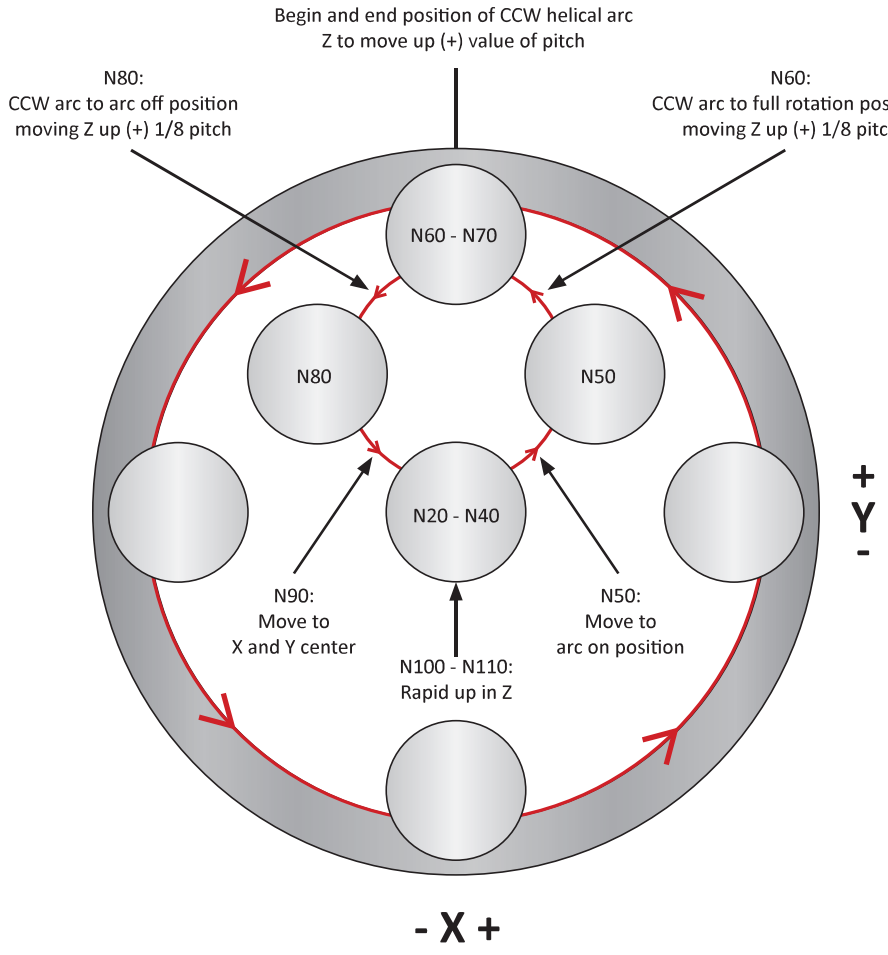
Thread Mill Programming Guide

What you need to know
- Thread milling can be easily accomplished with simple G code programming
- If your machine is capable of 3 axis (helical) interpolation, you can and **should** be thread milling
- Basic programming of a one pass thread mill can be achieved in 6 basic steps

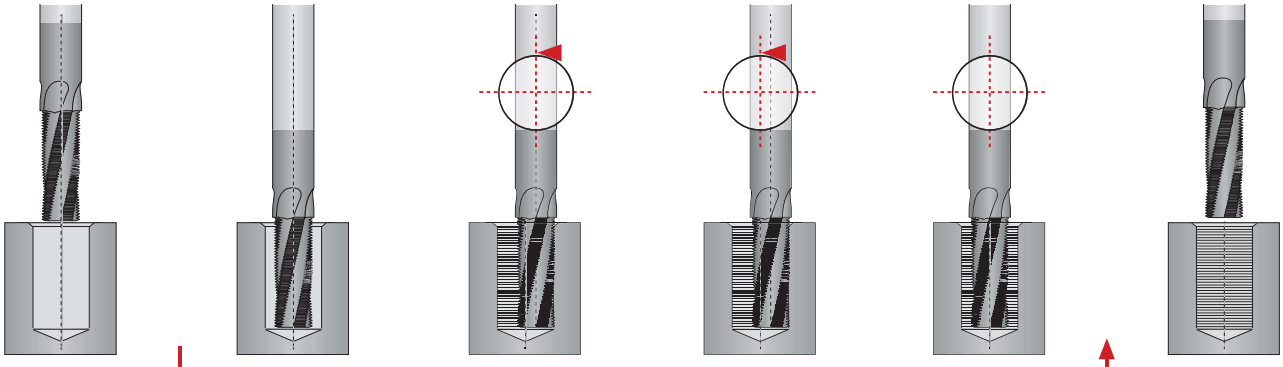
The following are examples of how to calculate and program a 7/16-20 right hand thread that will be 1/2 deep produced in one pass

Major thread diameter	**0.4375**	Major diameter of thread (7/16 = 0.4375)
Threads per inch	**20**	Number of threads per inch (20 is from 7/16-20 designation)
Length of thread	**0.5**	Desired length of cut
SFM	**475**	Recommended surface footage for material to be cut
Feed per flute	**0.0025**	Recommended feed rate per cutting edge
Number of flutes	**4**	Number of flutes on tool to be used
Tool diameter	**0.335**	Diameter of cutting tool
Using the information above, the values can be calculated:		
Pitch	**0.05**	= 1 / thread per inch
RPM	**5416**	(SFM • 3.82) / Tool diameter
Linear feed	**54.16**	RPM • Feed per flute • Number of flutes
Feed rate for thread milling	**12.69**	Linear feed • ((Major thread diameter - Tool diameter) / Major thread diameter)
Z axis move on arc on	**0.0063**	(Pitch / 8)
Z axis move for full thread	**0.5063**	(Pitch / 8) + Length of cut
Arc on/off	**0.0256**	(Major thread diameter - Tool diameter) / 4
Full rotation value	**0.05125**	(Major thread diameter - Tool diameter) / 2

Major thread diameter	0.4375	**Feed rate for thread milling**	12.69	**Arc on/off value**	0.0256
Cutter diameter	0.335	**Z axis depth for full thread**	0.5063	**Full rotation value**	0.05125
Length of thread	0.5	**Z axis for arc on/off**	0.0063	**Pitch value**	0.05



	N10	S		5416	M03			
1			Absolute position in rapid to center of hole in X and Y, then rapid to Z0 (level with surface of hole)(assumed to be X0, Y0, Z0 for demonstration purposes). To be done by customer.					
	N20	G90	G00	X 0.0000	Y 0.0000			
	N30					Z 0.0000		
2	N 40	G91	Switch to incremental positioning and high feed to Z axis depth for full thread.					
			G01			Z -0.5063		F 50.0
	N50	G41	Activate left diameter compensation for tool number 1 (needs to be whatever tool number you are using) and feed to arc on/off position at 1/4 feed rate for thread milling.					
			G01	X 0.0256	Y 0.0256		D1	F 3.17
3			CCW arc from full rotation from the arc on position at the calculated thread milling feed rate moving Z up (+) 1/8 pitch value (Z axis move up for arc on/off). X and Y positions are the incremental distance from where tool is to where it will be after arc (arc on/off value). 1 is the incremental X value of center of rotation from where tool currently is arc on/off value *-1. J is the incremental Y value from current tool position to center of rotation.					
	N60		G03	X -0.0256	Y 0.0256	Z 0.0063	I -0.0256	J 0.0000 F 12.69
4	N70		One complete CCW arc from the full arc rotation position at the calculated thread milling feed rate moving Z up (positive pitch value). I and J values are calculated same as above. I will be 0.0 and J will be full rotation value *-1.					
			G03	X 0.0000	Y 0.0000	Z 0.0500	I 0.0000	J -0.0513 F 12.69
	N80		CCw arc from full rotation diameter to arc off position at double the calculated thread milling feed rate moving Z up (+) 1/8 pitch value (Z axis move up for arc on/off). I and J values are calculated same as above.					
			G03	X -0.0256	Y -0.0256	Z 0.0063	I 0.0000	J -0.0256 F 25.38
5			Shut off cutter comp and move from arc off position to center of hole in X (arc on/off value -1) and Y (arc on/off value *-1) at high feed rate.					
	N90	G40	G01	X 0.0256	Y -0.0256			
			Rapid Z up incremental value (length of thread - all Z values in G03 arc commands).					
	N100		G00			Z 0.4438		
6			Switch back to absolute positioning and rapid to a safe point in Z above part level (assumed to be 1 above part level for demonstration purposes).					
	N110	G90	G00				Z 1.0000	



Step 1 N10 - N30	Step 2 N40	Step 3 N50 - N60	Step 4 N70	Step 5 N80 - N90	Step 6 N100 - N110
• Preparatory commands • Positioning above hole center and at hole level in Z • In absolute position mode	• Change to incremental • Feed to bottom of hole • Z axis depth for full thread	• Activate left cutter comp • Feed to arc on position • Arc to full rotation value while moving Z up 1/8 pitch • Z axis move for arc on	• One complete CCW rotation at full arc rotation value while moving Z up 1 pitch value	• CCW arc from full rotation value to the arc on/off value while moving Z up 1/8 pitch (Z axis move for arc off)	• Rapid up in Z

Thread Mill Troubleshooting Guide

Causes		Problem										
		Thread mill is showing accelerated or excessive wear	Cutting edges are chipping	Thread mill is breaking in the first hole of part	Thread mill is creating excessive chatter	Out of round thread is produced	Bell mouthed thread form (small at bottom, big at top)	Part rejection because of rough flank finish	Steps in thread profile	Gauge difference from part to part	Machine not making correct paths to create thread profile	Control not accepting the program
Catalog	Incorrect tool selection			1	1							
	Incorrect speed and feed selection	2, 3	2, 3		2, 3			2, 3				
Speed and Feed	RPM too high	5										
	RPM too low				4		4	4				
	Machine tool specifications restrict RPMs			5, 19								
	Feed rate too high		7	7			7	7	7			
	Feed rate too low	6										
	Incorrect adjusted feed rate adjustment ratio			12								
	Machine tool specification restricts feed rate					7, 19						
	Ramp-in is programmed as an axial move			20					20			
Tool	Thread mill moved or slipped in its holding device	13	13	13	13			13	13			
	Tool is sticking out of the holder too far	15	15	15	15			15	15	15		
	Runout between thread mill and holder				10			10				
	Incorrect coating creating built up edge	8, 17								8, 17		
	Helix angle too low				9			9				
	Excessive thread mill wear								11	11		
	Excessive tool pressure	7, 11, 14					7, 11, 14					
Machine	Workpiece moving in its fixturing	16	16	16	16			16		16		
	Insufficient coolant pressure or flow	17	17									
	Lack of machine rigidity	16	16		16		16	16				
Programming	Incorrect number of passes			22			22					
	Incorrect program variables			18, 26							18, 26	
	Did not account for X/Y radial moves for tapered threads										24, 26	
	Incorrect cutter compensation variables			23, 26								23, 26
	Helical interpolation option not on machine or turned off										21, 26	21, 26
	Machine tool control is not formatted to standard EIA/ASCII/ISO Code											25, 26

Troubleshooting Solutions

1. Refer to catalog to ensure proper tool selection.
2. Verify the correct speed was selected from the catalog speed and feed chart.
3. Verify the correct feed rate was selected from the catalog speed and feed chart.
4. Increase the spindle speed (RPM).
5. Decrease the spindle speed (RPM).
6. Increase feed per tooth.
7. Decrease feed per tooth.
8. Investigate other coatings.
9. Increase the tool helix.
10. Gauge runout between thread mill and tool holder.
11. Perform tool change at quicker intervals.
12. Adjust the feed rate ratio properly to the correct actual penetration rate for internal threads. Refer to speed and feed pages for formula.
13. Use hydraulic clamping chuck.
14. Check the tool for excessive wear. Beginning threads will wear the fastest.
15. Make the amount of overhang in the holding device as short as possible.
16. Verify the workpiece is properly clamped. Re-tighten or increase stability if necessary.
17. Increase the coolant flow and volume.
18. Check the milling program variables, especially the positive or negative value associated with I and J values.
19. Make sure the machine has the appropriate axis and path speed capabilities.
20. Make sure the thread mill is arcing in the major diameter instead of making a radial move.
21. Make sure the machine tool has a helical interpolation option that is on.
22. Increase the number of thread mill passes.
23. Make sure the cutter compensation variables are input into the G41 program line.
24. Adjust the program for pipe tap threads to taper out on diameter in X/Y directions to create proper form.
25. Request information from the machine tool builder regarding its programming formats.
26. Scan and email a copy of your program to the Application Engineering department at **appeng@alliedmachine.com**.