

Acorn - Rigid Tapping Setup for Mill and Lathe

Overview

Rigid Tapping setup and test procedure to control accuracy of depth of cut and quality of threads in various working materials. There is no set in stone specification values for over-travel or under-travel associated with rigid tapping because each machine has different inertia properties for the spindle/motor system and different VFD settings for braking and acceleration. Therefore setting up rigid tapping must be considered an "art" and requires some experimentation to tweak-in the both the VFD and CNC12 Rigid tapping parameters for good results. Since the Rigid Tap feature works only as good as the integrator "tunes" it, Centroid makes no warranty or guarantee regarding rigid tapping, that being said, once it is tuned properly it works quite well.

Rigid Tapping requires feedback from a spindle encoder, this encoder must be connected with the spindle at a 1:1 ratio. This means for each revolution of the spindle the encoder must also rotate exactly one revolution in sync with the spindle. (The spindle encoder is also used to display the actual spindle speed rather than the commanded spindle speed).

Step 1.) Use the Centroid CNC Control Configuration Wizard to configure the CNC12 software for use with a spindle encoder. The Wizard turns on the spindle encoder and sets the spindle encoder counts per revolution and direction of rotation. Whether you are setting up for a mill or lathe start the Centroid Setup Wizard and go to tab #3 "Spindle Setup". Toggle "Spindle Encoder" to "Yes". Then scroll down to Spindle Encoder Counts and enter the encoder counts of the encoder connected. (common values are: 8000 (a 2000 line encoder), 8192 (a 2048 line encoder) Here is a video showing how to setup a spindle encoder with the Wizard. https://youtu.be/Gk1T_9-7_4g

Step 2.) Determine if the encoder signals match the direction of the spindle. Start CNC12 and go to PID menu. F1 Setup, F3 Config (type in the password 137 press enter), F4 PID. Now the Rotate the spindle clockwise and observe the 5th axis 'Abs Pos' column are the encoder counts increasing in a positive value or negative value? If value is increasing in a positive value then do nothing and press ESC three time to return to the main menu. If the encoder count value is increasing in the negative direction then shut down CNC12, start the Wizard and edit the encoder value with a negative sign in front of the count. For instance if the count was 8000, change that to -8000. If the encoder counts up when the spindle turns CW then the encoder count per revolution value entered in the wizard should be positive. If the encoder counts up when the spindle turns CCW then the value should be negative.

Step 3.) Verify and Tune the VFD so that the commanded RPM matches the actual RPM.

1. Enter in the desired maximum spindle speed in the Wizard and write settings to CNC config.
2. Following the instructions in the inverter manual and set the inverter maximum frequency to 120hz.
3. Using the Virtual Control Panel switch to MANUAL SPINDLE MODE and turn the SPINDLE Speed OVERRIDE all the way down.
4. Turn the spindle on CW and make sure it is turning the correct direction. If it is turning the wrong direction correct this by turning off power and swapping 2 legs of the 3 phase cable between the inverter and the motor.
5. Repeat step 3 and 4. 6. Monitor the actual spindle speed on the screen or use a tachometer and slowly increase the SPINDLE OVERRIDE until the correct maximum spindle speed is reached. If the maximum speed cannot be reached turn the spindle off and restart at step 2 with a higher maximum frequency.
7. Write down the actual frequency that the inverter is outputting (from the control panel on the front inverter) and then turn off the spindle.
8. Then following the instructions in the inverter manual set the inverter maximum frequency to the frequency in step recorded in step 7.
9. Using the VCP turn on AUTO SPINDLE MODE; and with the Spindle speed OVERRIDE at 100%, in MDI turn on the spindle at 640 RPM (use this command "M3 S640") Now with the spindle running. Adjust the maximum frequency of the VFD so the actual spindle speed is within 1 rpm of 640 RPM.
10. Spot check the spindle speed across the full speed range commanded vs. actual reported by the spindle encoder.
11. It is more critical for the spindle speed to be exact at lower speeds and it is normal for the actual spindle speed to be correct at low speeds and up to 50 rpm slower at max rpm then the commanded speed.
12. If there is a multi-range spindle see the operating manual for setting up parameters 65-67.

Step 4.) Tune the VFD for best acell and decell performance. Whenever possible, use a braking resistor in conjunction with the VFD for the fastest possible stops. See the VFD manual for proper size and type of braking resistor and the associated VFD parameter settings to activate the feature (DC injection braking). Tune VFD for the fastest possible stops from Max RPM without a VFD fault. Typical VFD Spindle Motor braking and accelerating values to shoot for are 3 second acceleration and 3 second deceleration times for good rigid tapping results.

Step 5.) Set the Rigid Tapping CNC12 parameters at base line values using the values in Table 1 (below) as a guide.

Step 6.) Test Tap holes in machinable wax. Using wax as a test medium does two things. It saves the tap from breakage but more importantly also quickly exposes bad threads better than metal. So using wax is critical to dialing in the rigid tapping parameters where if you tapped in metal the test results not be so obvious when fine tuning. For instance, if the Z axis was pulling out of the hole a little too soon and running ahead of the spindle this puts stress on the threads in metal this might not show up as a problem at first..but in wax the wax threads will give way at a much lower force than metal and it will be obvious that things are not perfect. If rigid tap parameters are not set perfectly you will not get good threads in wax and if you are getting good threads in wax you will get good threads in metal. PVC and some other plastics can be used as alternatives to machinable wax but wax is preferred for fine tuning Rigid Tapping.

Use Centroid's Intercon Conversational programming to program the test tapping program or download sample 1/4-20 Rigid Tapping Intercon programs for Mill and Lathe here. Notice the programmed tapping RPM is 640 RPM and the spindle is in hi range.

Use the trouble shooting guidelines below to adjust the Rigid Tapping Parameters as necessary for good results in wax.

Making Threads Notes:

A.) Typical Tapping RPM. Use 640 RPM for most all taps.

B.) If the spindle has multiple gear ranges Tapping is typically performed in High range! (Using very low gear ranges drastically slows down the ability of the VFD to decelerate, stop and accelerate).

C.) Once Tapped hole sizes approach a certain size alternative methods of creating threads are often used and are recommended. Threadmilling using a milling machine or router can easily create internal and external threads as small as .080! (Harvey Tools makes a .080 threadmill). Threadmilling is also simple to program and is an included feature of Centroid's Mill Intercon Conversational programming software. And threadmilling is also preferred method for creating threads in hard materials. And for a Lathe, once internal thread size reaches around 1/2" it time to start using the threading cycle and turn the threads. So, when possible use a internal threading tool and use the threading canned cycle in Centroid's Lathe Intercon Conversational programming software.

**Table 1. Base Line Rigid Tapping Parameters for Acorn CNC12 software.
F1 Setup, F3 Config, F3 Parameters**

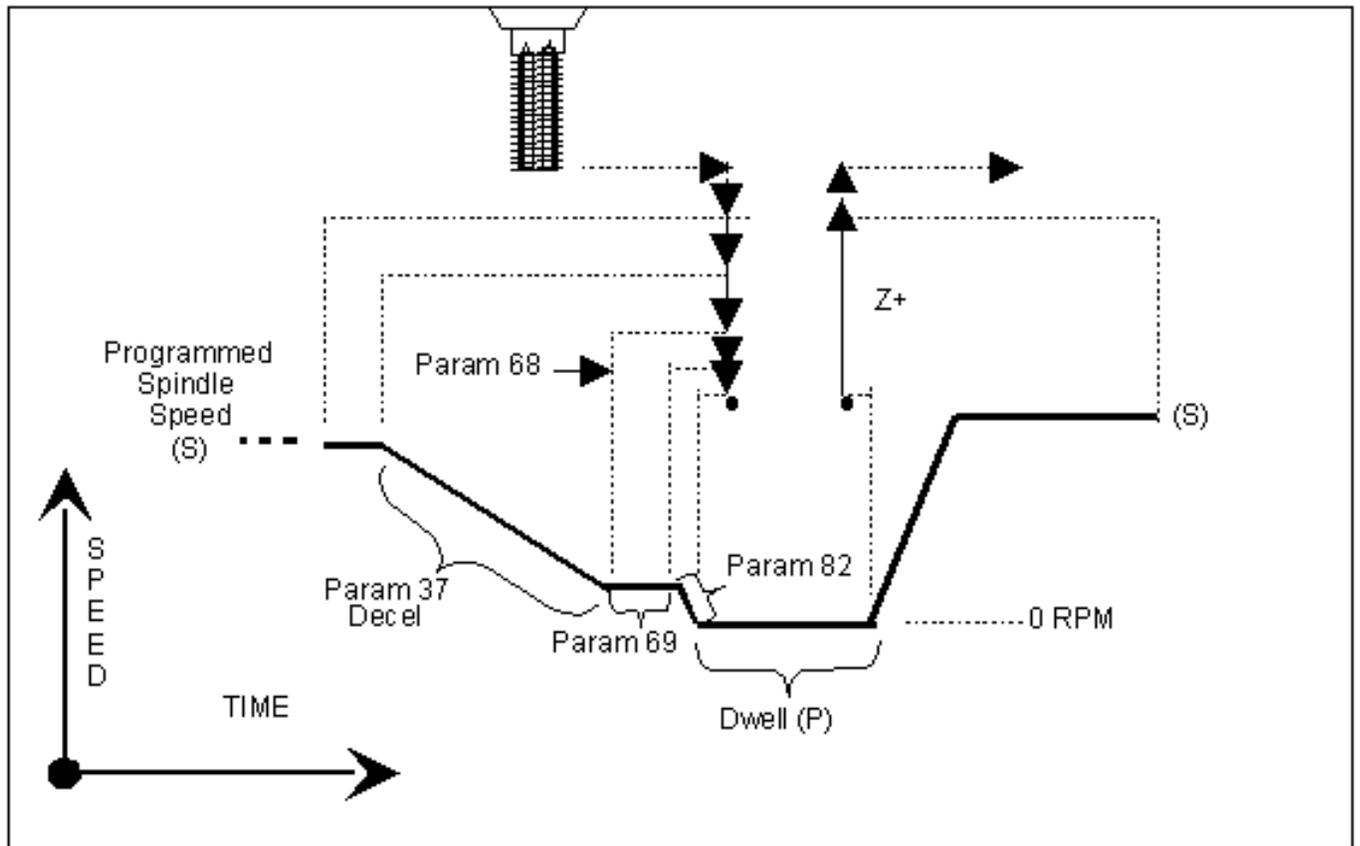
Parameter	Value	Function
36	1	Bit 0: 0-Disable rigid tapping, 1-Enable rigid tapping Bit 1: 0-Wait for index pulse during rigid tapping, 2-Do not wait for index pulse Bit 2: 0-Do not allow spindle override, 4-Allow spindle override <i>Example:</i> A value of 3 will enable rigid tapping (bit 0 = 1) and during execution will not wait for the index pulse to start (bit 1 = 2) and the spindle override will not change the spindle speed (bit 2 = 0). Note: Please check the operator's manual for a detailed description of this parameter.
37	3-10	Spindle Deceleration Time - This value is used for setting the spindle deceleration rate from the programmed spindle speed (S) down to the spindle speed of Parameter 68. Check the value of your inverter setting and enter it. (i.e. if the inverter is set to 3 seconds deceleration, enter 3).
68	640 RPM	Minimum Rigid Tapping Spindle Speed (RPM) - This parameter is the speed that the spindle slows down to from the programmed spindle speed towards the end of the tapping cycle. The lower the value, the more accurately the Z axis will land on target but at the expense of possibly stalling the motor which in turn will cause Z to stop short. If this value is too large, the off-target error increases.
69	1.75 sec	Duration for Minimum Spindle Speed Mode (Seconds) - This is a buffer time value to allow the spindle time to decelerate to the spindle speed defined by parameter 68. If the number is too small, overshoot may occur. If it's too large, the user waits longer for the hole to be tapped at the slow speed specified by parameter 68.
74	4	Spindle M function to be run at the bottom of the hole for G84 tapping. Spindle M function to be run at the top of the hole to for G74 counter tapping.
78	1	Bit 0: 0-Displayed speed is commanded speed, 1-Displayed speed is actual speed from spindle encoder Bit 1: 0-Does not prorates feedrate, 2-Prorates programmed feedrate proportional to spindle speed if speed is lower than threshold percent set by parameter 149 Bit 2: 0-Does not wait for spindle to get up to speed, 4-Waits for the spindle speed to get up to speed threshold set by parameter 149 before continuing on with the next line in the program
82	500	Spindle Drift (in degrees of rotation, i.e. 360 = 1 full rotation, 90 = 1/4 rotation) - This value is the number of degrees that the spindle moves to coast to a stop when it is shut off at the speed specified by parameter 68. This value is proportional to the distance above the Z target at which the spindle motor must be shut off in order for Z to land on target. (Remember that Z is slaved to the spindle speed during rigid tapping.) Increasing this value will cause Z to stop sooner when the spindle stops. Decrease this value will cause Z to travel further when the spindle stops.
84	3	Spindle M function to be run at the bottom of the hole for G74 tapping. (left hand taps) Spindle M function to be run at the top of the hole to for G84 counter tapping.(right hand taps)
240	.1" (inches)	Rigid Tapping Accel Distance for Tapping and Threading. This is sacrificial lead in distance that is used to sync up Z axis with Spindle Encoder. Typical open loop stepper motor based systems use a value around .1" (close loop typical distance is .05") Smaller number = shorter sync up distance there for more abrupt z axis movement. Larger number = more gentle sync up. Too large and sync up distance will start to interfere with threads. Each machine is a little different some experimentation is needed to determine best value.
241	20	Parameter 241 is used to allow the Z axis to accelerate up to spindle encoder synchronization in a smooth way. P241 is the amount in which the Z axis is allowed to use to get in sync with the spindle encoder for Tapping and Threading The default value is 10 and the minimum value is 10. P241 is in degrees of rotation that the Z axis accelerates to get Z axis sinked up with the spindle at the beginning of a Tapping cycle. This helps eliminate banging (cause by instantaneous accel) to get sinked) at the start and end of the tap and thread. Basically you are specifying how much rotational sacrificial movement is allowed to gently accelerate the Z axis up to the spindle encoder synchronization. A smaller value makes Z axis accelerate over a shorter distance therefore is a faster sync up, a larger value is for more gentle sync up (less abrupt synchronization). For examples: with high RPM and a course pitch threads P241 may have to be increase to eliminate the banging. OR for Open loop stepper

		motor based systems P241 is typically set to 20 for a more gentle sync up move so as to not lose steps. Parameters 240 and 241 are
149	1	Waits for spindle to get up to speed before tapping. 1 = 100% of programmed speed.

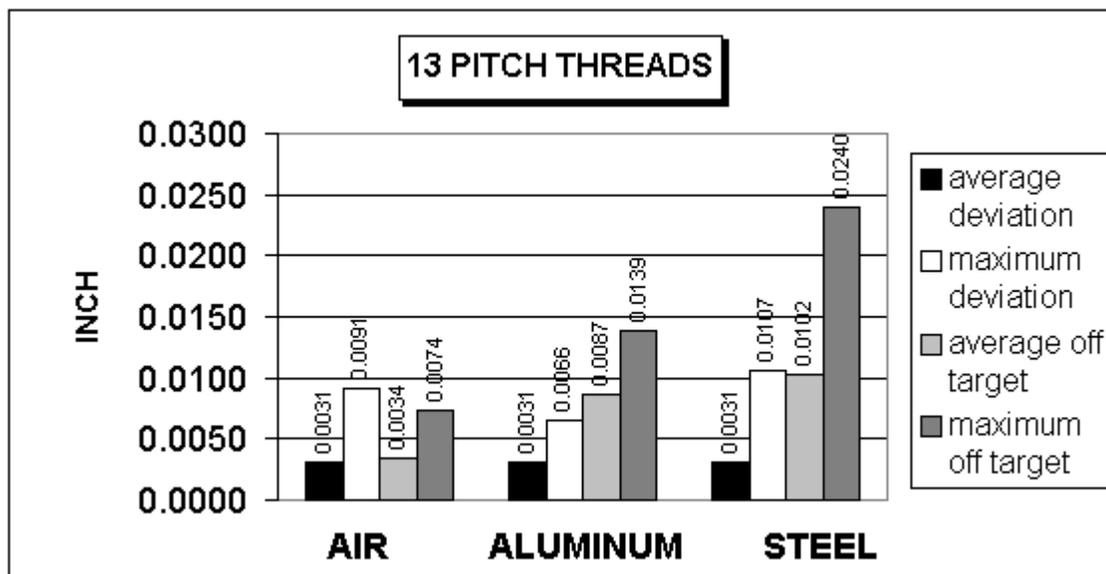
Rigid Tapping Troubleshooting guidelines

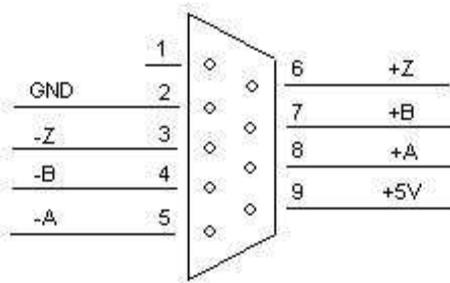
Symptoms	Possible Problem	Solution/Troubleshoot
Rigid tapping "Option not available, line xx" appears.	Acorn CNC Pro License not installed	From the main screen press F7 (Utility) ->F8 (Options)> F2 Import License
When running a G74 or G84 rigid tap cycle, the control stops above the hole to be tapped with a message in the status window "moving" and DOES NOT continue.	<ol style="list-style-type: none"> 1. No spindle encoder feedback. 2. No spindle encoder index pulse. 	<ol style="list-style-type: none"> 1. Verify encoder is setup correctly in Wizard. 2. Turn the spindle by hand. Check for encoder movement in PID menu shown in figure 1. If there isn't any movement, check the spindle encoder, the cable, and the connections. Figure 2 shows the DB9 pin out for the encoder. 3. Verify the system sees the index pulse from the spindle encoder. Turn the spindle by hand and look for the asterisk, *, next to the axis label that is designated for the spindle as shown in figure 1. 4. If the asterisk is not seen from step 3, then check the spindle encoder, the cable, and the connections. Figure 2 shows the DB9 pin out for the encoder.
When running a G74 or G84 rigid tap cycle, you get a "410: Z axis (3) position error".	<ol style="list-style-type: none"> 1. Spindle encoder turning opposite direction of spindle motor. 2. Spindle speed too high for tapping. 3. Wrong number of spindle encoder counts. 	<ol style="list-style-type: none"> 1. Change the sign of the Encoder counts value in the Wizard as described above. 2. Try tapping at a reduced spindle speed. 3. See figure 1 below to verify encoder counts for the spindle encoder is set correctly. When the spindle is turned 1 revolution, the number of counts should change by the value that's in parameter 34.
Tapping does not make it to the bottom of the hole. Not deep enough.	<ol style="list-style-type: none"> 1. Not programmed deep enough. 2. Spindle is turning off too soon. 	<ol style="list-style-type: none"> 1. Check Intercon or G code program for depth. 2. Parameter 68, 69, and/or 82 may need adjusted as described above.
Breaking taps in blind holes or tapping too deep.	<ol style="list-style-type: none"> 1. Programmed too deep. 2. Spindle is turning off too late. 3. Spindle speed is too high. 	<ol style="list-style-type: none"> 1. Check Intercon or G code program for depth. 2. Parameter 68, 69, and/or 82 may need adjusted as described above. 3. Slow down spindle speed.
Ripping up threads and or Breaking taps in blind holes or tapping too deep.	<ol style="list-style-type: none"> 1.) VFD not setup properly for good accel decell rates. 2.) Machine has too much inertia and overpowers VFD and VFD and spindle motor are under sized. 	<p>Install braking resistor and turn on braking features in VFD setup parameters. (DC injection braking) see VFD manual for details.</p> <p>Use a Floating Tap Holder so the tap has some compliance to compensate for overshoot.</p>

Graphic representation of Rigid Tapping control parameters



Graphic Representation of Test Results for Precision





Document History

Rev0 Created on 2018-01-02 by *jkm*