

## Centroid CNC12 v5.20 Mill, Lathe, Router, Plasma Release Notes:

CNC software is now available for use with Acorn, AcornSix, Hickory, Allin1DC, Oak and MPU11. Please [Follow the installation instructions](#). Do not use "restore report" using a report from an earlier version.

Download CNC12 v5.20 here.

[https://www.centroidcnc.com/centroid\\_diy/centroid\\_cnc\\_software\\_downloads.html](https://www.centroidcnc.com/centroid_diy/centroid_cnc_software_downloads.html)

This is a free CNC software update! All previous license files file work with this new version of CNC12 software (in their respective categories: Acorn mill licenses work with Acorn mill, Oak lathe licenses work with Oak lathe CNC12, etc.)

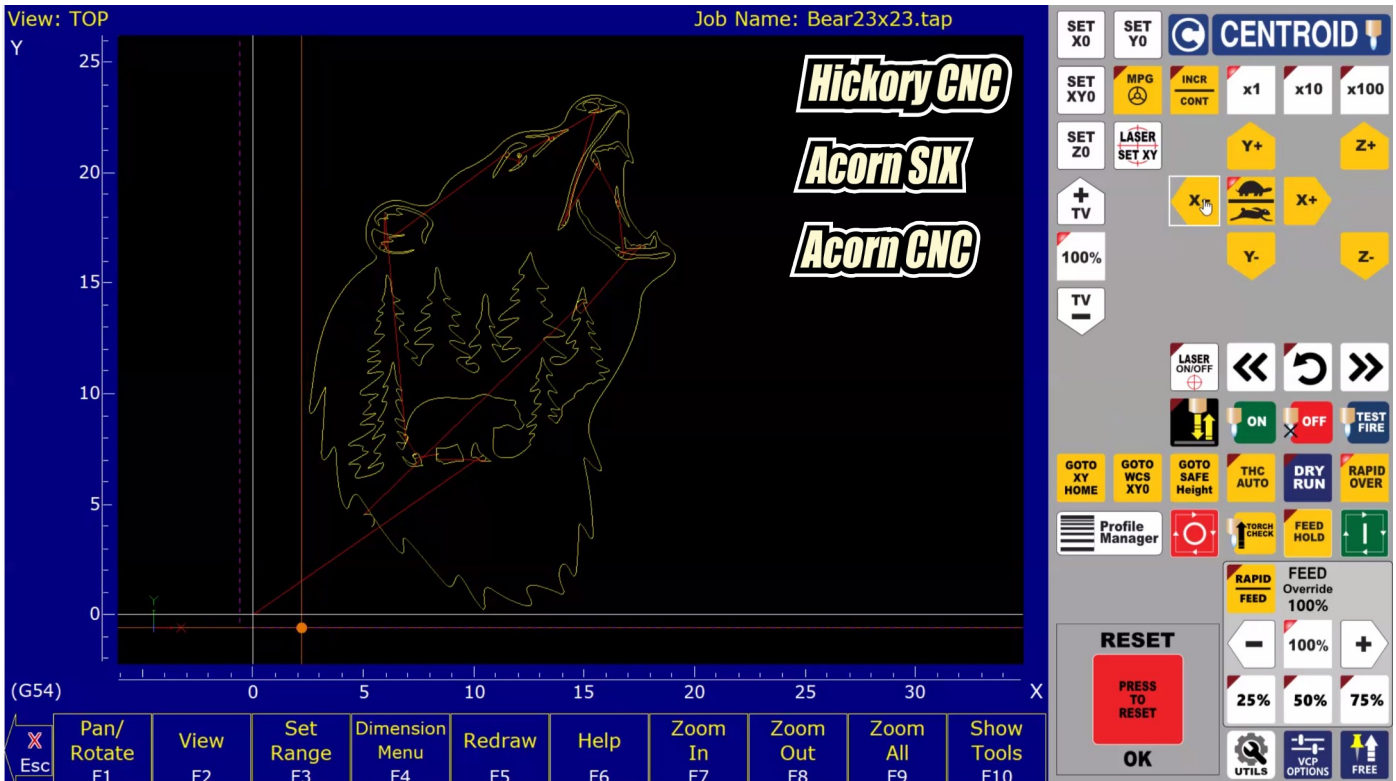
### Major Additions

1.) The Hickory EtherCat based CNC Controller now has a Centroid CNC Controller setup Wizard for Mill, Lathe, Router and Plasma versions of CNC12 which is very similar to the popular Acorn and AcornSix setup Wizard with simple drag and drop, fill-out-the-menu type CNC machine tool setup configuration Launch the new Hickory Wizard from the CNC12 Utility menu. (Note: Use of the Wizard is not required for those that are used to configuring without.)

### Hickory Router setup Wizard Axis Configuration Setup Menu

	Axis 1	Axis 2	Axis 3	Axis 4	Axis 5	Axis 6
Linear or Rotary	Linear	Linear	Linear	Linear	Linear	Linear
Label	N	N	N	N	N	N
Encoder Counts / Revolution	1048576	1048576	1048576	1048576	1048576	1048576
Overall Turns Ratio <i>turns/in. or rev/deg or turns/rev</i>	1.25	1.25	2.5	5	5	5
Lash Comp. <i>inches</i>	0	0	0	0	0	0
Max Rate <i>in./min or deg/min or rev/min</i>	2000	2000	1000	200	200	200
Fast Jog Plus Direction <i>in./min or deg/min or rev/min</i>	750	750	300	50	50	50
Fast Jog Minus Direction <i>in./min or deg/min or rev/min</i>	750	750	150	50	50	50
Slow Jog <i>in./min or deg/min or rev/min</i>	100	100	50	10	10	10
Accel / Decel <i>seconds</i>	0.375	0.375	0.375	0.5	0.5	0.5
Direction Reversal	N	N	N	N	N	N
Drive Enable Delay <i>milliseconds</i>	250	250	250	250	250	250
Travel Limit (+)	52	100	0	0	0	0
Travel Limit (-)	0	0	-11	0	0	0
x1 Linear Jog Increment <i>inches</i>	0.001					
x1 Rotary Jog Increment <i>degrees</i>	0.1					

2.) Plasma CNC12 software with Centroid Torch Height Controller (THC) hardware support is now available for use with the AcornSix and Hickory CNC controllers.



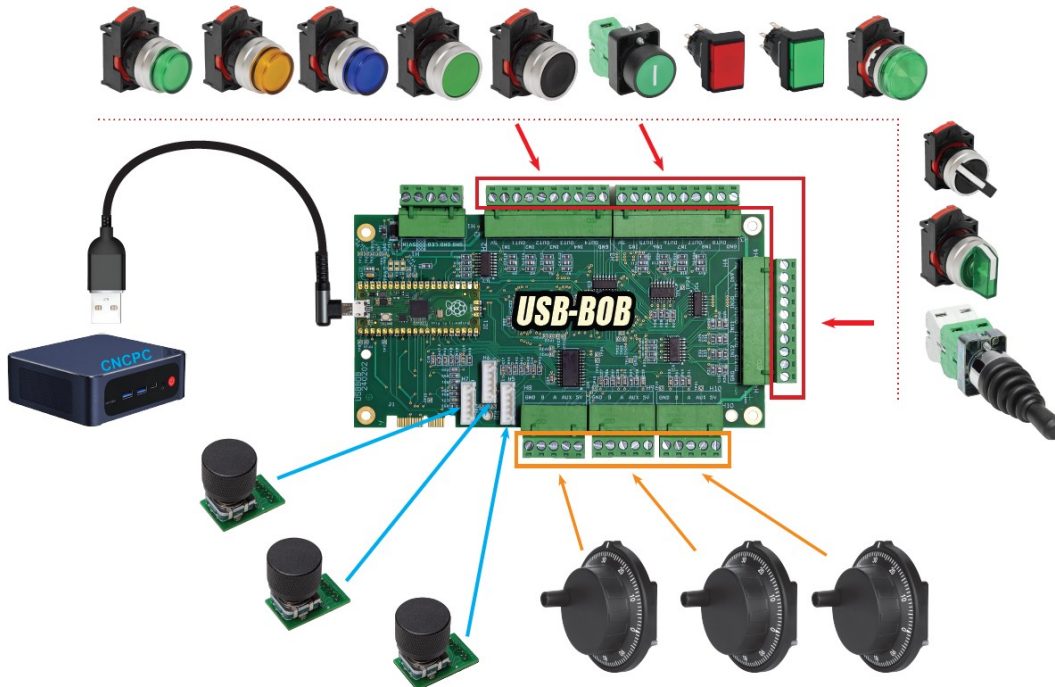
Now you can control a CNC Plasma Machine with the AcornSix or Hickory CNC Controllers in addition to the Acorn controller.



<https://shopcentroidcnc.com/acornsix-cnc-controller/>

<https://shopcentroidcnc.com/hickory-cnc-controller/>

3.) USB-BOB support in CNC12 and the Wizard (Acorn, AcornSix and Hickory) . Create your own CNC control operators console with the USB-BOB!



Router CNC Control Configuration Wizard

### Centroid USB-BOB CNC Operator Control Panel Breakout Board

Input Type

- FeedHold
- FeedRate 100%
- FeedRate 25%
- FeedRate 50%
- FeedRate 75%
- Flood
- JogDisable
- Limit Defeat
- Mist
- MPG Axis 4 Select
- MPG Mode Toggle
- PopUpPins
- RapidFeedLink
- RapidRate 100%
- RapidRate 25%
- RapidRate 50%
- RapidRate 75%
- SingleBlock
- Spindle Brake
- Spindle Range High
- Spindle Range Low
- Spindle Range Med
- Spindle Speed 100%
- Spindle Speed Minus
- Spindle Speed Plus

USB-BOB #1 Connected

USB-BOB #2 Not Found

Input	Definition
1 IN1	CycleStart
2 IN2	ToolCheck
3 IN3	CycleCancel
4 IN4	Spindle Auto/Manual
5 IN5	Spindle On Toggle
6 IN6	Spindle CW
7 IN7	Spindle CCW
8 IN8	Incremental/Continuous Jog
9 IN9	MPG Axis 1 Select
10 IN10	MPG Axis 2 Select
11 IN11	MPG Axis 3 Select
12 IN12	ToolRelease
13 KB13	Feedrate Custom %
14 KB14	Spindle Speed Custom %
15 KB15	Rapidrate Custom %

MPG (H8)    MPG (H9)    MPG (H10)

Use Selector Switch    Select Axis    Select Axis

**Override Encoder Knob Speed Multiplier 1x-20x**

Feedrate Override (H5)    1 x

Spindle Speed Override (H6)    1 x

Rapid Override (H7)    1 x

**Values for the Encoder Knob Push Button when using the "Custom" % inputs**

Value for use with the "Feedrate Custom %" input    2 %

Value for use with the "Spindle Speed Custom %" input    100 %

Value for use with the "Rapidrate Custom %" input    50 %

Connected to CNC12

Write Settings to CNC Control Configuration

More information on USB-BOB is here: [https://shopcentroidcnc.com/shop/cnc-accessories/centroid\\_usb-bob\\_operator\\_control\\_panel/](https://shopcentroidcnc.com/shop/cnc-accessories/centroid_usb-bob_operator_control_panel/)



4.) New CNC12 Router specific CNC12 software for Acorn, AcornSix and Hickory!

- CNC12 Function Key tree rearranged for improved Router Work Flow (less keystrokes, and new tree puts the menus you use the most often at your finger tips)

WCS #1 (G54) Current Position (Inches)

Job Name: NO\_JOB\_LOADED.cnc  
 Tool: T1 H1  
 Feedrate: 100% 0.0 ipm  
 Spindle: 0 A  
 Rapid Rate: 100%

335 Emergency stop released  
 347 Reset Cleared  
 490 Reset Initiated, Press Reset to Clear  
 347 Reset Cleared

Press CYCLE START to start job

--- Warning: Machine Home Position Not Set ---

1) Jog all axes to Machine Home Position  
 2) Press CYCLE START to set the home position

Machine Coord.  
 X +0.0000  
 Y +0.0000  
 Z +0.0000

MPU12 SYSTEM ID  
 38D2694D64EF-0406180825  
 CNC12 Acorn Router v5.20  
 ACORN ROUTER PRO

Buttons: Set Part Zeros (F1), Load Job (F2), MDI (F3), Tool/ATC (F5), Edit G-code (F6), Utility Menu (F7), Smoothing (F9), Shut Down Menu (F10)

Control Panel: AUTO SPINDLE MAN, 100%, M3, M4, M58, SET WCS XY0, SET WCS Z0, Auto Z to PLATE, PARK, SET Rotary WCS, AXES PAIRED, GOTO WCS XY0, LIMIT SWITCH DEFEAT, LASER SET XY, LASER ON/OFF, RESET HOME, AUTO MAN, DUST Collector, Hold Down VAC, Air Blow, Change TOOL, INCR CONT, x1, x10, x100, MPG, Y+, Z+, X-, X+, Y-, Z-, SINGLE BLOCK, TOOL CHECK, FEED HOLD, RAPID FEED, FEED Override 100%, 25%, 50%, 75%, UTILS, VCP OPTIONS, FREE, RESET PRESS TO RESET, OK

- All new Router Part Setup Menu with menu driven support for Part Setups for all common Router specific methods such as but not limited to: Touch Plate, Cross Hair Laser, Touch Probe, and various Manual methods all in one quick to navigate menu. Here are a few samples of the new part setup menus.

WCS #1 (G54) Current Position (Inches)

Job Name: NO\_JOB\_LOADED.cnc  
 Tool: T1 H1  
 Feedrate: 100% 0.0 ipm  
 Spindle: 0 A  
 Rapid Rate: 100%

335 Emergency stop released  
 347 Reset Cleared  
 490 Reset Initiated, Press Reset to Clear  
 347 Reset Cleared

Press CYCLE START to start job

Distance to Go  
 X +0.0000  
 Y +0.0000  
 Z +0.0000

Machine Coord.  
 X +0.0000  
 Y +0.0000  
 Z +0.0000

Buttons: Set Part Zeros (F1), Load Job (F2), MDI (F3), Run Job Options (F4), Tool/ATC (F5), Edit G-code (F6), Utility Menu (F7), Graph Job (F8), Smoothing (F9), Shut Down Menu (F10)

Control Panel: AUTO SPINDLE MAN, 100%, M3, M4, M58, SET WCS XY0, SET WCS Z0, Auto Z to PLATE, PARK, SET Rotary WCS, AXES PAIRED, GOTO WCS XY0, LIMIT SWITCH DEFEAT, LASER SET XY, LASER ON/OFF, RESET HOME, AUTO MAN, DUST Collector, Hold Down VAC, Air Blow, Change TOOL, INCR CONT, x1, x10, x100, MPG, Y+, Z+, X-, X+, Y-, Z-, SINGLE BLOCK, TOOL CHECK, FEED HOLD, RAPID FEED, FEED Override 100%, 25%, 50%, 75%, UTILS, VCP OPTIONS, FREE, RESET PRESS TO RESET, OK



A few select Router Part Setup Menu Screen Shots

WCS #1 (G54) Current Position (Inches) Job Name: Non ATC Gasket Groove.cnc  
 X +1.2400 Tool: T2 H---  
 Y +0.7900 Feedrate: 100% 0.0 ipm  
 Z -0.1569 Spindle: 0 A  
 Rapid Rate: 100%

335 Emergency stop released  
 347 Reset Cleared  
 490 Reset Initiated, Press Reset to Clear  
 347 Reset Cleared

Press CYCLE START to start job

Touch Plate XYZ Position Auto Probing  
 1.) Position Tool, adjust options  
 2.) Press Cycle Start to run the Probing Cycle

Bit Diameter 1.0000 Magnet Reminder Yes  
 Z Clearance Amount 1.0000 Flute Reminder Yes  
 Return Z to home Yes Set Z Zero Yes

F2 to Select Corner

Esc	Inside/Outside F1	Corner F2	Side F3	Z Only F5	Bore F6	Set CSR Angle F7	Plate Setup F8	WCS CSR Table F9
-----	-------------------	-----------	---------	-----------	---------	------------------	----------------	------------------

Avid CNC Touch Plate Part Zero setup menu.

WCS #1 (G54) Current Position (Inches) Job Name: Non ATC Gasket Groove.cnc  
 X +0.0000 Tool: T2 H---  
 Y +0.0000 Feedrate: 100% 0.0 ipm  
 Z +0.0000 Spindle: 0 A  
 Rapid Rate: 100%

335 Emergency stop released  
 347 Reset Cleared  
 490 Reset Initiated, Press Reset to Clear  
 347 Reset Cleared

Press CYCLE START to start job

Touch Plate XYZ Position Auto Probing  
 1.) Position Tool, adjust options  
 2.) Press Cycle Start to run the Probing Cycle

Bit Diameter 1.0000 Magnet Reminder Yes  
 Z Clearance Amount 1.0000 Flute Reminder Yes  
 Return Z to home Yes Set Z Zero Yes

F2 to Select Corner

Esc	Inside/Outside F1	Corner F2	Side F3	Z Only F5	Bore F6	Set CSR Angle F7	Plate Setup F8	WCS CSR Table F9
-----	-------------------	-----------	---------	-----------	---------	------------------	----------------	------------------

Touch Plate menus

WCS #1 (G54) Current Position (Inches) Job Name: Non ATC Gasket Groove.cnc  
 Tool: T2 H--- Feedrate: 100% 0.0 ipm Spindle: 0 A Rapid Rate: 100%

X +1.2400  
 Y +0.7900  
 Z -0.1569

335 Emergency stop released  
 347 Reset Cleared  
 490 Reset Initiated, Press Reset to Clear  
 347 Reset Cleared  
 Press CYCLE START to start job

Cross Hair LASER Set Part Zero Location  
 Jog to position, enter values press F10 Set

Axis	Part Position
X	0.0000
Y	0.0000

X Laser to Spindle C/L Offset -4.0000  
 Y Laser to Spindle C/L Offset 1.5000

Manual Laser Probe Plate CSR Laser On Teach WCS CSR Set  
 F1 F2 F3 F4 F5 F7 F8 F9 F10

ACORN CNC Control Panel: AUTO SPINDLE MAN, SET WCS XY, SET WCS Z, Auto Z to PLATE, PARK, 100%, M3, SET Rotary WCS, M55, GOTO WCS XY, LIMIT SWITCH DEFEAT, M58, LASER SET XY, LASER ON/OFF, RESET HOME, AUTO MAN, DUST Collector, Hold Down VAC, Air Blow, Change TOOL, INCR CONT, x1, x10, x100, MPG, 4th+, 4th-, SINGLE BLOCK, TOOL CHECK, FEED HOLD, RAPID FEED, FEED Override 100%, 25%, 50%, 75%, RESET, PRESS TO RESET, OK, UTILS, VCP OPTIONS, FREE.

Cross Hair Laser to set X and Y zero position

WCS #1 (G54) Current Position (Inches) Job Name: Non ATC Gasket Groove.cnc  
 Tool: T2 H--- Feedrate: 100% 0.0 ipm Spindle: 0 A Rapid Rate: 100%

X +1.2400  
 Y +0.7900  
 Z -0.1569

335 Emergency stop released  
 347 Reset Cleared  
 490 Reset Initiated, Press Reset to Clear  
 347 Reset Cleared  
 Press CYCLE START to start job

Set Part Zero Position

Axis	Part Position	Edge Finder Dia.	Approach from
X	0.0000	0.2500	Left (-)
Y	0.0000	0.2500	Back (+)
Z	0.0000	Tool #: 2	

Help: Shift F12

Manual Laser Probe Plate CSR Auto Zero Set XY WCS CSR Set  
 F1 F2 F3 F4 F5 F7 F8 F9 F10

ACORN CNC Control Panel: AUTO SPINDLE MAN, SET WCS XY, SET WCS Z, Auto Z to PLATE, PARK, 100%, M3, SET Rotary WCS, M55, GOTO WCS XY, LIMIT SWITCH DEFEAT, M58, LASER SET XY, LASER ON/OFF, RESET HOME, AUTO MAN, DUST Collector, Hold Down VAC, Air Blow, Change TOOL, INCR CONT, x1, x10, x100, MPG, 4th+, 4th-, SINGLE BLOCK, TOOL CHECK, FEED HOLD, RAPID FEED, FEED Override 100%, 25%, 50%, 75%, RESET, PRESS TO RESET, OK, UTILS, VCP OPTIONS, FREE.

Manually touch part edge and set X zero position

WCS #1 (G54) Current Position (Inches) Job Name: Non ATC Gasket Groove.cnc  
 X +1.2400 Tool: T2 H--- Feedrate: 100% 0.0 ipm  
 Y +0.7900 Spindle: 0 A Rapid Rate: 100%  
 Z -0.1569

335 Emergency stop released  
 347 Reset Cleared  
 490 Reset Initiated, Press Reset to Clear  
 347 Reset Cleared  
 Press CYCLE START to start job

Set Part Zero Position

Axis	Part Position	Edge Finder Dia.	Approach from
X	0.0000	0.2500	Left (-)
Y	0.0000	0.2500	Back (+)
Z	.75	Tool #: 2	

Help: Shift F12

Manual Laser Probe Plate CSR  
 F1 F2 F3 F4 F5 Auto Zero Set XY WCS CSR Set  
 F7 F8 F9 F10

Acorn CNC  
 + AUTO SPINDLE MAN  
 100% M3 SET WCS XY0 SET WCS Z0 Auto Z to PLATE PARK  
 - M4 SET Rotary WCS M55 GOTO WCS XY0 LIMIT SWITCH DEFEAT  
 STOP M58 LASER SET XY LASER ON/OFF RESET HOME  
 AUTO MAN DUST Collector Hold Down VAC Air Blow  
 Change TOOL INCR CONT x1 x10 x100 MPG  
 4th+ X- Y+ Z+  
 4th X- Y- Z-  
 SINGLE BLOCK TOOL CHECK FEED HOLD  
 RAPID FEED FEED Override 100%  
 25% 100% 75%  
 RESET PRESS TO RESET OK  
 UTILS VCP OPTIONS FREE

Manually set Z part zero position

WCS #1 (G54) Current Position (Inches) Job Name: Non ATC Gasket Groove.cnc  
 X +1.2400 Tool: T2 H--- Feedrate: 100% 0.0 ipm  
 Y +0.7900 Spindle: 0 A Rapid Rate: 100%  
 Z -0.1569

335 Emergency stop released  
 347 Reset Cleared  
 490 Reset Initiated, Press Reset to Clear  
 347 Reset Cleared  
 Press CYCLE START to start job

Set Part Zero Position

Axis	Part Position	Edge Finder Dia.	Approach from
X	0.0000	0.2500	Left (-)
Y	0.0000	0.2500	Center
Z	0.7500	Tool #: 10	

Help: Shift F12

Manual Laser Probe Plate CSR  
 F1 F2 F3 F4 F5 Auto Zero Set XY WCS CSR Set  
 F7 F8 F9 F10

Acorn CNC  
 + AUTO SPINDLE MAN  
 100% M3 SET WCS XY0 SET WCS Z0 Auto Z to PLATE PARK  
 - M4 SET Rotary WCS M55 GOTO WCS XY0 LIMIT SWITCH DEFEAT  
 STOP M58 LASER SET XY LASER ON/OFF RESET HOME  
 AUTO MAN DUST Collector Hold Down VAC Air Blow  
 Change TOOL INCR CONT x1 x10 x100 MPG  
 4th+ X- Y+ Z+  
 4th X- Y- Z-  
 SINGLE BLOCK TOOL CHECK FEED HOLD  
 RAPID FEED FEED Override 100%  
 25% 100% 75%  
 RESET PRESS TO RESET OK  
 UTILS VCP OPTIONS FREE

Set Part Edge X0 with Touch Probe



WCS #1 (G54) Current Position (Inches)

X +1.5731

Y +1.0900

Z -0.1569

Job Name: Non ATC Gasket Groove.cnc  
 Tool: T2 H---  
 Feedrate: 100% 0.0 ipm  
 Spindle: 0 A  
 Rapid Rate: 100%

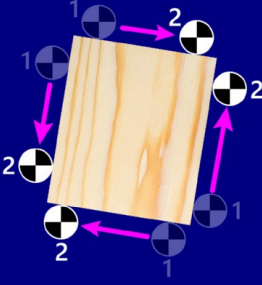
335 Emergency stop released  
 347 Reset Cleared  
 490 Reset Initiated, Press Reset to Clear  
 347 Reset Cleared

Press CYCLE START to start job

Manual CSR Coordinate System Rotation

3.) Jog to SECOND position  
 4.) Press F10 to accept SECOND position

Measured CSR Angle 0.0000  
 Last Probed CSR Angle 0.0000



Manual Teach CSR F2	Laser Teach CSR F3	Probe CSR F4	Touch Plate CSR F5	Probe Cycles F6	Zero CSR F7	MDI F8	WCS CSR Table F9	Accept F10
---------------------	--------------------	--------------	--------------------	-----------------	-------------	--------	------------------	------------



Manual "jog to position to set Coordinate System Rotation (CSR)"

WCS #1 (G54) Current Position (Inches)

X +1.5731

Y +1.0900

Z -0.1569

Job Name: Non ATC Gasket Groove.cnc  
 Tool: T2 H---  
 Feedrate: 100% 0.0 ipm  
 Spindle: 0 A  
 Rapid Rate: 100%

335 Emergency stop released  
 347 Reset Cleared  
 490 Reset Initiated, Press Reset to Clear  
 347 Reset Cleared

Press CYCLE START to start job

Laser Coordinate System Rotation

3.) Jog Cross Hair Laser to SECOND position  
 4.) Press F10 to set SECOND position

Measured CSR Angle 0.0000  
 Last Probed CSR Angle 0.0000



Laser On F1	Manual Teach CSR F2	Laser Teach CSR F3	Probe CSR F4	Touch Plate CSR F5	Probe Cycles F6	Zero CSR F7	MDI F8	WCS CSR Table F9	Accept F10
-------------	---------------------	--------------------	--------------	--------------------	-----------------	-------------	--------	------------------	------------



Cross Hair Laser set Coordinate System Rotation (CSR)

WCS #1 (G54) Current Position (Inches) Job Name: NO\_JOB\_LOADED.cnc

X +1.5731 Tool: T2 H---  
 Feedrate: 100% 0.0 ipm  
 Spindle: 0 A  
 Rapid Rate: 100%

Y +1.0900

Z -0.1569

335 Emergency stop released  
 347 Reset Cleared  
 490 Reset Initiated, Press Reset to Clear  
 347 Reset Cleared

Press CYCLE START to start job

Probing Cycles (Upgrade to Pro for all Probing Cycles)

Bore Boss Slot Web  
 Inside Corner Outside Corner Single Axis Find Angle

Probe diameter (Tool #10): 0.2500 Record Probing Data (.\\probe\_cycles\_history.txt) On

Probe Setup F9 Stylus Calibration F10

Acorn CNC

AUTO SPINDLE MAN  
 100% M3 SET WCS XYO SET WCS ZO Auto Z to PLATE PARK  
 M4 SET Rotary WCS M55 GOTO WCS XYO LIMIT SWITCH DEFEAT  
 M58 LASER SET XY LASER ON/OFF RESET HOME

AUTO MAN DUST Collector Hold Down VAC Air Blow

Change TOOL INCR CONT x1 x10 x100 MPG  
 4th+ 4th- Y+ Y- X+ X- Z+ Z-  
 SINGLE BLOCK TOOL CHECK FEED HOLD

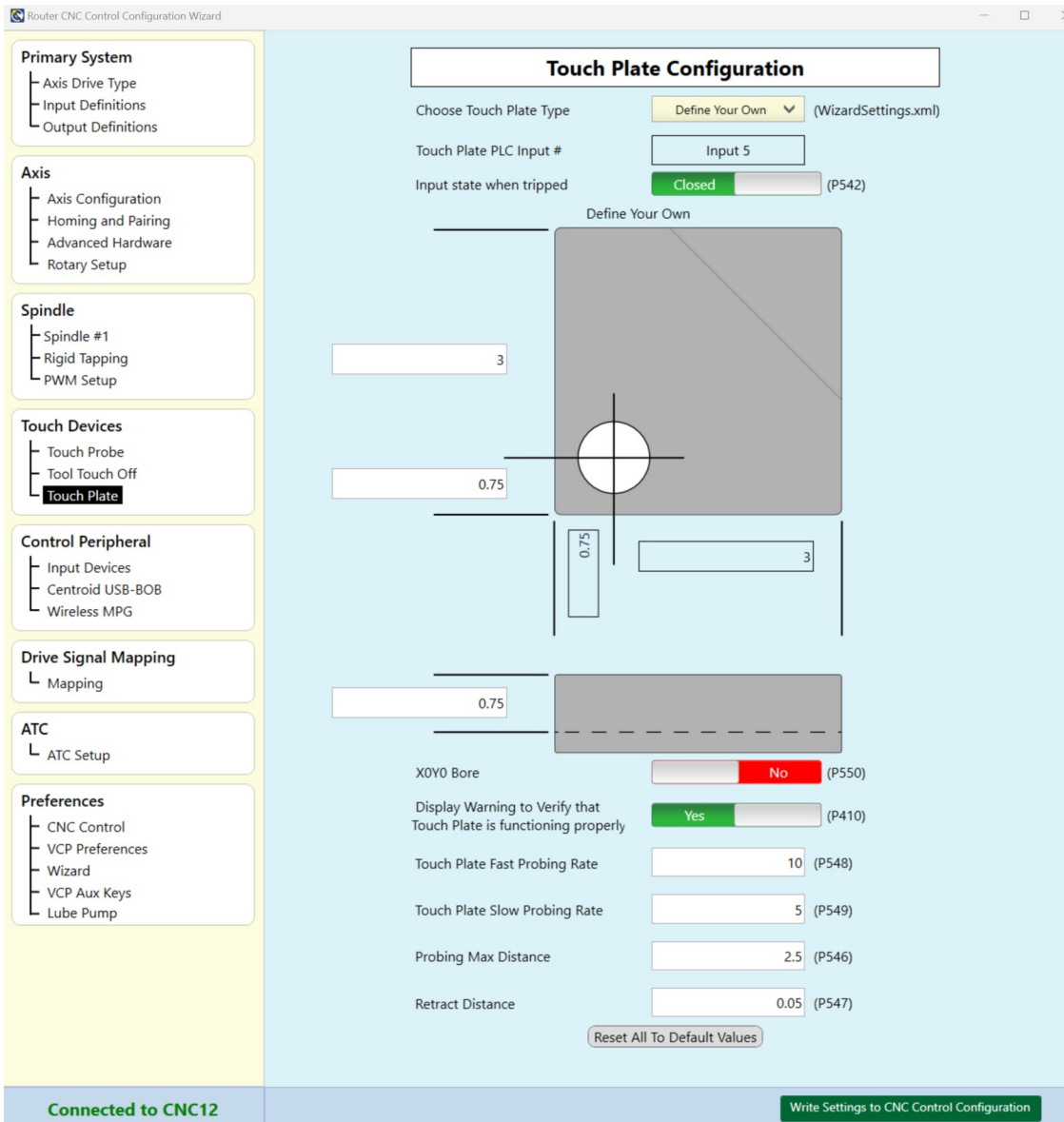
RAPID FEED FEED Override 100%  
 25% 100% 75%

RESET  
 PRESS TO RESET  
 OK

UTILS VCP OPTIONS FREE

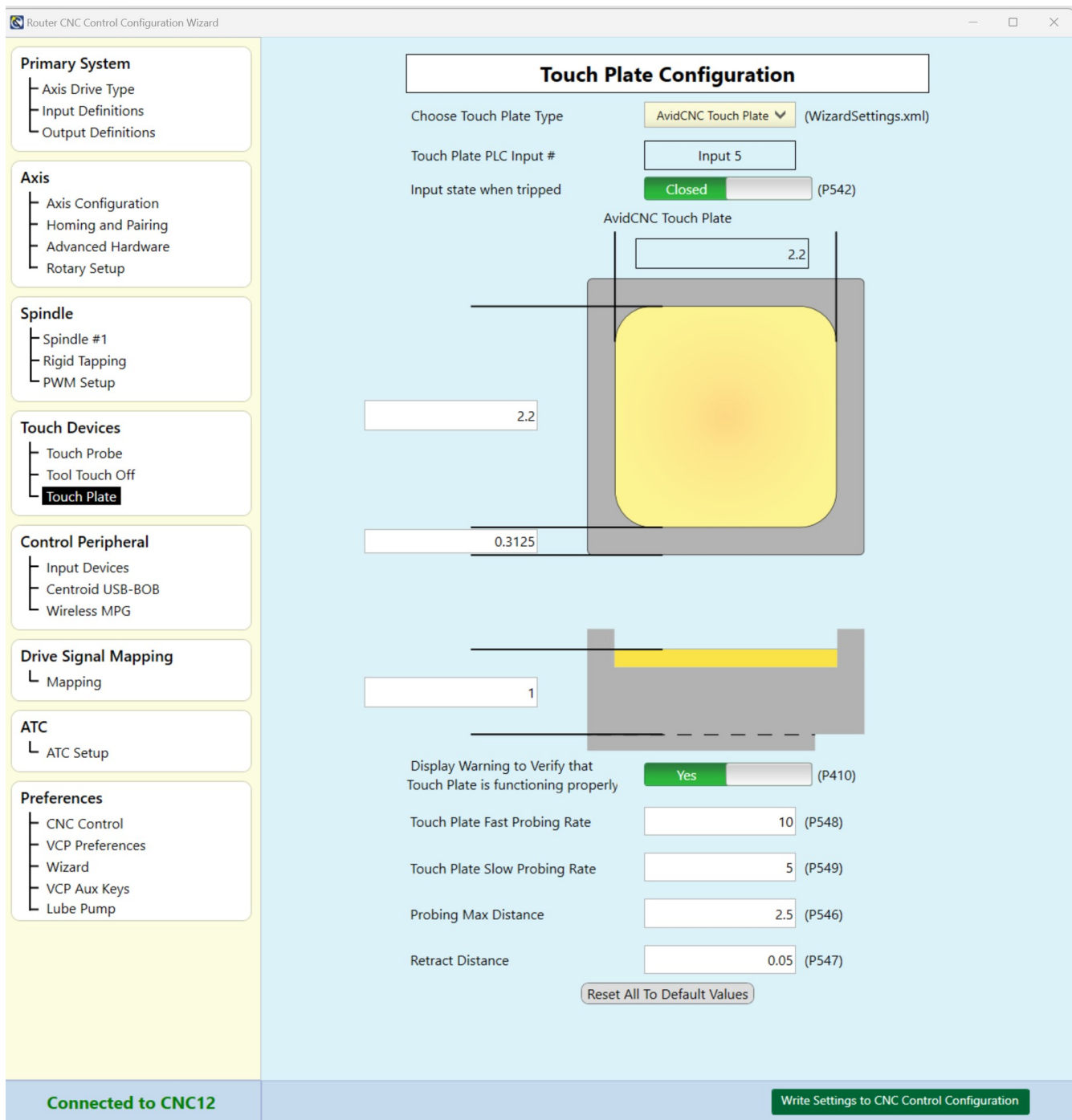
Touch Probe Menu, find positions, set part zeros, calibrate probe pre-travel automatically, record probed data to a text file.

- Router Touch Plate setup menus added to the Router Wizard, Preset AvidCNC Touch Plate selection along with "Define your own Touch Plate Geometry" for simple touch plate setup.



Define your own Touch Plate geometry, with or without a bore.





Avid CNC Touch Plate is pre-filled in with the correct data, just select it and press “Write Settings to CNC control configuration”

Router CNC Control Configuration Wizard

**Primary System**

- └ Axis Drive Type
- └ Input Definitions
- └ Output Definitions

**Axis**

- └ Axis Configuration
- └ Homing and Pairing
- └ Advanced Hardware
- └ Rotary Setup

**Spindle**

- └ Spindle #1
- └ Rigid Tapping
- └ PWM Setup

**Touch Devices**

- └ Touch Probe
- └ Tool Touch Off
- └ **Touch Plate**

**Control Peripheral**

- └ Input Devices
- └ Centroid USB-BOB
- └ Wireless MPG

**Drive Signal Mapping**

- └ Mapping

**ATC**

- └ ATC Setup

**Preferences**

- └ CNC Control
- └ VCP Preferences
- └ Wizard
- └ VCP Aux Keys
- └ Lube Pump

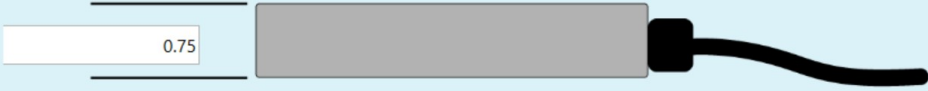
### Touch Plate Configuration

Choose Touch Plate Type Surface Plate (WizardSettings.xml) ▼

Touch Plate PLC Input # Input 5

Input state when tripped Closed (P542)

Surface Plate



Display Warning to Verify that Touch Plate is functioning properly Yes (P410)

Touch Plate Fast Probing Rate 10 (P548)

Touch Plate Slow Probing Rate 5 (P549)

Probing Max Distance 2.5 (P546)

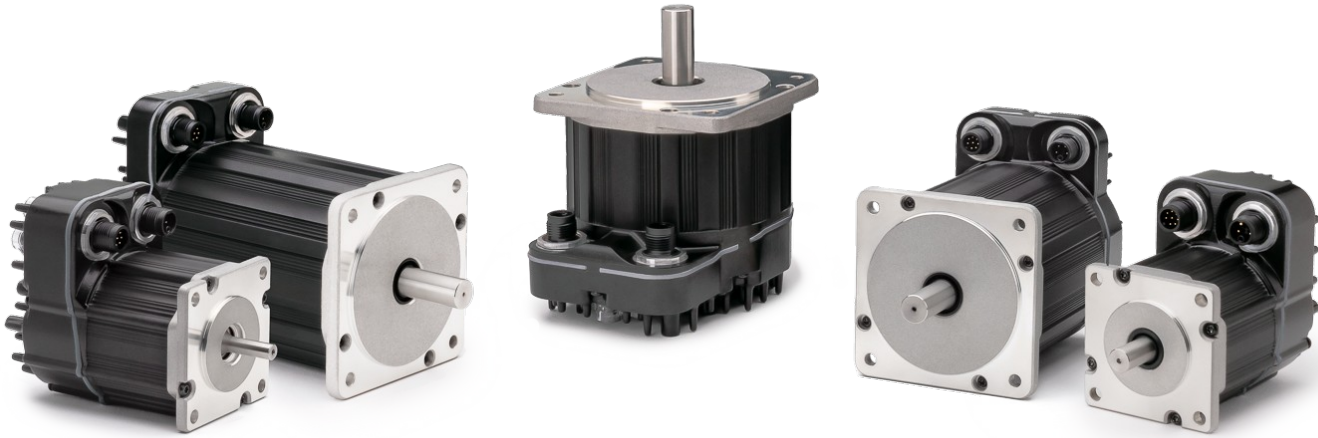
Retract Distance 0.05 (P547)

[Reset All To Default Values](#)

Connected to CNC12
Write Settings to CNC Control Configuration

and for those using a simple piece of metal as a touch off to set part Z zeros rather than a XYZ touch plate.

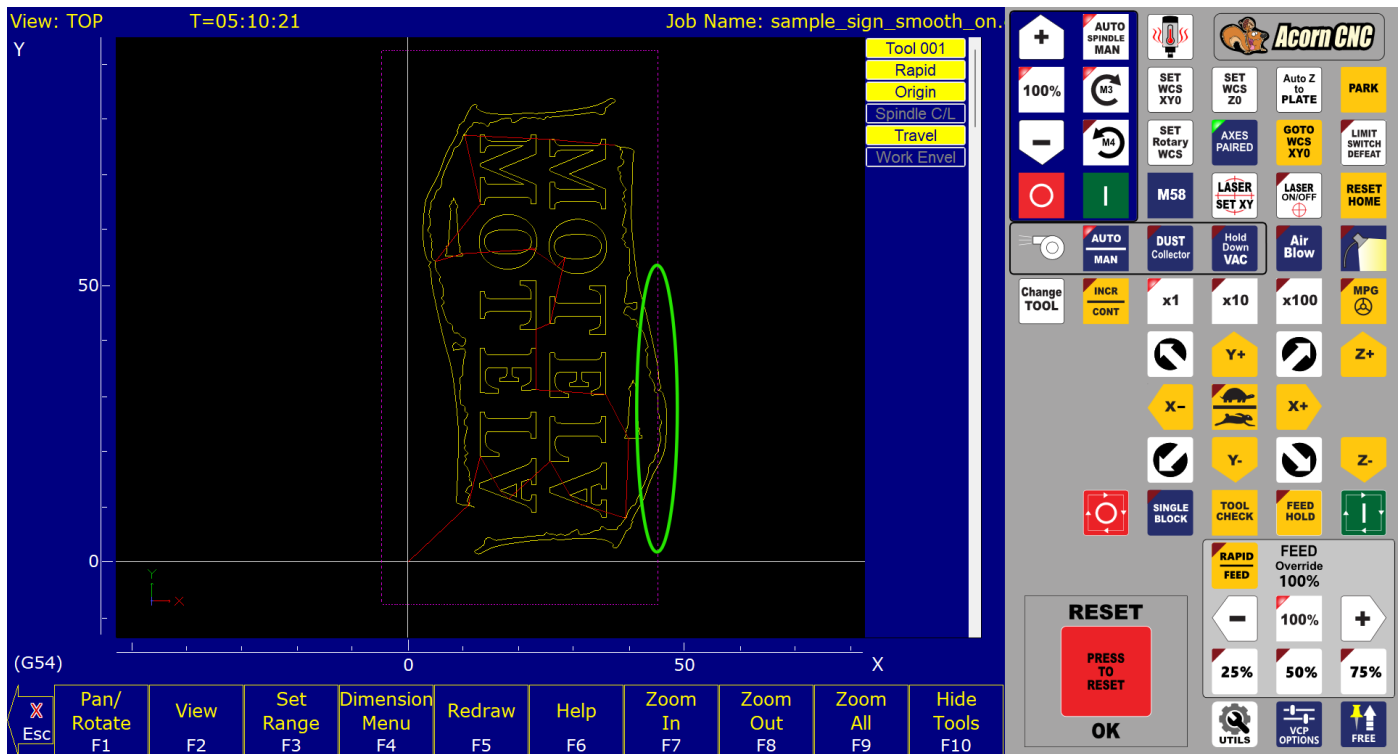
5.) The Hickory CNC controller now supports Clearpath EC EtherCat servo motors.  
Link to the Centroid Hickory Clearpath setup Tech Bulletin:  
[https://www.centroidcnc.com/dealersupport/tech\\_bulletins/uploads/332.pdf](https://www.centroidcnc.com/dealersupport/tech_bulletins/uploads/332.pdf)



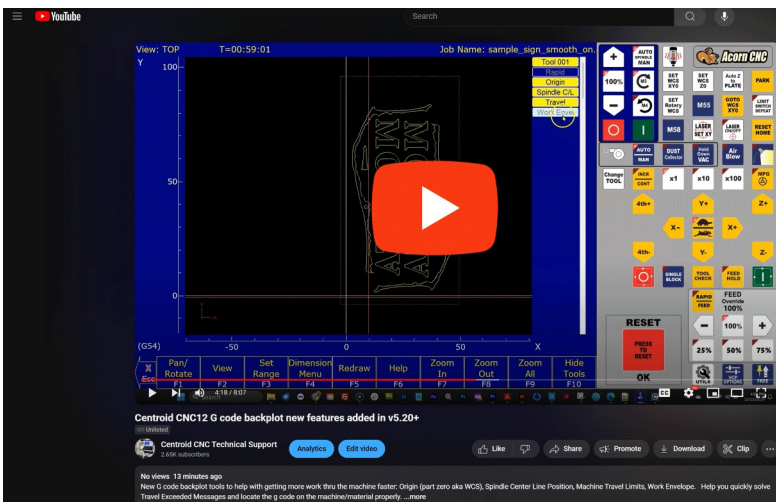
Link to Teknic web page.  
<https://teknic.com/products/clearpath-brushless-dc-servo-motors/ethercat-servo/>



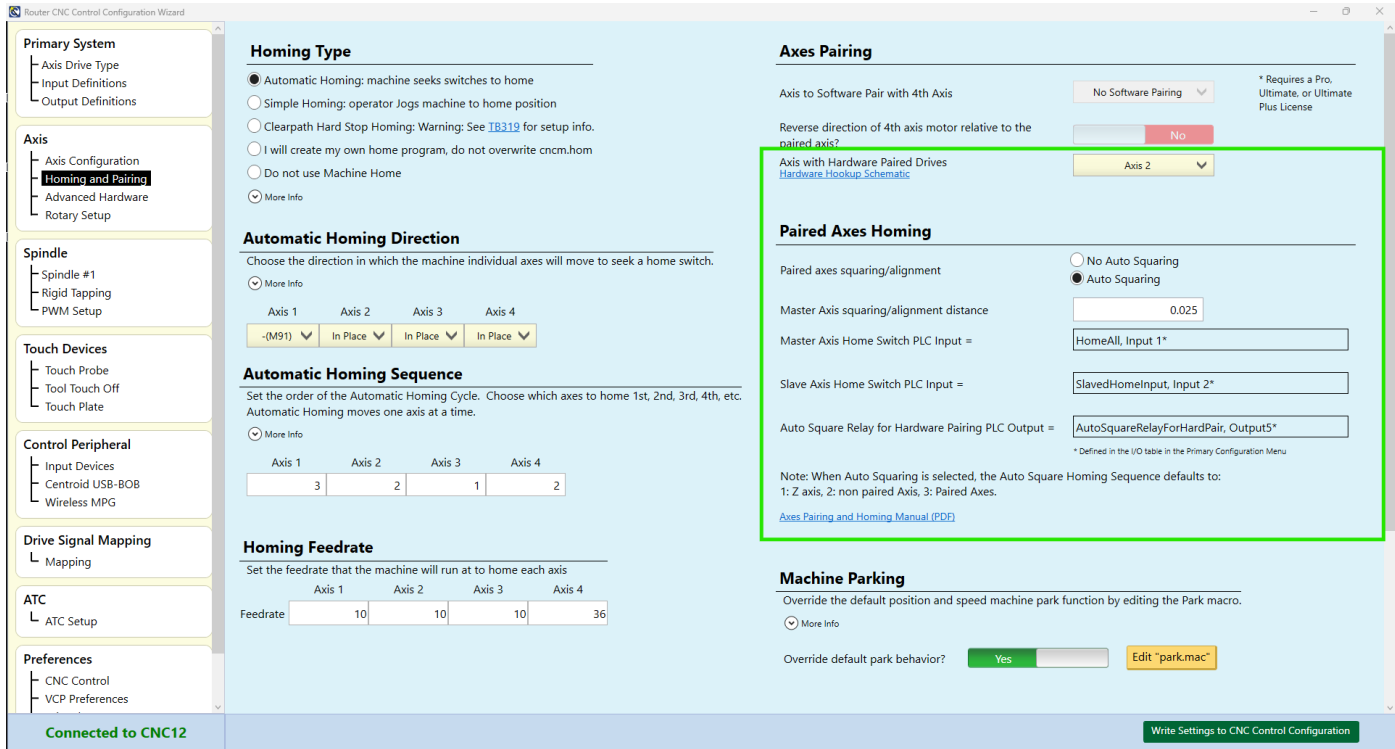
6.) Useful new G-code backplot features such as Travel Limit, Work Envelope, Dynamic Spindle Position indicator, WCS position. Example below shows the G code job exceeding the machine travel limits with the current Part zero location. Available in Mill, Router and Plasma. (Lathe will get these new features in the next version release)



Video demonstration is here [https://youtu.be/dhe\\_YG7Fsc4](https://youtu.be/dhe_YG7Fsc4)



7.) Hardware Pairing with Autosquaring is now a “canned” feature and has been added to the Acorn and AcornSix Wizard. Now you can hardware pair an axis (two axis motors hardwired together on one axis, typically a gantry) and the Wizard will write the homing program for you that will home and auto square a hardware paired axis! (no custom code needed but, you can still customize it if you want.)



Link to typical Hardware Paired CNC Router Gantry machine schematic with Autosquaring. (S15200)

[https://www.centroidcnc.com/centroid\\_diy/schematics/pbrowse.php?term=hardware](https://www.centroidcnc.com/centroid_diy/schematics/pbrowse.php?term=hardware)

8.) Lathe: G64 smoothing added to Lathe CNC12 (was previously only available in Mill, Router and Plasma).

**Smoothing Setup Menu**

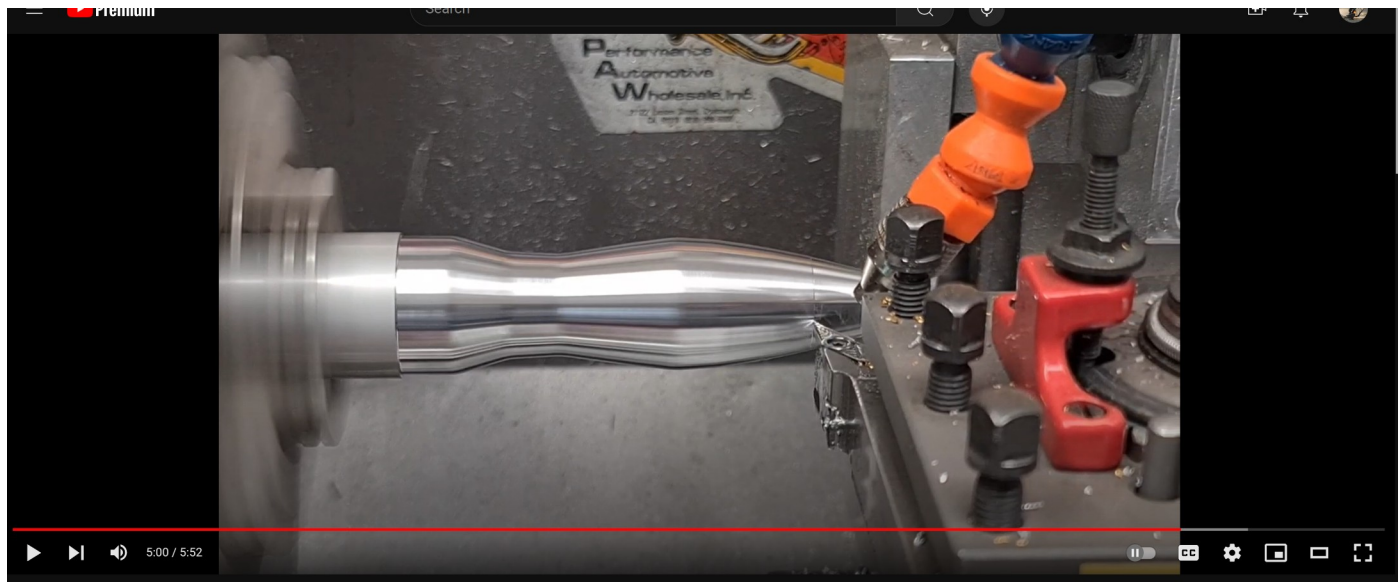
Smoothing on/off:	<input type="checkbox"/> OFF <input checked="" type="checkbox"/> ON		Smoothing is ON (P220) *
Slower Curves	<input type="range"/>	Faster Curves	Curve Feedrate Multiplier at 100% (P230)
Round All	<input type="range"/>	Sharpen All	Angles up to 95 degrees will be sharpened (P227)
Gentler Stops	<input type="range"/>	Faster Stops	Acceleration Multiplier at 100% (P231)

**Quick Setups**  
\* F2 Precision Lathe is selected

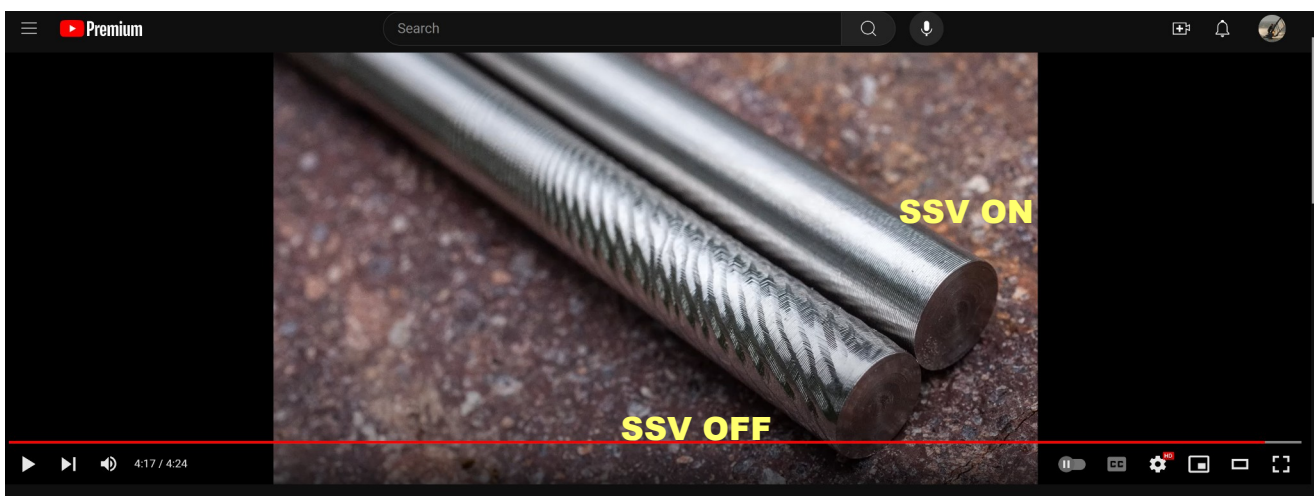
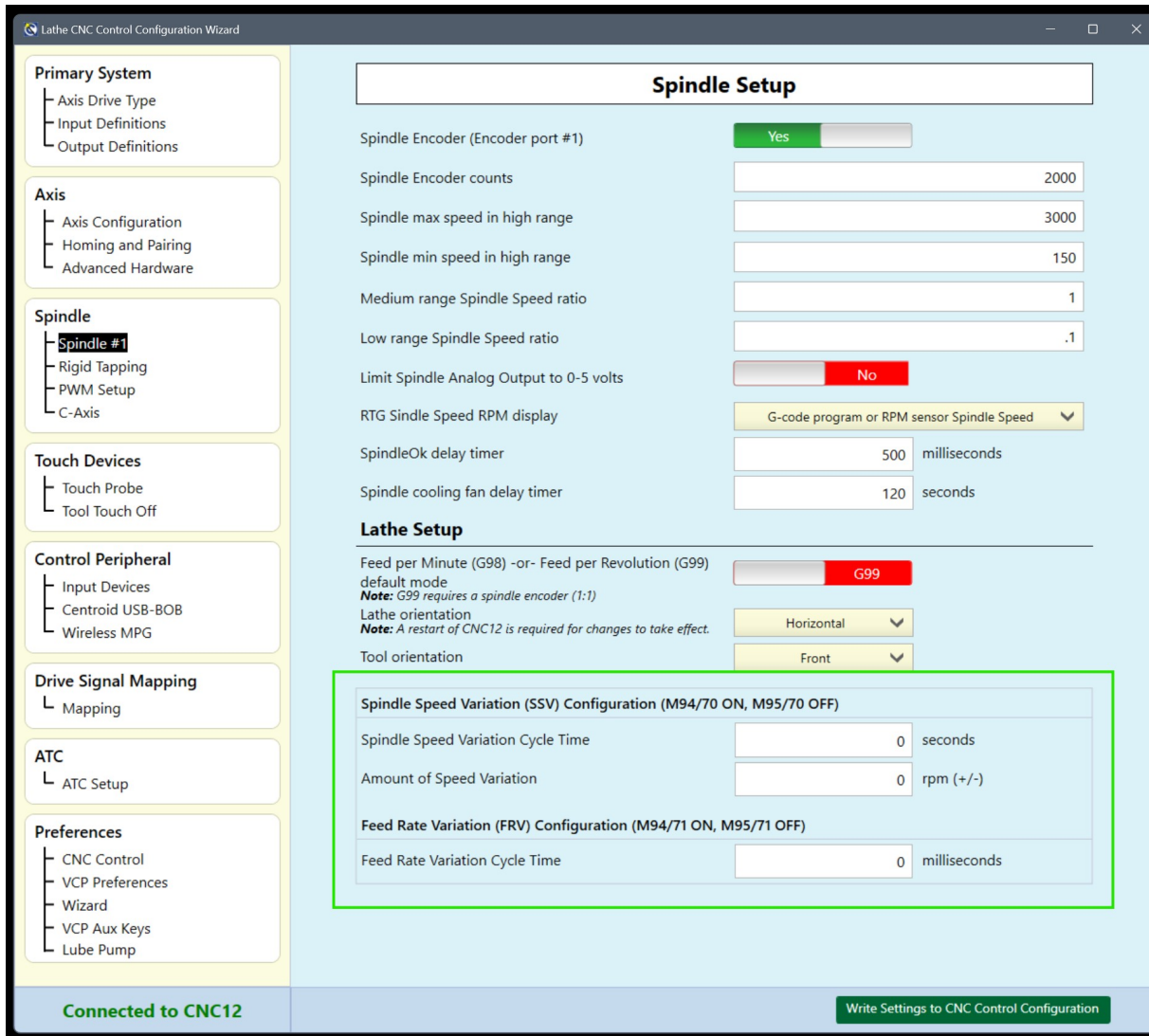
X Esc	Exact Stop F1	Precision Lathe F2	Contouring Lathe F3		Customize Presets F9	Save F10
----------	---------------------	--------------------------	---------------------------	--	----------------------------	-------------

Uwe Mattern made a video showing his unique way of using G64 on a Lathe Part.  
<https://youtu.be/HNRWMMaluq4?si=IFqPM2RwDXPVJ3tu>



9.) Lathe: Spindle Speed Variation and Feedrate Speed Variation is now a standard feature of Mill, Lathe , Router versions of CNC12. SSV and FRV are most useful for Lathe applications. Setup SSV and FRV parameters in the Wizard or via CNC12 Parameters.

Simple M code to turn either feature on and off.



Video showing SSV OFF vs. SSV ON <https://youtu.be/o6YsrY0MDoA>

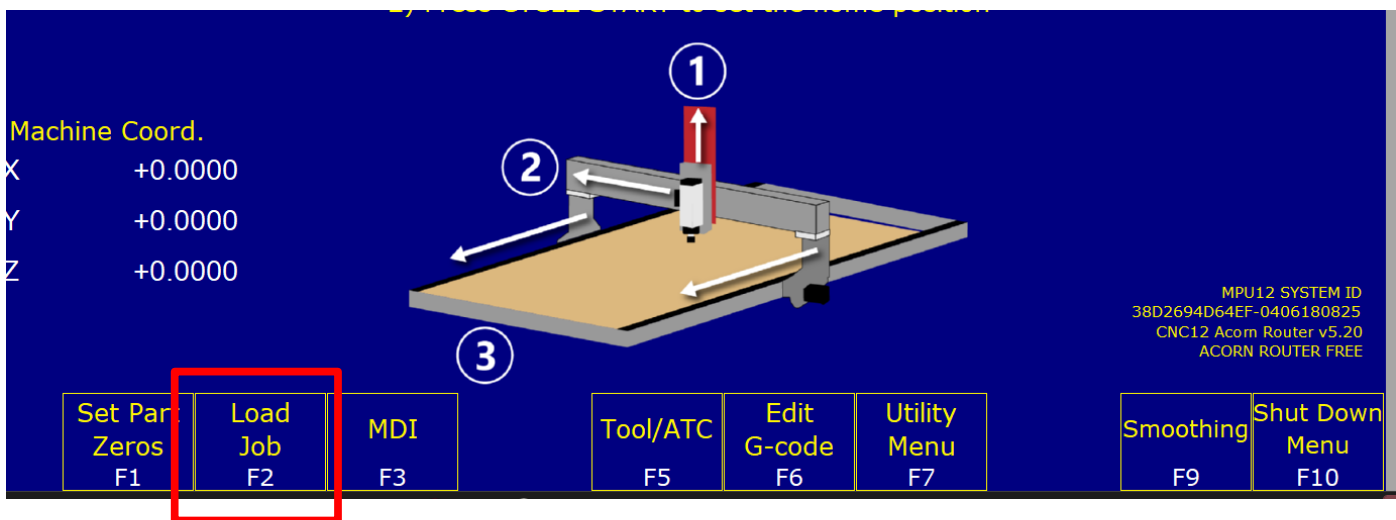


10.) CNC12 On screen text/words/labels/language file editing improvements.

Users can always change any words in CNC12 by simply editing the Centroid provided language.msg text file. What's new is now you can leave the Centroid provided language.msg text file ( the default file included with the CNC12 installation) alone and create a 'modifier' file called "user\_language.msg" this modifier file overrides the default file with just the changes you want.

The main advantage of this method is that it allows the user/oem to modify CNC12 words by creating a user language file that contains just the changes they want and easily copy those changes to a new installation of CNC12 without have to edit/re edit the Centroid default file, this allows Centroid to continue to make improvements and additions to the CNC12 language file with each new version while the user/oem can continue to use/reuse their same file with their custom changes with any new versions of CNC12. This new user language file makes modifying the words on CNC12 more convenient by saving time when installing multiple CNC12 installations and 'upgrading' to a new version of CNC12.

Any text present in CNC12 can be changed with the new file. For example, using the new Router version of CNC12, if we wanted to change the Function Key F2 words "Load Job" to "Open File"

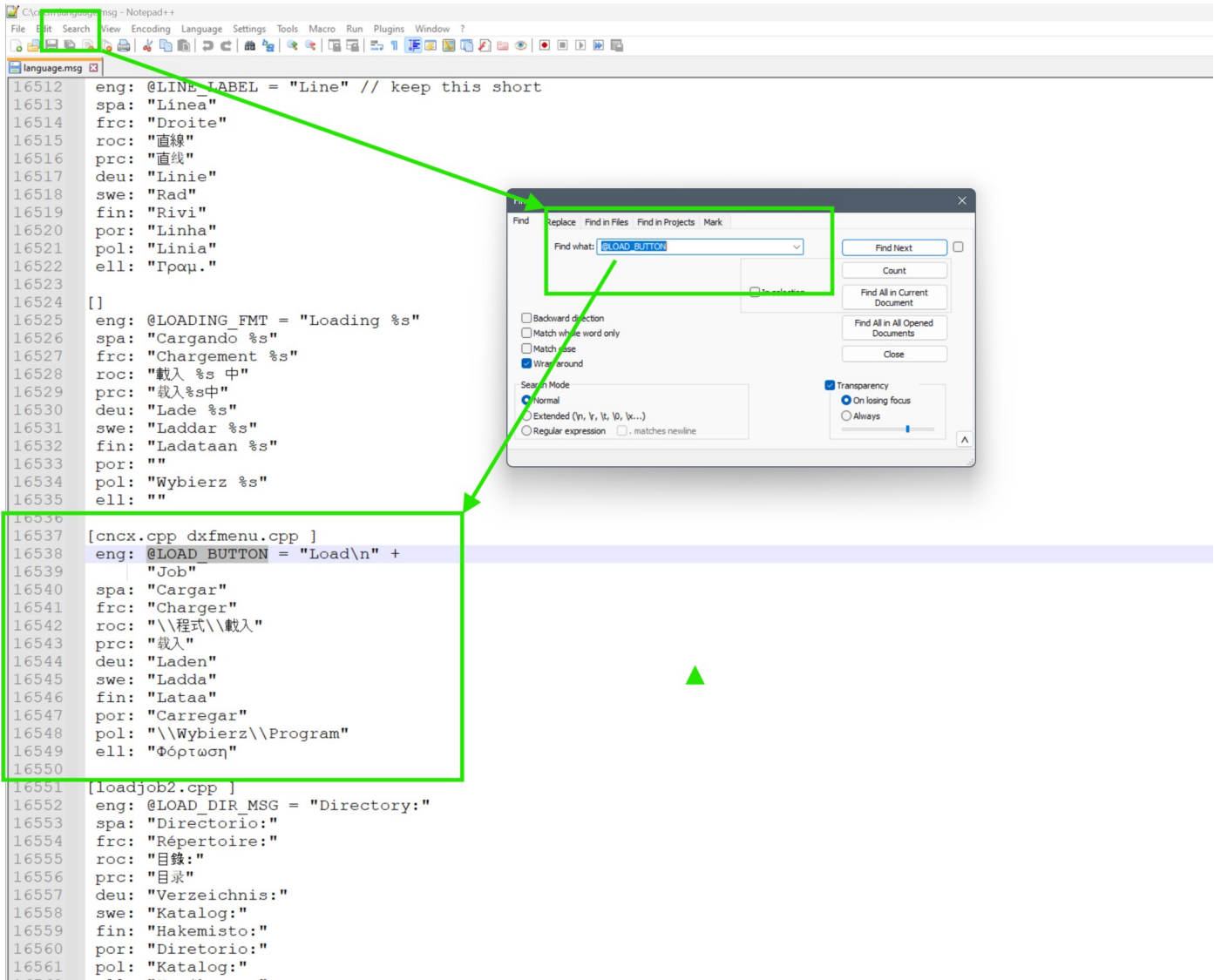


a.) Open Notepad ++ and create a new file. " user\_language.msg " and save it in the root CNC12 directory ( cncm or cnct etc)

b.) Open the stock c:\cncm\language.msg file

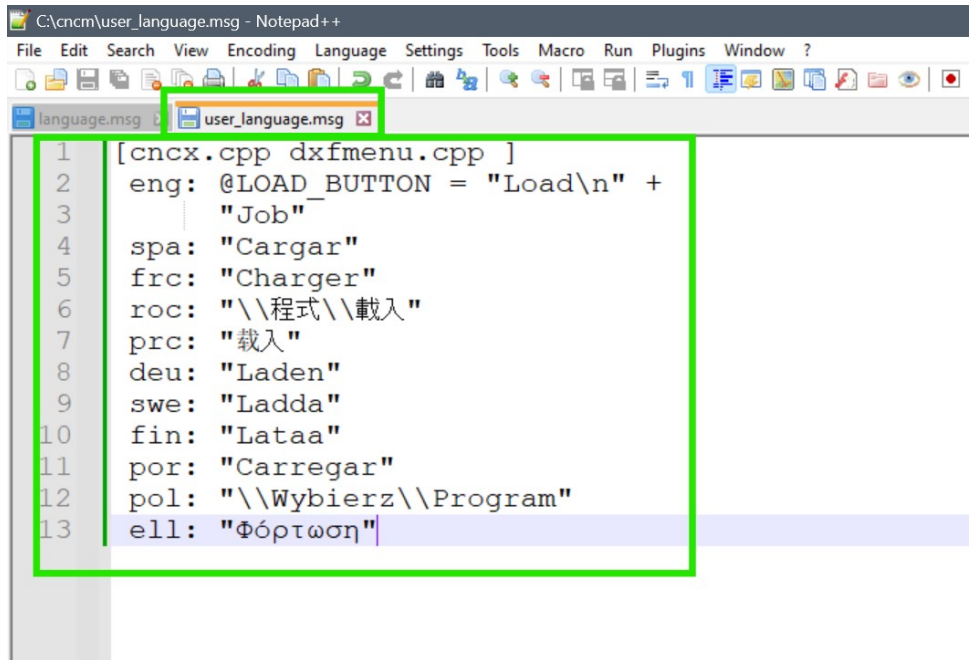
Use the search feature of Notepad ++ find the words you want to change.

In this case I searched on @LOAD\_BUTTON



c.) The format for the user\_language.msg file is the same as that of default Centroid language.msg file with the additional field of 'replacement:'.

Copy the entire Load Button section of the default Centroid language.msg file and paste it into the new user\_language.msg file, below is the Load Button section that I copied into the new user\_language.msg file



```
C:\cncm\user_language.msg - Notepad++
File Edit Search View Encoding Language Settings Tools Macro Run Plugins Window ?
language.msg user_language.msg
1 [cncx.cpp dxfmenu.cpp ]
2 eng: @LOAD_BUTTON = "Load\n" +
3     "Job"
4 spa: "Cargar"
5 frc: "Charger"
6 roc: "\\程式\\載入"
7 prc: "載入"
8 deu: "Laden"
9 swe: "Ladda"
10 fin: "Lataa"
11 por: "Carregar"
12 pol: "\\Wybierz\\Program"
13 ell: "Φόρτωση"
```

```
[cncx.cpp dxfmenu.cpp ]
eng: @LOAD_BUTTON = "Load\n" +
    "Job"
spa: "Cargar"
frc: "Charger"
roc: "\\程式\\載入"
prc: "載入"
deu: "Laden"
swe: "Ladda"
fin: "Lataa"
por: "Carregar"
pol: "\\Wybierz\\Program"
ell: "Φόρτωση"
```

d.) In the user\_language.msg file echo what you want to replace in this section by adding the "replacement:" line directly under the header. Highlighted in orange

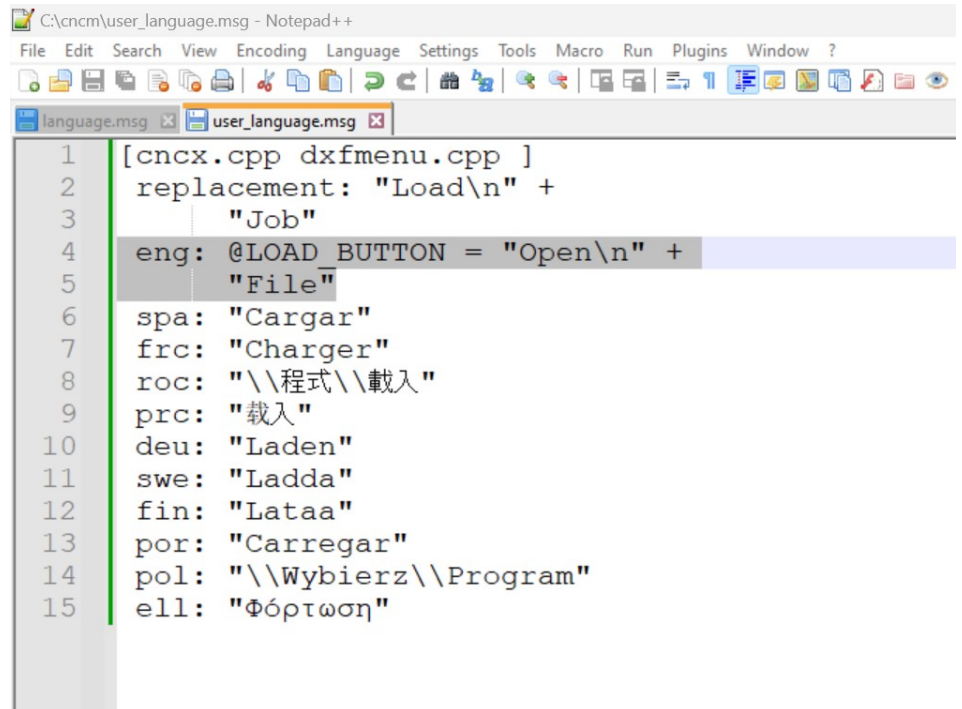
```
[cncx.cpp dxfmenu.cpp ]
replacement: "Load\n" +
    "Job"
eng: @LOAD_BUTTON = "Load\n" +
    "Job"
spa: "Cargar"
frc: "Charger"
.....
```

The replacement line we just added in orange above tells CNC12 that you want to replace the words Load Job "Load\n" + "Job"

Now edit the words in the Load Button section of the user\_language.msg file that you want to change.

In this example, the English words for the F2 Load Job Function key are changed from “Load Job” to “Open File” on the eng: line highlighted in yellow below

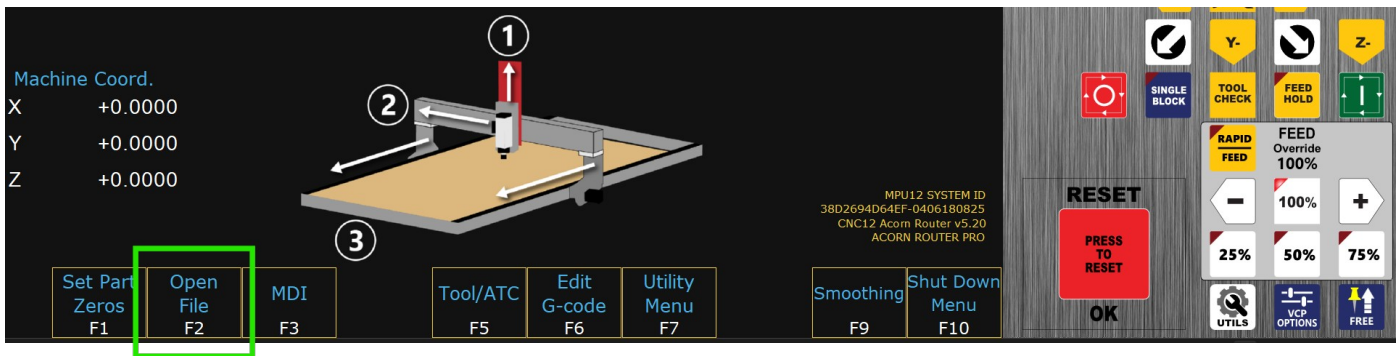
```
[cncx.cpp dxfmnu.cpp ]  
replacement: "Load\n" +  
  "Job"  
eng: @LOAD_BUTTON = "Open\n" +  
  "File"  
spa: "Cargar"  
frc: "Charger"  
roc: "\\程式\\載入"  
prc: "載入"  
deu: "Laden"  
swe: "Ladda"  
fin: "Lataa"  
por: "Carregar"  
pol: "\\Wybierz\\Program"  
ell: "Φόρτωση"
```



```
C:\cncm\user_language.msg - Notepad++  
File Edit Search View Encoding Language Settings Tools Macro Run Plugins Window ?  
language.msg user_language.msg  
1 [cncx.cpp dxfmnu.cpp ]  
2 replacement: "Load\n" +  
3   "Job"  
4 eng: @LOAD_BUTTON = "Open\n" +  
5   "File"  
6 spa: "Cargar"  
7 frc: "Charger"  
8 roc: "\\程式\\載入"  
9 prc: "載入"  
10 deu: "Laden"  
11 swe: "Ladda"  
12 fin: "Lataa"  
13 por: "Carregar"  
14 pol: "\\Wybierz\\Program"  
15 ell: "Φόρτωση"
```

Note: “\n” = new line (carriage return) so the word “File” appears on a new line (under the word “Open”)

Save the file and restart CNC12 and see the result.



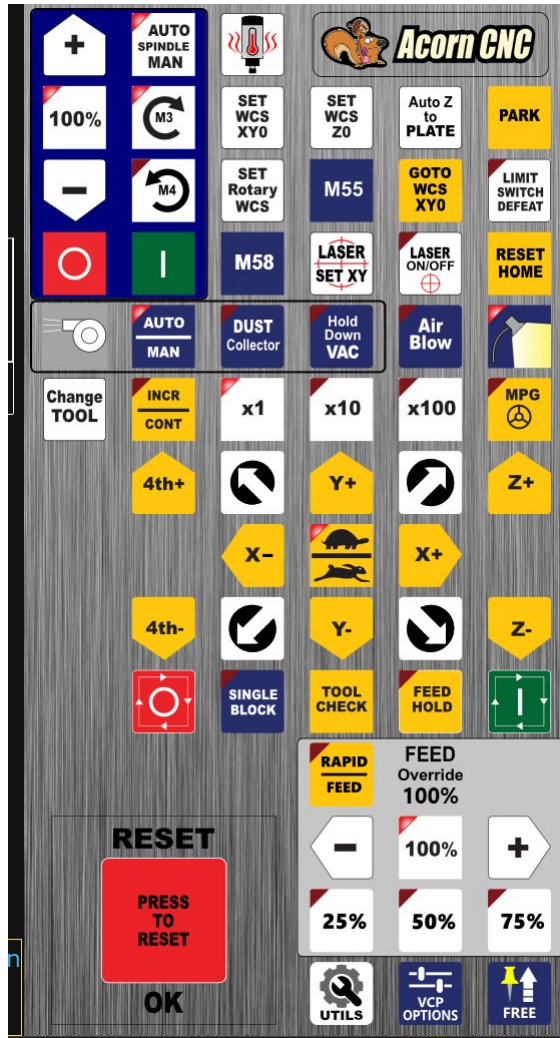
The advantage here is:

- Now we have a language file that tells CNC12 what to change without having to edit the Centroid default language file. This allows us to use the user\_language.msg file across all future versions of CNC12.
- Makes updating to a new version of CNC12 with your customizations easy. Simply copy your existing “user\_language.msg” file into the root CNC12 directory after a new CNC12 installation to have your changes take effect.
- Makes setting up multiple machine tools with a custom language file easy. Simply copy your “user\_language.msg” file into the root CNC12 directory after CNC12 installation.
- You can share your “user\_language.msg” with others using version v5.20+ of CNC12

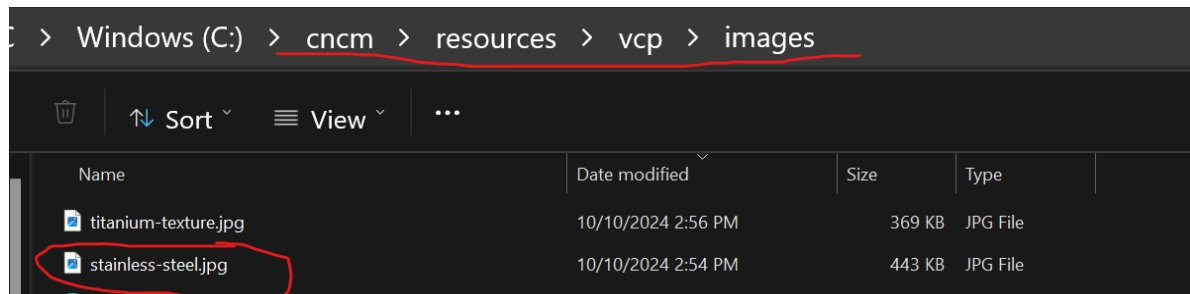


## 11.) Operators Virtual Control Panel (VCP) additions

The VCP now supports using an image as the background in addition to solid colors. Below is an example of the VCP using a .jpg as the background. The .JPG being used is a brushed stainless steel texture image.



This sample .jpg file is included in the c:\ncm\resources\vcp\images folder



To instruct the VCP to use an image as a background:

Edit the VCP skin and add the new background node <background>

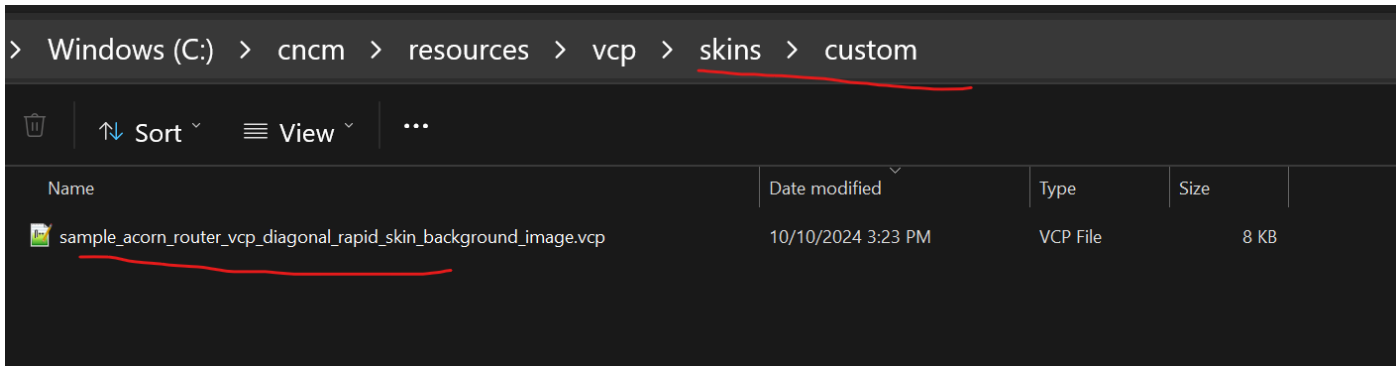
under <vcp\_skin> with the path to the image.

Highlighted in yellow below.

```
<vcp_skin>  
  <background>  
    c:\cncm\resources\vcpl\images\stainless-steel.jpg  
  </background>  
<border>  
  <column_span>2</column_span>  
  <column_start>1</column_start>  
  <fill>#00007F</fill>  
  <row_span>4</row_span>  
  <row_start>1</row_start>  
  <outline_color>#000000</outline_color>  
  <outline_thickness>2</outline_thickness>  
</border>
```

#### Notes:

- VCP image nodes can now accept .jpg and .png (and possibly other, untested) image formats.
- The image should be larger in size (pixels) than the VCP for best results.
- The image upper right corner will be placed in the upper right corner of the VCP background.
- Any parts of the image that are larger than the VCP are simply cropped.
- An example skin has been included in the skins\custom\folder



- The VCP group switching feature can now be triggered through parameters via

```
<trigger>
  <parameter number="750">
    <value>17</value>
  </parameter>
</trigger>
```

--- OR ---

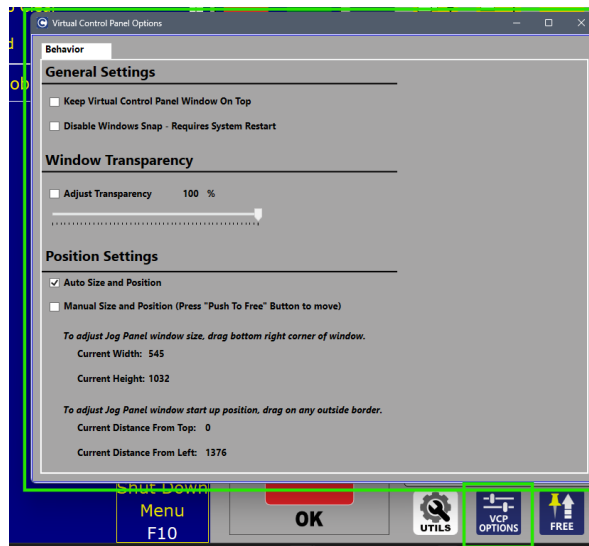
```
<trigger>
  <parameter number="750">
    <bit>0</bit>
  </parameter>
</trigger>
```

in addition to the previous switching methods of: clicking on the vcp button and using a PLC output number state.

- VCP Preferences Menu: When notifying the VCP that you have customized the VCP using the Custom VCP YES/NO selection. There is now a choice to “link” the Rapid and Feedrate Override control for both on one control. This sets the parameters so that your custom VCP can use the new way of linking feedrate and rapid rate override control if desired.

Custom VCP Configuration	
Customized VCP skin in use	<input checked="" type="checkbox"/> Yes
Custom VCP Enable Rapid Override	<input checked="" type="checkbox"/> Yes
Custom VCP Enable Rapid Feed Link	<input type="checkbox"/> No

- The VCP Options menu will now open based on location of the VCP's lower right corner so its easier to see and less likely to be under CNC12 itself.



12.) Organized and grouped PLC messages into types so its easier to identify and manage. Users can edit which PLC message falls into which category by editing \cncm\Plcmmsg.txt if desired.

Fault messages use the 9000's, (Note: legacy 452 and 406 are included here as well). 9000 default colors are: Red and white.

Error messages use the 8000's, default colors: Yellow and Black

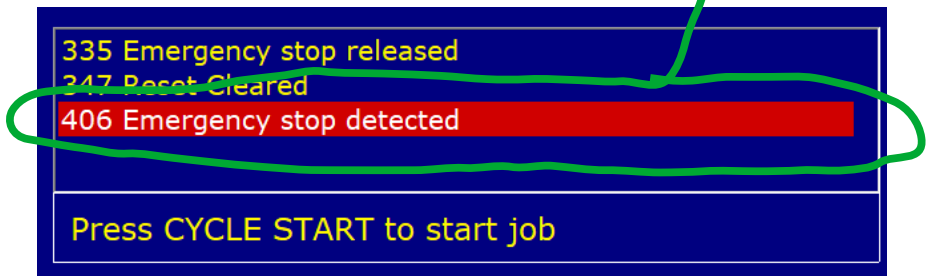
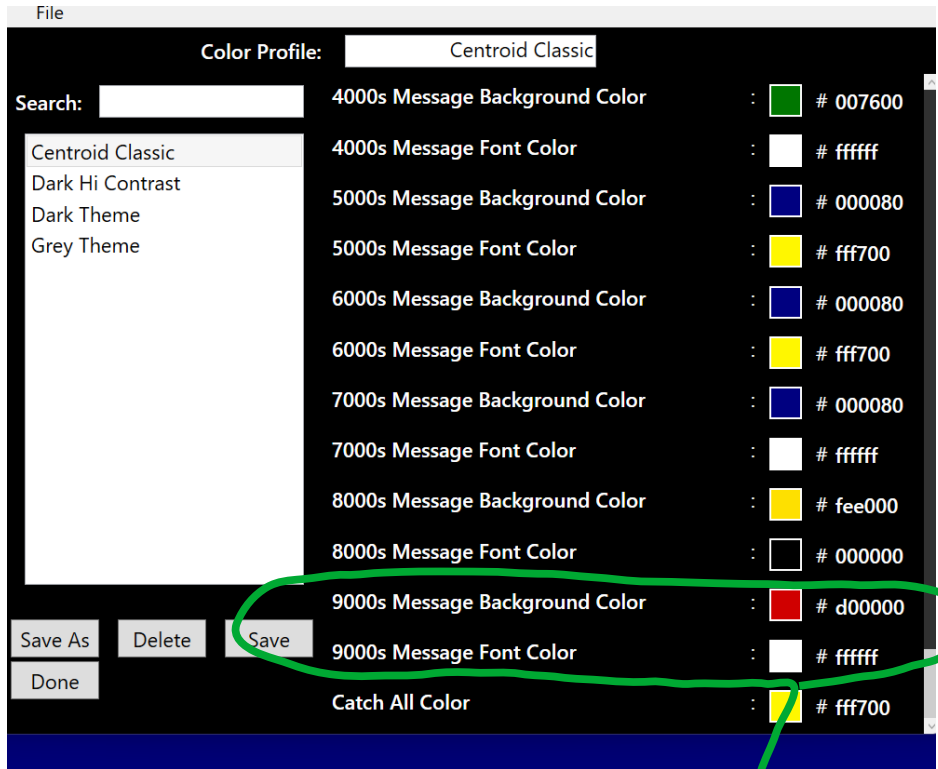
Warning messages use the 7000's, default colors: Blue and White

Other types of Messages use the 5000's default colors: Blue and Yellow

Ok Cleared type Messages use the 4000's  
4000 default colors: Green and White.

(Refer to CNC12 Operator Manuals for more information on all types of messages.)

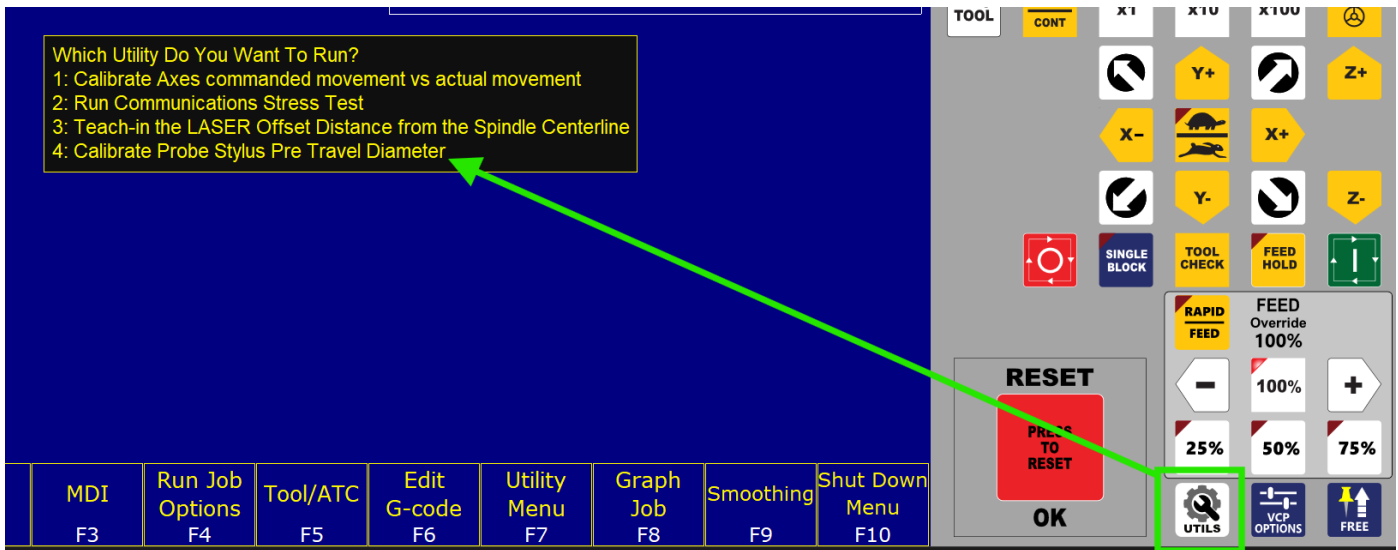
We also added them to the Color Picker so users have full control over the color of the types of messages CNC12 produces.



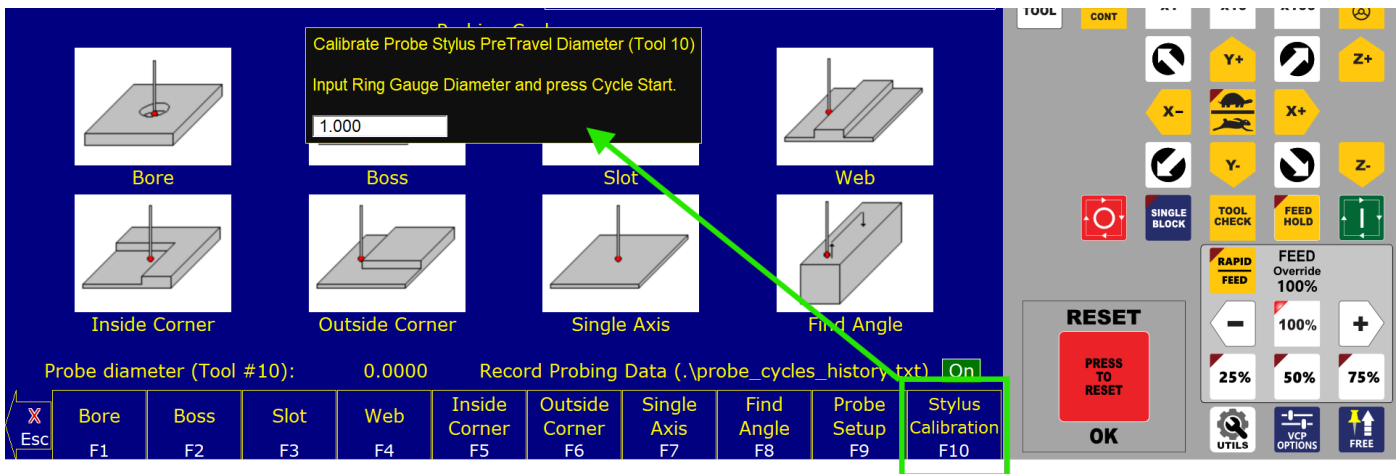


### 13.) Probing

- Added Probe Stylus Pre-travel calibration macro to the Probing menu and to the VCP Utils menu. This is a quick way to automatically calibrate and compensate for any probe stylus pre-travel amount.



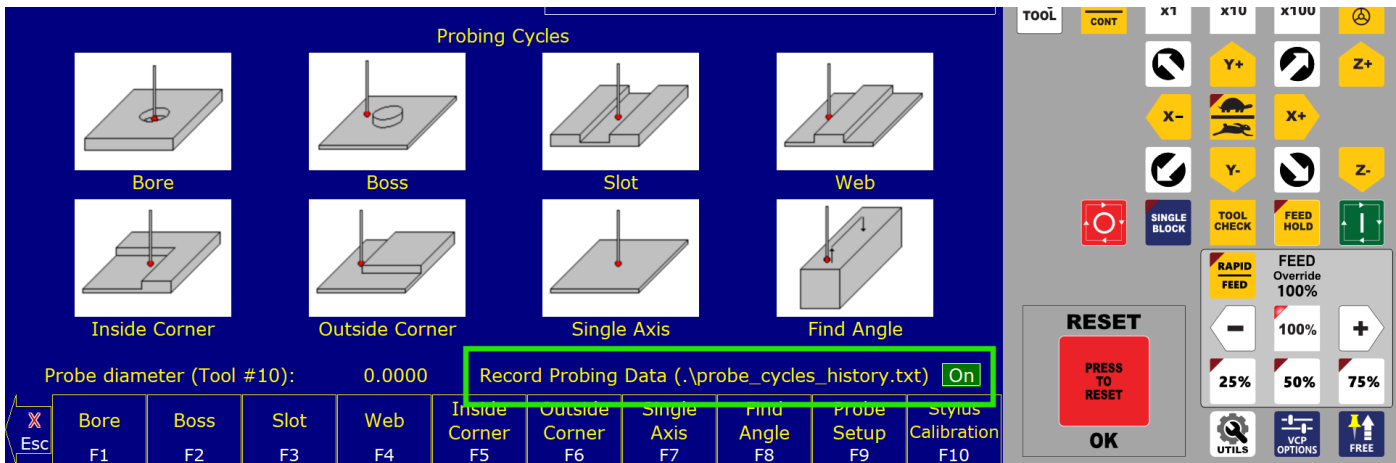
and directly in the probing menu as well.



- The Probing menu can now record probing data to a text file.

File name and location are: \cncm\probe\_cycles\_history.txt.

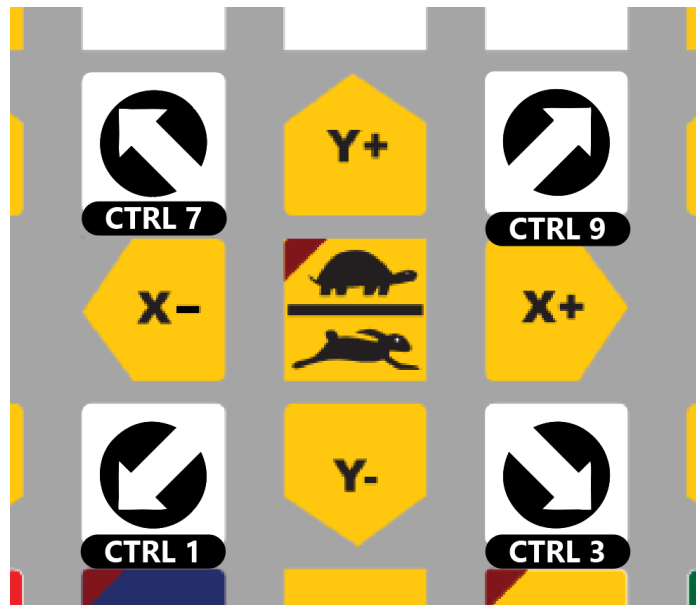
This feature can be turned on and off by clicking on the On/Off button in the Probing Menu.



Open the file with any text editor to see the gathered data, edit and manipulate as desired for importation into CAD/CAM systems, Spreadsheets etc.

## Other Changes, Additions and Improvements

- Moved the Manual Run toggle in Lathe Run Menu to right hand column to fix graphical issue
- Added checks to part and feed menus to ensure at least once axis is set before entering menu
- Utility menu now uses an HTML file to display information which is customizable by users and oem alike.
- Run Menus rewritten which allows the toggle buttons to be clickable with or without the use of function keys.
- Added Reference Mark Homing wizard option to Hickory
- Added "Return Z to Home" option in the router touch plate menu
- Combined the Homing and Travel Menu and the Axes Pairing Menu in the wizard. Software travel limits moved to the Axis Configuration menu. These changes also removed the need for parameter 414.
- Added M25 to the goto\_part\_xy\_zero.cnc macro which will raise Z before it moves to the XY0 position
- Keyboard Jogging:
  - a.) Now you can jog machines diagonally using the PC keyboard using numbers 1,7,9 and 3.



### PC Keyboard

CTRL 7 = X -, Y + at the same time

CTRL 9 = X +, Y+ at the same time

CTRL 3 = X +, Y - at the same time

CTRL 1 = X - , Y – at the same time

b.) Centroid default/Wizard generated PLC program change for improved keyboard jogging safety: When using PC Keyboard machine tool jogging, the PC Keyboard CTRL key activates the jog keys on the keyboard for safe Machine Axis Jogging via the keyboard. The CTRL key acts as a deadman safety switch. Preventing unintended movement of the machine tool while using the computer keyboard.



As always we recommend using the Centroid Operators control panel (either the Virtual one or the Hard panel), Your own hard wired Jog buttons ([see the USB-BOB web page for more information](#) on how to do that), a Hard wired MPG or the Wireless MPG ([see this page for more info on the Centroid Wireless MPG](#))

The Centroid PLC program is user editable, human readable, open source and the tools to customize/create/edit/debug/compile your own are free.



PC Keyboard Hot Keys Legend.

<b>+</b> Ctrl >	<b>AUTO SPINDLE MAN</b> Ctrl A	<b>AUX 1</b> Ctrl F1	<b>AUX 2</b> Ctrl F2	<b>AUX 3</b> Ctrl F3
<b>100%</b> Ctrl ?	<b>M3</b> Ctrl C	<b>AUX 4</b> Ctrl F4	<b>AUX 5</b> Ctrl F5	<b>AUX 6</b> Ctrl F6
<b>-</b> Ctrl <	<b>M4</b> Ctrl W	<b>AUX 7</b> Ctrl F7	<b>AUX 8</b> Ctrl F8	<b>AUX 9</b> Ctrl F9
<b>○</b> Ctrl Q	<b>I</b> Ctrl S	<b>AUTO MAN</b> Ctrl M	<b>DUST Collector</b> Ctrl N	<b>Hold Down VAC</b> Ctrl K
<b>INCR CONT</b> Ctrl I	<b>x1</b> Delete	<b>x10</b> Del\Ins	<b>x100</b> Insert	
<b>4th+</b> Ctrl Home	<b>Y+</b> Ctrl ↑	<b>Z+</b> Ctrl Pg Up		
	<b>X-</b> Ctrl ←	<b>X+</b> Ctrl →		
<b>4th-</b> Ctrl End	<b>Y-</b> Ctrl ↓	<b>Z-</b> Ctrl Pg Dn		
<b>○</b> Esc	<b>SINGLE BLOCK</b> Ctrl B	<b>TOOL CHECK</b> Ctrl T	<b>FEED HOLD</b> Space	<b>I</b> Alt S
	<b>-</b> Ctrl -	<b>100%</b> Ctrl \	<b>+</b> Ctrl +	
<b>PUSH TO PIN</b>		<b>PRINT</b>		

- Removed the license requirement for advanced macro functions.
- Deprecated API calls GetUserSystemVariable and GetUserStringSystemVariable. Added overloaded call GetSystemVariable to handle all system variables from #1 to #31999
- Added P181 to specify an additional ratio of a full motor revolution that an axis is allowed to move while searching for an index pulse before faulting out.
- Added bit 1 (add 2) P425 "Unhome machine after estop" to be able to unhome the machine after the reset button on the vcp is pressed.
- Added an "InverseTimeMode" to display 3D surface feedrate when G93.1 is in use. Rotary axes will display deg/min for rotary-only moves while G93.1 is not in use.
- Added Plasma Restart Mode debug mode when P389 == 1
- Increased which axes can be reversed by Plasma Restart Mode. Rotary jobs now work with Plasma restart traverse buttons
- Added horizontal auto scrolling for menus that don't fit horizontally on the screen. This is most useful when dragging/dropping inputs/outputs and multiple expansion boards are hooked up
- Limit switches will implicitly be set as home switches if no home or home limit switches are found in the wizard
- Removed Parameter 821 as it was unused for it's stated purpose
- Added dedicated Rack engage speed for Hole Rackmount ATC type
- Split Fork Engage/Disengage speed for Fork Rackmount and Sliding Carousel ATC types
- Removed unnecessary truncation of input/output names when dragging/dropping IO
- Added Simple PLC Diagnostics to Hickory
- Fixed issue where the SV\_?\_AXIS\_DRIVE\_ONLINE system variable, where ? is a single character axis label ABCXYZUVW, was not working correctly for ACDC drive types.
- PLC Installer: the button "Install - and keep the existing machine configuration" will not be enabled/visible if an mpu.plc program is not found in the corresponding cnct or cncm directory. It is assumed that if there is no mpu.plc program found, then this is a new install so the choice to keep the existing machine configuration would be a mistake. In addition, when choosing the "Install - and keep the existing machine configuration" option, we now copy over any plcmsg.txt since it is tightly coupled to mpu.plc.
- OEM use for custom CNC12 screen. P113 Bit 2
- Add 2 will hide F5 and F7 in CNC12 Router
- Add 4 will hide F1, F2, and F3 (the measurement buttons)
- Add 8 will hide Press Shift F12 help messages
- M222 is a new M code that uses key input. M222 is similar to M224 but takes any key input as an input. The file cncm\system\setup\_key\_defines.cnc lists the keys and the values that end up in the requested variable.

For example:

*M222 #101 "Verify Tool Touch Off device is functioning properly.\nPress any key to continue."*

use covered

- Add a new Mcode: M221. M221 is the same as M225 but doesn't stop the job.

*M221 #100 "Homing X axis"* (displays "homing X axis" as the g code still runs, does this until the job ends, the timer runs out or another message is triggered.)

- Added a P0 value to M115/116, M125/126 will move the probe until any touch device input is triggered.

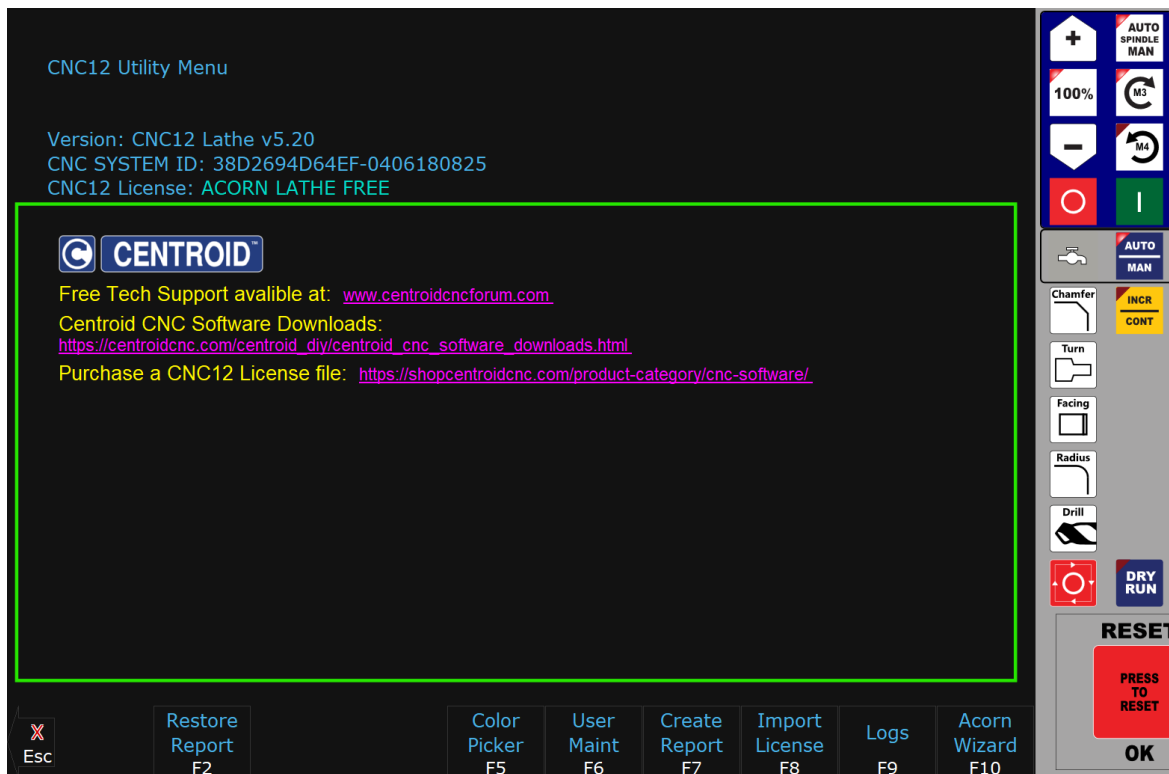
*M115/X P0 F10* Move X axis in the negative direction until any touch device input is triggered. Works for any touch device, any touch input is triggered.

- Added Inline hyperlinks for Message Macros including but not limited to: M222,221,224,225  
you can now use the html a href tag directly within the message m code.

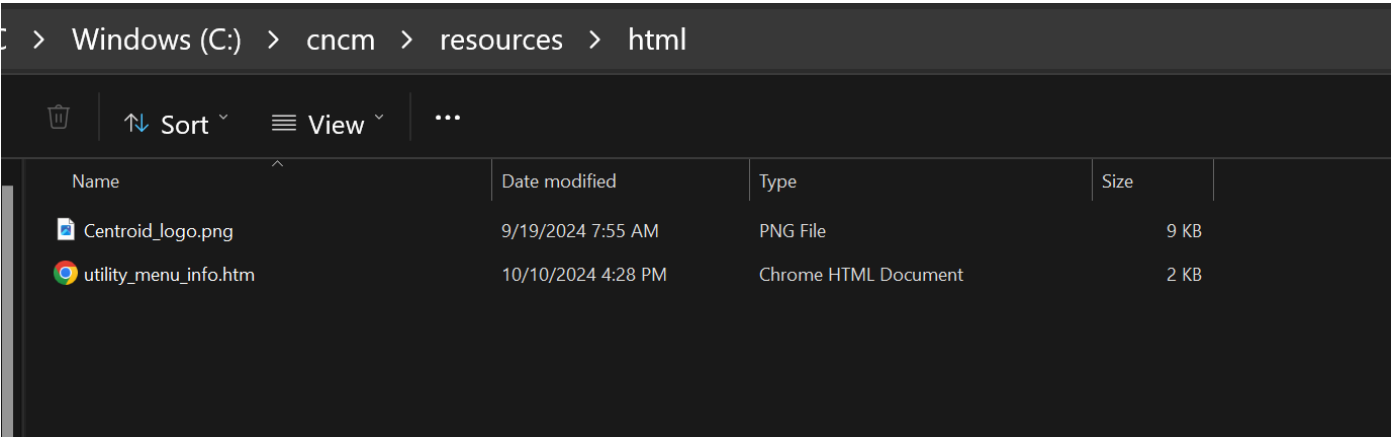
*M221 #100 "<a href='centroidcnc.com'>This will take you to the best website ever</a>"*

only one hyper link per line. \n for a new line.

- Utility Menu now displays user editable HTML file. Typically used by OEM's to present their specific messaging. You can change any words, add images and hyperlinks.



Edit the file c:\cncm\utility\_menu\_info.htm , restart CNC12 to see the changes.

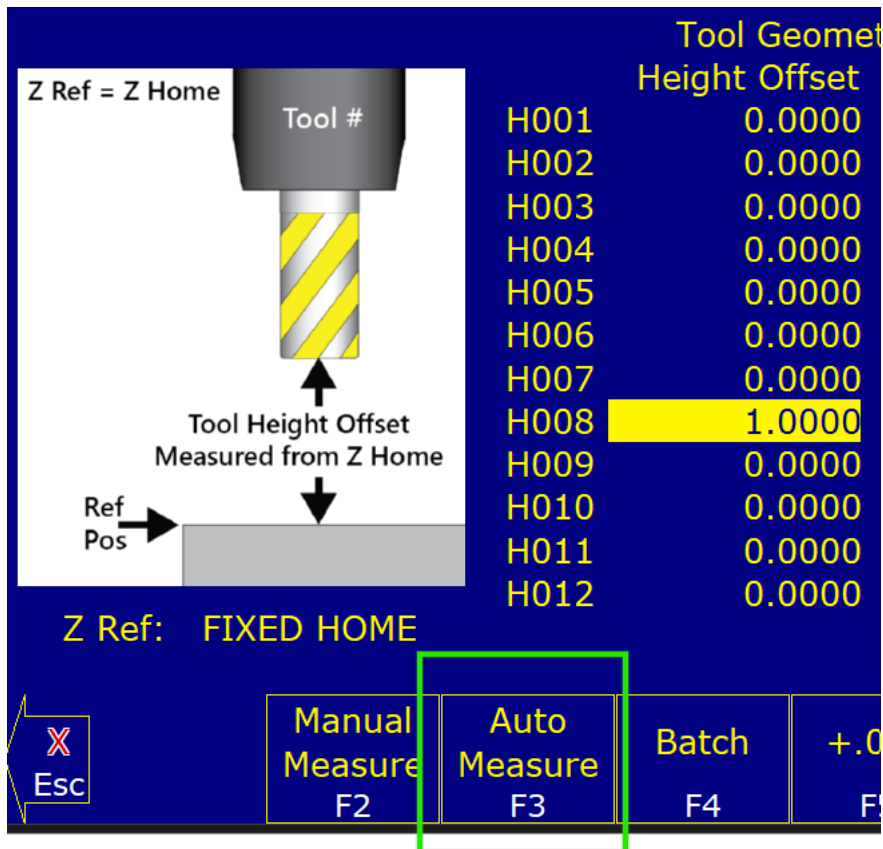


- Added Support for the Applied Motion EtherCat open loop stepper drive STF06-EC to Hickory CNC12  
This drive is only recommended use is for accessory motion (think ATC Carousel or Rack in/out type applications) this drive is NOT for cutting motion.

- New EtherCat Leadshine EL7 hex file improvements for better startup, axis pairing and homing performance.

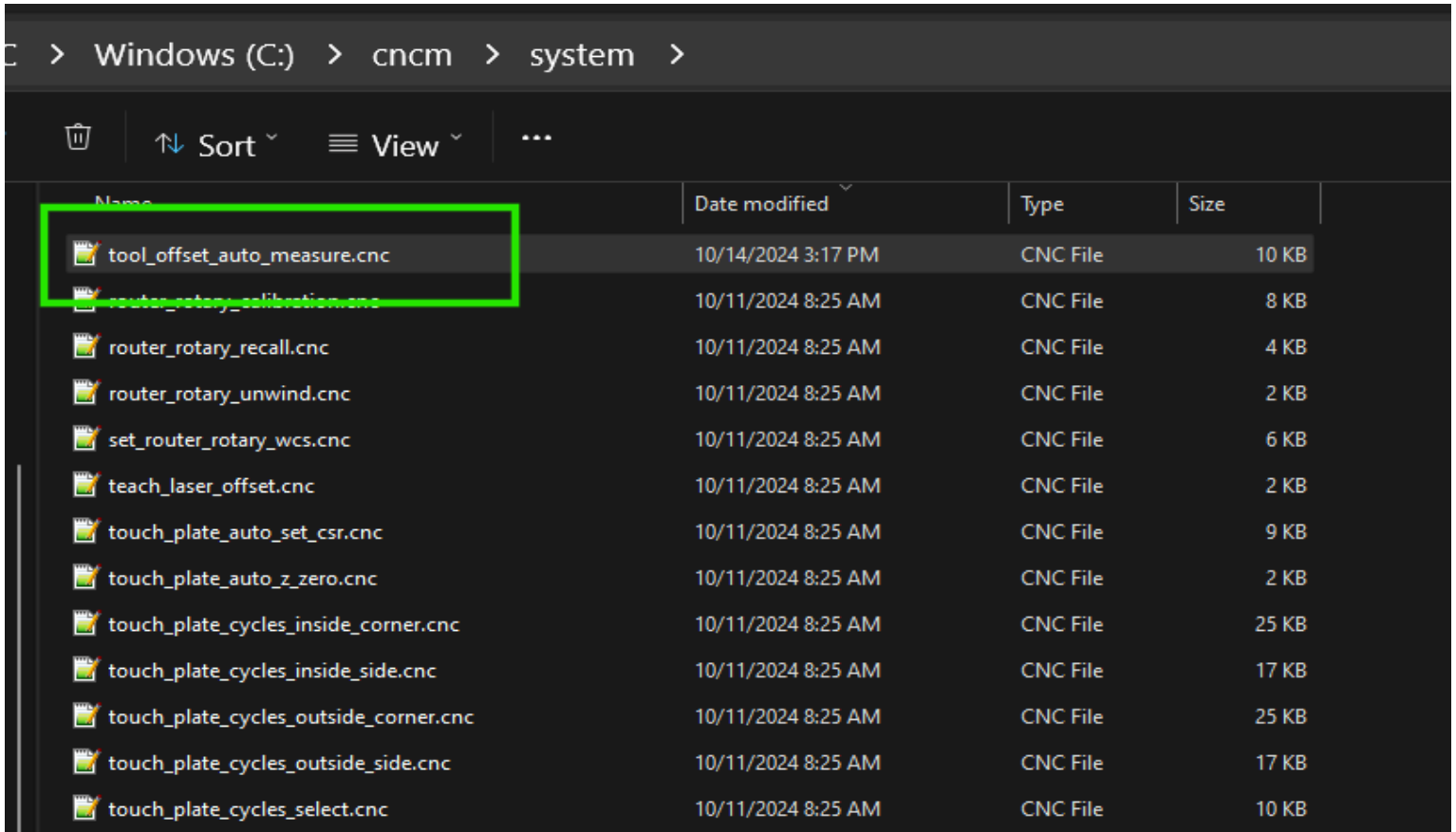
- Improved the Wizards Print (CTRL+P) feature so the Wizard screens print nearly a full page width of 8.5x11 piece of paper

- The Function Key F3 Auto Measure in the Tool Library is now a user editable Macro.



The CNC12 Router Version of this macro ask the operator if they would like to Automatically go to the Tool Touch Off fixed position or Manually Jog the Tool over the TT. Auto is used for measuring tools that can come straight down on the TT's location. Manually jog the tool over the TT position and then measure from that point is used for tools like Fly Cutters that can not come straight down on the TT for Auto Height Measurement. Manual in this case is actually semi automatic, the user simply jogs the Fly Cutter to a good XY position to measure the tool height from and then presses cycle start and the Auto Tool Height measurement TT cycle takes over.

Anyone can now make F3 do what every they want, just edit the macro located in the c:\cncm\system\tool\_offset\_auto\_measure.cnc





Related to that, we have been working through as many Function keys inside of CNC12 to break them out to user editable macros so they can be customized. These new macros can be found here. C:\cncm\system

This PC > Windows (C:) > cncm > system >

Name	Date modified	Type	Size
tool_offset_auto_measure.cnc	10/14/2024 3:17 PM	CNC File	10 KB
router_rotary_calibration.cnc	10/11/2024 8:25 AM	CNC File	8 KB
router_rotary_recall.cnc	10/11/2024 8:25 AM	CNC File	4 KB
router_rotary_unwind.cnc	10/11/2024 8:25 AM	CNC File	2 KB
set_router_rotary_wcs.cnc	10/11/2024 8:25 AM	CNC File	6 KB
teach_laser_offset.cnc	10/11/2024 8:25 AM	CNC File	2 KB
touch_plate_auto_set_csr.cnc	10/11/2024 8:25 AM	CNC File	9 KB
touch_plate_auto_z_zero.cnc	10/11/2024 8:25 AM	CNC File	2 KB
touch_plate_cycles_inside_corner.cnc	10/11/2024 8:25 AM	CNC File	25 KB
touch_plate_cycles_inside_side.cnc	10/11/2024 8:25 AM	CNC File	17 KB
touch_plate_cycles_outside_corner.cnc	10/11/2024 8:25 AM	CNC File	25 KB
touch_plate_cycles_outside_side.cnc	10/11/2024 8:25 AM	CNC File	17 KB
touch_plate_cycles_select.cnc	10/11/2024 8:25 AM	CNC File	10 KB
touch_plate_get_variables.cnc	10/11/2024 8:25 AM	CNC File	4 KB
touch_plate_move.cnc	10/11/2024 8:25 AM	CNC File	8 KB
Axis_Calibration.cnc	10/11/2024 8:25 AM	CNC File	10 KB
goto_machine_xy_zero.cnc	10/11/2024 8:25 AM	CNC File	2 KB
goto_machine_xyz_zero.cnc	10/11/2024 8:25 AM	CNC File	2 KB
goto_part_xy_zero.cnc	10/11/2024 8:25 AM	CNC File	2 KB
home_machine_cycle.cnc	10/11/2024 8:25 AM	CNC File	2 KB
homing_relay_square.cnc	10/11/2024 8:25 AM	CNC File	18 KB
MPGmacro1.mac	10/11/2024 8:25 AM	MAC File	2 KB
MPGmacro2.mac	10/11/2024 8:25 AM	MAC File	2 KB
MPGmacro3.mac	10/11/2024 8:25 AM	MAC File	2 KB
MPGmacro4.mac	10/11/2024 8:25 AM	MAC File	2 KB
perform_park.cnc	10/11/2024 8:25 AM	CNC File	2 KB
perform_spindle_warmup.cnc	10/11/2024 8:25 AM	CNC File	3 KB
perform_tool_change.cnc	10/11/2024 8:25 AM	CNC File	3 KB
probe_check_configuration.cnc	10/11/2024 8:25 AM	CNC File	6 KB
probe_stylus_calibration.cnc	10/11/2024 8:25 AM	CNC File	5 KB
select_laser_macros.cnc	10/11/2024 8:25 AM	CNC File	3 KB
set_laser_csr.cnc	10/11/2024 8:25 AM	CNC File	2 KB
set_part_all_zero.cnc	10/11/2024 8:25 AM	CNC File	4 KB
set_part_axis_zero.cnc	10/11/2024 8:25 AM	CNC File	7 KB
set_part_x_zero.cnc	10/11/2024 8:25 AM	CNC File	2 KB
set_part_xy_zero.cnc	10/11/2024 8:25 AM	CNC File	2 KB
set_part_y_zero.cnc	10/11/2024 8:25 AM	CNC File	2 KB
set_part_z_zero.cnc	10/11/2024 8:25 AM	CNC File	2 KB
set_tool_number.cnc	10/11/2024 8:25 AM	CNC File	2 KB
setup_defines.cnc	10/11/2024 8:25 AM	CNC File	19 KB
tool_library_tool_change.cnc	10/11/2024 8:25 AM	CNC File	2 KB
tool_offset_man_measure.cnc	10/11/2024 8:25 AM	CNC File	2 KB

- If no ATC F7 = Operator Manual Tool Change

Put the tool you want in the spindle and let CNC12 know which tool you loaded in the spindle manually.

WCS #1 (G54) Current Position (Inches) Job Name: NO\_JOB\_LOADED.cnc  
Tool: T1 H102 20001  
Feedrate: 100% 0.0 ipm Part Cnt: 0  
Spindle: 0 A Part #: 0  
Rapid Rate: 100% 0:00:05

335 Emergency stop released  
347 Reset Cleared  
490 Reset Initiated, Press Reset to Clear  
347 Reset Cleared  
Press CYCLE CANCEL to cancel

Operator Manual Tool Change  
Enter Tool Number and press Cycle Start to continue.  
Input desired Tool #:

Tool #	H	D	D
H005	0.0000	D005	0.0500
H006	0.0000	D006	0.2500
H007	0.0000	D007	0.0000
H008	0.0000	D008	0.0000
H009	0.0000	D009	0.0000
H010	0.0000	D010	0.0000
H011	0.0000	D011	0.0000
H012	0.0000	D012	0.0000

Z Ref: FIXED HOME

Change Tool F7

If ATC present F7 Auto Tool Change will put and get tools automatically

WCS #1 (G54) Current Position (Inches) Job Name: NO\_JOB\_LOADED.cnc  
Tool: T1 H102 20001  
Feedrate: 100% 0.0 ipm Part Cnt: 0  
Spindle: 0 A Part #: 0  
Rapid Rate: 100% 0:02:17

347 Reset Cleared  
490 Reset Initiated, Press Reset to Clear  
347 Reset Cleared  
313 Waiting for dwell time  
Press CYCLE CANCEL to cancel

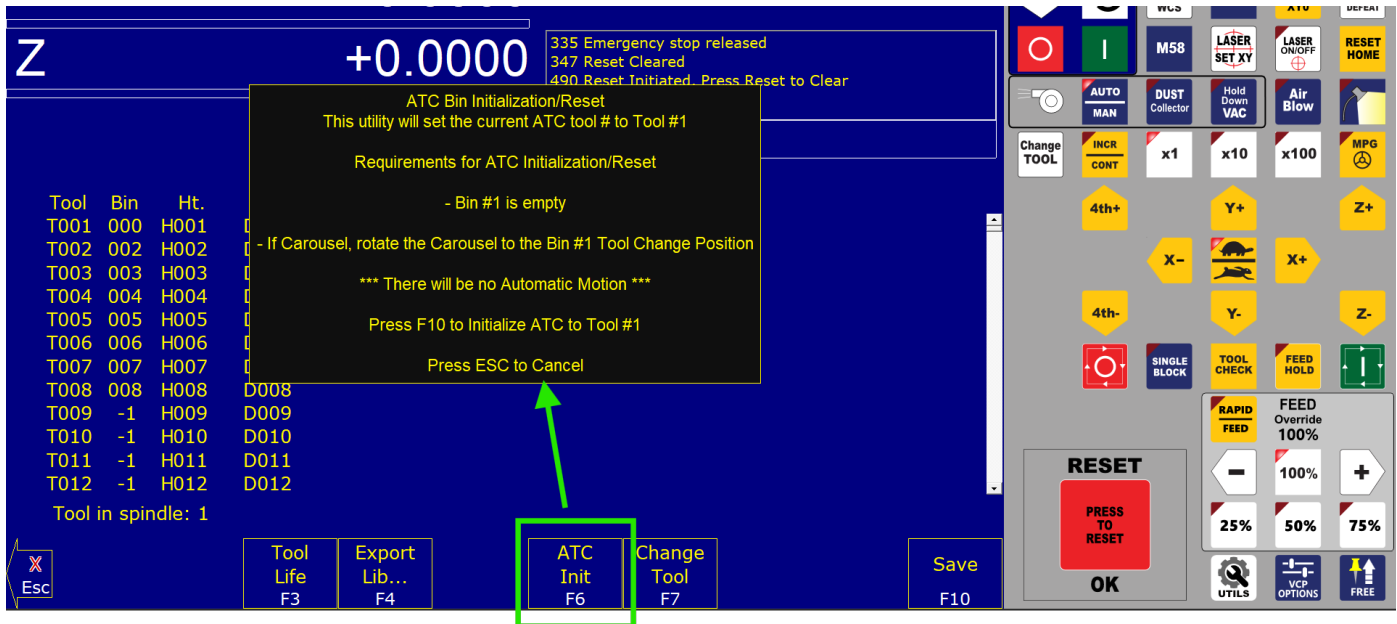
Perform Automatic Tool Change  
Enter Tool Number and Press Cycle Start.  
(Machine will Put away the tool currently in the spindle and go Get the specified tool number.)  
Get Tool #:

Tool #	H	D	D
H006	0.0000	D006	0.2500
H007	0.0000	D007	0.0000
H008	0.0000	D008	0.0000
H009	0.0000	D009	0.0000
H010	0.0000	D010	0.0000
H011	0.0000	D011	0.0000
H012	0.0000	D012	0.0000

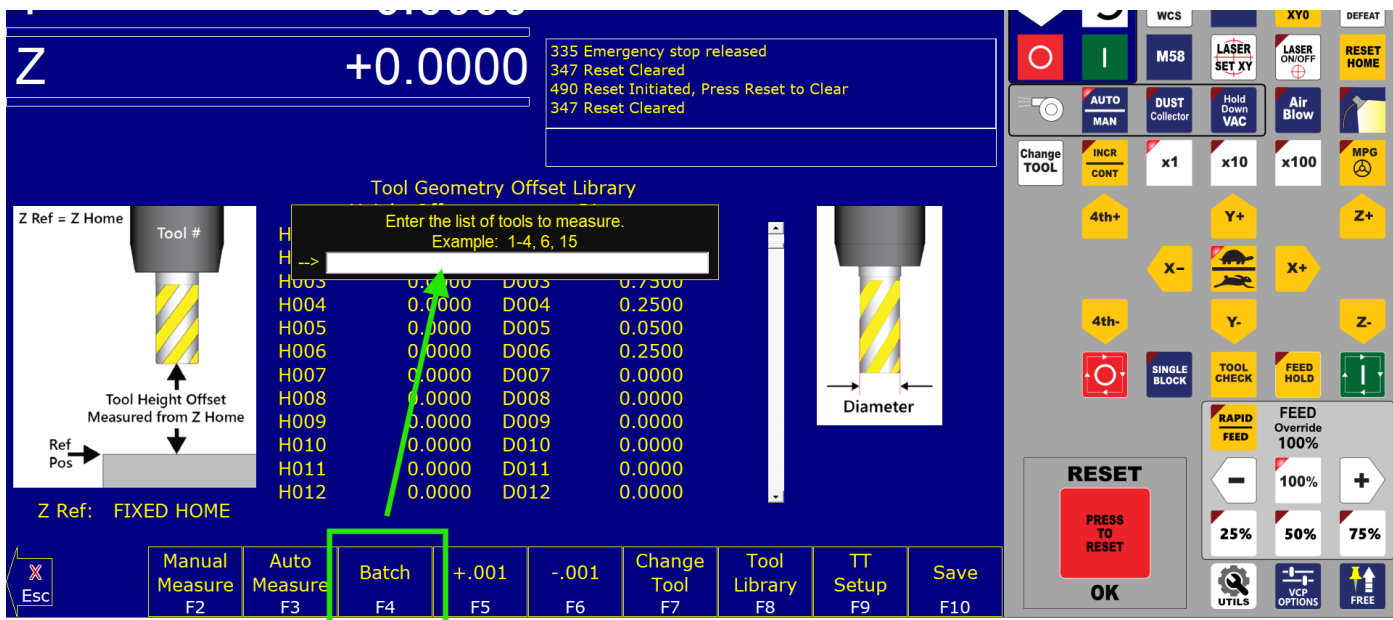
Z Ref: FIXED HOME

Change Tool F7

- Revised copy on the F6 ATC initialization button for clarity.



- With ATC and Fixed TT Batch Measure will automatically measure one or more tools at a time fully automatically.



Examples:

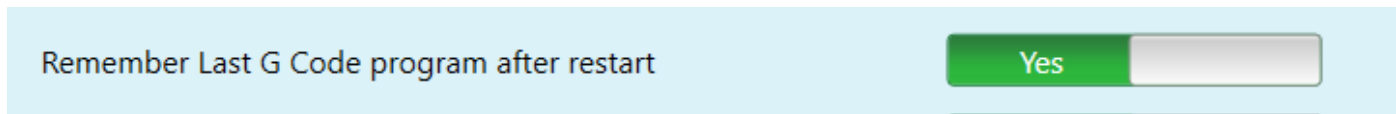
- Enter "1" and press cycle start and machine will put current Tool away , pick up Tool #1 and go touch it off the fixed TT.

- Enter "1,3,5,6" and press cycle start and machine will put current tool away , pick up Tool #1 and go touch it off the fixed TT, then machine will put Tool #1 away, pick up Tool #3 and go touch it off the fixed TT, then machine will put Tool #3 away, pick up Tool #5 and go touch it off the fixed TT, then machine will put Tool #5 away, pick up Tool #6 and go touch it off the fixed TT

- Enter "1-4" and press cycle start and machine will put current tool away , pick up Tool #1 and go touch it off the fixed TT, then machine will put Tool #1 away, pick up Tool #2 and go touch it off the fixed TT, then

machine will put Tool #2 away, pick up Tool #3 and go touch it off the fixed TT, then machine will put Tool #3 away, pick up Tool #4 and go touch it off the fixed TT

- Changed this default setting to “Yes” so CNC12 will keep the current job loaded even in the event of a power loss. (Change this slider to NO if you don’t want this feature).



Added Default Jobs (g code programs) that will be loaded upon the very first start up of CNC12, instead of defaulting to “no\_job\_loaded.cnc” Any subsequent start/restart of CNC12 will of course remember what ever job you had loaded upon the next start/restart of CNC12.

These jobs below only load the very first time CNC12 starts so new users have a good g-code program to graph/edit/run etc the very first time they start and use CNC12.

Lathe default first time only job: PAWN.cnc (no encoder required version)

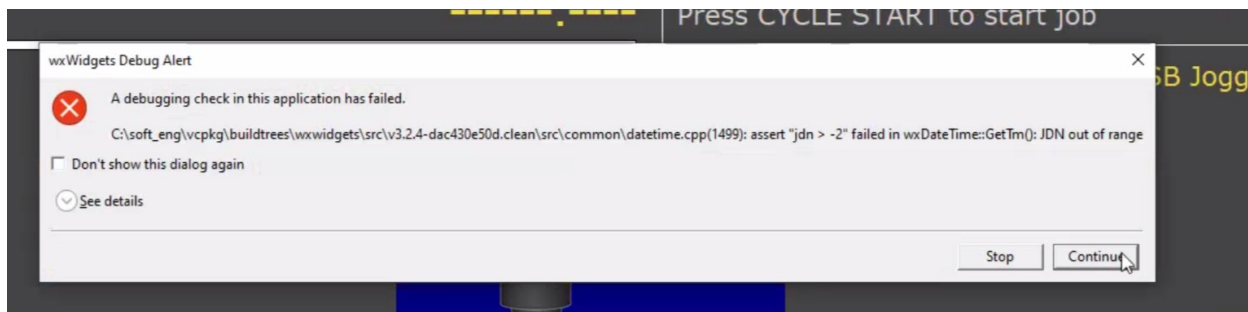
Plasma default first time only job: Bear23x23.tap

Mill default first time only job: sample\_atlanta.cnc

Router default first time only job: sample\_sign\_smooth\_on.cnc

If you want “no\_job\_loaded.cnc” to be loaded automatically upon a CNC12 start/restart change the slider to No.

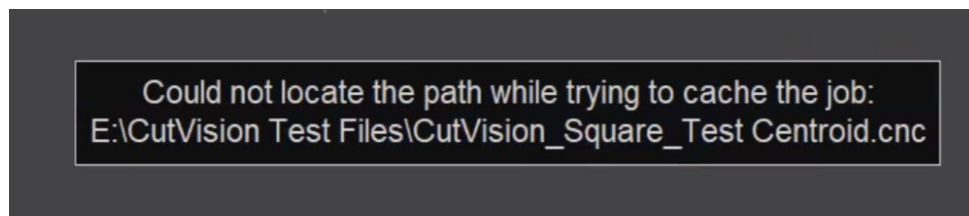
- Fixed a rare but annoying windows wxwidgets message related to loading a g code file remotely.



When Setting Parameter 4 = 26 (default is 25) this will sometimes cause a windows wxwidgets error message when the date/time stamp or path of the remote file was missing or corrupted.

When using P4=26, CNC12 caches a copy of a g-code program locally from a remote drive, this feature was useful back in the day of small hard drive space, not so much anymore. ( Recommended setting for Parameter 4 these days is 25 and the default is P4=25.

New CNC12 Message added to let the operator know that the remote file is missing or corrupted etc.

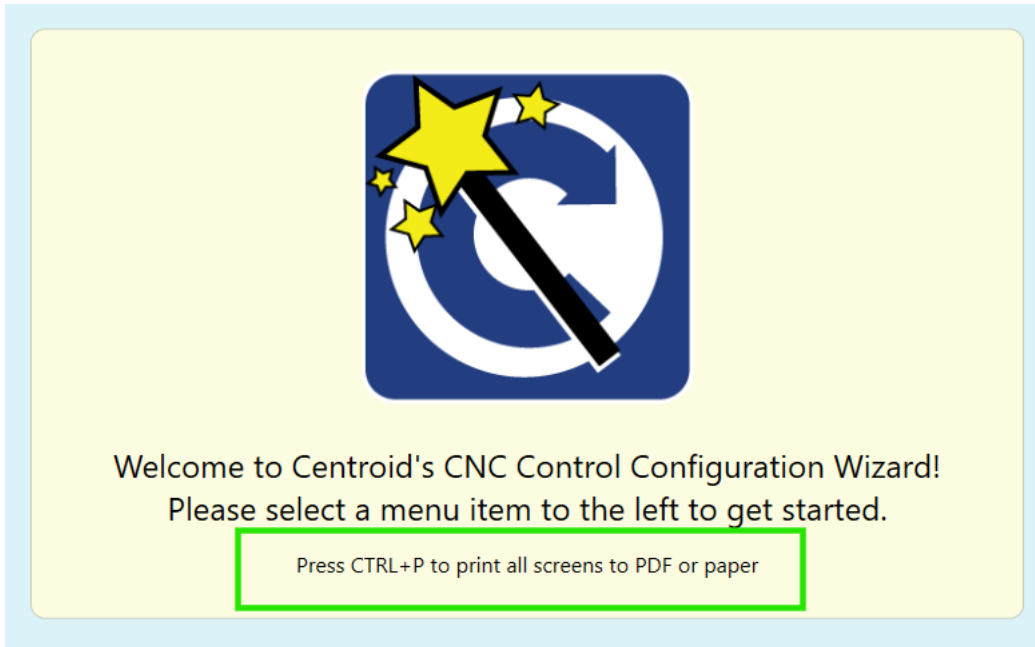


More information on Parameter 4 can be found in the CNC12 Operator manuals.





- Added a note on the Wizard start up screen about undocumented Wizard feature "Print to paper or PDF"



- Moved the Wizard Axis travel limits into the Axis Configuration menu

Router CNC Control Configuration Wizard

**Primary System**

- Axis Drive Type
- Input Definitions
- Output Definitions

**Axis**

- Axis Configuration**
- Homing and Pairing
- Advanced Hardware
- Rotary Setup

**Spindle**

- Spindle #1
- Rigid Tapping
- PWM Setup

**Touch Devices**

- Touch Probe
- Tool Touch Off
- Touch Plate

**Control Peripheral**

- Input Devices
- Centroid USB-BOB
- Wireless MPG

**Drive Signal Mapping**

- Mapping

**ATC**

- ATC Setup

**Preferences**

- CNC Control
- VCP Preferences

**Axis Configuration**

	Axis 1	Axis 2	Axis 3	Axis 4
Linear or Rotary	Linear	Linear	Linear	Rotary
Label	X	Y	Z	N
Steps / Revolution	2000	2000	2000	1600
Overall Turns Ratio <small>mm/turn or deg/rev or revs/turn</small>	1.25	1.25	2.5	4
Lash Comp. <small>millimeters</small>	0	0	0	0
Max Rate <small>mm/min or deg/min or rev/min</small>	1200	1200	600	360
Fast Jog Plus Direction <small>mm/min or deg/min or rev/min</small>	1000	1000	500	360
Fast Jog Minus Direction <small>mm/min or deg/min or rev/min</small>	1000	1000	250	360
Slow Jog <small>mm/min or deg/min or rev/min</small>	100	100	80	36
Accel / Decel <small>seconds</small>	.375	.375	.375	0.5
Direction Reversal	N	N	N	N
Drive Enable Delay <small>milliseconds</small>	250	250	250	250
Travel Limit (+)	52	102	0	0
Travel Limit (-)	0	0	-11	0

x1 Linear Jog Increment millimeters .01 (P040)

x1 Rotary Jog Increment degrees 0.1 (P041)

Maximum number of steps that can be pulsed per second (P968) 200,000 steps/second

Axis Motor Drive fault delay time (P991) 1000 milliseconds

Connected to CNC12

Write Settings to CNC Control Configuration

- Made the offset library increment parameter #70 unit-less so now when switching from inches to mm CNC12 will NOT convert this value from mm to inches or vs. verse.

Default value for P70 is .001, (when in inches this is .001" when in mm this is .001 mm.)

009	0.0000	029	100.0000	049	0.0000	069	1.7500	089
010	0.0000	030	126.6667	050	0.0000	070	0.0010	090
011	0.0000	031	0.0000	051	0.0000	071	0.0000	091
012	10.0000	032	25.0000	052	0.0000	072	0.0000	092
013	1.2700	033	1.0000	053	0.0000	073	1.2700	093
014	304.8000	034	8000.0000	054	0.0000	074	4.0000	094
015	127.0000	035	0.0000	055	0.0000	075	0.0000	095
016	254.0000	036	0.0000	056	5.0000	076	0.0000	096
017	0.0000	037	3.0000	057	0.0000	077	0.0000	097
018	0.0000	038	0.0000	058	0.0000	078	0.0000	098
019	0.0000	039	100.0000	059	0.0000	079	0.0000	099

Offset Library Inc/Decrement Amount

Tool Geometry Offset Library

Height Offset		Diameter	
H001	0.0000	D001	0.2500
H002	0.0000	D002	0.5000
H003	0.0000	D003	0.7500
H004	0.0000	D004	0.2500
H005	0.0000	D005	0.0500
H006	0.0000	D006	0.2500
H007	0.0000	D007	0.0000
H008	0.0000	D008	0.0000
H009	0.0000	D009	0.0000
H010	0.0000	D010	0.0000
H011	0.0000	D011	0.0000
H012	0.0000	D012	0.0000

Z Ref: FIXED HOME

Manual Measure F2    Auto Measure F3

+ .001 F5    - .001 F6    Change Tool F7    Tool Library F8    TT Setup F9    Save F10

Change the value in P70 to change the amount the F5 and F6 adjustment keys increment the values by.

Example below is millimeters with P70 set to .01 value.

The screenshot displays the 'Tool Geometry Offset Library' on a CNC control screen. It features a table with columns for 'Height Offset' and 'Diameter', listing tools H001 through H012 and D001 through D012. The H001 Height Offset value of 0.000 is highlighted in yellow. To the left, a diagram shows a tool with a 'Tool Height Offset Measured from Z Home' and a 'Ref Pos' arrow. To the right, a diagram shows a tool with a 'Diameter' measurement. At the bottom, a function key menu includes 'Manual Measure (F2)', 'Auto Measure (F3)', '+.01 (F5)', '-.01 (F6)', 'Change Tool (F7)', 'Tool Library (F8)', 'TT Setup (F9)', and 'Save (F10)'. The '+.01' and '-.01' keys are enclosed in a green box. A 'TOC' button is visible in the top right corner.

Tool Geometry Offset Library			
	Height Offset		Diameter
H001	0.000	D001	6.350
H002	0.000	D002	12.700
H003	0.000	D003	19.050
H004	0.000	D004	6.350
H005	0.000	D005	1.270
H006	0.000	D006	6.350
H007	0.000	D007	0.000
H008	0.000	D008	0.000
H009	0.000	D009	0.000
H010	0.000	D010	0.000
H011	0.000	D011	0.000
H012	0.000	D012	0.000

Z Ref: FIXED HOME

Manual Measure (F2)    Auto Measure (F3)    **+.01 (F5)**    **-.01 (F6)**    Change Tool (F7)    Tool Library (F8)    TT Setup (F9)    Save (F10)

- Added 'CTRL-G' to the PLC Detective, which opens a Goto Line Dialog
- PLC Detective can now search both up and down using F6 – Search
- Fixed missing Touch Plate PLC messaging

## Known Issues

1.) Oak, Allin1DC and MPU11 users should be aware: running the CNC12 v5.20 on top of the existing cncm\ directory will overwrite your WCS file with a new, blank file.

That will remove your software travel limits; zero out any custom return points (such as a fixed tool setter location, or the G30 Z coordinate for an umbrella or swing-arm tool changer); and zero out all your work offsets.

But have no fear it is simple to recover the .wcs file.

You can recover normal operation by restoring the last-known-good report from before the update, or by extracting the cncm.wcs file from such a report and copying it into the c:\cncm directory.

So as always be sure to create a fresh report.zip file BEFORE running the CNC12 installer.