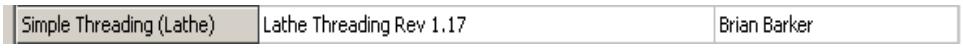
CUTTING A NTPF USING THE SIMPLE THREADING WIZARD

The table below provides input values for the wizard to cut an NPTF thread on the end of a shaft.

NTPF	LENGTH	Z START	PITCH	CHAMFER	X START	X END	TAPER
1/8 - 27	0.51830	0.01750	0.03210	1080.00000	0.38840	0.32440	3.57334
1/4 - 18	0.75030	0.02630	0.05555	1080.00000	0.51540	0.41918	3.57334
3/8 - 18			0.05555	1080.00000			
1/2 - 14			0.07143	1080.00000			
3/4 - 14			0.07143	1080.00000			

FIGURE 1

WIZARD SCREENS

Select 

This is the Wizard input screen with values (red underlined) from the table for a ¼-18 NPTF tapered thread. (All the following figures show values for a ¼-18 NPTF)

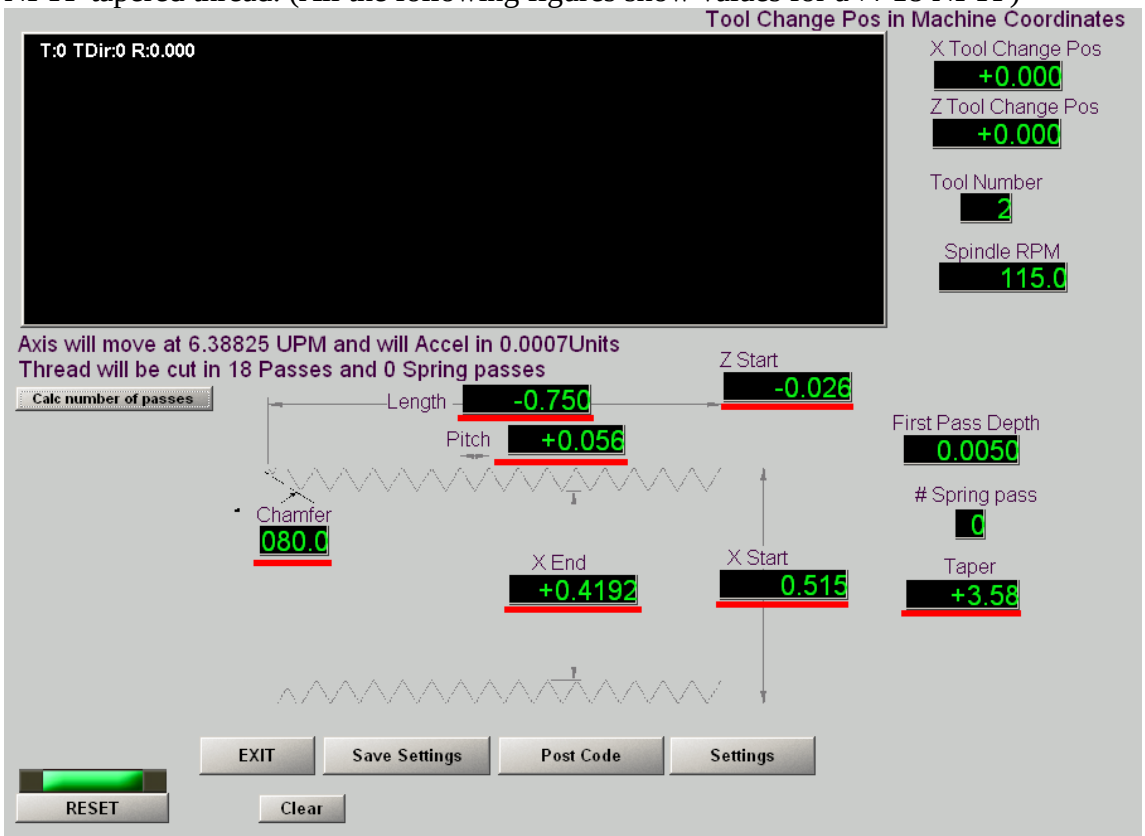


FIGURE 2

Clicking the settings button 

provides additional settings which are used when generating the gcode. In particular is the infeed angle. 30 degrees provides for radial cutting and increasing to 31.8 degrees will provide for flank cutting. The other inputs are users choice.

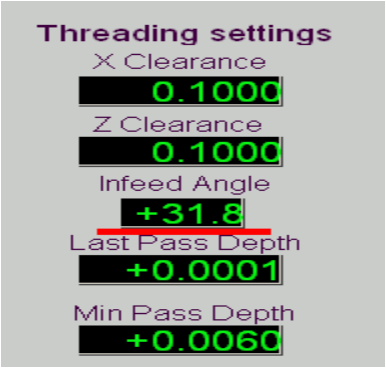


FIGURE 3

INPUT DEFINITIONS:

TAPER

This is the "total" taper on the end of the pipe. The actual input value was 3.576334 deg per code but after entering the angle Mach will round out the angle in the screen, but, will use what was entered for the gcode.

Z START

The intent of the provided table is to cut a thread such that when one hand tightens the joint, it will take 4.1 turns +- 1 turn from basic when removed, makes full flank contact with a crest at L1 length. The crest of the V is located based on E1 location with the pitch line running through the pitch diameter for E1 as shown in Figure 4. Note that L1 is also the gauging line for an L1 gauge. Thus the user will need to setup the lathe properly to achieve the cut. If cut properly, then when the joint is made up, make up distance will equal L1 length for a rather closely known assembly distance.

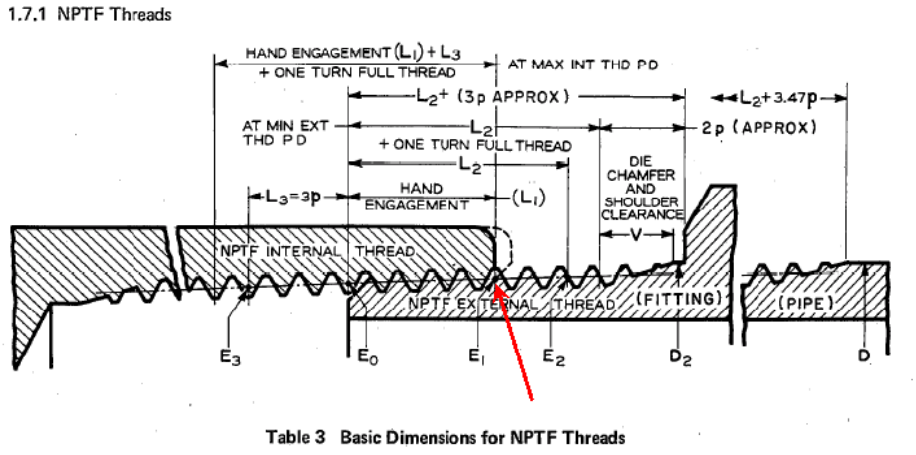


Table 3 Basic Dimensions for NPTF Threads

FIGURE 4

If no offset value is input in the wizard it will generate code based on the root of the V as shown below. If a negative value is used, the wizard will offset where the tool tip / controlled point will start the thread. Standard practice in cutting a thread is to provide 3 to 5 x pitch length for axis acceleration and spindle synchronization. The input value was - 0.02630 and again Mach rounds the value but uses the input value for the Gcode. Note actual cut starts at approx 3x pitch away from the end of the prepared piece.

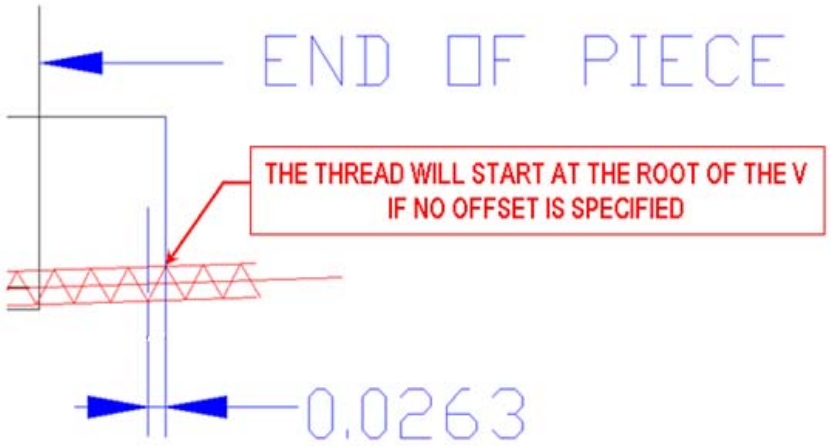


FIGURE 5

X START & X END

XStart:

These two settings are not straight forward. The start of the thread is related to E1 location as shown in the pathing plot of Figure 6. Basically the pitch diameter and threads are projected beyond the end of the pipe and the values determined such that contact at the pitch line at E1 is accomplished. The depth of thread is 0.866 x pitch. Note that you will be air cutting and doing so ensures time for Mach to adjust velocity and acceleration properly compared to spindle rpm as noted before. That sets the X Start & X End diameters. Note that crests should be truncated properly and what is shown does not address truncation. All depends on how you want to do the machining, ie; do the truncation machining before or after the thread is cut. Also note that the threading tool is set perpendicular to the center of the pipe.

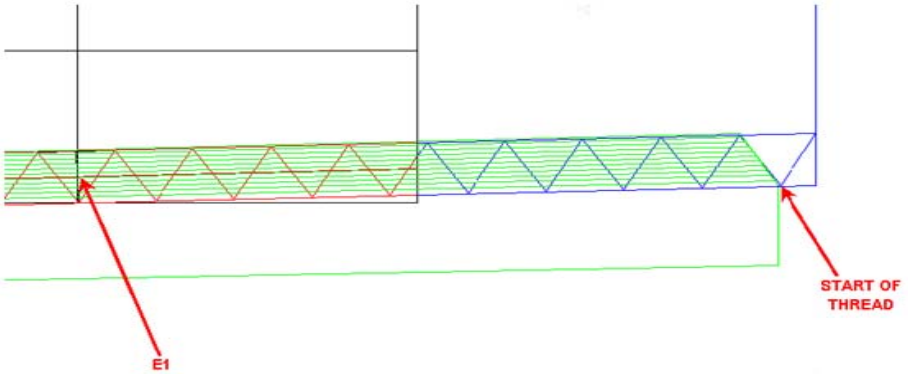


FIGURE 6

PITCH
Pitch is 1/TPI. You should use 5 decimal places for the input. Again, Mach will round out the input but uses the pitch as input. Mach uses the pitch to set the axis feed rate. I must note the more precise the spindle rpm the better the thread lead will be controlled. For NPTF (Dryseal pipe threads) the flanks of the thread will mate and additionally the crests and roots will compress to provide the seal. Thus the NTPF is a more accurately produced thread as compared to the rather loose NPT thread.

CHAMFER
Chamfer is the pull out of the tool at the end of the thread cycle. It is specified in ***degree's.*** At high feed rates it unlikely that you can pull out in say 90 degrees. Additionally the user can produce tapering at the end of the thread. The start of the chamfer is related to how many additional threads you want along the taper. The code provides basic guidelines and the end of the thread can mimic threads cut with a die. Two to three threads should be provided for (joint makeup can be +1 turn).

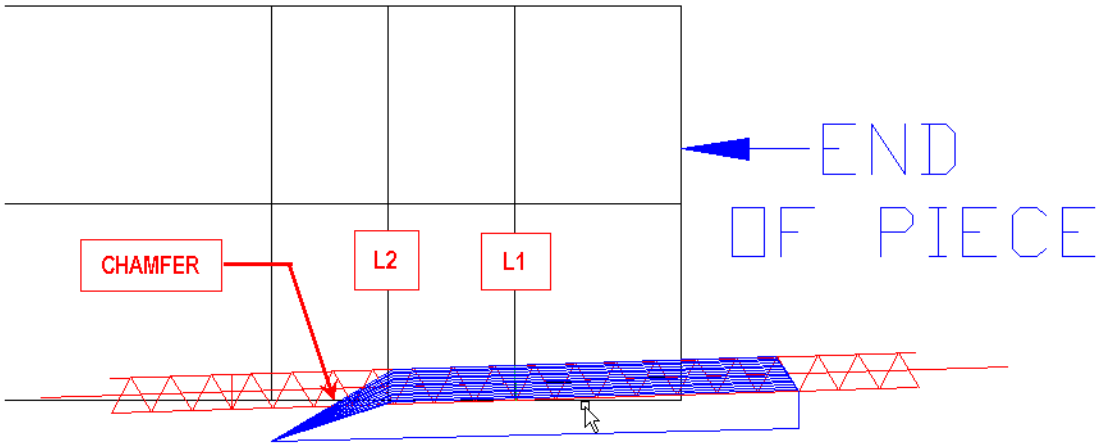


FIGURE 7

LENGTH
The length is the distance from Z=0 to the end of the threading cutting cycle which includes the chamfer. If the length is a negative value the threading will be outward towards the outside diameter of the pipe as shown below. If a positive value is used the threading will be done inwards towards the bore of the pipe. The length is considered parallel to the center line of the pipe.

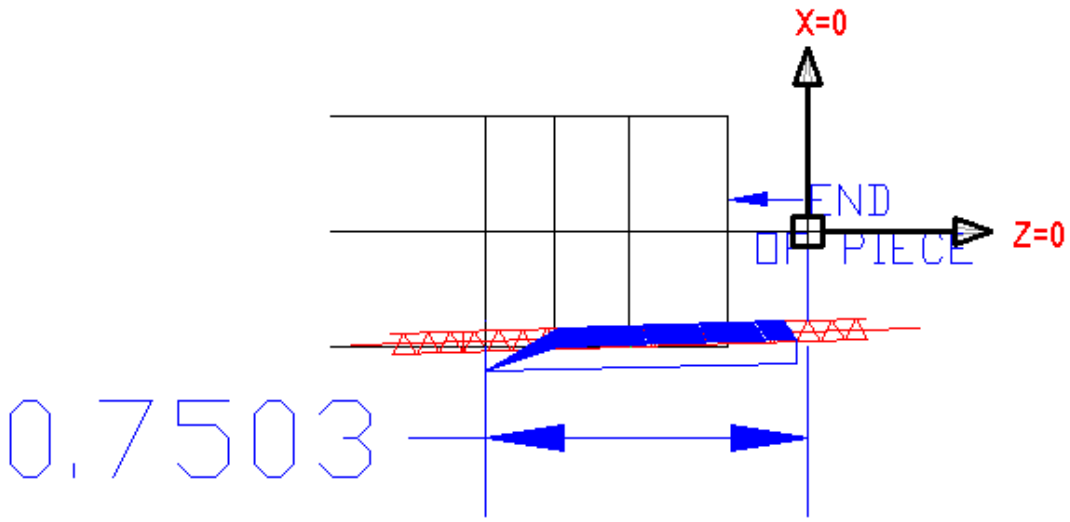


FIGURE 8