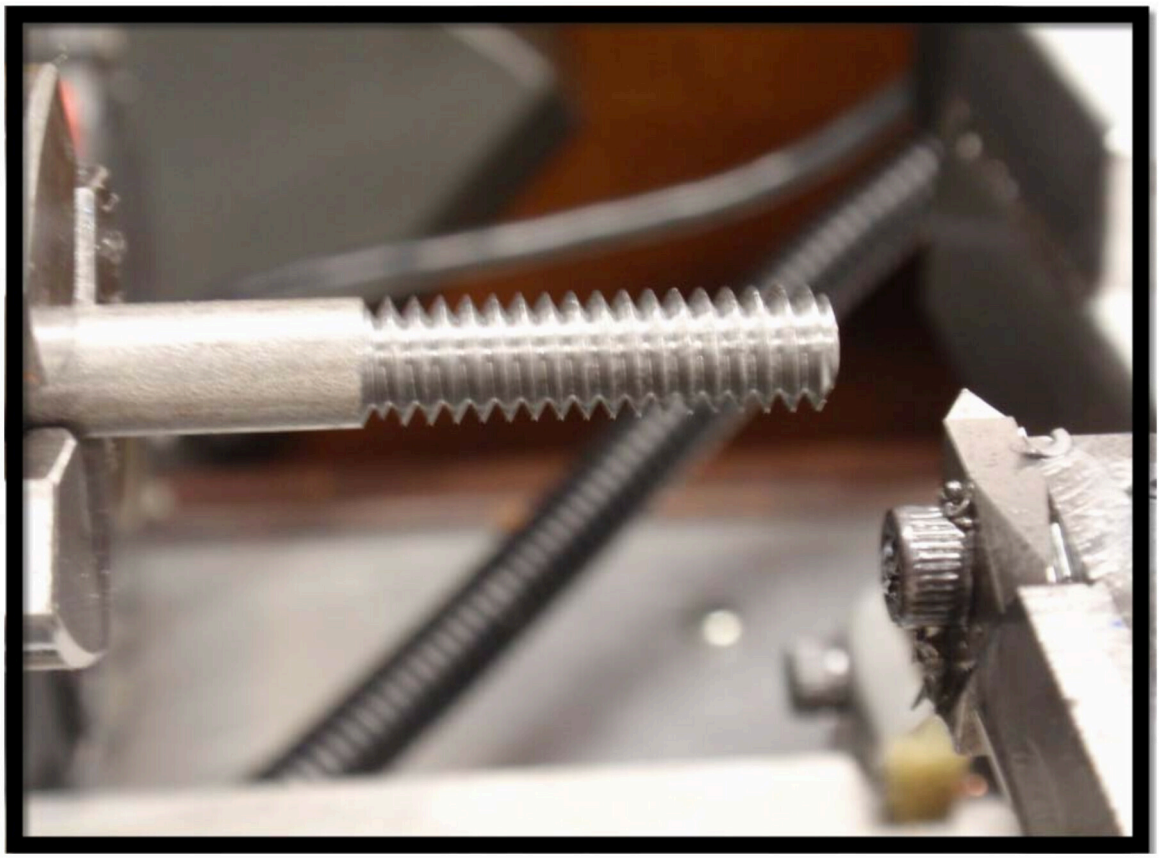
THREADING ON THE LATHE-MACH3 TURN



5.0 THREAD CUTTING

5.1 THREAD CUTTING FEED METHODS & FORMULAS

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

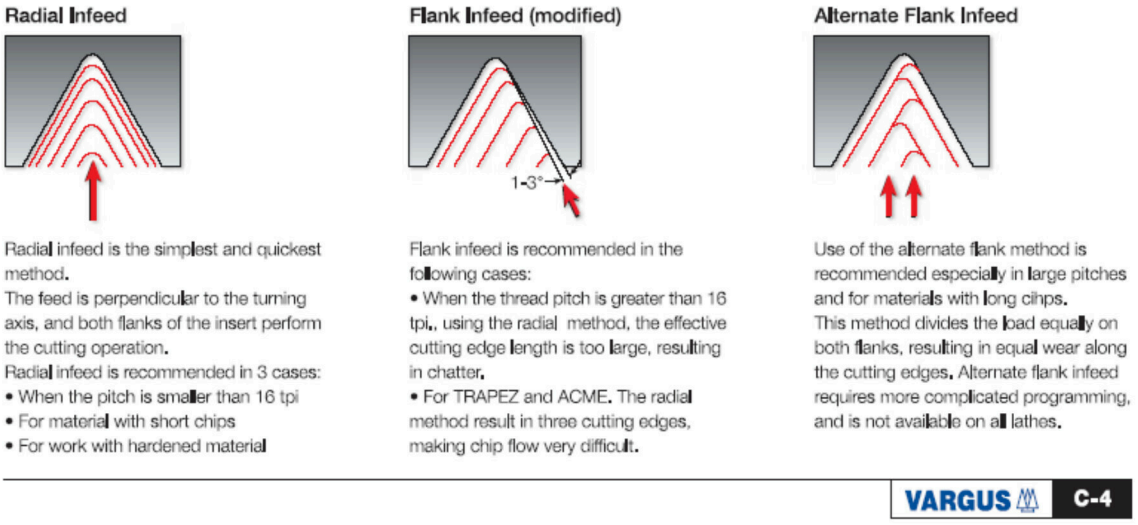


FIGURE 5.1

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

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

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

WW:

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

7.0 MACH3 TURN CONFIGURATION

Mach version 30.042.032 implemented a completely new threading code which fixed all earlier versions.
Mach version 30.042.033 provided for CPU's with a fast bus speed > 2.1 GHz to eliminate a (-) spindle speed in the DRO
Mach version 30.043.037 is recommended if using MSM and it is available from the CVI web site download area at *WWW.CalyopsoVentures .com*

The writer recommends the user upgrade to a Mach Version greater than 3.042.034 or the current lockdown version, but use 30.043.037 if using MSM. Should the user have a threading problem post on the Artsoft Forum.

NOTE the following:

- You can only use a **SINGLE INDEX** and **timing** is not provided

WW

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). The updated version of threading changed the current 'timing" configuration such that a multi slotted disc with a wider index slot used for timing will no longer be available.

- Diagnostics for threading is now outdated / do not use it / and should not be enabled.

WW

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

- Do not use CSS when threading as it can screw up the thread lead

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.

WW

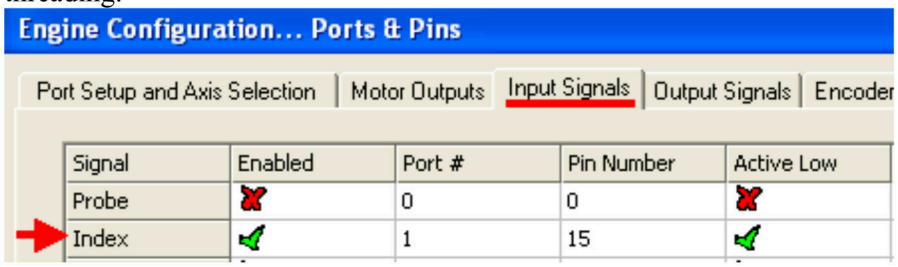
The "future" Advanced Threading requires update per the new threading code and will allow for a slotted wheel with index or an encoder. When using an encoder additional parameters will need to be defined to MACH. If using an encoder, you'll have to set the number of quads per rev and should work fine 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 went through to fix threading.

7.1 CONFIGURATION

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



The timing function is disabled in versions greater than 3.042.032 and will not function. DO NOT Enable it!

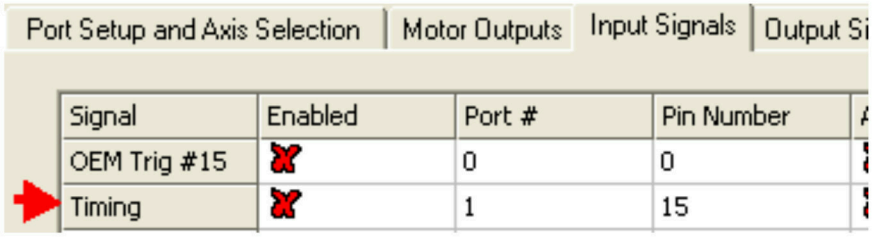


FIGURE 7.1.1

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

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

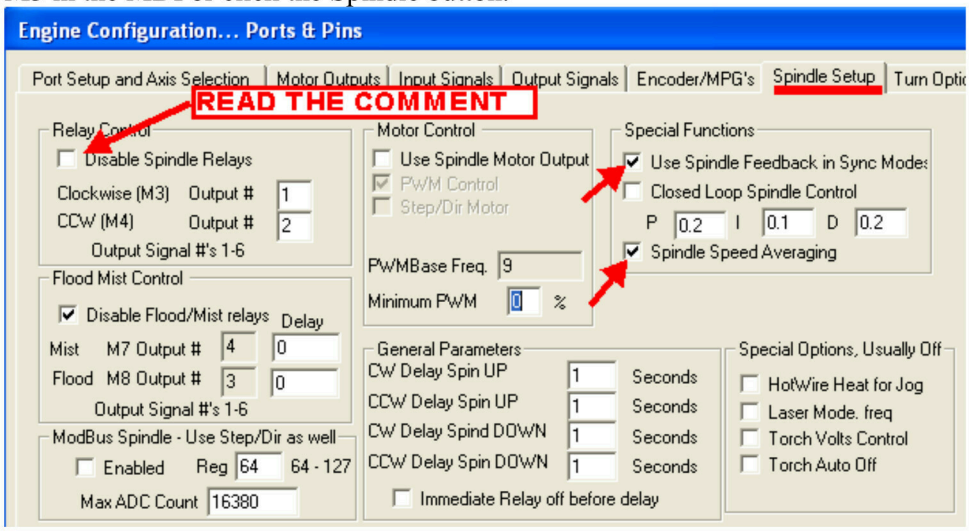


FIGURE 7.1.2

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

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

Cut Type: defines type of threading method
0 - Flank cutting (default)
1 - Alternate Flank cutting
2 – Back Flank cutting

InFeed Type: defines how the passes are calculated
0 – Constant volume threading (default)
1 – Cut the thread in X number of passes (set in the wizard DRO 1022)

NOTE: THE ABOVE SETTINGS WILL DEFAULT BACK TO "0" WHEN YOU EXIT MACH3 TURN.
The remainder of the settings shown in the dotted square provide default values for G76 if it is missing.

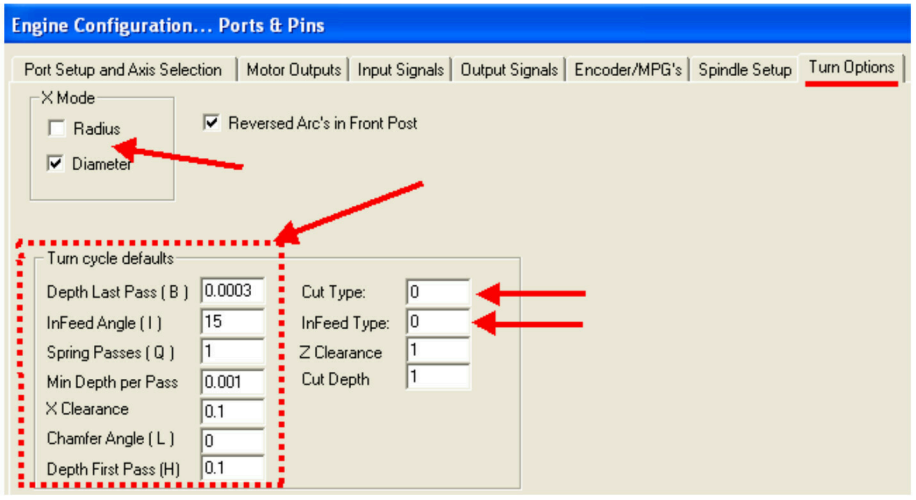


FIGURE 7.1.3

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

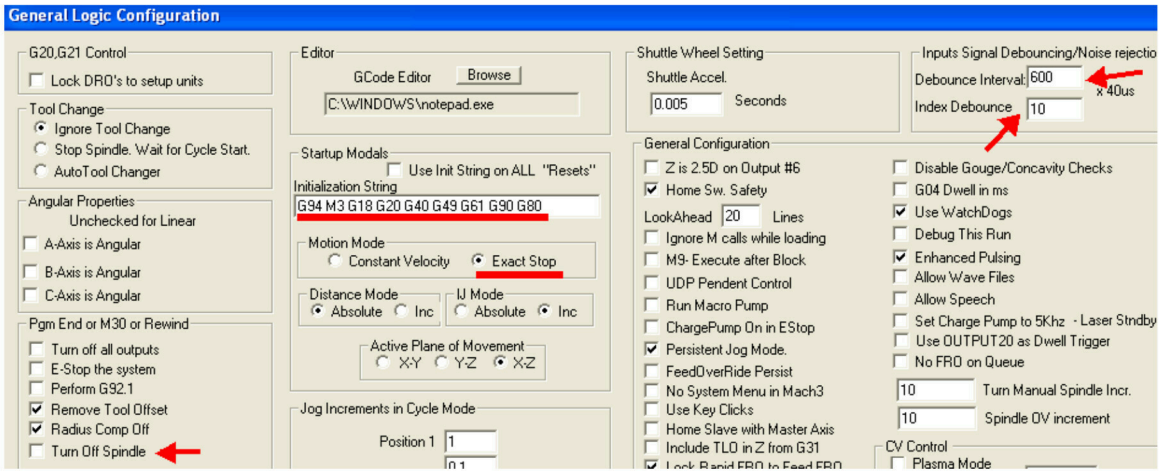


FIGURE 7.1.4

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

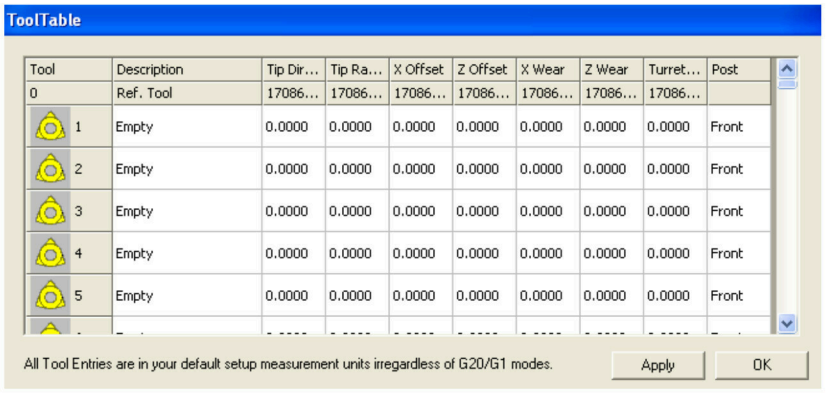


FIGURE 7.1.5

APPENDIX "D" – REVISIONS

REV – PAGE – DESCRIPTION

REV 1- 2/22/2010 - MUST TURN ON SPINDLE TO GET RPM READING
- TURN DIAGNOSTICS NO LONGER FUNCTIONAL / UN-ENABLE PLUG IN
- PAGE 19,42,43,45,46 REVISED TO REFLECT ABOVE AND REVISION LIST ADDED

REV 2- 1/21/2012 GENERAL REVIEW AND UPDATE AS FOLLOWS:
PG 2 – ADDED **APPENDIX D-** REVISION LIST
PG 3 – ADDED NOTE **NOTE:** The user must have a Mach3 license to do actual threading on lathe. Threading is disabled in the demo version. Pirated licenses of Mach will create problems in Mach Turn.
PG18 - DELETED USE OF TURN DIAGNOSTICS PLUG IN

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

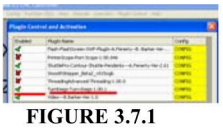


FIGURE 3.7.1

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

FIGURE 3.7.2

PG 22 – ADDED See Figure 4.2.1 extracted from the Machinery's Handbook
– ADDED If not accounted for,
PG 32 – REVISED FIGURE 6.1.0
PG 34 – ADDED The Z axis will slow down during pullout, but, at fast feedrates there must be enough time to accomplish the pullout.
PG 40 - DELETED reference to THREAD 22 (LINK NO LONGER AVAILABLE)
Another program is THREAD22 (Figure 6.6.2). The freeware threading program is a single point threading pass calculator. It provides number of passes based solely on a calculated cross-sectional area of material cut per pass. Here is the link: http://www.vectorcam.com/thread.html

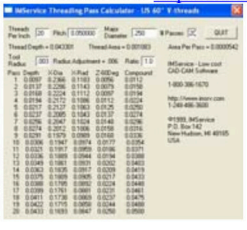


FIGURE 6.6.2

PG 42 – REVISED / ADDED WORDING , complete rewrite of page

PG 43 – DELETED FIGURE 7.1.5 AND ASSOCIATED WORDING

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

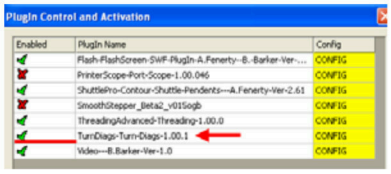


FIGURE 7.1.5

PG 44 – DELETED FIGURE 7.1.6 AND ASSOCIATED WORDING, NOW A BLANK PAGE

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.

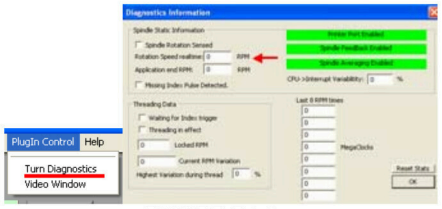


FIGURE 7.1.6