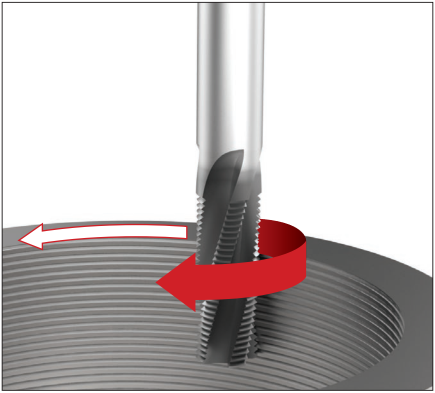
Eliminate the wait. Get your program now.

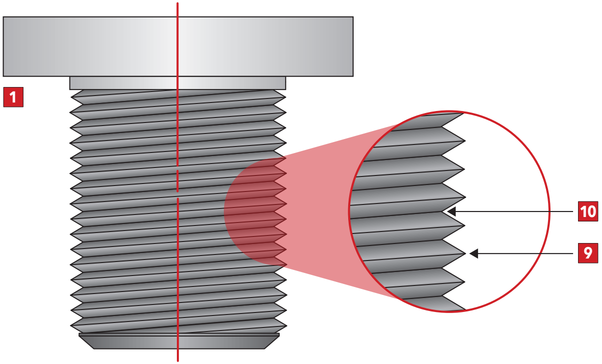
Online Version **Download Version** **Offline Version Update**

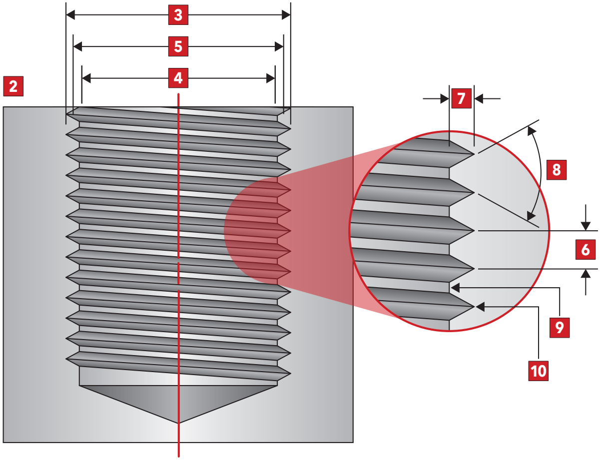
A thread mill can be thought of as an end mill with the profile of the thread on the side. Using multiple axes of a CNC machine allows you to helically interpolate the thread profile into a workpiece.

Unlike a tap, a thread mill has a smaller diameter than the thread size being produced because the mill must be able to enter into the drilled hole and interpolate.

Thread milling provides excellent control of the thread's major diameter because it can be controlled using the machine's cutter compensation.




1
10
9


3
5
4
2
7
8
6
9
10

1 **External Thread:** The mating part with threads on the external surface.

2 **Internal Thread:** The mating part with threads on the internal surface.

(***Nominal Diameter:*** *The theoretical diameter of the threaded assembly.*)

3 **Major Diameter:** The largest diameter of either the internal or external thread.

4 **Minor Diameter:** The smallest diameter of either the internal or external thread.

5 **Pitch Diameter:** A theoretical diameter used to correlate the dimensions between an internal and external thread assembly.

6 **Pitch:** The distance between like points on a thread and is typically measured between crests.

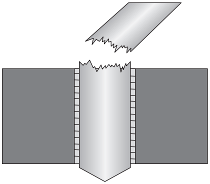
7 **Thread Depth:** The distance from root to crest when measuring perpendicular to the thread's axis.

8 **Thread Angle:** The included angle from crest to root of each thread.

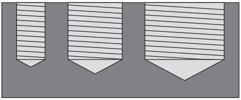
9 **Crest:** The surface area of each thread's peak on either an internal or external thread.

10 **Root:** The surface area of each thread's valley on either an internal or external thread.

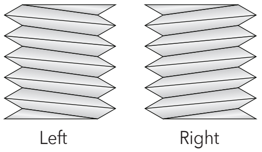
Right on size!
Using the machine's cutter compensation, a thread's major diameter can easily be altered to produce precise threads. Tool life is increased over common tapping because wear can be compensated when using thread mills.



No more broken taps!
When taps break, it can be very costly to salvage the workpiece. If a thread mill breaks during a cut, a new thread mill can reenter into the hole and start from the beginning with very little time lost. Producing threads in large or expensive parts is no longer a risk. A presetter is suggested for tool height accuracy.

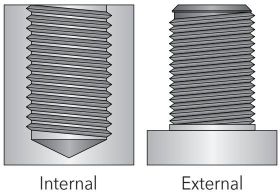


Multiple diameters? No problem.
A thread mill is simply an end mill with the thread form and a specific pitch. The machine can be programmed to helically interpolate to the desired diameter. A 1/4-20 thread mill can produce a 1/2-20 or even a 7/8-20 thread making it an excellent choice for uncommon thread sizes or large diameter threads.



Left- and right-handed threads

Thread mills are not left-hand or right-hand specific. While the thread mill is interpolating, the thread mill must move up or down one pitch in the Z-axis to create the thread's helix. If the Z-axis is not moving, the thread mill will just produce grooves inside of the hole. No more ordering a special left-handed tap--one thread mill provides multiple solutions.



Internal and external threads

Unlike tapping, threads can be milled inside of the hole or on the outside of the material; however, not all thread forms have identical details for their internal and external mating surfaces. Some threads may require an internal or external specific thread mill.

Low horsepower required

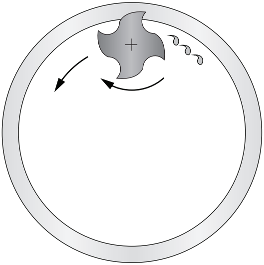
The diameter of a tap is completely engaged in the hole and requires more torque especially on large diameter threads. Cutting large diameter threads with coarse pitches becomes even more difficult. Thread mills only cut a portion of the circumference at a time, which significantly reduces the torque and horse power requirements.

G-Codes	
G00	Send to position with rapid feed
G01	Send to position with linear movement and control by feed
G02	Clockwise circular interpolation
G03	Counterclockwise circular interpolation
G40	Cutter compensation cancel
G41	Turn on left-hand cutter compensation
G42	Turn on right-hand cutter compensation
G54-59	Available workpiece coordinate settings
G90	Absolute positioning
G91	Incremental positioning

M-Codes	
M00	Program stop
M01	Program optional stop
M03	Turn on spindle clockwise direction
M04	Turn on spindle counterclockwise direction
M05	Turn off spindle rotation
M06	Tool change
M08	Coolant on
M09	Coolant off
M30	Program end and reset to start of program

When milling a workpiece, the cutting tool can be fed in different directions along the workpiece. Both directions will achieve material removal but can have significant effects on the cutting tool and the milled surface.

Climb Milling

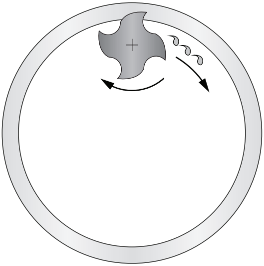


Improved Finish and Tool Life

Less Heat:
Chips evacuate away from cutter faster.

Less Deflection:
Cutting tooth enters the workpiece in a manner that creates a chip that's thicker at the beginning and thinner at the end.

Conventional Milling

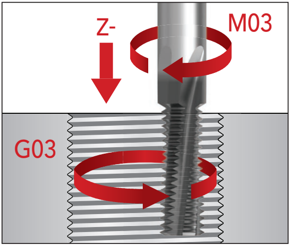


Poor Finish and Tool Life

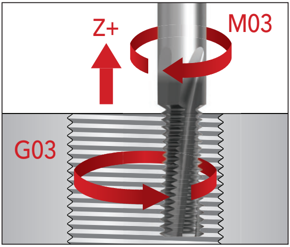
Increased Heat:
Chips evacuate in front of the cutting tool, allowing for the chips to be recut and generate heat.

Less Deflection:
Cutting tooth enters the workpiece in a rubbing manner that creates a chip that's thinner at the beginning and thicker at the end.

Climb Milling

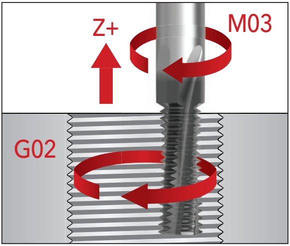


Internal left-hand thread

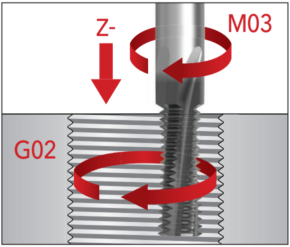


Internal right-hand thread

Conventional Milling

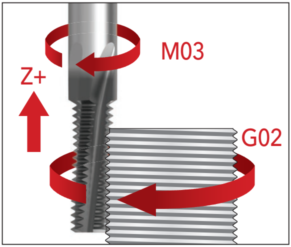


Internal left-hand thread

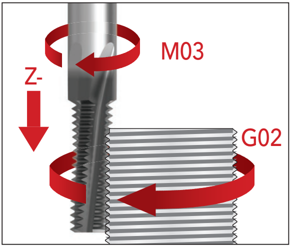


Internal right-hand thread

Climb Milling

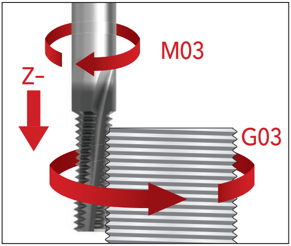


External left-hand thread

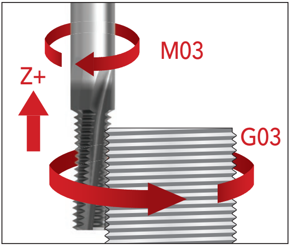


External right-hand thread

Conventional Milling



External left-hand thread



External right-hand thread

What you need to know

- Thread milling can be easily accomplished with simple G-code programming.
- If your machine is capable of three-axis (helical) interpolation, you can utilize thread milling.
- Basic programming of a one pass thread mill can be achieved in six basic steps.

The following are examples of how to calculate and program a 7/16-20 right-hand thread that will be 1/2 deep produced in one pass.

Major thread diameter *Major diameter of thread (7/16 = 0.4375)*	**0.4375**
TPI *Number of threads per inch (20 is from 7/16-20 designation)*	**20**
Length of thread *Desired length of cut*	**0.5**
SFM *Recommended surface footage for material to be cut*	**475**
Feed per flute *Recommended feed rate per cutting edge*	**0.0025**
Number of flutes *Number of flutes on tool to be used*	**4**
Tool diameter *Diameter of cutting tool*	**0.335**

Pre-program notes: The start point for the AMEC program is the X, Y, and Z center of the top of the hole. Your program should change to the thread mill tool and move it into position. Insert the thread mill program at each location where the thread mill sequence is desired. The AMEC program will switch the machine to incremental, machine one pitch, return to the top/center of the hole, and switch the machine back to absolute.

Helical interpolation reference: When using a G02 or G03 for helical interpolation, the X and Y values are the landing location for the cutting tool. The I and J values are the center point for the rotation. The I value is relative to the X starting point, and the J value is relative to the Y starting point.

Using the information on the previous page, the values can be calculated:

Pitch *= 1 / TPI*	**0.05**
RPM *(SFM • 3.82) / tool diameter*	**5416**
Linear feed *RPM • feed per flute • number of flutes*	**54.16**
Feed rate for thread milling *linear feed • ((major thread Ø - tool Ø) / major thread Ø)*	**12.69**
Z-axis move on arc on *(pitch / 8)*	**0.0063**
Z-axis move for full thread *(pitch / 8) +length of cut*	**0.5063**
Arc on/off *(major thread diameter - tool diameter) / 4*	**0.0256**
Full rotation value *(major thread diameter - tool diameter) / 2*	**0.05125**

Major thread diameter	0.4375
Tool diameter	0.335
Length of thread	0.5
Feed rate for thread milling	12.69
Z-axis depth for full thread	0.5063

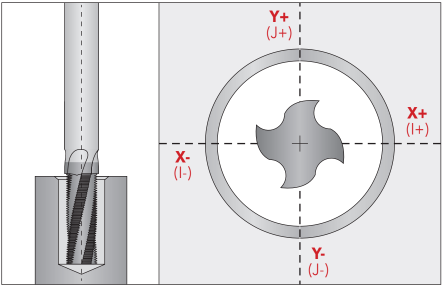
Z-axis for arc on/off	0.0063
Arc on/off value (X,Y)	0.0256
Full rotation value (X,Y)	0.05125
Pitch value	0.05

WARNING: The program is based off the center of the tool. No diameter value should be stored in the machine's controller for the thread mill. This will cause a machine controller error.

1 **Spindle on** | ▸ **S5416 M03** | *Turn on spindle in clockwise direction.*

2 **Lower tool**

▸ **G91 G01 Z-0.5063 F50.0**

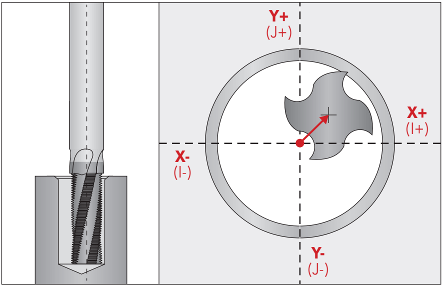


Lower tool the depth required plus an additional 1/8 pitch for the arc on.

Tool remains in center line of hole.

3 **Position for arc on**

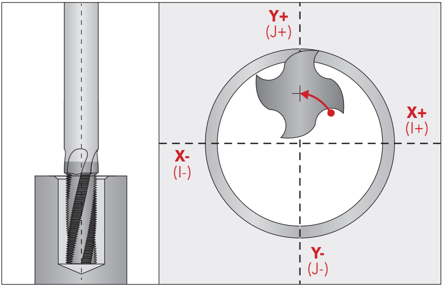
▸ **G41 G01 X0.0256 Y0.0256 D1 F3.17**



Position tool for the arc on motion and turn on cutter compensation.

4 Arc on

▸ G03 X-0.0256 Y0.0256 Z0.0063 I-0.0256 J0.0000 F12.69

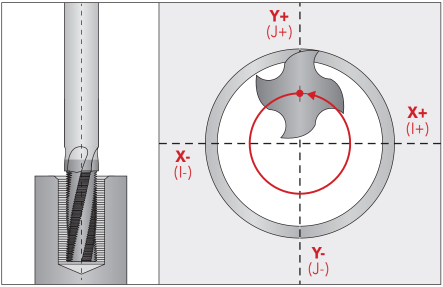


Arc on to engage the tool to the major thread Ø while moving the tool up in the Z-axis 1/8 pitch.

NOTE: X and Y is the end point. I and J is the center point of the arc.

5 **Full pass** *(see page #'s for tapered thread example)*

▸ G03 X0.0000 Y0.0000 Z0.0500 I0.0000 J-0.0513 F12.69

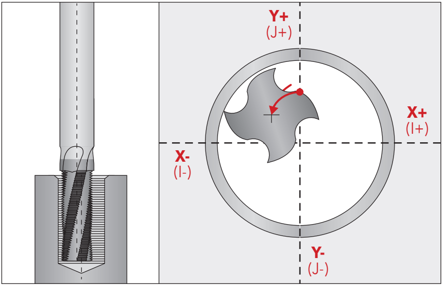


Interpolate the thread mill inside the major thread Ø while moving the tool up one pitch in the Z-axis.

CONTINUE

6 Arc off

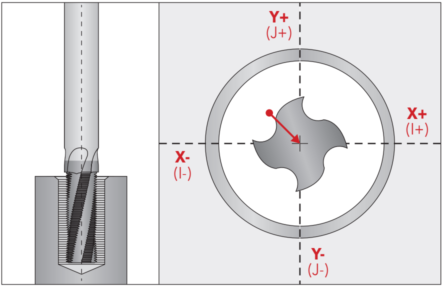
▸ G03 X-0.0256 Y-0.0256 Z0.0063 I0.0000 J-0.0256 F25.38



Arc off to disengage the tool from the major thread Ø while moving the tool up in the Z-axis 1/8 pitch.

7 Return to center

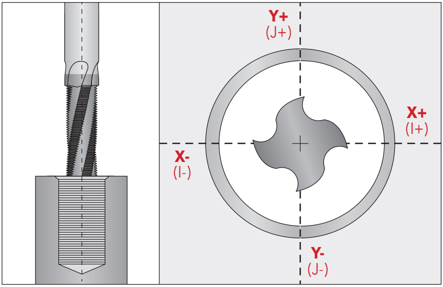
▸ G40 G01 X0.0256 Y-0.0256 Z0.0000 F50.0



Turn off cutter compensation and move the tool back to center.

8 Return to top

▸ **G00 Z0.4438**



Rapid the tool back to the top of the threaded hole.

9 Back to absolute

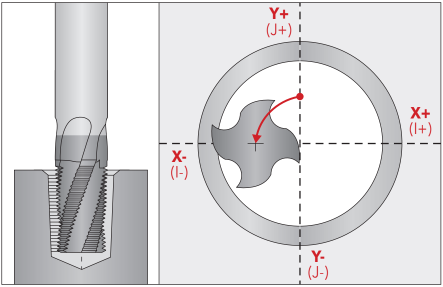
▸ **G90**

Switch back to absolute to continue.

Even though the thread mill is tapered, the thread mill must move out in X and Y as it moves up in Z. This will allow the tool to follow the taper being created. The helical interpolation shown in **Step 5** on the previous page must be broken into segments to make the diameter adjustments. Typically, this adjustment is done in quadrants for simplicity:

5A First quadrant

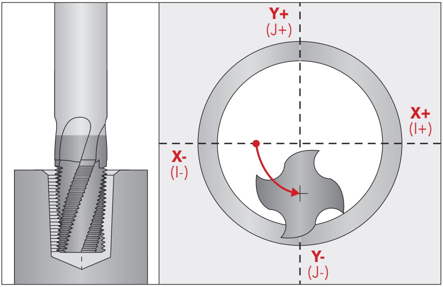
▸ G03 X-0.1819 Y-0.1815 Z0.0139 I0.0000 J-0.1819



Move to 9 o'clock position, move Z-axis 1/4 of the pitch.

5B Second quadrant

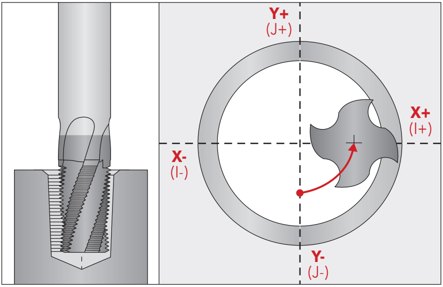
▸ G03 X0.1819 Y-0.1824 Z0.0139 I0.1824 J0.0000



Move to 6 o'clock position, move Z-axis 1/4 of the pitch.

5C Third quadrant

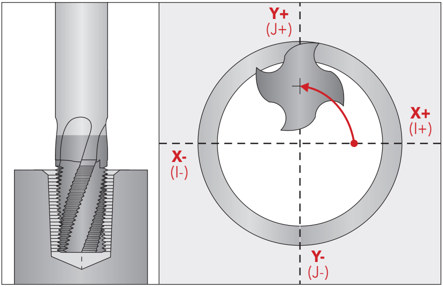
▸ **G03 X0.1828 Y0.1824 Z0.0139 I0.0000 J0.1828**



Move to 3 o'clock position, move Z-axis 1/4 of the pitch.

5D Fourth quadrant

▸ **G03 X-0.1828 Y0.1832 Z0.0139 I-0.1832 J0.0000**



Move to 12 o'clock position, move Z-axis 1/4 of the pitch.

Example values are from 3/8-18 NPT right-hand thread.

NOTE: The radial adjustment needed for producing one thread can be found by multiplying the taper by one pitch. Divide by four to segment this adjustment into quadrants.

AccuThread T3 is designed to cut minimal threads at a time to reduce side deflection in hard or difficult-to-machine materials.

To improve rigidity, they are intended to be programmed to cut each thread starting at the top and progressing down in Z-axis.

Because right-hand threads are most common, AccuThread T3 thread mills are left-hand cutting (M04) to maintain the climb milling method.

1 Spindle on | ▸ **S5416 M04**

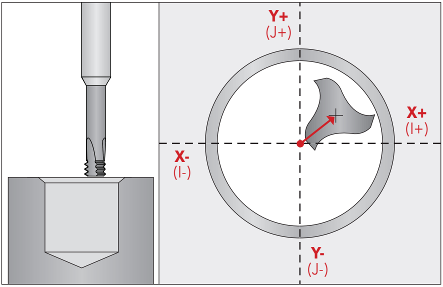
Turn on spindle in counterclockwise direction.

2 Lower tool | ▸ **G91 G42 D1**

Switch to incremental programming and turn on cutter compensation.

3 Position for arc on

▸ **G01 X0.0314 Y0.0314 Z0 F23.6**

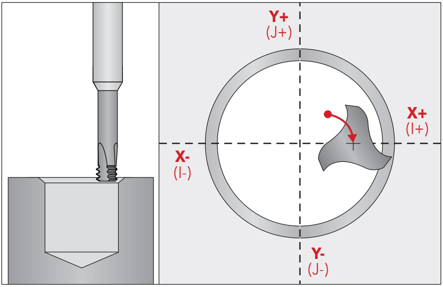


Position tool for the arc on motion.

Example shows right-hand thread.

4 Arc on

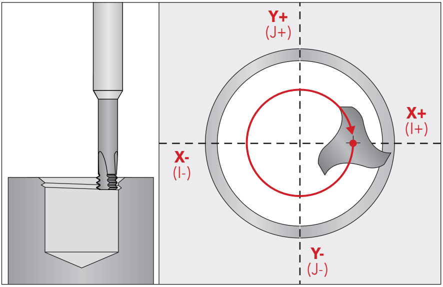
▸ G02 X0.0314 Y-0.0314 Z-0.0063 I0 J-0.0314 F7.87



Arc on to engage the tool to the major thread Ø while moving the tool down in the Z-axis 1/8 pitch.

5 Full pass

▸ G02 X0 Y0 Z-0.0500 I-0.0628 J0

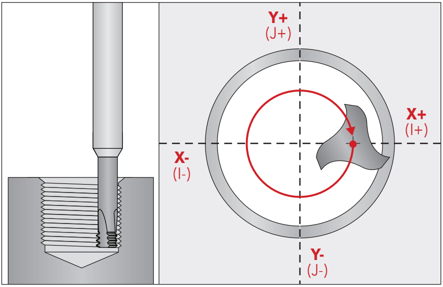


Interpolate the thread mill inside the major thread diameter while moving the tool down one pitch in the Z-axis.

CONTINUE

5A Repeat full pass

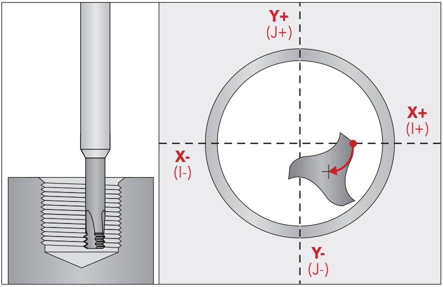
▸ G02 X0 Y0 Z-0.0500 I-0.0628 J0



*Repeat interpolation the number of times to create length of thread needed. Use the **length of thread / pitch** and round the result to determine the number of Z-axis passes required.*

6 Arc off

▸ G02 X-0.0314 Y-0.0314 Z-0.0063 I-0.0314 J0

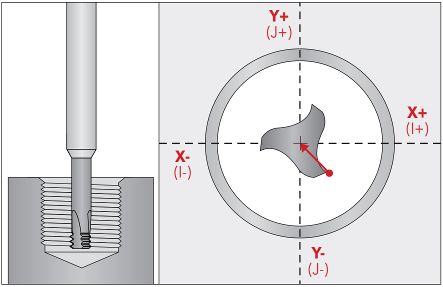


Arc off to disengage the tool from the major thread Ø while moving the tool up in the Z-axis 1/8 pitch.

Example shows right-hand thread.

7 Return to center

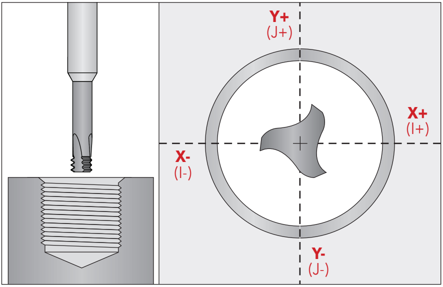
▸ G01 G40 X-0.0314 Y0.0314 F39.34



Return tool back to center and turn off cutter compensation.

8 Return to top

▸ G00 Z1.0126



Return tool to one pitch above the part.

9 Back to absolute

▸ G90

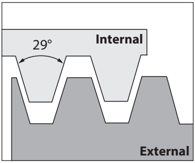
Switch back to absolute to continue.

Conversion Chart

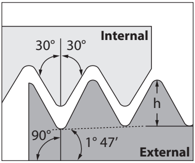
FRACTIONS	DECIMALS	MM
1/64	.0156	0.396
1/32	.0312	0.793
3/64	.0468	1.190
1/16	.0625	1.587
5/64	.0781	1.984
3/32	.0937	2.381
7/64	.1093	2.778
1/8	**.1250**	**3.175**
9/64	.1406	3.571
5/32	.1562	3.968
11/64	.1718	4.365
3/16	.1875	4.762
13/64	.2031	5.159
7/32	.2187	5.556
15/64	.2343	5.953
1/4	**.2500**	**6.350**
17/64	.2656	6.746
9/32	.2812	7.143
19/64	.2968	7.540
5/16	.3125	7.937
21/64	.3281	8.334
11/32	.3437	8.731
23/64	.3593	9.128
3/8	**.3750**	**9.525**
25/64	.3906	9.921
13/32	.4026	10.318
27/64	.4218	10.715
7/16	.4375	11.112
29/64	.4531	11.509
15/32	.4687	11.906
31/64	.4843	12.303
1/2	**.5000**	**12.700**

FRACTIONS	DECIMALS	MM
33/64	.5156	13.096
17/32	.5312	13.493
35/64	.5468	13.890
9/16	.5625	14.287
37/64	.5781	14.684
19/32	.5937	15.081
39/64	.6093	15.478
5/8	**.6250**	**15.875**
41/64	.6406	16.271
21/32	.6562	16.668
43/64	.6718	17.065
11/16	.6875	17.462
45/64	.7031	17.859
23/32	.7187	18.256
47/64	.7343	18.653
3/4	**.7500**	**19.050**
49/64	.7656	19.446
25/32	.7812	19.843
51/64	.7968	20.240
13/16	.8125	20.637
53/64	.8281	21.034
27/32	.8437	21.431
55/64	.8593	21.828
7/8	**.8750**	**22.225**
57/64	.8906	22.621
29/32	.9062	23.018
59/64	.9218	23.415
15/16	.9375	23.812
61/64	.9531	24.209
31/32	.9687	24.606
63/64	.9843	25.003
1	**1.0000**	**25.400**

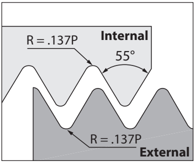
Common Thread Forms



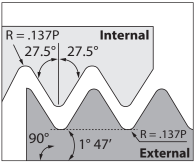
ACME (full profile)



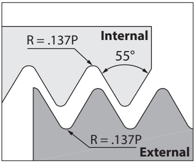
API Round



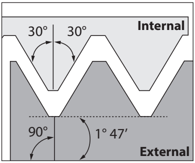
BSPP



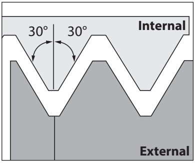
BSPT



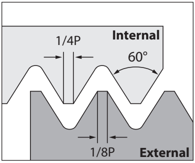
BSW



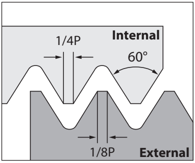
NPT / NPTF



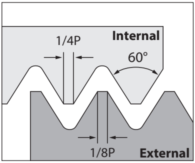
NPS / NPSF



UNC / UNF



UNJC / UNJF



ISO

Ø	UNC		UNF	
	Pitch (TPI)	Drill Size (in)	Pitch (TPI)	Drill Size (in)
#0	–	–	80	3/64
#1	64	0.059 (#53)	72	0.059 (#53)
#2	56	0.067 (#51)	64	0.070 (#50)
#3	48	5/64	56	0.081 (#46)
#4	40	0.089 (#43)	48	0.093 (#42)
#5	40	0.099 (#39)	44	0.104 (#37)
#6	32	0.106 (#36)	40	0.113 (#33)
#8	32	0.136 (#29)	36	0.136 (#29)
#10	24	0.149 (#25)	32	0.159 (#21)
#12	24	0.173 (#17)	28	0.180 (#15)
1/4	20	0.201 (#7)	28	0.213 (#3)
5/16	18	0.257 (F)	24	0.272 (I)
3/8	16	5/16	24	0.332 (Q)
7/16	14	0.368 (U)	20	0.386 (W)
1/2	13	27/64	20	29/64
9/16	12	31/64	18	33/64
5/8	11	17/32	18	37/64
3/4	10	21/32	16	11/16
7/8	9	49/64	14	13/16
1	8	7/8	12	15/16
1-1/8	7	63/64	12	1-1/32
1-1/4	7	1-7/64	12	1-11/64
1-3/8	6	1-7/32	12	1-19/64
1-1/2	6	1-11/32	12	1-27/64
1-3/4	5	1-35/64	12	1-43/64
2	4-1/2	1-25/32	12	1-59/64

UNJC		Minor Ø (in)		Recommended Drill Size	
Ø	Pitch (TPI)	Min	Max	Drill Size	Decimal (in)
#4	40	.0877	.0942	2.30 mm	.0906
#5	40	.1007	.1072	2.60 mm	.1024
#6	32	.1076	.1157	#33	.1130
#8	32	.1336	.1417	3.50 mm	.1378
#10	24	.1494	.1600	3.90 mm	.1535
#12	24	.1754	.1852	4.60 mm	.1811
1/4	20	.2013	.2121	5.30 mm	.2087
5/16	18	.2584	.2690	6.70 mm	.2638
3/8	16	.3141	.3250	8.10 mm	.3189
7/16	14	.3680	.3795	9.50 mm	.3740
1/2	13	.4251	.4368	10.90 mm	.4291
9/16	12	.4814	.4914	31/64"	.4844
5/8	11	.5365	.5474	13.80 mm	.5433
3/4	10	.6526	.6646	16.75 mm	.6594
7/8	9	.7668	.7801	19.60 mm	.7717
1	8	.8783	.8933	22.50 mm	.8858

UNJF		Minor Ø (in)		Recommended Drill Size	
Ø	Pitch (TPI)	Min	Max	Drill Size	Decimal (in)
#4	48	.0917	.0971	2.40 mm	.0945
#5	44	.1029	.1088	2.70 mm	.1063
#6	40	.1137	.1202	3.00 mm	.1181
#8	36	.1370	.1442	#28	.1405
#10	32	.1596	.1675	4.20 mm	.1654
#12	28	.1812	.1896	#13	.1850
1/4	28	.2152	.2229	7/32"	.2188
5/16	24	.2719	.2799	7.00 mm	.2756
3/8	24	.3344	.3417	8.60 mm	.3386
7/16	20	.3888	.3970	10.00 mm	.3937
1/2	20	.4513	.4591	11.60 mm	.4567
9/16	18	.5084	.5166	13.00 mm	.5118
5/8	18	.5709	.5788	14.60 mm	.5748
3/4	16	.6892	.6977	17.60 mm	.6929
7/8	14	.8055	.8152	13/16"	.8125
1	12	.9189	.9289	59/64"	.9219

	Metric			Metric Fine		
		Drill Size			Drill Size	
Ø	Pitch	mm	in	Pitch	mm	in
M1.6	0.35	1.25	3/64	–	–	–
M1.8	0.35	1.45	0.059 (#53)	–	–	–
M2	0.4	1.60	0.063 (#52)	–	–	–
M2.2	0.45	1.75	0.070 (#50)	–	–	–
M2.5	0.45	2.05	0.081 (#46)	–	–	–
M3	0.5	2.50	0.099 (#39)	–	–	–
M3.5	0.6	2.90	0.116 (#32)	–	–	–
M4	0.7	3.30	0.128 (#30)	–	–	–
M4.5	0.75	3.70	0.147 (#26)	–	–	–
M5	0.8	4.20	0.166 (#19)	–	–	–
M6	1	5.00	0.199 (#8)	–	–	–
M7	1	6.00	0.238 (B)	–	–	–
M8	1.25	6.80	0.266 (H)	1	7.00	0.277 (J)
M10	1.5	8.50	0.339 (R)	1.25	8.80	11/32
M12	1.75	10.20	0.404 (Y)	1.50	10.50	0.413 (Z)
M14	2	12.00	15/32	1.50	12.50	1/2
M16	2	14.00	9/16	1.5	14.50	37/64
M18	2.5	15.50	39/64	1.5	16.50	21/32
M20	2.5	17.50	11/16	1.5	18.50	47/64
M22	2.5	19.50	49/64	1.5	20.50	13/16
M24	3	21.00	53/64	2	22.00	7/8
M27	3	24.00	61/64	2	25.00	63/64
M30	3.5	26.50	1-3/64	2	28.00	1-7/64
M33	3.5	29.50	1-11/64	2	31.00	1-7/32
M36	4	32.00	1-17/64	3	33.00	1-19/64
M39	4	35.00	1-25/64	3	36.00	1-27/64

	Metric		Metric Fine	
Ø	Pitch	Drill Size (mm)	Pitch	Drill Size (mm)
M2	0.4	1.65	–	–
M2.5	0.45	2.10	–	–
M3	0.5	2.60	–	–
M3.5	0.6	3.00	–	–
M4	0.7	3.40	–	–
M5	0.8	4.30	–	–
M6	1	5.10	–	–
M7	1	6.10	–	–
M8	1.25	6.90	1	7.10
M10	1.5	8.70	1.25	8.90
M12	1.75	10.50	1.25	10.90
M14	–	–	1.5	12.60
M16	2	14.30	1.5	14.60
M18	–	–	1.5	16.60
M20	–	–	1.5	18.60

	NPT & NPTF		NPS & NPSF	
Pipe Tap Size	Pitch (TPI)	Drill Size* (in)	Pitch (TPI)	Drill Size (in)
1/16	27	0.242 (C)	–	–
1/8	27	0.332 (Q)	27	0.348 (S)
1/4	18	7/16	18	29/64
3/8	18	9/16	18	19/32
1/2	14	45/64	14	47/64
3/4	14	29/32	14	15/16
1	11-1/2	1-9/64	11-1/2	1-3/16
1-1/4	11-1/2	1-31/64	11-1/2	1-33/64
1-1/2	11-1/2	1-23/32	11-1/2	1-3/4
2	11-1/2	2-7/32	11-1/2	2-7/32
2-1/2	8	2-5/8	8	2-21/32
3	8	3-1/4	8	3-9/32
3-1/2	8	3-3/4	8	3-25/32
4	8	4-1/4	8	4-9/32

*Without taper reamer.

	BSW		BSPP		BSPT	
Pipe Tap Size	Pitch (TPI)	Drill Size (in)	Pitch (TPI)	Drill Size (in)	Pitch (TPI)	Drill Size* (in)
1/16	60	0.046 (#56)	28	0.261 (G)	–	–
1/8	40	0.098 (#40)	28	11/32	28	21/64
1/4	20	0.196 (#9)	19	29/64	19	7/16
5/16	18	1/4	–	–	–	–
3/8	16	5/16	19	19/32	19	37/64
7/16	14	23/64	–	–	–	–
1/2	12	0.413 (Z)	14	3/4	14	23/32
9/16	12	17/32	–	–	–	–
5/8	11	35/64	14	53/64	–	–
3/4	10	41/64	14	31/32	14	15/16
7/8	9	3/4	14	1-7/64	–	–
1	8	55/64	11	1-13/64	11	1-11/64
1-1/4	7	1-3/32	11	1-35/64	11	1-33/64
1-1/2	6	1-5/16	11	1-25/32	11	1-3/4
1-3/4	5	1-17/32	11	2	11	–
2	4.5	1-3/4	11	2-1/4	11	2-3/16

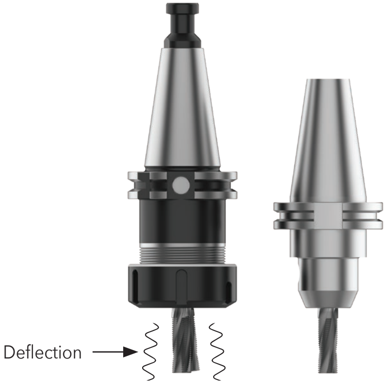
*Without taper reamer.

	ACME		
Nominal Ø	Pitch (TPI)	Min (in)	Max (in)
1/4	16	.1875	.1925
5/16	14	.2411	.2461
3/8	12	.2917	.2967
7/16	12	.3542	.3592
1/2	10	.4000	.4050
5/8	8	.5000	.5062
3/4	6	.5833	.5916
7/8	6	.7083	.7166
1	5	.8000	.8100
1-1/8	5	.9250	.9350
1-1/4	5	1.0500	1.0600
1-3/8	4	1.1250	1.1375
1-1/2	4	1.2500	1.2625
1-3/4	4	1.5000	1.5125
2	4	1.7500	1.7625
2-1/4	3	1.9167	1.9334
2-1/2	3	2.1667	2.1834
2-3/4	3	2.4167	2.4334
3	2	2.5000	2.5250
3-1/2	2	3.0000	3.0250
4	2	3.5000	3.5250
4-1/2	2	4.0000	4.0250
5	2	4.5000	4.5250

When applying side pressure on any milling tool, rigidity is important for success. For best results, use tool holders with accurate and secure clamping to reduce tool deflection. Collet chucks should be avoided for applications with side loading pressure.

	Solid Carbide	Indexable
Collet Chuck		
Hydraulic Chuck	●	●
Shrink Fit	●	
Milling Chuck	●	●
End Mill Holder	○	●
Shell Mill Holder		●

● = Recommended ○ = Use with caution ***Blank*** = Not recommended



ISO	Material	Hardness (BHN)	Machinability*	Speed (SFM)
P	**Free-Machining Steel** 1118, 1215, 12L14, etc.	100 - 150	Easy	725
		150 - 200	Easy	550
		200 - 250	Easy	450
	Low-Carbon Steel 1010, 1020, 1025, 1522, 1144	85 - 125	Average	725
		125 - 175	Average	550
		175 - 225	Average	450
		225 - 275	Average	400
	Medium-Carbon Steel 1010, 1040, 1050, 1527, 1140	125 - 175	Average	450
		175 - 225	Average	400
		225 - 275	Average	350
		275 - 325	Average	300
	Alloy Steel 4140, 5140, 8640	125 - 175	Average	450
		175 - 225	Average	400
		225 - 275	Average	350
		275 - 325	Difficult	300
		325 - 375	Difficult	250
	High-Strength Alloy 4340, 4330V, 300M	225 - 300	Average	350
		300 - 350	Difficult	300
		350 - 400	Difficult	250
	Structural Steel A36, A285, A516	100 - 150	Average	450
		150 - 250	Average	400
		250 - 350	Difficult	300

NOTE: Feed rates provided are safe starting recommendations and may be increased to reduce cycle times. Solid carbide thread mills may perform at double or triple these feed recommendations.

Recommended Feed (inch/tooth) by Cutter Diameter							
0.060 to 0.125	0.126 to 0.188	0.189 to 0.250	0.251 to 0.312	0.313 to 0.375	0.376 to 0.500	0.501 to 0.625	0.626 to 0.750
0.0004	0.0005	0.0007	0.0009	0.0010	0.0015	0.0020	0.0025
0.0004	0.0005	0.0007	0.0009	0.0010	0.0015	0.0020	0.0025
0.0004	0.0005	0.0007	0.0009	0.0010	0.0015	0.0020	0.0025
0.0004	0.0005	0.0007	0.0009	0.0010	0.0015	0.0020	0.0025
0.0004	0.0005	0.0007	0.0009	0.0010	0.0015	0.0020	0.0025
0.0004	0.0005	0.0007	0.0009	0.0010	0.0015	0.0020	0.0025
0.0004	0.0005	0.0007	0.0009	0.0010	0.0015	0.0020	0.0025
0.0004	0.0005	0.0006	0.0008	0.0010	0.0013	0.0018	0.0020
0.0004	0.0005	0.0006	0.0008	0.0010	0.0013	0.0018	0.0020
0.0004	0.0005	0.0006	0.0008	0.0010	0.0013	0.0018	0.0020
0.0004	0.0005	0.0006	0.0008	0.0010	0.0013	0.0018	0.0020
0.0004	0.0005	0.0006	0.0008	0.0010	0.0013	0.0018	0.0020
0.0004	0.0005	0.0006	0.0008	0.0010	0.0013	0.0018	0.0020
0.0004	0.0005	0.0006	0.0008	0.0010	0.0013	0.0018	0.0020
0.0004	0.0005	0.0006	0.0008	0.0010	0.0013	0.0018	0.0020
0.0004	0.0005	0.0006	0.0008	0.0010	0.0013	0.0018	0.0020
0.0004	0.0005	0.0006	0.0008	0.0010	0.0013	0.0018	0.0020
0.0004	0.0005	0.0006	0.0008	0.0010	0.0013	0.0018	0.0020
0.0004	0.0005	0.0006	0.0008	0.0010	0.0013	0.0018	0.0020
0.0004	0.0005	0.0007	0.0009	0.0010	0.0015	0.0020	0.0025
0.0004	0.0005	0.0007	0.0009	0.0010	0.0015	0.0020	0.0025
0.0004	0.0005	0.0007	0.0009	0.0010	0.0015	0.0020	0.0025

*Refer to recommended pass chart on page 43 when referencing material machinability.

ISO	 Material	Hardness (BHN)	Machinability*	Speed (SFM)
S	**High-Temp Alloy**	140 - 220	Difficult	100
	Hastelloy B, Inconel 600	220 - 310	Difficult	75
M	**Stainless Steel**	135 - 185	Difficult	425
	303, 416, 420	185 - 275	Difficult	400
	Stainless Steel PH	185 - 275	Difficult	250
	17-4	275 - 325	Difficult	125
	Tool Steel	150 - 200	Difficult	325
	H-13, H21, A-4	200 - 250	Difficult	225
K	**Cast Iron**	120 - 150	Easy	550
	Grey, Ductile, Nodular	150 - 200	Easy	500
		200 - 220	Easy	450
		220 - 260	Average	400
		260 - 320	Average	375
N	**Wrought Aluminum**	30	Easy	1000
	6061 T6	180	Easy	900
	Cast Aluminum** up to 10% silicon	120	Easy	500
	Brass	30 - 125	Easy	1000

NOTE: Feed rates provided are safe starting recommendations and may be increased to reduce cycle times. Solid carbide thread mills may perform at double or triple these feed recommendations.

Recommended Feed (inch/tooth) by Cutter Diameter							
0.060 to 0.125	0.126 to 0.188	0.189 to 0.250	0.251 to 0.312	0.313 to 0.375	0.376 to 0.500	0.501 to 0.625	0.626 to 0.750
0.0003	0.0004	0.0006	0.0008	0.0009	0.0010	0.0012	0.0015
0.0003	0.0004	0.0006	0.0008	0.0009	0.0010	0.0012	0.0015
0.0004	0.0005	0.0006	0.0008	0.0009	0.0010	0.0015	0.0020
0.0004	0.0005	0.0006	0.0008	0.0009	0.0010	0.0015	0.0020
0.0004	0.0005	0.0006	0.0008	0.0009	0.0010	0.0015	0.0020
0.0004	0.0005	0.0006	0.0008	0.0009	0.0010	0.0015	0.0020
0.0004	0.0005	0.0007	0.0008	0.0010	0.0015	0.0020	0.0025
0.0004	0.0005	0.0007	0.0008	0.0010	0.0015	0.0020	0.0025
0.0004	0.0005	0.0007	0.0009	0.0010	0.0015	0.0020	0.0025
0.0004	0.0005	0.0007	0.0009	0.0010	0.0015	0.0020	0.0025
0.0004	0.0005	0.0007	0.0009	0.0010	0.0015	0.0020	0.0025
0.0004	0.0005	0.0007	0.0009	0.0010	0.0015	0.0020	0.0025
0.0004	0.0005	0.0007	0.0009	0.0010	0.0015	0.0020	0.0025
0.0005	0.0006	0.0009	0.0010	0.0015	0.0020	0.0025	0.0030
0.0005	0.0006	0.0009	0.0010	0.0015	0.0020	0.0025	0.0030
0.0005	0.0006	0.0009	0.0010	0.0015	0.0020	0.0025	0.0030
0.0005	0.0006	0.0009	0.0010	0.0015	0.0020	0.0025	0.0030

*Refer to recommended pass chart on page 43 when referencing material machinability.

**Uncoated thread mills are recommended for cast aluminum applications.

ISO	 Material	Hardness (BHN)	Machinability*	Speed (M/min)
P	**Free-Machining Steel** 1118, 1215, 12L14, etc.	100 - 150	Easy	221
		150 - 200	Easy	168
		200 - 250	Easy	137
	Low-Carbon Steel 1010, 1020, 1025, 1522, 1144	85 - 125	Average	221
		125 - 175	Average	168
		175 - 225	Average	137
		225 - 275	Average	122
	Medium-Carbon Steel 1010, 1040, 1050, 1527, 1140	125 - 175	Average	137
		175 - 225	Average	122
		225 - 275	Average	107
		275 - 325	Average	91
	Alloy Steel 4140, 5140, 8640	125 - 175	Average	137
		175 - 225	Average	122
		225 - 275	Average	107
		275 - 325	Difficult	91
		325 - 375	Difficult	76
	High-Strength Alloy 4340, 4330V, 300M	225 - 300	Average	107
		300 - 350	Difficult	91
		350 - 400	Difficult	76
	Structural Steel A36, A285, A516	100 - 150	Average	137
		150 - 250	Average	122
		250 - 350	Difficult	91

NOTE: Feed rates provided are safe starting recommendations and may be increased to reduce cycle times. Solid carbide thread mills may perform at double or triple these feed recommendations.

Recommended Feed (mm/tooth) by Cutter Diameter							
1.50 to 3.18	3.19 to 4.76	4.77 to 6.35	6.36 to 7.94	7.95 to 9.53	9.54 to 12.70	12.71 to 15.88	15.89 to 19.05
0.010	0.013	0.018	0.023	0.025	0.038	0.051	0.064
0.010	0.013	0.018	0.023	0.025	0.038	0.051	0.064
0.010	0.013	0.018	0.023	0.025	0.038	0.051	0.064
0.010	0.013	0.018	0.023	0.025	0.038	0.051	0.064
0.010	0.013	0.018	0.023	0.025	0.038	0.051	0.064
0.010	0.013	0.018	0.023	0.025	0.038	0.051	0.064
0.010	0.013	0.018	0.023	0.025	0.038	0.051	0.064
0.010	0.013	0.015	0.020	0.025	0.038	0.046	0.051
0.010	0.013	0.015	0.020	0.025	0.038	0.046	0.051
0.010	0.013	0.015	0.020	0.025	0.038	0.046	0.051
0.010	0.013	0.015	0.020	0.025	0.038	0.046	0.051
0.010	0.013	0.015	0.020	0.025	0.038	0.046	0.051
0.010	0.013	0.015	0.020	0.025	0.038	0.046	0.051
0.010	0.013	0.015	0.020	0.025	0.038	0.046	0.051
0.010	0.013	0.015	0.020	0.025	0.038	0.046	0.051
0.010	0.013	0.015	0.020	0.025	0.038	0.046	0.051
0.010	0.013	0.015	0.020	0.025	0.038	0.046	0.051
0.010	0.013	0.015	0.020	0.025	0.038	0.046	0.051
0.010	0.013	0.015	0.020	0.025	0.038	0.046	0.051
0.010	0.013	0.018	0.023	0.025	0.038	0.051	0.064
0.010	0.013	0.018	0.023	0.025	0.038	0.051	0.064
0.010	0.013	0.018	0.023	0.025	0.038	0.051	0.064

*Refer to recommended pass chart on page 43 when referencing material machinability.

ISO	 Material	Hardness (BHN)	Machinability*	Speed (M/min)
S	**High-Temp Alloy**	140 - 220	Difficult	30
	Hastelloy B, Inconel 600	220 - 310	Difficult	23
M	**Stainless Steel**	135 - 185	Difficult	130
	303, 416, 420	185 - 275	Difficult	122
	Stainless Steel PH	185 - 275	Difficult	76
	17-4	275 - 325	Difficult	38
	Tool Steel	150 - 200	Difficult	99
	H-13, H21, A-4	200 - 250	Difficult	69
K	**Cast Iron**	120 - 150	Easy	168
	Grey, Ductile, Nodular	150 - 200	Easy	152
		200 - 220	Easy	137
		220 - 260	Average	122
		260 - 320	Average	114
N	**Wrought Aluminum**	30	Easy	305
	6061 T6	180	Easy	274
	Cast Aluminum** up to 10% silicon	120	Easy	152
	Brass	30 - 125	Easy	305

NOTE: Feed rates provided are safe starting recommendations and may be increased to reduce cycle times. Solid carbide thread mills may perform at double or triple these feed recommendations.

Recommended Feed (mm/tooth) by Cutter Diameter							
1.50 to 3.18	3.19 to 4.76	4.77 to 6.35	6.36 to 7.94	7.95 to 9.53	9.54 to 12.70	12.71 to 15.88	15.89 to 19.05
0.008	0.010	0.015	0.020	0.023	0.025	0.030	0.038
0.008	0.010	0.015	0.020	0.023	0.025	0.030	0.038
0.010	0.013	0.015	0.020	0.023	0.025	0.038	0.051
0.010	0.013	0.015	0.020	0.023	0.025	0.038	0.051
0.010	0.013	0.015	0.020	0.023	0.025	0.038	0.051
0.010	0.013	0.015	0.020	0.023	0.025	0.038	0.051
0.010	0.013	0.018	0.023	0.025	0.038	0.051	0.064
0.010	0.013	0.018	0.023	0.025	0.038	0.051	0.064
0.010	0.013	0.018	0.023	0.025	0.038	0.051	0.064
0.010	0.013	0.018	0.023	0.025	0.038	0.051	0.064
0.010	0.013	0.018	0.023	0.025	0.038	0.051	0.064
0.010	0.013	0.018	0.023	0.025	0.038	0.051	0.064
0.010	0.013	0.018	0.023	0.025	0.038	0.051	0.064
0.013	0.015	0.023	0.025	0.038	0.051	0.064	0.076
0.013	0.015	0.023	0.025	0.038	0.051	0.064	0.076
0.013	0.015	0.023	0.025	0.038	0.051	0.064	0.076
0.013	0.015	0.023	0.025	0.038	0.051	0.064	0.076

*Refer to recommended pass chart on page 43 when referencing material machinability.

**Uncoated thread mills are recommended for cast aluminum applications.

ISO	 Material	Hardness (BHN)	Speed (M/min)
P	**Free-Machining Steel**	100 - 150	375
	1118, 1215, 12L14, etc.	150 - 200	275
		200 - 250	225
	Low-Carbon Steel	85 - 125	375
	1010, 1020, 1025,	125 - 175	275
	1522, 1144, etc.	175 - 225	225
		225 - 275	200
	Medium-Carbon Steel	125 - 175	225
	1010, 1040, 1050,	175 - 225	200
	1527, 1140, 1151, etc.	225 - 275	175
		275 - 325	150
	Alloy Steel	125 - 175	225
	4140, 5140, 8640, etc.	175 - 225	200
		225 - 275	175
		275 - 325	150
		325 - 375	125
	High-Strength Alloy	225 - 300	175
	4340, 4330V, 300M, etc.	300 - 350	150
		350 - 400	125
	Structural Steel	100 - 150	225
	A36, A285, A516, etc.	150 - 250	200
		250 - 350	150
	Tool Steel	150 - 200	175
	H-13, H-21, A-4, 0-2, S-3, etc.	200 - 250	125

Chipload per Tooth (IPT) by Cutter Diameter						
0.055 to 0.125	0.126 to 0.188	0.189 to 0.250	0.251 to 0.312	0.313 to 0.375	0.376 to 0.500	0.501 to 0.750
0.0008	0.0010	0.0014	0.0018	0.0020	0.0030	0.0035
0.0008	0.0010	0.0014	0.0018	0.0020	0.0030	0.0035
0.0008	0.0010	0.0014	0.0018	0.0020	0.0030	0.0035
0.0008	0.0010	0.0014	0.0018	0.0020	0.0030	0.0035
0.0008	0.0010	0.0014	0.0018	0.0020	0.0030	0.0035
0.0008	0.0010	0.0014	0.0018	0.0020	0.0030	0.0035
0.0008	0.0010	0.0014	0.0018	0.0020	0.0030	0.0035
0.0008	0.0010	0.0012	0.0016	0.0020	0.0026	0.0031
0.0008	0.0010	0.0012	0.0016	0.0020	0.0026	0.0031
0.0008	0.0010	0.0012	0.0016	0.0020	0.0026	0.0031
0.0008	0.0010	0.0012	0.0016	0.0020	0.0026	0.0031
0.0008	0.0010	0.0012	0.0016	0.0020	0.0026	0.0031
0.0008	0.0010	0.0012	0.0016	0.0020	0.0026	0.0031
0.0008	0.0010	0.0012	0.0016	0.0020	0.0026	0.0031
0.0008	0.0010	0.0012	0.0016	0.0020	0.0026	0.0031
0.0008	0.0010	0.0012	0.0016	0.0020	0.0026	0.0031
0.0008	0.0010	0.0012	0.0016	0.0020	0.0026	0.0031
0.0008	0.0010	0.0012	0.0016	0.0020	0.0026	0.0031
0.0008	0.0010	0.0012	0.0016	0.0020	0.0026	0.0031
0.0008	0.0010	0.0014	0.0018	0.0020	0.0030	0.0035
0.0008	0.0010	0.0014	0.0018	0.0020	0.0030	0.0035
0.0008	0.0010	0.0014	0.0018	0.0020	0.0030	0.0035
0.0008	0.0010	0.0012	0.0016	0.0020	0.0026	0.0031
0.0008	0.0010	0.0012	0.0016	0.0020	0.0026	0.0031

AccuThread® T3 Speeds and Feeds | Imperial

ISO	Material	Hardness (BHN)	Speed (SFM)
S	**High-Temp Alloy**	140 - 220	100
S	Hastelloy B, Inconel 600	220 - 310	75
S	**Titanium Alloy**	140 - 220	100
S		220 - 310	75
S	**Aerospace Alloy**	185 - 275	100
S	S82	275 - 350	75
M	**Stainless Steel**	185 - 275	225
M	416, 420, etc.	275 - 350	200
M	**Stainless Steel 300 Series**	135 - 185	125
M	304, 316, 17-4, etc.	185 - 275	75
M	**Super Duplex Stainless Steel**	135 - 185	125
M		185 - 275	75

Chipload per Tooth (IPT) by Cutter Diameter						
0.055 to 0.125	0.126 to 0.188	0.189 to 0.250	0.251 to 0.312	0.313 to 0.375	0.376 to 0.500	0.501 to 0.750
0.0006	0.0008	0.0012	0.0016	0.0018	0.0020	0.0025
0.0006	0.0008	0.0012	0.0016	0.0018	0.0020	0.0025
0.0006	0.0008	0.0012	0.0016	0.0018	0.0020	0.0025
0.0006	0.0008	0.0012	0.0016	0.0018	0.0020	0.0025
0.0006	0.0008	0.0012	0.0016	0.0018	0.0020	0.0025
0.0006	0.0008	0.0012	0.0016	0.0018	0.0020	0.0025
0.0008	0.0010	0.0012	0.0016	0.0018	0.0020	0.0025
0.0008	0.0010	0.0012	0.0016	0.0018	0.0020	0.0025
0.0008	0.0010	0.0012	0.0016	0.0018	0.0020	0.0025
0.0008	0.0010	0.0012	0.0016	0.0018	0.0020	0.0025
0.0006	0.0008	0.0012	0.0016	0.0018	0.0020	0.0025
0.0006	0.0008	0.0012	0.0016	0.0018	0.0020	0.0025

ISO	 Material	Hardness (BHN)	Speed (SFM)
H	**Hardened Steels**	140 - 220	100
		220 - 310	75
K	**Cast Iron** Grey, Ductile, Nodular	185 - 275	225
		275 - 350	200
		135 - 185	125
		185 - 275	75
		135 - 185	125
N	**Wrought Aluminum**	30	500
		180	450
	Cast Aluminum	30 - 180	250
	Brass	30 - 100	500

Chipload per Tooth (IPT) by Cutter Diameter						
0.055 to 0.125	0.126 to 0.188	0.189 to 0.250	0.251 to 0.312	0.313 to 0.375	0.376 to 0.500	0.501 to 0.750
0.0006	0.0008	0.0012	0.0016	0.0018	0.0020	0.0025
0.0006	0.0008	0.0012	0.0016	0.0018	0.0020	0.0025
0.0008	0.0010	0.0014	0.0018	0.0020	0.0030	0.0035
0.0008	0.0010	0.0014	0.0018	0.0020	0.0030	0.0035
0.0008	0.0010	0.0014	0.0018	0.0020	0.0030	0.0035
0.0008	0.0010	0.0014	0.0018	0.0020	0.0030	0.0035
0.0008	0.0010	0.0014	0.0018	0.0020	0.0030	0.0035
0.0010	0.0012	0.0018	0.0020	0.0030	0.0040	0.0048
0.0010	0.0012	0.0018	0.0020	0.0030	0.0040	0.0048
0.0010	0.0012	0.0018	0.0020	0.0030	0.0040	0.0048
0.0010	0.0012	0.0018	0.0020	0.0030	0.0040	0.0048

ISO	Material	Hardness (BHN)	Speed (M/min)
P	**Free-Machining Steel**	100 - 150	115
	1118, 1215, 12L14, etc.	150 - 200	85
		200 - 250	70
	Low-Carbon Steel	85 - 125	115
	1010, 1020, 1025,	125 - 175	85
	1522, 1144, etc.	175 - 225	70
		225 - 275	60
	Medium-Carbon Steel	125 - 175	70
	1010, 1040, 1050,	175 - 225	60
	1527, 1140, 1151, etc.	225 - 275	50
		275 - 325	45
	Alloy Steel	125 - 175	70
	4140, 5140, 8640, etc.	175 - 225	60
		225 - 275	50
		275 - 325	45
		325 - 375	38
	High-Strength Alloy	225 - 300	50
	4340, 4330V, 300M, etc.	300 - 350	45
		350 - 400	38
	Structural Steel	100 - 150	70
	A36, A285, A516, etc.	150 - 250	60
		250 - 350	45
	Tool Steel	150 - 200	50
	H-13, H-21, A-4, 0-2, S-3, etc.	200 - 250	38

Chipload per Tooth (mm/tooth) by Cutter Diameter						
1.40 to 3.17	3.18 to 4.77	4.78 to 6.35	6.36 to 7.92	7.93 to 9.52	9.53 to 12.70	12.71 to 19.05
0.020	0.025	0.035	0.045	0.050	0.075	0.090
0.020	0.025	0.035	0.045	0.050	0.075	0.090
0.020	0.025	0.035	0.045	0.050	0.075	0.090
0.020	0.025	0.035	0.045	0.050	0.075	0.090
0.020	0.025	0.035	0.045	0.050	0.075	0.090
0.020	0.025	0.035	0.045	0.050	0.075	0.090
0.020	0.025	0.035	0.045	0.050	0.075	0.090
0.020	0.025	0.030	0.040	0.050	0.065	0.080
0.020	0.025	0.030	0.040	0.050	0.065	0.080
0.020	0.025	0.030	0.040	0.050	0.065	0.080
0.020	0.025	0.030	0.040	0.050	0.065	0.080
0.020	0.025	0.030	0.040	0.050	0.065	0.080
0.020	0.025	0.030	0.040	0.050	0.065	0.080
0.020	0.025	0.030	0.040	0.050	0.065	0.080
0.020	0.025	0.030	0.040	0.050	0.065	0.080
0.020	0.025	0.030	0.040	0.050	0.065	0.080
0.020	0.025	0.030	0.040	0.050	0.065	0.080
0.020	0.025	0.030	0.040	0.050	0.065	0.080
0.020	0.025	0.030	0.040	0.050	0.065	0.080
0.020	0.025	0.035	0.045	0.050	0.075	0.090
0.020	0.025	0.035	0.045	0.050	0.075	0.090
0.020	0.025	0.035	0.045	0.050	0.075	0.090
0.020	0.025	0.030	0.040	0.050	0.065	0.080
0.020	0.025	0.030	0.040	0.050	0.065	0.080

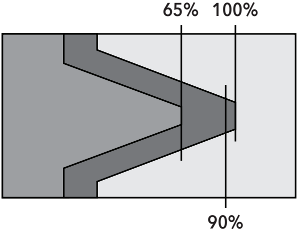
ISO	Material	Hardness (BHN)	Speed (M/min)
S	**High-Temp Alloy**	140 - 220	30
	Hastelloy B, Inconel 600	220 - 310	23
	Titanium Alloy	140 - 220	30
		220 - 310	23
	Aerospace Alloy	185 - 275	30
	S82	275 - 350	23
M	**Stainless Steel**	185 - 275	70
	416, 420, etc.	275 - 350	60
	Stainless Steel 300 Series	135 - 185	38
	304, 316, 17-4, etc.	185 - 275	23
	Super Duplex Stainless Steel	135 - 185	38
		185 - 275	23

Chipload per Tooth (mm/tooth) by Cutter Diameter						
1.40 to 3.17	3.18 to 4.77	4.78 to 6.35	6.36 to 7.92	7.93 to 9.52	9.53 to 12.70	12.71 to 19.05
0.015	0.020	0.030	0.040	0.045	0.050	0.065
0.015	0.020	0.030	0.040	0.045	0.050	0.065
0.015	0.020	0.030	0.040	0.045	0.050	0.065
0.015	0.020	0.030	0.040	0.045	0.050	0.065
0.015	0.020	0.030	0.040	0.045	0.050	0.065
0.015	0.020	0.030	0.040	0.045	0.050	0.065
0.020	0.025	0.030	0.040	0.045	0.050	0.065
0.020	0.025	0.030	0.040	0.045	0.050	0.065
0.020	0.025	0.030	0.040	0.045	0.050	0.065
0.020	0.025	0.030	0.040	0.045	0.050	0.065
0.015	0.020	0.030	0.040	0.045	0.050	0.065
0.015	0.020	0.030	0.040	0.045	0.050	0.065

ISO	Material	Hardness (BHN)	Speed (M/min)
H	**Hardened Steels**	140 - 220	50
		220 - 310	38
K	**Cast Iron** Grey, Ductile, Nodular	185 - 275	85
		275 - 350	75
		135 - 185	70
		185 - 275	60
		135 - 185	60
N	**Wrought Aluminum**	30	150
		180	135
	Cast Aluminum	30 - 180	75
	Brass	30 - 100	150

Chipload per Tooth (mm/tooth) by Cutter Diameter						
1.40 to 3.17	3.18 to 4.77	4.78 to 6.35	6.36 to 7.92	7.93 to 9.52	9.53 to 12.70	12.71 to 19.05
0.015	0.020	0.030	0.040	0.045	0.050	0.065
0.015	0.020	0.030	0.040	0.045	0.050	0.065
0.020	0.025	0.035	0.045	0.050	0.075	0.090
0.020	0.025	0.035	0.045	0.050	0.075	0.090
0.020	0.025	0.035	0.045	0.050	0.075	0.090
0.020	0.025	0.035	0.045	0.050	0.075	0.090
0.020	0.025	0.035	0.045	0.050	0.075	0.090
0.025	0.030	0.045	0.050	0.075	0.100	0.120
0.025	0.030	0.045	0.050	0.075	0.100	0.120
0.025	0.030	0.045	0.050	0.075	0.100	0.120
0.025	0.030	0.045	0.050	0.075	0.100	0.120

Thread milling is like any other material removal process. Depending on how much material needs to be removed and the material's machinability, multiple machining passes may be required. The coarser the pitch, the more material that needs removed. Use the chart below for the suggested number of radial passes.



The percentage of material removal for each machining pass is determined based off total thread height. Percentage of material removal varies depending on the application, but general starting percentages are:

1 pass	100%			
2 passes	75%	100%		
3 passes	60%	80%	100%	
4 passes	60%	80%	90%	100%

NPT / NPTF / BSPT / API			
Pitch Size	Machinability		
	E	A	D
28	1	1	2
27	1	1	2
19	1	1	2
18	1	1	2
14	1	2	3
11.5	1	2	3
11	1	2	3
10	1	2	3
8	2	3	4

ISO			
Pitch Size	Machinability		
	E	A	D
0.40	1	1	2
0.45	1	1	2
0.50	1	1	2
0.70	1	1	2
0.75	1	1	2
0.80	1	1	2
1.00	1	1	2
1.25	1	2	3
1.50	1	2	3
1.75	1	2	3
2.00	1	2	3
2.50	2	3	4
3.00	2	3	4
3.50	2	3	4
4.00	2	3	4
4.50	2	3	4
5.00	2	3	4
6.00	2	3	4

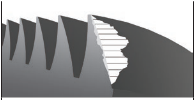
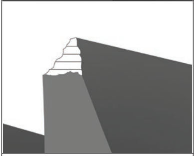
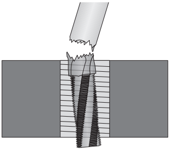
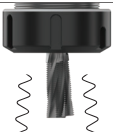
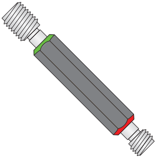
UN / UNJ / BSPP BSW / NPS / NPSF			
Pitch Size	Machinability		
	E	A	D
64	1	1	2
56	1	1	2
48	1	1	2
44	1	1	2
40	1	1	2
36	1	1	2
32	1	1	2
28	1	1	2
27	1	1	2
24	1	1	2
20	1	2	3
19	1	2	3
18	1	2	3
16	1	2	3
14	1	2	3
13	1	2	3
12	1	2	3
11.5	2	2	4
11	2	2	4
10	2	3	4
9	2	3	4
8	2	3	4
7	2	3	4
6	2	3	4

1 Pass
2 Passes
3 Passes
4 Passes

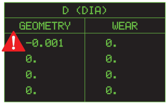
Machinability codes:

E = Easy   A = Average   D = Difficult

Cutting Tool Problems

	**Excessive Wear** • Decrease speed. • Use preferred tool holder (see page 33).
	**Chipping Cutting Edges** • Reduce feed rate. • Add additional radial pass. • Reduce tool overhang. • Use preferred tool holder (see page 33).
	**Immediate Tool Failure** • Check tool selection (reduce cutter diameter). • Reduce feed rate. • Use arc on programming method. • Check cutter compensation D value in program and registrar value. • Check program variables (use Insta-Code® for program).
	**Excessive Chatter** • Reduce tool overhang. • Reduce feed rate. • Add additional radial pass. • Use preferred tool holder (see page 33).
	**Go and No-Go Gage Fit** • Reduce feed rate. • Add additional radial pass. • Reduce tool overhang. • Use preferred tool holder (see page 33). • Tool is worn, replace tool.

Machine Controller Problems

	**Spindle Alarm** ● Check RPM maximum for machine.
	**Negative Cutter Compensation** ● If machine does not accept negative cutter compensation, change compensation direction and use positive offset in registrar.
	**Alarm Regarding Tool Path** ● Remove tool diameter from controller's tool page. Most thread mill programs are based off center of tool, and diameter is not necessary for controller.

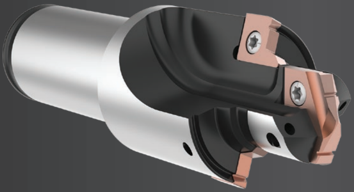
***Watch** this video for more details on threading!*

Notes

Notes

WE HAVE A KIT FOR THAT

Kits aren't for everyone, but if you work on different projects from day to day, you need to be prepared for the work tomorrow will bring.



4 IN 1

One Tool,
Four Operations

PORT KITS

- *One* AccuPort 432® body.
- *Two* T-A® inserts & *two* port contour inserts.
- *One* AccuThread® 856 solid carbide thread mill.

Allied Machine & Engineering
120 Deeds Drive
Dover, Ohio 44622
United States
330.343.4283

Allied Machine & Engineering Co. (Europe) Ltd.
93 Vantage Point
Pensnett Estate
Kingswinford
West Midlands
DY6 7FR England
+44 (0) 1384 400900

Wohlhaupter® GmbH
Maybachstrasse 4
Postfach 1264
72636 Frickenhausen
Germany
011.49.7022.408.0

Wohlhaupter® India Pvt. Ltd.
B-23, 3rd Floor
B Block Community Centre
Janakpuri, New Delhi - 110058
India
+91.11.41827044

ALLIED MACHINE & ENGINEERING

www.alliedmachine.com

Allied Machine & Engineering is registered by DQS to **ISO 9001:2015**

Literature Order Number: **TMPG**
Print Date: August 2024