

THREADING ON THE LATHE LATHE-MACH3 TURN



TABLE OF CONTENTS

SECTION	TOPIC	PAGE
1.0	PREFACE	3
2.0	MACH THREADING	4
2.1	HOW IT WORKS	4
3.0	TESTING YOUR LATHE	5
3.1	TEST EQUIPMENT	
3.2	STEPS PER UNIT VALUE – USING MACH MILL	5
3.3	AXIS TESTS	6
3.4	X & Z AXIS TESTS	7
	Z AXIS TEST	
	AXIS LOADING TEST	
	X AXIS TEST	
3.5	TRIGGERING TEST	9
3.6	SCRIBING	10
3.6.1	LEAD ERROR TESTING	10
3.6.2	ALTERNATE FLANK THREAD CUTTING TEST	13
3.6.3	MULTIPLE THREADS TEST	13
3.6.4	PICKING UP A THREAD SCRIBE TEST	15
3.7	TESTING – CS / AL RESULTS	16
3.8	SPINDLE RPM	19
3.9	MOTOR- GENERAL SLOWDOWN / POWER / EFFECT ON TREAD'G	20
4.0	THREAD BASICS	20
4.1	STANDARDS – DEFINITONS	20
4.2	DEPTH OF CUT BASIS	22
4.3	MEASURING THE THREAD	23
4.4	TOLERANCE	24
5.0	THREAD CUTTING	27
5.1	THREAD CUTTING FEED METHODS	27
5.2	SPINDLE MOTION / TURNING METHODS	28
5.3	CHIP FORMATION	29
5.4	FORMULAS	30
5.5	THREAD CUTTERS / TIP RADIUS	30
5.6	WORK HOLDING	31
6.0	GCODE – MACH THREADING WIZARDS & CANNED CYCLES	32
6.1	WIZARDS	32
6.2	G76 THREADING CYCLE	33
6.2.1	THREADING DEFAULTS	
6.3	METHOD CHOICE	34
6.4	SIMPLE THREADING (LATHE) WIZARD	35
6.5	QUICK THREADS WIZARD	37
6.6	HELPFUL INFO / PROGRAMS	39
7.0	MACH3 TURN CONFIGURATION	42
7.1	CONFIGURATION	43
7.2	MODIFYING M1076 MACRO	47
8.0	MULTI START THREADING	48
9.0	HOW TO PICK UP A THREAD	49
10.0	REFERENCES	51
11.0	APPENDIX LIST	
A –	LATHE SPECIFICATION , TESTING & TOLERANCES	52
B –	JIS STANDARD TOLERANCES LATHES	55
C –	INITIALIZATION MACRO	57

1.0 PREFACE

This writing is done to provide a general insight into threading. Threading is a complex machining operation if you look at the big picture of what is involved. Hopefully this will provide some insight into on how it is all related, and thus the Mach user can be successful at machining threads on the lathe.

The document is a collection of many threads and replies on the Mach Forum and is supplemented by a lot of information from manufactures, books, and experience. There are books and plenty of reference sources available for reading. This only covers single point threading. The writing is tailored to the user of MACH3 TURN, and in that light, you will find some undocumented information and answers to questions that otherwise would be difficult to search for.

I plagiarized and borrowed pictures with pride through out the write-up. So don't think for a moment that I am expert on what is not a simple subject.

You will find in the write-up "WW" which stands for "WISHY WASHY". Some things are not straight forward and vary because of how they are related. So WW just provides discussion on some subject matter. It will be in a finer print.

This content of this writing is limited in subject matter and should be used as a supplement to the existing "Using Mach3 Turn Manual". The user should also read the test file named "MachTurn" which can be found in the in the Mach3 directory for a quick "get started" guide on the lathe.

Have Fun Doing Threads,
RICH

2.0 MACH THREADING

2.1 HOW IT WORKS

CNC threading is just like manual threading only the process is automated.

A gcode file defining the axis moves along with related thread information is read by MACH. The index pulse provides Mach with rpm data and the program controls the Z axis to a move appropriately from a dead start, accelerate to a defined distance, and then maintain a feedrate such that the cutting tool produces a spiral cut along a cylinder representing the lead of the thread. The start of the Z motion happens when and if a timing pulse is seen. If no pulse is seen the threading will not begin or continue. The timing pulse synchronizes the z axis location to the spindle rotation the same as closing of the half nuts on a manual lathe would do. So, the threading is activated with an index pulse. As the defined length of thread is reached the controller moves the tool out of the thread based on the gcode file. Thus the X axis retracts while the Z axis is still moving but over a defined distance. The Z axis moves the tool back to some location, the X axis moves the tool in or out, the Z axis moves to the original starting location. Axis movement now stops until an index again “triggers” Mach to repeat the threading cycle.

One complete thread cycle or pass is basically composed of the following:

Trigger – index pulse is seen and activate start of movement

Accelerate – move to an exact Z location relative to the turning spindle

Threading – move / control the tool such that the feedrate is correct relative to spindle rpm

Pullout – the tool is removed at the end of the thread

Retract – the tool is moved back to a starting point for repeat of the cycle

During the threading the rpm is monitored by the controller for variations and Mach plans on how to modify the next threading pass such that the Z axis movement will maintain the lead of the screw. Testing has shown that the lead is tightly controlled to a fine tolerance such that a near perfect thread can be produced if the lathe system is capable of it. Should the spindle slow down, Mach will change the Z movement to try maintaining the lead. Spindle slowdown in the range of 10 to 75% may be the range, but, as of this writing has not been tested. Past testing of past Mach versions on spindle slowdown is relative but not definitive for the new threading version.

To accomplish the necessary axis movements a gcode file is written or generated using a particular threading method. There are different gcodes and threading cutting methods all of which define the X and Z axis movements used in the threading cycle and how many passes / cycles will occur.

The remainder of this write-up provides additional information which influences threading.

WW:

You can't compare different controller programs. The control scheme may use an external device / hardware and doesn't mean anything, other than to say, with another system you get some kind of threading. It would be like comparing apples and oranges. Same goes for higher end CNC lathe systems. A statement saying that perfect threading was done is a different “fruit” many times. This writing only covers using the PP (parallel port) along with the threading application.

Don't care what your CNC lathe “SYSTEM” is like, your thread lead will only be as good as the screw / ball screw that drives the axis, how well that movement is implemented by Mach, and all the other electronic / mechanical items associated with that movement. It can quickly get complex, the stepper motor, the pulleys and their belts, the timing sensor, spindle motor and belts (variations in the motors rpm's and power / torque), backlash, etc. So it becomes a matter of degree as to the influences of those items. Checking each

of the items may not even be practical or even possible for the average user. To simplify it all you can check the lathe ‘system’ and the controlling system.

If you can confirm cutting, such that scribing of many passes, provides a single cut line, and is repeatable and measurable, then the lathe as a system is refined to a rather high level and can be used as a base relative to the controlling software. There will always be inaccuracy in both the lathe system and the controlling system. As the inaccuracy decreases it gets more difficult to identify the cause such that a change on the software side may not be perfect based on a non perfect lathe system.

One could say that if the nut goes on the thread it’s fine while another would say the nut needs to track the thread perfectly with no play. Yet neither of those may meet a designed intent. I guess it’s a matter of degree. There are a lot variables ie; the lathe, the type of cutter, experience, etc. that can have a big influence on the actual cutting of the thread .So it comes down to standards and not personal opinion.

Lets say the lathe is perfect. Then the software side of it needs to be able to control to some level such that it can control to suit some standard. The software side has been tested and as such can be transparent to the user. Thus there is no need to dwell into software details on how Mach threading works.

3.0 TESTING YOUR LATHE

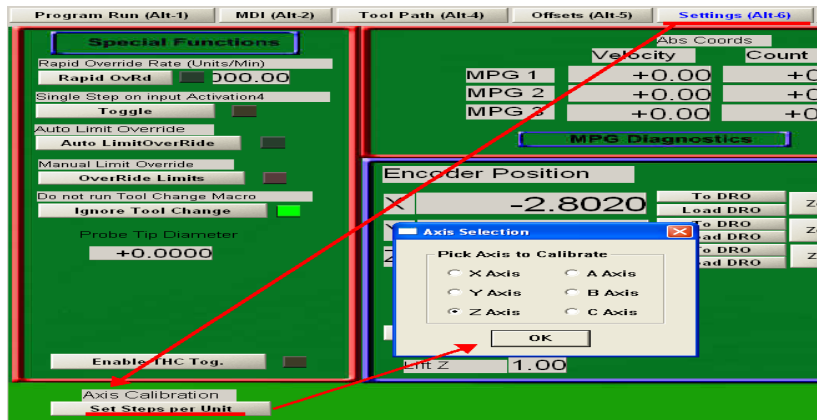
There are numerous sources of information which explain how to test and adjust a lathe. Manufactures do lathe tests based on standards and may provide an inspection report. See the attachments and references in the appendix section for standards, testing and tolerances. This information can be used to assess your lathe. What someone else has is of no value.

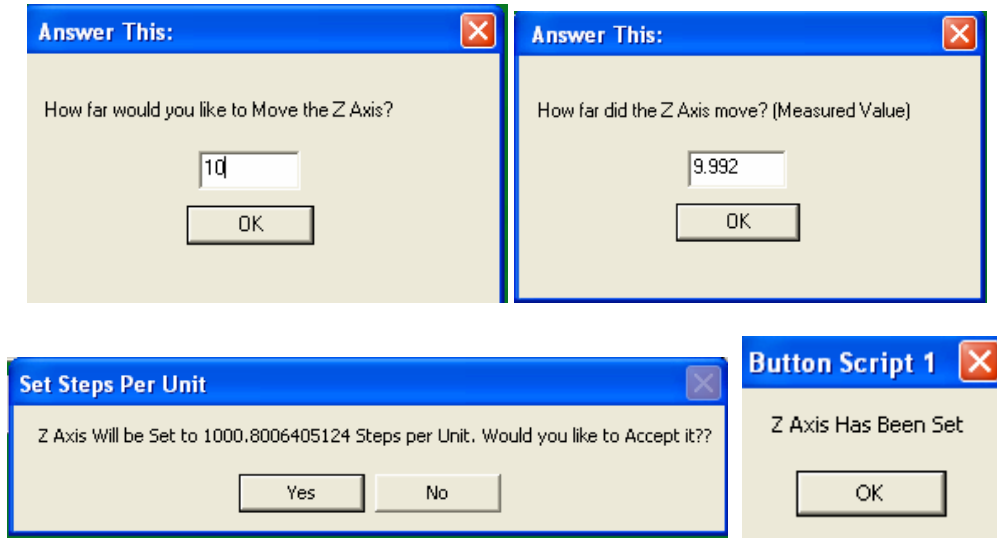
3.1 TEST EQUIPMENT

It’s assumed you have at least a 1” dial indicator and micrometer which reads to 0.0001”. Additionally, you should have a quality 20-30X magnifier.

3.2 STEPS PER UNIT VALUE – USING MACH MILL

You need to set the steps per unit for your axis accurately and that is covered in the Using Mach3 Turn Manual. For longer and even short steps per unit checks you can use the axis calibration in Mach Mill. You just use the Settings tab and click on Set Steps per Unit, tell it how far you want to move, and then how far the axis actually moved. Mach calculates the steps per unit. If you accept Mach’s calculation then the settings will appear in the motor tuning for that axis. You can then use the value in Mach 3 Turn. This is all shown in the figures below. You can use an accurate scale that reads in 100ths and read the scale within .005” easily for longer distances.





3.3 AXIS TESTS

You can tell a lot about what your lathe system will do with just a 20-30x magnifier and inspection of some scribing. A pocket comparator with a scale in 0.001” increments is also handy. So you do not need a lot fancy equipment.

Turning tests on the lathe should be done as noted in Appendix “A”. The tests that will follow are done for a different reason and relate to threading on a CNC lathe.

Note the following:

So the motor tuning is all done and the axis steps per unit are correct. You have checked for backlash and maybe you need or choose to use it. Of course before you did all that you adjusted any gibs, checked all the belts and pulleys, etc. You know what your spindle run out is and also how well you can turn to diameter over a distance. You may as well check how good the chuck centers a ground test bar of various sizes. Adjusted the head and tail stock if possible. Know the center height of the axis so you can set a threading tool accurately. How to do all that is beyond this writing, but, threading will only be as good as your “lathe system”.

In any testing, safety is important, so irrelevant of what is written, think before you do anything. Safety is 100% your responsibility!

WW:

Threading is a true test of your equipment and the finished threading will show it. Consider this: For a Class 3 external ¼-20 UNC x 1” long thread the pitch diameter can only vary by 0.0026”. If the lathe taper cuts 0.001” / inch then that only leaves the remainder of what’s involved in threading 0.0016” and you have not even cut the screw. That won’t leave much for inexperience on setting the tool, flex of the material during cutting, backlash, or anything else. So you need everything going for you before you even start.

3.4 X & Z AXIS TESTS

These tests check how an axis is working as a “system”. You will need a dial indicator that reads to 0.0001”. They are different because they include triggering, acceleration / deceleration, positioning, etc. They don’t isolate one particular part of a the movement. You can use any spindle rpm, but, the axis must be able to move at the requested feedrate. You can confirm this by using the Simple Threading Wizard (section 6.4) since it will warn you if you exceed the settings in motor tuning.

Z AXIS TEST

The program just runs the Z axis back and forth for a distance of 1”, twenty times, and will stop for 4 seconds so you can see the axis position on the indicator in test #1 and will stop twice in test #2. Test#2 relates to alternate flank cutting and in this test the difference between readings should be 0.0005”. Figure 3.4.1 shows the indicator set to zero.

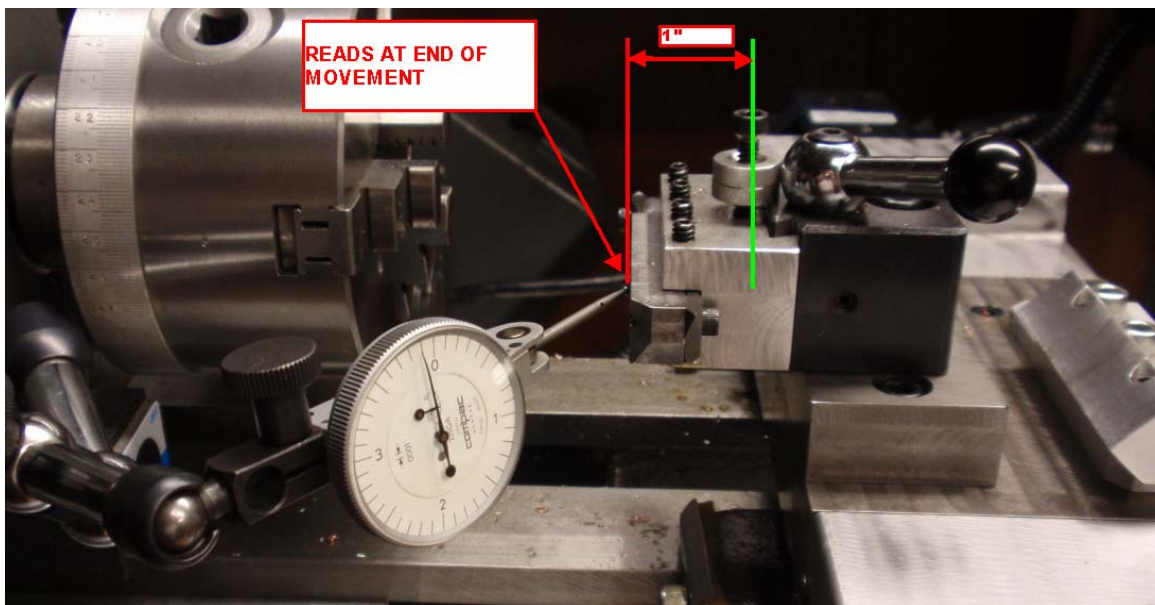


FIGURE 3.4.1

```

N10 (Z AXIS TEST NO 1 )
N20 M3 G18 G20 G40 G49 G61 G80 G90 G94
N30 M98 P01 L10
N40 M30
O01
G32 Z-1 F 0.1
G95
G4 P4
G00 Z 0.001
G94
G32 Z-1.0 F.1
G95
G00 Z0.0
N130 M99

```

```

N10 (Z AXIS TEST NO 2 )
N20 M3 G18 G20 G40 G49 G61 G80 G90 G94
N30 M98 P02 L10
N40 M30
O02
G32 Z-0.9995 F 0.1
G95
G4 P4
G00 Z 0.001
G94
G32 Z-1.0 F 0.1
G95
G4 P4
G00 Z0.0
N140 M99
    
```

How much the readings vary will give you an indication of your lathe “system”. My test’s show a change in the reading of 0.0001” for test #1 and the difference in test # 2 is also just 0.0001” (ie; 0.0004” instead of 0.0005”). In alternate flank cutting, the gcode change in Z may only be .001”, so if the axis movement can’t hold below that, alternate flank cutting may not be a good threading method for your use.

AXIS LOAD TEST

Here is just another simple test. Push into the axis as shown in figure 3.4.2. A hard push will be in the range of 30 to 45#. When a deep thread is cut the axial load can easily be 2 to 4x that amount. SO, if you can see indicator movement, and you will, that same movement will occur during threading. It will have an effect on the cutting, thread finish, rpm stability, etc. So even if the axis tests showed no variation, there will play due to lack of equipment rigidity. This becomes more important with smaller lathes. Don’t confuse this with backlash. The user should do this for Z & X and also push directly down. Tool forces are shown in figure 3.4.3.

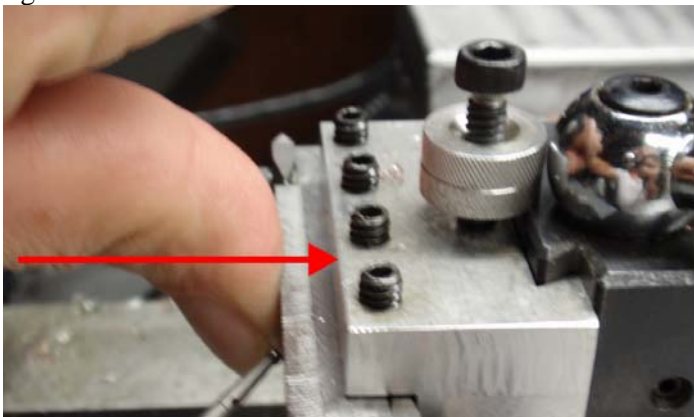


FIGURE 3.4.2

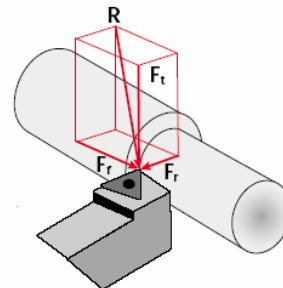


FIGURE 3.4.3

You may also want to mount a piece into the chuck and do the equivalent by pushing on the work piece to see just how rigid different setups are (see Work Holding in Section 5.6).

X AXIS TEST

3.5 TRIGGERING TEST

You can test if triggering is functional. More advanced testing is beyond this write up. The Turn Diagnostics (see section 3.7) confirms that triggering is functioning during threading. You can use the diagnostics screen since the indicating light will turn on and off as you manually turn the spindle as shown in Figure 3.5.1 when in the G94 or G95 mode.

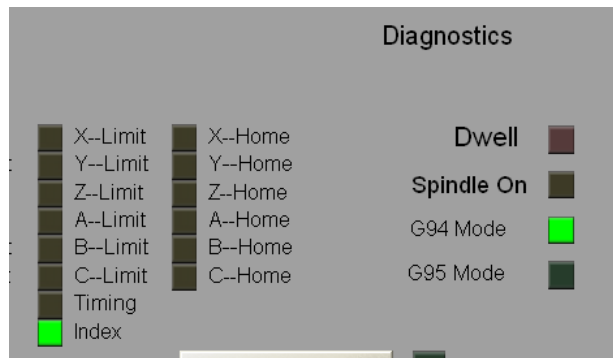


FIGURE 3.5.1

The user should check the triggering as to when it just turns on and off as exercise. Watch the diagnostics screen while manually turning the spindle, and when it just turns on off, place marks say on a piece of tape and note the midway distance between the marks. This is shown in figure 3.3.2. An indexing circle is attached to the chuck in the picture and provides a rather precise measurement. You don't know where the "exact trigger occurs" but the marks are relative and quite repeatable. Why do this? Later in the write it will be used to quickly "get in the ball park" for picking up a thread.



FIGURE 3.5.2

3.6 SCRIBING

The following equipment was used for the scribing tests. Axis movement was checked “as noted above” but the z axis movement was confirmed over a range using calibrated optical alignment equipment and scales for incremental movements along with calibrated indicators. So measurements were viewed at 40X and to 0.0001”. Besides the steps per unit setting, the ball screw was actually profiled incrementally for a number of typical pitch’s. Scribing movement was monitored via a 30x microscope with 0.001” scale divisions and mounted on the carriage. Scribed lines representing the lead were measured using a Gartner Toolmakers microscope. Multiple scribed lines were at times measured / distinguished using a stereo microscope with a calibrated fical micrometer and scaled eyepieces.

3.6.1 LEAD ERROR TESTING

The following test results were posted on the forum for five scribing tests as shown in Figure 3.6.1.1. Approx 20 passes @.0002" / pass Spindle Speed Averaging / Constant Velocity / Debounce Interval=600 Index Debounce=10 Z=60 IPM @ 6 accel / X =80 IPM @ 8 accel

402RPM / 20 PASSES / 0.1, .050, & .025 PITCH
115RPM / 20 PASSES / 0.1, 0.50

The pitch error is for those tests were for practical purposes "0". ie; at 1.5" there may be a slight lead error of -0.0004 or +0.0002 total.

Note that two test were done on two of the pieces but the individual lines are single scribed lines.

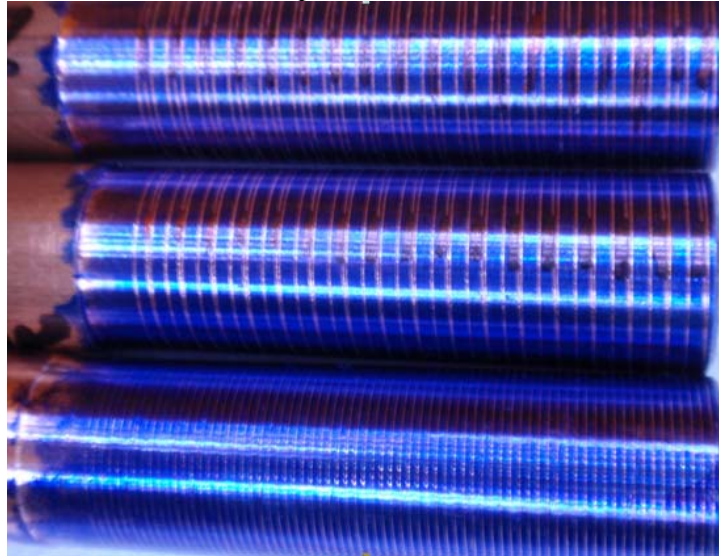


FIGURE 3.6.1.1

The tool used for scribing should have an extremely sharp pointed tip. As shown in figure 3.6.1.2 The material used for the threading test was ½” (.625 OD) copper tubing which scribes nicely, is easy to machine, and it can be used multiple times.

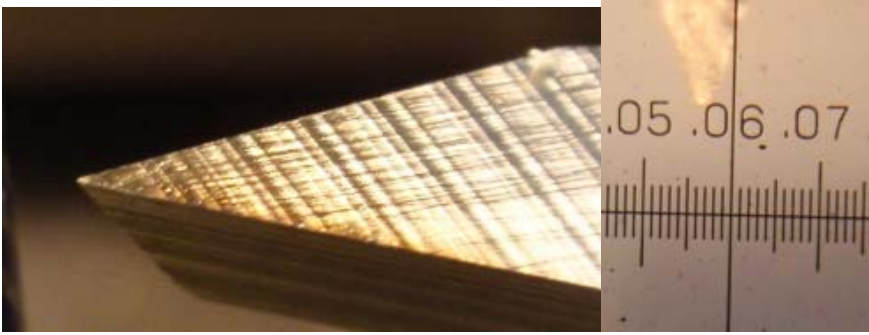


FIGURE 3.6.1.2

The scribed line should be a single line as shown and magnified in figure 3.6.1.3. Note that the single line is the result of doing 20 passes. Widening of the scribed line from beginning to end should not occur as shown in figure 3.6.1.4. If this is the case then you have backlash, a poor screw, or even a setup / rigidity problem and the lathe should be looked at very carefully.



FIGURE 3.6.1.3

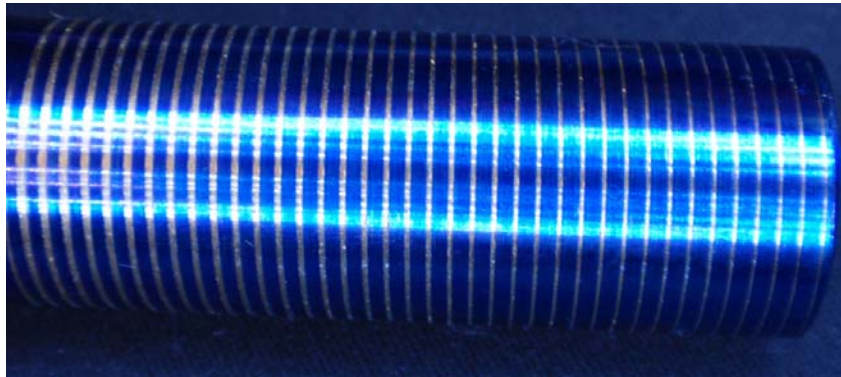


FIGURE 3.6.1.4

Figure 3.6.1.5 & 6 shows a magnified view of a multiple and single line scribe. Both are the result of 20 passes and only 0.004” deep.

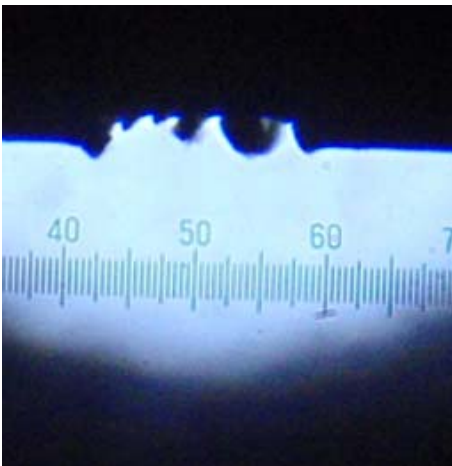


FIGURE 3.6.1.5

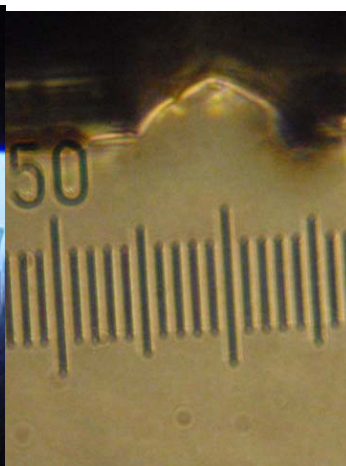


FIGURE 3.6.1.6

Figure 3.6.1.7 shows single scribed lines which were done at a pitch of .1 and 2.0. Testing covered 80 to greater than 10 TPI.

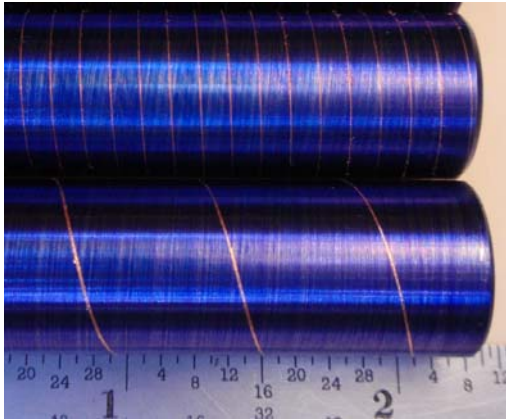


FIGURE 3.6.1.7

Figure 3.6.1.8 shows a magnified shadow graph of a good thread. Note that you can plainly see a small step change on the back flank which was due to backlash.

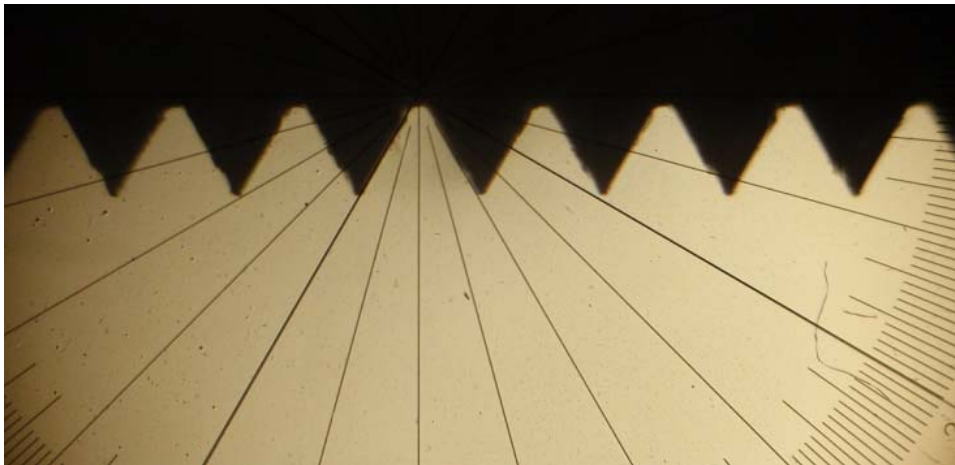


FIGURE 3.6.1.8

A users scribing will be only as good as your “lathe system” and can tell you a lot about what may be right or wrong. The lead will only be as good as the lead of your ball screw ie; for a ball screw it can be in the rough range of 0.0003”/foot to 0.003” per foot, the lathe screw can be ground or may be just a threaded rod. My 35 year old manual 11” Delta lathe has a ground screw, is in good condition, and scribed a single line with a lead error of only 0.0003” over a 6” length.

3.6.2 ALTERNATE THREAD CUTTING TEST

Alternate thread cutting can be an effective way of machining deeper cut threads. Recommended use is for 8 TPI and under, but on a small lathe that general rule doesn't apply! The 3/8 -16 UNC & 1/2-13 UNC in figure 3.6.2.1 show good results / Class 2A. The 3/4-10 UNC in figure 3.6.2.2 is Class 1A but the front and back flank finish is poor and worthy of some comments.

WW:

For a thread with deep cutting (3/4 -10 was 0.0866 deep) your lathe needs to be tight. Any looseness will show up on the thread finish. The waviness on the flanks, while not much, was a result of play in the Z axis gibs, and could be misinterpreted as deflection of the material. You can also have some deflection due to how the headstock bearings were preloaded or even in the chuck mount. What type cutting method, rpm used , and cutting fluid all play into threading and the resulting finish. The scribing test noted here will not show lathe play, but, will show if you lathe is accurately positioning the axis for the alternating cuts. The user should visually monitor the chip produced as cutting progresses since you can see if it is cutting the front or back side of the thread flank. A lathe with backlash / or incorrect backlash settings can create problems. The Z offset in the gcode may only be 0.001" ,thus, the cutting is not actually being done as the code specifies. Incorrect cutting by "your lathe system will create lead and pitch error even though the and OD and ID of the thread is correct / in tolerance.

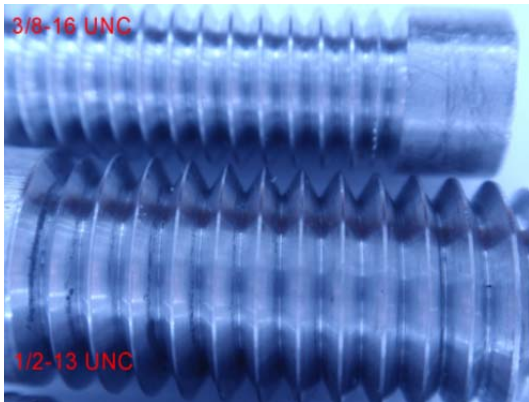


FIGURE 3.6.2.1

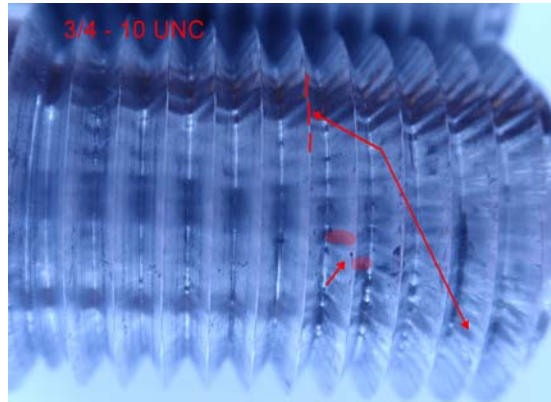


FIGURE 3.6.2.2

You should have done the Z Axis Test #2 in section 3.4, but, nothing will be better than to actually try cutting a deep thread. Pay attention to the curl of the produced chip during each pass.

3.6.3 MULTIPLE THREADS TEST

This test will show if your lathe system is capable of multiple threads. Multiple threads require the Z start point be shifted or the triggering time shifted ie; for two threads, triggering would be delayed or advanced by $360 / 2 = 180$ degrees whatever the method used for the Gcode program. It is important that leads are maintained and cutting quality duplicated for each completed threading cycle. One "quick indicator" is to look at the front of the thread and see if the individual starting points are scribed equally about the circumference. You should check the spacing between each thread scribe as shown in figure 3.6.3.1

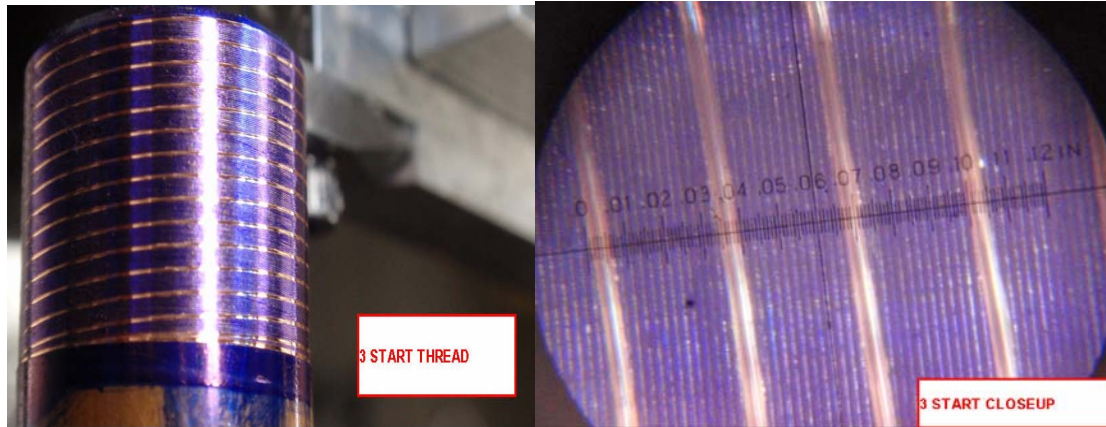


FIGURE 3.6.3.1

Do not exceed your max feedrate when doing a multiple start thread. ie; If it's a 4 start lead then the feedrate is 4x of just a single start thread. There no warning if the feedrate exceeds your defaults and the machine will move as fast as it can but lead and pitch will be incorrect. See Section 8 for additional info on multiple cut threads.

Here is the code used in this test:

```
( SCRIBE TEST FOR 3 START THREAD )
N10 M3 G18 G20 G40 G49 G61 G80 G90 G94
N20 G00 X0.625 ( OD OF TUBE )
N30 G00 Z0.3 ( 3X PITCH AND ASSUME THREAD STARTS AT Z=0)
N40 G76 X0.615 Z-0.625 Q0 P0.1 J0.001 L0 H0.001 I30 C0.1 T0
N50 G00 X0.625
N60 G00 Z0.03333 ( Z OFFSET )
N70 G76 X0.615 Z-0.625 Q0 P0.1 J0.001 L0 H0.001 I30 C0.1 T0
N80 G00 X0.625
N80 G00 Z0.06666 ( Z OFFSET )
N90 G76 X0.615 Z-0.625 Q0 P0.1 J0.001 L0 H0.001 I30 C0.1 T0
N100 M30
```

WW:

Thing about this.....for a 2 start thread you basically have double the error on the external thread and if you include cutting the nut then it could be fours times the error, and then some! Don't know what standards cover multiple cut threads, and frankly plug or ring gauges are not even shown in vendors literature. Now if the intent of multiple threading is for optic assemblies "the nut screws on " attitude amounts to worthless threading!

3.6.4 PICKING UP A THREAD SCRIBE TEST

This test will show how well the cutter was aligned to the piece using the method to pick up a thread described in Section 9. The same Gcode program used in scribing the lines should be used trying to pick a thread. You simply do a scribe test, align the cutter, then re-run the code to see if there is any difference in the scribing between the two as shown in the figures below. In this case you are aligning to a single scribed line per figure 3.6.4.1. If it was threaded piece the alignment point could be the root of a single thread, front or rear flank, it all depends on what you trying to do.

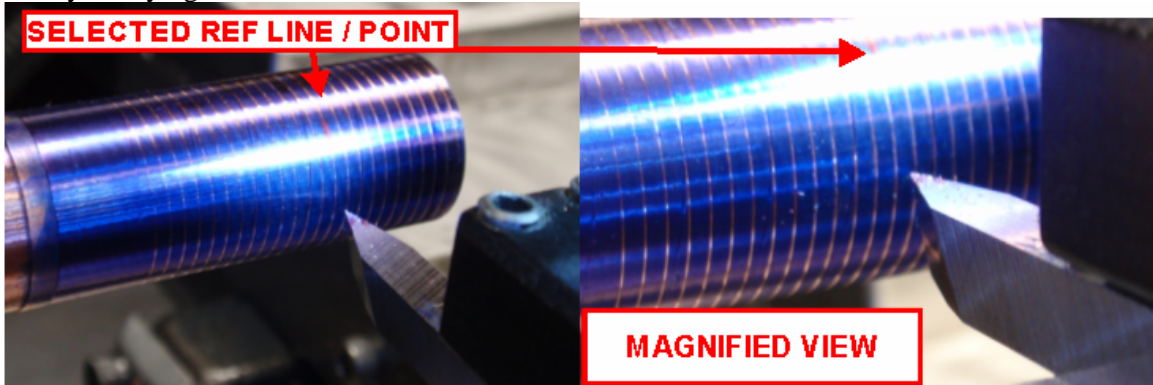


FIGURE 3.6.4.1

The figure below shows that “the over all procedure and cutting “ amounted to an error” in lead of 0.004”. Ideally there should be no error.

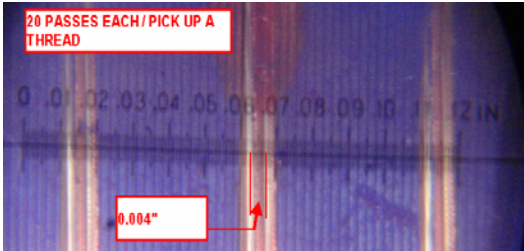


FIGURE 3.6.4.2

WW:

There will always be an error due to backlash, measurement, point selection along the thread but the most significant error is likely due to tool alignment. It will have an effect on the tolerance of the thread since the pitch has changed even though the lead is correct. The test doesn’t “isolate” the reason for the error but gives you a flavor of what you could expect based on user experience and your lathe system.

3.7 TESTING – CS / AL RESULTS

This is a summary of eight threads 4 in Al and 4 in CS.

All were set up as shown in Figure #1 and all were cut using the same Gcode.

1/4-20-UNC / 402 RPM / 44 PASSES (.004 first pass - .002 remainder – except #43 was .0004 and #44 was .0002)

The lead error on the tests were excellent.

Figure 1 shows the set up. 1/4 -20 was selected since visually you can see a bad pass and what is going on and not trying so much to measure, etc. When you do this thread and the setup is as shown, the 100# to 150# threading force will deflect the piece some 0.010 to 0.020". If the index timing / accel, etc is off some you may catch the end. (If it's consistent the cutting is will be very smooth.) That could make for a ruined thread but it also gives you a good indication on how well the planning will take care of practical situations. The start of the threading is short ie; normally in a scribe test I would allow 3 to 5x pitch for acceleration.

Just for you techie guys. Since the same Gcode is being used, the rpm will slow down at a different pass# / point since the difference in the modulus is offset by the difference in machinability, etc, etc . So calculate if wish, your wasting your time. Enough said!

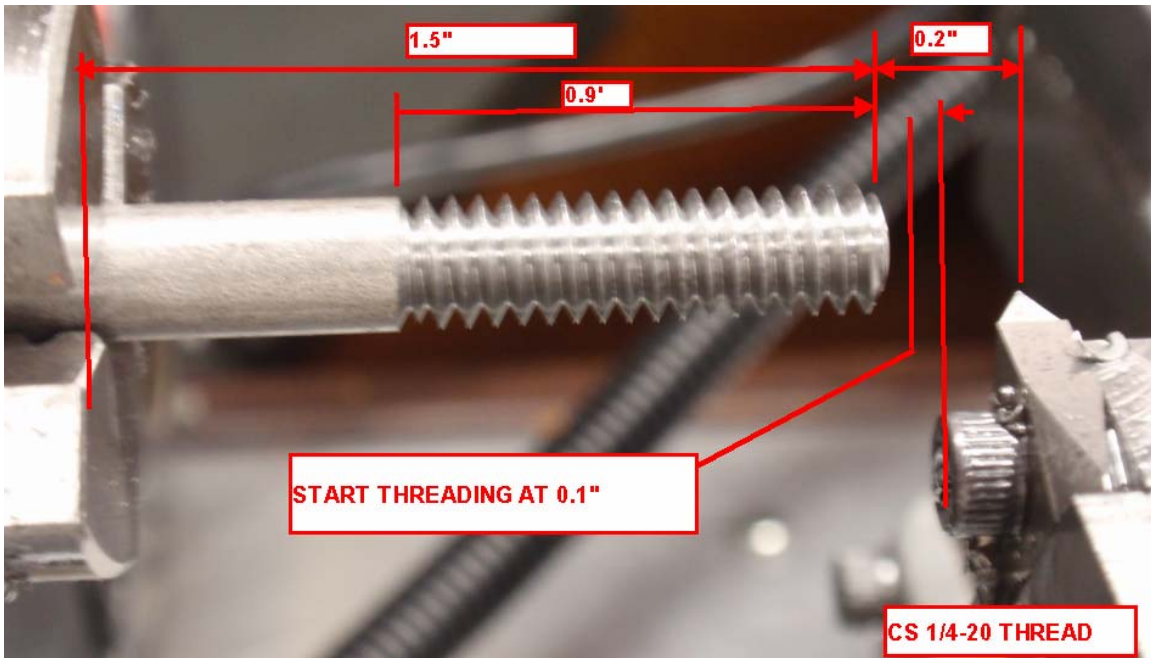


FIGURE 1

You can see in Figure 2 that both the Al and CS threaded well.

But there are basic differences. When measured the lead error was approx within .001"/inch for both. Now the AL thread is a class 2B and the CS is a 1A. Not because of lead error, but because of change in the pitch diameter, OD is in spec. Yes the gage goes on both of them but one is loose and the other (Al) is a nice fit. Actually the CS tapers due to deflection. So on the practical end of things this just reflects setup and has nothing much to do with the software side of things.

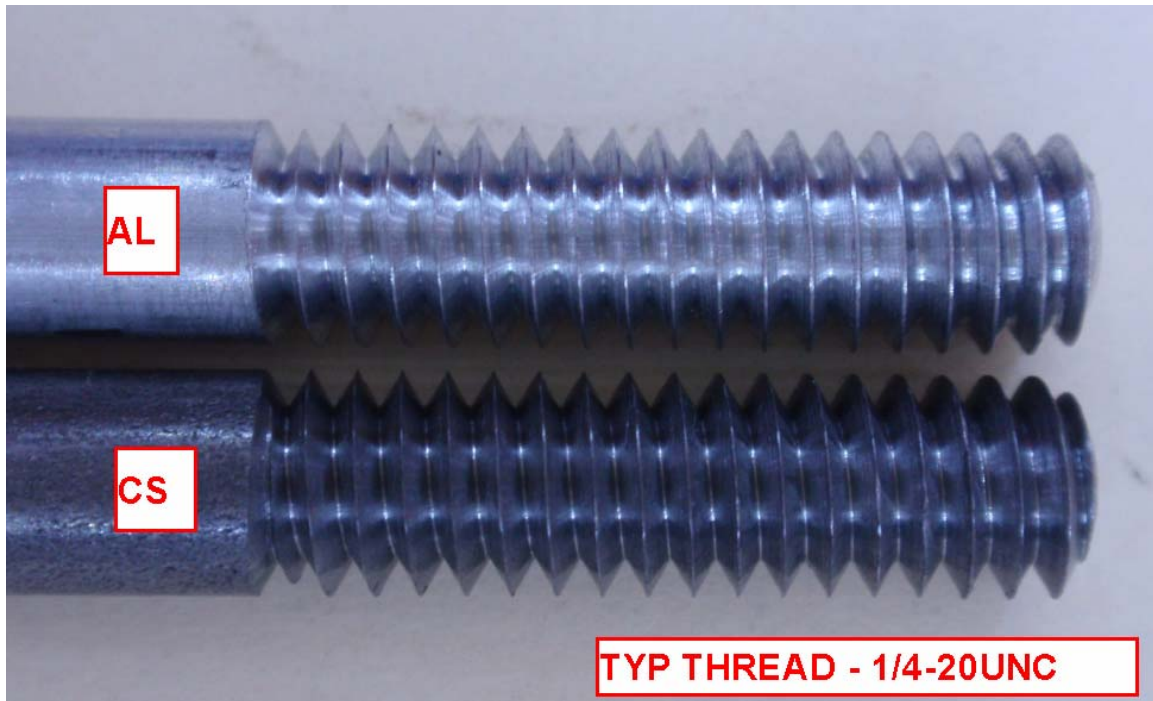


FIGURE 2

Figure 3 shows the profile of the CS more towards the anchored end of the stock.

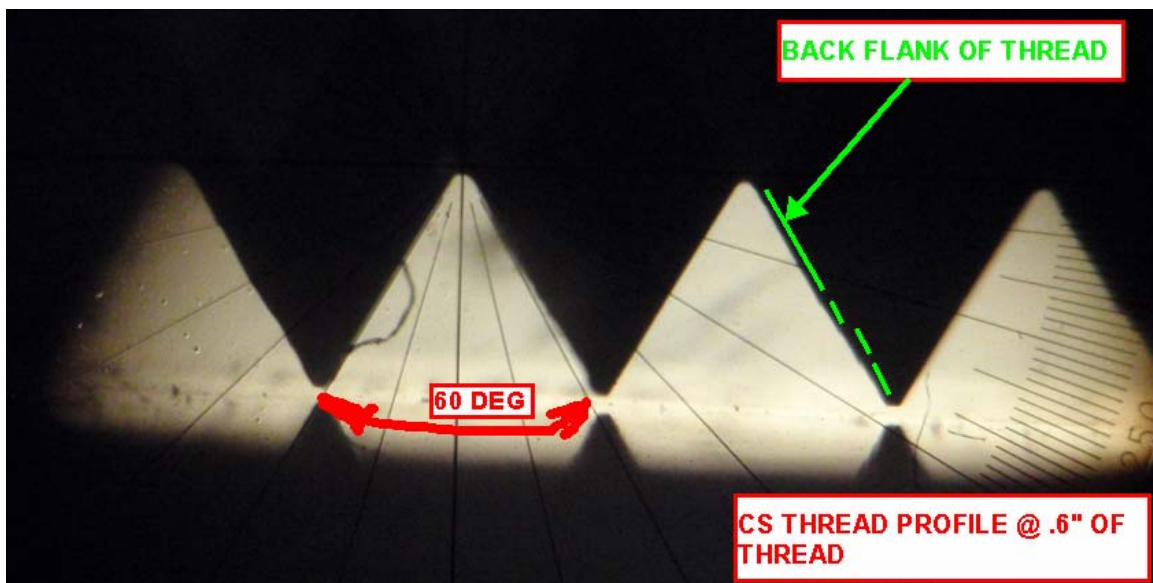


FIGURE 3

Figure 4 looks down on the thread showing the result of deflection. It is not chatter. The next picture provides explanation.

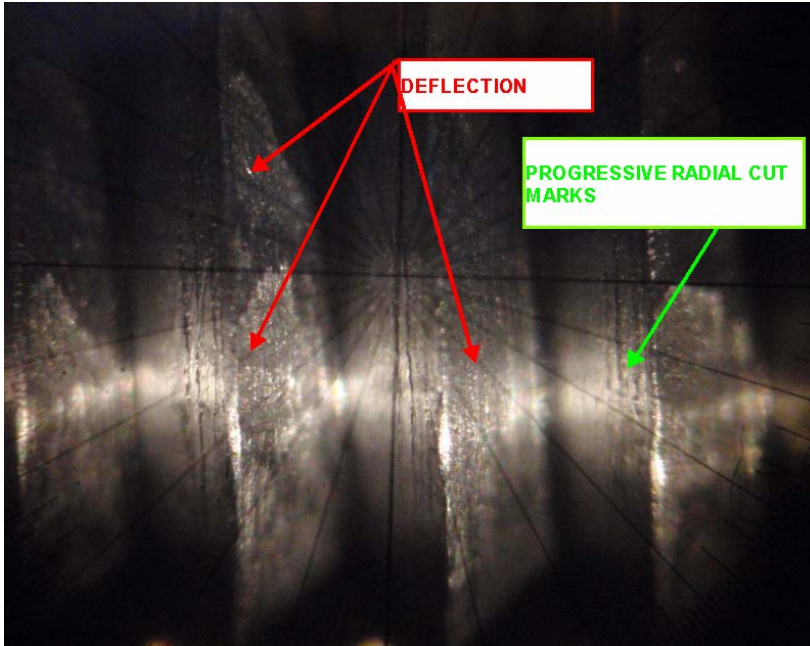


FIGURE 4

Figure 5 shows the finished CS thread. At pass #33 the deflection was rather great, the piece deflects, the spindle slows down some, the cutter cuts more deeply at the beginning of the thread, the spring reduces as you approach the anchored end and the cutting / shearing action of the tool progressively changes until the piece now longer is deflecting and normal cutting is restored. Now in the past, chances are, that the compensation / threading would have just trashed the thread in the next runs. Visually, passes 35 to 38 cleaned the thread up with clean cutting, pass #40 & 41 were more or less spring passes, but also fixed any lead error. The damage was due to deflection, but most important is that the software did it's job, BTW all four times!

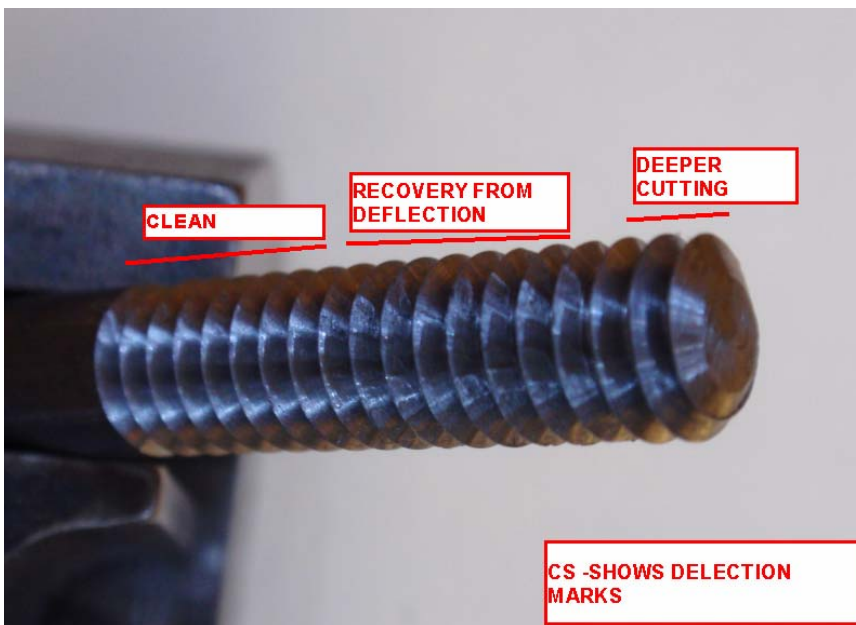


FIGURE 5

3.8 SPINDLE RPM

Spindle rpm can be read using a number of devices, such as, speed indicators, tachometers, oscilloscope input via the index pulse index, etc.

WW: The device most uses will have are not accurate enough, not calibrated, or can be misleading for a very accurate measurement. Look at the specs on the tach as it may be 0.5% +- 1 rpm. Time measurement with a low quality oscilloscope (even high quality digital storage used without consideration) just won't give you the info your looking. The device though will give a relative measurement of your rpm.

Those with or without any device should use the TurnDiags-Turn-Diag-1.00.1 plug in called Turn Diagnostics which is located in Mach3 Turn under the PlugIn Control tab. You may need to enable it and no configuration of the plugin is required. See figure 3.7.1. The plugin is currently loaded when a new or updated Mach installation is done.

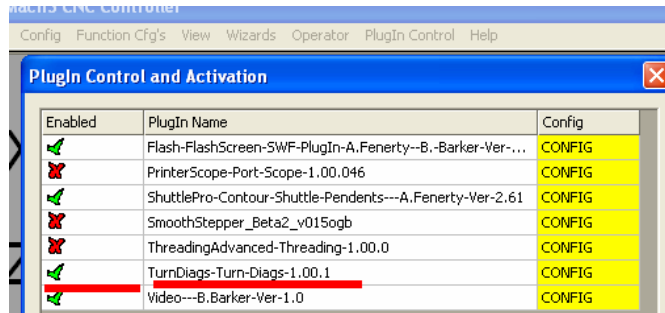


FIGURE 3.7.1

The plug-in will probably show your rotation speed real time as floating over a range and the higher of the range should be used in the wizard as an rpm input. Threading is based on what Mach sees as an input from the index. During threading the feedrate is adjusted and can be adjusted downward but not upward relative to the spindle rpm. Some testing using a specialty time based device (using the index pulse from the sensor) showed the plug in to be very accurate. BTW, the actual pulse signal can trigger differently in time even when conditioned. What is important is the “lathe system”. Odds are the average user will not have the required equipment nor expertise to analyze the index signal. Use the diagnostics information as shown in Figure 3.7.2 and don't get hung up on only one piece of the lathe system.

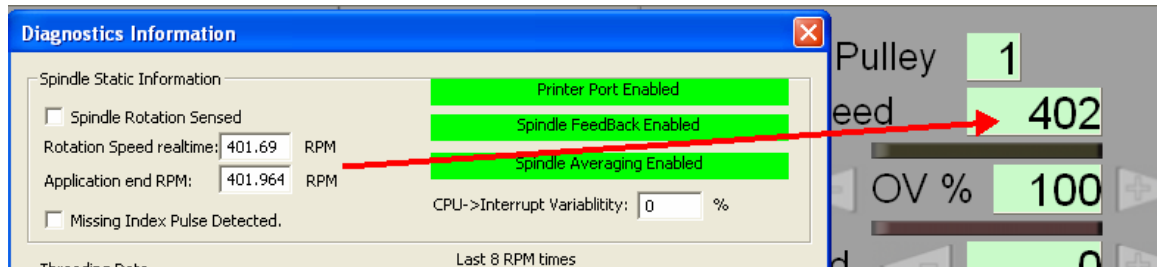


FIGURE 3.7.2

3.9 MOTOR - GENERAL SLOWDOWN / POWER / EFFECT ON TREAD'G WW:

The rpm stability and power delivered to the spindle will affect how Mach plans the Z motion for threading. Motor rpm does change and in threading it can have a dramatic effect during the threading cycle. The horsepower required for making a cut can be calculated, and actual cutting tests by the Society of Manufacturing Engineers have provided practical ways of calculating the power. General formulas for horsepower are helpful for comparisons, ie; stepper hp delivered verses spindle hp, but, calculations are not “exact” / subjective, and frankly is beyond the average users understanding or application of them.

The stepper motor needs to have adequate power to move the Z axis during threading. Thus, during threading, the combination is a “chain” so to speak, and the application of the power is only as good as the weakest link in the chain. Changing gearing / belt ratio's for either motor along with driver setup (ie; voltage / amperage , etc) can improve the operating range of the “system”. The stepper must be able to accelerate / decelerate within the parameters the user defines in the Gcode. Experience gained by just cutting a range of threads, using different cut depths, rpm, cutting methods, etc is highly suggested.

4.0 THREAD BASICS

4.1 STANDARDS & DEFINITIONS

There are numerous forms of threads and various standards which govern thread tolerances. The following are a partial listing of designations:

- UN - unified screw thread constant pitch series
- UNC - unified screw thread coarse pitch series
- UNF - unified screw thread fine pitch series
- UNEF - unified screw thread extra fine pitch series
- UNJ - unified screw thread constant pitch series, with rounded root
- UNJC - unified screw thread coarse pitch series, rounded root
- UNS - unified screw thread Special diameter, pitch or length of engagement
- UNJF - unified screw thread fine pitch series, rounded root
- UNJEF - unified screw thread extra fine pitch series, rounded root
- M - Metric Screw Threads- M profile with basic ISO 68 profile
- MJ - Metric Screw Threads- MJ profile with rounded root
- MJS - Metric Screw Threads- MJ profile profile special series

There are many more thread types such as National Pipe. For detailed information you can obtain the screw thread specifications from organizations such as ASME.

Many sources of information are available to enlighten oneself to any degree they wish. I am going to just limit the info presented here to the 60 deg V - thread at a high level.

The following are some definitions from different sources:

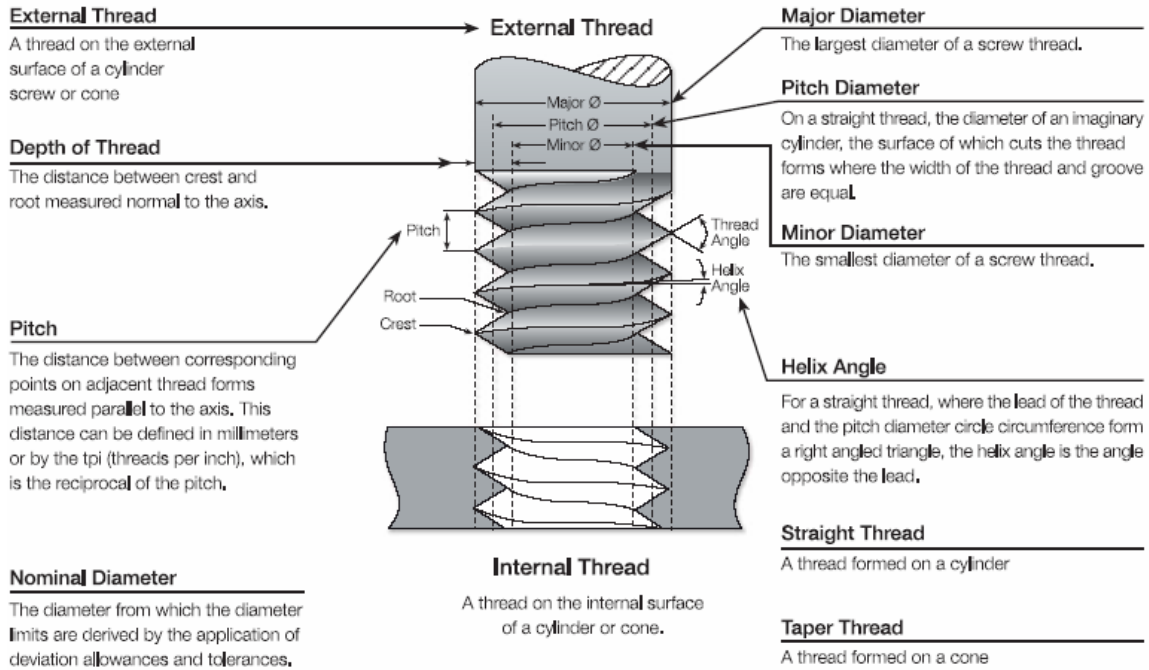
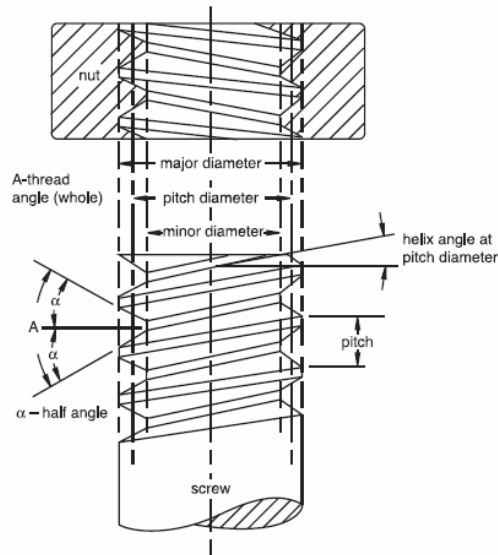


FIGURE 4.1.0



It is generally recognized that, in mass production, it is impossible to reproduce in exact detail the theoretically perfect product as laid out on the drawing board. The allowed slight variation between the theoretically perfect product and each unit of the actual product is called the TOLERANCE.

Allowance

An intentional difference in correlated dimensions of mating parts. It is the minimum clearance or maximum interference between such parts.

Angle of Thread

The angle included between the flanks of the thread measured in an axial plane.

Half Angle of Thread

The angle included between a flank of the thread and the normal (90°) to the axis, measured in an axial plane.

Lead of Thread

The distance a screw thread advances axially in one turn. On a single-thread screw, the lead and pitch are identical. On a double thread, the lead is 2X pitch; on a triple thread, the lead is 3X pitch, etc.

Major Diameter

The largest diameter of a straight screw thread.

Minor Diameter

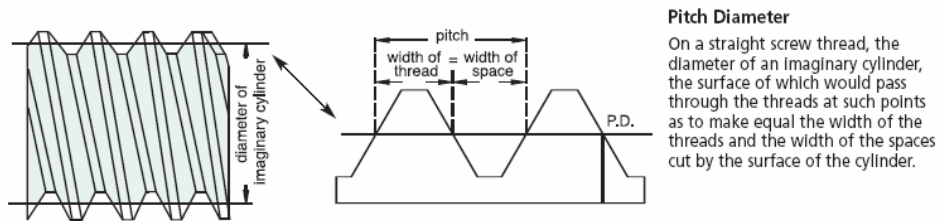
The smallest diameter of a straight screw thread.

Pitch

The distance from a point on a screw thread to a corresponding point on the next thread measured parallel to the axis.

The pitch in inches = $\frac{1}{\text{Number of threads per inch}}$

FIGURE 4.1.1



Pitch Diameter

On a straight screw thread, the diameter of an imaginary cylinder, the surface of which would pass through the threads at such points as to make equal the width of the threads and the width of the spaces cut by the surface of the cylinder.

FIGURE 4.1.2

There is another definition which is worth defining as shown and defined in the following picture, namely, Basic Pitch Diameter. Do not confuse it with pitch Diameter of a thread. Basic P.D defines a basic line about which thread tolerances are based. This will be discussed later on.

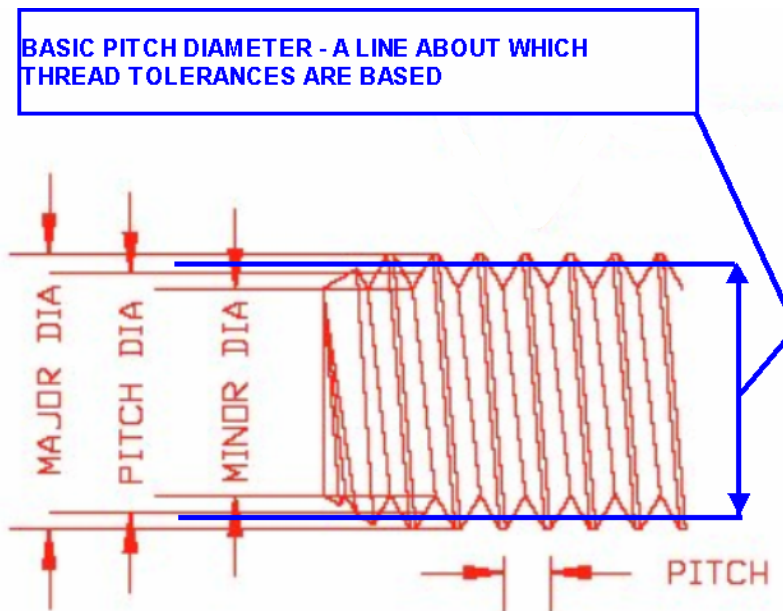


FIGURE 4.1.3

4.2 DEPTH OF CUT BASIS

If you are using a sharp V tool, then $DEPTH = .86603 \times Pitch$.

Check out a Machinist Handbook. Take a close look at the different forms. Peak to Peak of the sharp thread crests are defined as $H = 0.86603 \times Pitch$ for Unified Screw Threads. If you take away some of the sharp crests (top and bottom) then the remaining depth of the thread is defined as $0.61343 \times pitch$ or $17H / 24$ where H is as stated.

You also account for actual outside diameter and any tip radius of the sharp v tool if setting to an outside turned diameter. There are tolerances on the major, minor and pitch diameter. If not the, the pitch diameter may not be in tolerance post cutting. If the threading tool is not ground to the correct angle, then thread form will not be correct.

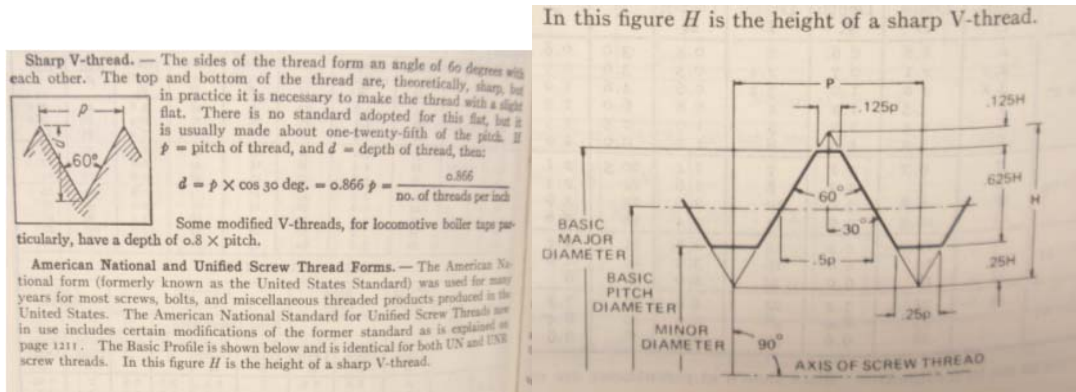


FIGURE 4.2.1

4.3 MEASURING THE THREAD

There are a number of methods to measure or test inside or outside threads depending on the tools used. Screw thread micrometers, caliper attachments, plug and ring gauges (Figure 4.3.1), microscopes, comparators, three wire method (Figure 4.3.2), thread triangles for use with a micrometer and even specialty tools. Some standards required a calibrated accurate comparator, and lets not leave out the specialty devices. Tools such as pocket comparators and the bladed screw thread tools are for “visual” checking only and are not for measuring. The three wire method is accurate but a PITA, plug and ring gauges are nice but you need an assortment and are expensive.



FIGURE 4.3.1

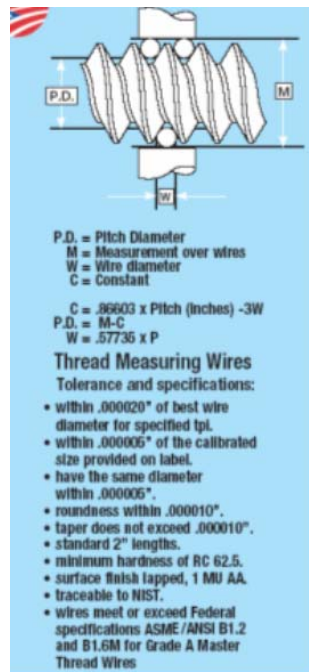


FIGURE 4.3.2

Use of best size wires
 In measuring screw threads it is desirable to use wires which touch the thread at or near the pitch diameter for the reason that with such "Best Size" wires the measurement of pitch diameter is least effected by an error in the included angle of thread.

	Max.	Best	Min.
60°	1.01062p	.57735p	.505182p
55°	.82573p	.563692p	.50568p
53°-8°	.97828p	.55902p	.54076p
47½°	.72889p	.54625p	.49852p
29° Acme	.650013p	.51645p	.487263p

p=Pitch n=TPH P= 1/n

Notes: Maximum and minimum wires should be used which are safety within the limiting dimensions given by the above formulas.

There are reasons for measuring a thread. Measuring a thread with go-no go plug and ring gauges along with diameter checks just say that a something can be assembled based on meeting some basic criteria for form. The real intent is about satisfying the intent that the nut doesn't strip and the bolt / screw breaks first. Design calculations may be based on factors which require quality threads and materials. The threading operation needs to meet that design specification. Example; One job required very high mill spec'd bolt's and nuts to satisfy a condition. The 120 - 3/4" bolts alone cost \$18000, so we figured we could save money by having the bolting made and heat treated. Five machine shops provided samples and every nut stripped due to heat treatment or thread form. So there is more to threading than just the "nut goes on the thread".

WW:

Lead is important and may be governed by the software, but then the lead can change depending on the lathe system due to cutting a taper or not holding a diameter, so that even if the programming is true, then the class of fit can suffer. So in threading, a user may experience "the dog chasing his tail", if ALL is not appropriate when threading to meet a standard.

4.4 TOLERANCE

There are many classes of fit along with combinations that will assemble or not assemble. Figure 4.4.1 shows the relationship for taps.

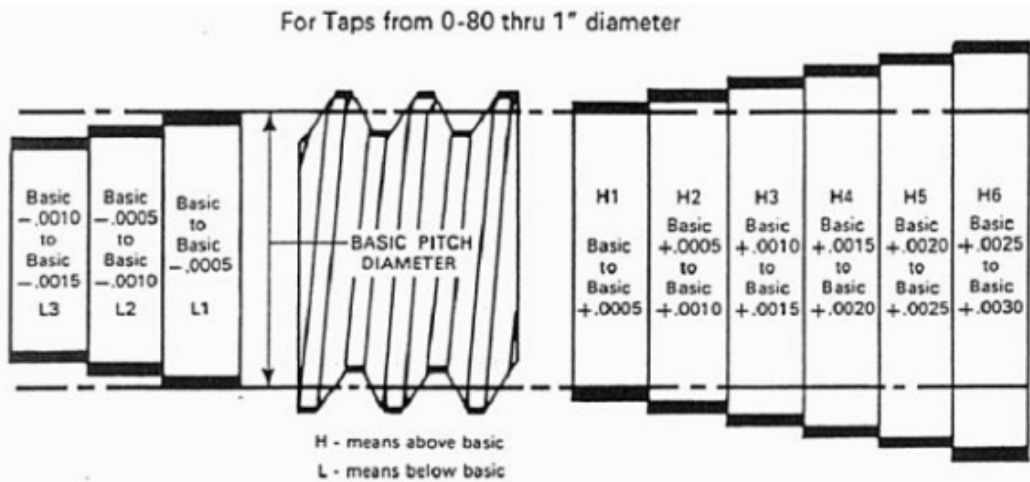


FIGURE 4.4.1

Figure 4.4.2 provides overall tolerances of internal and external threads for one thread form about the Basic Pitch inclusive of taps. External threads have tolerances below the basic P.D. while internal is above it. The info is from the Greenfield Screw Thread Manual.

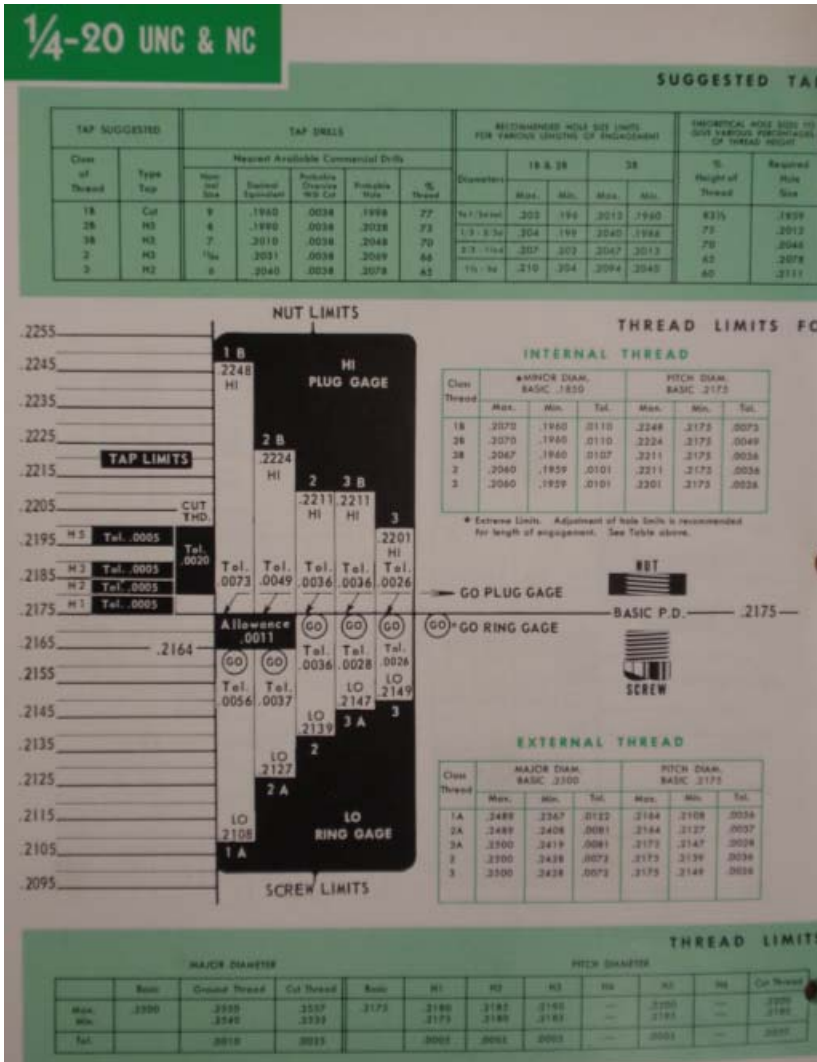


FIGURE 4.4.2

So if the user wanted to make an inexpensive ring gauge he could use a ground H1 or H2 limit tap to thread a drilled and reamed hole into some stock. Here is a homemade gauge 3/4" long that was checked and satisfies a 3B plug gauge check for a few cents in cost. It is a lot better than using a nut since the home made gauge is of a defined tolerance only starts and stops as compared to using a nut of unknown class as shown in figure 4.4.4.



FIGURE 4.4.3



FIGURE 4.4.4

This table shows a lead tolerance for a length in order to meet a 2A or 3A thread and was the basis for which Mach software needed to achieve or exceed. (See Section 3.6.1 for test results)

2	THREAD SERIES	SIZE	TPI	PITCH	(NOTE#2) LENGTH	(NOTE#1) ROUNDED 2A+- ALLOW	ACTUAL 2A+- ALLOW	(NOTE#1) ROUNDED 3A+- ALLOW	3A+-ALLOW
3	UNC	#1	64	0.01563	0.1	0.001	0.00058	0.000	0.00043
4	UNC	#2	56	0.01786	0.1	0.001	0.00061	0.000	0.00046
5	UNC	#3	48	0.02083	0.2	0.001	0.00066	0.000	0.00049
6	UNC	#4	40	0.025	0.2	0.001	0.00072	0.001	0.00055
7	UNC	#5	40	0.025	0.2	0.001	0.00075	0.001	0.00055
8	UNC	#6	32	0.03125	0.2	0.001	0.00081	0.001	0.00061
9	UNC	#8	32	0.03125	0.2	0.001	0.00084	0.001	0.00064
10	UNC	#10	24	0.04167	0.2	0.001	0.00095	0.001	0.00072
11	UNC	#12	24	0.04167	0.2	0.001	0.00098	0.001	0.00075
12	UNC	1/4	20	0.05	0.3	0.001	0.00107	0.001	0.00081
13	UNC	5/16	18	0.05556	0.3	0.001	0.00115	0.001	0.00087
14	UNC	3/8	16	0.0625	0.4	0.001	0.00127	0.001	0.00095
15	UNC	7/16	14	0.07143	0.4	0.001	0.00136	0.001	0.00101
16	UNC	1/2	13	0.07692	0.4	0.001	0.00144	0.000	0.00111
17	UNC	9/16	12	0.08333	0.6	0.002	0.0015	0.001	0.00113
18	UNC	5/8	11	0.09091	0.8	0.002	0.00159	0.001	0.00118
19	UNC	3/4	10	0.1	0.8	0.002	0.0017	0.001	0.00127
20	UNC	7/8	9	0.11111	0.9	0.002	0.00182	0.001	0.00136
21	UNC	1"	8	0.125	0.9	0.002	0.00196	0.001	0.00147
22									
23	UNF	#0	80	0.0125	0.1	0.001	0.00052	0.000	0.00038
24	UNF	#1	72	0.01389	0.1	0.001	0.00055	0.000	0.0004
25	UNF	#2	64	0.01563	0.1	0.001	0.00058	0.000	0.00043
26	UNF	#3	56	0.01786	0.2	0.001	0.00064	0.000	0.00046
27	UNF	#4	48	0.02083	0.2	0.001	0.00069	0.001	0.00052
28	UNF	#5	44	0.02273	0.2	0.001	0.00072	0.001	0.00055
29	UNF	#6	40	0.025	0.2	0.001	0.00075	0.001	0.00058
30	UNF	#8	36	0.02778	0.2	0.001	0.00081	0.001	0.00061
31	UNF	#10	32	0.03125	0.2	0.001	0.00087	0.001	0.00066
32	UNF	#12	28	0.03571	0.2	0.001	0.00092	0.007	0.0069
33	UNF	1/4	28	0.03571	0.3	0.001	0.00095	0.001	0.00072
34	UNF	5/16	24	0.04167	0.3	0.001	0.00107	0.001	0.00078
35	UNF	3/8	24	0.04167	0.4	0.001	0.0011	0.001	0.00084
36	UNF	7/16	20	0.05	0.4	0.001	0.00121	0.001	0.00089
37	UNF	1/2	20	0.05	0.4	0.001	0.00124	0.001	0.00092
38	UNF	9/16	18	0.05556	0.6	0.001	0.0013	0.001	0.00098
39	UNF	5/8	18	0.05556	0.6	0.001	0.00136	0.001	0.00101
40	UNF	3/4	16	0.0625	0.6	0.001	0.00144	0.001	0.0011
41	UNF	7/8	14	0.07143	0.7	0.002	0.00156	0.001	0.00118
42	UNF	1"	12	0.83333	0.7	0.002	0.0017	0.000	0.000127
43	NOTE#1 - ACTUAL ALLOWABLE ERROR ROUNDED TO THREE DECIMAL PLACES FOR REFERENCE								
44	NOTE#2 - REFERENCE LENGTH FOR TESTING								

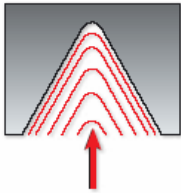
FIGURE 4.4.5

5.0 THREAD CUTTING

5.1 THREAD CUTTING FEED METHODS & FORMULAS

There are numerous methods to actually cut a thread. The Gcode is similar but different for each of the cutting methods. There are advantages to using the different methods. The figures below show the different methods along with comments for each. Alternate flank cutting is available in Mach Turn. The flank in feed provides excellent chip control since the chip flow is away from the tool.

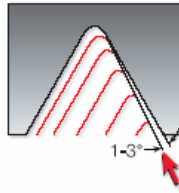
Radial Infeed



Radial infeed is the simplest and quickest method. The feed is perpendicular to the turning axis, and both flanks of the insert perform the cutting operation. Radial infeed is recommended in 3 cases:

- When the pitch is smaller than 16 tpi
- For material with short chips
- For work with hardened material

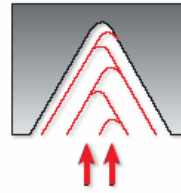
Flank Infeed (modified)



Flank infeed is recommended in the following cases:

- When the thread pitch is greater than 16 tpi, using the radial method, the effective cutting edge length is too large, resulting in chatter.
- For TRAPEZ and ACME. The radial method result in three cutting edges, making chip flow very difficult.

Alternate Flank Infeed



Use of the alternate flank method is recommended especially in large pitches and for materials with long chips. This method divides the load equally on both flanks, resulting in equal wear along the cutting edges. Alternate flank infeed requires more complicated programming, and is not available on all lathes.



FIGURE 5.1

Now there are variations on the theme of the above noted thread methods. Constant volume, where in addition to radial feeding the depths are varied to maintain that volume and the same applies to the other methods. Increasing the first pass depth can reduce the number of passes required to cut the thread and also provides a more constant load on the motor from start to finish.

WW:

If you look at various manufactures information you will find some differences in the use and name of the methods. They are just variations on the ones shown in Figure 5.1. You vary the method to suite the material, setup rigidity, and experiment some! On a puny lathe this is very important. Mach3 Turn threading specifics are in Sections 6,7, & 8.

5.2 SPINDLE MOTION / TURNING METHODS

Figure 5.2 relates spindle motion to different turning methods.

Thread Turning Methods

Thread	Inserts & Toolholder	Rotation	Feed Direction	Helix Method	Drawing No.
Right Hand External	EX RH	Anticlockwise	Towards chuck	Regular	1
	EX LH	Clockwise	From chuck	Reversed	2
Right Hand Internal	IN RH	Anticlockwise	Towards chuck	Regular	3
	IN LH	Clockwise	From chuck	Reversed	4
Left Hand External	EX LH	Clockwise	Towards chuck	Regular	5
	EX RH	Anticlockwise	From chuck	Reversed	6
Left Hand Internal	IN LH	Clockwise	Towards chuck	Regular	7
	IN RH	Anticlockwise	From chuck	Reversed	8

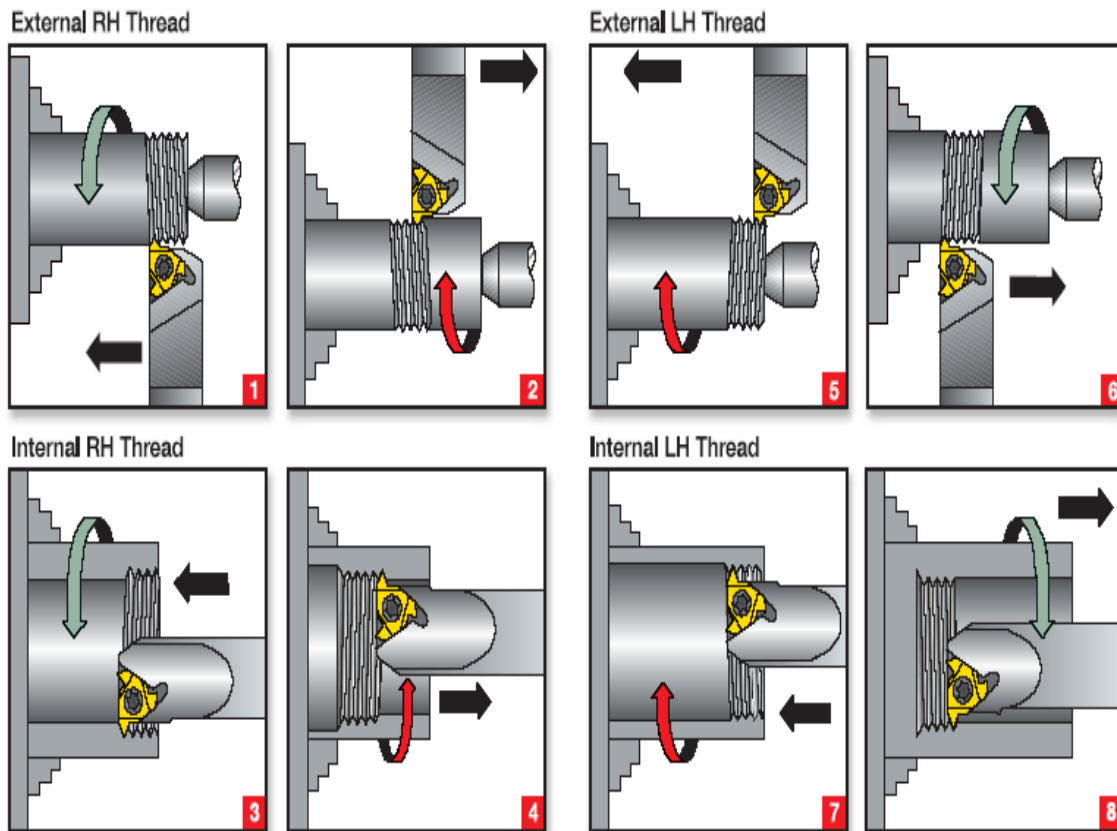


FIGURE 5.2

5.3 CHIP FORMATION

Figure 5.3 shows examples of the chip produced when using the different methods of threading.

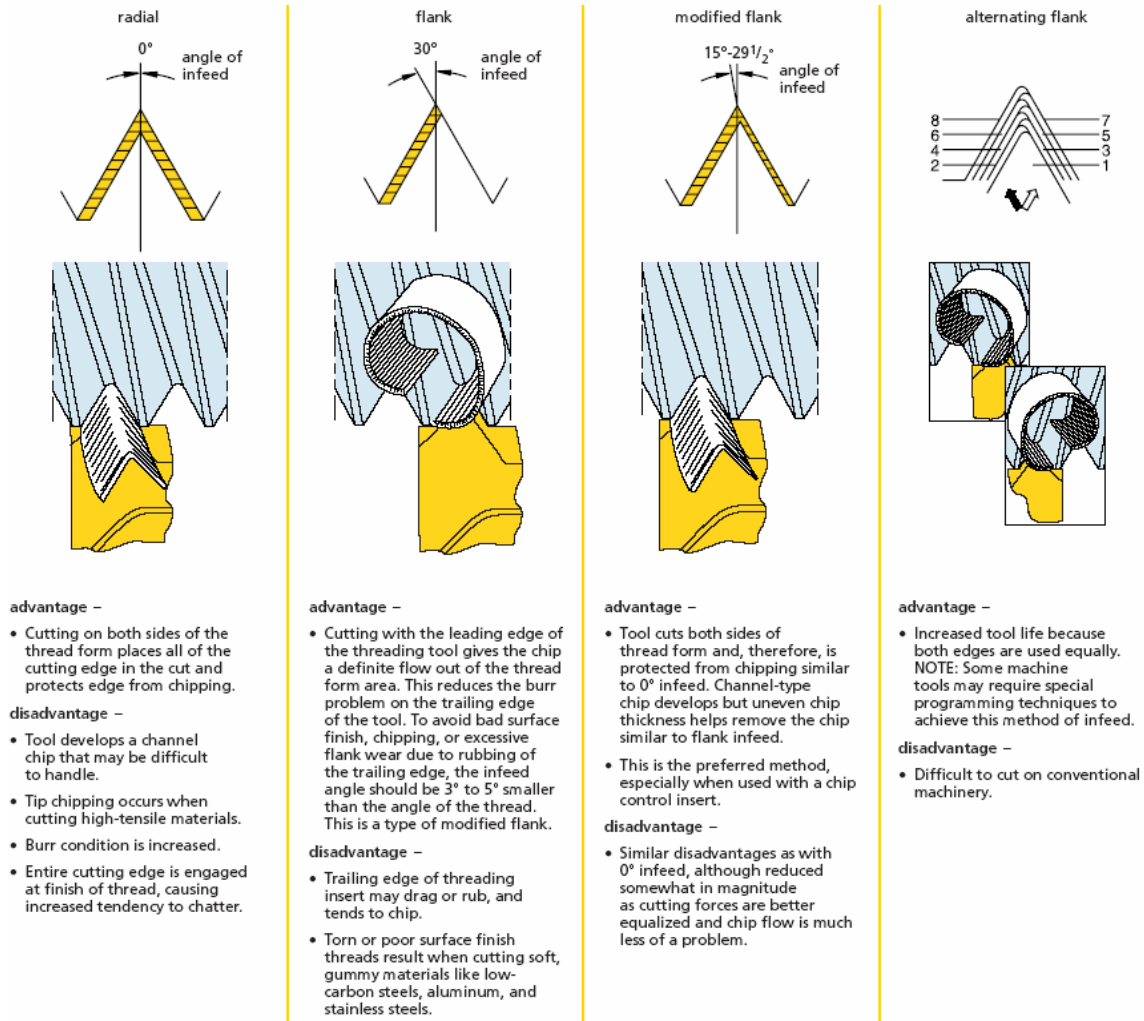


FIGURE 5.3

5.4 FORMULAS

Here are some formulas related to threading which you will find useful.

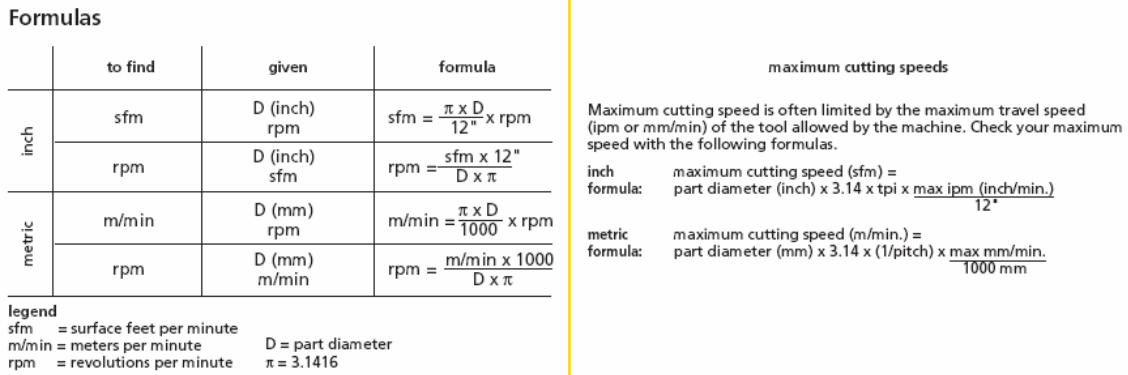


FIGURE 5.4

5.5 THREAD CUTTERS / TIP RADIUS

Cutters come in many shapes and forms. There are hand ground, machine ground, inserts, forming, preformed shapes and bars, different materials of construction, etc. I will remark that the thread profile will be a product of the cutter you have chosen to use.

The tip radius (more importantly the offset from a point) on a cutting tool is important to know when setting up a piece to be threaded. If the offset is not taken into account, the resulting pitch diameter could be out of spec for the desired thread. Tolerances and measuring threads are discussed later.

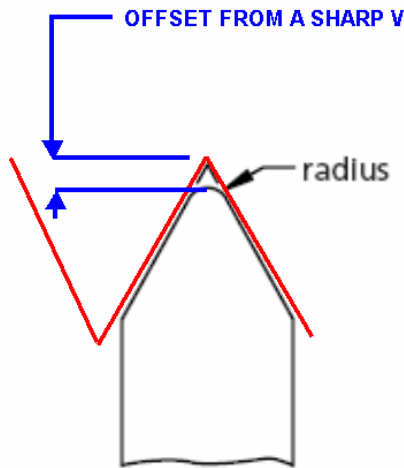


FIGURE 5.5

Tip radius, if just machining a piece to size will influence the surface finish, but for threading, there is much more. A extremely sharp root from a non-radius cutter is a stress intensifier in calculations, yet, for a small 0-80 a very sharp cutter is desirable as you don't have much depth to that thread. Use a sharp tip on hard material and it will break off, use a large tip on a small lathe and the you may have trouble cutting the thread. The cutter geometry plays a big role in chip formation and can greatly influence the actual loads generated.

5.6 WORK HOLDING

Rigidity should be taken into consideration when threading and the holding of material should be as strong a condition as possible. The cutting speed and feed should be modified to suite holding conditions. The attached provides a better understanding than the old rule of thumb of no greater than say 5 dia out from the chuck which in some cases is not a good guide. Note the support conditions and relative strength of the different holding conditions in Figure 5.6.1 below.

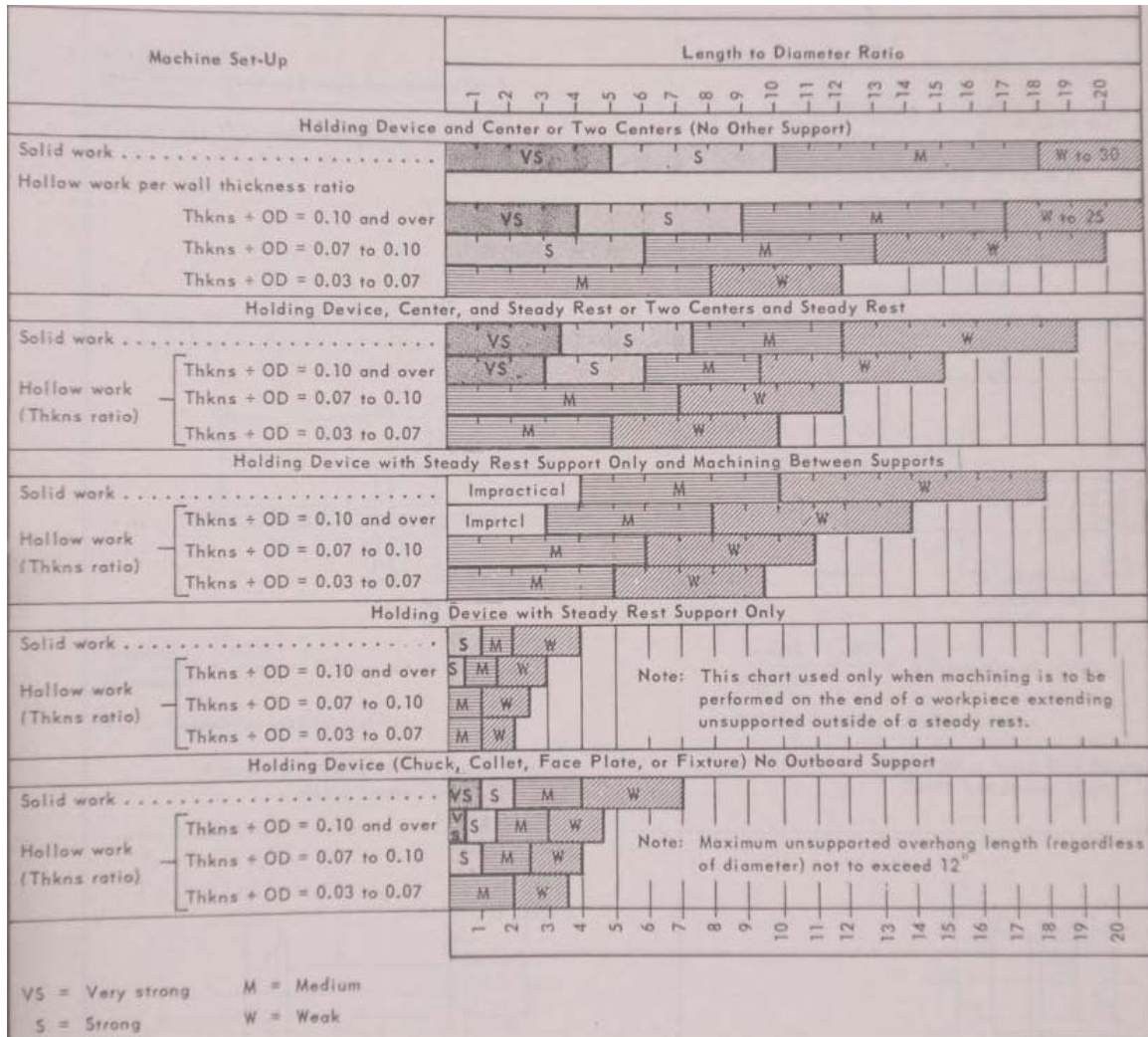


FIGURE 5.6.1

6.0 GCODE – MACH THREADING WIZARDS & CANNED CYCLES

6.1 WIZARDS

The GCode for threading can be provided by hand coding, wizards, or use of a canned cycle. Hand coding and CAM is not covered since most users will use a wizard or the G76 canned cycle. The two primary threading wizards are as follows:

Function Name	Description	Author
Quick Threads	Quick Threads: complex threadi	Scott Shafer
Simple Threading (Lathe)	Lathe Threading Rev 1.17	Brian Barker

FIGURE 6.1.0

The Simple Threading wizard is installed along with MACH. The Quick Threads wizard can be found and downloaded from the following location. It is suggested that you read through the thread. Information for installing the wizard is included in the download.

Machsupport Forum
 Mach Discussion
 VB and the development of wizards

Visual Basic and the development of wizards

Pages: 1 2 3 4 5 »

	Subject	Started by	Replies	Views
NORMAL POST	Milling Roughing and Finishing macro -- M79	melee	0	8
NORMAL POST	How To Tell When A G-Code Program Is Running?	HimyKabibble	7	193
NORMAL POST	No Menu Button?	sunangel	6	59
NORMAL POST	Axis Offset	Mojito	3	40
NORMAL POST	Running Mach3 via VB.NET	andrewk	2	63
NORMAL POST	Controlling a stepper motor from VB	andy692003	5	225
NORMAL POST	Cutting from Template	StefanV	2	100
VERY HOT	New Multiple and Complex thread Turning Wizard!!! < 1 2 ... 5 6 >	poppabear	53	5863

FIGURE 6.1.1

6.2 G76 THREADING CYCLE

Canned Cycle – Threading G76 (Using Mach3Turn 10-16 Rev 1.84-A2)

Program G76 X~ Z~ Q~ P~ H~ I~ R~ K~ L~ C~ B~ T~ J~ to cut a complete thread.

X -XEnd
 Z -ZEnd
 Q - Spring Passes. (optional)
 P -Pitch
 H -Depth of first pass
 I -Infeed angle
 R -XStart (optional)
 K -Z Start (optional)
 L -Chamfer (optional)
 C -X Clearance
 B -Depth Last Pass (optional)
 T -Taper (optional)
 J -Minimum depth per pass (optional)

The controlled point should give a suitable clearance for air cutting on the Z axis and have X equal to the crest diameter (radius). This is not necessary if R (the starting X value) and/or K (the starting Z value) are specified.

The length and depth of the thread are given by X and Z (in relation to the start values). If the "depth" is negative then the thread is taken as internal.

The C word gives the X clearance for the rapid return motion. It is always specified as positive and Mach3 handles the reversed direction for an internal thread

The P word gives the pitch of the thread (use 1/TPI for imperial threads)

The H word determines not only the first pass but, because the G76 ensures that subsequent passes cut the same volume of material per revolution determine the cuts for the entire thread and hence the number of passes.

The B word determines the depth of the last pass. If this is set too large then it can interact in an unexpected way with the constant volume calculations and the number of actual passes used.

The Q word gives the number of spring passes (i.e. cuts at the X end position).

The I word gives the infeed angle in degrees. This is Zero for radial infeed and typically 29 or 29.5 degrees for a unified or ISO metric thread or 27 degrees for 55 degree angle forms like BSW or BSF.

The values of words which are omitted are taken from the values set on the Turn Options tab of Config>Ports & Pins.

Note: This cycle is actually implemented by the macro M1076.M1S which can be inspected for implementation details. Note that one instance of this macro (usually in C:\Mach3\Macros) is used for all profiles.

NOTE:

1. The “L” retract or chamfer angle is in "angle of revolution"or a "rotational angle". and not an actually a chamfer. L90 would retract over 1/4 of a revolution, L360 would be 1 rev., 720, two revs. and so on. The following topic on the forum talks about the L setting in depth. <http://www.machsupport.com/forum/index.php/topic,5250.msg36097/topicseen.html#msg36097>

The above also applies to the chamfer in the wizards.

2. The mach G76 is not exactly like that described in Smid’s book. ie; Q for example is not used for a shift in degrees for multiple start threading.

6.2.1 Threading defaults

Most users will not need to be concerned with these as the CAM post processor or Mach3 Wizard will generally provide the system with all the required information to define a thread. There are default values (see Section 7.1 figure 7.1.3) in Mach configuration that will be used if in a given word a value is omitted in the G76 (threading) canned cycle.

6.3 METHOD CHOICE

Which method you use to generate the Gcode for threading is a user choice. Here are some general thoughts:

- Both wizards provide Gcode for threading (G76 or G32 code – see Mach configuration). The user should just try each to see which wizards he prefers for the threading that will done. Section 5.4 and 5.5 gives an overview of each wizard. I have only provided few comments in those sections for user awareness.

The Simple Threading wizard may be your first choice since it is easier to use, but, the Quick Thread Wizard provides additional features not found in the Simple Threading.

- The G32 is not a cycle and each step of the threading is defined in the code. If you change the diameter of the part, then you will need to go back and rerun the wizard with a new diameter input, same is true for any of the other parameters which affect the code.

The G32, in general, does not allow change to the code “on the fly”. You may have a program which generates passes based on volume, or alternate flank cutting, such that the code can be manipulated easily from a G33 (some other program output) to G32. So the G32 satisfies some needs that the G76 will not do (you have full control). The code will provide descriptive information for each and every pass.

- The G76 provides all the parameters for threading on a single line of code. It does not provide information about each pass. There is more room for error as a bad input may not be caught, but, that is true with any coding. G76 can do straight threads, tapered threads, multi-start threads, for external or internal threads. It is great way to do single point threading using the MDI line.

6.4 SIMPLE THREADING (LATHE) WIZARD

Use of this wizard is covered in the Using Mach3 Turn Manual (see Section 6.5 and is not repeated here). The associated screens are shown in Figures 6.4.1 & 6.4.2

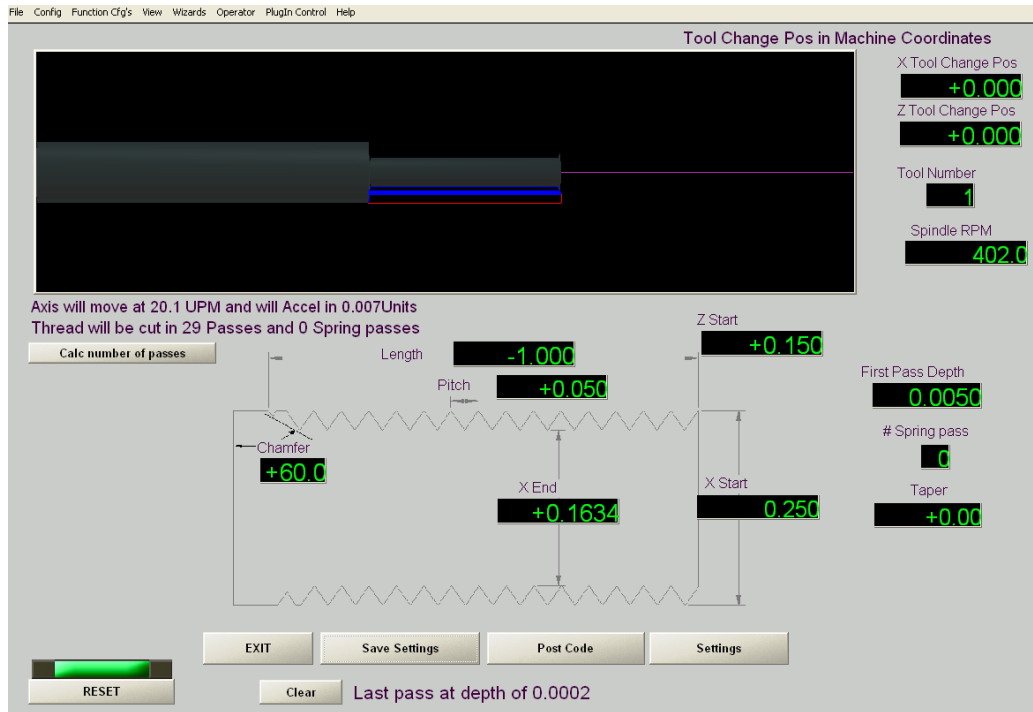


FIGURE 6.4.1

1. The “L” retract or chamfer angle is in "angle of revolution"or a "rotational angle". and not an actually a chamfer. L90 would retract over 1/4 of a revolution, L360 would be 1 rev, 720, two revs and so on.
2. If the length is – it will thread towards the headstock
If the length is + it will thread towards the tailstock
3. The number of digits for the pitch can be greater than what is displayed. ie; if 8 TPI then $1/8 = .125$, but, if 9 TPI $1/9 = .111111$ and 6 digits should be used for long or odd thread pitches for accuracy. The additional digits will be used in the Gcode generated even if not shown in the above screen.
4. See Section 6.6 for information relative to determining the first pass depth.
5. Use the Calc Number of Passes button to check if you motor setting is appropriate for the rpm and pitch entered. Remember that with a stepper as the velocity is increased the torque decreases / there is less power available and you want some “headroom” left for cutting the thread.



FIGURE 6.4.2

COMMENTS:

1. Infeed Angle

0 – provides radial cutting (This angle becomes important as the lathe size becomes smaller. ie; for deep cut threads you may want to go to 15 degrees and do “flank” cutting. The different cutting methods were discussed in Section #4.)

1 to 30 – will provide flank cutting

27 degrees for 55 degree angle forms like BSW or BSF

29 / 29.5- typically used for a unified or ISO metric thread and will modify the flank cutting providing a clearance on the back cutting edge to the back flank.

2. The min pass depth will determine the number of thread passes and also how the pass volume is calculated. It is suggested the user review and use the programs noted in See Section 6.6 to see the effects of different input values and read the WW in that section.

6.5 QUICK THREADS WIZARD

Information on this wizard is available via the “Help” tab in the program as shown below and is repeated here.



FIGURE 6.5.1

Quick Threads is a Wizard for doing multiple complex threading operations. It assumes a Spindle on your left hand as you face the lathe and the tool post on the front side. Start with the Set up page for the 1st operation and also each time you add another type of thread operation. If the new thread you want to cut, the settings change. Change them here for that next thread before going to the next thread. Set up your Coolant status for the current op. Set Right or left hand thread (yellow light off, is right hand), this assumes that CW (M3) will cut a right hand thread. Set what the maximum feed rate of your machine is in Inches per minute or mm/minute for the metric guys. *NOTE: The wizard assumes you are using Feed/Rev type of cutting.* You will have to have a index pulse coming back to Mach. Set your Obstacle Clearance. Use these moves to clear obstacles on complex objects. You can clear objects before the thread and after the thread, and you can move in one or two axis. You can set the amount per axis. There are Two Thread type wizards one Internal and one External. On each you can set Inch or mm setting and you must choose one. Set Your Tool holder number (or Tool number) and Tool offset. You can set your Work offsets from G54-59 with the quick set buttons, or just type in the number if higher than that. In the lower right hand you can just type in your TPI (for inch), and hit calculate and it will put the correct number in the "Pitch DRO". The common In-feed Angle button will also set your in-feed for typical stuff. Calculate Maximum RPM of spindle given the feed, put in Pitch, and hit Calculate Max spindle Left lower. If you want to set the Spindle to that max RPM just push the Set Spindle RPM to max button. Just fill in the requested information by looking at the Wizard Picture and Parameter. Posting multiple threads: If you just want one, thread routine, hit Post 1st thread, and then hit Done with Wizard. If you want multiple threads, then push the Post 1st thread then, after that, if you need to change your settings, go back to the settings page and make new settings (you can even change to a different measurement system if you want). Then go to the next thread type you want to cut (it could be the same type or not).

BUT!!! for the 2cd and subsequent threads press the "Add another thread op" on that page of the thread your currently wanting to add. You can keep doing this per thread op, (but remember both the Internal and External thread wizards pull off of the settings page, so make sure the settings or correct for the operation of the thread your doing before you push the "Add another thread op".

NOTE: There are "Global" LED's for the number of operations, it goes 1st Op, then 2cd, then 3rd, and lastly 4th, you can do more ops but the number of Op Done LEDs only go up to 4.

The Save Wizard does what it says.

The "Return to Mach" is to return to Mach with out posting anything. Once you hit "Done with Wizard" it will load the G-code to Mach and return you there.

You can View your Tool Path and G-code by pressing the "Tool Path" button and you can also Erase your operation to start over.

"Thread Calculator Table" button will pull up a "Free ware" which is a useful table with information for cutting all the various internal and external thread types.

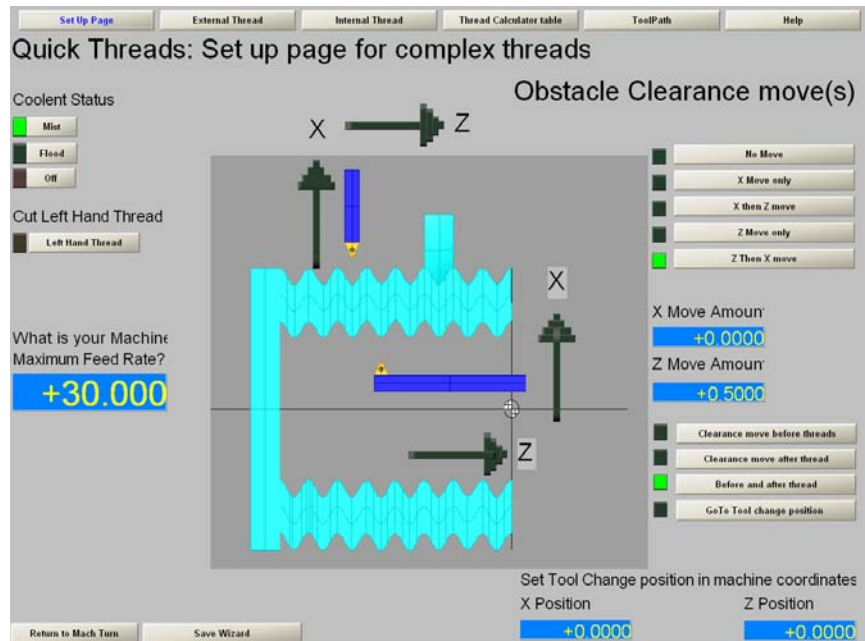


FIGURE 6.5.2

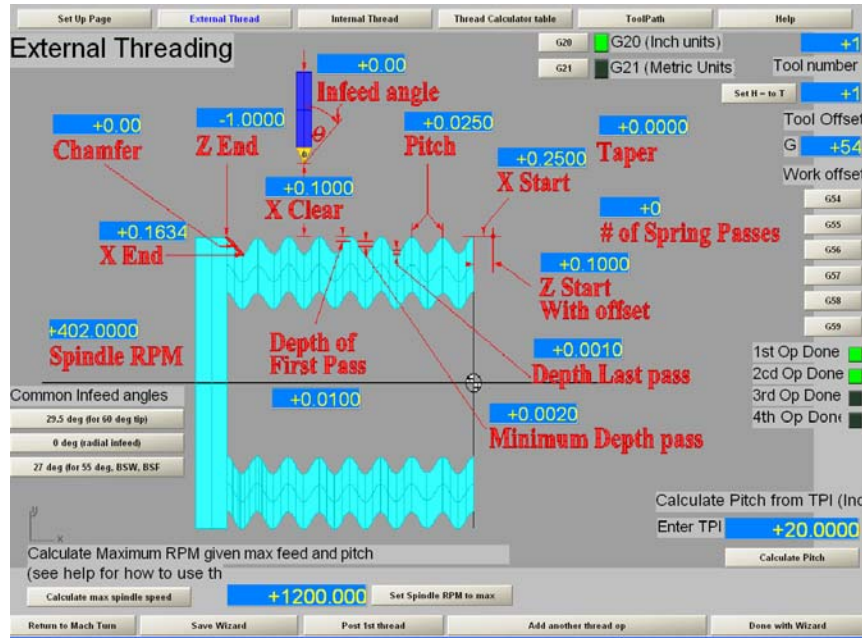


FIGURE 6.5.3

6.6 HELPFUL INFO / PROGRAMS

There are number of helpful programs related to threading.

The freeware program called THREADS (Figure 6.6.1), which is included with the Quick Threads program by Scott Shafer, provides information for various threads including metric and more.

NOTE: Use of the Minor Diameter “ref” will result in incorrect threads if using a SHARP “V” cutter. (see Section 4.2)

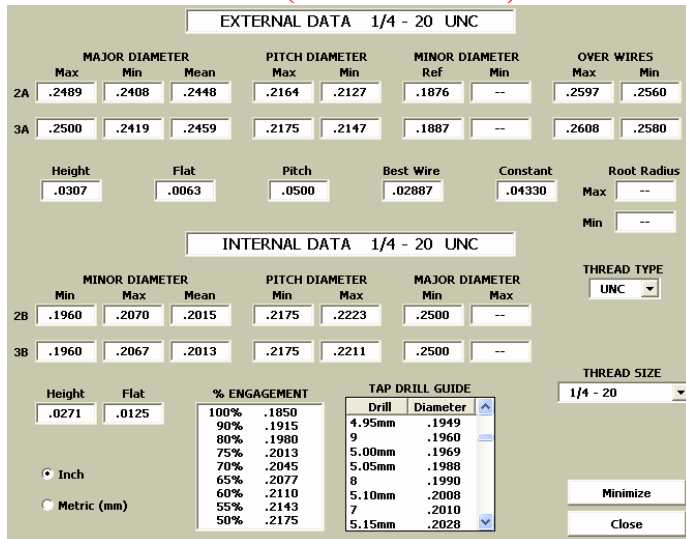


FIGURE 6.6.1

Another program is THREAD22 (Figure 6.6.2). The freeware threading program is a single point threading pass calculator. It provides number of passes based solely on a calculated cross-sectional area of material cut per pass.

Here is the link: <http://www.vectorcam.com/thread.html>

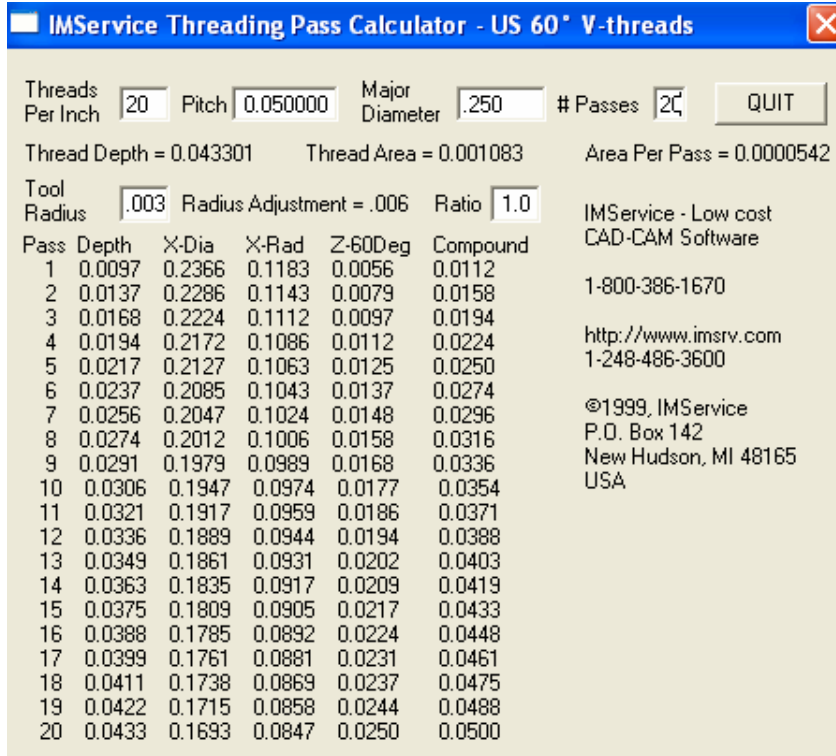
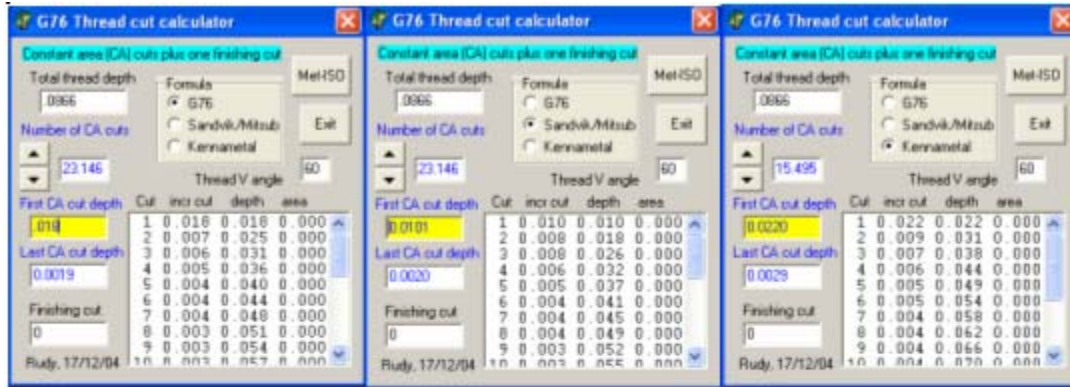


FIGURE 6.6.2

The G76_threads application (Figure 6.6.3) is included when Mach is installed and can be found in the MACH3 directory. There are three different formulas that can be used. The tabulation below shows the effect of a larger first pass for one side of the thread. How large the first pass should be is 20-30% of the total thread depth as a “general” guide line. Ideally you want a larger first pass cut depth , decreasing cut passes thereafter to maintain constant volume of cut (keeps rpm more consistent /same motor hp requirements) and a pass no less than 0.002”. Add a spring passes, but, depending on the material, it may want to be 0.002’ min in depth.

Note that thread cutting of different materials should be done at recommended SFM ranges along with an appropriate cutting fluid for the cutting tool that will be used The user needs to test to know what works with his lathe.



1	0.018	0.018
2	0.007	0.025
3	0.006	0.031
4	0.005	0.036
5	0.004	0.040
6	0.004	0.044
7	0.004	0.048
8	0.003	0.051
9	0.003	0.054
10	0.003	0.057
11	0.003	0.060
12	0.003	0.062
13	0.003	0.065
14	0.002	0.067
15	0.002	0.070
16	0.002	0.072
17	0.002	0.074
18	0.002	0.076
19	0.002	0.078
20	0.002	0.080
21	0.002	0.082
22	0.002	0.084
23	0.002	0.086

G76

1	0.010	0.010
2	0.008	0.018
3	0.008	0.026
4	0.006	0.032
5	0.005	0.037
6	0.004	0.041
7	0.004	0.045
8	0.004	0.049
9	0.003	0.052
10	0.003	0.055
11	0.003	0.058
12	0.003	0.061
13	0.003	0.064
14	0.003	0.066
15	0.003	0.069
16	0.002	0.071
17	0.002	0.074
18	0.002	0.076
19	0.002	0.078
20	0.002	0.080
21	0.002	0.082
22	0.002	0.084
23	0.002	0.086

SANDVIK

1	0.022	0.022
2	0.009	0.031
3	0.007	0.038
4	0.006	0.044
5	0.005	0.049
6	0.005	0.054
7	0.004	0.058
8	0.004	0.062
9	0.004	0.066
10	0.004	0.070
11	0.003	0.073
12	0.003	0.076
13	0.003	0.079
14	0.003	0.082
15	0.003	0.085

KENMETAL

FIGURE 6.6.3

User choice on which one to use. Read the WW!

WW:

For threading you should provide a larger first pass and the last pass should be smaller than all the others. Use of the programs will provide you a way of determining how deep that first pass should be such that a constant volume of material is removed. You do not want the last pass to be deeper than the others for obvious reasons. With the same volume, loading to the motor will be more consistent and thus Mach's threading planner will have a more representative rpm average to apply to the next pass. This relates to spindle slow down which defines Z axis feed rate adjustment. Having all the passes the same is not good, since in some cases, you can work harden the material. Some materials should not be threaded using .001 to .003" / small cut depths. Puny lathes will benefit from small cut depths. There are all kinds of information on thread cutting. You can calculate loading and hp requirements for threading, but, in the end, you need to just do some threading on your lathe to see what works for you and makes sense for what your trying to do with the lathe you have. You also have option on the type of cutting you can use. What someone else does "may not" even be relevant!

7.0 MACH3 TURN CONFIGURATION

Mach version 30.042.032 uses a completely new threading code versus all the other versions. Note that the version is only for a **single INDEX**.

There were two basic configurations for threading. Single index (single slotted disc / halls sensor, etc.) or timing (multi-slotted disc with index, or electronic equivalent / encoder). This version of threading will change the current ‘timing’ configuration such that a multi slotted disc with a wider index slot used for timing will no longer be available.

There are two current threading plug-ins. TurnDiags-Turn-Diag-1.00.1 called Turn Diagnostics in the PlugIn Control tab is currently loaded on a Mach installation. The second one, namely ThreadingAdvanced-Threading-1.00.0 was for testing and should not be used.

The new threading still addresses slow down of the spindle. How well it will hold the threading tolerance has not been tested yet, but ‘may be’ in the 75% range. You will not be able to completely stop the spindle and recover the threading cycle and prior tests for spindle slow down are no longer applicable.

The “future” Advanced Threading requires update per the new threading code and will allow for a slotted wheel with index or an encoder. When using an encoder additional parameters will need to be defined to MACH. If using an encoder, you’ll have to set the number of quads per rev and should work fine if one considers just how many lines can be seen per rev at top speed of threading used and limits their speed to that as a maximum. The Advanced Threading will not be available until after the updated version of Mach is released.

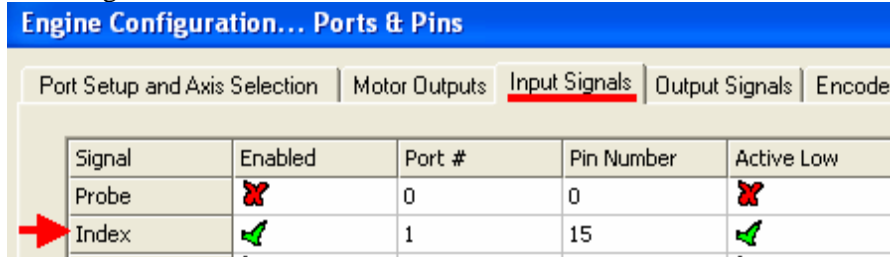
An encoder can be used for threading, and it IS on the books to allow an encoder geared output of the stream, but that may take longer to appear since rework of the code is first required to get it to a more manageable level of interconnection. As the printer port very slowly goes away, it becomes a question of what the new hardware guru's can put in their firmware with application support from ArtSoft's side.

I suspect Electronic gearing is now possible, but will require a defined development procedure similar to what we just went through with threading.

This writing current as of Mach version: 3.042.032

7.1 CONFIGURATION

The current version is for index only. The index must be enabled and port & pin assigned to do threading.



The timing function is disabled in version 3.042.032 and will not function.

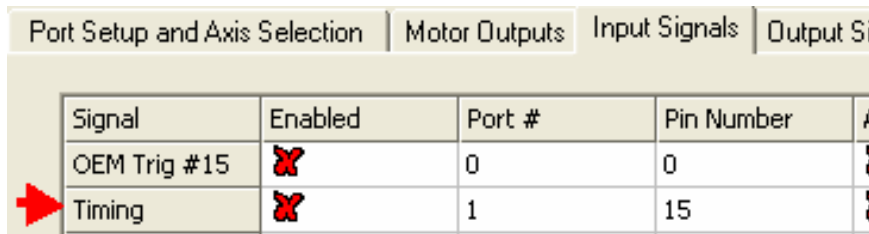


FIGURE 7.1.1

Spindle feedback should be checked. Spindle speed averaging will average the rpm over 8 readings and should be checked (IMHO).

COMMENT: If you do not use any motor control ie; PWM, VFD, just turning the spindle via belts and pulleys, uncheck the Relay Control. This is required in 3.042.032 in order to have the DRO display the rpm and have the timing work when threading. Later versions may change this, but for now it is required. Additionally, you must have the spindle turned on, so just type M3 in the MDI or click the Spindle button.

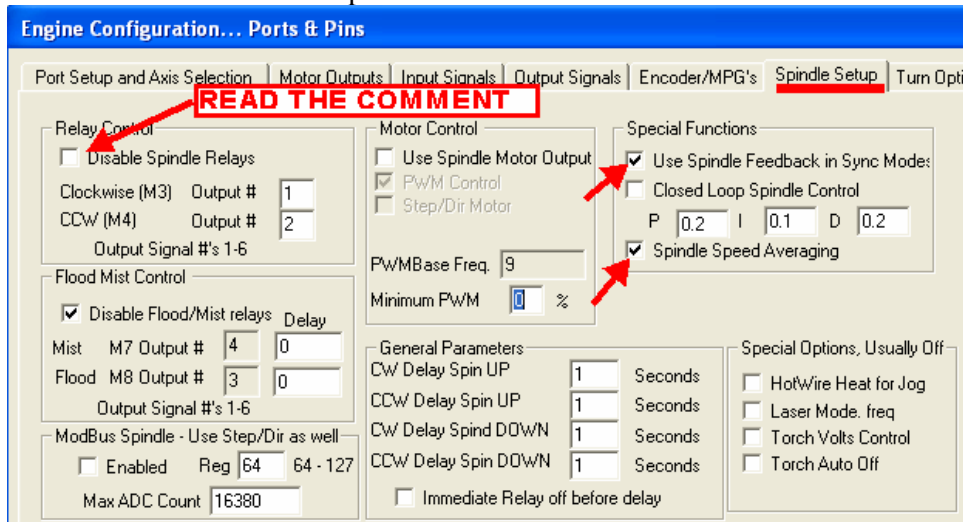


FIGURE 7.1.2

You should decide what mode (diameter or radius) you want to work in and be consistent when using Mach3 Turn or you will get confused sooner or later. Some turn wizards only work in radius but the threading wizards work in both. The Turn cycle defaults provide default values when not specified / missing in G76.

Note the following settings:

The cut type and infeed type settings will affect the Gcode posted by the wizards and the default is “0”. You change them and use any combination of them as follows:

Cut Type: defines type of threading method

- 0 - Flank cutting (default)
- 1 - Alternate Flank cutting
- 2 – Back Flank cutting

InFeed Type: defines how the passes are calculated

- 0 – Constant volume threading (default)
- 1 – Cut the thread in X number of passes (set in the wizard DRO 1022)

NOTE: THE ABOVE SETTINGS WILL DEFAULT BACK TO “0” WHEN YOU EXIT MACH3 TURN.

The remainder of the settings shown in the dotted square provide default values for G76 if it is missing.

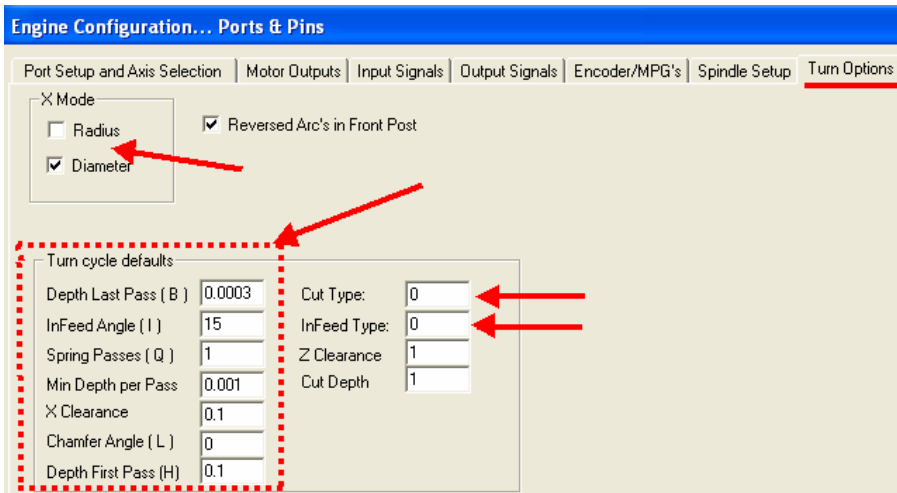


FIGURE 7.1.3

You should have use an initialization string to set Mach3 Turn “startup switches” to a state suitable for *how you work* as shown in figure 7.1.4. Do not just copy what is shown in figure 7.1.4, but review each and every code for *your* application. See Appendix “C” for an initialization macro. Debounce and index may need to be changed to get a full rpm readout in the DRO. Exact stop mode should be used when threading. If you use CV then the thread can become tapered towards the end of the cut due to cv blending corners.

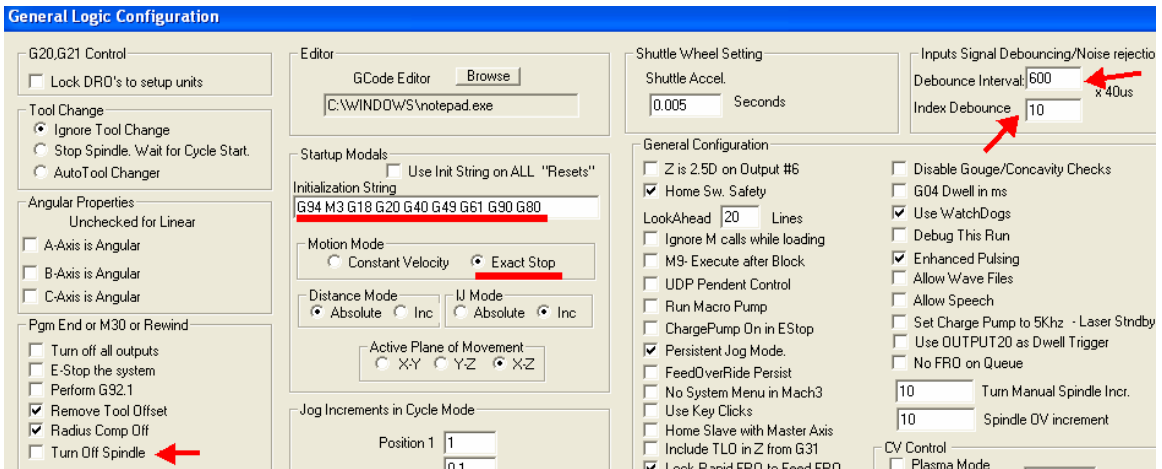


FIGURE 7.1.4

I don't use radius compensation and account for any tip radius via axis location after setting / checking the tool for threading. To each their own!

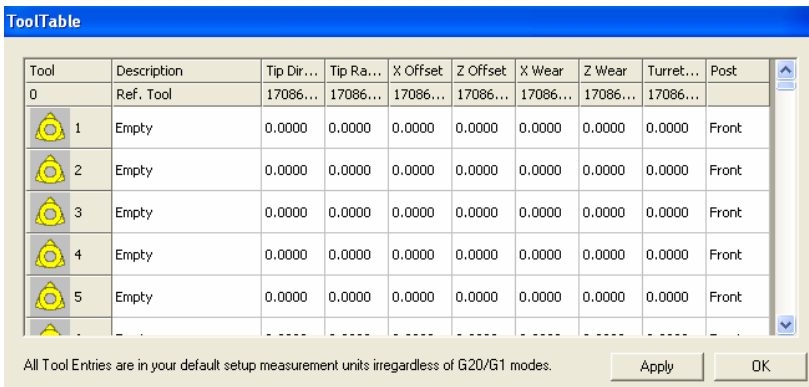


FIGURE 7.1.5

The turn diagnostics must be enabled by checking the box. No configuration is required.

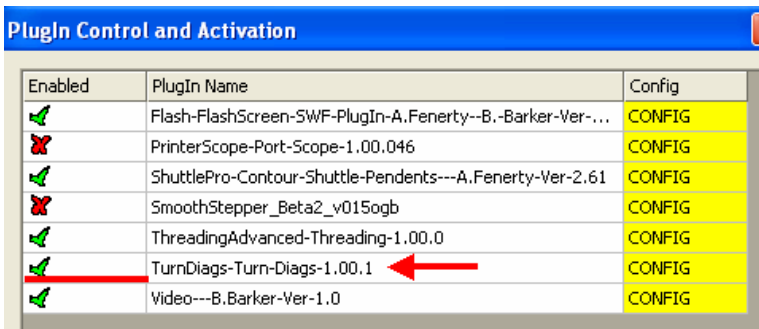


FIGURE 7.1.5

Turn diagnostics is available in the PlugIn Control. Your spindle must be turning. It will confirm that the index / spindle rotation is sensed, and show basic configuration status. Real time speed is shown and that is the rpm the user may want to use in the Wizard as it very accurate. CPU interrupt may no longer be applicable. as the new code is RPM based. Treading data will not be shown until a Gcode for threading is loaded and in effect. During threading it will show the variation in your rpm and the variation in rpm while threading. This is all shown in Figure 7.1.6.

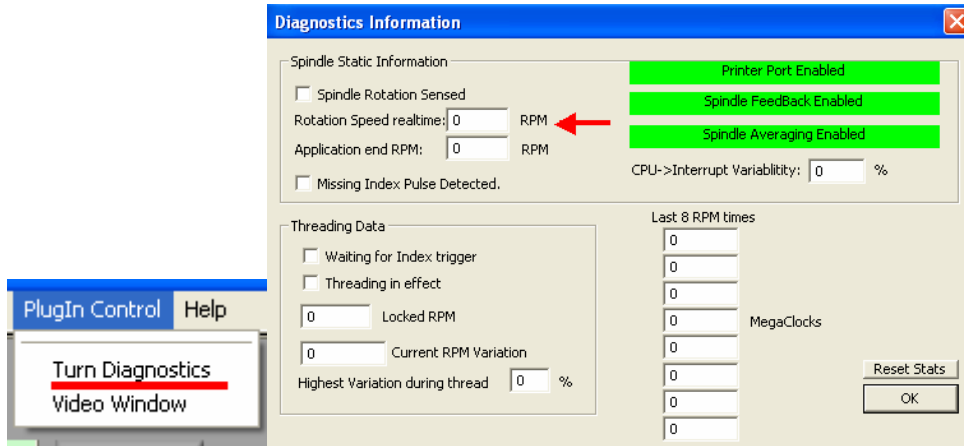


FIGURE 7.1.6

7.2 MOFIFYING M1076 MACRO

You can change how the Wizard will post Gcode and is a user preference. The code can be posted as a G76 with it's parameters or as the expanded G32 which will list all of the passes along with comments. To change the Macro for G32 posting of code, open the VB Script Editor as shown in Figure 7.2.1 and select the m1076.mis macro as shown in Figure 7.2.2.



FIGURE 7.2.1

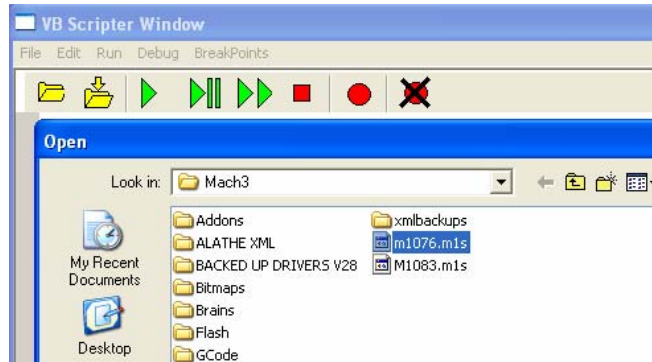


FIGURE 7.2.2

Scroll down to the line shown in Figure 7.2.3 and change the text to “true” and then save the file.

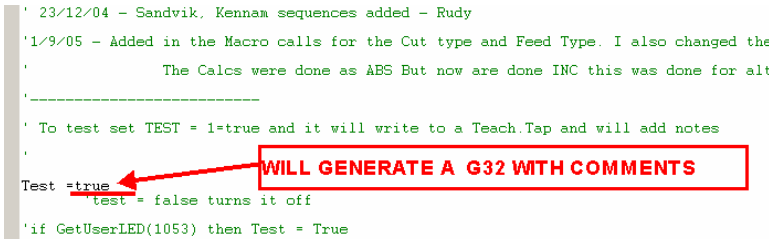


FIGURE 7.2.3

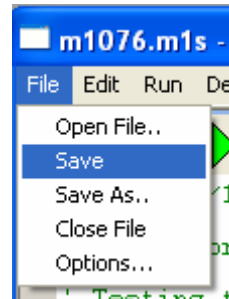


FIGURE 7.2.4

8.0 MULTI START THREADING

In order to cut a multi start thread it's necessary to first understand the nomenclature and how it works. Pitch, P, is defined as the distance between two adjacent crests (for metric threads). Lead, is the distance the screw advances when it's turned one revolution. For a single start thread, lead and pitch are equal. On a double thread , the lead is 2x pitch. (see section 4.1)

Figure 8.1 provides a good example of a three start thread.

A thread in which the lead is an integral multiple, greater than one, of the pitch. A multi-start thread permits a more rapid advance without a coarser (larger) thread form,

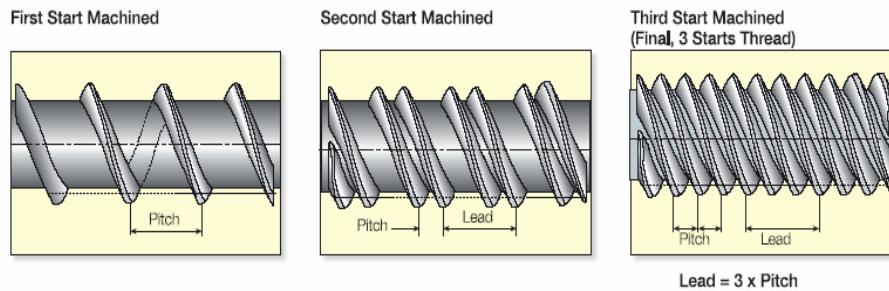


FIGURE 8.1

There are few different ways to cut multiple start threads. A simple way to do multi start threads would be to just add an offset (1/2,1/3,1/4 etc of pitch) to the Z starting position. That is how the scribe test in Section 3.6.3 was coded. Thus if the thread started at Z=0, then the next thread to be cut is offset by .1/3=0.3333” , etc as shown in red below for the scribe test code below. The G76 line is the same.

```

N10 M3 G18 G20 G40 G49 G61 G80 G90 G94
N20 G00 X0.625 ( OD OF TUBE )
N30 G00 Z0.3 ( 3X PITCH AND ASSUME THREAD STARTS AT Z=0)
N40 G76 X0.615 Z-0.625 Q0 P0.1 J0.001 L0 H0.001 I30 C0.1 T0
N50 G00 X0.625
N60 G00 Z0.03333 ( 1ST Z OFFSET )
N70 G76 X0.615 Z-0.625 Q0 P0.1 J0.001 L0 H0.001 I30 C0.1 T0
N80 G00 X0.625
N80 G00 Z0.06666 ( 2ND Z OFFSET )
N90 G76 X0.615 Z-0.625 Q0 P0.1 J0.001 L0 H0.001 I30 C0.1 T0
N100 M30
    
```

The machine will cut three individual threads. The user may want to consider adding additional code such that three single spring passes are done in sequence since the same x axis position (it doesn't move / same accuracy of position is applied to each thread) would be used to cut the spring passes. I have never tried applying the G92 for the spring passes (see gcode reference in Using Mach3 Turn manual).

WW:

You can just past together a number of Gcode files generated by the Simple Threading wizard (See section 6) along with minor editing. Use the Quick Threads Wizard which allows for posting multiple threads but make sure you provide for an offset of the Z. You can also change the pitch, multiply the pitch by the number of starts, but use 6 digits.

Do not exceed your max feedrate when doing a multiple start thread. ie; If it's a 4 start lead then the feedrate is 4x of just a single start thread. There no warnings if the feedrate exceeds your defaults and the machine will move as fast as it can but lead and pitch will be incorrect.

There is another way of using G76 canned cycle and triggering based on angular sensing as shown below. **Note:** This is not currently work (hangs on the second G76) in version 3.042.032 but is noted here, and, a fix in the new code is required. It was based on interrupt code of past versions. For a three start thread $360 \text{ deg} / 3 = 120$ as shown in red below.

```
#99066 = 0 ( Start angle of the first thread )
N40 G76 X0.615 Z-0.625 Q0 P0.1 J0.001 L0 H0.001 I30 C0.1 T0
#99066 = 120 ( start angle pf the second thread )
N40 G76 X0.615 Z-0.625 Q0 P0.1 J0.001 L0 H0.001 I30 C0.1 T0
#99066 = 240 (start angle of the third thread )
N40 G76 X0.615 Z-0.625 Q0 P0.1 J0.001 L0 H0.001 I30 C0.1 T0
```

For info purposes: 66 is the DRO number of the start angle for a thread. When used with #99(DRO number) the DRO is accessible in Gcode.

9.0 HOW TO PICK UP A THREAD

Picking up a thread on a CNC lathe is similar to what you would do on a manual lathe. The big difference is you need to confirm that the Z axis location is at the correct position relative to the thread and index timing / triggering. The user should have done the triggering test (section 3.5) and scribe test (section 3.6.4) .

The user should consider the following:

- 1.Picking up a thread will only be as accurate as the “lathe system”
2. Scenarios
 - piece not removed from chuck and threading is complete – maybe you need to tweak the thread, then use the DRO values to appropriately move the Z & X back to the correct starting position and just use the G32 Z.....F..... and step in the X as required. (user can write a small Gcode file). If you don't know the current axis positions then you need to pick up and confirm axis locations.
 - new piece inserted into chuck for repair- need to accurately set up the piece, pick up the thread, and be careful since you need to select a good thread (I would not use the first or last thread) and your lathe may cut the thread to a different tolerance.
3. The user must decide what he wants to pick up ie; center of the V or either flank side but comment 1 above applies.

PROCEDURE TO PICK UP A THREAD

The spindle is off, the cutter tip is not in the thread.

1. Manually turn the spindle to midway between the on/off triggering positions of the index (see section 3.5) and leave it in that position

- If your in Mach3Turn you may want to use the MDI line and do a G80 & G94.

2. Move the thread cutter tip into alignment with the point you wish to pick up.

- Use the root at minor diameter of the thread V, “tweak” if desired to the front or back flank of the V

- Zero the Z axis DRO

Note: Move to your point such that backlash is taken out. MPG use along with a magnifier helps with the alignment.

3. Move the cutter clear of the thread outside diameter, now make + Z moves in increments of the thread lead. ie; for 20 TPI you will move $1/20 = .050$ increments. You should move beyond the end of the shaft by approx 3 to 5X the lead (ie; $3 \times .05 = .150$ ") to allow for acceleration. Single start thread, so pitch =lead. Make note of the total distance you moved the Z axis.

- Zero the Z axis DRO and now make one more additional +Z move and return back to Z =0 to remove any backlash.

4. Turn the spindle on, now do a G32 Z -***** F.050 from Z=0 where ***** is the distance you made note of in step 3 above (you want to go back to the same point) and F value is the thread lead using the MDI.

- Turn off the spindle, reset the lathe by typing G80 & G94 in the MDI, set the DRO to Z=0

5. Realign the spindle as in the first step. Move the X axis to see if the cutter tip aligns with the pick up point. If all aligned then no need for a Z adjustment. If not adjust the Z again and retest.

6. Move the Z and X into the appropriate positions and now you can use G32 moves to accomplish what you want to do.

What you have done is the equivalent of picking up the thread on a manual lathe with an indexer dial. This doesn't take 5 minutes, and if you know what your doing it is rather quick to do. Simply analogy to a manual lathe, you have closed the half nuts, picked up the thread using the compound slide, tested it by dry running.

10.0 REFERENCES

AMERICAN NATIONAL STANDARDS INSTITUTE (ANSI)

ANSI B5.10 - Machine Engine Lathes, Turret Lathes, and Automatic Lathes

ANSI B5.16 - Accuracy of Engine and Tool room Lathes

AMERICAN SOCIETY OF MECHANICAL ENGINEERS (ASME)

ASME B1.1 - Unified Inch Screw Threads (UN and UNR Thread Form)

ASME B1.13M - Metric Screw Threads – M Profile

ASME B1.21M - Metric Screw Threads – MJ Profile R

INTERNATIONAL ORGANIZATION FOR STANDARDIZATION (ISO)

ISO 1708 - Acceptance Conditions for General Purpose Parallel Lathes Testing of the Accuracy

ISO 230-1 - Test Code for Machine Tools – Part 1: Geometric Accuracy of Machines Operating Under No-Load or Finishing Conditions

JIS – JAPANESE INDUSTRIAL STANDARD

MISCLEANOUS / COMPANY LITERATURE

GREENFIELD SCREW THREAD MANUAL

KENNAMETAL

VARGUS

MACHINERY'S HANDBOOK

CNC PROGRAMING TECHNIQUES – PETER SMID

SME – SOCIETY of MANUFACTURING ENGINEERS

USING MACH 3 TURN MANUAL

ARTSOFT MACH3 FORUM

MACH3 TURN VER:\3.042.031

11.0 APPENDIX LIST

A – LATHE SPECIFICATION , TESTING & TOLERANCES

B – JIS STANDARD TOLERANCES

C – INITIALIZATION MACRO

APPENDIX “A” LATHE SPECIFICATION, TESTING & ACCURACIES / TOLERANCES

Table II is lathe specification to a manufacturer.

TABLE II -- ACCURACIES

Requirements	Tolerance
Bed (Verification of Leveling of Slide ways)	
Longitudinal Verification (In vertical Plane)	0.0008" (Convex)
Local Tolerance	0.0003"/10"
Transverse Verification (In Vertical Plane)	0.0008"/20"
Carriage (Straightness of Movement in Horizontal Plane)	0.0008"
Parallelism of Tailstock to Carriage Movement	
In the Horizontal Plane	0.0012"
In the Vertical Plane	0.0012"
Local Tolerance (*)	0.0008"/20"
Headstock Spindle	
Periodic axial slip	0.0006"
Camming (**) of the Face Plate Resting Surface	0.0008"
Runout of Spindle Nose Centering Sleeve	0.0006"
Runout of Axis of Center	
At the Spindle Nose of the Housing	0.0004"
At 12 inches from the Spindle Nose	0.0008"
Parallelism of Spindle Axis to Carriage	
Longitudinal Movement on a Length of 12 inches	
In the Horizontal Plane	0.0006" Frontwards
In the Vertical Plane	0.0008" Upwards
Runout of Center	0.0006"
Tailstock	
Parallelism of the Axis of the Outside of Sleeve to Carriage Movement at 4 Inches	
In the Horizontal Plane	0.0006" Frontwards
In the Vertical Plane	0.0008" Upwards
Parallelism of Taper Bore of Sleeve to Carriage	
Movement at 12 inches	
In the Horizontal Plane	0.0012" Frontwards
In the Vertical Plane	0.0012" Upwards
Centers (Difference of Height Between Headstock and Tailstock Centers)	0.0016"
Upper Slide (Parallelism of the Slide Longitudinal Movement to the Spindle Axis)	0.0016"/12"
Cross Slide (Squareness of the Transverse Movement of the Slide to the Spindle Axis)	0.0008"/12" at 90 Degrees
Leadscrew	
Periodic Axial Slip	0.0006"
Cumulative Error of the Lead Screw	

For any Measured Length of 12 Inches	0.0016"
For any Measured Length of 2-1/2 or 3 Inches	0.0006"
Face plate runout,	
On outside Diameter	0 to 0.001"
On Face at Nominal Diameter	0 to 0.0015"
Three-Jaw Chuck Runout,	
Face and Periphery	0.0006"
Face of Steps	0.0008"
Bar test 3 inches from End of Jaw with the Test	0.0012"
Bar Diameter the Same as the Spindle Hole	
Four-Jaw Chuck Runout,	
Face and Periphery	0.0006"
Face of Steps	0.0008"
Bar test 3 inches from end of Jaw with the Test	0.0012"
Bar Diameter the Same as the Spindle Hole	
Collet Chuck, Runout, 1 inch from Collet Chuck	0 to 0.0008"

Round Rod Turning Requirement. The lathe shall develop the power required to perform rough and finish cutting operations under the conditions stated herein. The turning operation shall be performed on a round, low carbon, steel (1020) bar, no less than 2.00 inches in diameter, no less than 6 inches long, and mounted in the 3-jaw chuck supplied with the lathe.

Rough Cutting Requirement. The rough cut shall be no less than 0.040 inches deep and no less than 3.00 inches long. The cut shall be made at a feed rate of not less than 0.010 inches per revolution at a spindle speed of not less than 700 revolutions per minute (rpm). The rough turned diameter shall show no evidence of chatter and shall meet a total tolerance requirement of 0.0005 inches or less for both out-of-round and taper per foot.

Finish Cutting Requirements. Finish cutting operations shall be performed on the same steel bar as the rough cut. The finish cut shall be no less than .020 inches deep and no less than 3.00 inches long. The finish cut shall be made at a feed rate of not less than 0.005 inches per revolution at a cutting speed of not less than 1,000 rpm. The finished turned diameter shall meet a total tolerance requirement of 0.0005. The finish of the machined diameter shall be no less than 63 micro-inches aa.

Cylindrical Turning Requirement. The lathe shall machine cylindrical diameters, on a low-carbon, steel (1020) bar, no less than 2.00 inches in diameter and no less than 15 inches long. The bar shall be held in the 4-jaw chuck and supported by a live tailstock center. The lathe shall machine no less than three diameters on the bar, as illustrated in test P1 of ISO Standard 1708. The L, dimension of test P1 shall be no less than 12 inches. The machined diameter shall be no less than 0.250 inches less than the D diameter. The test piece shall be machined at a spindle speed of not less than 700 revolutions per minute, a cutting depth of no less than 0.020 inches, and a feed rate of not less than 0.005 inches per revolution. A single point carbide-cutting tool shall be used. The variation in the machined diameter, at the tailstock end of the test piece, shall be not greater than 0.00025 inches. No less than four readings shall be taken. See clause 14.3 of ISO Standard 1101 for a definition of circularity tolerance. The variation between machined diameters at either end of the test piece shall be not greater than 0.0005 inches, measured in a single axial plane. Any taper noticed in the test piece shall have the major diameter near the headstock end of the test piece. The general testing guidance provided in ISO 230/1, clauses 3.1, 3.22, 4.1, and 4.2, is applicable to this requirement. The lathe shall machine a finish cut across the collars of the test piece machined as stated herein. The across-the-collars machine cut shall be made at a lathe spindle speed of no less than 1500 rpm, a cutting depth of no less than 0.002 inches, and a feed rate of no less than 0.002 inches per revolution. The cut shall be made with a single point carbide-cutting tool. The variation in the three machined collar diameters shall be not greater than 0.0004 inches. The finish cut shall have a surface finish of no less than 20 micro-inches aa.

Threading Requirement. The lathe shall machine threads on a steel test bar. Two threads, at least four inches long, shall be cut, a 1/2-13 UNC thread and a 1/2-32 UNF thread as defined by ASME B1.1. The machined threads shall meet the size and shape requirements of ASME B1.1.

Center Sleeve. One headstock center sleeve shall be provided for adapting the taper in the spindle nose to the taper of the headstock spindle center supplied. Concentricity shall be within 0.0002 inches total indicator reading. Tapers shall be in accordance with ANSI B5.10 for self-holding tapers.

Lathe Centers. Two lathe centers shall be furnished, one headstock spindle center and one tailstock spindle center. All centers shall conform to ANSI B5.10. The centers shall be hardened to Rockwell C-62 to C-68 and ground to within 0.0002 inch total indicator reading for concentricity.

Chuck, 4-jaw, Independent. The chuck body shall be steel and shall fit to the spindle nose without adapters other than as a part of the chuck. Reversible step jaws with adjusting screws and wrench shall be included with each chuck. All working parts of the chuck shall be heat-treated. The total indicator reading on the periphery of the chuck, face of chuck body, and the face of the jaw steps shall be in accordance with the requirements of Table II and ANSI B5.8.

Chuck, 3-jaw, Universal. The chuck shall be steel, universal geared, and self-centering. All working parts shall be heat-treated. Run out of the chuck periphery, face of the body, and jaw steps shall be in accordance with the requirements of Table II and ANSI B5.8.

Spindle Nose Collet Chuck. The spindle nose collet chuck shall meet or exceed the accuracy requirement of Table II.

Alignment Accuracy Tests. The lathe shall be examined for conformance to the measurements listed in Table II. Where applicable, the methods of measurement shall be as explained in ISO 1708, but the tolerance limits shall be as specified in Table II.

Center Sleeve Concentricity. The center sleeve shall be inserted in the headstock and shall be checked for internal concentricity.

Lathe Center Concentricity. The two lathe centers supplied with the lathe shall be inserted in the headstock and tailstock and checked for concentricity over their entire surfaces (centers contacting work-piece).

Performance Tests

The lathe shall be subjected to the following turning tests after the machine has reached operating temperature.

Round Turning Test. A bar of low carbon steel (1020) not less than 2 inches in diameter and not less than 6 inches long shall be mounted in the 3-jaw chuck supplied with the lathe. A rough cut shall be made at a tool depth of not less than 0.040 inches with a feed rate of not less than 0.010 inches per revolution and a spindle speed of not less than 700 revolutions per minute (rpm). The rough cut shall be made for a distance of not less than 3 inches along the length of the test bar. Upon completion of the rough cut, with the same cutting tool, a finish cut shall be made at a tool depth of not less than 0.020 inches with a feed rate of not less than 0.005 inches per revolution and a spindle speed of not less than 1000 revolutions per minute (rpm). The finish cut shall be made for a distance of not less than 3 inches along the length of the test bar. The finished turned diameter shall be round within 0.0005 inch.

Metal Removal Turning Test. A bar of low carbon steel (1020) approximately 5 inches in diameter and 15 inches long shall be held in the 4-jaw chuck supplied with the lathe, and supported by the tailstock center. A cut shall be made for a length of not less than 12 inches using a single point carbide tip turning tool. The metal removal rate shall be at least 1 cubic inch per minute per horsepower rating of the motor.

Cylindrical Turning Test. Using the same test bar as for the round turning test, the specimen shall be semi-finished to the configuration shown in Test P1 of ISO 1708. A finish cut shall be taken over the three collars in one pass using a single point carbide tip turning tool and a cutting speed of not less than 350 surface feet per minute. The turned diameters shall conform to the accuracy of P1, ISO 1708. Upon completion of the test, the specimen shall be prepared for turning between centers. A single cut shall be taken over the three collars, and the turned diameters of the three bands shall conform to the accuracy of test P1, ISO 1708.






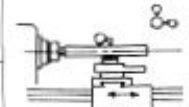




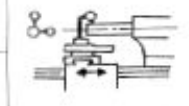
Collet and Drill Chuck Test. A length of 3/16 inch diameter low carbon steel (1020) shall be gripped by an appropriate collet furnished with the lathe. A #70 drill bit shall be inserted in the drill chuck furnished with the lathe, and the chuck inserted in the tailstock. Using the tailstock hand wheel, drill a #70 hole 1/4 inch deep. Remove the collet and drill bit. Install another collet furnished with the lathe and grip a length of 3/4 inch round low carbon steel (1020). Insert a 1/2 inch drill bit in the drill chuck and drill a hole 1 inch deep in the steel. Using the hand wheel, back the drill out of the work until the drill chuck is ejected. Failure of the collets to prevent slippage of the work pieces, failure of the drill chuck to hold the bit securely, or failure of the chuck to eject when the tailstock was retracted shall be cause for rejection.

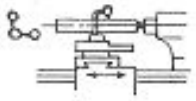
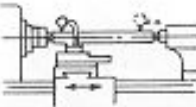
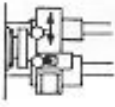
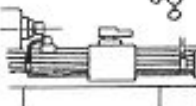
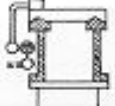
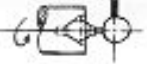

Threading Test. Provide two 1/2 inch bars of low carbon steel (1020) of sufficient length so no less than 4 inches of threads can be cut. Set the lathe to cut 1/2-13UNC threads over a length of at least 4 inches. Repeat cutting 1/2-32UNF threads on the second bar. Failure of the screw threads to meet the acceptability requirements of ASME B1.1, ASME B1.13M, and ASME B1.21 shall be cause for rejection.

APPENDIX “B” – JIS STANDARD TOLERANCES

12. Accuracy Test of Lathe

Unit: mm

Test NO.	Test to be applied		JIS tolerances (Swing up to 500)	Test result
1	Bed level Longitudinal direction in vertical plane		0.04/1000 (Bed to be convex)	
	Transverse direction in vertical plane		0.04/1000	
	Longitudinal direction in horizontal plane		0.02 Per 2000 (For machines of not over 2000 center distance)	
2	Parallelism of bed slide ways in over-all length		0.02	
3	Spindle nose runout		0.01	
4	Spindle taper hole runout		at end of spindle nose 0.01	
			at end of 300 test bar 0.02	
5	Head-stock alignment in vertical plane		0.01 per 300 high at free end of test bar	
	Head-stock alignment in horizontal plane		0.01 per 300 forward at free end of test bar	
6	Upper slide alignment in vertical plane		0.03 per 150	
7	Spindle sleeve hole runout		at end of spindle nose 0.02	
			at end of 300 test bar 0.03	
8	Cam action of spindle		0.015	
9	Spindle center runout		0.02	
10	Tail-stock spindle align- ment in vertical plane		0.02 per 150 high at free end	
	Tail-stock spindle align- ment in horizontal plane		0.01 per 150 forward at free end	

Test NO.	Test to be applied		JIS tolerances (Swing up to 500)	Test result
11	Tail-stock taper alignment in vertical plane		0.02 per 300 high at free end of test bar	
	Tail-stock taper alignment in horizontal plane		0.02 per 300 forward at free end of test bar	
12	Vertical alignment of head and tail-stock center		0.02 high at tail-stock	
13	Cross slide alignment		0.02 per 300 to face, concave only	
14	Lead screw alignment in vertical plane		0.1	
	Lead screw alignment in horizontal plane		0.1	
15	Alignment of lead screw with half nut closed in vertical plane		0.15	
	Alignment of lead screw with half nut closed in horizontal plane		0.15	
16	Cam action lead screw		0.01	
17	Pitch error of lead screw		0.03 per 12in.	

This test was performed according to the Japanese Industrial Standard and the accuracy was shown in metric size. For the approximate conversion into inch system, each figure should be divided by 25.4.

APPENDIX “C” – INITIALIZATION MACRO

This is a macro that can be used to “reset” the lathe to a user preferred state. The macro should be reviewed and then modified for the users working preference. The macro must be placed in the proper turn profile folder and can be used as an Initialization String, called in a program or implemented via the MDI line.

To modify the macro use the VB script editor available in Mach. See Section 10.9 in the “Using Mach3 Turn Manual” for additional info on macros.

This is the macro:

```
' ( M1111.M1S Macro )
' ( Needs to be installed in C:\Mach3\Macros\Mach3Turn Folder)
' ( Basic Default Location or as needed )
' ( Change it as you need to )
' ( For MM Units Replace G20 with G21 )
' ( For CV Mode Replace G61 with G64 )

Code "G18" '      Set X,Z Plane
Code "G20" '      Set INCH Units Mode
Code "G40" '      Cancel Cutter Radius Comp.
Code "G49" '      Cancel Tool Length Offset
Code "G50" '      Re-Set All Scale Factors To 1.0
Code "G61" '      Set Exact Stop Mode
Code "G69" '      Cancel G68 Rotate Coordinate System
Code "G80" '      Cancel Canned Cycle Mode
Code "G90" '      Set ABS Mode
Code "G91.1" '    Set IJ's INC Mode
Code "G94" '      Set Feed Per Minute Mode
Code "M30" '      Program End and Rewind G-Code
```