# THREADING ON THE LATHE-MACH3 TURN

# TABLE OF CONTENTS

## 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 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 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.

## 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, nut, 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 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 quess 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 100th's and read the scale within .005" easily for longer distances.
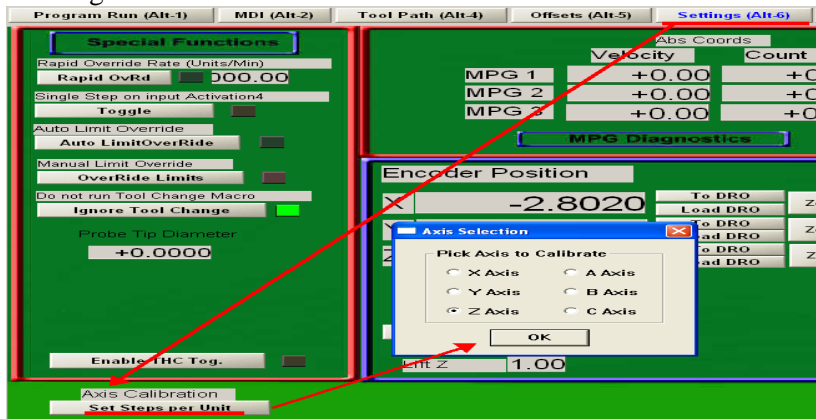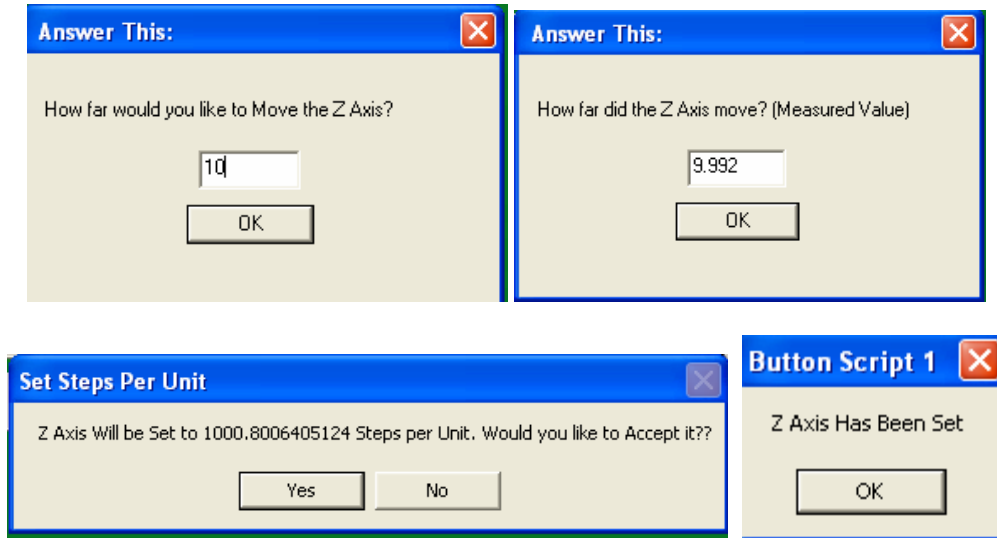
## 3.3 AXIS TESTS

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". These tests are done for
a different reason and relate to threading on a CNC'd lathe.

So the motor tuning is all done and the axis steps per unit is 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 runout
is and also how well you can turn to diameter over a distance. You may as well check how
good the new 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".

### WW:
Threadng is true test of your equipment and the finished threading will show.
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

Turn a piece of stock, say a ½" in diameter, for an inch using a fine feedrate with high spindle speed. Take two cuts if necessary such that one is a finishing cut. Measure the diameter in four quadrants near both ends of the cut end. There should only a very small difference and that is due to spindle runout. Using a strip of 400 grit abrasive cloth and lightly remove any finish marks. Clean and apply a thin colored coating. A permanent blue colored marker works great. Now bring an extremely pointed cutter to within say a 0.001".

LATER

## 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 G94 or G95 mode.
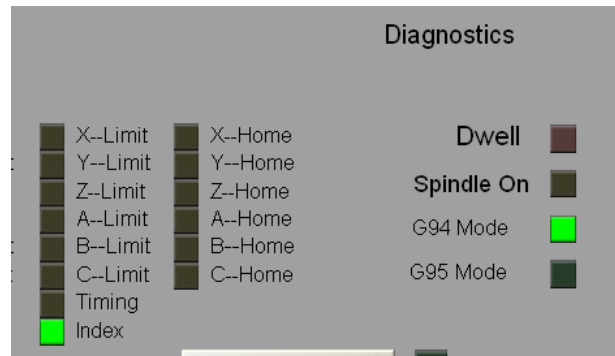


## FIGURE 3.5.1

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.
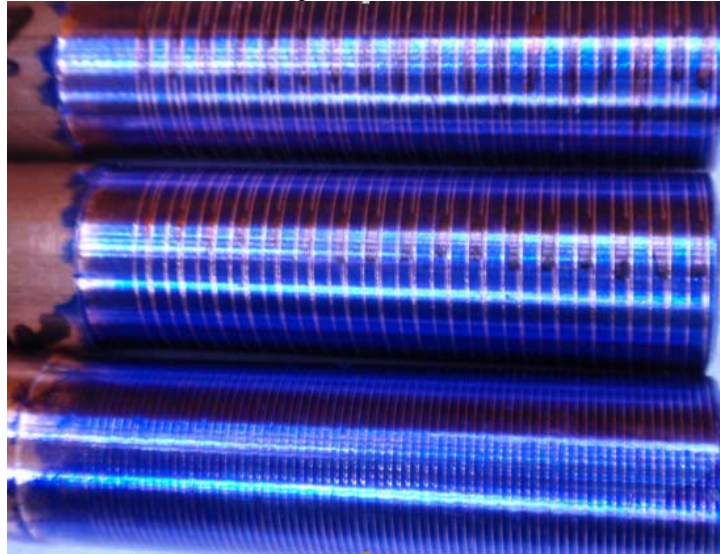


**FIGURE 3.6.1.1**

The tool used for scribing should have an extremely sharp pointed tip. As shown in figure3.6.1.2 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.
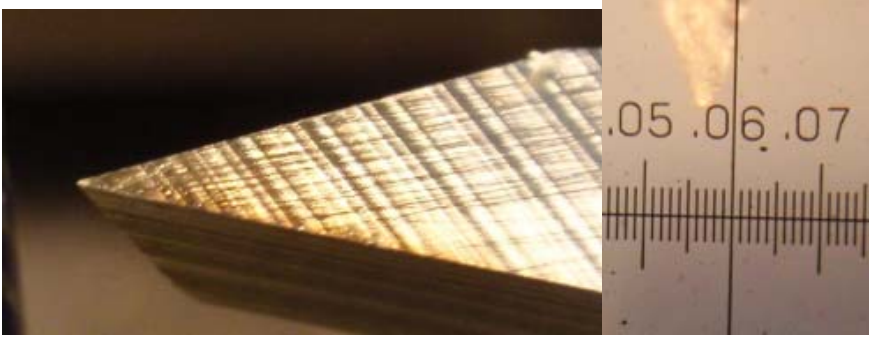
**FIGURE 3.6.1.2**

The scribed line should be a single line as shown and magnified in figure 3.6.1.3.Note that that single line is 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.
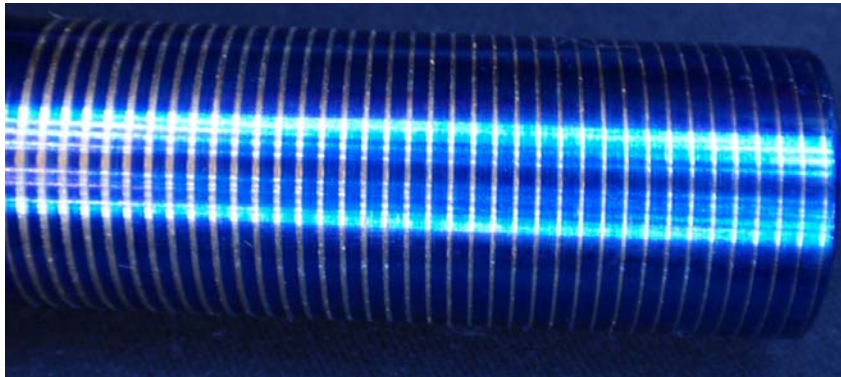


**FIGURE 3.6.1.3**                    **FIGURE 3.6.1.4**

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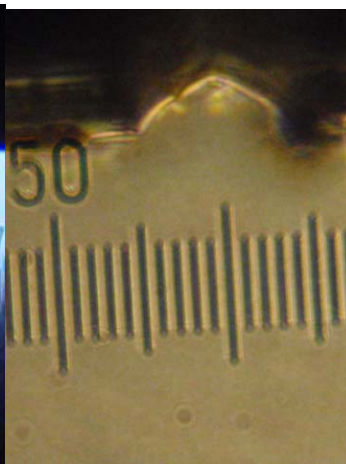


**FIGURE 3.6.1.5**                    **FIGURE 3.6.1.6**

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.
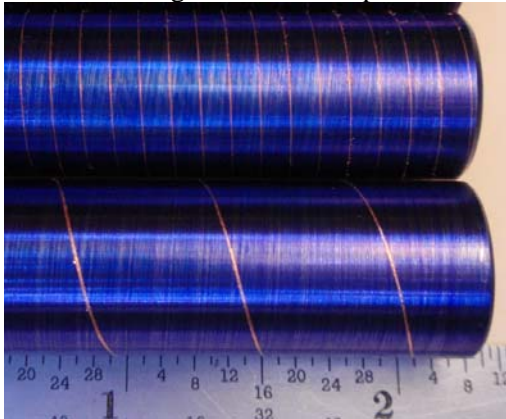


**FIGURE 3.6.1.7**

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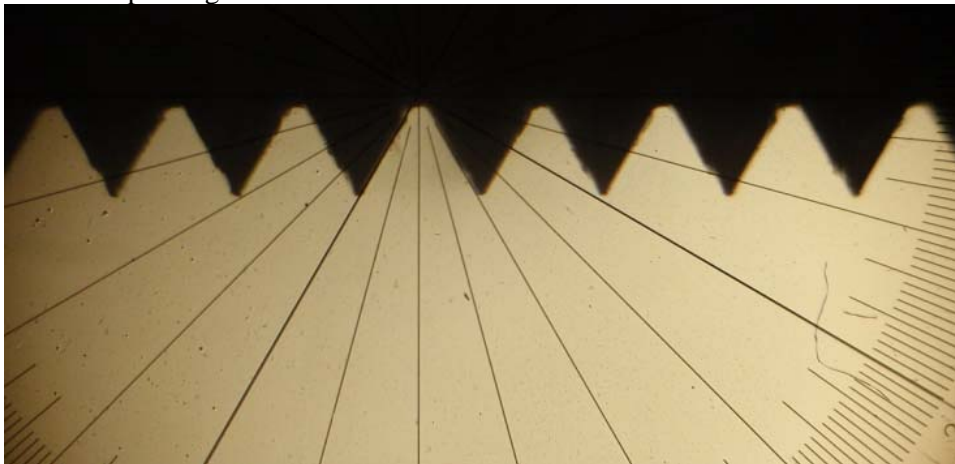


**FIGURE 3.6.1.8**

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 a good as the lead of 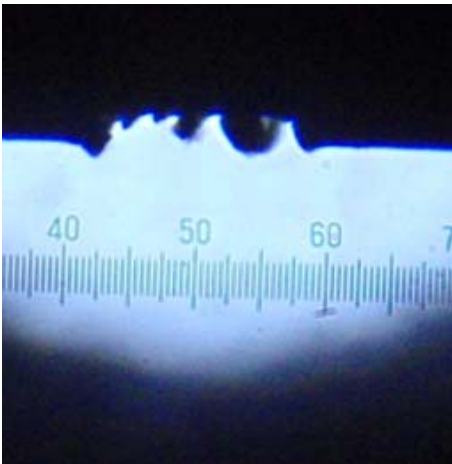your ball screw ie; for a ball screw it can be in the rough range of  0.0003"/foot to 0.003" per foot, the lathe screw can be ground or may be just a threaded rod. My 35 year old manual 11" Delta lathe has a ground screw, is in good condition, and scribed a single line with a lead error of only 0.0003" over a 6" length.

**3.6.2 ALTERNATE THREAD CUTTING TEST**
**LATER**
**3.6.3 MULTIPLE THREADS TEST**
This test will show if your lathe system is even 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 leads are maintained and cutting quality duplicated for each completed threading cycle. The best "quick indicator" is to look at the front of the thread and see if the individual starting points are scribed equally.

Here is the code used in this test:
LATER

**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 should be used. 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 your trying to do.
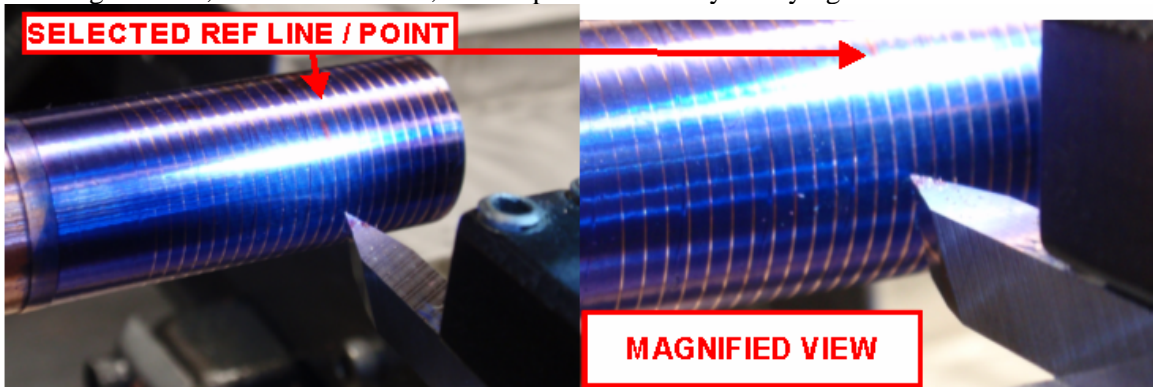


**FIGURE 3.6.4.1**

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.
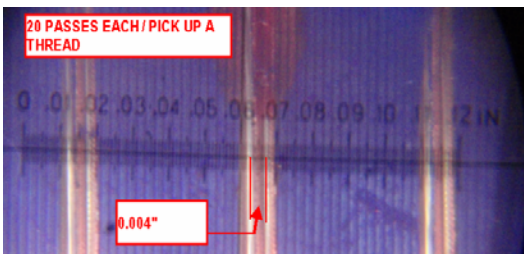


**FIGURE 3.6.4.2**

**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.
¼-20-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. ¼ -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 / acell, etc is off some you may catch the end. If it's consistant
the cutting is 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; normaly 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
machinabilty, etc, etc . So calculate if wish,  your wasting  your time. Enough said!
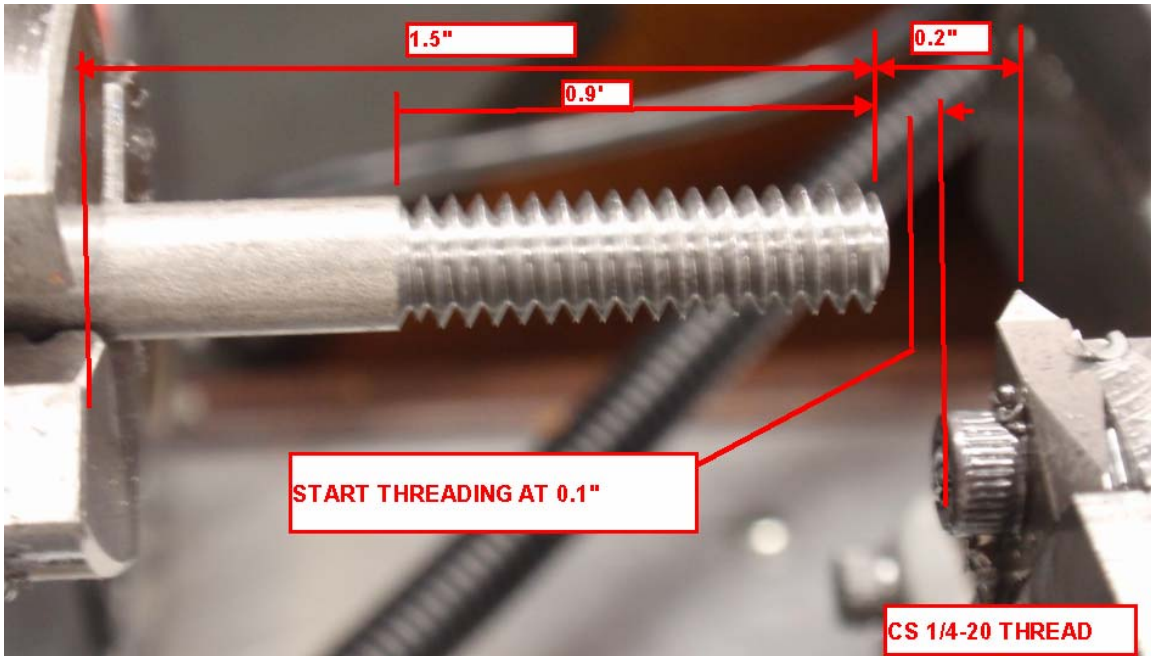


FIGURE 1

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. Actualy 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.
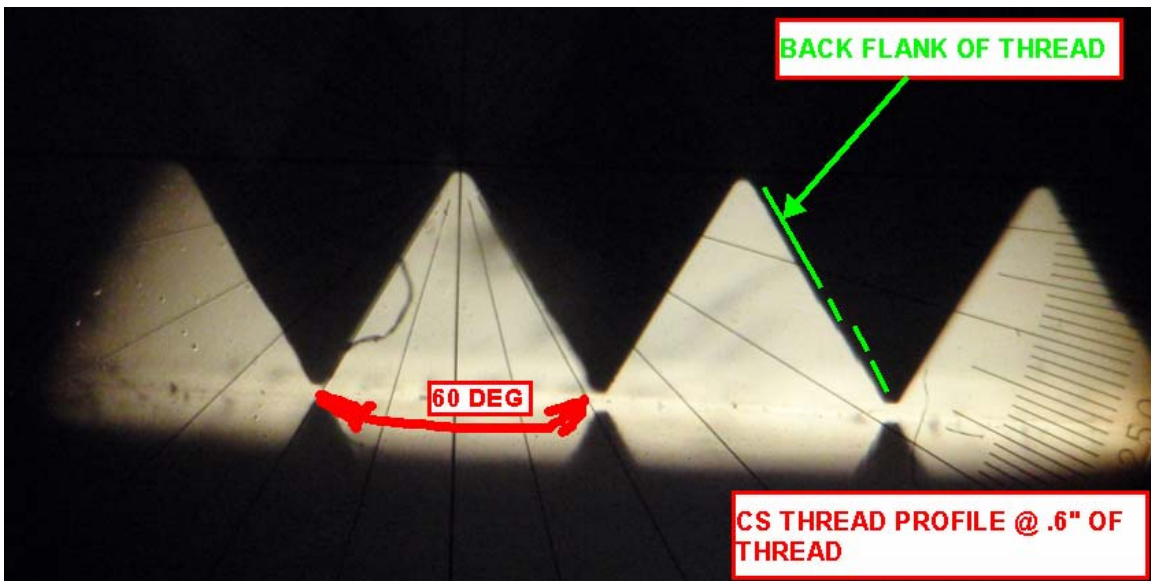


FIGURE 3

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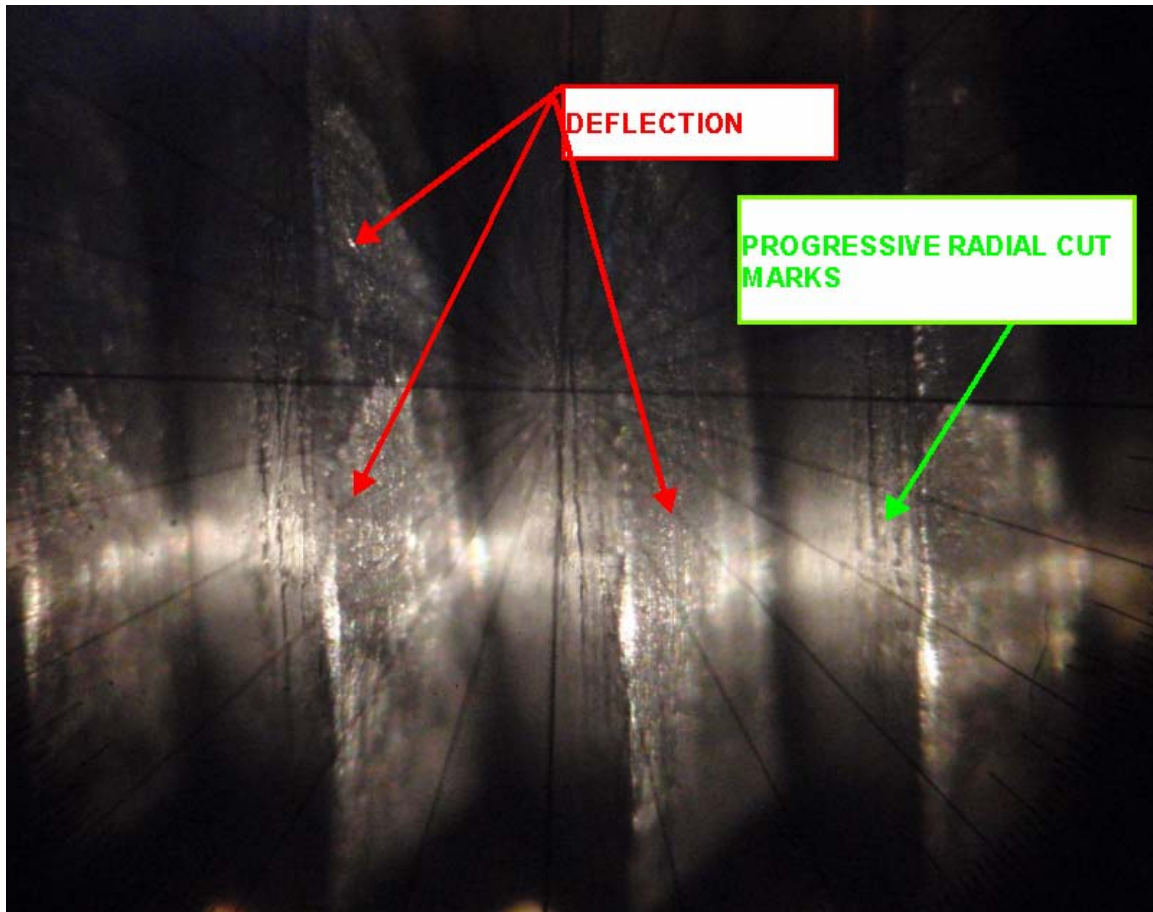
FIGURE 4

**Fi**gure 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. Visualy, 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!
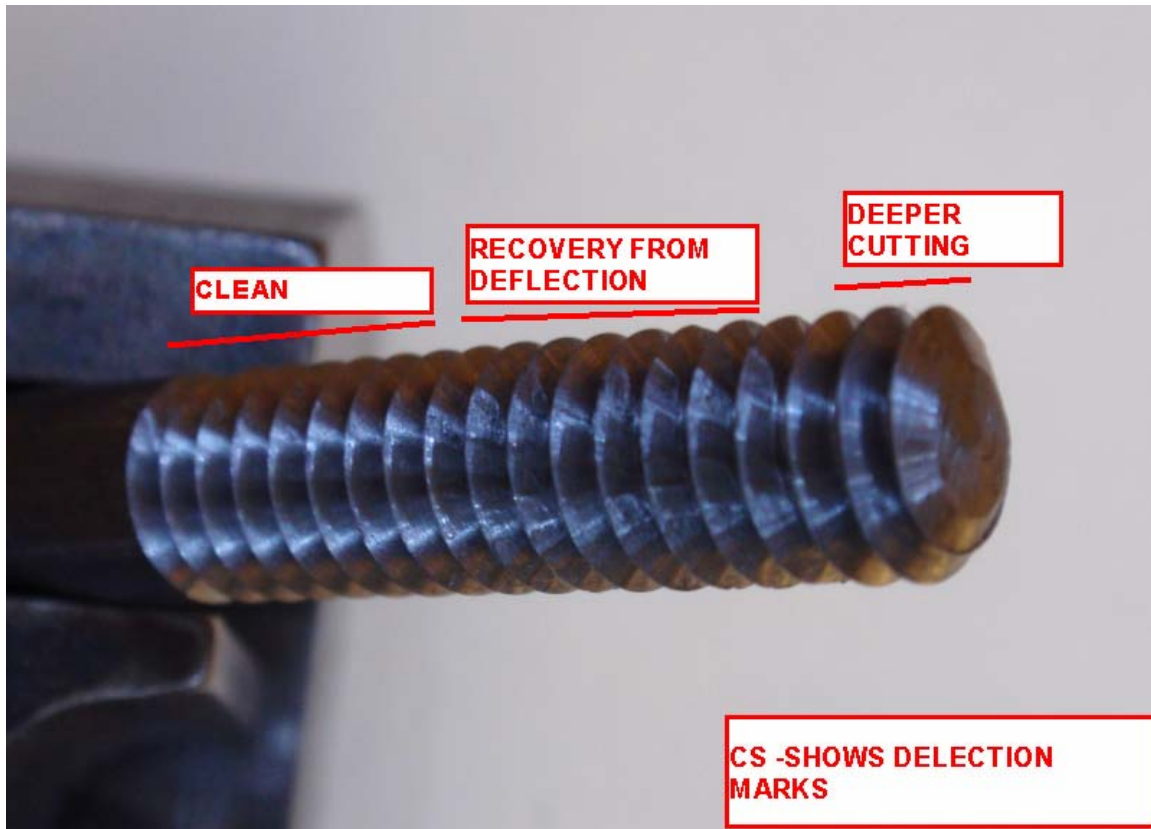
FIGURE 5

**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 plug in is currently loaded when a new or updated Mach installation is done.
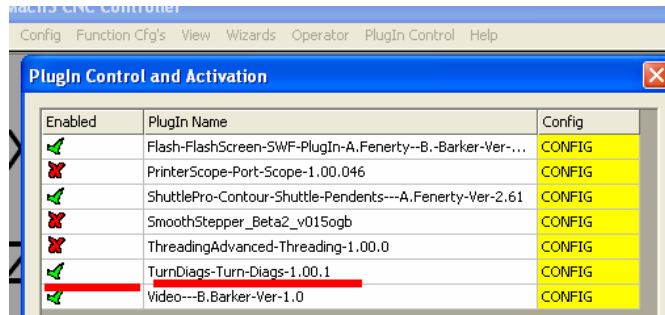
**FIGURE 3.7.1**

The plug-in will probably show your rotation speed realtime 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.
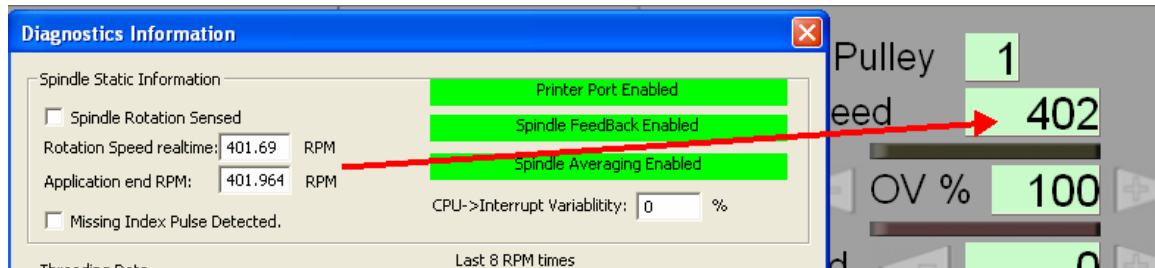

**FIGURE 3.7.2**

### 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:



**External Thread**
A thread on the external surface of a cylinder screw or cone

**Depth of Thread**
The distance between crest and root measured normal to the axis.

**Pitch**
The distance between corresponding points on adjacent thread forms measured parallel to the axis. This distance can be defined in millimeters or by the tpi (threads per inch), which is the reciprocal of the pitch.

**Nominal Diameter**
The diameter from which the diameter limits are derived by the application of deviation allowances and tolerances.

**Internal Thread**
A thread on the internal surface of a cylinder or cone.

**Major Diameter**
The largest diameter of a screw thread.

**Pitch Diameter**
On a straight thread, the diameter of an imaginary cylinder, the surface of which cuts the thread forms where the width of the thread and groove are equal.

**Minor Diameter**
The smallest diameter of a screw thread.

**Helix Angle**
For a straight thread, where the lead of the thread and the pitch diameter circle circumference form a right angled triangle, the helix angle is the angle opposite the lead.

**Straight Thread**
A thread formed on a cylinder
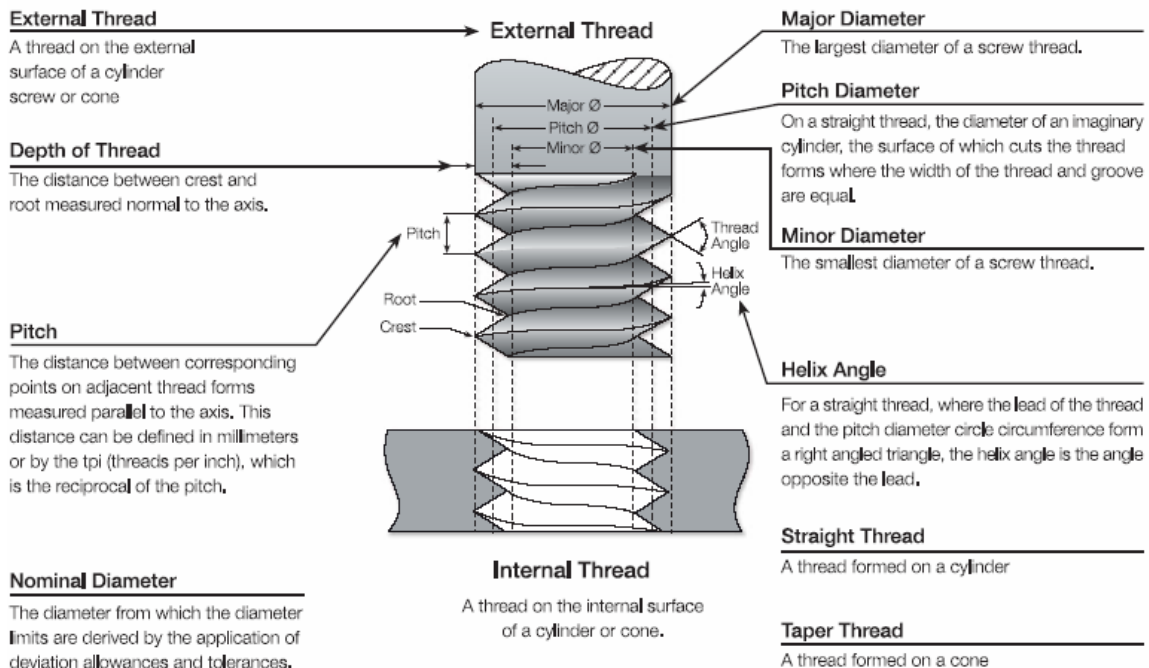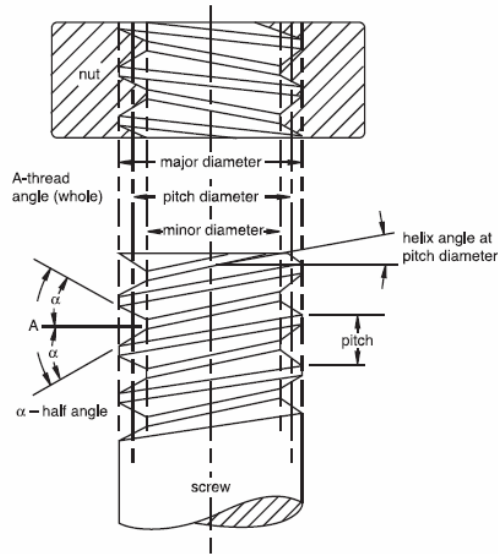
**Taper Thread**
A thread formed on a cone

**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**
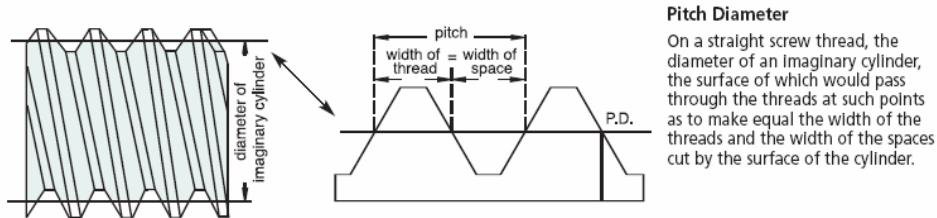
**Pitch Diameter**

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.

BASIC PITCH DIAMETER - A LINE ABOUT WHICH
THREAD TOLERANCES ARE BASED

MAJOR DIA

PITCH DIA

MINOR DIA
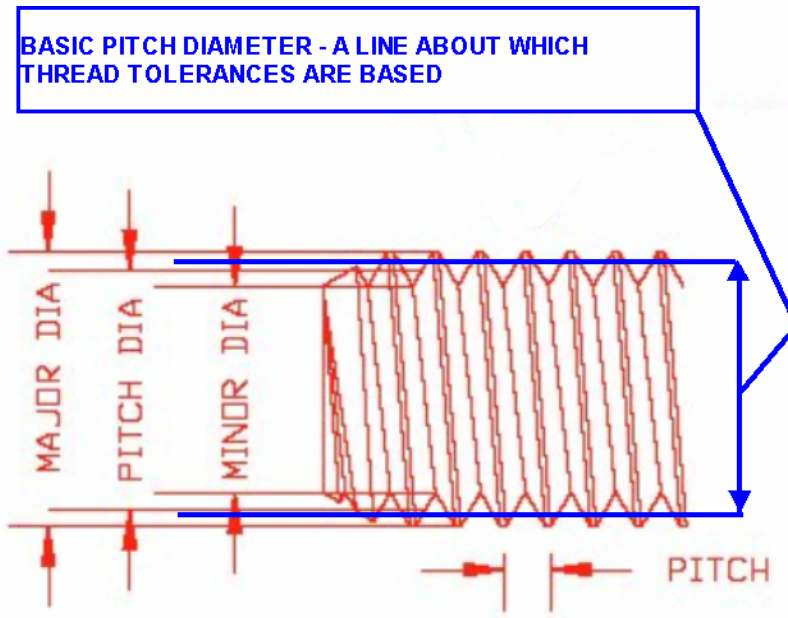
PITCH

**FIGURE 4.1.3**

## 4.2 DEPTH OF CUT BASIS

If you are using a sharp SHARP V tool, then DEPTH=.86603 X Pitch.

Check out a Machinst Handbook. Take a close look at the different forms. Peak to Peak of the sharp thread crests are defined as H=0.86603 x 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 x 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.
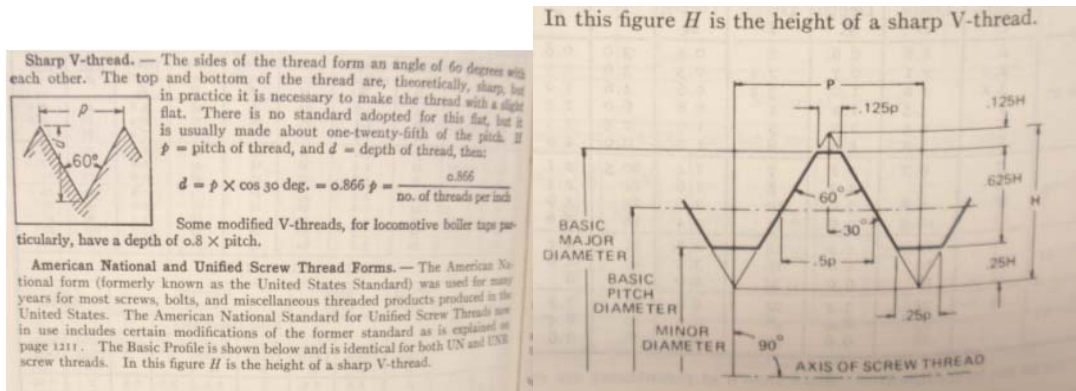


**FIGURE 4.2.1**

### 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.

P.D. = Pitch Diameter
M = Measurement over wires
W = Wire diameter
C = Constant

C = .86603 x Pitch (Inches) -3W
P.D. = M-C
W = .57735 x P

**Thread Measuring Wires**
Tolerance and specifications:
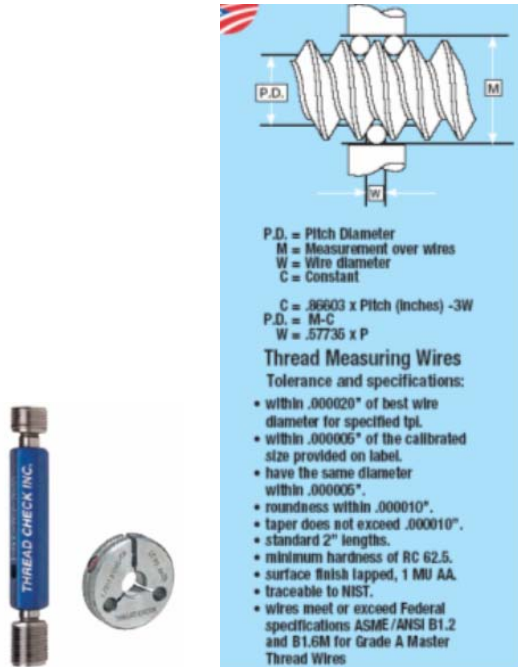- within .000020" of best wire diameter for specified tpi.
- within .000005" of the calibrated size provided on label.
- have the same diameter within .000005".
- roundness within .000010".
- taper does not exceed .000010".
- standard 2" lengths.
- minimum hardness of RC 62.5.
- surface finish lapped, 1 MU AA.
- traceable to NIST.
- wires meet or exceed Federal specifications ASME/ANSI B1.2 and B1.6M for Grade A Master Thread Wires

**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°    | 1.010362p | .57735p  | .505182p |
| 55°    | .82573p   | .563692p | .50568p  |
| 53°-8' | .97828p   | .55902p  | .54076p  |
| 47½°   | .72889p   | .54625p  | .49852p  |
| 29° Acme | .650013p | .51645p | .487263p |

p=Pitch   n=TPI   P= 1/n

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

**FIGURE 4.3.1                          FIGURE 4.3.2**

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 - ¾" 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.
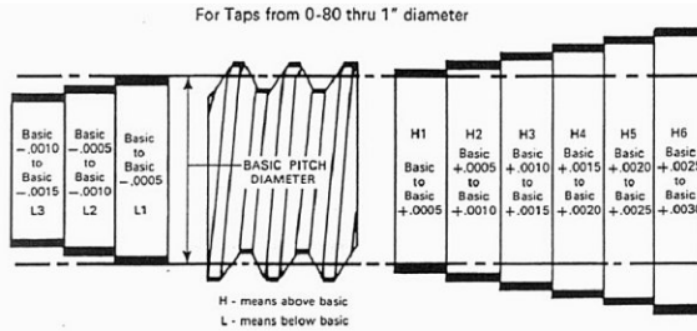Figure 4.4.1 shows the relationship for taps.



**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
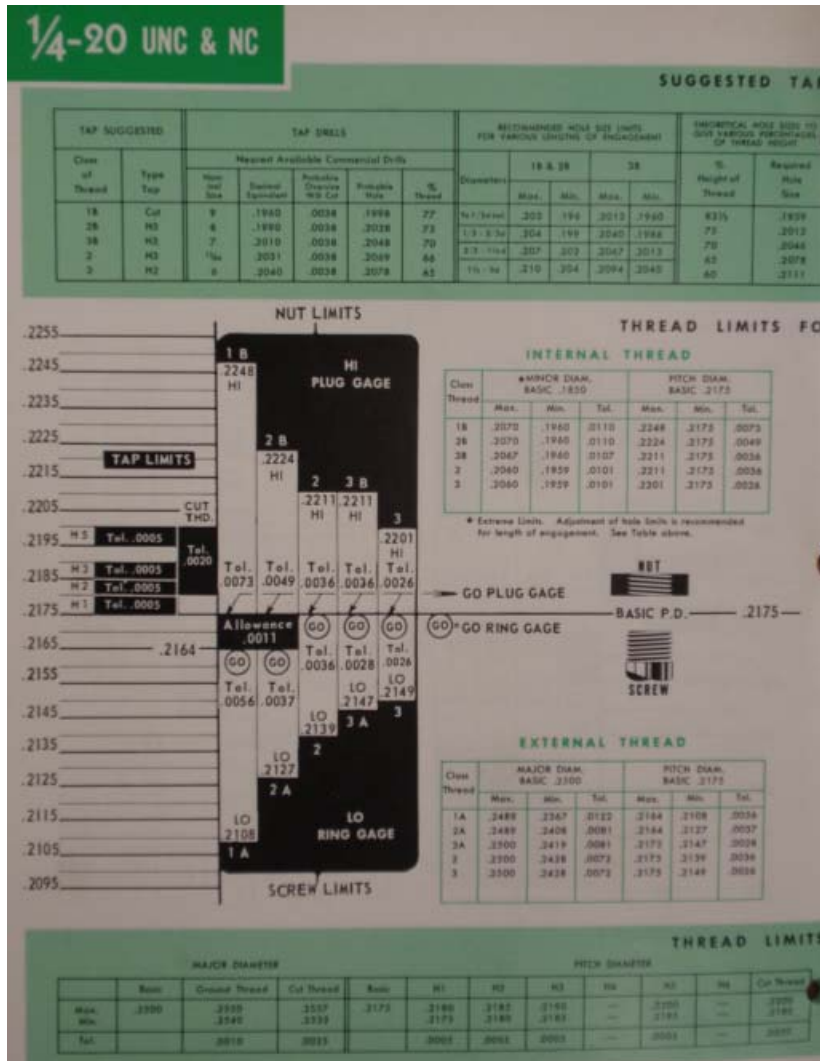internal is above it. The info is from the Greenfield Screw Thread Manual.



**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  ¾" 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 only starts and stops as compared to using a nut on an cut thread.**
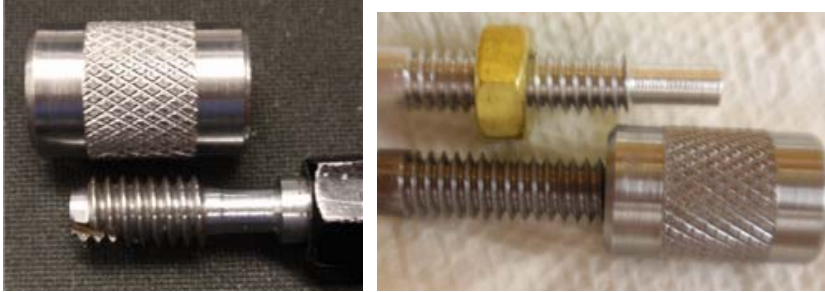


**FIGURE 4.4.3**

**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 )**

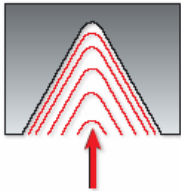| 2 | THREAD SERIES | SIZE | TPI | PITCH | (NOTE#2) LENGTH | (NOTE#1) ROUNDED 2A+- ALLOW | ACTUAL 2A +- ALLOW | (NOTE#1) ROUNDED 3A+- ALLOW | 3A+-ALLOW |
|---|---|---|---|---|---|---|---|---|---|
| 3 | UNC | #1 | 64 | 0.01563 | 0.1 | 0.001 | 0.00058 | 0.000 | 0.00043 |
| 4 | UNC | #2 | 56 | 0.01786 | 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 | 0.00011 |
| 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.**

## 5.0 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 ship control since the chip flow away from the tool.



**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 form start to finish.

**WW:**

So you now take a look at the Vargus info and there seems to be some difference in the use and name of the methods. 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.
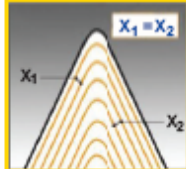
## VARDEX Tips 14 — Types of Passes
### December 2006

### Tip 1 — Thread turning - CONSTANT DEPTH chip method

- The insert cuts a constant depth $(X_1, X_2...)$ in each pass

- The high load on the last passes reduces the tool life

- Easy to program the machine

- As the insert advances, more material is removed. As a result, the load on the cutting edge is increased

- Not recommended

$X_1 = X_2$
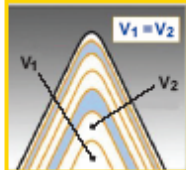
### Tip 2 — Thread turning - CONSTANT VOLUME chip method

- The insert removes the same amount of material $(V_1, V_2...)$ in each pass

- The load on the cutting edge is constant

- Tool life is improved in comparison to constant depth chip method

- Since each pass removes the identical volume, the first pass requires a deeper cut. This sometimes causes a higher load on the first pass

$V_1 = V_2$
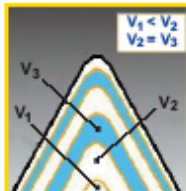
### Tip 3 — Thread turning - MODIFIED VOLUME chip method

**Recommended!**

- Modified constant volume chip passes uses the same principle as the constant volume method, except that the first pass uses a smaller depth of cut (smaller volume)

Advantages:

- Constant load on all passes

- No overload on the first pass

- Improved tool life

$V_1 < V_2$
$V_2 = V_3$

### One Final Tip — Super finish - Zero cut

- One or two additional passes taken at the same position as the final pass can provide improved surface finish

$V_1 < V_2$
$V_2 = V_3$
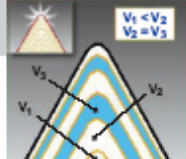
**FIGURE 5.1.1**

### 5.2 SPNDLE 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.



**radial**

advantage –
- Cutting on both sides of the thread form places all of the cutting edge in the cut and protects edge from chipping.

disadvantage –
- Tool develops a channel chip that may be difficult to handle.
- Tip chipping occurs when cutting high-tensile materials.
- Burr condition is increased.
- Entire cutting edge is engaged at finish of thread, causing increased tendency to chatter.

**flank**

advantage –
- Cutting with the leading edge of the threading tool gives the chip a definite flow out of the thread form area. This reduces the burr problem on the trailing edge of the tool. To avoid bad surface finish, chipping, or excessive flank wear due to rubbing of the trailing edge, the infeed angle should be 3° to 5° smaller than the angle of the thread. This is a type of modified flank.

disadvantage –
- Trailing edge of threading insert may drag or rub, and tends to chip.
- Torn or poor surface finish threads result when cutting soft, gummy materials like low-carbon steels, aluminum, and stainless steels.

**modified flank**

advantage –
- Tool cuts both sides of thread form and, therefore, is protected from chipping similar to 0° infeed. Channel-type chip develops but uneven chip thickness helps remove the chip similar to flank infeed.
- This is the preferred method, especially when used with a chip control insert.

disadvantage –
- Similar disadvantages as with 0° infeed, although reduced somewhat in magnitude as cutting forces are better equalized and chip flow is much less of a problem.

**alternating flank**

advantage –
- Increased tool life because both edges are used equally. NOTE: Some machine tools may require special programming techniques to achieve this method of infeed.

disadvantage –
- Difficult to cut on conventional machinery.

## FIGURE 5.3

## 5.4 FORMULAS

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

**Formulas**

|  | to find | given | formula |
|---|---|---|---|
| inch | sfm | D (inch) rpm | $sfm = \frac{\pi \times D}{12"} \times rpm$ |
| inch | rpm | D (inch) sfm | $rpm = \frac{sfm \times 12"}{D \times \pi}$ |
| metric | m/min | D (mm) rpm | $m/min = \frac{\pi \times D}{1000} \times rpm$ |
| metric | rpm | D (mm) m/min | $rpm = \frac{m/min \times 1000}{D \times \pi}$ |

**maximum cutting speeds**

Maximum cutting speed is often limited by the maximum travel speed (ipm or mm/min) of the tool allowed by the machine. Check your maximum speed with the following formulas.

inch formula: maximum cutting speed (sfm) = part diameter (inch) x 3.14 x tpi x $\frac{max\ ipm\ (inch/min.)}{12"}$

metric formula: maximum cutting speed (m/min.) = part diameter (mm) x 3.14 x (1/pitch) x $\frac{max\ mm/min.}{1000\ mm}$

legend
sfm    = surface feet per minute
m/min = meters per minute          D = part diameter
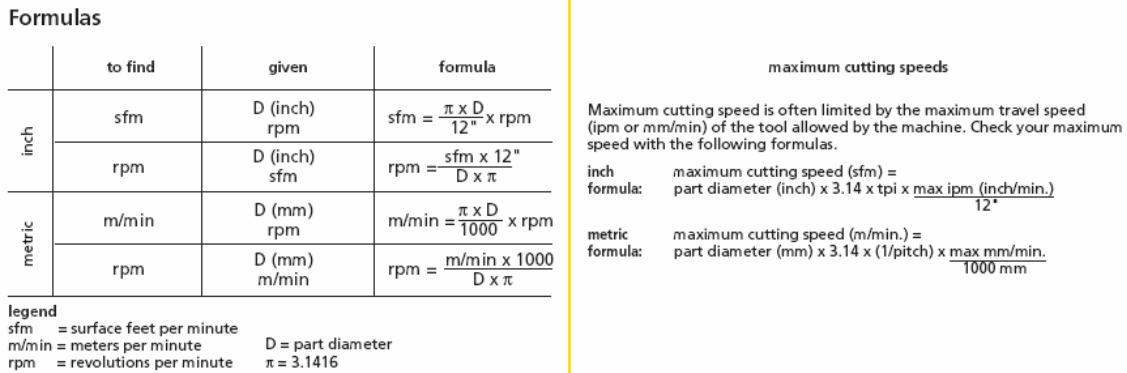rpm   = revolutions per minute      π = 3.1416

**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.
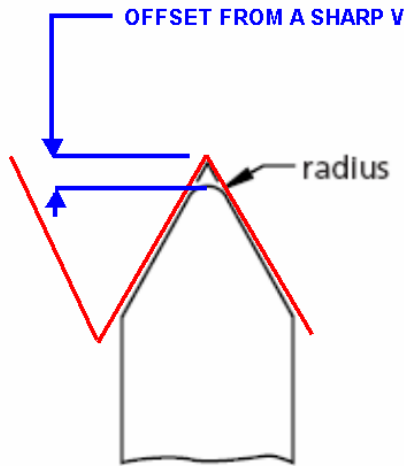


**FIGURE 5.5**

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-radiused 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.
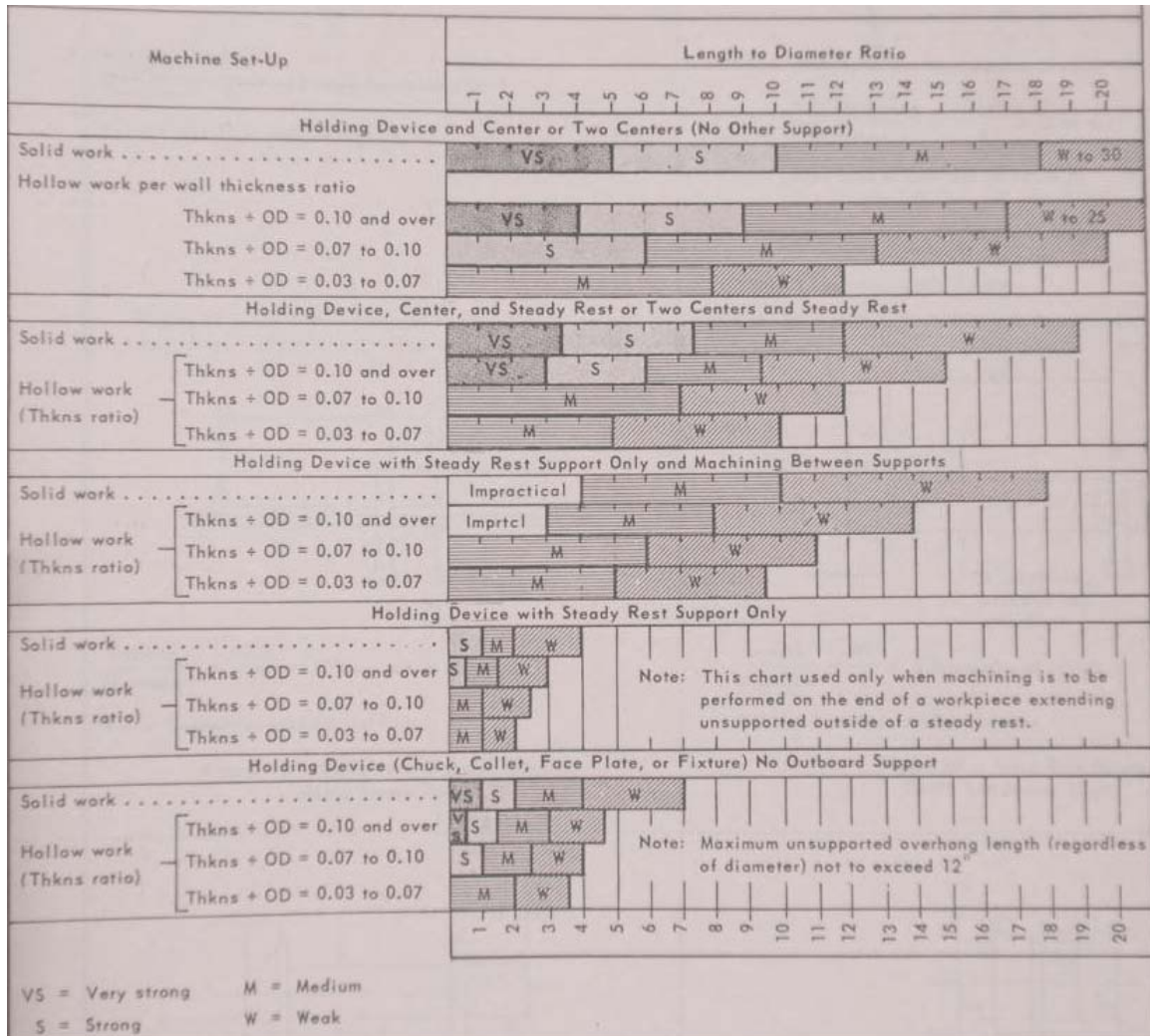


**FIGURE 5.6.1**

## 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 »  GO DOWN                                    MARK AS READ

| | | 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 | 5869 |

**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.

Here is a sample text file for a ¼-20 UNC ,1" long thread using the G76 canned cycle.

G18 G20 G40 G49 G80 G90 G94
 (Total_Depth =0.0866 minor dia=0.1634 )
(StartZ = 0.1 End Z=-1)
 (Dia Mode )
G0 X0.350  Z0.3
G0 X0.25
G76 X0.1634 Z-1.0 Q1 P0.050 H0.005 I30 K0.1 L360 C0.1 B0.0002  J0.002

## 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.

### 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.
These default values are, however, the ones that will be used in a given word is omitted in a
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
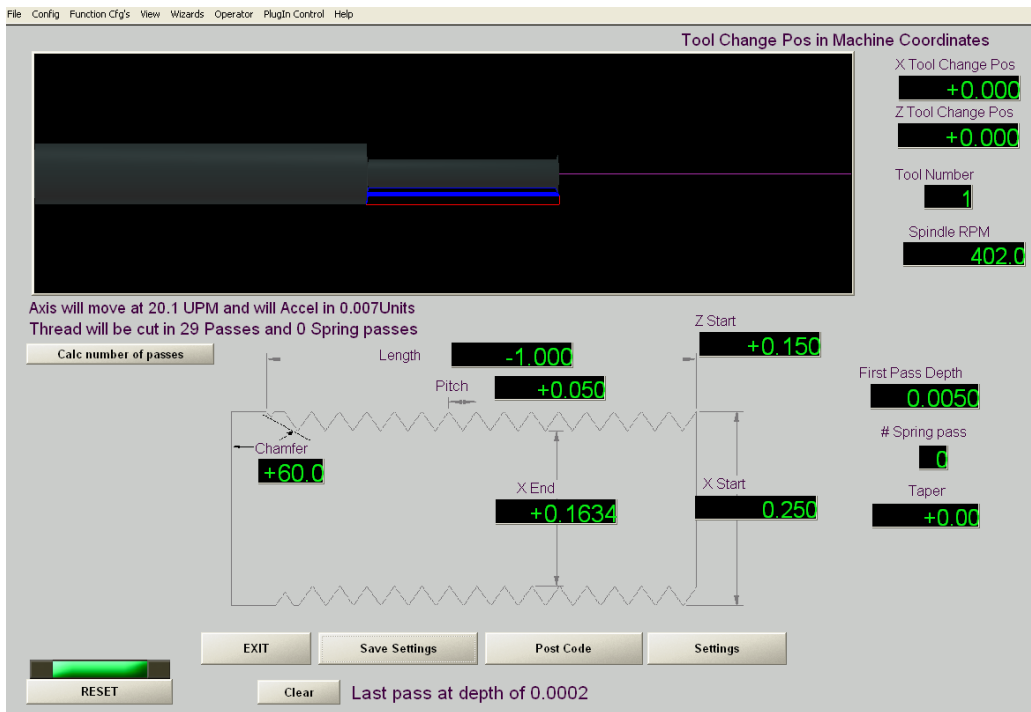


**FIGURE 6.4.1**

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. The following topic on the forum talks about the L setting in depth.
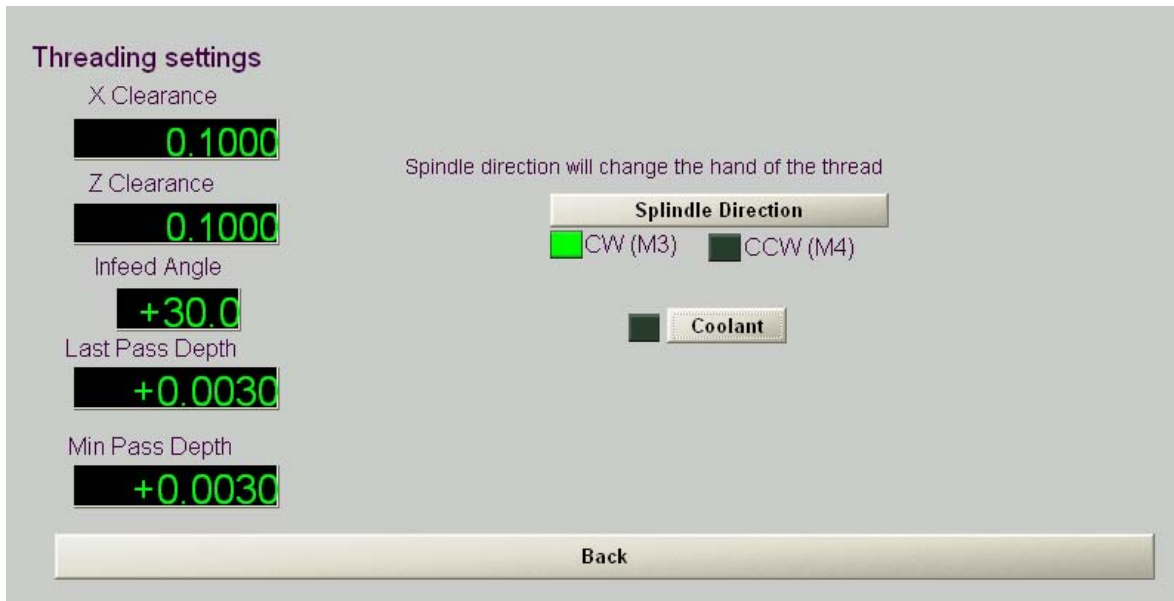
## FIGURE 6.4.2

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.)

   30 – will provide flank cutting

   29 / 29.5-  typically used for a unified or ISO metric thread and will modify the flank cutting
       providing a clearance on back cutting edge to the back   flank.  15 degrees even
       increases the change of X &Z ratio

   27 degrees for 55 degree angle forms like BSW or BSF

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 se the programs noted in See Section 6.6 to see the effects
   of different input values and read the WW in that section.

Note that G32 Z… F…. WITHOUT THE X WOULD GENERATE CODE WITH NO X
VALUE THUS CAN BE SKIPPED SEE PICKIMG UP A THREAD SECTION 8

**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.
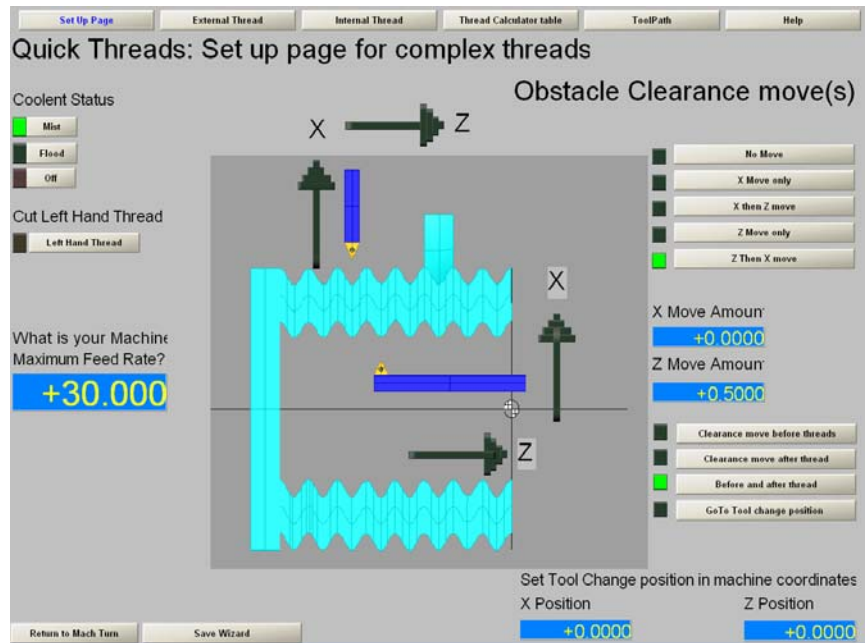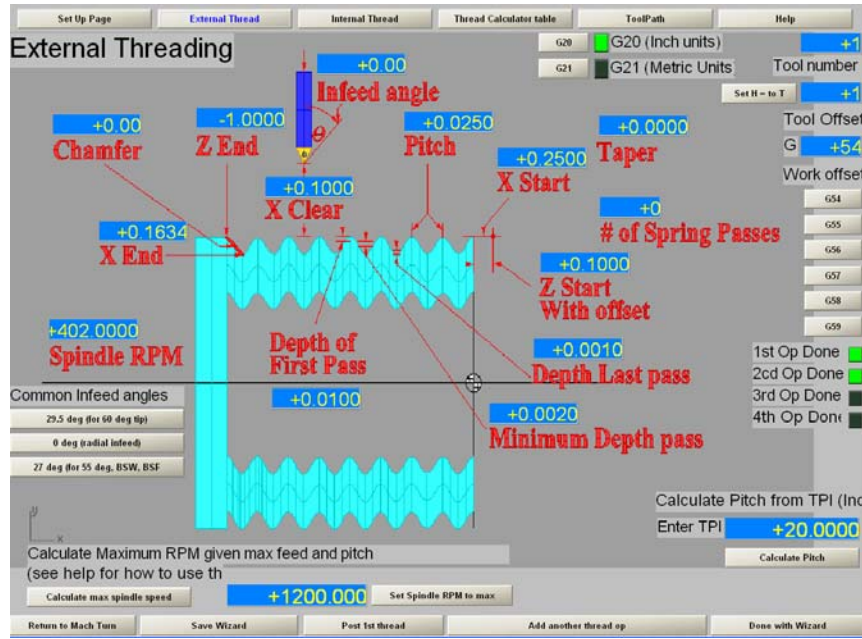


**FIGURE 6.5.2**

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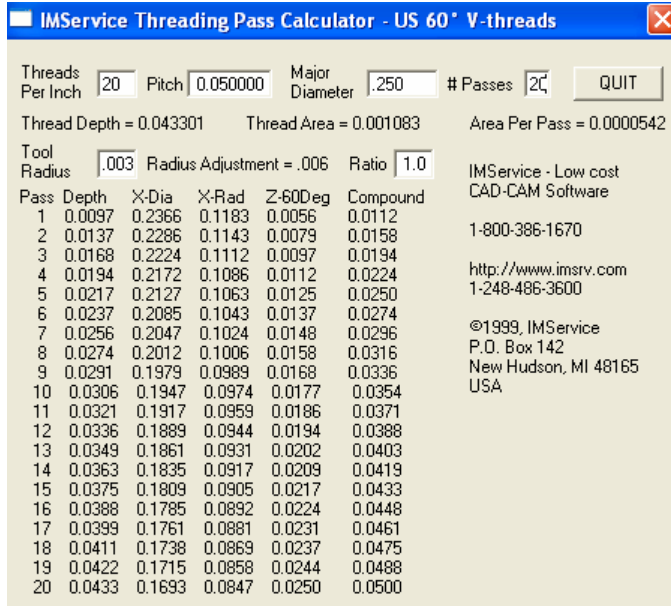Here is the link: http://www.vectorcam.com/thread.html



**IMService Threading Pass Calculator - US 60° V-threads**

Threads Per Inch 20    Pitch 0.050000    Major Diameter .250    # Passes 20    QUIT

Thread Depth = 0.043301        Thread Area = 0.001083        Area Per Pass = 0.0000542

Tool Radius .003    Radius Adjustment = .006    Ratio 1.0

IMService - Low cost CAD-CAM Software

1-800-386-1670

http://www.imsrv.com
1-248-486-3600

©1999, IMService
P.O. Box 142
New Hudson, MI 48165
USA

| Pass | Depth | X-Dia | X-Rad | Z-60Deg | Compound |
|---|---|---|---|---|---|
| 1 | 0.0097 | 0.2366 | 0.1183 | 0.0056 | 0.0112 |
| 2 | 0.0137 | 0.2286 | 0.1143 | 0.0079 | 0.0158 |
| 3 | 0.0168 | 0.2224 | 0.1112 | 0.0097 | 0.0194 |
| 4 | 0.0194 | 0.2172 | 0.1086 | 0.0112 | 0.0224 |
| 5 | 0.0217 | 0.2127 | 0.1063 | 0.0125 | 0.0250 |
| 6 | 0.0237 | 0.2085 | 0.1043 | 0.0137 | 0.0274 |
| 7 | 0.0256 | 0.2047 | 0.1024 | 0.0148 | 0.0296 |
| 8 | 0.0274 | 0.2012 | 0.1006 | 0.0158 | 0.0316 |
| 9 | 0.0291 | 0.1979 | 0.0989 | 0.0168 | 0.0336 |
| 10 | 0.0306 | 0.1947 | 0.0974 | 0.0177 | 0.0354 |
| 11 | 0.0321 | 0.1917 | 0.0959 | 0.0186 | 0.0371 |
| 12 | 0.0336 | 0.1889 | 0.0944 | 0.0194 | 0.0388 |
| 13 | 0.0349 | 0.1861 | 0.0931 | 0.0202 | 0.0403 |
| 14 | 0.0363 | 0.1835 | 0.0917 | 0.0209 | 0.0419 |
| 15 | 0.0375 | 0.1809 | 0.0905 | 0.0217 | 0.0433 |
| 16 | 0.0388 | 0.1785 | 0.0892 | 0.0224 | 0.0448 |
| 17 | 0.0399 | 0.1761 | 0.0881 | 0.0231 | 0.0461 |
| 18 | 0.0411 | 0.1738 | 0.0869 | 0.0237 | 0.0475 |
| 19 | 0.0422 | 0.1715 | 0.0858 | 0.0244 | 0.0488 |
| 20 | 0.0433 | 0.1693 | 0.0847 | 0.0250 | 0.0500 |

**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. Figure 6.6.4 shows the incremental cutting.
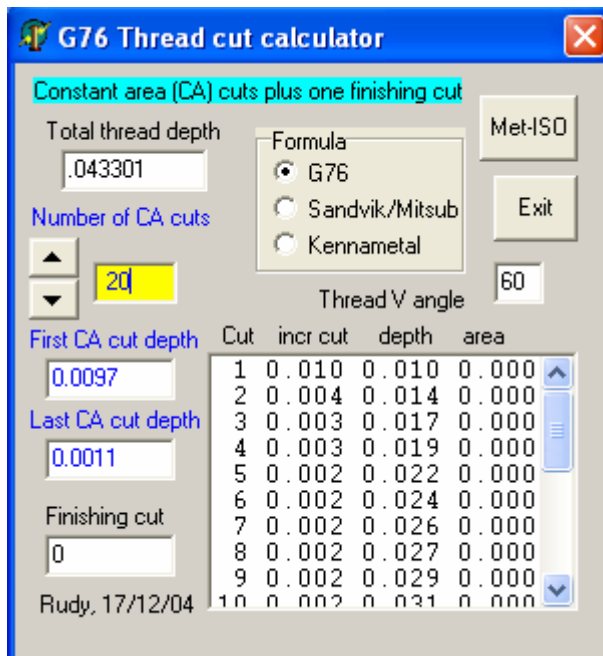


**G76 Thread cut calculator**

Constant area (CA) cuts plus one finishing cut

Total thread depth
.043301

Formula
● G76
○ Sandvik/Mitsub
○ Kennametal

Met-ISO

Exit

Number of CA cuts
▲
20
▼

Thread V angle 60

First CA cut depth
0.0097

Last CA cut depth
0.0011

Finishing cut
0

Rudy, 17/12/04

| Cut | incr cut | depth | area |
|---|---|---|---|
| 1 | 0.010 | 0.010 | 0.000 |
| 2 | 0.004 | 0.014 | 0.000 |
| 3 | 0.003 | 0.017 | 0.000 |
| 4 | 0.003 | 0.019 | 0.000 |
| 5 | 0.002 | 0.022 | 0.000 |
| 6 | 0.002 | 0.024 | 0.000 |
| 7 | 0.002 | 0.026 | 0.000 |
| 8 | 0.002 | 0.027 | 0.000 |
| 9 | 0.002 | 0.029 | 0.000 |
| 10 | 0.002 | 0.031 | 0.000 |

| | | | |
|---|---|---|---|
| 1 | 0.010 | 0.010 | 0.000 |
| 2 | 0.004 | 0.014 | 0.000 |
| 3 | 0.003 | 0.017 | 0.000 |
| 4 | 0.003 | 0.019 | 0.000 |
| 5 | 0.002 | 0.022 | 0.000 |
| 6 | 0.002 | 0.024 | 0.000 |
| 7 | 0.002 | 0.026 | 0.000 |
| 8 | 0.002 | 0.027 | 0.000 |
| 9 | 0.002 | 0.029 | 0.000 |
| 10 | 0.002 | 0.031 | 0.000 |
| 11 | 0.001 | 0.032 | 0.000 |
| 12 | 0.001 | 0.034 | 0.000 |
| 13 | 0.001 | 0.035 | 0.000 |
| 14 | 0.001 | 0.036 | 0.000 |
| 15 | 0.001 | 0.037 | 0.000 |
| 16 | 0.001 | 0.039 | 0.000 |
| 17 | 0.001 | 0.040 | 0.000 |
| 18 | 0.001 | 0.041 | 0.000 |
| 19 | 0.001 | 0.042 | 0.000 |
| 20 | 0.001 | 0.043 | 0.000 |

**FIGURE 6.6.3**                    **FIGURE 6.6.4**

Note that cutting will vary some depending which of the three formulas are used.

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 threading 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 30.042.031 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 in 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: 3.042.031

## 7.1 CONFIGURATION
**I HAVE NOT EVEN LOADED VER .030 YET SO DON'T KNOW IF THERE WERE ANY SCREEN CHANGES  BUT THE FOLLOWING STILL APPPLY AND WILL UPDATE IN THE NEXT POSTING .**
The current version is for index only. The index must be enabled and port & pin assigned to do threading.
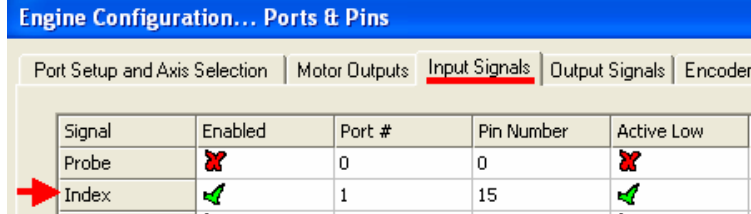


The timing function is disabled in version 3.042.20 and will not function.
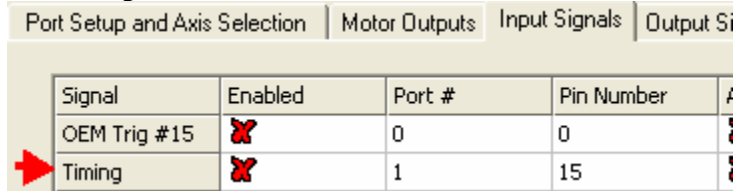


**FIGURE 7.1.1**

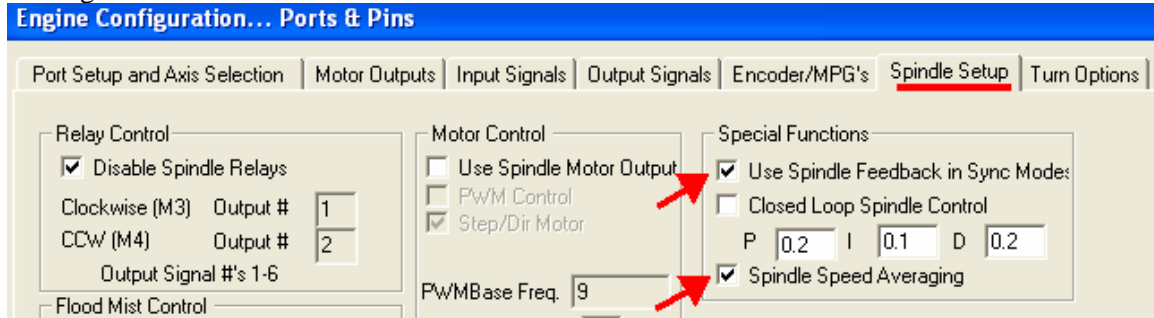Spindle feedback should be checked. Spindle speed averaging will average the rpm over 8 readings and should be checked.



**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 threading works in both. The Turn cycle defaults provide default values when not specified / missing in G76.

Note the following settings: **LATER**
Cut Type: Specifies type of threading method
0  is for Flank cutting
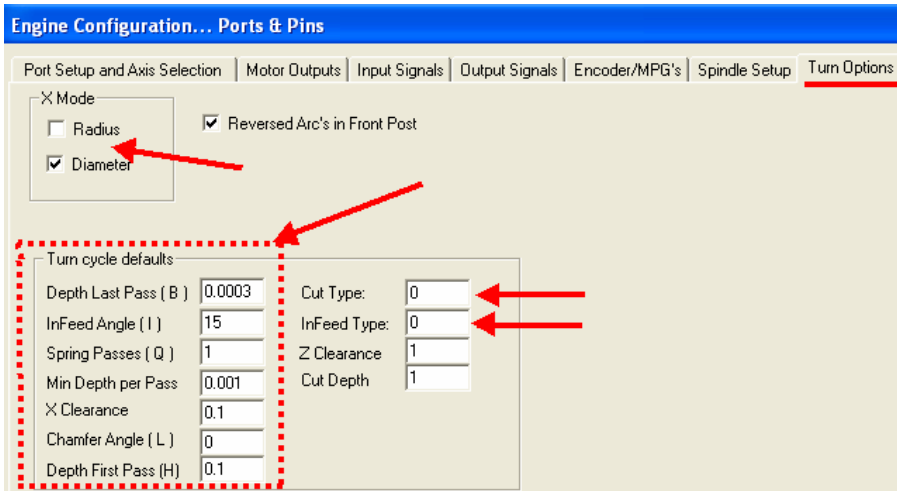1 is for Alternate Flank cutting
InFeed Type:
0  is for
? is for

**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 to avoid problems. 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.
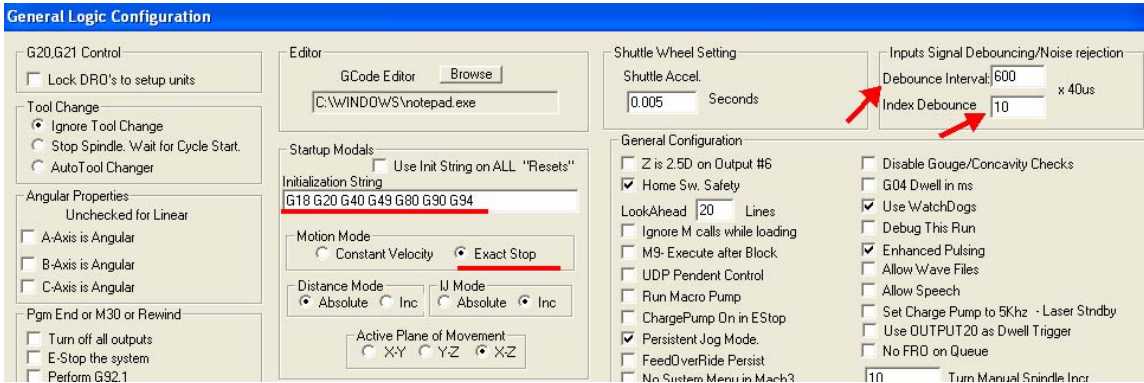


**FIGURE 7.1.4**

**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!  LATER**



**FIGURE 7.1.5**

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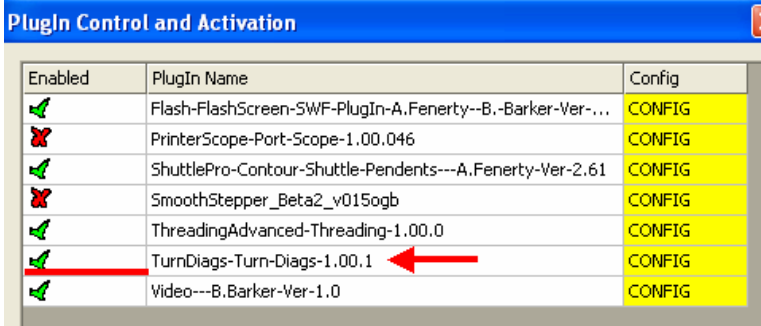


**FIGURE 7.1.5**

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.
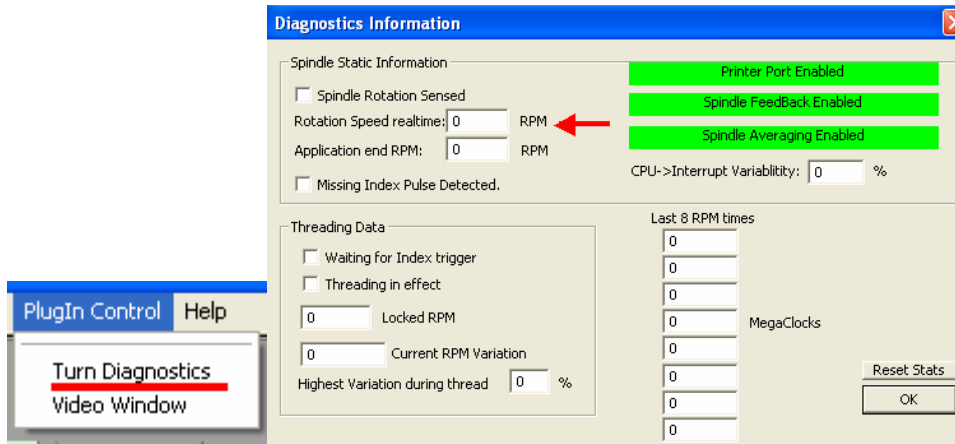


**FIGURE 7.1.6**

## 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.
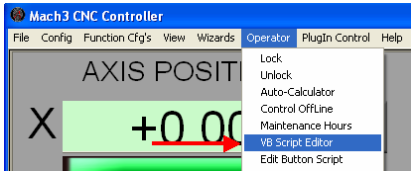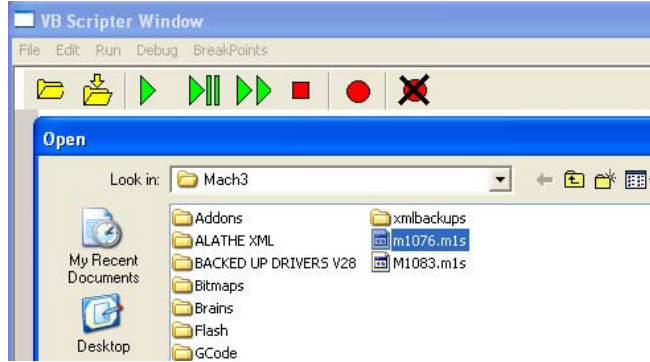

**FIGURE 7.2.1**


**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.
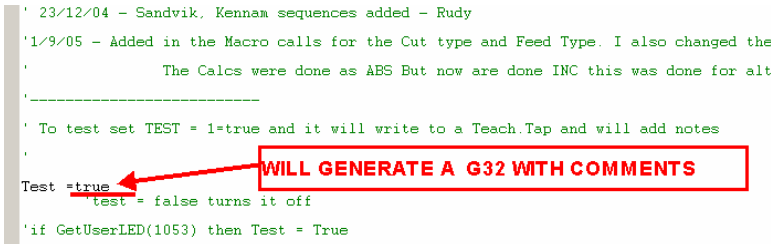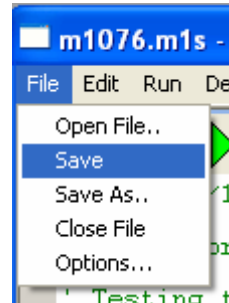

**FIGURE 7.2.3**


**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.

Thus, when you cut a (multi start) thread what you have to consider is the LEAD and NOT the PITCH. So if you want to cut a 1mm pitch double start thread, you have to program a feed of 2mm/Rev for each of the starts. Obviously, the two starts are 1mm apart, so after cutting the first lead you have to move the tool 1mm in Z and start the move from there for the second lead. Note that the depth of thread has to be the same as if you were cutting a single start thread with the same pitch (which is 1mm for example).

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.
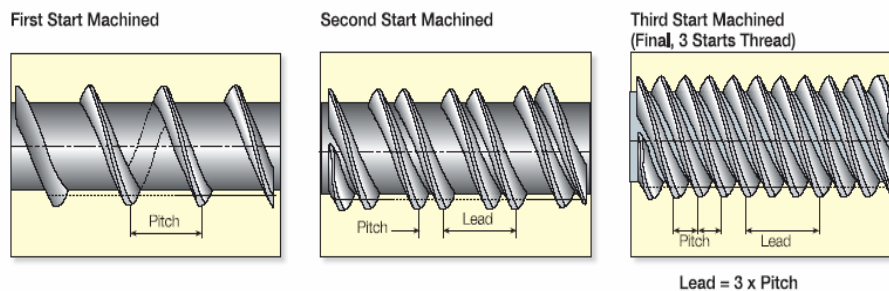


**FIGURE 8.1**

Multi start threads are produced by adding an offset ( ½,1/3,1/4 etc of pitch ) to the Z starting position.
-You can just past together a number of Gcode files generated by the Simple Threading wizard ( See section 6)along with minor editing.

-You can also use the Quick Threads Wizard which allows for posting multiple threads but make sure you provide for an offset of the Z,, thus no need to edit the text??

- You can use the G76 canned cycle as follows:

Put in #99066 = start angle for the thread in the code, so if you want to do 3 start thread then
#99066 = 0
threading code
#99066 = 120
same threading code
#99066 = 240
same threading code

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

G76 X*** Z*** Q* P* etc , you would put it before each thread start.


#99066 = 0
G76 X*** Z*** Q* P* etc
#99066 = 120
G76 X*** Z*** Q* P* etc
#99066 = 240
G76 X*** Z*** Q* P* etc

## EXAMPLES OF MLTI START THREAD CODES
http://www.machsupport.com/forum/index.php/topic,8314.msg68974.html#msg68974
----------------
pic A -This will cut the thread three individual times but a single thread.
#99066=0
G76 X0.33 Z-0.625 Q1 P0.08865 J0.004 L0 H0.005 I30 C0.01 B0.0002 T0
#99066=120
G76 X0.33 Z-0.625 Q1 P0.0591 J0.004 L0 H0.005 I30 C0.01 B0.0002 T0
#99066=240
G76 X0.33 Z-0.625 Q1 P0.02955 J0.004 L0 H0.005 I30 C0.01 B0.0002 T0


-----------------------------
pic B - This will cut the thread  three times but a single thread.Note that it
looks like a Acme thread.
#99066=0
G76 X0.33 Z-0.5 Q1 P0.08865 J0.004 L0 H0.005 I30 C0.01 B0.0002 T0
#99066=120
G76 X0.33 Z-0.5 Q1 P0.08865 J0.004 L0 H0.005 I30 C0.01 B0.0002 T0
#99066=240
G76 X0.33 Z-0.5 Q1 P0.08865 J0.004 L0 H0.005 I30 C0.01 B0.0002 T0


---------------------------
pic C - this cuts a three pass, but looking at the end of the thread you can see that the
start point is not even spaced, thus the third pass lead is out.
G76 X0.33 Z-0.625 Q1 P0.08865 J0.004 L0 H0.005 I30 C0.01 B0.0002 T0
G00 Z-.0296
G76 X0.33 Z-0.655 Q1 P0.08865 J0.004 L0 H0.005 I30 C0.01 B0.0002 T0
G00 Z-.0592
G76 X0.33 Z-0.685 Q1 P0.08865 J0.004 L0 H0.005 I30 C0.01 B0.0002 T0


-------------------------
pic D - same as pic c but start points are out ( just a refinement of the numbers in the code)

G76 X0.33 Z-0.600 Q1 P0.08865 J0.004 L0 H0.005 I30 C0.01 B0.0002 T0
G00 Z-.02955
G76 X0.33 Z-0.62955 Q1 P0.08865 J0.004 L0 H0.005 I30 C0.01 B0.0002 T0
G00 Z-.0591
G76 X0.33 Z-0.6591 Q1 P0.08865 J0.004 L0 H0.005 I30 C0.01 B0.0002 T0


-------------------------
pic F - this is a two start thread using just a Z offset and it turns out fime . The start points
are 180 degrees out as expected.
G76 X0.33 Z-0.600 Q1 P0.0591 J0.004 L0 H0.005 I30 C0.01 B0.0002 T0
G00 Z.02955
G76 X0.33 Z-0.62955 Q1 P0.0591 J0.004 L0 H0.005 I30 C0.01 B0.0002 T0


-------------------------
pic G - This is a three start thread, and the third pass screws up the thread.

G00 Z.0591
G76 X0.33 Z-0.600 Q1 P0.08865 J0.004 L0 H0.005 I30 C0.01 B0.0002 T0
G00 Z.02955
G76 X0.33 Z-0.600 Q1 P0.08865 J0.004 L0 H0.005 I30 C0.01 B0.0002 T0
G00 Z0.0
G76 X0.33 Z-0.600 Q1 P0.08865 J0.004 L0 H0.005 I30 C0.01 B0.0002 T

## 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 carefull 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, "tweek" 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; 3x.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 spindel 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 equivilant of picking up the thread on a manual lathe with an indexer dial. This dosn't take 5 minutes, and if you know what your doing it is rather quick to do. Simply put, 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 Toolroom 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
MACHINERYS HANDBOOK
CNC PROGRAMING TECHNIQUES – PETER SMID
SME – SOCIETY of MANUFACTURING ENGINEERS
USING MACH 3 TURN MANUAL
ARTSOFT MACH3 FORUM

# 11.0 APPENDIX LIST

**A –  LATHE SPECIFICATION , TESTING & TOLERANCES**
**B –  JIS  STANDARD TOLERANCES**

## APPENDIX "A"   LATHE SPECIFICATION, TESTING  & ACCURACES / TOLERANCES

Table II is lathe specification to a manufacturer.

TABLE II -- ACCURACIES

| Requirements | Tolerance |
| --- | --- |
| Bed (Verification of Leveling of Slideways) | |
| 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 Bar Diameter the Same as the Spindle Hole | 0.0012" |
| 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 Bar Diameter the Same as the Spindle Hole | 0.0012" |
| 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.
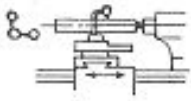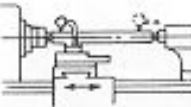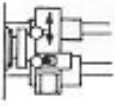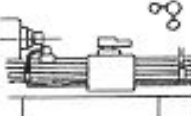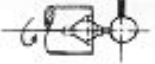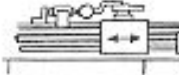
**Threading Test**. Provide two 1/2 inch bars of low carbon steel (1020) of sufficient length so no less than 4 inches of threads can be cut. Set the lathe to cut 1/2-13UNC threads over a length of at least 4 inches. Repeat cutting 1/2-32UNF threads on the second bar. Failure of the screw threads to 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 alignment in vertical plane | | 0.02   per 150 high at free end | |
| | Tail-stock spindle alignment 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 | Pich 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.