Deluxe Thread Milling CNC Code Generator for machining centers

The thread milling program for CNC machining centers features an incredible array of highly customizable programming parameters for threading machined parts. This highly configurable program **instantly** produces NC code to thread mill most any thread specification using multi-tooth or single-point style threading tooling.

CNC Machining Center Thread Milling Program Features

The thread milling program for CNC machining centers features a wide range of programmable variables, including:

- Designed for most Fanuc, Okuma and Siemens CNC controls
- Thread milling of internal or external threads
- Thread milling of right or left hand threads
- Thread milling using full form multi-tooth cutter in 1 full helical circle
- Thread milling using single point tool in continuous full helical circles (or full spiral circles for tapered threads)
- Single point threading allows for 1 tool to cut many different pitches (Greatly reduces tooling costs!)
- Can cut multiple arc segments per revolution with constant, increasing or decreasing radius per segment
- Emulates spiral interpolation with multiple arc segments per revolution for tapered threads
- Support for standard and non-standard inch and metric, NPT pipe and most other thread forms
- Allows for multiple radial thread passes using constant depth or constant volume thread depths
- Can perform single or multiple clean-up passes
- Supports multiple thread starts
- Thread mills in +Z or -Z direction
- Supports (G41 & G42) tool radius cutter compensation
- Allows for auto positioning to center of hole or set distance from hole wall prior to thread cut
- Calculates feedrate factor from tool centerline to edge of cutter for circular moves
- Has separate user specified feedrates for cutting, XY approach and Z approach (feed or rapid) moves
- Can intermix any of the above combinations to produce any desired thread profile
- Performs various validations of inputted data to ensure accuracy of NC code
- Software instantly re-calculates all data and thread milling program after changing any input variables

Thread Milling G-Code Generator

This deluxe version includes below enhancements:

- Can select thread specs from stored table containing major & minor diameter & thread pitch & depth values
- Selecting a thread spec entry automatically loads relevant parameters for new instantly generated NC code
- Can modify existing thread spec entries or add new entries to (or remove entries from) thread specs table
- More than 90 internal and 90 external thread spec entries come pre-loaded
- Supports option to thread mill several holes or bosses at various XY coordinates
- Calculates cutting time for entire thread milling operation (0h 0m 0s, 0m 0s, 0s and 0.00m formats)
- Offers extremely flexible way to add, modify and/or balance radial thread passes & clean-up passes
- Allows for multiple axial passes when thread depth is greater than the insert height (works w/ NPT also)
- Supports both forward (M03) and reverse (M04) spindle directions
- Support for absolute (G90) program coordinates (in main or sub program)
- Support for incremental (G91) program coordinates (as a sub program)
- Can intermix any of the above combinations to produce any desired thread profile

This deluxe version of the software uses tangential arc approach as the infeed method causing the tool load to build up gradually which minimizes vibration and promotes longer tool life. The thread mill cutter positions to set distance from wall of hole (or center of hole), positions in Z, moves in XY half the distance to hole wall in straight line, arcs remaining distance to full engage of cut, does single (or multiple) full circle(s) while ramping up or down one pitch (or lead if multiple start thread), arcs off of the part, does straight line back to the XY start point and then retracts out of the hole.

The thread milling NC code generator program works with a wide range of thread milling tools, including both multi-tooth and single-tooth thread milling tools. The diverse tooling utilization capabilities allow for different methods and styles of milling, such as highly precise corkscrewing and multiple revolutions for tapered threads.

The thread milling program even inspects and validates inputted data to guarantee the accuracy of the NC code, ensuring optimal functionality and performance for every custom program. This functionality allows for greatly reduced programming times and drastically reduces the potential for human error.
Simply enter all desired thread mill specifications into the appropriate cells. Spreadsheet will instantly re-calculate all data and thread milling program after changing any input variables. To generate (NC code) text file of current output results, press “Write Code to Text File” button and the file (Thread_Mill_Code.txt or any other name) will be created in the user specified folder.

Note: All information and comments regarding holes (internal threading) are implied to also pertain to thread milling bosses (external threading).

General Information

This powerful and extremely flexible thread milling CNC code generator allows you to use this one common thread milling program for tooling from **any** thread mill tooling vendor.

Unlock yourself from using a tooling vendor's thread milling software where, in many cases, you are forced to select from **only** their expensive multi-tooth cutters and inserts. That's why their thread milling software is free! You buy **all** of your $$$ tooling from them!

Maybe you have a single tooth thread mill cutter on hand and want to use it for a rush threading job. You are locked out from using **your existing tooling** with their software. Is their **free software** really worth it? Is their free software doing what you need it to do? This user friendly thread milling software is likely to be less costly than just one of their thread mill cutters you probably didn't need to buy. Unlike some other thread milling programs, this software allows you to input **your cutter diameter** (and all other parameters) so you can use your existing tooling.

This thread milling CNC code generator utility is MS Excel event driven software relying heavily on VBA for Excel.
If a relevant cell (parameter) or combo box (drop-down list box) changes, all appropriate cells are instantly re-calculated including the NC code ready to output to a text file.

You will notice the tool call-up, tool pre-select, fixture offset, coolant codes and whatever else are missing for the beginning of the thread milling program stub. This is due to the many, many variations of all of the these codes. If I included all of above mentioned code it would probably be wrong for your particular machine and you would be editing it anyways. This is the same with handling the end of the tool; there are many variations to send a tool back to it's home position.

Both the regular and premium versions of the thread mill software use tangential arc approach as the infeed method causing the tool load to build up gradually which minimizes vibration and promotes longer tool life. The thread mill cutter positions to set distance from wall of hole (or center of hole), positions in Z, moves in XY half the distance to hole wall in straight line, arcs remaining distance to full engage of cut, does single (or multiple) full circle(s) while ramping up or down one pitch (or lead if multiple start thread), arcs off of the part, does straight line back to the XY start point and then retracts out of the hole.

IMPORTANT NOTE: If you (activate "Freeze Panes") turn on the "header labels" row by double-clicking the left mouse button, you may need to **turn it off** (by double-clicking on most any cell again) if you **want to scroll to the top of the sheet again**.

Many rules and data validation checks take place throughout the thread milling NC code generator program.
Some examples are:

Major Diameter must be greater than Minor Diameter.

For internal threads, Tool Diameter must be less than Minor Diameter.

If changing to a larger thread size, it's easiest to edit the Major Diameter first, then the Minor Diameter and then lastly the Tool Diameter. If changing to a smaller thread size, it's easiest to edit the Tool Diameter first, then the Minor Diameter and then lastly the Major Diameter.

VERY IMPORTANT: For **internal tapered threads**, the “Major Diameter” and "Minor Diameter" should be specified at **large end of taper** (i.e. diameter at top of hole). For **external tapered threads**, the “Major Diameter” and "Minor Diameter" should be specified at **large end of taper** (i.e. diameter at bottom of tapered boss [= thread depth = bottom of threads]).

The “Tool Diameter” should be specified at the **theoretical** crest (top) of the tooth. A large crest radius would otherwise "throw off" the accuracy of the thread diameter. For tapered threads, the “Tool Diameter” should be specified at the theoretical crest of the **1st** tooth (at small end of taper).

For straight threads, the "Gage Length" should be specified at end of tool or you can set “Gage Length” at crest of 1st end tooth for a slightly more accurate thread depth. For tapered threads, you can set “Gage Length” at crest of the 1st tooth (small end of taper) for a slightly more accurate thread depth **and** thread diameter. If setting “Gage Length” at crest of the 1st tooth, then allow for an extra half tooth (1/2 thread pitch) when determining threading depth to bottom of blind hole.

NOTE: For internal tapered threads the software accounts for, based on the thread depth, that you defined the tool gage diameter and length at the **bottom of the tool** yet you defined the hole (major & minor) diameters at the **large end** (**top) of the hole**.
For external tapered threads the software acknowledges that you defined the tool gage diameter and length at the (**bottom) end of the tool** and you defined the boss (major & minor) diameters at the **large end (bottom) of the boss**.

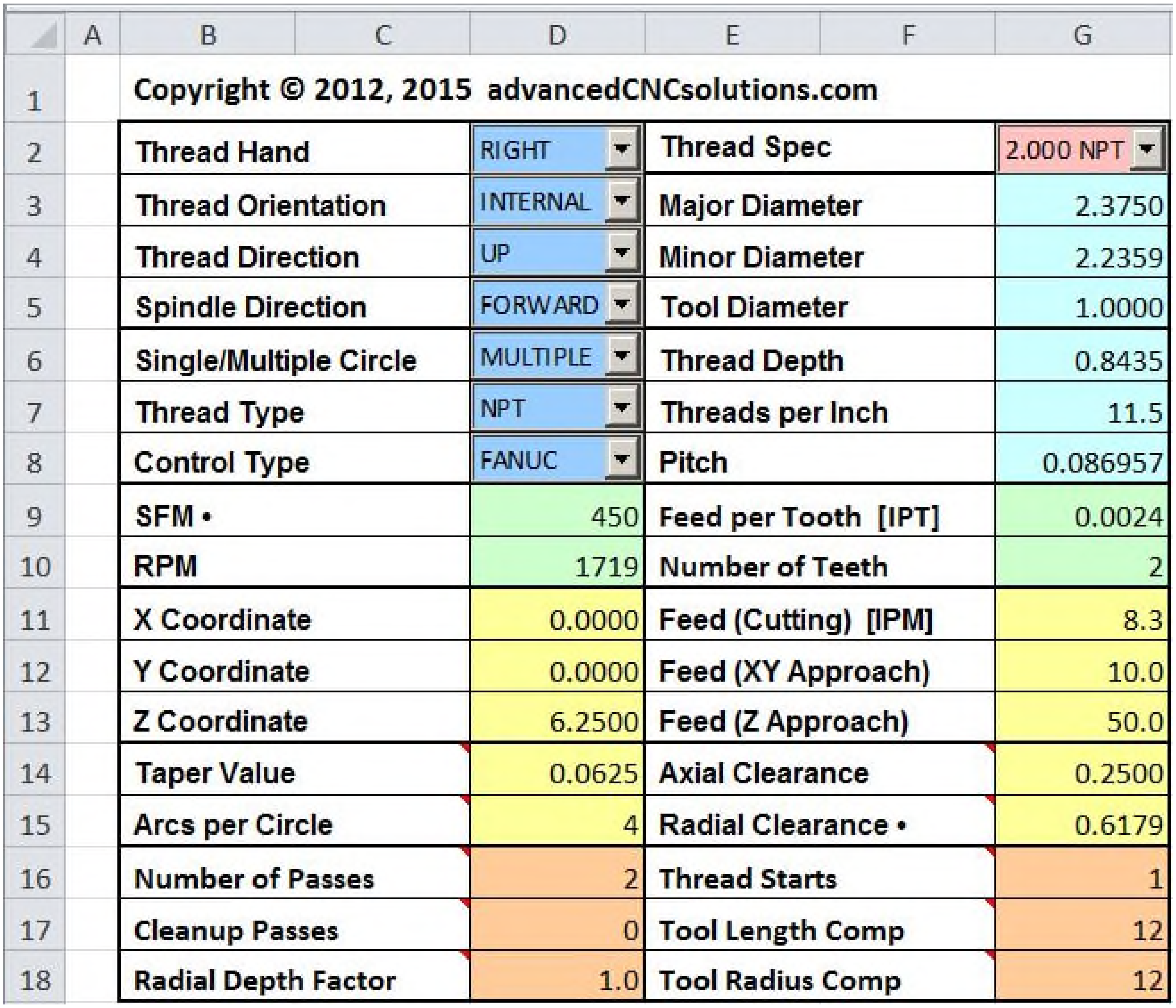
VERY IMPORTANT: When using cutter compensation, **only enter deviation** (from nominal diameter to actual cutter diameter) **into cutter radius compensation offset** for the tool**.**
(Example: if you enter 0.500" as "Tool Diameter" (within this software only) and the cutter in the machine is 0.490" then the cutter radius compensation offset value (within the NC machine control) would be **-.005**)
Since the actual cutter is smaller than the nominal cutter size, the offset value will be negative.
(0.490-.500)/2= **-.005**) (Note: it is divided by 2 because the cutter compensation value is usually a radius value.)

"Taper Ratio" parameter is set to 0.0625, which is 1 degree 47 minutes (on the radius). The taper rate for all NPT threads is 1/16 (0.0625) measured by the change of diameter per length of pipe thread. Other taper values can be used, values between 0 and 0.125 (3 degrees 34 minutes) are allowed. Note: Inverse Tangent (0.0625/2) = 1 degree 47 minutes.

Care must be taken to refer to the manufacturers tooling and/or insert recommendations where needed.

I & J output follows standard convention (signed incremental value from startpoint of arc back to center of arc).

"Main_Code" sheet



	A	B	C	D	E	F	G
1		Copyright © 2012, 2015 advancedCNCsolutions.com					
2		Thread Hand		RIGHT ▾	Thread Spec		2.000 NPT ▾
3		Thread Orientation		INTERNAL ▾	Major Diameter		2.3750
4		Thread Direction		UP ▾	Minor Diameter		2.2359
5		Spindle Direction		FORWARD ▾	Tool Diameter		1.0000
6		Single/Multiple Circle		MULTIPLE ▾	Thread Depth		0.8435
7		Thread Type		NPT ▾	Threads per Inch		11.5
8		Control Type		FANUC ▾	Pitch		0.086957
9		SFM •		450	Feed per Tooth [IPT]		0.0024
10		RPM		1719	Number of Teeth		2
11		X Coordinate		0.0000	Feed (Cutting) [IPM]		8.3
12		Y Coordinate		0.0000	Feed (XY Approach)		10.0
13		Z Coordinate		6.2500	Feed (Z Approach)		50.0
14		Taper Value		0.0625	Axial Clearance		0.2500
15		Arcs per Circle		4	Radial Clearance •		0.6179
16		Number of Passes		2	Thread Starts		1
17		Cleanup Passes		0	Tool Length Comp		12
18		Radial Depth Factor		1.0	Tool Radius Comp		12

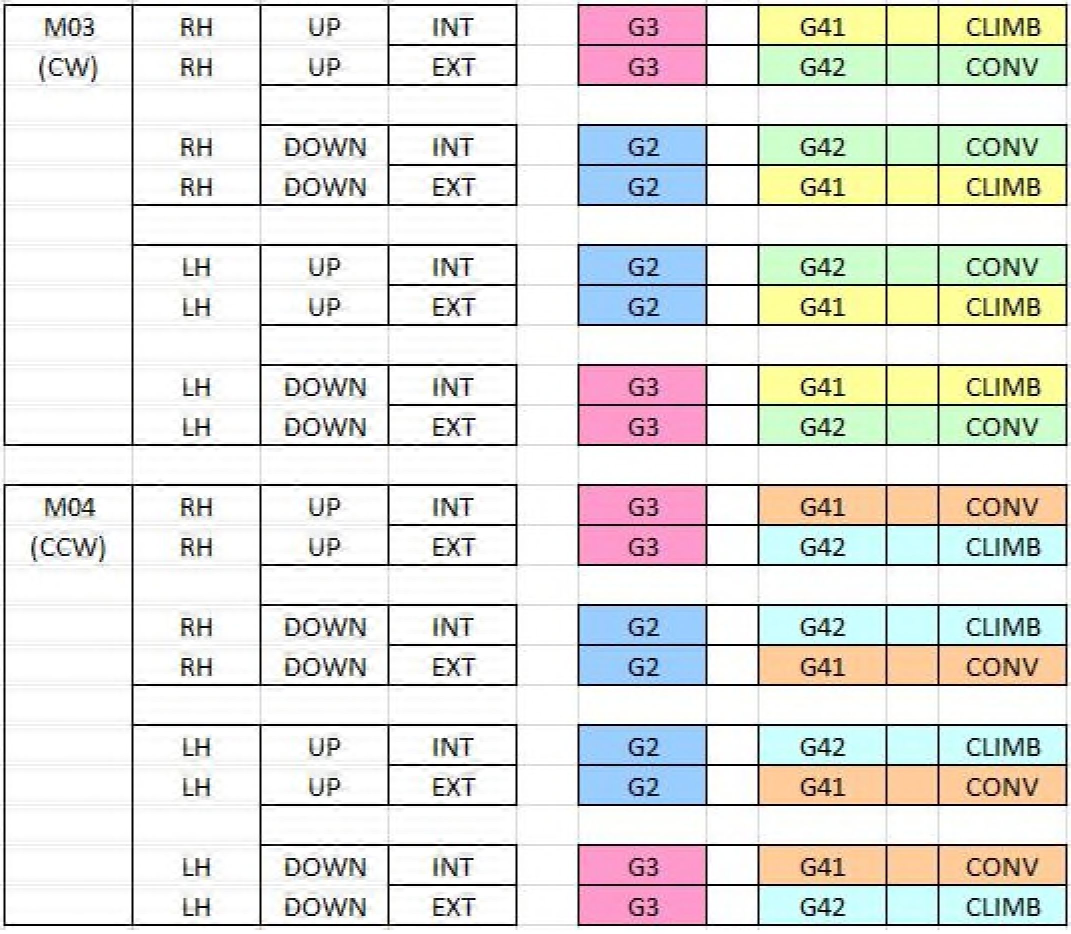
"**Thread Hand**" simply has 2 choices: "**RIGHT**" for right handed (RH) threads and "**LEFT**" for left handed (LH) threads.

"**Thread Orientation**" has 2 choices: "**INTERNAL**" for internal threads and "**EXTERNAL**" for external threads.

"**Thread Direction**" has 2 choices: "**UP**" for Z+ motion and "**DOWN**" for Z**-** motion during thread milling.

"**Spindle Direction**" also has 2 choices: "**FORWARD**" for M3 (CW) and "**REVERSE**" for M4 (CCW) direction.

Based on these 4 above settings, the software **automatically** determines if G2 (CW) or G3 (CCW) circular cutter direction is needed along with G41 (cutter compensation Left) or G42 (cutter compensation Right). These 4 settings also determine if the cutter is Climb milling or Conventional milling. On the "Main_Code" sheet, a cell will indicate if you are currently climb milling (preferred) or conventional milling.



M03 (CW)	RH	UP	INT	G3	G41	CLIMB
	RH	UP	EXT	G3	G42	CONV
	RH	DOWN	INT	G2	G42	CONV
	RH	DOWN	EXT	G2	G41	CLIMB
	LH	UP	INT	G2	G42	CONV
	LH	UP	EXT	G2	G41	CLIMB
	LH	DOWN	INT	G3	G41	CLIMB
	LH	DOWN	EXT	G3	G42	CONV
M04 (CCW)	RH	UP	INT	G3	G41	CONV
	RH	UP	EXT	G3	G42	CLIMB
	RH	DOWN	INT	G2	G42	CLIMB
	RH	DOWN	EXT	G2	G41	CONV
	LH	UP	INT	G2	G42	CLIMB
	LH	UP	EXT	G2	G41	CONV
	LH	DOWN	INT	G3	G41	CONV
	LH	DOWN	EXT	G3	G42	CLIMB

At least one (clever) company makes a thread milling cutter with the first tooth for roughing and the second tooth for finishing on hard materials. With this configuration the internal thread cutter **must proceed down** the hole (roughing tooth first) with several helical circle moves. The cutter makes a right handed thread, but it is designed for M04 (CCW) reverse spindle rotation to produce the preferred climb milling cutting strategy. See 11th entry in above chart: **M04**, **RH**, **DOWN**, **INT**ernal thread = Climb milling. **If** it were a standard (CW) rotation cutter that's thread milling in the required downward direction, the cutter would have been conventional milling the threads. See 3rd row of chart: **M03**, **RH**, **DOWN**, **INT**ernal thread = **Conv**entional milling.

If some company makes "reverse rotation" thread mill cutters, one advantage would be you could internally thread mill left handed internal threads while "Climb" milling (less chatter and a smoother thread finish). The other advantage would be using a +Z direction of cut (less chip re-cutting because the cutter would be climbing up out of the hole away from the chips. See 13th row of above chart.

"**Single/Multiple Circle**" has 4 settings: "**SINGLE**", "**USE AXIAL**", "**MULTIPLE**" and "**MULT FULL**".

If you are using a **multi-tooth thread milling tool**, you should set the "**Single/Multiple Circle**" parameter to "**SINGLE**" or "**USE AXIAL**". This tells the software you want to make a **single** helical complete circle cutter move (with the included arc on and arc off moves).

When set to "**SINGLE**" the cutter will travel **one single full** helical circle from full depth (thread bottom) advancing 1 thread upwards if "Thread Direction" is set to "UP". If "Thread Direction" is set to "DOWN" then the tool will position to 1 thread above bottom and proceed to arc in and then travel one full circle while advancing downwards in the Z axis by 1 thread.

When "**Single/Multiple Circle** " is set to "**USE AXIAL**" (meaning **Use Axial** Thread Pass Table) the cutter will **also** travel one full helical circle (up or down depending on the same "Thread Direction" setting) **for each** axial thread pass (cut) in the table on the "Axial_Passes" sheet. This gives you the ability to axially "divide" up the thread cuts at the depths you choose. This threading method is unique to this deluxe version of the software.

For example let's say your thread depth is 2.000" and your insert height (depth) is only 1.250". This means you would need to take a thread pass at full depth and another pass somewhere just below 1/2 of full thread depth. I say "just below" because you need some thread overlap between the 2 axial thread passes. The "Axial Thread Pass Table" on the "Axial_Passes" sheet is specifically designed for this situation. I just mentioned vaguely "just below" 1/2 way down, but this thread pass must be **exactly** 1 or more pitches (threads) above the full depth pass for them to cut a good thread (no thread mismatch) between the two of them. The table allows you to select any one or more of these exact heights in Z (full pitches). If the thread is "12 pitches" depth (in the example above) then you would select "12" pitches and also select probably "7" pitches causing the cutter to take a pass 12 threads down from the top of hole and another pass at 7 threads down. You can select these passes in any order and can cut them in the "upwards" or "downwards" Z axis direction. The selection of 12 and 7 pitches is what makes up the data in the Axial Thread Pass Table on the "Axial_Passes" sheet. There is more information on multiple axial thread passes further down in the documentation where "Axial_Passes" sheet is discussed.

Note: "**Single/Multiple Circle**" set to "**SINGLE**" is very similar to "**USE AXIAL**" except for "**USE AXIAL**" allows for the user to axially cut overlapping portions of the thread when the thread depth (height) is greater than the multi-tooth insert height.

If you are using a **single point thread milling tool**, you should set the "**Single/Multiple**" parameter to "**MULTIPLE**" or "**MULT FULL**". This tells the software you want to make **multiple** helical circle cutter moves from thread bottom to the thread top (or from thread top to the thread bottom) depending on if the "Thread Direction" setting is "UP" or "DOWN".

If thread milling upwards (Z+ direction), "**MULT FULL**" means the cutter will travel **full** helical circles, from thread bottom to the thread top plus 1 pitch and continue the arc until it completes its last full circle. If thread milling downwards (Z- direction), the cutter will travel **full** helical circles from thread top plus 1 pitch (or lead) or more if needed and then proceed to the thread bottom. In either case, if the Z axis travels higher than the "Axial Clearance" plane (possible with course or multiple start threads) the software will adjust this clearance plane upwards.

Whereas, "**MULTIPLE**" means the cutter will travel **full** helical circles, (if thread milling upwards) from thread bottom to thread top plus 1 pitch (or lead) (and stop **exactly** at that thread height even if it's a partial helical circle). If thread milling downwards (Z- direction), the cutter will travel **full** helical circles from thread top plus 1 pitch (or lead) to thread bottom (and stop **exactly** at thread bottom and then arc off the part with a small but necessary Z- move).

This software also supports multiple helical arc segment moves per full circle, which is the **only proper** way to thread mill tapered threads using a single point style thread mill cutter. It progressively increases (if Z+ direction) the radius of the arc as it advanced to the next arc segment emulating spiral interpolation. The "Arcs per Circle" thread milling technique works with all 4 "**Single/Multiple Circle**" settings ("**SINGLE**", "**MULTIPLE**", "**MULT FULL**" and "**USE AXIAL**"). If you set the "Arcs per Circle" parameter equal to 4, it would produce 4 (90 degree) arc segment moves for each full helical circle. The "Arcs per Circle" parameter has a maximum value of 360 (1 degree) arc segment moves. The "Thread Type" parameter should be set to "**TAPERED**" for the radius to progressively increase (or decrease) if tapered threads are required. With standard threads, the "Arcs per Circle" can be set to 2 or 4 if a machine requires semi-circle or quadrant output.

If you perform tapered thread milling using a single point tool (multiple helical arc segment moves per full circle), you will need to decide which number of "**Arcs per Circle**" works best for you. I am guessing a number between 4 and 10 will work fine. Note: Courser threads will need a higher number than fine threads. With 27 TPI, 4 arc segment moves shows (from arc to arc) a change in radius of 0.0003" and 8 TPI (still with 4 arc segments) shows a change in radius of 0.0010". If this "**Arcs per Circle**" number is set too low you risk imperfections in the thread form which could cause tapered "sealed" threads to leak! You will need to test these combinations of parameter settings to see how this works out.

This "multiple arc moves per full circle" technique saves greatly on tooling costs since a single tooth cutter can machine a tapered thread. You may need to taper ream the hole (internal threads) or taper turn the shaft (external threads) OR use a "Partial Profile" or "topping" style" single tooth insert. With **just a few** "Partial Profile" inserts on hand (in various pitch ranges) you could thread mill **most any** straight or NPT tapered pipe thread. For low volume parts, the tooling cost savings would far outweigh the increased cycle time due to multiple helical passes of a single point tool.

You should also use the multiple "Arcs per Circle" feature (by setting it’s value to more than 1) when cutting tapered threads even with multi-tooth thread mill cutters (with single circle method) for conical interpolation (increasing or decreasing radius) to occur.

All 4 of the above thread milling methods ("**SINGLE**", "**MULTIPLE**", "**MULT FULL**" and "**USE AXIAL**") also work in conjunction with multiple thread starts and/or multiple radial threading passes. There is a "multiple radial threading pass" table on the "Radial_Passes" sheet which allows you with ease to select the values for the multiple radial thread passes. There is more information on multiple radial thread passes further down in this documentation where the "Radial_Passes" sheet is discussed.

All features are designed to work with all other features except for the 2 features "multiple axial passes" ["**Single/Multiple Circle**" = "**USE AXIAL**"] (used for a multi-tooth cutter) AND "multiple helical circle moves" ["**Single/Multiple Circle**" = "**MULTIPLE**" or "**MULT FULL**"] (usually used for a single tooth cutter), which are mutually exclusive.

With this software (when using a multi-tooth cutter) it is possible to thread mill multiple holes (or bosses) using multiple radial passes with multiple axial passes with multiple thread starts with multiple arc segment moves per circle on a **single helical circle** cut up (or down) the straight or tapered (right or left handed) threaded hole (or boss).

Or with this software (when using a single point tool) it is possible to thread mill multiple holes (or bosses) using multiple radial passes with multiple thread starts with multiple arc segment moves per circle on **multiple (or single) helical circle** cuts up (or down) the straight or tapered (right or left handed) threaded hole (or boss).

The order in which the software performs all of these thread milling moves is as follows; first it thread mills all of the arc segment moves (if they exist) and completes one thread top to bottom (or bottom to top), then performs all of the thread starts, then thread mills all of the axial passes, then thread mills all of the radial passes and then lastly thread mills all of the holes (or bosses).

"**Thread Type**" has 2 choices: "**STRAIGHT**" for non-tapered threads and "**TAPERED**" for tapered threads such as NPT (National Pipe Tapered threads) style threads.

The "**Axial Clearance**" setting is simply the distance from the top of hole (or boss) to this Z clearance plane. This clearance plane is the approach to the hole in Z before the thread milling and Z departure from the hole after thread milling. It is usually best to make the "Axial Clearance" value greater than the pitch (or lead) of the thread. While the cutting of the threads, if the Z axis travels higher than the "Axial Clearance" plane (possible with course threads or "multiple thread starts") the software will automatically adjust this (before and after) clearance plane upwards.

If you manually set the "**Radial Clearance**" value to 0.100", the initial XY move will cause the **edge** of the tool to position to 0.100" from the (minor diameter) side wall of the hole. Or in the external world, will cause the edge of the tool to position to 0.100" from the (major diameter) side wall of the boss.

The first move is the initial XY (rapid) move dictated by the XY coordinate of hole or boss and the "Radial Clearance" value. Followed by a Z axis rapid move to 1" above the part or 1" above the "Axial Clearance" plane (whichever value is greater). Next a Z axis (rapid) move controlled by "Axial Clearance", then a Z axis (approach feed) move to thread top (or bottom), then a linear XY (approach feed) move while invoking cutter compensation. Then finally the "arc in" engage move (at 1/2 cutting feedrate).

When internal thread milling, the "Radial Clearance" value will always have an automatically calculated maximum value which, **if used**, would put the tool, regardless of tool diameter, at the center of the hole prior to cutting the thread. For internal threads, maximum possible radial clearance is always equal to half the difference between the "Minor Diameter" and the "Tool Diameter" (i.e. tool at center of hole – where the tool is furthest from the wall of the hole). If the Tool Diameter changes then the "Radial Clearance" value will also change. You can use any "Radial Clearance" value between 0 and this maximum radial clearance amount. If you set this "Radial Clearance" value to 0, it will **then reset** to and **display and use this maximum value**, which will force the cutter to **always start at the center of the hole**. This will stay in effect until you enter a non-zero number for the "Radial Clearance" value. The great thing about this feature is, if you change the tool diameter, the software will re-calculate and display this (now changed) radial clearance value and the tool will automatically maintain it's initial XY position at the center of the hole. If you set "Radial Clearance" = 0 (auto start at center of hole), you will also see a special symbol " • " to the right of the text (**Radial Clearance •**) indicating that auto positioning to center of hole is in effect. You will only see this special symbol when "Thread Orientation" is set to "INTERNAL". The external and internal threading have their own "Radial Clearance" values. I usually set the external "Radial Clearance" value to some number like 0.250" and let the internal "Radial Clearance" value set at "Auto" to start from hole center.

If you click on the "Radial Clearance" cell, it with display (for example) a message reading "*Value **must be less** than 0.609 and **should be greater** than 0.086*". This is telling you that **0.609"** is the **maximum** "Radial Clearance" value possible that can fit in the hole (positioning the cutter at center of the hole) for the current tool diameter, minor diameter and radial depth of cut. This also telling you that **0.086"** is the **minimum** "Radial Clearance" value possible so the XY linear positioning move (with it's fast **approach** feedrate) does not engage the part and so the cutter starts to arc (with it's 1/2 nominal **cutting** feedrate) towards full diameter **before** it gets into the cut. If you use a "Radial Clearance" value smaller than recommended value the software will automatically reduce the XY linear positioning move's feedrate to equal 1/4 the nominal cutting feedrate to prevent cutter damage.

If "Radial Clearance" cell displays a message like "*Value must be less than 0.186 and should be greater than 0.186*", it is telling you the tool diameter is nearly equal to the minor diameter. Or it can mean the total radial depth of cut is greater than the correlation between the tool diameter and "Radial Clearance" value. This condition is not necessarily a bad thing. This is also telling you the first XY linear move (which usually just fast feed positions close to the part wall) is going to cut into the part because the maximum "Radial Clearance" value (0.186") and minimum "Radial Clearance" value (0.186") have converged. Again the software will automatically reduce the XY linear positioning move's feedrate to equal 1/4 the nominal cutting feedrate to prevent cutter damage.

For very small threaded holes, sometimes the tool diameter is just slightly smaller (for cutter strength) than the minor diameter and the XY linear "positioning move" **will** engage the part (with protected low feedrate) prior to arcing towards full radial depth (towards major diameter). The software will override the "XY Approach" fast feedrate **down to** *1/4 normal cutting feedrate* on the XY positioning move if this move (edge of cutter) comes within 0.005" (radius value) of the hole wall. This means your **measured** "Gage Diameter" to **actual** "Gage Diameter" should have an error of less than 0.010" (diameter value).

For "Threads per Inch" simply enter 8 for 8 TPI, or you can enter 0.125 for the "Pitch" value. They both equate to the same thread form. If you enter a value for either "Threads per Inch" or "Pitch", the other value will automatically update.

For metric threads (using M12 X 1.5 for this example), you could enter the (metric to inch) calculation into either the "Threads per Inch" or "TPI" or you can enter in a formula instead. Entering a formula into the "Threads per Inch" field, it would be "**=25.4/1.5**" which then displays 16.93 after you pressed the "Enter" key (any Mxx X 1.5 metric thread has 16.93 threads per inch). Alternatively you could enter a formula into the "Pitch" field, it would be "**=1.5/25.4**" which then display 0.059055" after you press "Enter". Do not forget the "**=**" before the two numbers, this makes the expression a "formula" and automatically converts the expression into a direct value.

For a single start thread, lead is always equal to pitch. For multiple start threads (like the 4 thread starts on a plastic milk jug cap), the lead is equal to the pitch times the number of thread starts. If you had 8 TPI and "Thread Starts" was 4, then pitch (0.125") times 4 thread starts would equal the lead of 0.500". For "Thread Starts" > 1 the "**Pitch**" field **description changes** to "**Lead**" description because the lead amount is really the true movement of the Z axis per revolution of thread.

For multiple start threads, there is a 4 way correlation between "**Pitch**" and "**Lead**" and "**TPI**" and "**Thread Starts**".

For 8 TPI and 1 "Thread Starts" you would have a "Pitch" of 0.125" (and a "Lead" of 0.125")

For 8 TPI and 4 "Thread Starts" you would have a "Lead" of 0.500" (and still a "Pitch" of 0.125")

If you change (TPI) "Threads per Inch", then the "Lead" (or Pitch) would automatically change (and "Thread Starts" would not change).

If you change "Lead" (or Pitch), then the (TPI) "Threads per Inch" would automatically change (and "Thread Starts" would not change).

If you change "Thread Starts", then the "Lead" (or Pitch) would automatically change (and "Threads per Inch" would not change). Note: "Thread Starts" can be set up to a maximum of 12 thread starts.

There is a 3 way correlation between "**SFM**" and "**RPM**" and "**Tool Diameter**".

If you modify the "SFM" value, the "SFM" then becomes the primary variable and dictates the rules as such:
First the software auto calculates the "RPM" value based on the "SFM" and "Tool Diameter".
Then if the "Tool Diameter" is changed thereafter the software auto calculates the "RPM" value **keeping** the "SFM" value fixed. You will also see a special symbol " • " to the right of the text (**SFM** •). These rules stay in effect until you manually modify the "RPM" value.

If you modify the "RPM" value, the "RPM" then becomes the primary variable and dictates the rules as such:
First the software auto calculates the "SFM" value based on the "RPM" and "Tool Diameter".
Then if the "Tool Diameter" is changed thereafter the software auto calculates the "SFM" value **keeping** the "RPM" value fixed. You will also see a special symbol " • " to the right of the text (**RPM** •). These rules stay in effect until you manually modify the "SFM" value.

There is also a 3 way correlation between " **RPM**" and "**Feedrate per Tooth**" and "**Number of Teeth**".

Enter "RPM", "Feedrate per Tooth" and "Number of Teeth" then the software calculates a "linear" feedrate in inches per minute (IPM).

With circular interpolation and internal threading, the radial feedrate at the edge of the cutter is faster than the feed at the center of cutter. Some machine controls specify the feedrate at the center of the tool and some base the feedrate at the tool's outer edge.

If your machine control uses center of the tool feedrate, you need to turn "ON" the "**Feedrate Factor**" within this thread milling software. The "Feedrate Factor" will convert center of tool feedrate into edge of tool feedrate. If your machine control is using edge of tool feedrate, you will want the software's "Feedrate Factor" calculation turned "OFF". "OFF" sets the "Feedrate Factor" = 1 (= 100%). Simply click on the "Feedrate Factor" drop-down list box and then select "ON" or "OFF".
You can determine if your machine uses center of tool or edge of tool feed method by obtaining the cycle time of a test program. For example, if you cut a 2" diameter hole with a 1" diameter cutter (= 1" diameter programmed circular cutter path) at 3.14 IPM it should take 1 minute if your machine uses center of tool method. Otherwise the cycle time will be 2 minutes if your machine uses edge of tool feedrate.

For internal threads, Feedrate Factor = (Major Diameter – Tool Diameter) / Major Diameter

For external threads, Feedrate Factor = (Minor Diameter + Tool Diameter) / Minor Diameter

The "Feedrate Factor" is based on a calculated ratio between the tool diameter and (if internal threads) the major diameter. This value will be less than 1 for internal threads and greater than 1 for external threads.

If needed, the "Feedrate Factor" converts linear feedrate (in Inches per Minute) into circular feedrate so the cutter does not feed too fast for internal threads or does not feed too slow for external threads.

For the peripheral (edge of tool) feedrate to be **equal to** the linear straight line feedrate, the center of tool (programmed) feedrate must be adjusted based on the calculated "Feedrate Factor".

This "linear" cutting feedrate **multiplied by** the "Feedrate Factor" makes up the programmed feedrate required for the cutter to advance along the arc at the correct feedrate. This programmed feedrate is located in the "Feed (Cutting)" cell.

So the SFM (indirectly) or RPM X Feed per Tooth (IPT) X Number of Teeth X Feedrate Factor = Cutting Feed (IPM)

The Cutting Feed (IPM) **is usually the outputted calculation of all the other parameters** just mentioned, but you can **directly enter a "Cutting Feed (IPM)**" into it's cell, which temporarily overrides the calculation and backward calculates the "Feed per Tooth" from it.

There are also secondary feedrates for axial "**Feed (Z Approach)**" and radial "**Feed (XY Approach)**" moves towards the part. The "Feed (XY Approach)" feedrate you enter is for the first linear XY move (just after final positioning in Z axis) and just before the XY arc move into the workpiece. The XY arc move is always set to one half of "Cutting Feedrate". If you use too small of a "Radial Clearance" value, (meaning the linear XY move will engage the workpiece) then the software will force this XY linear move to also be one half of "Cutting Feedrate". This is to protect the cutter against using a fast **approach** feedrate when **engaging** the part. The "Feed (Z Approach)" value can be set at zero, which causes all the Z axis moves to rapid (G00) moves.

The rules for the "**Control Types**" parameter are as follows:

If "**FANUC**" is selected then a G43 & H value (for tool length compensation) are outputted with the first Z axis move. It also adds a D value (tool radius compensation) to the linear XY move just prior to the arc in move.

If "**OKUMA**" is selected then a G56 & H value (for tool length compensation) are outputted with the first Z axis move. It also adds a D value (tool radius compensation) to the linear XY move just prior to the arc in move.

If "**SIEMENS**" is selected then **no** G43 (Fanuc style) **nor** G56 (Okuma style) are outputted with the first Z axis move. It also discards the H value (for tool length compensation) and D value (tool radius compensation) since they are not needed.

The "**Cleanup Passes**" is simply the number of **extra** passes that will occur at full radial thread depth **after** all the other passes are completed.

The rules for "**Tool Length Comp**" and "**Tool Radius Comp**" parameters are as follows:

If you modify the value for **Tool Length Comp** then the **Tool Radius Comp** value will automatically default to the same number. Thereby saving the time of entering an identical number into **Tool Radius Comp**. If you need a different value for **Tool Radius Comp**, simply enter the value **after** the value for **Tool Length Comp**.

"Taper Angle" now has two choices. "**Taper Angle"** or **"Taper Ratio**". By double-clicking (the left mouse button) on it's description (cell "B14" or "C14") the description will toggle between **Taper Ratio** and **Taper Angle**. **Taper Ratio** where the value is equal to a ratio of the **diameter** versus length of thread (0.0625 for standard NPT pipe threads). The **Taper Ratio** is also equal to 2*TAN of the **Taper Angle**. The **Taper Angle** is the angle of the taper (based on the inverse tangent of **Taper Ratio/2**). The **Taper Angle** for a standard NPT thread is 1 degree 47 minutes 24 seconds. If you modify either of these 2 values; the other one will automatically update as well.

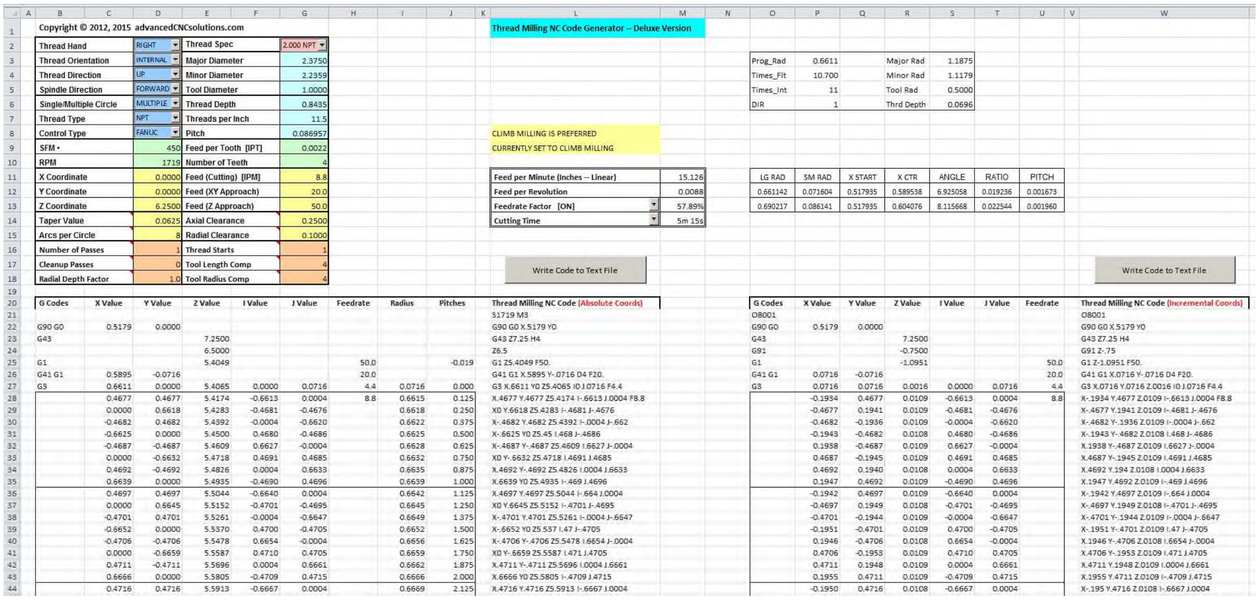
"**Thread Depth"** now has two choices. By double-clicking (the left mouse button) on it's description (cell "E6" or "F6") the description will toggle between "**Thread Depth"** and **"Thread Turns**". **Thread Depth** is obviously the thread depth. **Thread Turns** is simply the number of turns of the thread. It equates to **Thread Depth/Pitch** (or Lead if multiple starts) which is also equivalent to **Thread Depth*TPI/Starts**. If you modify either of these 2 values; the other one will automatically update as well.

Another new feature is the ability to turn the "Cell Comments" on or off. There are cell comments defined for cells C14 through C18 and cells F14 through F18. They are designated by a little red triangle in the upper right corner of the mentioned cells. When the cursor is hovered over one of these cells a cell comment "pops" up. By double-clicking on cell "**A1**" the comments will toggle between on (visible) or off (not visible).

You can toggle on or off the "Freeze Panes" feature on the "Code_Main" sheet by double-clicking the left mouse button. This freezes the window "pane" so the "header labels" (**G Codes   X Value   Y Value** ...) are always visible. When this feature is activated Excel will display a horizontal line across the window (and all rows above this line are "frozen" in place. When performing this task, it's best to double-click near the center of most any cell (and not near any edge of cell). You should experiment with this very handy feature; it becomes quite useful when there are many lines of code and you need a reference to which columns are which. It is also useful if (when row 1 is visible) you invoke the "Freeze Panes" feature to allow you to always see all of parameter settings and the header row while scrolling down through the code. If column "A" is not visible when you double-click to turn "on" the freeze panes feature, then instead of 2 panes visible (upper and lower) there will be 4 panes visible (splitting the screen horizontally as well as vertically). Because of these above new rules (a few paragraphs back), 5 cells (to double-click on) no longer work for the "Freeze Panes" feature. They are the previously mentioned cells “A1”, “B14”, “C14”, “E6” & “F6”.

To generate (NC code) text file of current output results, press one of the “**Write Code to Text File**” buttons and the file Thread_Mill_Code.txt (or any user defined filename) will be created in the user specified folder. The software generates NC code for absolute coordinates (left "**Write Code to Text File**" button") and incremental coordinates (right "Write Code to Text File" button"). The absolute coordinates version would most likely be used in (appended to) a main program and the incremental coordinates version works well as a sub-program. You could use a call to the sub-program to keep the main program considerably shorter. You could call the sub-program once per hole if need be. The flexibility is there for you to use as you see fit. Note: You are **allowed to edit** the absolute and/or incremental NC code (column "L" o column "W") prior to clicking "Write Code to Text File" button.

Zoom in



Copyright © 2012, 2015 advancedCNCsolutions.com

Thread Hand	RIGHT	Thread Spec	2.000 NPT
Thread Orientation	INTERNAL	Major Diameter	2.3750
Thread Direction	UP	Minor Diameter	2.2359
Spindle Direction	FORWARD	Tool Diameter	1.0000
Single/Multiple Circle	MULTIPLE	Thread Depth	0.8435
Thread Type	NPT	Threads per Inch	11.5
Control Type	FANUC	Pitch	0.086957
SFM •	450	Feed per Tooth [IPT]	0.0022
RPM	1719	Number of Teeth	4
X Coordinate	0.0000	Feed (Cutting) [IPM]	8.8
Y Coordinate	0.0000	Feed (XY Approach)	20.0
Z Coordinate	6.2500	Feed (Z Approach)	50.0
Taper Value	0.0625	Axial Clearance	0.2500
Arcs per Circle	8	Radial Clearance	0.1000
Number of Passes	1	Thread Starts	1
Cleanup Passes	0	Tool Length Comp	4
Radial Depth Factor	1.0	Tool Radius Comp	4

Thread Milling NC Code Generator -- Deluxe Version

CLIMB MILLING IS PREFERRED
CURRENTLY SET TO CLIMB MILLING

Feed per Minute (Inches -- Linear)	15.126
Feed per Revolution	0.0088
Feedrate Factor [ON]	57.89%
Cutting Time	5m 15s

Prog_Rad	0.6611	Major Rad	1.1875
Times_Flt	10.700	Minor Rad	1.1179
Times_Int	11	Tool Rad	0.5000
DIR	1	Thrd Depth	0.0696

LG RAD	SM RAD	X START	X CTR	ANGLE	RATIO	PITCH
0.661142	0.071604	0.517935	0.589538	6.925058	0.019236	0.001673
0.690217	0.086141	0.517935	0.604076	8.115668	0.022544	0.001960

Write Code to Text File

G Codes	X Value	Y Value	Z Value	I Value	J Value	Feedrate	Radius	Pitches
G90 G0	0.5179	0.0000						
G43			7.2500					
			6.5000					
G1			5.4049			50.0		-0.019
G41 G1	0.5895	-0.0716				20.0		
G3	0.6611	0.0000	5.4065	0.0000	0.0716	4.4	0.0716	0.000
	0.4677	0.4677	5.4174	-0.6613	0.0004	8.8	0.6615	0.125
	0.0000	0.6618	5.4283	-0.4681	-0.4676		0.6618	0.250
	-0.4682	0.4682	5.4392	-0.0004	-0.6620		0.6622	0.375
	-0.6625	0.0000	5.4500	0.4680	-0.4686		0.6625	0.500
	-0.4687	-0.4687	5.4609	0.6627	-0.0004		0.6628	0.625
	0.0000	-0.6632	5.4718	0.4691	0.4685		0.6632	0.750
	0.4692	-0.4692	5.4826	0.0004	0.6633		0.6635	0.875
	0.6639	0.0000	5.4935	-0.4690	0.4696		0.6639	1.000
	0.4697	0.4697	5.5044	-0.6640	0.0004		0.6642	1.125
	0.0000	0.6645	5.5152	-0.4701	-0.4695		0.6645	1.250
	-0.4701	0.4701	5.5261	-0.0004	-0.6647		0.6649	1.375
	-0.6652	0.0000	5.5370	0.4700	-0.4705		0.6652	1.500
	-0.4706	-0.4706	5.5478	0.6654	-0.0004		0.6656	1.625
	0.0000	-0.6659	5.5587	0.4710	0.4705		0.6659	1.750
	0.4711	-0.4711	5.5696	0.0004	0.6661		0.6662	1.875
	0.6666	0.0000	5.5805	-0.4709	0.4715		0.6666	2.000
	0.4716	0.4716	5.5913	-0.6667	0.0004		0.6669	2.125

Thread Milling NC Code (Absolute Coords)

```
S1719 M3
G90 G0 X.5179 Y0
G43 Z7.25 H4
Z6.5
G1 Z5.4049 F50.
G41 G1 X.5895 Y-.0716 D4 F20.
G3 X.6611 Y0 Z5.4065 I0 J.0716 F4.4
X.4677 Y.4677 Z5.4174 I-.6613 J.0004 F8.8
X0 Y.6618 Z5.4283 I-.4681 J-.4676
X-.4682 Y.4682 Z5.4392 I-.0004 J-.662
X-.6625 Y0 Z5.45 I.468 J-.4686
X-.4687 Y-.4687 Z5.4609 I.6627 J-.0004
X0 Y-.6632 Z5.4718 I.4691 J.4685
X.4692 Y-.4692 Z5.4826 I.0004 J.6633
X.6639 Y0 Z5.4935 I-.469 J.4696
X.4697 Y.4697 Z5.5044 I-.664 J.0004
X0 Y.6645 Z5.5152 I-.4701 J-.4695
X-.4701 Y.4701 Z5.5261 I-.0004 J-.6647
X-.6652 Y0 Z5.537 I.47 J-.4705
X-.4706 Y-.4706 Z5.5478 I.6654 J-.0004
X0 Y-.6659 Z5.5587 I.471 J.4705
X.4711 Y-.4711 Z5.5696 I.0004 J.6661
X.6666 Y0 Z5.5805 I-.4709 J.4715
X.4716 Y.4716 Z5.5913 I-.6667 J.0004
```

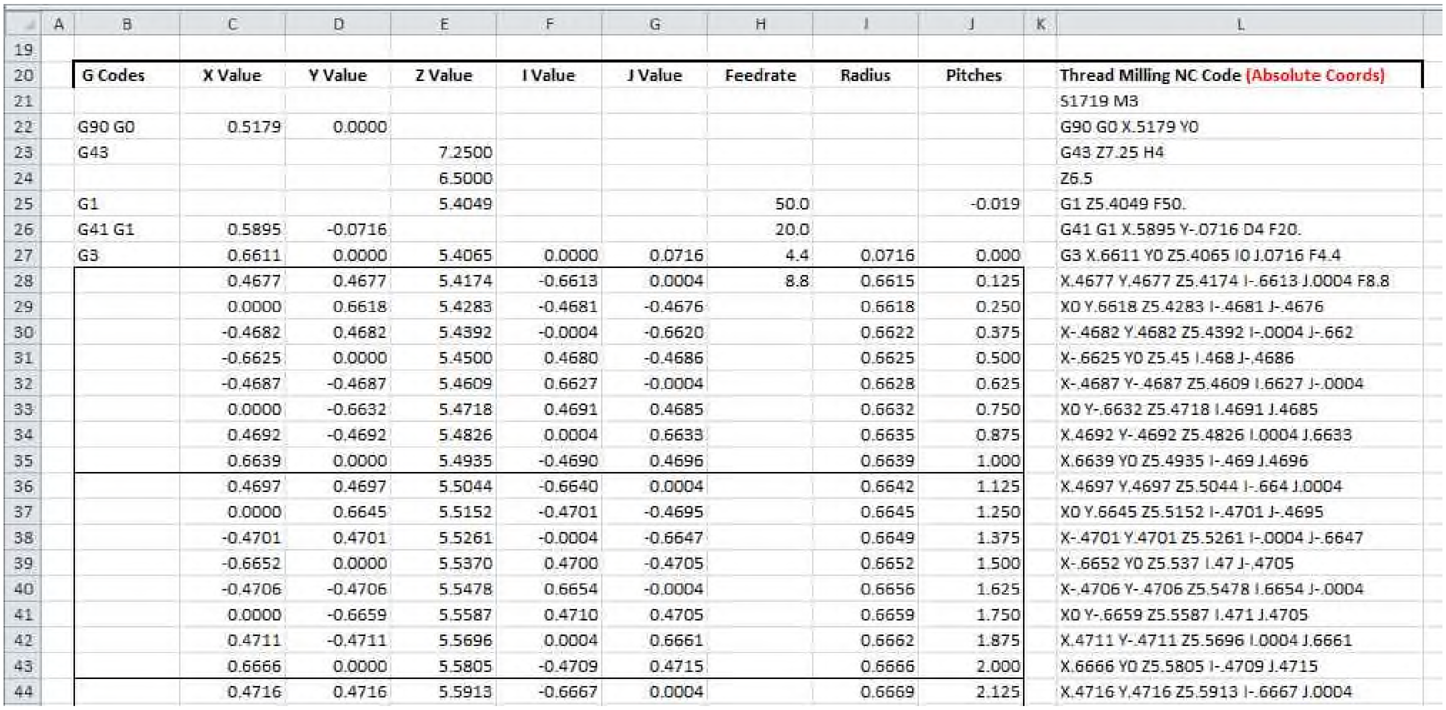
Write Code to Text File

G Codes	X Value	Y Value	Z Value	I Value	J Value	Feedrate
O8001						
G90 G0	0.5179	0.0000				
G43			7.2500			
G91			-0.7500			
G1			-1.0951			50.0
G41 G1	0.0716	-0.0716				20.0
G3	0.0716	0.0716	0.0016	0.0000	0.0716	4.4
	-0.1934	0.4677	0.0109	-0.6613	0.0004	8.8
	-0.4677	0.1941	0.0109	-0.4681	-0.4676	
	-0.4682	-0.1936	0.0109	-0.0004	-0.6620	
	-0.1943	-0.4682	0.0108	0.4680	-0.4686	
	0.1938	-0.4687	0.0109	0.6627	-0.0004	
	0.4687	-0.1945	0.0109	0.4691	0.4685	
	0.4692	0.1940	0.0108	0.0004	0.6633	
	0.1947	0.4692	0.0109	-0.4690	0.4696	
	-0.1942	0.4697	0.0109	-0.6640	0.0004	
	-0.4697	0.1949	0.0108	-0.4701	-0.4695	
	-0.4701	-0.1944	0.0109	-0.0004	-0.6647	
	-0.1951	-0.4701	0.0109	0.4700	-0.4705	
	0.1946	-0.4706	0.0108	0.6654	-0.0004	
	0.4706	-0.1953	0.0109	0.4710	0.4705	
	0.4711	0.1948	0.0109	0.0004	0.6661	
	0.1955	0.4711	0.0109	-0.4709	0.4715	
	-0.1950	0.4716	0.0108	-0.6667	0.0004	

Thread Milling NC Code (Incremental Coords)

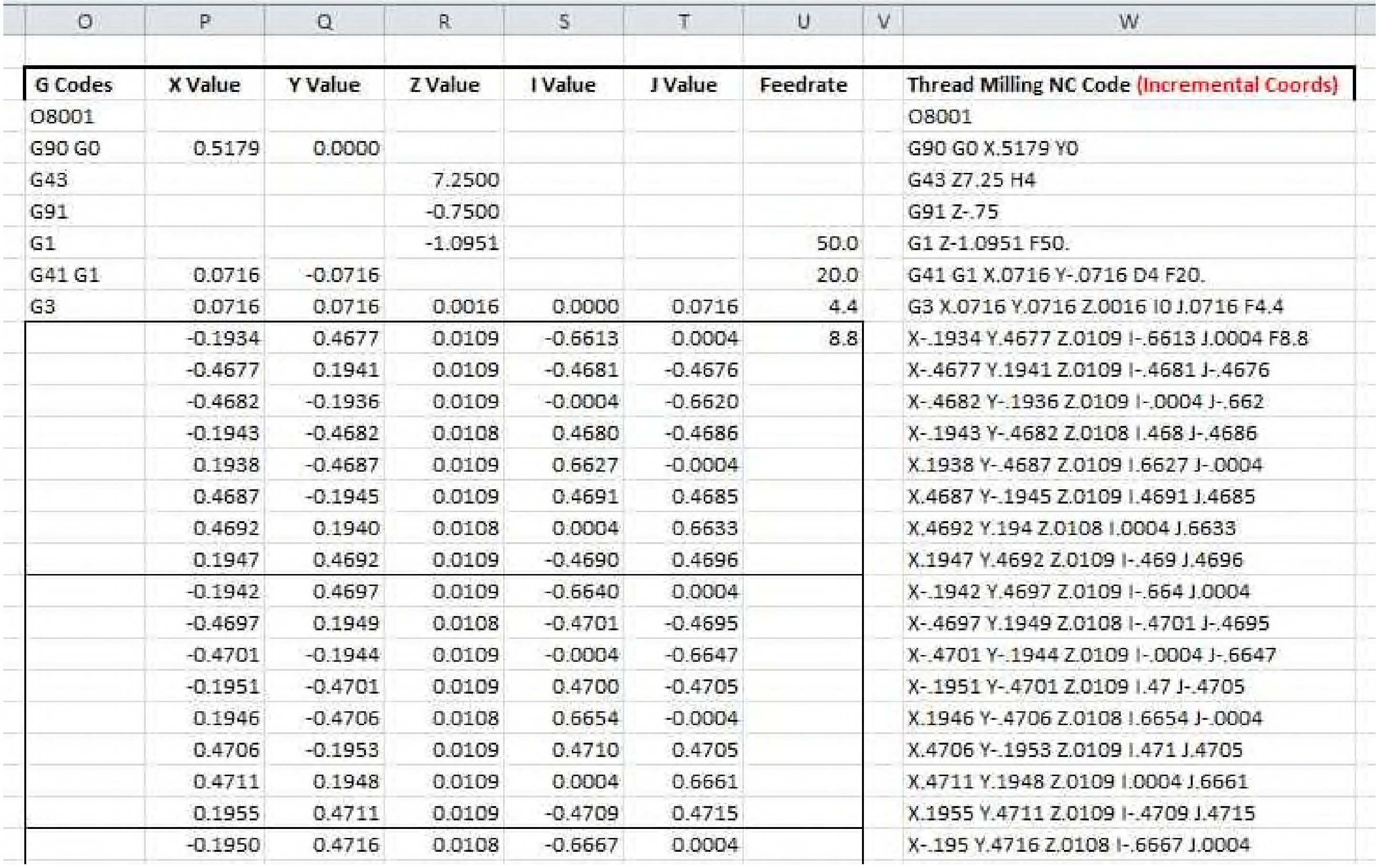
```
O8001
G90 G0 X.5179 Y0
G43 Z7.25 H4
G91 Z-.75
G1 Z-1.0951 F50.
G41 G1 X.0716 Y-.0716 D4 F20.
G3 X.0716 Y.0716 Z.0016 I0 J.0716 F4.4
X-.1934 Y.4677 Z.0109 I-.6613 J.0004 F8.8
X-.4677 Y.1941 Z.0109 I-.4681 J-.4676
X-.4682 Y-.1936 Z.0109 I-.0004 J-.662
X-.1943 Y-.4682 Z.0108 I.468 J-.4686
X.1938 Y-.4687 Z.0109 I.6627 J-.0004
X.4687 Y-.1945 Z.0109 I.4691 J.4685
X.4692 Y.194 Z.0108 I.0004 J.6633
X.1947 Y.4692 Z.0109 I-.469 J.4696
X-.1942 Y.4697 Z.0109 I-.664 J.0004
X-.4697 Y.1949 Z.0108 I-.4701 J-.4695
X-.4701 Y-.1944 Z.0109 I-.0004 J-.6647
X-.1951 Y-.4701 Z.0109 I.47 J-.4705
X.1946 Y-.4706 Z.0108 I.6654 J-.0004
X.4706 Y-.1953 Z.0109 I.471 J.4705
X.4711 Y.1948 Z.0109 I.0004 J.6661
X.1955 Y.4711 Z.0109 I-.4709 J.4715
X-.195 Y.4716 Z.0108 I-.6667 J.0004
```

Below is sample parameter settings with the sample NC code using absolute coordinates.



	A	B	C	D	E	F	G	H	I	J	K	L
19												
20		G Codes	X Value	Y Value	Z Value	I Value	J Value	Feedrate	Radius	Pitches		Thread Milling NC Code (Absolute Coords)
21												S1719 M3
22		G90 G0	0.5179	0.0000								G90 G0 X.5179 Y0
23		G43			7.2500							G43 Z7.25 H4
24					6.5000							Z6.5
25		G1			5.4049			50.0		-0.019		G1 Z5.4049 F50.
26		G41 G1	0.5895	-0.0716				20.0				G41 G1 X.5895 Y-.0716 D4 F20.
27		G3	0.6611	0.0000	5.4065	0.0000	0.0716	4.4	0.0716	0.000		G3 X.6611 Y0 Z5.4065 I0 J.0716 F4.4
28			0.4677	0.4677	5.4174	-0.6613	0.0004	8.8	0.6615	0.125		X.4677 Y.4677 Z5.4174 I-.6613 J.0004 F8.8
29			0.0000	0.6618	5.4283	-0.4681	-0.4676		0.6618	0.250		X0 Y.6618 Z5.4283 I-.4681 J-.4676
30			-0.4682	0.4682	5.4392	-0.0004	-0.6620		0.6622	0.375		X-.4682 Y.4682 Z5.4392 I-.0004 J-.662
31			-0.6625	0.0000	5.4500	0.4680	-0.4686		0.6625	0.500		X-.6625 Y0 Z5.45 I.468 J-.4686
32			-0.4687	-0.4687	5.4609	0.6627	-0.0004		0.6628	0.625		X-.4687 Y-.4687 Z5.4609 I.6627 J-.0004
33			0.0000	-0.6632	5.4718	0.4691	0.4685		0.6632	0.750		X0 Y-.6632 Z5.4718 I.4691 J.4685
34			0.4692	-0.4692	5.4826	0.0004	0.6633		0.6635	0.875		X.4692 Y-.4692 Z5.4826 I.0004 J.6633
35			0.6639	0.0000	5.4935	-0.4690	0.4696		0.6639	1.000		X.6639 Y0 Z5.4935 I-.469 J.4696
36			0.4697	0.4697	5.5044	-0.6640	0.0004		0.6642	1.125		X.4697 Y.4697 Z5.5044 I-.664 J.0004
37			0.0000	0.6645	5.5152	-0.4701	-0.4695		0.6645	1.250		X0 Y.6645 Z5.5152 I-.4701 J-.4695
38			-0.4701	0.4701	5.5261	-0.0004	-0.6647		0.6649	1.375		X-.4701 Y.4701 Z5.5261 I-.0004 J-.6647
39			-0.6652	0.0000	5.5370	0.4700	-0.4705		0.6652	1.500		X-.6652 Y0 Z5.537 I.47 J-.4705
40			-0.4706	-0.4706	5.5478	0.6654	-0.0004		0.6656	1.625		X-.4706 Y-.4706 Z5.5478 I.6654 J-.0004
41			0.0000	-0.6659	5.5587	0.4710	0.4705		0.6659	1.750		X0 Y-.6659 Z5.5587 I.471 J.4705
42			0.4711	-0.4711	5.5696	0.0004	0.6661		0.6662	1.875		X.4711 Y-.4711 Z5.5696 I.0004 J.6661
43			0.6666	0.0000	5.5805	-0.4709	0.4715		0.6666	2.000		X.6666 Y0 Z5.5805 I-.4709 J.4715
44			0.4716	0.4716	5.5913	-0.6667	0.0004		0.6669	2.125		X.4716 Y.4716 Z5.5913 I-.6667 J.0004

Below is the same sample parameter settings with the NC code using incremental coordinates as a subprogram.



O	P	Q	R	S	T	U	V	W
G Codes	X Value	Y Value	Z Value	I Value	J Value	Feedrate		Thread Milling NC Code (Incremental Coords)
O8001								O8001
G90 G0	0.5179	0.0000						G90 G0 X.5179 Y0
G43			7.2500					G43 Z7.25 H4
G91			-0.7500					G91 Z-.75
G1			-1.0951			50.0		G1 Z-1.0951 F50.
G41 G1	0.0716	-0.0716				20.0		G41 G1 X.0716 Y-.0716 D4 F20.
G3	0.0716	0.0716	0.0016	0.0000	0.0716	4.4		G3 X.0716 Y.0716 Z.0016 I0 J.0716 F4.4
	-0.1934	0.4677	0.0109	-0.6613	0.0004	8.8		X-.1934 Y.4677 Z.0109 I-.6613 J.0004 F8.8
	-0.4677	0.1941	0.0109	-0.4681	-0.4676			X-.4677 Y.1941 Z.0109 I-.4681 J-.4676
	-0.4682	-0.1936	0.0109	-0.0004	-0.6620			X-.4682 Y-.1936 Z.0109 I-.0004 J-.662
	-0.1943	-0.4682	0.0108	0.4680	-0.4686			X-.1943 Y-.4682 Z.0108 I.468 J-.4686
	0.1938	-0.4687	0.0109	0.6627	-0.0004			X.1938 Y-.4687 Z.0109 I.6627 J-.0004
	0.4687	-0.1945	0.0109	0.4691	0.4685			X.4687 Y-.1945 Z.0109 I.4691 J.4685
	0.4692	0.1940	0.0108	0.0004	0.6633			X.4692 Y.194 Z.0108 I.0004 J.6633
	0.1947	0.4692	0.0109	-0.4690	0.4696			X.1947 Y.4692 Z.0109 I-.469 J.4696
	-0.1942	0.4697	0.0109	-0.6640	0.0004			X-.1942 Y.4697 Z.0109 I-.664 J.0004
	-0.4697	0.1949	0.0108	-0.4701	-0.4695			X-.4697 Y.1949 Z.0108 I-.4701 J-.4695
	-0.4701	-0.1944	0.0109	-0.0004	-0.6647			X-.4701 Y-.1944 Z.0109 I-.0004 J-.6647
	-0.1951	-0.4701	0.0109	0.4700	-0.4705			X-.1951 Y-.4701 Z.0109 I.47 J-.4705
	0.1946	-0.4706	0.0108	0.6654	-0.0004			X.1946 Y-.4706 Z.0108 I.6654 J-.0004
	0.4706	-0.1953	0.0109	0.4710	0.4705			X.4706 Y-.1953 Z.0109 I.471 J.4705
	0.4711	0.1948	0.0109	0.0004	0.6661			X.4711 Y.1948 Z.0109 I.0004 J.6661
	0.1955	0.4711	0.0109	-0.4709	0.4715			X.1955 Y.4711 Z.0109 I-.4709 J.4715
	-0.1950	0.4716	0.0108	-0.6667	0.0004			X-.195 Y.4716 Z.0108 I-.6667 J.0004

"XY_Coords" sheet

The "XY_Coords" sheet contains a table of X&Y coordinates for the thread milling of multiple holes (or bosses externally). The software supports a maximum of 40 holes or bosses. The first XY coordinate for the holes/bosses is also on the "Main_Code" sheet . This (first) XY coordinate can be modified on this sheet ("XY_Coords") or on the "Main_Code" sheet and other one will auto update to the one that was last changed. With this version of the software all holes must be on the same Z plane. This table of XY coordinates for thread milling multiple holes (or bosses) is unique to this deluxe version of the software.

"Thread_Specs" sheet

The "Thread_Specs" sheet contains the threading specifications for each particular thread type. More than 90 internal and 90 external thread type entries have come pre-loaded. You can modify any of the existing thread spec values and you can add new (or remove) thread spec entries to the list. This thread spec's list was designed for a maximum of 500 entries. This table of thread specifications for thread milling is unique to this deluxe version of the software.

The 1st column contains the description of the particular thread (and this text is what you see on the "Main_Code" sheet "Thread Spec" drop-down list box). If you add to this list, columns B through G (minimum) must be **valid** for it to show up in the "Thread Specs" list. If you do not have a valid row 3 thread spec entry, a default one (named "TMP THRD") will be auto loaded for you. This default (row 3) entry can be modified at any time.

Reading down the table, if you have a continuous group of **valid** rows (of thread specs) then an invalid row(s) followed by valid rows, the software only recognizes the first **continuous** group of valid rows. You will only see "Main_Code" sheet thread descriptions (in it's drop down list) for the first continuous group of valid thread spec entries on the "Thread_Specs" sheet.

On the "Main_Code" sheet you can select any thread specification from this table and all of the relevant parameters are loaded into their respective fields (cells) automatically. If you modify the entry in the table you currently have selected (on the "Main_Code" sheet), you **may need** to re-select it in the "Thread_Spec" drop-down list box on the "Main_Code" sheet and, if used, you **may then need** to re-select any values on the "Axial_Passes" sheet.

Once you select a thread spec from the "Thread Spec" drop-down list box, the box will be reddish in color meaning it is ("live") because all of the parameters (for the entry selected) are automatically copied from the "Thread Spec" sheet to the respective cells on the "Main_Code" sheet.

These parameters (from the "Thread_Specs" sheet) are major & minor diameters, TPI, STD/NPT (standard straight or National Pipe Tapered) designator and if it exists the thread depth. You should double check that my pre-loaded values are correct if you are to use them. Pay particular attention to pre-loaded NPT thread specs for you most **LIKELY** will want to adjust/change these. These major diameter NPT thread specs are based off of the **theoretical** (crest) top of the tooth. If on the "Main_Code" sheet the "Thread Orientation" was already set to "INTERNAL" then the internal major and minor diameters will load (and likewise for the external thread). If you change from external orientation to internal (or vice-versa) later the major and minor diameters (and thread depth if in table) will automatically update in the proper cells. This is provided the "Thread Spec" choice is still "live". If it is still "live" the "Thread Spec" text box will be a reddish color and if it is "Dead" it will be a grayish color. The "Thread Spec" text box will go "live" (reddish) when you select one of the choices from it's drop-down list. It will go "dead" (gray) once you **manually** change **any** parameter that is tied to the thread specs table, such as major or minor diameter or pitch or straight standard (STD) vs. tapered (NPT) or the depth (if used in table) or the "Taper Ratio" setting not equal to the default 0.0625 (if NPT thread). When it goes "dead" (**turns gray**) you are no longer using **ALL of the values from the thread specs table**.

IMPORTANT: Stated in a different way, if the "Thread Spec" text box is reddish in color ("live"), you know that you are using **ALL of the preloaded values from the thread specs table** and if it is gray ("dead") you **are responsible** for setting all of the RELATED parameters **manually** on the "Main_Code" sheet .

If you need to re-select a specific thread spec again (to make it "live" again), you will need to select a different thread spec **first and then re-select the one you need**. This is just the way Excel works, you cannot select something that is already selected. In the table there is a column for standard (straight) versus NPT tapered pipe threads; (standard) STD = 0 and (tapered) NPT =1. If no number is in the cell, it's assumed to be 0 (standard straight thread). Since this table contains strictly inch values, any metric numbers will need to be converted to inch format. The TPI (threads per inch) for metric threads can be entered into the table one of two ways. One way (for M12 x 1.75 thread for example) is to take the results of 25.4/1.75 which is 14.5143 and enter it into the table and the second way is to simply type in **=25.4/1.75** (don't forget to include the equals symbol as well) and this Excel formula will then display the same results as first method. I have some examples of this already in the table.

Note: For straight threads, external minor diameter values are formulas using 0.7083 (70.8%) of the thread height from major diameter down to minor diameter) and internal threads use 0.6250 (62.5%) of the thread height to obtain the minor diameter values. You can simply modify the thread height percentages within the formulas or they can be modified to any direct values instead of the formulas if need be.

The 0.7083 (externally) & 0.6250 (internally) are commonly used values found on many different websites. These websites will use the formulas:

0.61343 * Pitch = external thread height which is = 0.61343 / Sin 60° = **0.7083** of thread height
0.54127 * Pitch = internal thread height which is = 0.54127 / Sin 60° = **0.6250** of thread height

The “**Tool Diameter**” value on the "Main_Code" sheet should be specified at the **theoretical** crest (top) of the tooth and for tapered threads, the “Tool Diameter” should be specified at the theoretical crest of the **1st** tooth (at small end of taper). A large crest radius would otherwise "throw off" the accuracy of the thread diameter. The theoretical crest needs to be used since all of the "Thread_ Spec" table major (and minor) diameter values are derived from the "Basic" (sharp "V") major diameters. Using this theoretical crest will then produce an accurate major diameter (and therefore pitch diameter) when thread milling **regardless** of the size of tool tip radius. If you used the top of the radius (instead of the theoretical crest) for the “**Tool Diameter**” value, then you would need an additional thread spec entry (new "tweaked" major & minor diameters) for each tool tip radius size used.

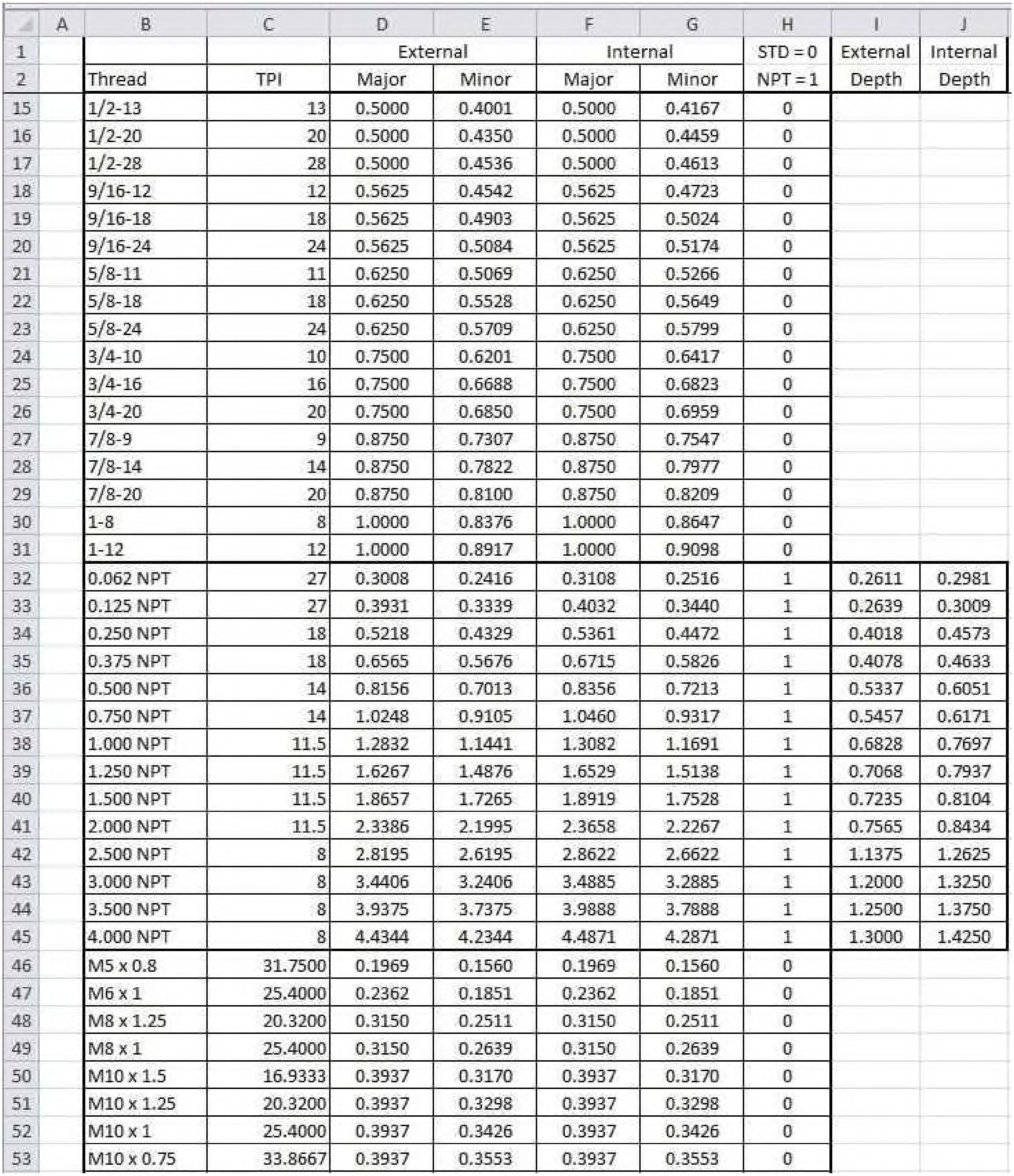
The minor diameter values used in the table are also derived from the "Basic" (sharp "V") major diameters values. The reason being if you take 4 (for example) equal radial thread passes the first pass will more accurately represent 1/4th (25%) of the total radial thread depth. If the minor diameter in the table were large (in relation to the major diameter) you would have a shallow thread depth in ***theory*** (in relation to the ***physical*** "tap" drill diameter you used) causing the first threading pass to cut **more** than the intended 25% of thread depth.

If you use too small of thread height in the table (minor diameter too large), but you drill the hole as to create a larger physical thread height than the table values represent, then the "**Radial Clearance**" value you set to the side wall of the hole will be less accurate.

Note: Because the minor diameter values used in the table are also derived from the "Basic" (sharp "V") values, they do not accurately reflect the minor diameter size you would drill to prior to thread milling.

Below is a sample of the "Thread Specs" table with the fields (parameters) which make up any given (external and internal) thread specification. Note a "Thread Spec" entry (one row) can include the thread depths where appropriate.

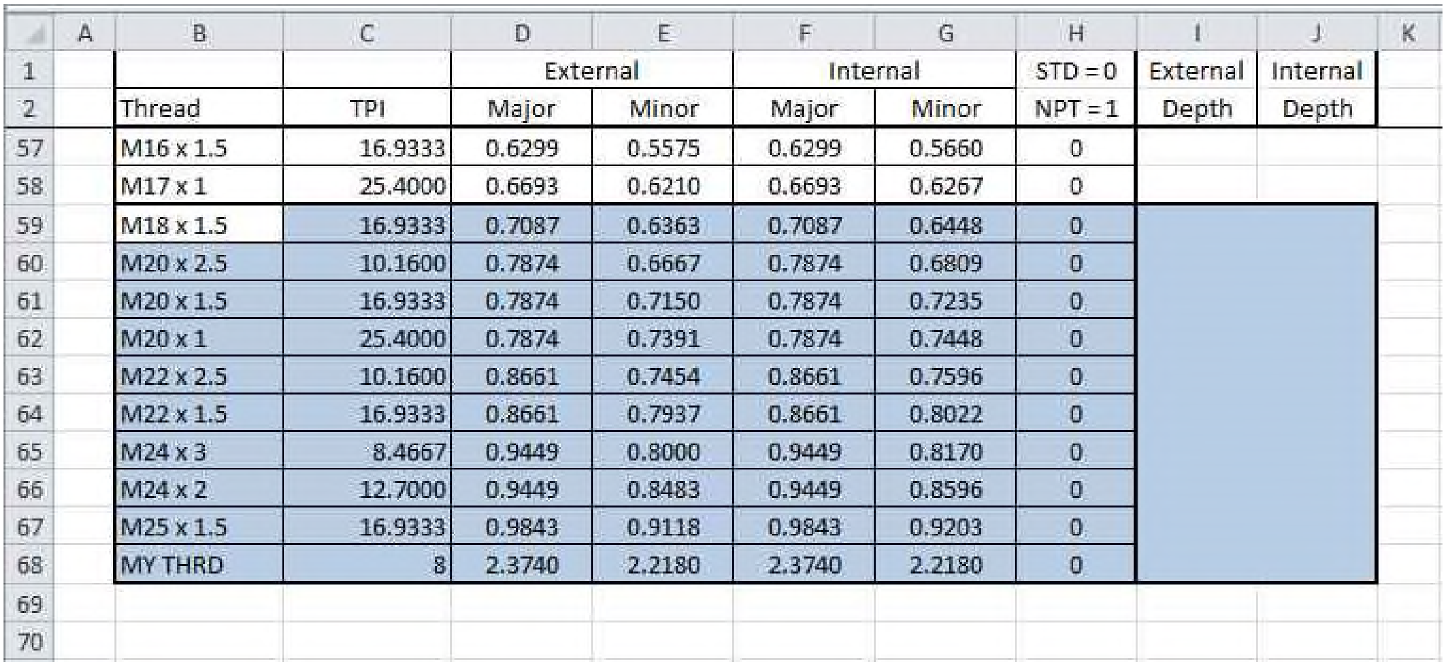
The text in column "B" is the descriptions you will see in the "Thread Spec" drop-down list on the "Main_Code" sheet.



	A	B	C	D	E	F	G	H	I	J
1				External		Internal		STD = 0	External	Internal
2		Thread	TPI	Major	Minor	Major	Minor	NPT = 1	Depth	Depth
15		1/2-13	13	0.5000	0.4001	0.5000	0.4167	0		
16		1/2-20	20	0.5000	0.4350	0.5000	0.4459	0		
17		1/2-28	28	0.5000	0.4536	0.5000	0.4613	0		
18		9/16-12	12	0.5625	0.4542	0.5625	0.4723	0		
19		9/16-18	18	0.5625	0.4903	0.5625	0.5024	0		
20		9/16-24	24	0.5625	0.5084	0.5625	0.5174	0		
21		5/8-11	11	0.6250	0.5069	0.6250	0.5266	0		
22		5/8-18	18	0.6250	0.5528	0.6250	0.5649	0		
23		5/8-24	24	0.6250	0.5709	0.6250	0.5799	0		
24		3/4-10	10	0.7500	0.6201	0.7500	0.6417	0		
25		3/4-16	16	0.7500	0.6688	0.7500	0.6823	0		
26		3/4-20	20	0.7500	0.6850	0.7500	0.6959	0		
27		7/8-9	9	0.8750	0.7307	0.8750	0.7547	0		
28		7/8-14	14	0.8750	0.7822	0.8750	0.7977	0		
29		7/8-20	20	0.8750	0.8100	0.8750	0.8209	0		
30		1-8	8	1.0000	0.8376	1.0000	0.8647	0		
31		1-12	12	1.0000	0.8917	1.0000	0.9098	0		
32		0.062 NPT	27	0.3008	0.2416	0.3108	0.2516	1	0.2611	0.2981
33		0.125 NPT	27	0.3931	0.3339	0.4032	0.3440	1	0.2639	0.3009
34		0.250 NPT	18	0.5218	0.4329	0.5361	0.4472	1	0.4018	0.4573
35		0.375 NPT	18	0.6565	0.5676	0.6715	0.5826	1	0.4078	0.4633
36		0.500 NPT	14	0.8156	0.7013	0.8356	0.7213	1	0.5337	0.6051
37		0.750 NPT	14	1.0248	0.9105	1.0460	0.9317	1	0.5457	0.6171
38		1.000 NPT	11.5	1.2832	1.1441	1.3082	1.1691	1	0.6828	0.7697
39		1.250 NPT	11.5	1.6267	1.4876	1.6529	1.5138	1	0.7068	0.7937
40		1.500 NPT	11.5	1.8657	1.7265	1.8919	1.7528	1	0.7235	0.8104
41		2.000 NPT	11.5	2.3386	2.1995	2.3658	2.2267	1	0.7565	0.8434
42		2.500 NPT	8	2.8195	2.6195	2.8622	2.6622	1	1.1375	1.2625
43		3.000 NPT	8	3.4406	3.2406	3.4885	3.2885	1	1.2000	1.3250
44		3.500 NPT	8	3.9375	3.7375	3.9888	3.7888	1	1.2500	1.3750
45		4.000 NPT	8	4.4344	4.2344	4.4871	4.2871	1	1.3000	1.4250
46		M5 x 0.8	31.7500	0.1969	0.1560	0.1969	0.1560	0		
47		M6 x 1	25.4000	0.2362	0.1851	0.2362	0.1851	0		
48		M8 x 1.25	20.3200	0.3150	0.2511	0.3150	0.2511	0		
49		M8 x 1	25.4000	0.3150	0.2639	0.3150	0.2639	0		
50		M10 x 1.5	16.9333	0.3937	0.3170	0.3937	0.3170	0		
51		M10 x 1.25	20.3200	0.3937	0.3298	0.3937	0.3298	0		
52		M10 x 1	25.4000	0.3937	0.3426	0.3937	0.3426	0		
53		M10 x 0.75	33.8667	0.3937	0.3553	0.3937	0.3553	0		

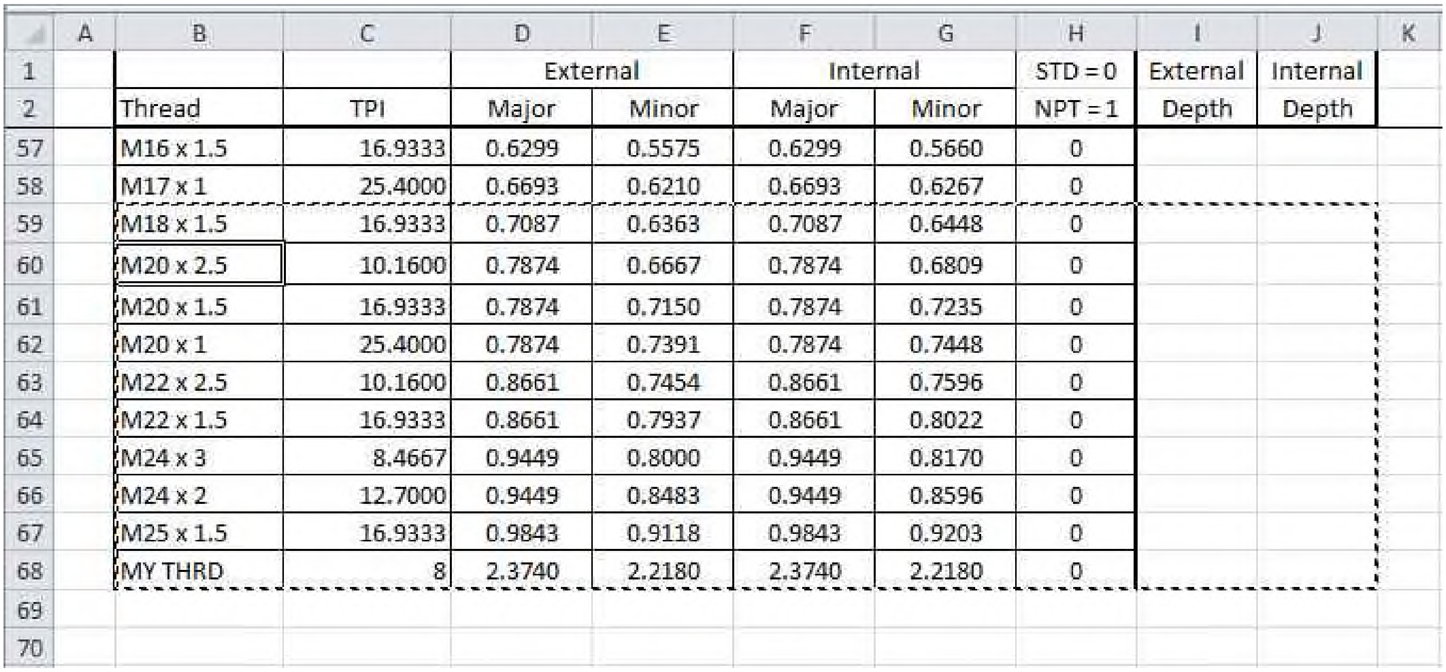
To add a new thread spec entry to list:

In this example we are adding a new thread spec entry (a fictitious M19 x 2 thread). Select cells "B59" through "J68" (to end of list) as shown. Now right-click and select "Copy" (or Ctrl-C) to copy selected entries.



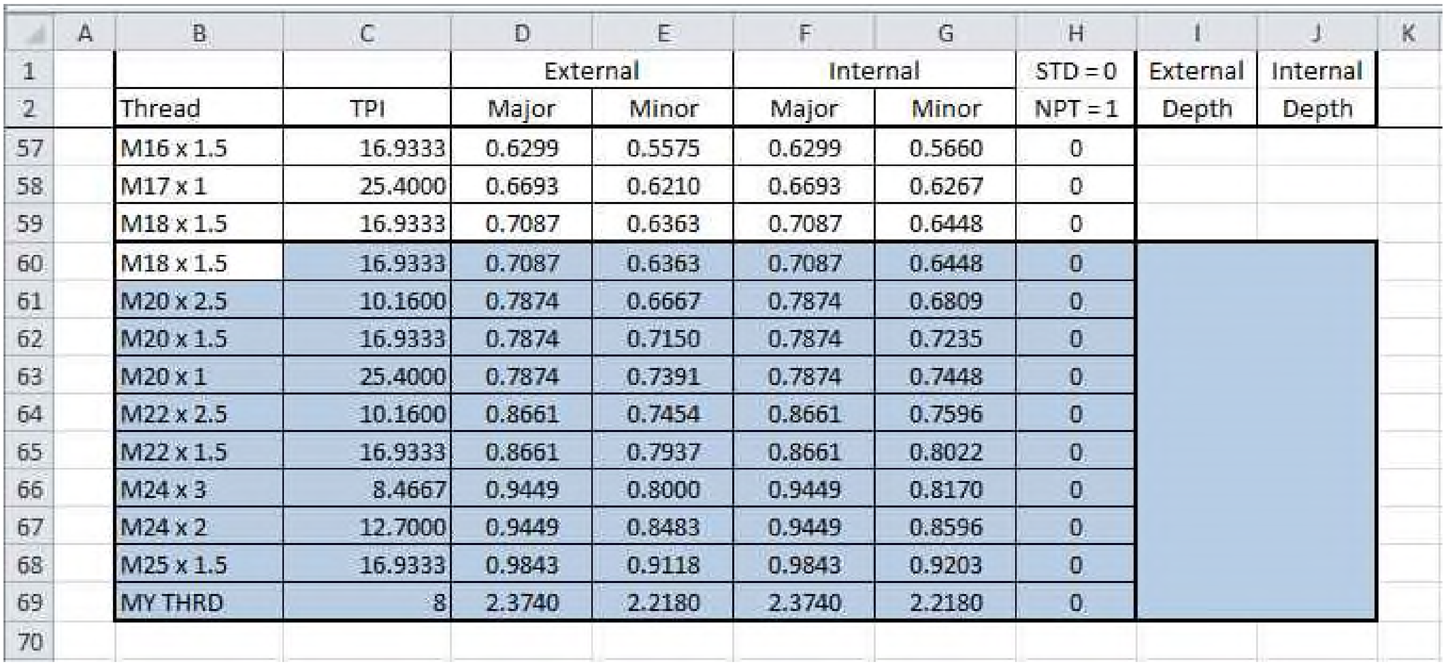
	A	B	C	D	E	F	G	H	I	J	K
1				External		Internal		STD = 0	External	Internal	
2		Thread	TPI	Major	Minor	Major	Minor	NPT = 1	Depth	Depth	
57		M16 x 1.5	16.9333	0.6299	0.5575	0.6299	0.5660	0			
58		M17 x 1	25.4000	0.6693	0.6210	0.6693	0.6267	0			
59		M18 x 1.5	16.9333	0.7087	0.6363	0.7087	0.6448	0			
60		M20 x 2.5	10.1600	0.7874	0.6667	0.7874	0.6809	0			
61		M20 x 1.5	16.9333	0.7874	0.7150	0.7874	0.7235	0			
62		M20 x 1	25.4000	0.7874	0.7391	0.7874	0.7448	0			
63		M22 x 2.5	10.1600	0.8661	0.7454	0.8661	0.7596	0			
64		M22 x 1.5	16.9333	0.8661	0.7937	0.8661	0.8022	0			
65		M24 x 3	8.4667	0.9449	0.8000	0.9449	0.8170	0			
66		M24 x 2	12.7000	0.9449	0.8483	0.9449	0.8596	0			
67		M25 x 1.5	16.9333	0.9843	0.9118	0.9843	0.9203	0			
68		MY THRD	8	2.3740	2.2180	2.3740	2.2180	0			
69											
70											

Next select cell "B60" (or arrow/cursor down one row) .



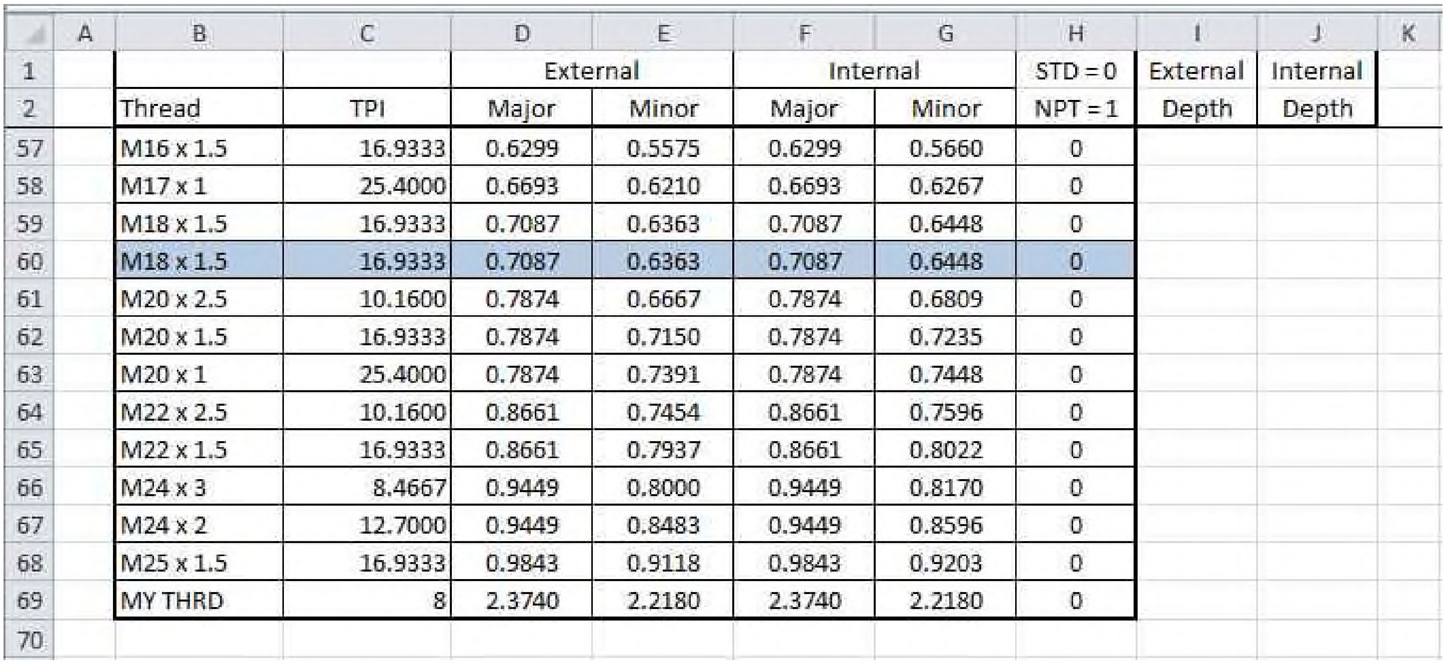
	A	B	C	D	E	F	G	H	I	J	K
1				External		Internal		STD = 0	External	Internal	
2		Thread	TPI	Major	Minor	Major	Minor	NPT = 1	Depth	Depth	
57		M16 x 1.5	16.9333	0.6299	0.5575	0.6299	0.5660	0			
58		M17 x 1	25.4000	0.6693	0.6210	0.6693	0.6267	0			
59		M18 x 1.5	16.9333	0.7087	0.6363	0.7087	0.6448	0			
60		M20 x 2.5	10.1600	0.7874	0.6667	0.7874	0.6809	0			
61		M20 x 1.5	16.9333	0.7874	0.7150	0.7874	0.7235	0			
62		M20 x 1	25.4000	0.7874	0.7391	0.7874	0.7448	0			
63		M22 x 2.5	10.1600	0.8661	0.7454	0.8661	0.7596	0			
64		M22 x 1.5	16.9333	0.8661	0.7937	0.8661	0.8022	0			
65		M24 x 3	8.4667	0.9449	0.8000	0.9449	0.8170	0			
66		M24 x 2	12.7000	0.9449	0.8483	0.9449	0.8596	0			
67		M25 x 1.5	16.9333	0.9843	0.9118	0.9843	0.9203	0			
68		MY THRD	8	2.3740	2.2180	2.3740	2.2180	0			
69											
70											

Next right-click and select "Paste" [left paste button] (or Ctrl-V) to paste selected entries down one row. All rows are "pushed" down by 1 row and row 60 becomes the new row for the new thread spec entry.



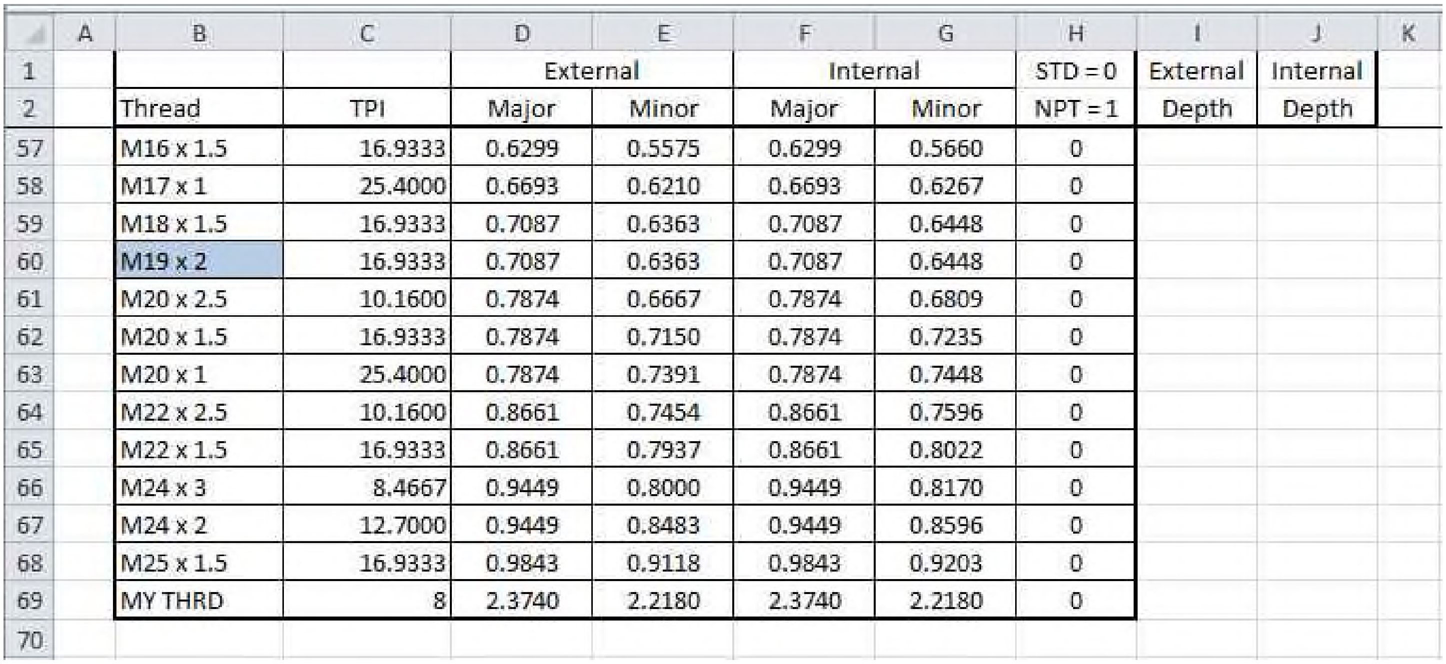
	A	B	C	D	E	F	G	H	I	J	K
1				External		Internal		STD = 0	External	Internal	
2		Thread	TPI	Major	Minor	Major	Minor	NPT = 1	Depth	Depth	
57		M16 x 1.5	16.9333	0.6299	0.5575	0.6299	0.5660	0			
58		M17 x 1	25.4000	0.6693	0.6210	0.6693	0.6267	0			
59		M18 x 1.5	16.9333	0.7087	0.6363	0.7087	0.6448	0			
60		M18 x 1.5	16.9333	0.7087	0.6363	0.7087	0.6448	0			
61		M20 x 2.5	10.1600	0.7874	0.6667	0.7874	0.6809	0			
62		M20 x 1.5	16.9333	0.7874	0.7150	0.7874	0.7235	0			
63		M20 x 1	25.4000	0.7874	0.7391	0.7874	0.7448	0			
64		M22 x 2.5	10.1600	0.8661	0.7454	0.8661	0.7596	0			
65		M22 x 1.5	16.9333	0.8661	0.7937	0.8661	0.8022	0			
66		M24 x 3	8.4667	0.9449	0.8000	0.9449	0.8170	0			
67		M24 x 2	12.7000	0.9449	0.8483	0.9449	0.8596	0			
68		M25 x 1.5	16.9333	0.9843	0.9118	0.9843	0.9203	0			
69		MY THRD	8	2.3740	2.2180	2.3740	2.2180	0			
70											

A few steps back we choose to "Copy" (instead of "Cut") so all settings are carried forward from row 59 to the newly created row 60. Row 60 will inherit all cell formatting from row 59.



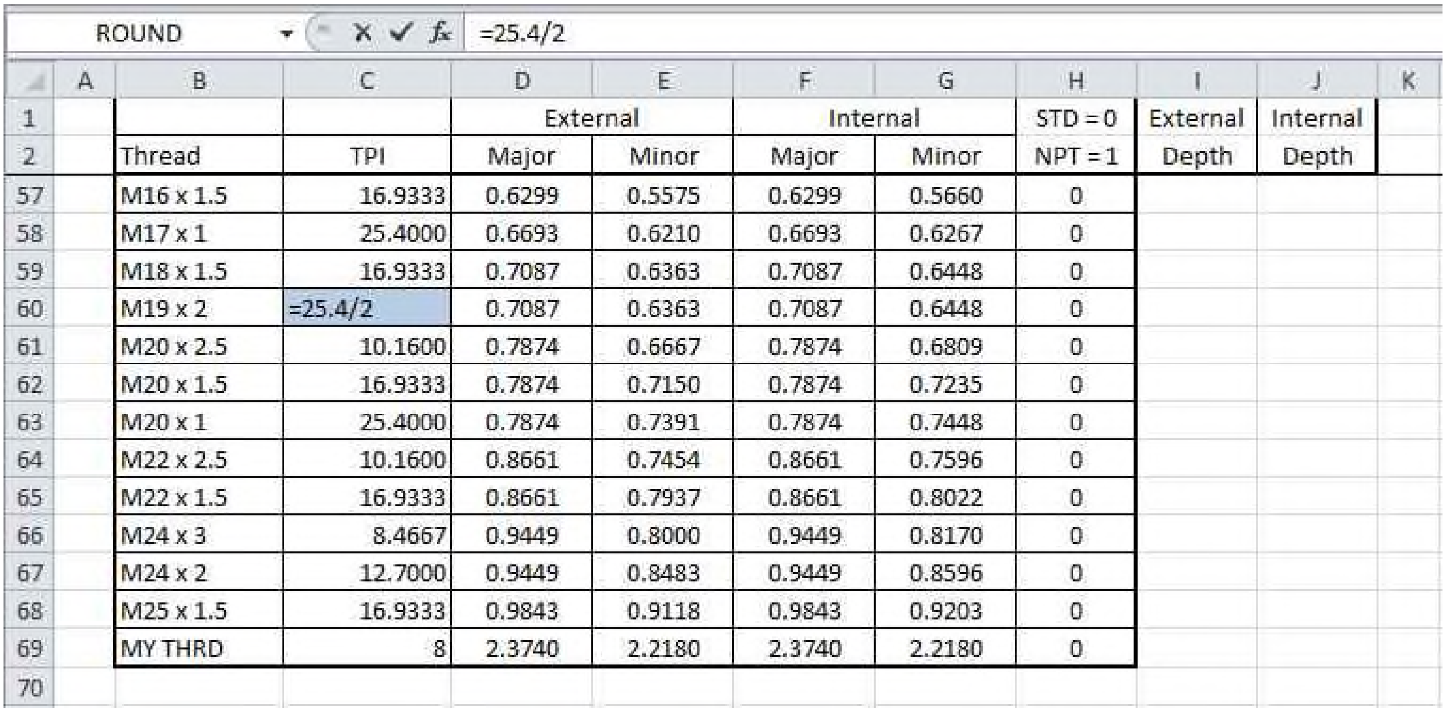
	A	B	C	D	E	F	G	H	I	J	K
1				External		Internal		STD = 0	External	Internal	
2		Thread	TPI	Major	Minor	Major	Minor	NPT = 1	Depth	Depth	
57		M16 x 1.5	16.9333	0.6299	0.5575	0.6299	0.5660	0			
58		M17 x 1	25.4000	0.6693	0.6210	0.6693	0.6267	0			
59		M18 x 1.5	16.9333	0.7087	0.6363	0.7087	0.6448	0			
60		M18 x 1.5	16.9333	0.7087	0.6363	0.7087	0.6448	0			
61		M20 x 2.5	10.1600	0.7874	0.6667	0.7874	0.6809	0			
62		M20 x 1.5	16.9333	0.7874	0.7150	0.7874	0.7235	0			
63		M20 x 1	25.4000	0.7874	0.7391	0.7874	0.7448	0			
64		M22 x 2.5	10.1600	0.8661	0.7454	0.8661	0.7596	0			
65		M22 x 1.5	16.9333	0.8661	0.7937	0.8661	0.8022	0			
66		M24 x 3	8.4667	0.9449	0.8000	0.9449	0.8170	0			
67		M24 x 2	12.7000	0.9449	0.8483	0.9449	0.8596	0			
68		M25 x 1.5	16.9333	0.9843	0.9118	0.9843	0.9203	0			
69		MY THRD	8	2.3740	2.2180	2.3740	2.2180	0			
70											

Next edit the values in the new row 60. First change cell "B60" to M19 x 2.



	A	B	C	D	E	F	G	H	I	J	K
1				External		Internal		STD = 0	External	Internal	
2		Thread	TPI	Major	Minor	Major	Minor	NPT = 1	Depth	Depth	
57		M16 x 1.5	16.9333	0.6299	0.5575	0.6299	0.5660	0			
58		M17 x 1	25.4000	0.6693	0.6210	0.6693	0.6267	0			
59		M18 x 1.5	16.9333	0.7087	0.6363	0.7087	0.6448	0			
60		M19 x 2	16.9333	0.7087	0.6363	0.7087	0.6448	0			
61		M20 x 2.5	10.1600	0.7874	0.6667	0.7874	0.6809	0			
62		M20 x 1.5	16.9333	0.7874	0.7150	0.7874	0.7235	0			
63		M20 x 1	25.4000	0.7874	0.7391	0.7874	0.7448	0			
64		M22 x 2.5	10.1600	0.8661	0.7454	0.8661	0.7596	0			
65		M22 x 1.5	16.9333	0.8661	0.7937	0.8661	0.8022	0			
66		M24 x 3	8.4667	0.9449	0.8000	0.9449	0.8170	0			
67		M24 x 2	12.7000	0.9449	0.8483	0.9449	0.8596	0			
68		M25 x 1.5	16.9333	0.9843	0.9118	0.9843	0.9203	0			
69		MY THRD	8	2.3740	2.2180	2.3740	2.2180	0			
70											

Then change cell "C60" to 12.7 (or in the cell or "formula bar" enter =25.4/2). Old formula was =25.4/1.5 for 1.5mm pitch. Note: Don't forget to add the "**=**" symbol when entering a formula. A 2mm pitch to equivalent to 12.7 **T**hreads **P**er **I**nch.



=25.4/2

	A	B	C	D	E	F	G	H	I	J	K
1				External		Internal		STD = 0	External	Internal	
2		Thread	TPI	Major	Minor	Major	Minor	NPT = 1	Depth	Depth	
57		M16 x 1.5	16.9333	0.6299	0.5575	0.6299	0.5660	0			
58		M17 x 1	25.4000	0.6693	0.6210	0.6693	0.6267	0			
59		M18 x 1.5	16.9333	0.7087	0.6363	0.7087	0.6448	0			
60		M19 x 2	=25.4/2	0.7087	0.6363	0.7087	0.6448	0			
61		M20 x 2.5	10.1600	0.7874	0.6667	0.7874	0.6809	0			
62		M20 x 1.5	16.9333	0.7874	0.7150	0.7874	0.7235	0			
63		M20 x 1	25.4000	0.7874	0.7391	0.7874	0.7448	0			
64		M22 x 2.5	10.1600	0.8661	0.7454	0.8661	0.7596	0			
65		M22 x 1.5	16.9333	0.8661	0.7937	0.8661	0.8022	0			
66		M24 x 3	8.4667	0.9449	0.8000	0.9449	0.8170	0			
67		M24 x 2	12.7000	0.9449	0.8483	0.9449	0.8596	0			
68		M25 x 1.5	16.9333	0.9843	0.9118	0.9843	0.9203	0			
69		MY THRD	8	2.3740	2.2180	2.3740	2.2180	0			
70											

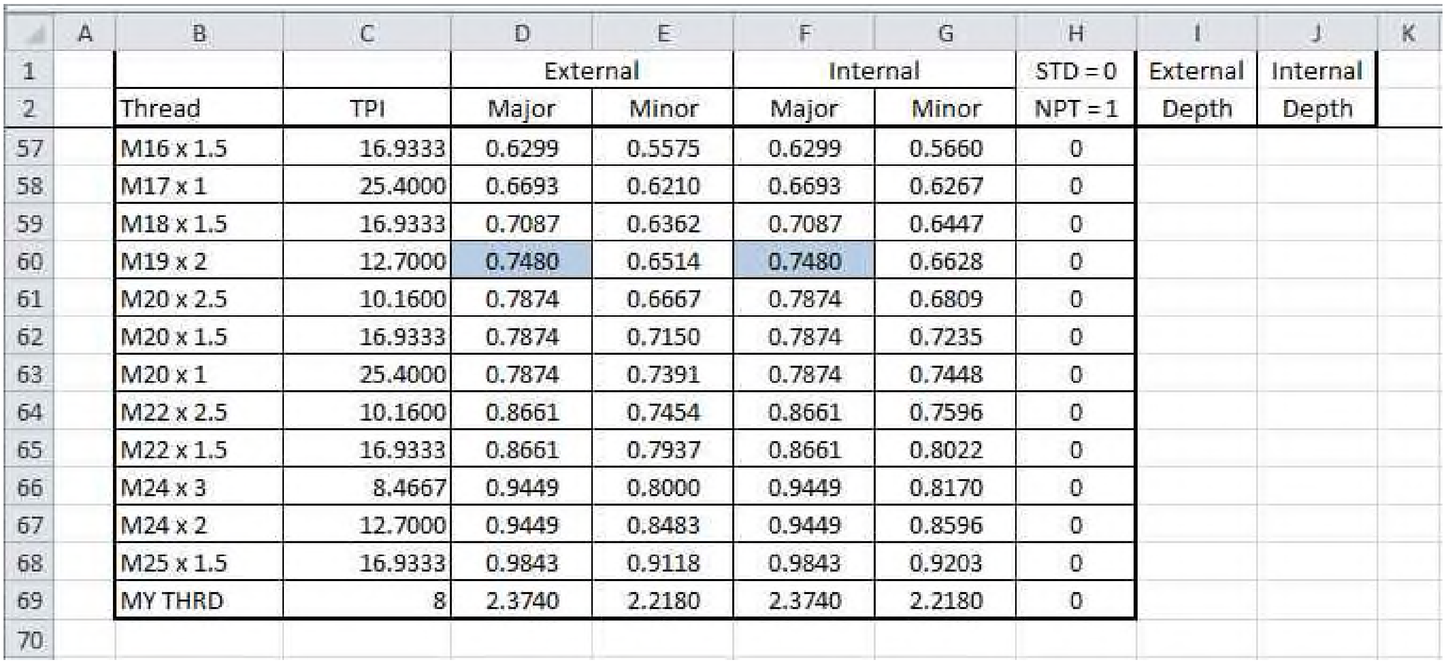
Then change cells "D60" and "F60" to the major diameter values for external & internal threads. 19mm/25.4 = 0.7480". If you are still using the (default) formulas for the minor diameters they will automatically self adjust to the correct values. If not, then enter your new minor diameter values directly into cells "E60" and "G60".

The formula for cell "E60" is =$D60-2*0.7083* SIN (RADIANS(60))/$C60

The formula for cell "G60" is =$F60-2*0.6250*SIN(RADIANS(30))/$C60

The 0.7083 is for 71% thread height and the SIN(RADIANS(60))/$C60 is the full thread height.

The $D60-2* means external major diameter – 2 times (the xx% of thread height) to obtain external minor diameter.



	A	B	C	D	E	F	G	H	I	J	K
1				External		Internal		STD = 0	External	Internal	
2		Thread	TPI	Major	Minor	Major	Minor	NPT = 1	Depth	Depth	
57		M16 x 1.5	16.9333	0.6299	0.5575	0.6299	0.5660	0			
58		M17 x 1	25.4000	0.6693	0.6210	0.6693	0.6267	0			
59		M18 x 1.5	16.9333	0.7087	0.6362	0.7087	0.6447	0			
60		M19 x 2	12.7000	0.7480	0.6514	0.7480	0.6628	0			
61		M20 x 2.5	10.1600	0.7874	0.6667	0.7874	0.6809	0			
62		M20 x 1.5	16.9333	0.7874	0.7150	0.7874	0.7235	0			
63		M20 x 1	25.4000	0.7874	0.7391	0.7874	0.7448	0			
64		M22 x 2.5	10.1600	0.8661	0.7454	0.8661	0.7596	0			
65		M22 x 1.5	16.9333	0.8661	0.7937	0.8661	0.8022	0			
66		M24 x 3	8.4667	0.9449	0.8000	0.9449	0.8170	0			
67		M24 x 2	12.7000	0.9449	0.8483	0.9449	0.8596	0			
68		M25 x 1.5	16.9333	0.9843	0.9118	0.9843	0.9203	0			
69		MY THRD	8	2.3740	2.2180	2.3740	2.2180	0			
70											

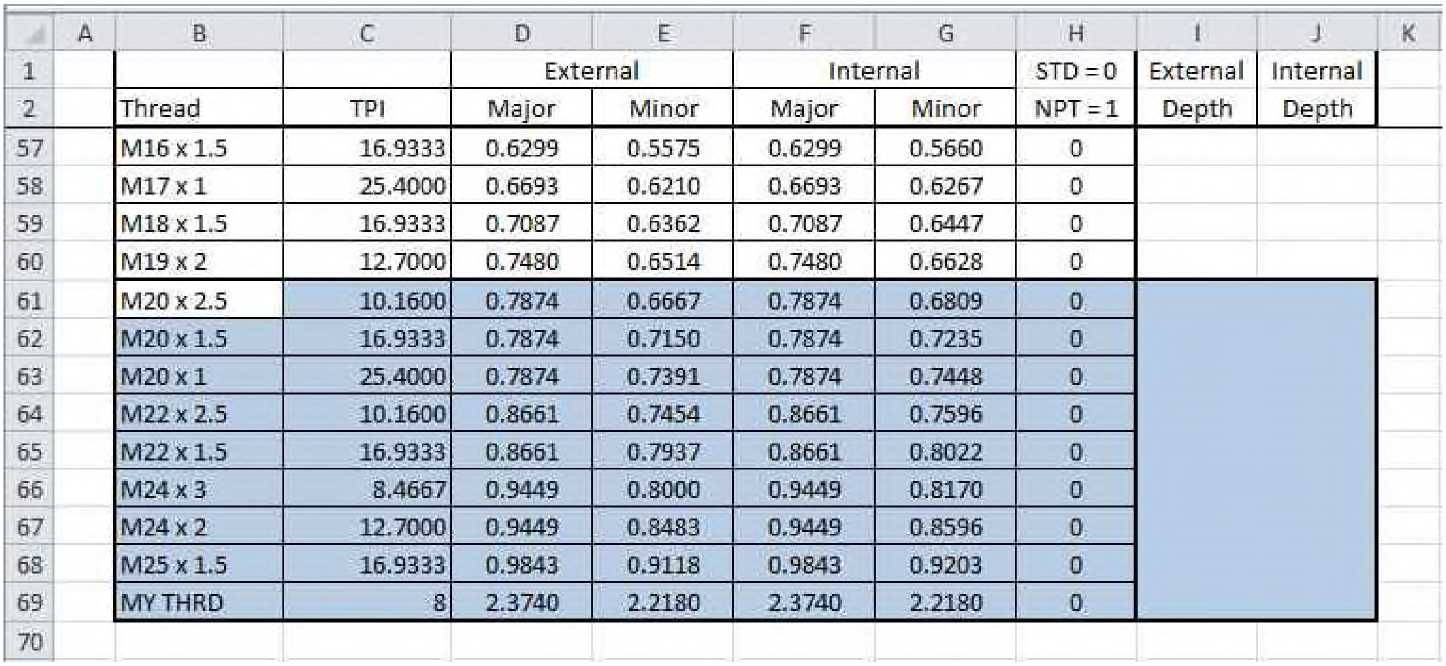
You can modify the formatting of the numbers. For example cell "C68" above shows 4 decimal places (to the right of the decimal point) and cell "C69" shows 0 decimal places. To modify the number of decimal places, select the cell(s) to modify, right-click, select "Format Cells", click the "Number" tab then (if needed click on "Number" under "Category:") then enter a new value for the number of decimal places (or click the little up or down arrows to the right of "Decimal places:").

You can also modify the borders if necessary to keep the thread spec entries in organized easy to read groups. Select the cell(s) to modify, right-click, select "Format Cells", click the "Border" tab then select line styles and borders at the lower right.

There is a remote chance you may need to modify the formatting of (thread spec description) column "B". If you enter something like **1/2**-20. Excel may treat the "**/**" as a "Date" format and display something like **1/2**/2020. If so, simply change the cell in column "B" back to "Text" formatting. Select the cell(s) to modify, right-click, select "Format Cells", click the "Number" tab then click on "Text" (under "Category:") . Then re-enter your thread spec description.

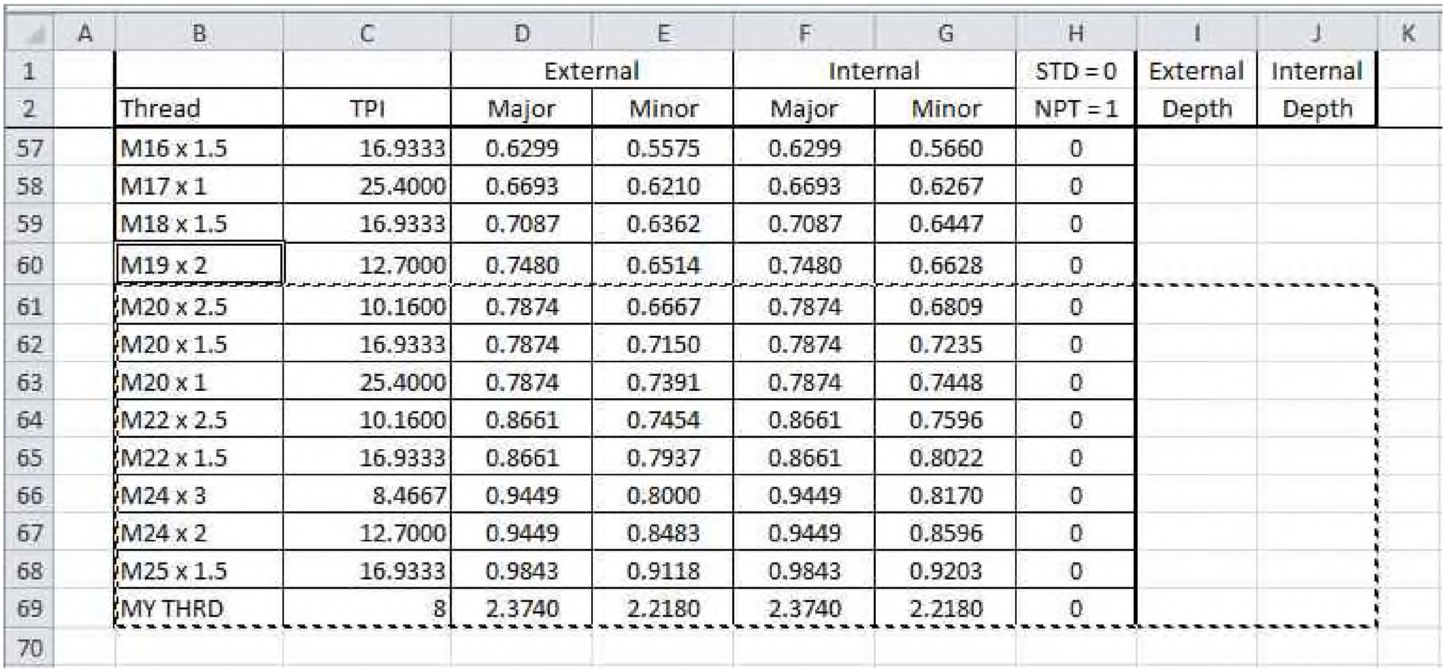
To delete an existing thread spec entry from list:

In this example we are removing the fictitious M19 x 1 thread spec entry in row 60. Select cells "B61" through "J69" (to end of list) as shown. Now right-click and select "Cut" (or Ctrl-X) to cut selected entries.



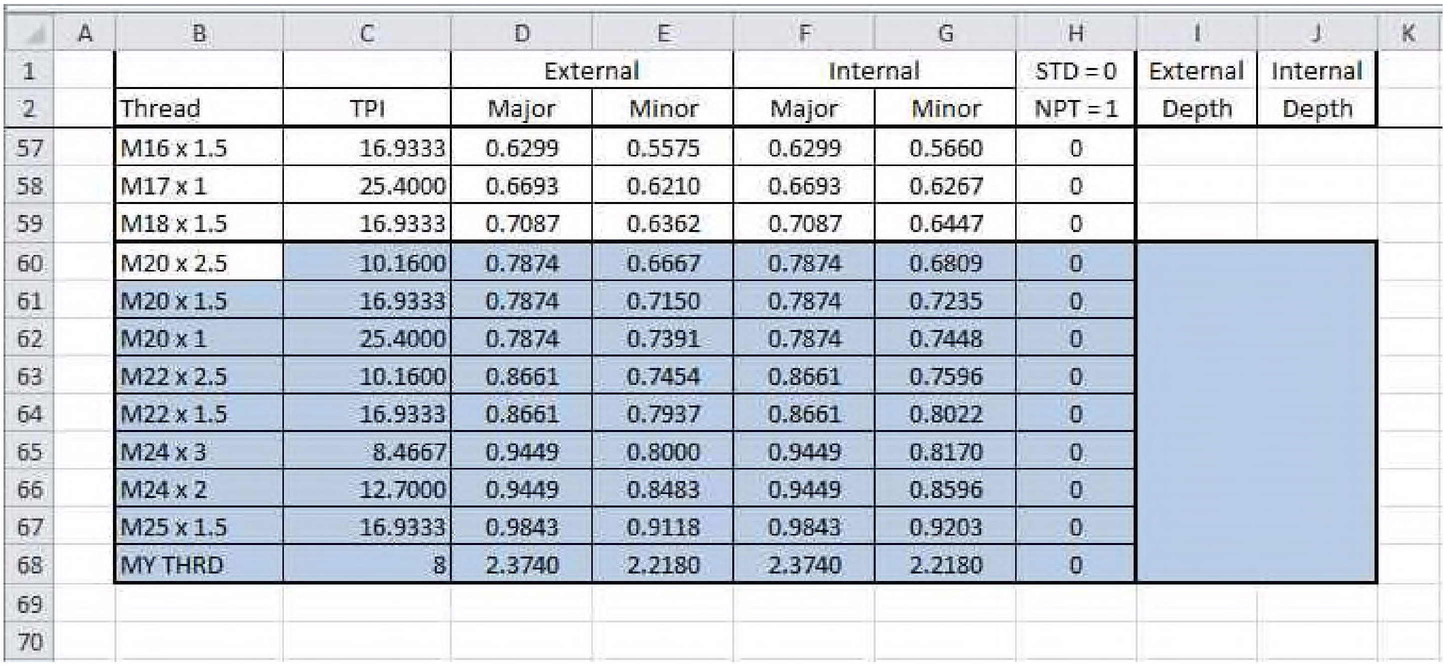
	A	B	C	D	E	F	G	H	I	J	K
1				External		Internal		STD = 0	External	Internal	
2		Thread	TPI	Major	Minor	Major	Minor	NPT = 1	Depth	Depth	
57		M16 x 1.5	16.9333	0.6299	0.5575	0.6299	0.5660	0			
58		M17 x 1	25.4000	0.6693	0.6210	0.6693	0.6267	0			
59		M18 x 1.5	16.9333	0.7087	0.6362	0.7087	0.6447	0			
60		M19 x 2	12.7000	0.7480	0.6514	0.7480	0.6628	0			
61		M20 x 2.5	10.1600	0.7874	0.6667	0.7874	0.6809	0			
62		M20 x 1.5	16.9333	0.7874	0.7150	0.7874	0.7235	0			
63		M20 x 1	25.4000	0.7874	0.7391	0.7874	0.7448	0			
64		M22 x 2.5	10.1600	0.8661	0.7454	0.8661	0.7596	0			
65		M22 x 1.5	16.9333	0.8661	0.7937	0.8661	0.8022	0			
66		M24 x 3	8.4667	0.9449	0.8000	0.9449	0.8170	0			
67		M24 x 2	12.7000	0.9449	0.8483	0.9449	0.8596	0			
68		M25 x 1.5	16.9333	0.9843	0.9118	0.9843	0.9203	0			
69		MY THRD	8	2.3740	2.2180	2.3740	2.2180	0			
70											

Next select cell "B60" (or arrow/cursor up one row) .



	A	B	C	D	E	F	G	H	I	J	K
1				External		Internal		STD = 0	External	Internal	
2		Thread	TPI	Major	Minor	Major	Minor	NPT = 1	Depth	Depth	
57		M16 x 1.5	16.9333	0.6299	0.5575	0.6299	0.5660	0			
58		M17 x 1	25.4000	0.6693	0.6210	0.6693	0.6267	0			
59		M18 x 1.5	16.9333	0.7087	0.6362	0.7087	0.6447	0			
60		M19 x 2	12.7000	0.7480	0.6514	0.7480	0.6628	0			
61		M20 x 2.5	10.1600	0.7874	0.6667	0.7874	0.6809	0			
62		M20 x 1.5	16.9333	0.7874	0.7150	0.7874	0.7235	0			
63		M20 x 1	25.4000	0.7874	0.7391	0.7874	0.7448	0			
64		M22 x 2.5	10.1600	0.8661	0.7454	0.8661	0.7596	0			
65		M22 x 1.5	16.9333	0.8661	0.7937	0.8661	0.8022	0			
66		M24 x 3	8.4667	0.9449	0.8000	0.9449	0.8170	0			
67		M24 x 2	12.7000	0.9449	0.8483	0.9449	0.8596	0			
68		M25 x 1.5	16.9333	0.9843	0.9118	0.9843	0.9203	0			
69		MY THRD	8	2.3740	2.2180	2.3740	2.2180	0			
70											

Next right-click and select "Paste" [left paste button] (or Ctrl-V) to paste selected entries up one row. Now the M19 x 1 entry has been removed from the thread specs list. A few steps back we choose to "Cut" (instead of "Copy") causing all the entries to carry upward so there aren't 2 copies of the last entry at the end of the list.



	A	B	C	D	E	F	G	H	I	J	K
1				External		Internal		STD = 0	External	Internal	
2		Thread	TPI	Major	Minor	Major	Minor	NPT = 1	Depth	Depth	
57		M16 x 1.5	16.9333	0.6299	0.5575	0.6299	0.5660	0			
58		M17 x 1	25.4000	0.6693	0.6210	0.6693	0.6267	0			
59		M18 x 1.5	16.9333	0.7087	0.6362	0.7087	0.6447	0			
60		M20 x 2.5	10.1600	0.7874	0.6667	0.7874	0.6809	0			
61		M20 x 1.5	16.9333	0.7874	0.7150	0.7874	0.7235	0			
62		M20 x 1	25.4000	0.7874	0.7391	0.7874	0.7448	0			
63		M22 x 2.5	10.1600	0.8661	0.7454	0.8661	0.7596	0			
64		M22 x 1.5	16.9333	0.8661	0.7937	0.8661	0.8022	0			
65		M24 x 3	8.4667	0.9449	0.8000	0.9449	0.8170	0			
66		M24 x 2	12.7000	0.9449	0.8483	0.9449	0.8596	0			
67		M25 x 1.5	16.9333	0.9843	0.9118	0.9843	0.9203	0			
68		MY THRD	8	2.3740	2.2180	2.3740	2.2180	0			
69											
70											

"Radial_Passes" sheet

The "Radial_Passes" sheet contains the "Radial Thread Depth" table used for multiple radial thread passes. The features within this table of radial thread milling passes have been greatly enhanced in this deluxe version of the software.

To take multiple radial thread passes you can enter “**Number of Passes**” and “**Radial Depth Factor**” (RDF). Both of these parameters exist here (“Radial_Passes" sheet) and on the “Main_Code” sheet. These parameters can be modified on either sheet and the corresponding parameter on the other sheet will update automatically.

Multiple radial thread passes works with all "Single/Multiple" settings ("SINGLE", "MULTIPLE", "MULT FULL" and "USE AXIAL") .

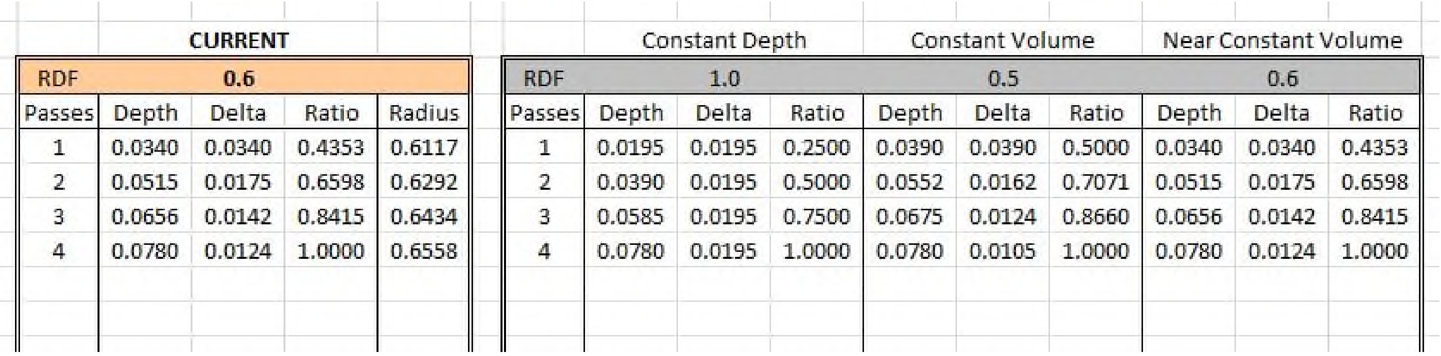
"Number of Passes" can be up to 24 radial thread milling passes.
The “Radial Depth Factor” (RDF) value must be between 0.5 and 1.0.
The RDF is used to control the progression of thread passes towards full radial thread depth.

RDF value = 1.0 = Constant Depth
RDF value = 0.5 = Constant Volume (for basically any 60 degree thread form)
RDF value = 0.6 = Nearly Constant Volume (Usually works best)
RDF value = 0.83 = Constant Volume (0.83 is the approximate value for any 29 degree Acme thread form)

Constant depth can cause the first few passes to be too light and the last few passes to be too heavy.
Constant volume can cause the last few passes to be too light.
I have used RDF = 0.6 quite a lot and it seems to be a very good balance of radial thread depths.

If I use a large number of passes (very course thread), I may tweak the RDF value to 0.7 or higher.
This ensures the last few passes are not too light.

The below example is for .078” thread depth (radial engagement) with 4 passes. It shows how the RDF value (set to 0.6) controls the thread depths for each pass. The table contains rows of Pass numbers,"Depth" (current depth so far), "Delta" (incremental depth of current pass), "Ratio" (ratio of how far to full depth so far with 1.0 representing full depth) and "Radius” (being the current pass programmed radius in the program). Notice the decreasing depths from one pass to the next on the left live working table.



	CURRENT			
RDF	0.6			
Passes	Depth	Delta	Ratio	Radius
1	0.0340	0.0340	0.4353	0.6117
2	0.0515	0.0175	0.6598	0.6292
3	0.0656	0.0142	0.8415	0.6434
4	0.0780	0.0124	1.0000	0.6558

	Constant Depth			Constant Volume			Near Constant Volume		
RDF	1.0			0.5			0.6		
Passes	Depth	Delta	Ratio	Depth	Delta	Ratio	Depth	Delta	Ratio
1	0.0195	0.0195	0.2500	0.0390	0.0390	0.5000	0.0340	0.0340	0.4353
2	0.0390	0.0195	0.5000	0.0552	0.0162	0.7071	0.0515	0.0175	0.6598
3	0.0585	0.0195	0.7500	0.0675	0.0124	0.8660	0.0656	0.0142	0.8415
4	0.0780	0.0195	1.0000	0.0780	0.0105	1.0000	0.0780	0.0124	1.0000

The "Radial_Passes" table will show the above(left) table automatically populated with all of the data based on the settings of “Number of Passes” and “Radial Depth Factor” on the "Main_Code" sheet. The software would then use these 4 "Depth" values to **automatically** create the multiple radial threading passes and NC code on the "Main_Code" sheet. Here you see how the "RDF value" affect the various radial depths of cut for each pass. Also shown strictly as a reference are the 3 (right tables) most common RDF settings. With "Constant Volume" and "Near Constant Volume" you can see the thread passes getting progressive lighter at the tool approaches full depth. The "Number of Passes" and the "Radial Depth Factor" parameters give you a lot of control over the thread milling NC code.

The "Main_Code" sheet uses the radial thread depth passes from the "Radial_Passes" sheet, which includes the current + new + clean-up pass values. You can have up to a total of 24 radial thread passes plus any "clean-up passes" from the "Main_Code" sheet.

You can edit any blue, green or yellow cells.

If needed, **first** edit the **blue cells first** (which are "Number of Passes" and "Radial Depth Factor"). Since they clear out any edits to green cells ("new" passes) and yellow cells ("modified" passes).

Secondly, edit **green cells** if new clean-up passes are needed. Green cell (clean-up pass) values can even be 0, which would add to the number of clean-up passes (if some clean-up passes were already entered on the "Main_Code" sheet). Adding new clean-up passes "pushes" the previous passes to smaller depths, since this is necessary to preserve the total radial depth of cut. The addition of new passes must be done in the "Delta" column of the table.

Thirdly, you can modify **yellow cells** (if needed) to re-balance the pre-existing depths of cut. Changes to pre-existing passes must be done in the "Depth" or "Ratio" columns of the table. If any yellow cells are modified, the current radial depth factor (RDF) is no longer in effect.

Note: Modifying blue cells ("Number of Passes" and "Radial Depth Factor") after green or yellow cell changes were made, causes loss of green and yellow cell modifications and forces yellow cell passes back to standard Radial Depth Factor values.

Note: Modifying green cells (added passes) after yellow cell changes were made, causes loss of yellow cell changes and forces yellow cell passes back to standard Radial Depth Factor values.

On the "Main_Code" sheet "Number of Passes" will display "**Number of Passes**" if no radial depth modifications were made on the "Radial_Passes" sheet.

Or "Number of Passes" will display "**Number of Passes [+0]**" meaning modifications were made to existing (yellow cells) passes but NO new (green cells) thread passes were added on the "Radial_Passes" sheet.

Or "Number of Passes" will display "**Number of Passes [+3]**" meaning (3 for example) new passes were added with possible modifications to existing thread passes.

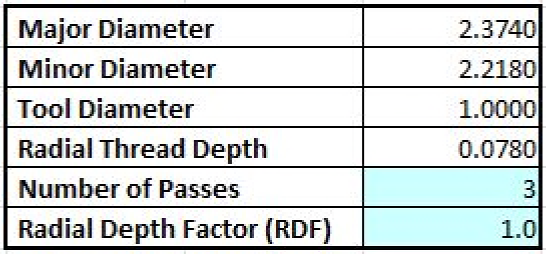
It is best to set the values on this sheet last, after all of the "Main_Code" sheet settings are completed.
If on the "Main_Code" sheet changes are made to the "Major Diameter", "Minor Diameter", "Number of Passes", or "Radial Depth Factor" cells OR changes to "Thread Spec" drop-down list box then all ("Radial_Passes" sheet) modifications to existing passes are **cleared out** and all new passes are removed.

On the "Radial_Passes" sheet, you can at any time, double-click on cell "H10" to remove all modifications to existing passes and remove all new passes. The data will revert back to the default values dictated by the "Number of Passes" and "Radial Depth Factor" value.

By using the "Number of Passes" and "Radial Depth Factor" settings (to get the thread passes correct or close to what you want) along with the ability to add any clean-up passes (of any depths) and/or the "tweaking" of pre-existing passes gives you infinite control of the thread milling operation. Sometimes it can be as simple as choosing 1 or 2 thread passes on the "Main_Code" sheet and then adding just 1 new clean-up pass of 0.005" or so for the last thread pass. This works particularly well on hard materials.

There are a few validation checks performed on this Sheet. The software checks for current pass depth being greater than full radial thread depth, if so it just reverts back to previous number. It checks for missing data (blank cells) where there should be a value entered, software simply auto re-enters value for you (without warning you of error). It also checks for the current threading pass being less than the previous pass (this check is performed only when leaving this sheet to make it easier to edit and adjust thread depth values without repeatedly getting error or warnings).

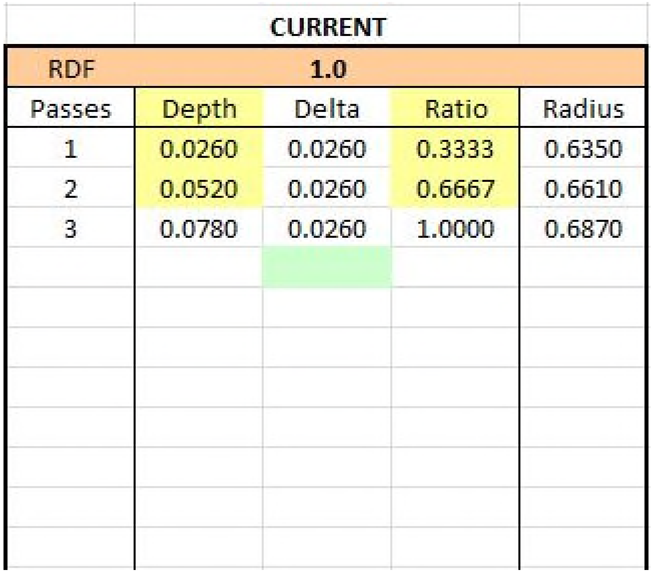
For a handy reference these parameters are displayed in the upper left corner of this sheet.



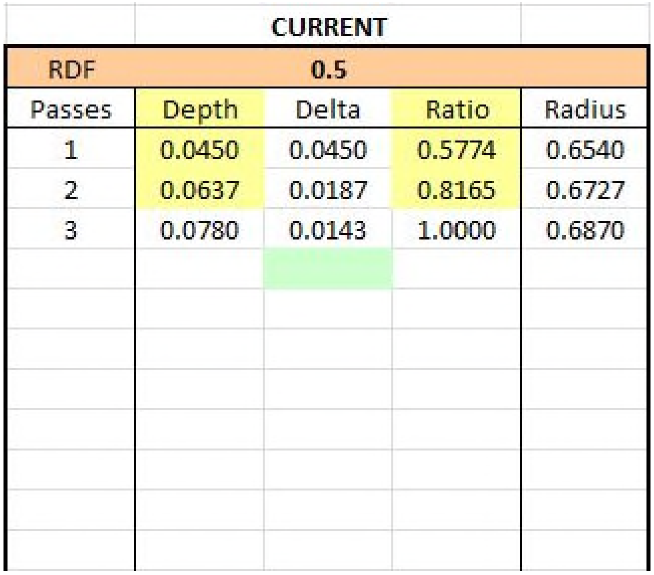
Major Diameter	2.3740
Minor Diameter	2.2180
Tool Diameter	1.0000
Radial Thread Depth	0.0780
Number of Passes	3
Radial Depth Factor (RDF)	1.0

Below left table illustrates a sample on how this sheet would look (automatically populated with these values) when first selecting it. It shows parameters already preset from the "Main_Code" sheet. "Number of Passes" = 3, the total depth of cut = 0.078" and "Radial Depth Factor" = 1.0 (constant depth passes). The total radial depth of cut value is derived from (Major Diameter – Minor Diameter)/2. In the left table, each threading pass (cut) removes 0.026" radially. Below right example shows same total depth with RDF set to 0.5 (constant volume passes). The software would use these 3 (left **OR** right) "Depth" values to **automatically** create the multiple radial threading passes and NC code on the "Main_Code" sheet. Displaying both “CURRENT” tables side by side (one with RDF = 1.0 and one with RDF = 0.5) is for illustration purposes to show the similarities and differences between the two RDF settings. Normally within Excel you would only see one of these “CURRENT” tables.

When you add a clean-up pass or modify a Depth, Delta or Ratio value, the software **instantly and automatically** adjusts all necessary surrounding Depth, Delta, Ratio and Radius values along with the NC code on the “Main_Code” sheet.

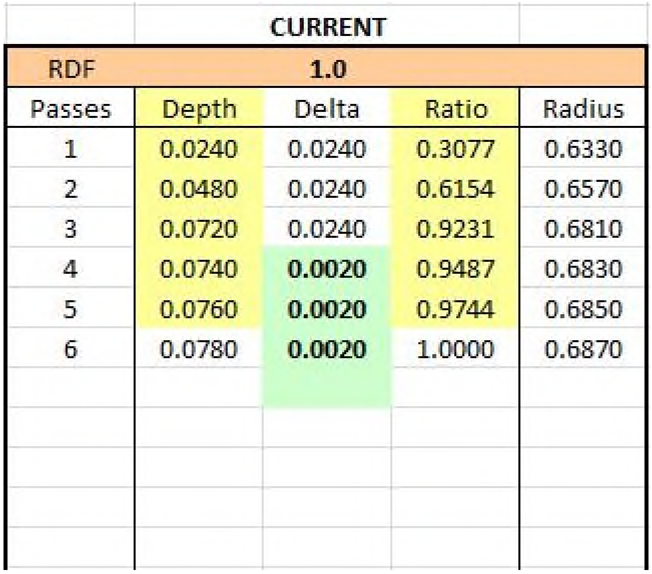


		CURRENT		
RDF		1.0		
Passes	Depth	Delta	Ratio	Radius
1	0.0260	0.0260	0.3333	0.6350
2	0.0520	0.0260	0.6667	0.6610
3	0.0780	0.0260	1.0000	0.6870

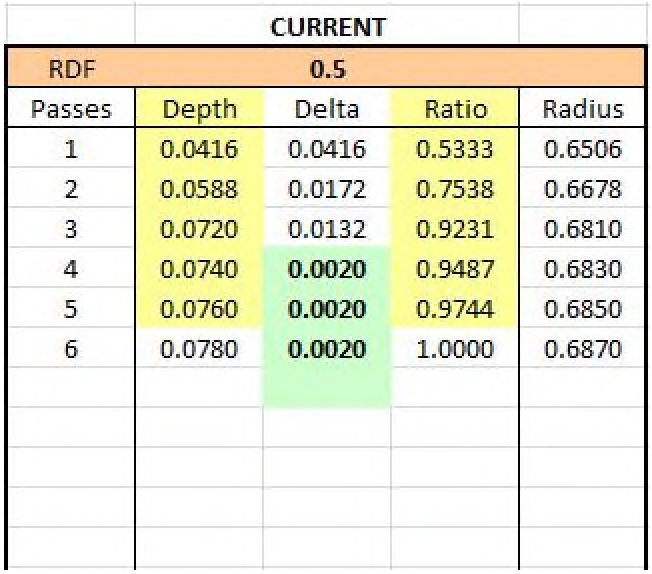


		CURRENT		
RDF		0.5		
Passes	Depth	Delta	Ratio	Radius
1	0.0450	0.0450	0.5774	0.6540
2	0.0637	0.0187	0.8165	0.6727
3	0.0780	0.0143	1.0000	0.6870

Next you decide (for example) to add 3 clean-up passes of 0.002" each. Notice how in both examples the software "pushes" the first 3 passes from 0.078" to 0.072" to make "room" for the 3 new passes. On the left, each of the first 3 passes are "pushed" by an equal amount (0.002") because of RDF = 1.0 (constant depth per pass). On the right, where RDF = 0.5 (constant volume), each of the first 3 passes are "pushed" by their respective constant volume amounts.

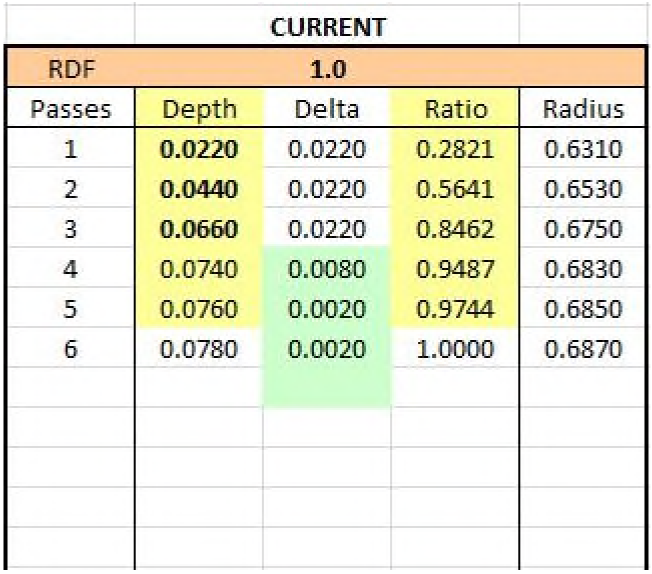


	CURRENT			
RDF	1.0			
Passes	Depth	Delta	Ratio	Radius
1	0.0240	0.0240	0.3077	0.6330
2	0.0480	0.0240	0.6154	0.6570
3	0.0720	0.0240	0.9231	0.6810
4	0.0740	**0.0020**	0.9487	0.6830
5	0.0760	**0.0020**	0.9744	0.6850
6	0.0780	**0.0020**	1.0000	0.6870



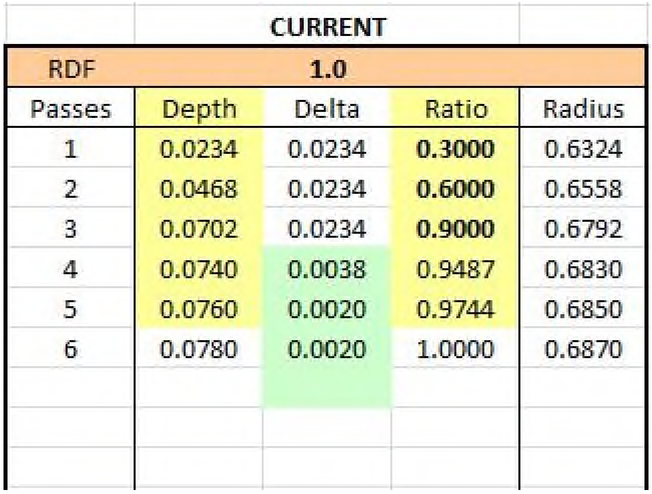
	CURRENT			
RDF	0.5			
Passes	Depth	Delta	Ratio	Radius
1	0.0416	0.0416	0.5333	0.6506
2	0.0588	0.0172	0.7538	0.6678
3	0.0720	0.0132	0.9231	0.6810
4	0.0740	**0.0020**	0.9487	0.6830
5	0.0760	**0.0020**	0.9744	0.6850
6	0.0780	**0.0020**	1.0000	0.6870

Next you decide to "tweak" the first 3 passes to 0.022" each by changing the "Depth" column cells to 0.022", 0.044" and 0.066". The 4th pass is then auto adjusted to force the integrity of full depth being = 0.078" and preserves the values of the last 2 added clean-up passes.



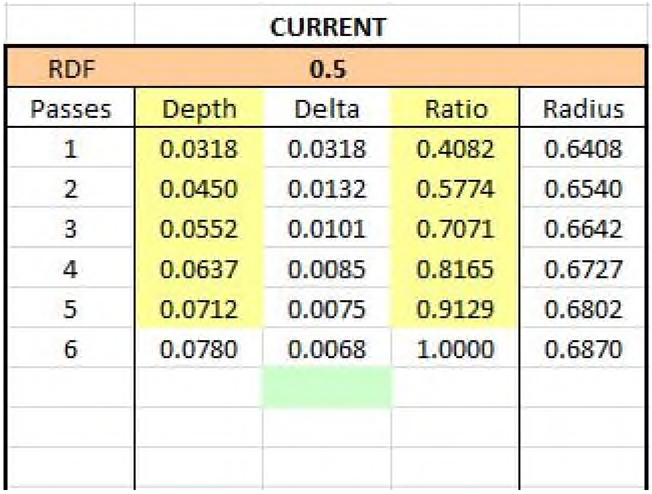
	CURRENT			
RDF	1.0			
Passes	Depth	Delta	Ratio	Radius
1	**0.0220**	0.0220	0.2821	0.6310
2	**0.0440**	0.0220	0.5641	0.6530
3	**0.0660**	0.0220	0.8462	0.6750
4	0.0740	0.0080	0.9487	0.6830
5	0.0760	0.0020	0.9744	0.6850
6	0.0780	0.0020	1.0000	0.6870

Next you decide to "tweak" the first 3 passes by making them equal to 30%, 60% and 90% of full thread depth by modifying cells in the "Ratio" column. Again the 4th pass is then auto adjusted to force the integrity of full depth being equal to 0.078".

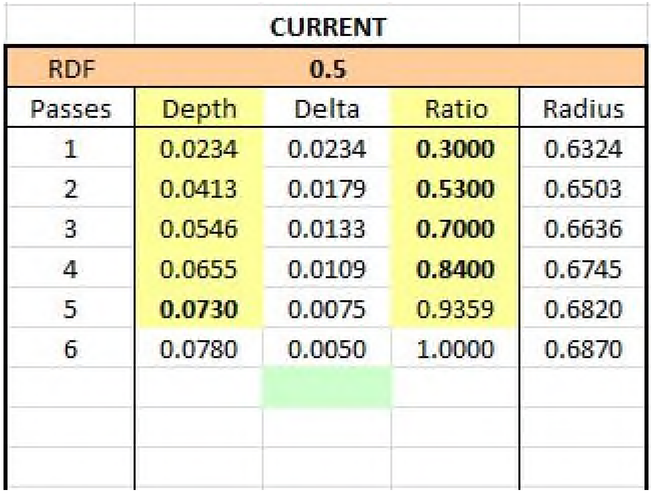


	CURRENT			
RDF	1.0			
Passes	Depth	Delta	Ratio	Radius
1	0.0234	0.0234	**0.3000**	0.6324
2	0.0468	0.0234	**0.6000**	0.6558
3	0.0702	0.0234	**0.9000**	0.6792
4	0.0740	0.0038	0.9487	0.6830
5	0.0760	0.0020	0.9744	0.6850
6	0.0780	0.0020	1.0000	0.6870

This below twin examples show no clean-up passes. When entering this sheet, the "Radial Depth Factor" was set to 0.5 with it's corresponding automatically populated **default** left table values, but the right table has been pretty much completely over-written with **user defined** values. The 5th "Depth" column value was set to 0.073" to force the 6th (last) pass to be a 0.005" depth of cut. Which is basically equivalent to adding a 0.005" clean-up pass. Then the 1st through the 4th "Ratio" column values were set to various values to control the progressively smaller depths of cut. This example would work great for very large radial thread depths (very course threads).



	CURRENT			
RDF	0.5			
Passes	Depth	Delta	Ratio	Radius
1	0.0318	0.0318	0.4082	0.6408
2	0.0450	0.0132	0.5774	0.6540
3	0.0552	0.0101	0.7071	0.6642
4	0.0637	0.0085	0.8165	0.6727
5	0.0712	0.0075	0.9129	0.6802
6	0.0780	0.0068	1.0000	0.6870



	CURRENT			
RDF	0.5			
Passes	Depth	Delta	Ratio	Radius
1	0.0234	0.0234	**0.3000**	0.6324
2	0.0413	0.0179	**0.5300**	0.6503
3	0.0546	0.0133	**0.7000**	0.6636
4	0.0655	0.0109	**0.8400**	0.6745
5	**0.0730**	0.0075	0.9359	0.6820
6	0.0780	0.0050	1.0000	0.6870

This utility does a great job of blending software generated passes (constant depth or volume or variation of) with user added clean-up passes . The user has the freedom to modify any software generated passes by various means. Through the use of the Depth, Delta and Ratio values you clearly see the intention of what each threading pass does in relation to the entire group of radial threading passes.

Having the ability to modify the parameters "Number of Passes" and "Radial Depth Factor" along with the nearly infinite possible "tweaking" of the individual passes (and/or adding clean-up passes) gives you complete control over the cutter passes for any thread milling application from standard 60 degree "V" threads to "ACME" threads and more.

"Axial_Passes" sheet

The "Axial_Passes" sheet contains the "Axial Thread Depth" table. **This table of multiple axial thread milling passes is unique to this deluxe version of the software**.

Multiple axial thread passes are **only possible** when, on the "Main_Code" sheet, parameter "Single/Multiple" is set to "**USE AXIAL**".

This table is used for when axial part thread depth is greater than the height of the multi-tooth thread mill insert.

This feature only works for threads that are a maximum of 24 threads deep.

You can have up to a total of 24 axial thread passes (one for each pitch/lead).

The software checks if already over maximum of 24 threads deep when selecting "Use Axial" on the "Main_Code" sheet. You will be prompted to address (threading depth is too deep) issue before allowing access to the "Axial_Passes" sheet (multiple axial thread passes table).

It is best to set the values on this sheet last, after all of the "Main_Code" sheet settings are completed.

If the "Thread Depth", "Threads per Inch", "Pitch", "Number of Starts" or "Thread Spec" value changes on the "Main_Code" sheet then additional axial depths are **cleared out** here and need to be re-selected.

You can select where (and in what order) the additional axial thread passes will be positioned.

Starting with 1, place a unique number at any depth to dictate order of cutting passes.

Typically you would select axial thread passes such that they would progress up out of the hole for straight threads, as to lessen the chance of chip re-cutting. You may also want to set the "Main_Code" sheet "Thread Direction" to "UP", so within each axial cut there is also upwards motion. For tapered threads, it may make sense to progress downwards for each axial thread pass.

You choose axial passes such that the cutter progresses down (or up) the hole, which is separate from threading up or downwards **within** each individual cut (within the "Thread Direction" setting).

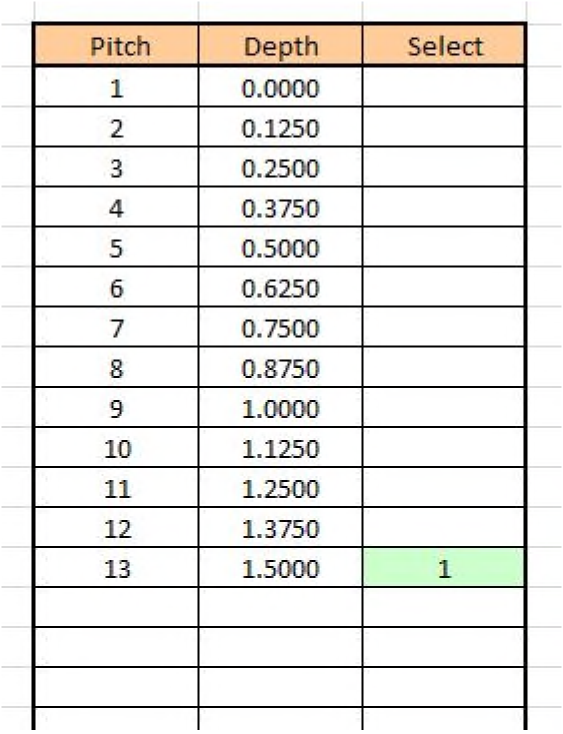
Note: Full axial depth pass (last in list) must contain a number.

Allow for some cutter overlap when selecting additional passes.

You can use axial thread passes for NPT tapered threads, the software will adjust the programmed radius (arc) based on the current axial depth.

For this sheet, the software checks for invalid data in the "Select" column. It also (upon leaving this sheet) warns you if you did not select an axial threading pass at full depth.

Default display when selecting "Axial Thread Passes" sheet. It show one axial pass at full thread depth.

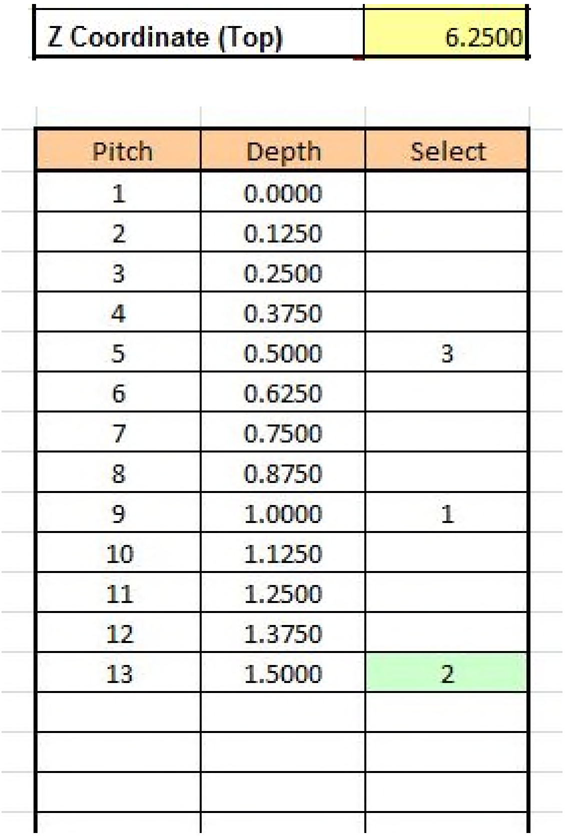


Pitch	Depth	Select
1	0.0000	
2	0.1250	
3	0.2500	
4	0.3750	
5	0.5000	
6	0.6250	
7	0.7500	
8	0.8750	
9	1.0000	
10	1.1250	
11	1.2500	
12	1.3750	
13	1.5000	1

For this below example, you could change the 13th thread (last pitch) to be cut 2nd in the order of thread milling, then make the 9th thread (down the hole) to be cut 1st and the 5th thread could be cut 3rd. This example is assuming the insert height is about 0.700" and then requires 3 thread passes to complete the entire thread. Most likely you would thread mill the axial threading passes from top down or bottom up; the 5th thread being the 1st pass, 9th thread being the 2nd pass and 13th thread being the 3rd pass.

The software would use these axial pass values on this "Axial_Passes" sheet to **automatically** create the multiple axial threading passes and NC code on the "Main_Code" sheet.

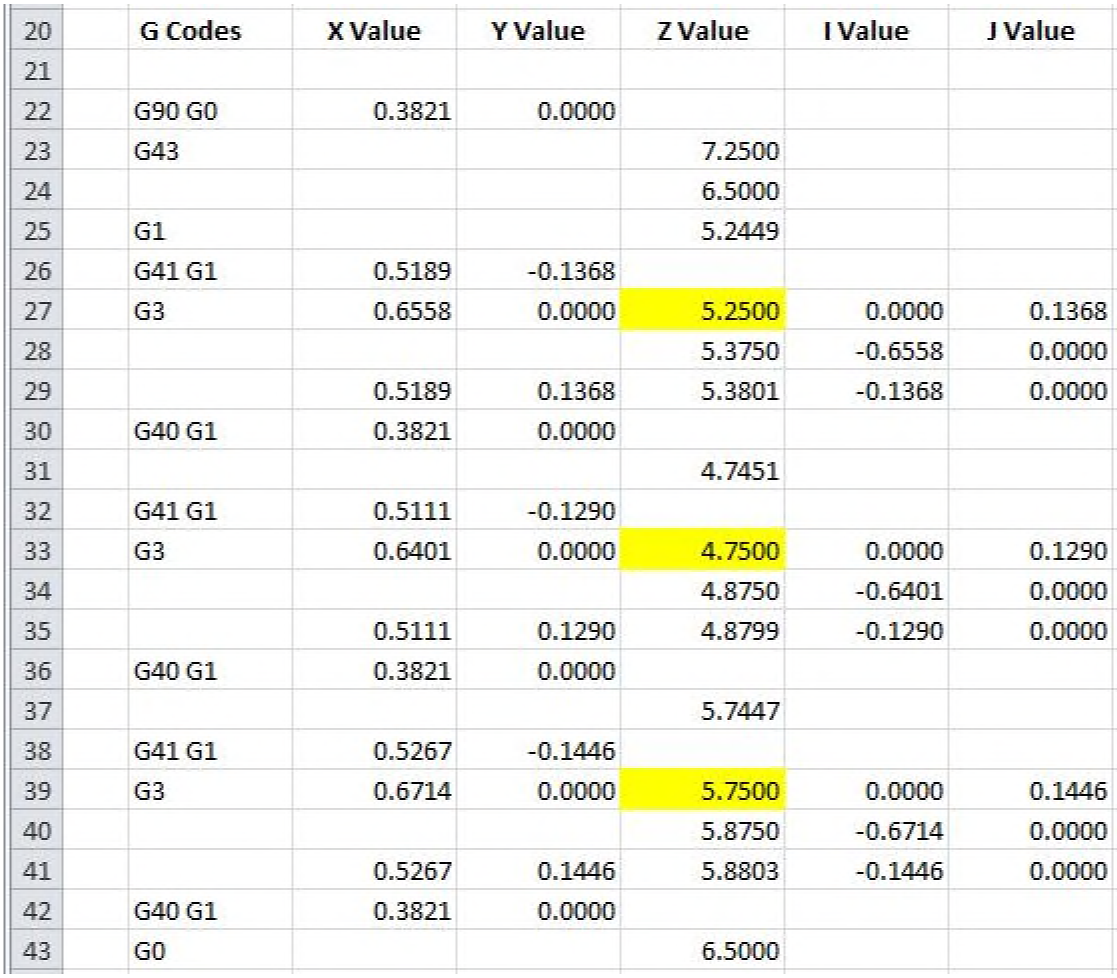
The multiple axial depths of cut shown here fully functions with **tapered** threads also. The software accounts for the change in programmed helical radius based on depth within the hole or boss.



Z Coordinate (Top) 6.2500

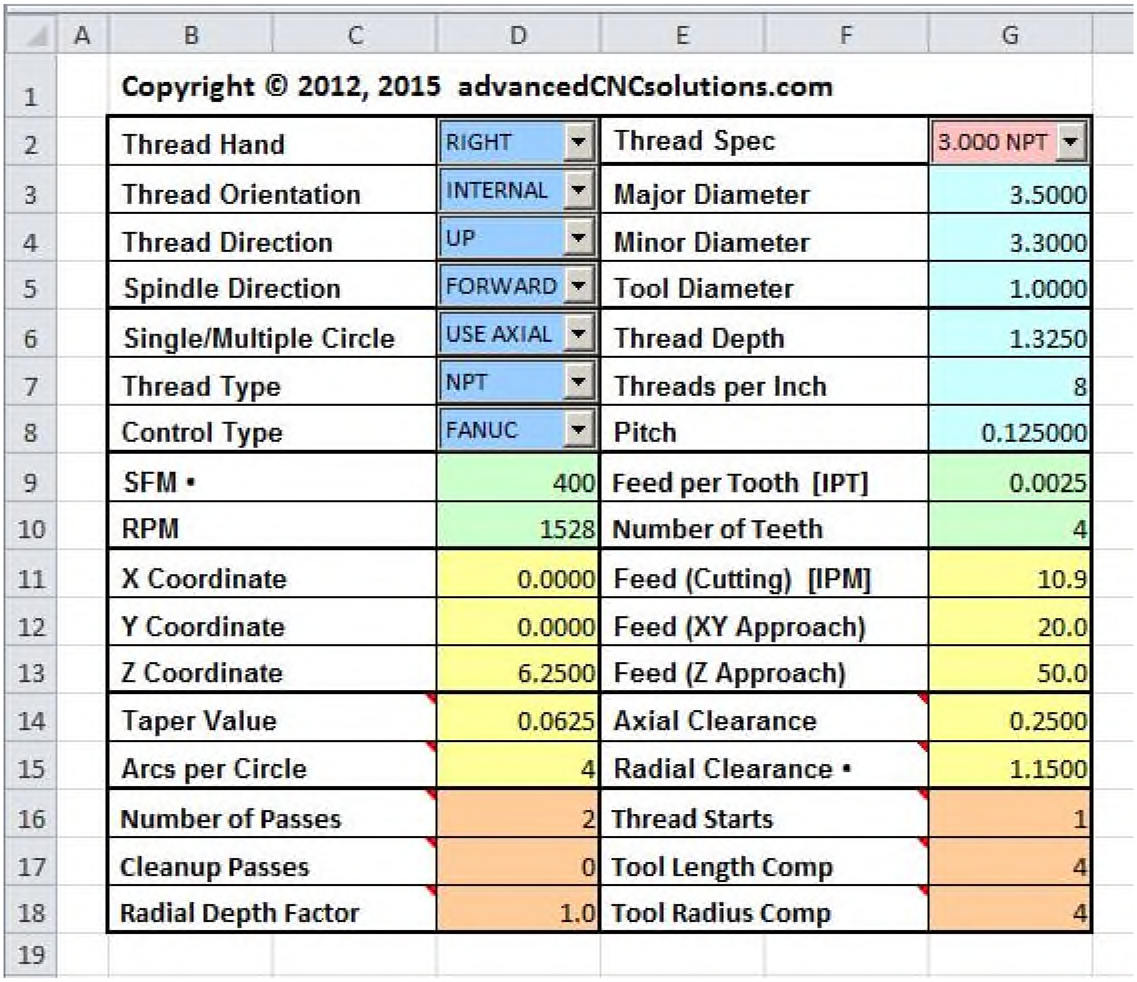
Pitch	Depth	Select
1	0.0000	
2	0.1250	
3	0.2500	
4	0.3750	
5	0.5000	3
6	0.6250	
7	0.7500	
8	0.8750	
9	1.0000	1
10	1.1250	
11	1.2500	
12	1.3750	
13	1.5000	2

Below (internal tapered thread) example shows selecting 1.000" depth (9 threads deep) as 1st axial thread pass, 1.500" depth (13 threads deep) as 2nd axial thread pass and 0.500" depth (5 threads deep) as 3rd and final pass. NC code shows corresponding Z axis values (in yellow) for the 3 axial thread depths. Notice the different "I" values for this tapered thread example. Note: Hole top is at Z = 6.2500".



20	G Codes	X Value	Y Value	Z Value	I Value	J Value
21						
22	G90 G0	0.3821	0.0000			
23	G43			7.2500		
24				6.5000		
25	G1			5.2449		
26	G41 G1	0.5189	-0.1368			
27	G3	0.6558	0.0000	5.2500	0.0000	0.1368
28				5.3750	-0.6558	0.0000
29		0.5189	0.1368	5.3801	-0.1368	0.0000
30	G40 G1	0.3821	0.0000			
31				4.7451		
32	G41 G1	0.5111	-0.1290			
33	G3	0.6401	0.0000	4.7500	0.0000	0.1290
34				4.8750	-0.6401	0.0000
35		0.5111	0.1290	4.8799	-0.1290	0.0000
36	G40 G1	0.3821	0.0000			
37				5.7447		
38	G41 G1	0.5267	-0.1446			
39	G3	0.6714	0.0000	5.7500	0.0000	0.1446
40				5.8750	-0.6714	0.0000
41		0.5267	0.1446	5.8803	-0.1446	0.0000
42	G40 G1	0.3821	0.0000			
43	G0			6.5000		

The below tapered thread milling example (for multi-tooth cutter) uses "**Arcs per Circle**" set to 4, with 3 axial passes and 2 radial passes. This example demonstrates the use of "axial dividing" up the cuts into cuts at 3 different depths.

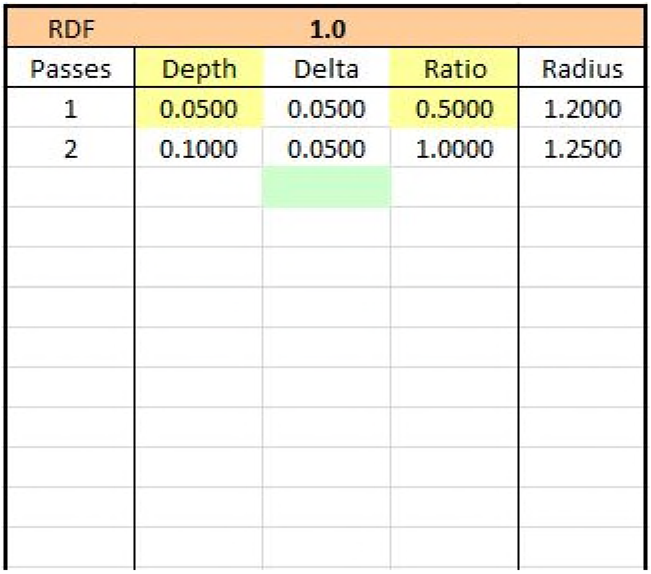


	A	B	C	D	E	F	G
1		**Copyright © 2012, 2015 advancedCNCsolutions.com**					
2		Thread Hand		RIGHT	Thread Spec		3.000 NPT
3		Thread Orientation		INTERNAL	Major Diameter		3.5000
4		Thread Direction		UP	Minor Diameter		3.3000
5		Spindle Direction		FORWARD	Tool Diameter		1.0000
6		Single/Multiple Circle		USE AXIAL	Thread Depth		1.3250
7		Thread Type		NPT	Threads per Inch		8
8		Control Type		FANUC	Pitch		0.125000
9		SFM •		400	Feed per Tooth [IPT]		0.0025
10		RPM		1528	Number of Teeth		4
11		X Coordinate		0.0000	Feed (Cutting) [IPM]		10.9
12		Y Coordinate		0.0000	Feed (XY Approach)		20.0
13		Z Coordinate		6.2500	Feed (Z Approach)		50.0
14		Taper Value		0.0625	Axial Clearance		0.2500
15		Arcs per Circle		4	Radial Clearance •		1.1500
16		Number of Passes		2	Thread Starts		1
17		Cleanup Passes		0	Tool Length Comp		4
18		Radial Depth Factor		1.0	Tool Radius Comp		4
19							

Radial passes set to 2.

Axial passes set to 3.

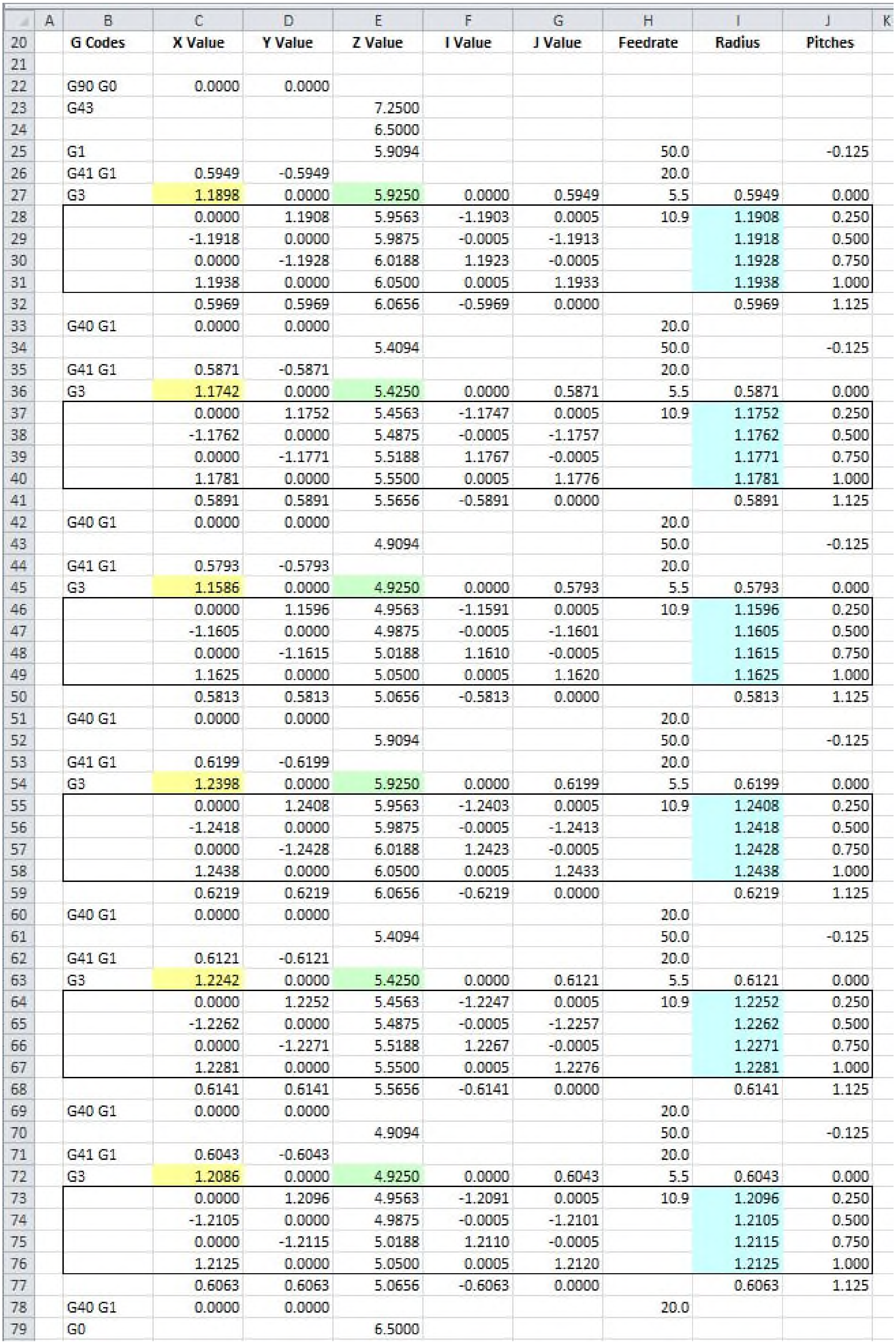
Thread Milling G-Code Generator



RDF	1.0			
Passes	Depth	Delta	Ratio	Radius
1	0.0500	0.0500	0.5000	1.2000
2	0.1000	0.0500	1.0000	1.2500

Pitch	Depth	Select
1	-0.0500	
2	0.0750	
3	0.2000	
4	0.3250	1
5	0.4500	
6	0.5750	
7	0.7000	
8	0.8250	2
9	0.9500	
10	1.0750	
11	1.2000	
12	1.3250	3

Thread Milling G-Code Generator



	A	B	C	D	E	F	G	H	I	J	K
20		G Codes	X Value	Y Value	Z Value	I Value	J Value	Feedrate	Radius	Pitches	
21											
22		G90 G0	0.0000	0.0000							
23		G43			7.2500						
24					6.5000						
25		G1			5.9094			50.0		-0.125	
26		G41 G1	0.5949	-0.5949				20.0			
27		G3	1.1898	0.0000	5.9250	0.0000	0.5949	5.5	0.5949	0.000	
28			0.0000	1.1908	5.9563	-1.1903	0.0005	10.9	1.1908	0.250	
29			-1.1918	0.0000	5.9875	-0.0005	-1.1913		1.1918	0.500	
30			0.0000	-1.1928	6.0188	1.1923	-0.0005		1.1928	0.750	
31			1.1938	0.0000	6.0500	0.0005	1.1933		1.1938	1.000	
32			0.5969	0.5969	6.0656	-0.5969	0.0000		0.5969	1.125	
33		G40 G1	0.0000	0.0000				20.0			
34					5.4094			50.0		-0.125	
35		G41 G1	0.5871	-0.5871				20.0			
36		G3	1.1742	0.0000	5.4250	0.0000	0.5871	5.5	0.5871	0.000	
37			0.0000	1.1752	5.4563	-1.1747	0.0005	10.9	1.1752	0.250	
38			-1.1762	0.0000	5.4875	-0.0005	-1.1757		1.1762	0.500	
39			0.0000	-1.1771	5.5188	1.1767	-0.0005		1.1771	0.750	
40			1.1781	0.0000	5.5500	0.0005	1.1776		1.1781	1.000	
41			0.5891	0.5891	5.5656	-0.5891	0.0000		0.5891	1.125	
42		G40 G1	0.0000	0.0000				20.0			
43					4.9094			50.0		-0.125	
44		G41 G1	0.5793	-0.5793				20.0			
45		G3	1.1586	0.0000	4.9250	0.0000	0.5793	5.5	0.5793	0.000	
46			0.0000	1.1596	4.9563	-1.1591	0.0005	10.9	1.1596	0.250	
47			-1.1605	0.0000	4.9875	-0.0005	-1.1601		1.1605	0.500	
48			0.0000	-1.1615	5.0188	1.1610	-0.0005		1.1615	0.750	
49			1.1625	0.0000	5.0500	0.0005	1.1620		1.1625	1.000	
50			0.5813	0.5813	5.0656	-0.5813	0.0000		0.5813	1.125	
51		G40 G1	0.0000	0.0000				20.0			
52					5.9094			50.0		-0.125	
53		G41 G1	0.6199	-0.6199				20.0			
54		G3	1.2398	0.0000	5.9250	0.0000	0.6199	5.5	0.6199	0.000	
55			0.0000	1.2408	5.9563	-1.2403	0.0005	10.9	1.2408	0.250	
56			-1.2418	0.0000	5.9875	-0.0005	-1.2413		1.2418	0.500	
57			0.0000	-1.2428	6.0188	1.2423	-0.0005		1.2428	0.750	
58			1.2438	0.0000	6.0500	0.0005	1.2433		1.2438	1.000	
59			0.6219	0.6219	6.0656	-0.6219	0.0000		0.6219	1.125	
60		G40 G1	0.0000	0.0000				20.0			
61					5.4094			50.0		-0.125	
62		G41 G1	0.6121	-0.6121				20.0			
63		G3	1.2242	0.0000	5.4250	0.0000	0.6121	5.5	0.6121	0.000	
64			0.0000	1.2252	5.4563	-1.2247	0.0005	10.9	1.2252	0.250	
65			-1.2262	0.0000	5.4875	-0.0005	-1.2257		1.2262	0.500	
66			0.0000	-1.2271	5.5188	1.2267	-0.0005		1.2271	0.750	
67			1.2281	0.0000	5.5500	0.0005	1.2276		1.2281	1.000	
68			0.6141	0.6141	5.5656	-0.6141	0.0000		0.6141	1.125	
69		G40 G1	0.0000	0.0000				20.0			
70					4.9094			50.0		-0.125	
71		G41 G1	0.6043	-0.6043				20.0			
72		G3	1.2086	0.0000	4.9250	0.0000	0.6043	5.5	0.6043	0.000	
73			0.0000	1.2096	4.9563	-1.2091	0.0005	10.9	1.2096	0.250	
74			-1.2105	0.0000	4.9875	-0.0005	-1.2101		1.2105	0.500	
75			0.0000	-1.2115	5.0188	1.2110	-0.0005		1.2115	0.750	
76			1.2125	0.0000	5.0500	0.0005	1.2120		1.2125	1.000	
77			0.6063	0.6063	5.0656	-0.6063	0.0000		0.6063	1.125	
78		G40 G1	0.0000	0.0000				20.0			
79		G0			6.5000						

The above example uses 4 arc segments per helical circle (each with an increasing radius) to thread mill the 3 axial passes at the first radial depth, then thread mill the 3 axial passes again at the second (final) radial depth.

With this course pitch (8 TPI), 8 or more arc segments per helical circle should have been used to keep the change in radius (from segment to segment) smaller. But this example used 4 arc segments as to only fill one page.

Using 4 arc segments per helical circle, the above example thread mills the 3 axial passes at the first radial depth, then thread mills the 3 axial passes at the second (final) radial depth.

The software calculates the correct cutter path radius based on the depth within the hole.

1.250 radius (at top of hole) = 1.2398 [4th thread pass radius] + (6.250 [top Z] – 5.925 [curr Z]) * 0.03125

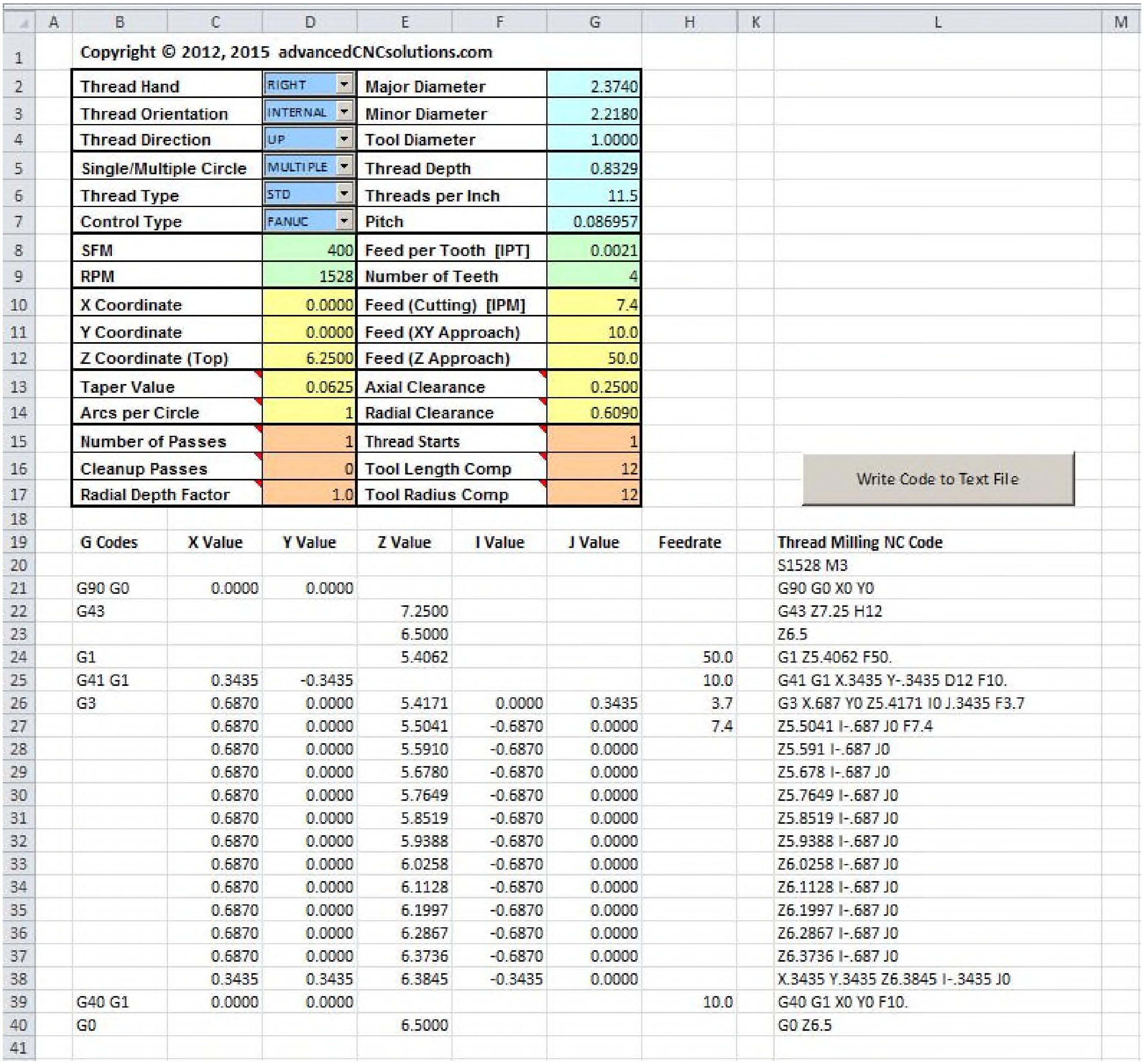
1.250 radius (at top of hole) = 1.2242 [5th thread pass radius] + (6.250 [top Z] – 5.425 [curr Z]) * 0.03125

1.250 radius (at top of hole) = 1.2086 [6th thread pass radius] + (6.250 [top Z] – 4.925 [curr Z]) * 0.03125

SEE VARIOUS THREAD MILLING EXAMPLES BELOW.

SAMPLE #1 (Fanuc control)

Right handed internal standard thread milling producing several full circle cuts for single-point thread milling tool.



Copyright © 2012, 2015 advancedCNCsolutions.com

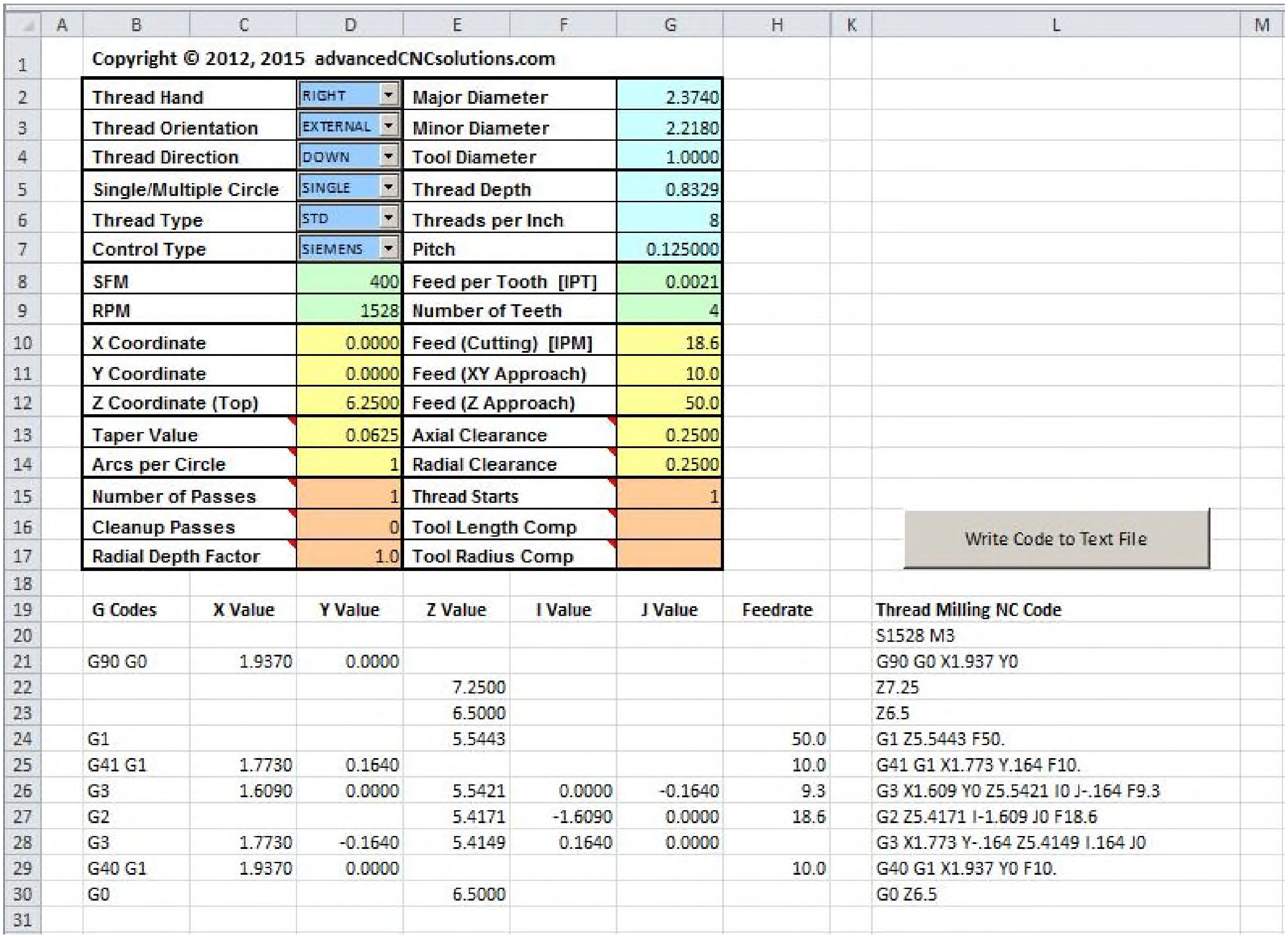
Thread Hand	RIGHT	Major Diameter	2.3740
Thread Orientation	INTERNAL	Minor Diameter	2.2180
Thread Direction	UP	Tool Diameter	1.0000
Single/Multiple Circle	MULTIPLE	Thread Depth	0.8329
Thread Type	STD	Threads per Inch	11.5
Control Type	FANUC	Pitch	0.086957
SFM	400	Feed per Tooth [IPT]	0.0021
RPM	1528	Number of Teeth	4
X Coordinate	0.0000	Feed (Cutting) [IPM]	7.4
Y Coordinate	0.0000	Feed (XY Approach)	10.0
Z Coordinate (Top)	6.2500	Feed (Z Approach)	50.0
Taper Value	0.0625	Axial Clearance	0.2500
Arcs per Circle	1	Radial Clearance	0.6090
Number of Passes	1	Thread Starts	1
Cleanup Passes	0	Tool Length Comp	12
Radial Depth Factor	1.0	Tool Radius Comp	12

Write Code to Text File

G Codes	X Value	Y Value	Z Value	I Value	J Value	Feedrate	Thread Milling NC Code
							S1528 M3
G90 G0	0.0000	0.0000					G90 G0 X0 Y0
G43			7.2500				G43 Z7.25 H12
			6.5000				Z6.5
G1			5.4062			50.0	G1 Z5.4062 F50.
G41 G1	0.3435	-0.3435				10.0	G41 G1 X.3435 Y-.3435 D12 F10.
G3	0.6870	0.0000	5.4171	0.0000	0.3435	3.7	G3 X.687 Y0 Z5.4171 I0 J.3435 F3.7
	0.6870	0.0000	5.5041	-0.6870	0.0000	7.4	Z5.5041 I-.687 J0 F7.4
	0.6870	0.0000	5.5910	-0.6870	0.0000		Z5.591 I-.687 J0
	0.6870	0.0000	5.6780	-0.6870	0.0000		Z5.678 I-.687 J0
	0.6870	0.0000	5.7649	-0.6870	0.0000		Z5.7649 I-.687 J0
	0.6870	0.0000	5.8519	-0.6870	0.0000		Z5.8519 I-.687 J0
	0.6870	0.0000	5.9388	-0.6870	0.0000		Z5.9388 I-.687 J0
	0.6870	0.0000	6.0258	-0.6870	0.0000		Z6.0258 I-.687 J0
	0.6870	0.0000	6.1128	-0.6870	0.0000		Z6.1128 I-.687 J0
	0.6870	0.0000	6.1997	-0.6870	0.0000		Z6.1997 I-.687 J0
	0.6870	0.0000	6.2867	-0.6870	0.0000		Z6.2867 I-.687 J0
	0.6870	0.0000	6.3736	-0.6870	0.0000		Z6.3736 I-.687 J0
	0.3435	0.3435	6.3845	-0.3435	0.0000		X.3435 Y.3435 Z6.3845 I-.3435 J0
G40 G1	0.0000	0.0000				10.0	G40 G1 X0 Y0 F10.
G0			6.5000				G0 Z6.5

SAMPLE #2 (Siemens control)

Right handed external standard thread milling producing single full circle cut for multi-tooth thread milling tool.



Copyright © 2012, 2015 advancedCNCsolutions.com

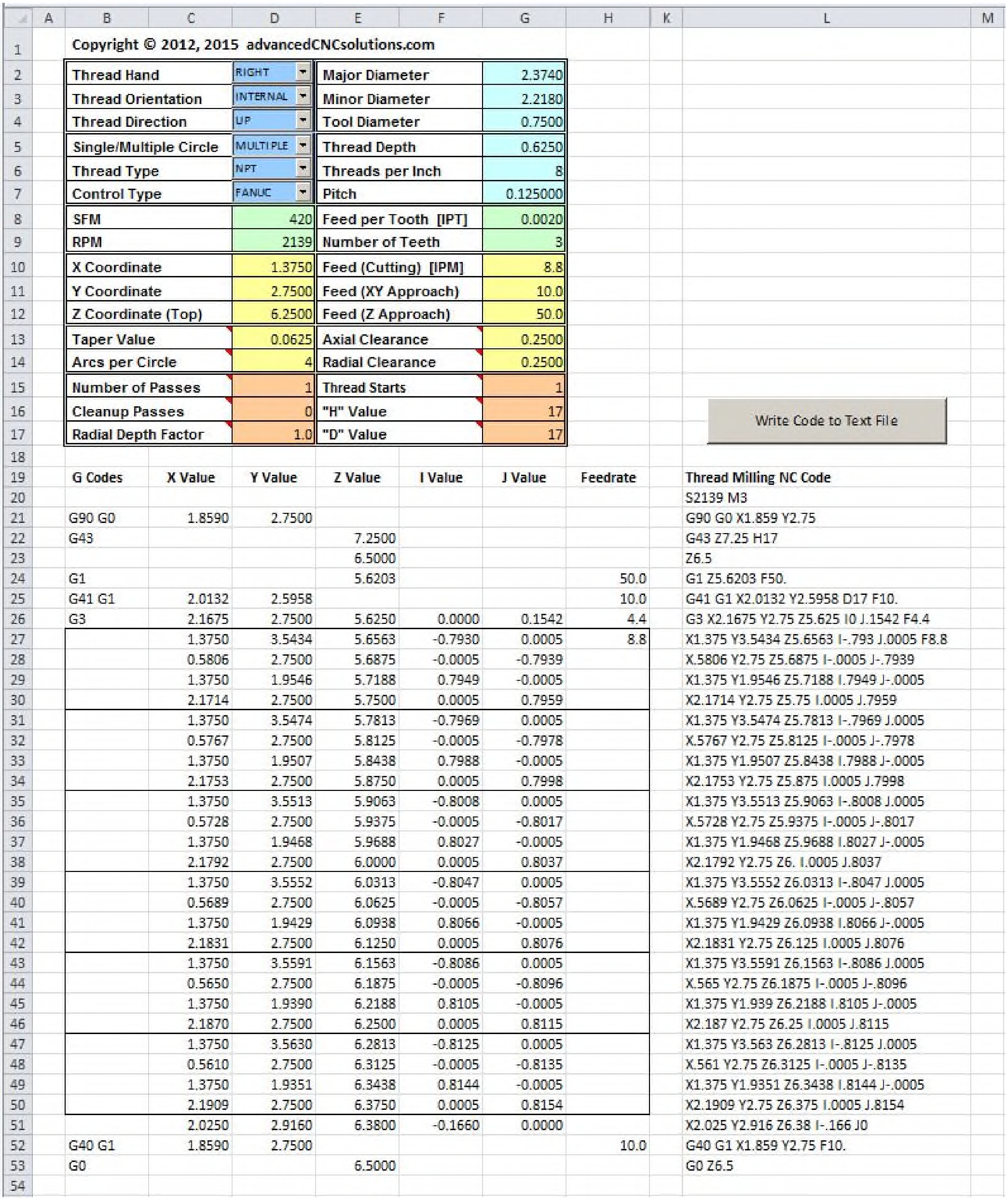
Thread Hand	RIGHT	Major Diameter	2.3740
Thread Orientation	EXTERNAL	Minor Diameter	2.2180
Thread Direction	DOWN	Tool Diameter	1.0000
Single/Multiple Circle	SINGLE	Thread Depth	0.8329
Thread Type	STD	Threads per Inch	8
Control Type	SIEMENS	Pitch	0.125000
SFM	400	Feed per Tooth [IPT]	0.0021
RPM	1528	Number of Teeth	4
X Coordinate	0.0000	Feed (Cutting) [IPM]	18.6
Y Coordinate	0.0000	Feed (XY Approach)	10.0
Z Coordinate (Top)	6.2500	Feed (Z Approach)	50.0
Taper Value	0.0625	Axial Clearance	0.2500
Arcs per Circle	1	Radial Clearance	0.2500
Number of Passes	1	Thread Starts	1
Cleanup Passes	0	Tool Length Comp	
Radial Depth Factor	1.0	Tool Radius Comp	

G Codes	X Value	Y Value	Z Value	I Value	J Value	Feedrate	Thread Milling NC Code
							S1528 M3
G90 G0	1.9370	0.0000					G90 G0 X1.937 Y0
			7.2500				Z7.25
			6.5000				Z6.5
G1			5.5443			50.0	G1 Z5.5443 F50.
G41 G1	1.7730	0.1640				10.0	G41 G1 X1.773 Y.164 F10.
G3	1.6090	0.0000	5.5421	0.0000	-0.1640	9.3	G3 X1.609 Y0 Z5.5421 I0 J-.164 F9.3
G2			5.4171	-1.6090	0.0000	18.6	G2 Z5.4171 I-1.609 J0 F18.6
G3	1.7730	-0.1640	5.4149	0.1640	0.0000		G3 X1.773 Y-.164 Z5.4149 I.164 J0
G40 G1	1.9370	0.0000				10.0	G40 G1 X1.937 Y0 F10.
G0			6.5000				G0 Z6.5

SAMPLE #3 (Fanuc control)

Right handed internal NPT tapered thread milling producing 4 helical arc segment moves per full circle using single-point thread milling tool. Notice the spiral interpolation arc moves to produce tapered threads.

Thread Milling G-Code Generator



Copyright © 2012, 2015 advancedCNCsolutions.com

Thread Hand	RIGHT	Major Diameter	2.3740
Thread Orientation	INTERNAL	Minor Diameter	2.2180
Thread Direction	UP	Tool Diameter	0.7500
Single/Multiple Circle	MULTIPLE	Thread Depth	0.6250
Thread Type	NPT	Threads per Inch	8
Control Type	FANUC	Pitch	0.125000
SFM	420	Feed per Tooth [IPT]	0.0020
RPM	2139	Number of Teeth	3
X Coordinate	1.3750	Feed (Cutting) [IPM]	8.8
Y Coordinate	2.7500	Feed (XY Approach)	10.0
Z Coordinate (Top)	6.2500	Feed (Z Approach)	50.0
Taper Value	0.0625	Axial Clearance	0.2500
Arcs per Circle	4	Radial Clearance	0.2500
Number of Passes	1	Thread Starts	1
Cleanup Passes	0	"H" Value	17
Radial Depth Factor	1.0	"D" Value	17

G Codes	X Value	Y Value	Z Value	I Value	J Value	Feedrate
G90 G0	1.8590	2.7500				
G43			7.2500			
			6.5000			
G1			5.6203			50.0
G41 G1	2.0132	2.5958				10.0
G3	2.1675	2.7500	5.6250	0.0000	0.1542	4.4
	1.3750	3.5434	5.6563	-0.7930	0.0005	8.8
	0.5806	2.7500	5.6875	-0.0005	-0.7939	
	1.3750	1.9546	5.7188	0.7949	-0.0005	
	2.1714	2.7500	5.7500	0.0005	0.7959	
	1.3750	3.5474	5.7813	-0.7969	0.0005	
	0.5767	2.7500	5.8125	-0.0005	-0.7978	
	1.3750	1.9507	5.8438	0.7988	-0.0005	
	2.1753	2.7500	5.8750	0.0005	0.7998	
	1.3750	3.5513	5.9063	-0.8008	0.0005	
	0.5728	2.7500	5.9375	-0.0005	-0.8017	
	1.3750	1.9468	5.9688	0.8027	-0.0005	
	2.1792	2.7500	6.0000	0.0005	0.8037	
	1.3750	3.5552	6.0313	-0.8047	0.0005	
	0.5689	2.7500	6.0625	-0.0005	-0.8057	
	1.3750	1.9429	6.0938	0.8066	-0.0005	
	2.1831	2.7500	6.1250	0.0005	0.8076	
	1.3750	3.5591	6.1563	-0.8086	0.0005	
	0.5650	2.7500	6.1875	-0.0005	-0.8096	
	1.3750	1.9390	6.2188	0.8105	-0.0005	
	2.1870	2.7500	6.2500	0.0005	0.8115	
	1.3750	3.5630	6.2813	-0.8125	0.0005	
	0.5610	2.7500	6.3125	-0.0005	-0.8135	
	1.3750	1.9351	6.3438	0.8144	-0.0005	
	2.1909	2.7500	6.3750	0.0005	0.8154	
	2.0250	2.9160	6.3800	-0.1660	0.0000	
G40 G1	1.8590	2.7500				10.0
G0			6.5000			

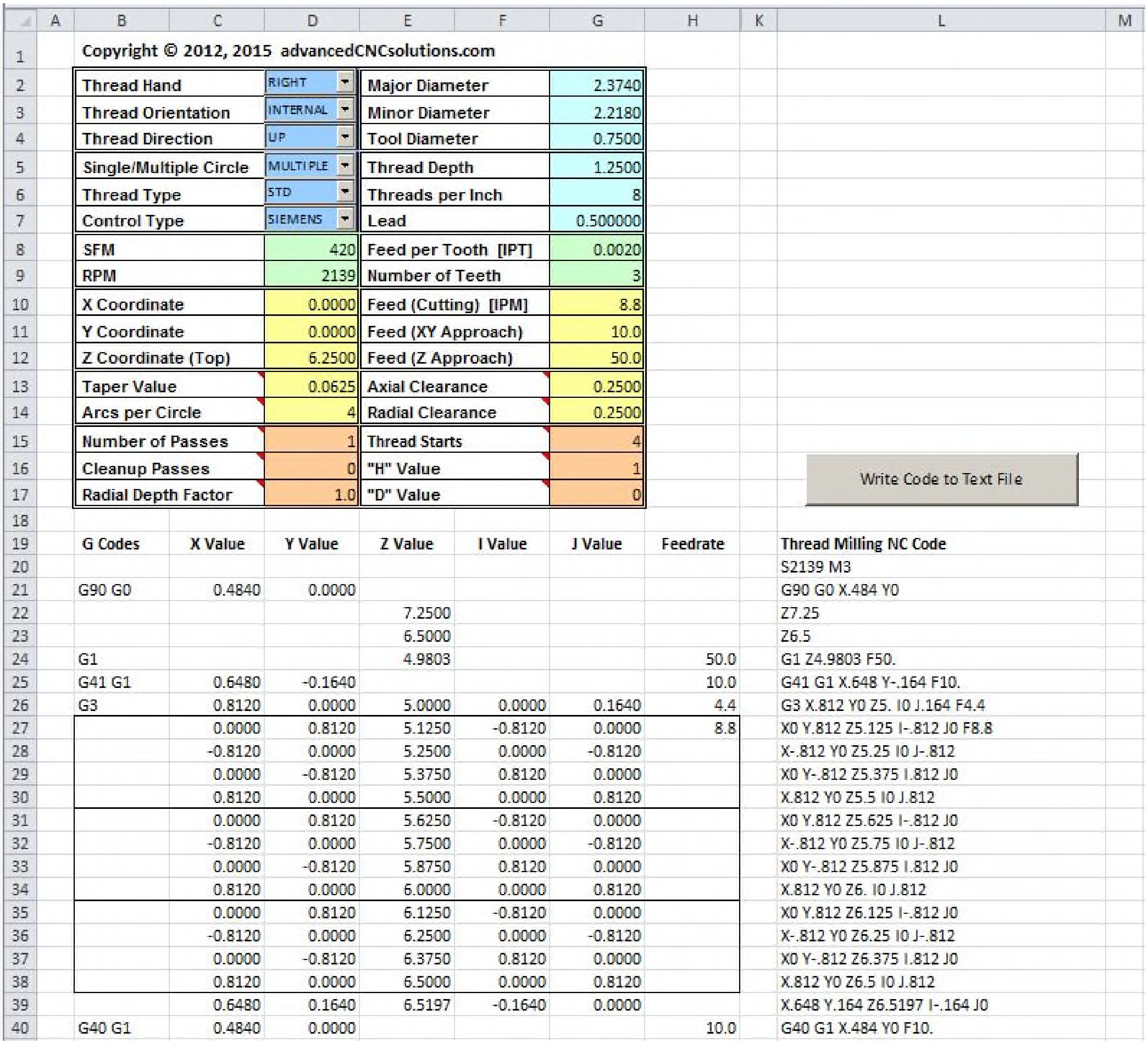
Thread Milling NC Code

```
S2139 M3
G90 G0 X1.859 Y2.75
G43 Z7.25 H17
Z6.5
G1 Z5.6203 F50.
G41 G1 X2.0132 Y2.5958 D17 F10.
G3 X2.1675 Y2.75 Z5.625 I0 J.1542 F4.4
X1.375 Y3.5434 Z5.6563 I-.793 J.0005 F8.8
X.5806 Y2.75 Z5.6875 I-.0005 J-.7939
X1.375 Y1.9546 Z5.7188 I.7949 J-.0005
X2.1714 Y2.75 Z5.75 I.0005 J.7959
X1.375 Y3.5474 Z5.7813 I-.7969 J.0005
X.5767 Y2.75 Z5.8125 I-.0005 J-.7978
X1.375 Y1.9507 Z5.8438 I.7988 J-.0005
X2.1753 Y2.75 Z5.875 I.0005 J.7998
X1.375 Y3.5513 Z5.9063 I-.8008 J.0005
X.5728 Y2.75 Z5.9375 I-.0005 J-.8017
X1.375 Y1.9468 Z5.9688 I.8027 J-.0005
X2.1792 Y2.75 Z6. I.0005 J.8037
X1.375 Y3.5552 Z6.0313 I-.8047 J.0005
X.5689 Y2.75 Z6.0625 I-.0005 J-.8057
X1.375 Y1.9429 Z6.0938 I.8066 J-.0005
X2.1831 Y2.75 Z6.125 I.0005 J.8076
X1.375 Y3.5591 Z6.1563 I-.8086 J.0005
X.565 Y2.75 Z6.1875 I-.0005 J-.8096
X1.375 Y1.939 Z6.2188 I.8105 J-.0005
X2.187 Y2.75 Z6.25 I.0005 J.8115
X1.375 Y3.563 Z6.2813 I-.8125 J.0005
X.561 Y2.75 Z6.3125 I-.0005 J-.8135
X1.375 Y1.9351 Z6.3438 I.8144 J-.0005
X2.1909 Y2.75 Z6.375 I.0005 J.8154
X2.025 Y2.916 Z6.38 I-.166 J0
G40 G1 X1.859 Y2.75 F10.
G0 Z6.5
```

SAMPLE #4 (Siemens control)

Thread Milling G-Code Generator

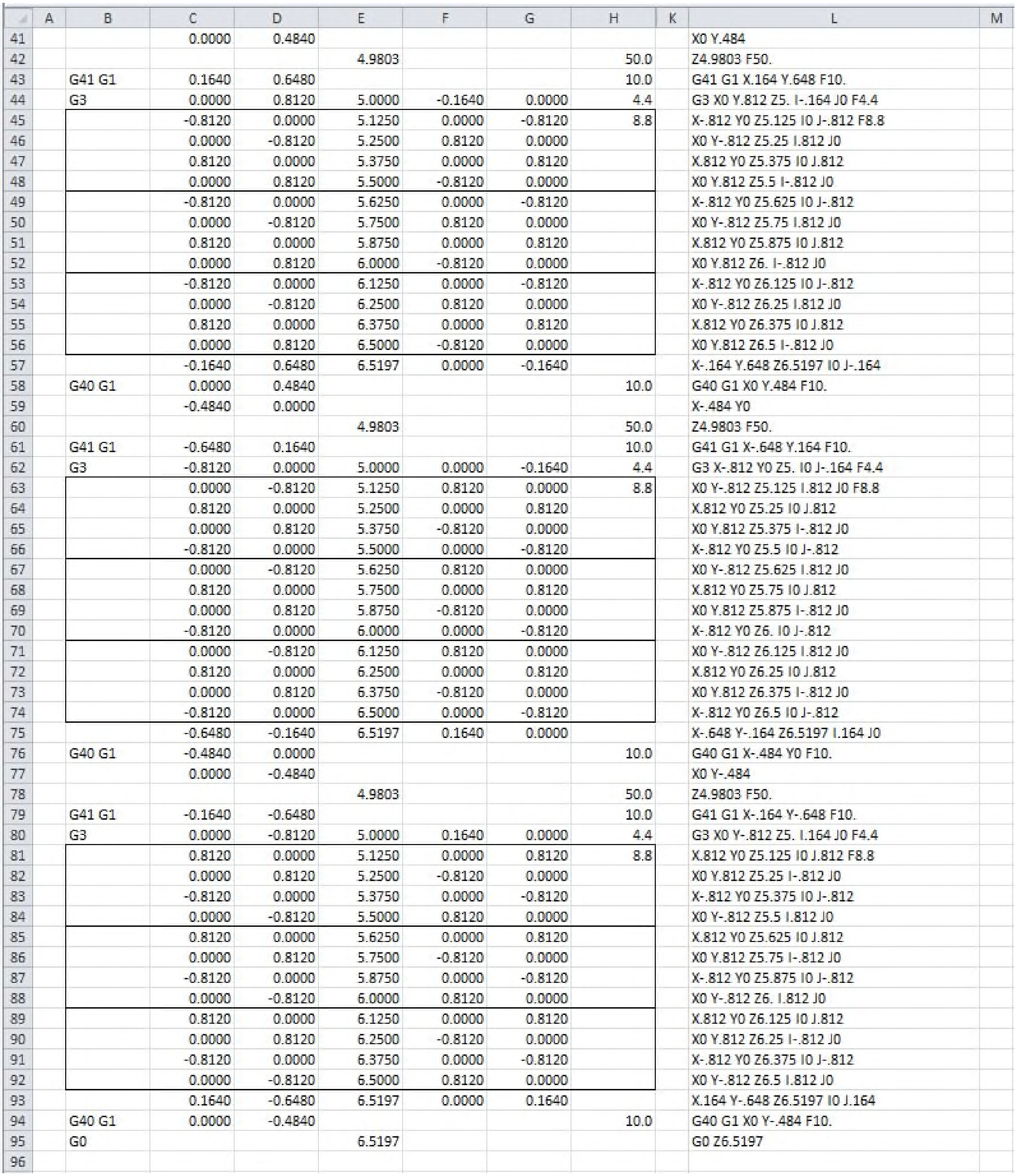
Right handed internal standard thread milling producing 4 helical arc segment moves per full circle using single-point thread milling tool performing 4 thread starts.



Copyright © 2012, 2015 advancedCNCsolutions.com

Thread Hand	RIGHT	Major Diameter	2.3740
Thread Orientation	INTERNAL	Minor Diameter	2.2180
Thread Direction	UP	Tool Diameter	0.7500
Single/Multiple Circle	MULTIPLE	Thread Depth	1.2500
Thread Type	STD	Threads per Inch	8
Control Type	SIEMENS	Lead	0.500000
SFM	420	Feed per Tooth [IPT]	0.0020
RPM	2139	Number of Teeth	3
X Coordinate	0.0000	Feed (Cutting) [IPM]	8.8
Y Coordinate	0.0000	Feed (XY Approach)	10.0
Z Coordinate (Top)	6.2500	Feed (Z Approach)	50.0
Taper Value	0.0625	Axial Clearance	0.2500
Arcs per Circle	4	Radial Clearance	0.2500
Number of Passes	1	Thread Starts	4
Cleanup Passes	0	"H" Value	1
Radial Depth Factor	1.0	"D" Value	0

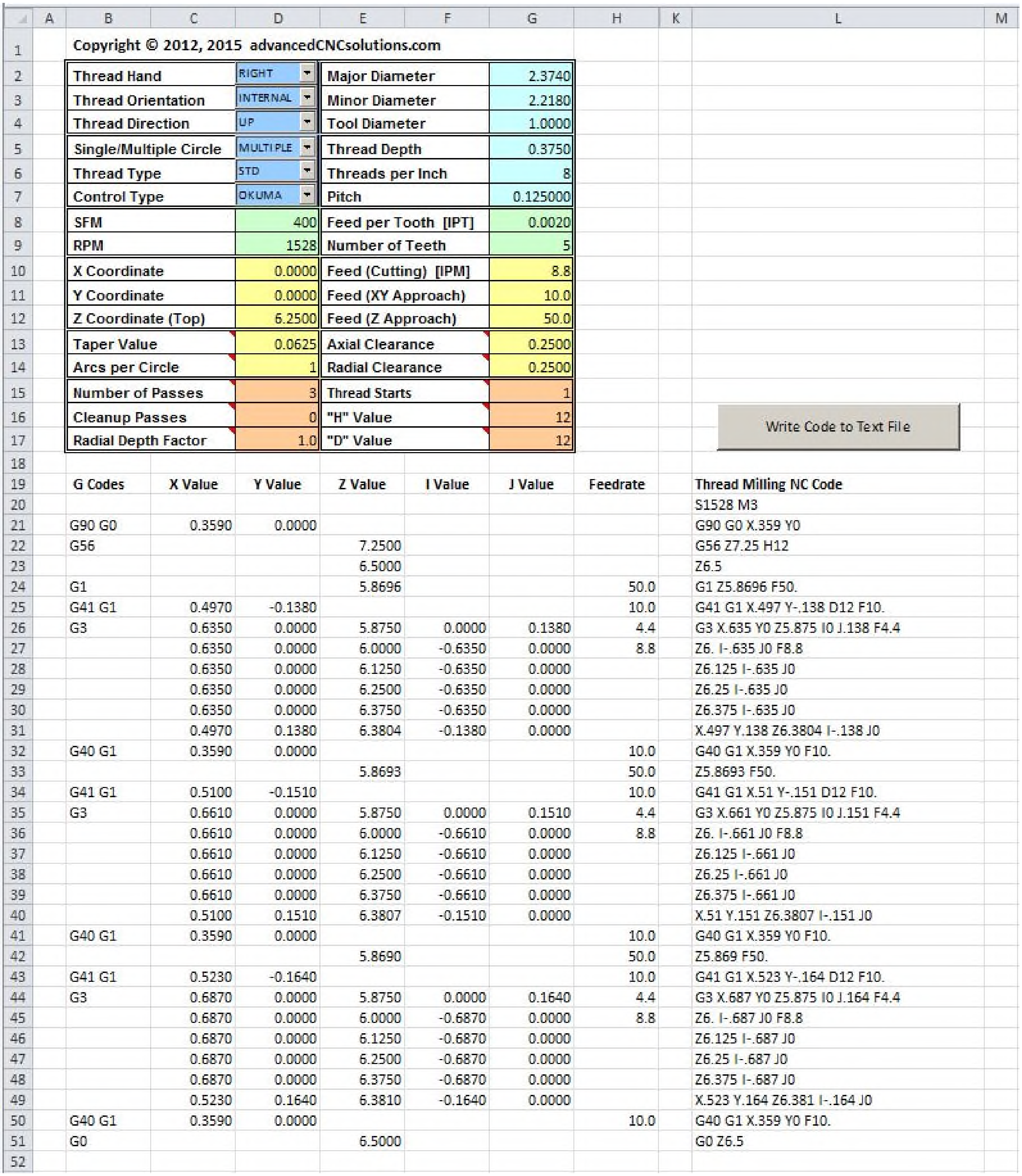
G Codes	X Value	Y Value	Z Value	I Value	J Value	Feedrate	Thread Milling NC Code
							S2139 M3
G90 G0	0.4840	0.0000					G90 G0 X.484 Y0
			7.2500				Z7.25
			6.5000				Z6.5
G1			4.9803			50.0	G1 Z4.9803 F50.
G41 G1	0.6480	-0.1640				10.0	G41 G1 X.648 Y-.164 F10.
G3	0.8120	0.0000	5.0000	0.0000	0.1640	4.4	G3 X.812 Y0 Z5. I0 J.164 F4.4
	0.0000	0.8120	5.1250	-0.8120	0.0000	8.8	X0 Y.812 Z5.125 I-.812 J0 F8.8
	-0.8120	0.0000	5.2500	0.0000	-0.8120		X-.812 Y0 Z5.25 I0 J-.812
	0.0000	-0.8120	5.3750	0.8120	0.0000		X0 Y-.812 Z5.375 I.812 J0
	0.8120	0.0000	5.5000	0.0000	0.8120		X.812 Y0 Z5.5 I0 J.812
	0.0000	0.8120	5.6250	-0.8120	0.0000		X0 Y.812 Z5.625 I-.812 J0
	-0.8120	0.0000	5.7500	0.0000	-0.8120		X-.812 Y0 Z5.75 I0 J-.812
	0.0000	-0.8120	5.8750	0.8120	0.0000		X0 Y-.812 Z5.875 I.812 J0
	0.8120	0.0000	6.0000	0.0000	0.8120		X.812 Y0 Z6. I0 J.812
	0.0000	0.8120	6.1250	-0.8120	0.0000		X0 Y.812 Z6.125 I-.812 J0
	-0.8120	0.0000	6.2500	0.0000	-0.8120		X-.812 Y0 Z6.25 I0 J-.812
	0.0000	-0.8120	6.3750	0.8120	0.0000		X0 Y-.812 Z6.375 I.812 J0
	0.8120	0.0000	6.5000	0.0000	0.8120		X.812 Y0 Z6.5 I0 J.812
	0.6480	0.1640	6.5197	-0.1640	0.0000		X.648 Y.164 Z6.5197 I-.164 J0
G40 G1	0.4840	0.0000				10.0	G40 G1 X.484 Y0 F10.



	A	B	C	D	E	F	G	H	K	L	M
41			0.0000	0.4840						X0 Y.484	
42					4.9803			50.0		Z4.9803 F50.	
43		G41 G1	0.1640	0.6480				10.0		G41 G1 X.164 Y.648 F10.	
44		G3	0.0000	0.8120	5.0000	-0.1640	0.0000	4.4		G3 X0 Y.812 Z5. I-.164 J0 F4.4	
45			-0.8120	0.0000	5.1250	0.0000	-0.8120	8.8		X-.812 Y0 Z5.125 I0 J-.812 F8.8	
46			0.0000	-0.8120	5.2500	0.8120	0.0000			X0 Y-.812 Z5.25 I.812 J0	
47			0.8120	0.0000	5.3750	0.0000	0.8120			X.812 Y0 Z5.375 I0 J.812	
48			0.0000	0.8120	5.5000	-0.8120	0.0000			X0 Y.812 Z5.5 I-.812 J0	
49			-0.8120	0.0000	5.6250	0.0000	-0.8120			X-.812 Y0 Z5.625 I0 J-.812	
50			0.0000	-0.8120	5.7500	0.8120	0.0000			X0 Y-.812 Z5.75 I.812 J0	
51			0.8120	0.0000	5.8750	0.0000	0.8120			X.812 Y0 Z5.875 I0 J.812	
52			0.0000	0.8120	6.0000	-0.8120	0.0000			X0 Y.812 Z6. I-.812 J0	
53			-0.8120	0.0000	6.1250	0.0000	-0.8120			X-.812 Y0 Z6.125 I0 J-.812	
54			0.0000	-0.8120	6.2500	0.8120	0.0000			X0 Y-.812 Z6.25 I.812 J0	
55			0.8120	0.0000	6.3750	0.0000	0.8120			X.812 Y0 Z6.375 I0 J.812	
56			0.0000	0.8120	6.5000	-0.8120	0.0000			X0 Y.812 Z6.5 I-.812 J0	
57			-0.1640	0.6480	6.5197	0.0000	-0.1640			X-.164 Y.648 Z6.5197 I0 J-.164	
58		G40 G1	0.0000	0.4840				10.0		G40 G1 X0 Y.484 F10.	
59			-0.4840	0.0000						X-.484 Y0	
60					4.9803			50.0		Z4.9803 F50.	
61		G41 G1	-0.6480	0.1640				10.0		G41 G1 X-.648 Y.164 F10.	
62		G3	-0.8120	0.0000	5.0000	0.0000	-0.1640	4.4		G3 X-.812 Y0 Z5. I0 J-.164 F4.4	
63			0.0000	-0.8120	5.1250	0.8120	0.0000	8.8		X0 Y-.812 Z5.125 I.812 J0 F8.8	
64			0.8120	0.0000	5.2500	0.0000	0.8120			X.812 Y0 Z5.25 I0 J.812	
65			0.0000	0.8120	5.3750	-0.8120	0.0000			X0 Y.812 Z5.375 I-.812 J0	
66			-0.8120	0.0000	5.5000	0.0000	-0.8120			X-.812 Y0 Z5.5 I0 J-.812	
67			0.0000	-0.8120	5.6250	0.8120	0.0000			X0 Y-.812 Z5.625 I.812 J0	
68			0.8120	0.0000	5.7500	0.0000	0.8120			X.812 Y0 Z5.75 I0 J.812	
69			0.0000	0.8120	5.8750	-0.8120	0.0000			X0 Y.812 Z5.875 I-.812 J0	
70			-0.8120	0.0000	6.0000	0.0000	-0.8120			X-.812 Y0 Z6. I0 J-.812	
71			0.0000	-0.8120	6.1250	0.8120	0.0000			X0 Y-.812 Z6.125 I.812 J0	
72			0.8120	0.0000	6.2500	0.0000	0.8120			X.812 Y0 Z6.25 I0 J.812	
73			0.0000	0.8120	6.3750	-0.8120	0.0000			X0 Y.812 Z6.375 I-.812 J0	
74			-0.8120	0.0000	6.5000	0.0000	-0.8120			X-.812 Y0 Z6.5 I0 J-.812	
75			-0.6480	-0.1640	6.5197	0.1640	0.0000			X-.648 Y-.164 Z6.5197 I.164 J0	
76		G40 G1	-0.4840	0.0000				10.0		G40 G1 X-.484 Y0 F10.	
77			0.0000	-0.4840						X0 Y-.484	
78					4.9803			50.0		Z4.9803 F50.	
79		G41 G1	-0.1640	-0.6480				10.0		G41 G1 X-.164 Y-.648 F10.	
80		G3	0.0000	-0.8120	5.0000	0.1640	0.0000	4.4		G3 X0 Y-.812 Z5. I.164 J0 F4.4	
81			0.8120	0.0000	5.1250	0.0000	0.8120	8.8		X.812 Y0 Z5.125 I0 J.812 F8.8	
82			0.0000	0.8120	5.2500	-0.8120	0.0000			X0 Y.812 Z5.25 I-.812 J0	
83			-0.8120	0.0000	5.3750	0.0000	-0.8120			X-.812 Y0 Z5.375 I0 J-.812	
84			0.0000	-0.8120	5.5000	0.8120	0.0000			X0 Y-.812 Z5.5 I.812 J0	
85			0.8120	0.0000	5.6250	0.0000	0.8120			X.812 Y0 Z5.625 I0 J.812	
86			0.0000	0.8120	5.7500	-0.8120	0.0000			X0 Y.812 Z5.75 I-.812 J0	
87			-0.8120	0.0000	5.8750	0.0000	-0.8120			X-.812 Y0 Z5.875 I0 J-.812	
88			0.0000	-0.8120	6.0000	0.8120	0.0000			X0 Y-.812 Z6. I.812 J0	
89			0.8120	0.0000	6.1250	0.0000	0.8120			X.812 Y0 Z6.125 I0 J.812	
90			0.0000	0.8120	6.2500	-0.8120	0.0000			X0 Y.812 Z6.25 I-.812 J0	
91			-0.8120	0.0000	6.3750	0.0000	-0.8120			X-.812 Y0 Z6.375 I0 J-.812	
92			0.0000	-0.8120	6.5000	0.8120	0.0000			X0 Y-.812 Z6.5 I.812 J0	
93			0.1640	-0.6480	6.5197	0.0000	0.1640			X.164 Y-.648 Z6.5197 I0 J.164	
94		G40 G1	0.0000	-0.4840				10.0		G40 G1 X0 Y-.484 F10.	
95		G0			6.5197					G0 Z6.5197	
96											

SAMPLE #5 (Okuma control)

Right handed internal standard thread milling producing 3 threading passes using single-point thread milling tool. "Radial Depth Factor" is set to 1.0 for constant (radial) depth of cut. Each pass removes 0.026" on the radius.



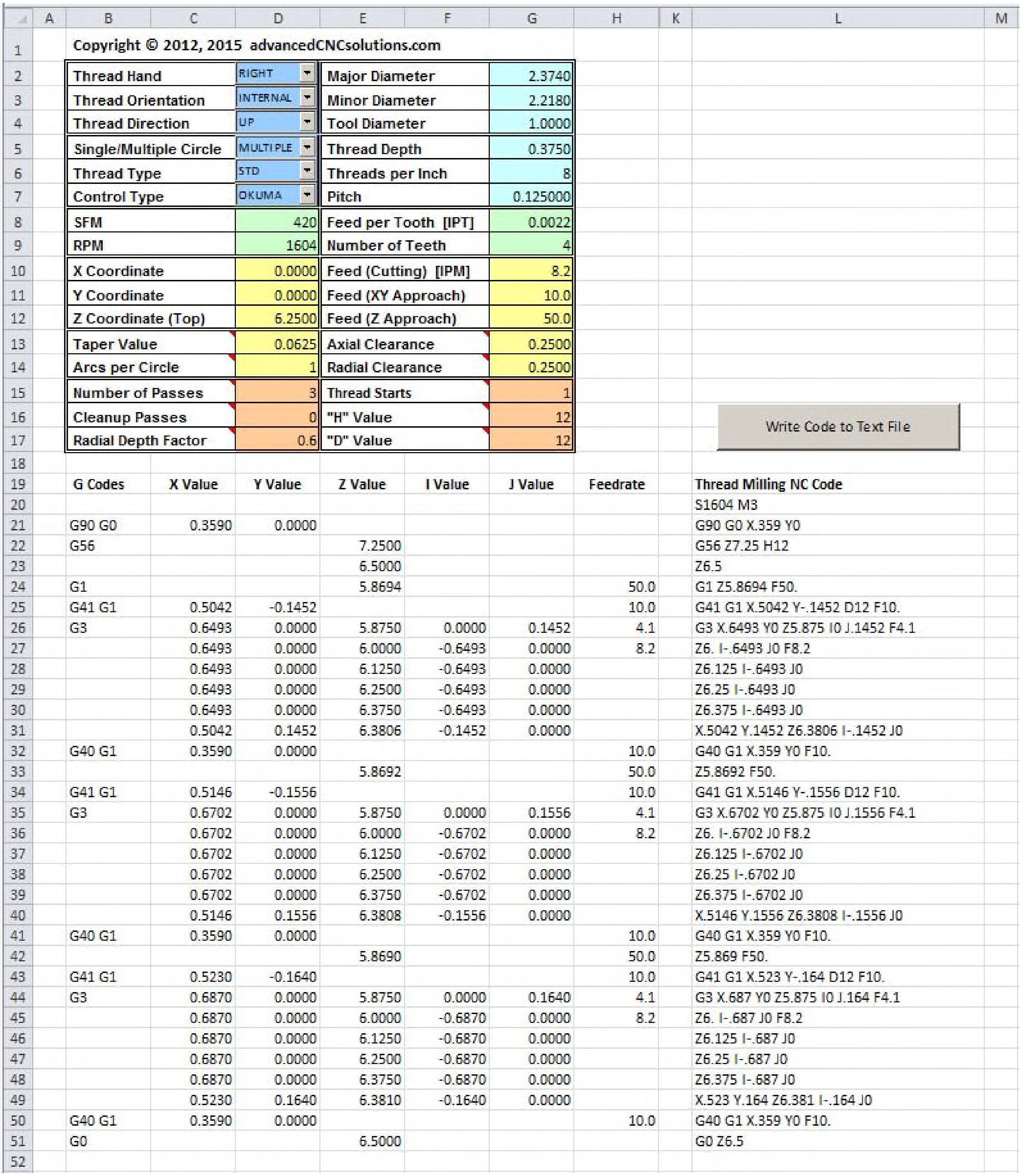
Copyright © 2012, 2015 advancedCNCsolutions.com

Thread Hand	RIGHT	Major Diameter	2.3740
Thread Orientation	INTERNAL	Minor Diameter	2.2180
Thread Direction	UP	Tool Diameter	1.0000
Single/Multiple Circle	MULTIPLE	Thread Depth	0.3750
Thread Type	STD	Threads per Inch	8
Control Type	OKUMA	Pitch	0.125000
SFM	400	Feed per Tooth [IPT]	0.0020
RPM	1528	Number of Teeth	5
X Coordinate	0.0000	Feed (Cutting) [IPM]	8.8
Y Coordinate	0.0000	Feed (XY Approach)	10.0
Z Coordinate (Top)	6.2500	Feed (Z Approach)	50.0
Taper Value	0.0625	Axial Clearance	0.2500
Arcs per Circle	1	Radial Clearance	0.2500
Number of Passes	3	Thread Starts	1
Cleanup Passes	0	"H" Value	12
Radial Depth Factor	1.0	"D" Value	12

G Codes	X Value	Y Value	Z Value	I Value	J Value	Feedrate	Thread Milling NC Code
							S1528 M3
G90 G0	0.3590	0.0000					G90 G0 X.359 Y0
G56			7.2500				G56 Z7.25 H12
			6.5000				Z6.5
G1			5.8696			50.0	G1 Z5.8696 F50.
G41 G1	0.4970	-0.1380				10.0	G41 G1 X.497 Y-.138 D12 F10.
G3	0.6350	0.0000	5.8750	0.0000	0.1380	4.4	G3 X.635 Y0 Z5.875 I0 J.138 F4.4
	0.6350	0.0000	6.0000	-0.6350	0.0000	8.8	Z6. I-.635 J0 F8.8
	0.6350	0.0000	6.1250	-0.6350	0.0000		Z6.125 I-.635 J0
	0.6350	0.0000	6.2500	-0.6350	0.0000		Z6.25 I-.635 J0
	0.6350	0.0000	6.3750	-0.6350	0.0000		Z6.375 I-.635 J0
	0.4970	0.1380	6.3804	-0.1380	0.0000		X.497 Y.138 Z6.3804 I-.138 J0
G40 G1	0.3590	0.0000				10.0	G40 G1 X.359 Y0 F10.
			5.8693			50.0	Z5.8693 F50.
G41 G1	0.5100	-0.1510				10.0	G41 G1 X.51 Y-.151 D12 F10.
G3	0.6610	0.0000	5.8750	0.0000	0.1510	4.4	G3 X.661 Y0 Z5.875 I0 J.151 F4.4
	0.6610	0.0000	6.0000	-0.6610	0.0000	8.8	Z6. I-.661 J0 F8.8
	0.6610	0.0000	6.1250	-0.6610	0.0000		Z6.125 I-.661 J0
	0.6610	0.0000	6.2500	-0.6610	0.0000		Z6.25 I-.661 J0
	0.6610	0.0000	6.3750	-0.6610	0.0000		Z6.375 I-.661 J0
	0.5100	0.1510	6.3807	-0.1510	0.0000		X.51 Y.151 Z6.3807 I-.151 J0
G40 G1	0.3590	0.0000				10.0	G40 G1 X.359 Y0 F10.
			5.8690			50.0	Z5.869 F50.
G41 G1	0.5230	-0.1640				10.0	G41 G1 X.523 Y-.164 D12 F10.
G3	0.6870	0.0000	5.8750	0.0000	0.1640	4.4	G3 X.687 Y0 Z5.875 I0 J.164 F4.4
	0.6870	0.0000	6.0000	-0.6870	0.0000	8.8	Z6. I-.687 J0 F8.8
	0.6870	0.0000	6.1250	-0.6870	0.0000		Z6.125 I-.687 J0
	0.6870	0.0000	6.2500	-0.6870	0.0000		Z6.25 I-.687 J0
	0.6870	0.0000	6.3750	-0.6870	0.0000		Z6.375 I-.687 J0
	0.5230	0.1640	6.3810	-0.1640	0.0000		X.523 Y.164 Z6.381 I-.164 J0
G40 G1	0.3590	0.0000				10.0	G40 G1 X.359 Y0 F10.
G0			6.5000				G0 Z6.5

SAMPLE #6 (Okuma control)

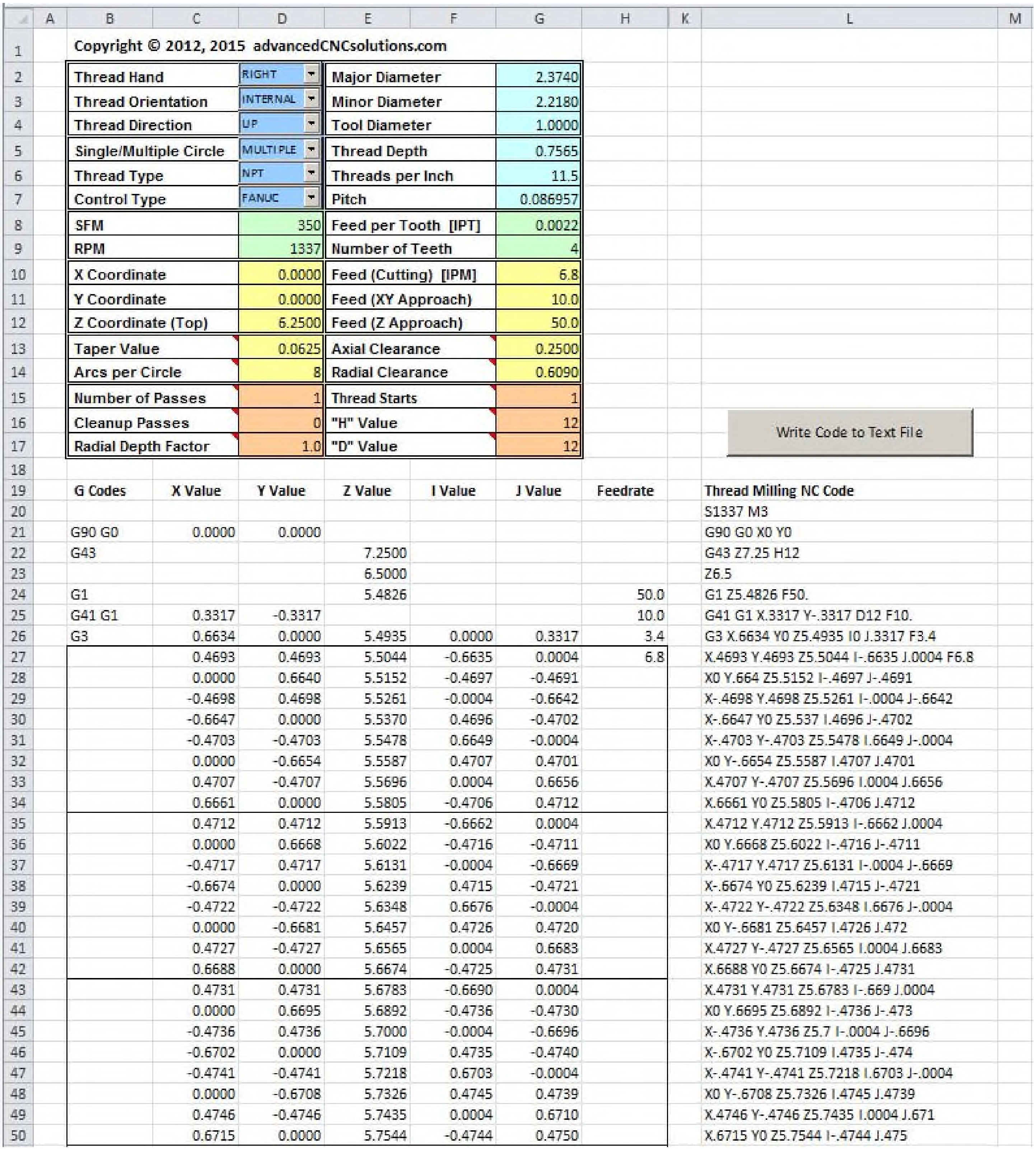
Right handed internal standard thread milling producing 3 passes using single-point thread milling tool. "Radial Depth Factor" is set to 0.6 for "modified constant volume" radial depths of cut. The 1st depth = 0.0403", 2nd depth = 0.0612" and the 3rd depth (full depth) = 0.0780" on the radius. This technique is ideal for hard to machine materials!



	A	B	C	D	E	F	G	H	K	L	M
1		Copyright © 2012, 2015 advancedCNCsolutions.com									
2		Thread Hand		RIGHT	Major Diameter		2.3740				
3		Thread Orientation		INTERNAL	Minor Diameter		2.2180				
4		Thread Direction		UP	Tool Diameter		1.0000				
5		Single/Multiple Circle		MULTIPLE	Thread Depth		0.3750				
6		Thread Type		STD	Threads per Inch		8				
7		Control Type		OKUMA	Pitch		0.125000				
8		SFM		420	Feed per Tooth [IPT]		0.0022				
9		RPM		1604	Number of Teeth		4				
10		X Coordinate		0.0000	Feed (Cutting) [IPM]		8.2				
11		Y Coordinate		0.0000	Feed (XY Approach)		10.0				
12		Z Coordinate (Top)		6.2500	Feed (Z Approach)		50.0				
13		Taper Value		0.0625	Axial Clearance		0.2500				
14		Arcs per Circle		1	Radial Clearance		0.2500				
15		Number of Passes		3	Thread Starts		1				
16		Cleanup Passes		0	"H" Value		12			Write Code to Text File	
17		Radial Depth Factor		0.6	"D" Value		12				
18											
19		G Codes	X Value	Y Value	Z Value	I Value	J Value	Feedrate		Thread Milling NC Code	
20										S1604 M3	
21		G90 G0	0.3590	0.0000						G90 G0 X.359 Y0	
22		G56			7.2500					G56 Z7.25 H12	
23					6.5000					Z6.5	
24		G1			5.8694			50.0		G1 Z5.8694 F50.	
25		G41 G1	0.5042	-0.1452				10.0		G41 G1 X.5042 Y-.1452 D12 F10.	
26		G3	0.6493	0.0000	5.8750	0.0000	0.1452	4.1		G3 X.6493 Y0 Z5.875 I0 J.1452 F4.1	
27			0.6493	0.0000	6.0000	-0.6493	0.0000	8.2		Z6. I-.6493 J0 F8.2	
28			0.6493	0.0000	6.1250	-0.6493	0.0000			Z6.125 I-.6493 J0	
29			0.6493	0.0000	6.2500	-0.6493	0.0000			Z6.25 I-.6493 J0	
30			0.6493	0.0000	6.3750	-0.6493	0.0000			Z6.375 I-.6493 J0	
31			0.5042	0.1452	6.3806	-0.1452	0.0000			X.5042 Y.1452 Z6.3806 I-.1452 J0	
32		G40 G1	0.3590	0.0000				10.0		G40 G1 X.359 Y0 F10.	
33					5.8692			50.0		Z5.8692 F50.	
34		G41 G1	0.5146	-0.1556				10.0		G41 G1 X.5146 Y-.1556 D12 F10.	
35		G3	0.6702	0.0000	5.8750	0.0000	0.1556	4.1		G3 X.6702 Y0 Z5.875 I0 J.1556 F4.1	
36			0.6702	0.0000	6.0000	-0.6702	0.0000	8.2		Z6. I-.6702 J0 F8.2	
37			0.6702	0.0000	6.1250	-0.6702	0.0000			Z6.125 I-.6702 J0	
38			0.6702	0.0000	6.2500	-0.6702	0.0000			Z6.25 I-.6702 J0	
39			0.6702	0.0000	6.3750	-0.6702	0.0000			Z6.375 I-.6702 J0	
40			0.5146	0.1556	6.3808	-0.1556	0.0000			X.5146 Y.1556 Z6.3808 I-.1556 J0	
41		G40 G1	0.3590	0.0000				10.0		G40 G1 X.359 Y0 F10.	
42					5.8690			50.0		Z5.869 F50.	
43		G41 G1	0.5230	-0.1640				10.0		G41 G1 X.523 Y-.164 D12 F10.	
44		G3	0.6870	0.0000	5.8750	0.0000	0.1640	4.1		G3 X.687 Y0 Z5.875 I0 J.164 F4.1	
45			0.6870	0.0000	6.0000	-0.6870	0.0000	8.2		Z6. I-.687 J0 F8.2	
46			0.6870	0.0000	6.1250	-0.6870	0.0000			Z6.125 I-.687 J0	
47			0.6870	0.0000	6.2500	-0.6870	0.0000			Z6.25 I-.687 J0	
48			0.6870	0.0000	6.3750	-0.6870	0.0000			Z6.375 I-.687 J0	
49			0.5230	0.1640	6.3810	-0.1640	0.0000			X.523 Y.164 Z6.381 I-.164 J0	
50		G40 G1	0.3590	0.0000				10.0		G40 G1 X.359 Y0 F10.	
51		G0			6.5000					G0 Z6.5	
52											

SAMPLE #7 (Fanuc control)

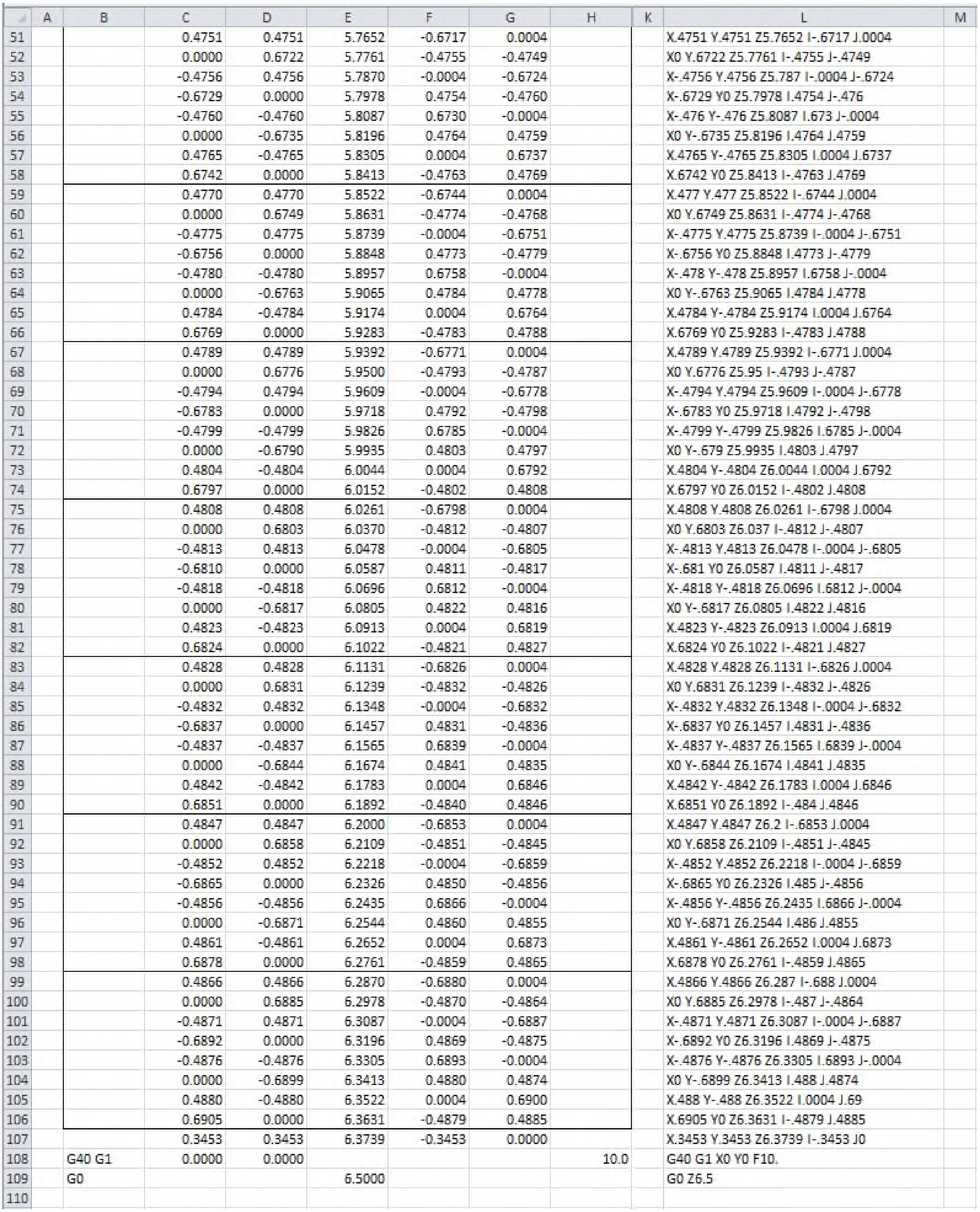
Right handed internal (2.0" x 11.5 NPT) tapered thread milling producing 8 spiral arc segment moves per full circle using single-point thread milling tool. Notice increasing spiral arc moves to produce tapered threads. With just a few "Partial Profile" style inserts on hand (in various pitch ranges) you could thread mill most any straight or NPT tapered pipe thread. This technique greatly reduces tooling costs!



Copyright © 2012, 2015 advancedCNCsolutions.com

Thread Hand	RIGHT	Major Diameter	2.3740
Thread Orientation	INTERNAL	Minor Diameter	2.2180
Thread Direction	UP	Tool Diameter	1.0000
Single/Multiple Circle	MULTIPLE	Thread Depth	0.7565
Thread Type	NPT	Threads per Inch	11.5
Control Type	FANUC	Pitch	0.086957
SFM	350	Feed per Tooth [IPT]	0.0022
RPM	1337	Number of Teeth	4
X Coordinate	0.0000	Feed (Cutting) [IPM]	6.8
Y Coordinate	0.0000	Feed (XY Approach)	10.0
Z Coordinate (Top)	6.2500	Feed (Z Approach)	50.0
Taper Value	0.0625	Axial Clearance	0.2500
Arcs per Circle	8	Radial Clearance	0.6090
Number of Passes	1	Thread Starts	1
Cleanup Passes	0	"H" Value	12
Radial Depth Factor	1.0	"D" Value	12

G Codes	X Value	Y Value	Z Value	I Value	J Value	Feedrate	Thread Milling NC Code
							S1337 M3
G90 G0	0.0000	0.0000					G90 G0 X0 Y0
G43			7.2500				G43 Z7.25 H12
			6.5000				Z6.5
G1			5.4826			50.0	G1 Z5.4826 F50.
G41 G1	0.3317	-0.3317				10.0	G41 G1 X.3317 Y-.3317 D12 F10.
G3	0.6634	0.0000	5.4935	0.0000	0.3317	3.4	G3 X.6634 Y0 Z5.4935 I0 J.3317 F3.4
	0.4693	0.4693	5.5044	-0.6635	0.0004	6.8	X.4693 Y.4693 Z5.5044 I-.6635 J.0004 F6.8
	0.0000	0.6640	5.5152	-0.4697	-0.4691		X0 Y.664 Z5.5152 I-.4697 J-.4691
	-0.4698	0.4698	5.5261	-0.0004	-0.6642		X-.4698 Y.4698 Z5.5261 I-.0004 J-.6642
	-0.6647	0.0000	5.5370	0.4696	-0.4702		X-.6647 Y0 Z5.537 I.4696 J-.4702
	-0.4703	-0.4703	5.5478	0.6649	-0.0004		X-.4703 Y-.4703 Z5.5478 I.6649 J-.0004
	0.0000	-0.6654	5.5587	0.4707	0.4701		X0 Y-.6654 Z5.5587 I.4707 J.4701
	0.4707	-0.4707	5.5696	0.0004	0.6656		X.4707 Y-.4707 Z5.5696 I.0004 J.6656
	0.6661	0.0000	5.5805	-0.4706	0.4712		X.6661 Y0 Z5.5805 I-.4706 J.4712
	0.4712	0.4712	5.5913	-0.6662	0.0004		X.4712 Y.4712 Z5.5913 I-.6662 J.0004
	0.0000	0.6668	5.6022	-0.4716	-0.4711		X0 Y.6668 Z5.6022 I-.4716 J-.4711
	-0.4717	0.4717	5.6131	-0.0004	-0.6669		X-.4717 Y.4717 Z5.6131 I-.0004 J-.6669
	-0.6674	0.0000	5.6239	0.4715	-0.4721		X-.6674 Y0 Z5.6239 I.4715 J-.4721
	-0.4722	-0.4722	5.6348	0.6676	-0.0004		X-.4722 Y-.4722 Z5.6348 I.6676 J-.0004
	0.0000	-0.6681	5.6457	0.4726	0.4720		X0 Y-.6681 Z5.6457 I.4726 J.472
	0.4727	-0.4727	5.6565	0.0004	0.6683		X.4727 Y-.4727 Z5.6565 I.0004 J.6683
	0.6688	0.0000	5.6674	-0.4725	0.4731		X.6688 Y0 Z5.6674 I-.4725 J.4731
	0.4731	0.4731	5.6783	-0.6690	0.0004		X.4731 Y.4731 Z5.6783 I-.669 J.0004
	0.0000	0.6695	5.6892	-0.4736	-0.4730		X0 Y.6695 Z5.6892 I-.4736 J-.473
	-0.4736	0.4736	5.7000	-0.0004	-0.6696		X-.4736 Y.4736 Z5.7 I-.0004 J-.6696
	-0.6702	0.0000	5.7109	0.4735	-0.4740		X-.6702 Y0 Z5.7109 I.4735 J-.474
	-0.4741	-0.4741	5.7218	0.6703	-0.0004		X-.4741 Y-.4741 Z5.7218 I.6703 J-.0004
	0.0000	-0.6708	5.7326	0.4745	0.4739		X0 Y-.6708 Z5.7326 I.4745 J.4739
	0.4746	-0.4746	5.7435	0.0004	0.6710		X.4746 Y-.4746 Z5.7435 I.0004 J.671
	0.6715	0.0000	5.7544	-0.4744	0.4750		X.6715 Y0 Z5.7544 I-.4744 J.475



	A	B	C	D	E	F	G	H	K	L	M
51			0.4751	0.4751	5.7652	-0.6717	0.0004			X.4751 Y.4751 Z5.7652 I-.6717 J.0004	
52			0.0000	0.6722	5.7761	-0.4755	-0.4749			X0 Y.6722 Z5.7761 I-.4755 J-.4749	
53			-0.4756	0.4756	5.7870	-0.0004	-0.6724			X-.4756 Y.4756 Z5.787 I-.0004 J-.6724	
54			-0.6729	0.0000	5.7978	0.4754	-0.4760			X-.6729 Y0 Z5.7978 I.4754 J-.476	
55			-0.4760	-0.4760	5.8087	0.6730	-0.0004			X-.476 Y-.476 Z5.8087 I.673 J-.0004	
56			0.0000	-0.6735	5.8196	0.4764	0.4759			X0 Y-.6735 Z5.8196 I.4764 J.4759	
57			0.4765	-0.4765	5.8305	0.0004	0.6737			X.4765 Y-.4765 Z5.8305 I.0004 J.6737	
58			0.6742	0.0000	5.8413	-0.4763	0.4769			X.6742 Y0 Z5.8413 I-.4763 J.4769	
59			0.4770	0.4770	5.8522	-0.6744	0.0004			X.477 Y.477 Z5.8522 I-.6744 J.0004	
60			0.0000	0.6749	5.8631	-0.4774	-0.4768			X0 Y.6749 Z5.8631 I-.4774 J-.4768	
61			-0.4775	0.4775	5.8739	-0.0004	-0.6751			X-.4775 Y.4775 Z5.8739 I-.0004 J-.6751	
62			-0.6756	0.0000	5.8848	0.4773	-0.4779			X-.6756 Y0 Z5.8848 I.4773 J-.4779	
63			-0.4780	-0.4780	5.8957	0.6758	-0.0004			X-.478 Y-.478 Z5.8957 I.6758 J-.0004	
64			0.0000	-0.6763	5.9065	0.4784	0.4778			X0 Y-.6763 Z5.9065 I.4784 J.4778	
65			0.4784	-0.4784	5.9174	0.0004	0.6764			X.4784 Y-.4784 Z5.9174 I.0004 J.6764	
66			0.6769	0.0000	5.9283	-0.4783	0.4788			X.6769 Y0 Z5.9283 I-.4783 J.4788	
67			0.4789	0.4789	5.9392	-0.6771	0.0004			X.4789 Y.4789 Z5.9392 I-.6771 J.0004	
68			0.0000	0.6776	5.9500	-0.4793	-0.4787			X0 Y.6776 Z5.95 I-.4793 J-.4787	
69			-0.4794	0.4794	5.9609	-0.0004	-0.6778			X-.4794 Y.4794 Z5.9609 I-.0004 J-.6778	
70			-0.6783	0.0000	5.9718	0.4792	-0.4798			X-.6783 Y0 Z5.9718 I.4792 J-.4798	
71			-0.4799	-0.4799	5.9826	0.6785	-0.0004			X-.4799 Y-.4799 Z5.9826 I.6785 J-.0004	
72			0.0000	-0.6790	5.9935	0.4803	0.4797			X0 Y-.679 Z5.9935 I.4803 J.4797	
73			0.4804	-0.4804	6.0044	0.0004	0.6792			X.4804 Y-.4804 Z6.0044 I.0004 J.6792	
74			0.6797	0.0000	6.0152	-0.4802	0.4808			X.6797 Y0 Z6.0152 I-.4802 J.4808	
75			0.4808	0.4808	6.0261	-0.6798	0.0004			X.4808 Y.4808 Z6.0261 I-.6798 J.0004	
76			0.0000	0.6803	6.0370	-0.4812	-0.4807			X0 Y.6803 Z6.037 I-.4812 J-.4807	
77			-0.4813	0.4813	6.0478	-0.0004	-0.6805			X-.4813 Y.4813 Z6.0478 I-.0004 J-.6805	
78			-0.6810	0.0000	6.0587	0.4811	-0.4817			X-.681 Y0 Z6.0587 I.4811 J-.4817	
79			-0.4818	-0.4818	6.0696	0.6812	-0.0004			X-.4818 Y-.4818 Z6.0696 I.6812 J-.0004	
80			0.0000	-0.6817	6.0805	0.4822	0.4816			X0 Y-.6817 Z6.0805 I.4822 J.4816	
81			0.4823	-0.4823	6.0913	0.0004	0.6819			X.4823 Y-.4823 Z6.0913 I.0004 J.6819	
82			0.6824	0.0000	6.1022	-0.4821	0.4827			X.6824 Y0 Z6.1022 I-.4821 J.4827	
83			0.4828	0.4828	6.1131	-0.6826	0.0004			X.4828 Y.4828 Z6.1131 I-.6826 J.0004	
84			0.0000	0.6831	6.1239	-0.4832	-0.4826			X0 Y.6831 Z6.1239 I-.4832 J-.4826	
85			-0.4832	0.4832	6.1348	-0.0004	-0.6832			X-.4832 Y.4832 Z6.1348 I-.0004 J-.6832	
86			-0.6837	0.0000	6.1457	0.4831	-0.4836			X-.6837 Y0 Z6.1457 I.4831 J-.4836	
87			-0.4837	-0.4837	6.1565	0.6839	-0.0004			X-.4837 Y-.4837 Z6.1565 I.6839 J-.0004	
88			0.0000	-0.6844	6.1674	0.4841	0.4835			X0 Y-.6844 Z6.1674 I.4841 J.4835	
89			0.4842	-0.4842	6.1783	0.0004	0.6846			X.4842 Y-.4842 Z6.1783 I.0004 J.6846	
90			0.6851	0.0000	6.1892	-0.4840	0.4846			X.6851 Y0 Z6.1892 I-.484 J.4846	
91			0.4847	0.4847	6.2000	-0.6853	0.0004			X.4847 Y.4847 Z6.2 I-.6853 J.0004	
92			0.0000	0.6858	6.2109	-0.4851	-0.4845			X0 Y.6858 Z6.2109 I-.4851 J-.4845	
93			-0.4852	0.4852	6.2218	-0.0004	-0.6859			X-.4852 Y.4852 Z6.2218 I-.0004 J-.6859	
94			-0.6865	0.0000	6.2326	0.4850	-0.4856			X-.6865 Y0 Z6.2326 I.485 J-.4856	
95			-0.4856	-0.4856	6.2435	0.6866	-0.0004			X-.4856 Y-.4856 Z6.2435 I.6866 J-.0004	
96			0.0000	-0.6871	6.2544	0.4860	0.4855			X0 Y-.6871 Z6.2544 I.486 J.4855	
97			0.4861	-0.4861	6.2652	0.0004	0.6873			X.4861 Y-.4861 Z6.2652 I.0004 J.6873	
98			0.6878	0.0000	6.2761	-0.4859	0.4865			X.6878 Y0 Z6.2761 I-.4859 J.4865	
99			0.4866	0.4866	6.2870	-0.6880	0.0004			X.4866 Y.4866 Z6.287 I-.688 J.0004	
100			0.0000	0.6885	6.2978	-0.4870	-0.4864			X0 Y.6885 Z6.2978 I-.487 J-.4864	
101			-0.4871	0.4871	6.3087	-0.0004	-0.6887			X-.4871 Y.4871 Z6.3087 I-.0004 J-.6887	
102			-0.6892	0.0000	6.3196	0.4869	-0.4875			X-.6892 Y0 Z6.3196 I.4869 J-.4875	
103			-0.4876	-0.4876	6.3305	0.6893	-0.0004			X-.4876 Y-.4876 Z6.3305 I.6893 J-.0004	
104			0.0000	-0.6899	6.3413	0.4880	0.4874			X0 Y-.6899 Z6.3413 I.488 J.4874	
105			0.4880	-0.4880	6.3522	0.0004	0.6900			X.488 Y-.488 Z6.3522 I.0004 J.69	
106			0.6905	0.0000	6.3631	-0.4879	0.4885			X.6905 Y0 Z6.3631 I-.4879 J.4885	
107			0.3453	0.3453	6.3739	-0.3453	0.0000			X.3453 Y.3453 Z6.3739 I-.3453 J0	
108		G40 G1	0.0000	0.0000				10.0		G40 G1 X0 Y0 F10.	
109		G0			6.5000					G0 Z6.5	
110											

advancedCNCsolutions.com