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.

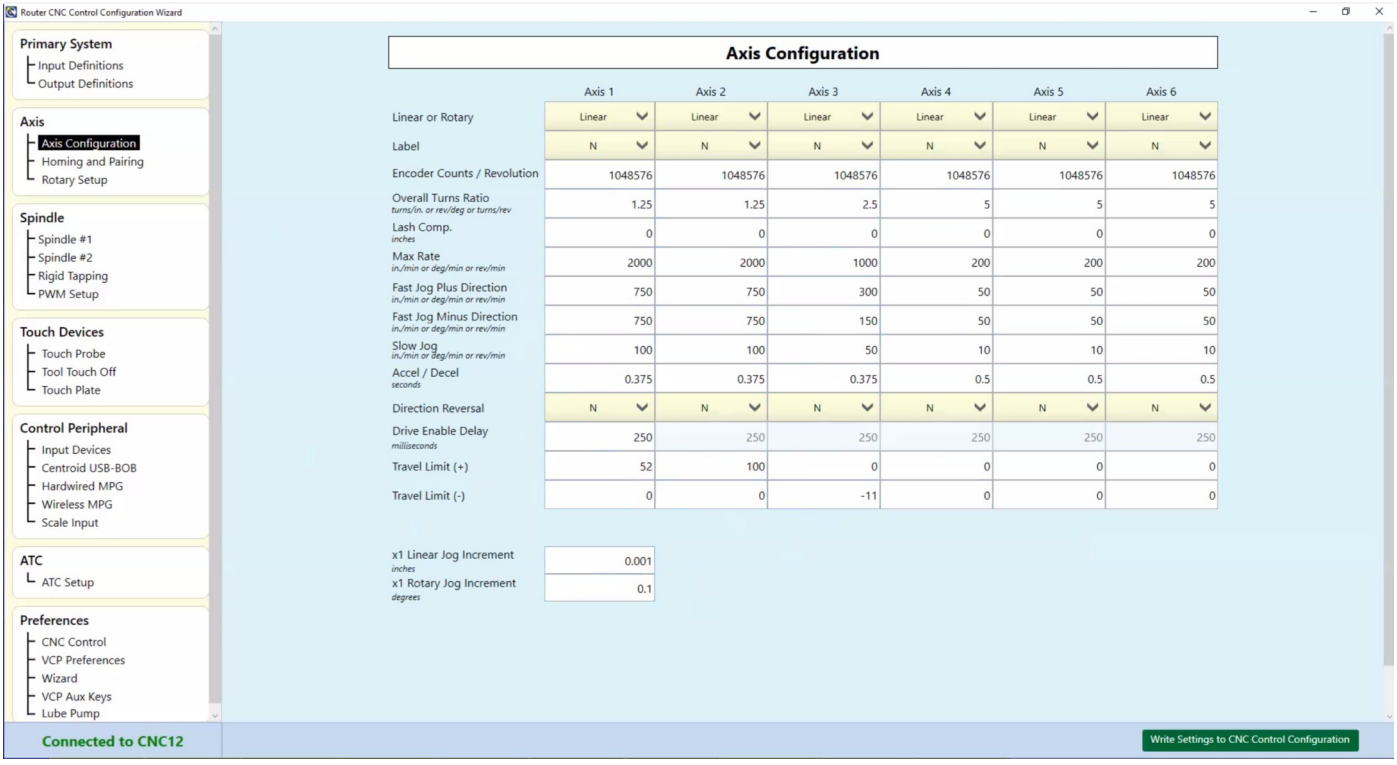
Download CNC12 v5.20 here.
https://www.centroidcnc.com/centroid_diy/centroid_cnc_software_downloads.html

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.)

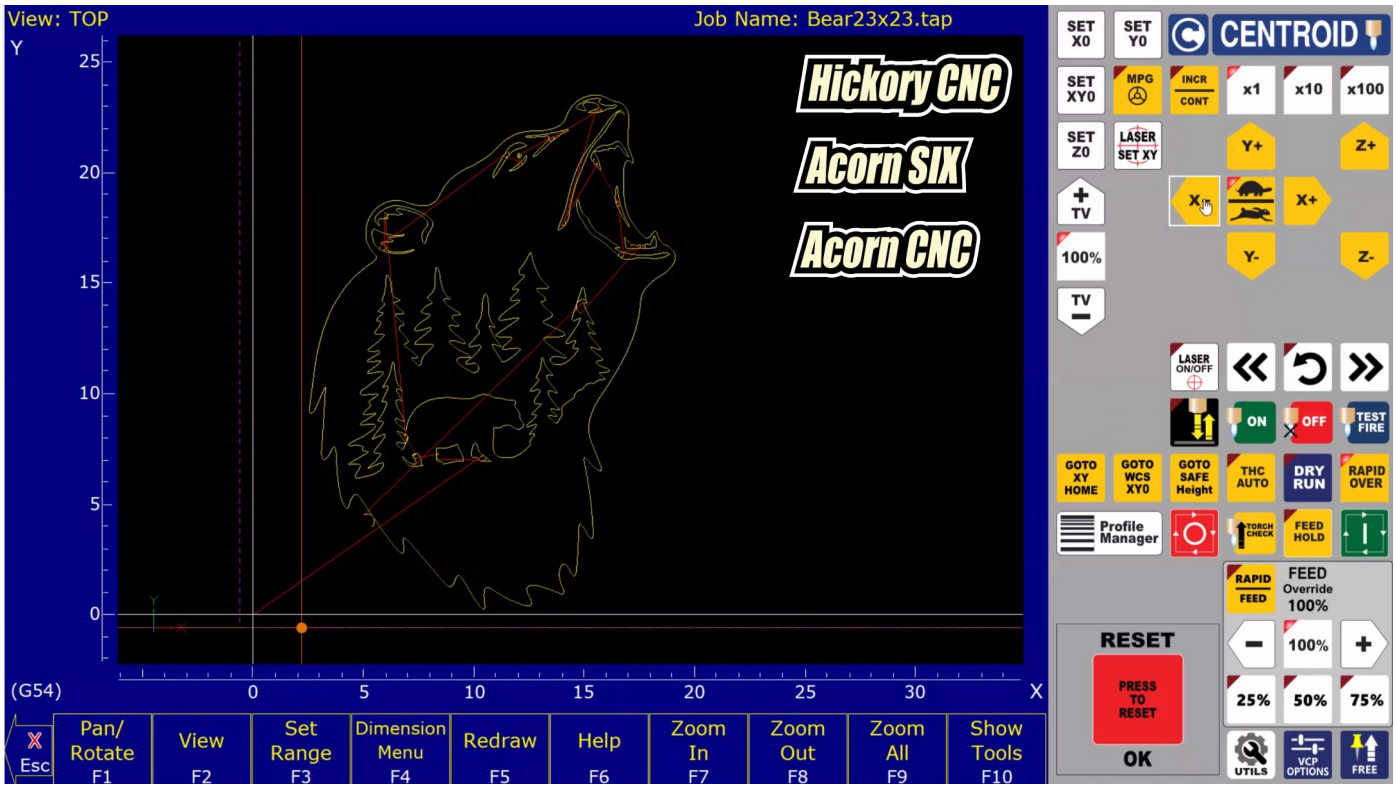
Major Additions

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



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

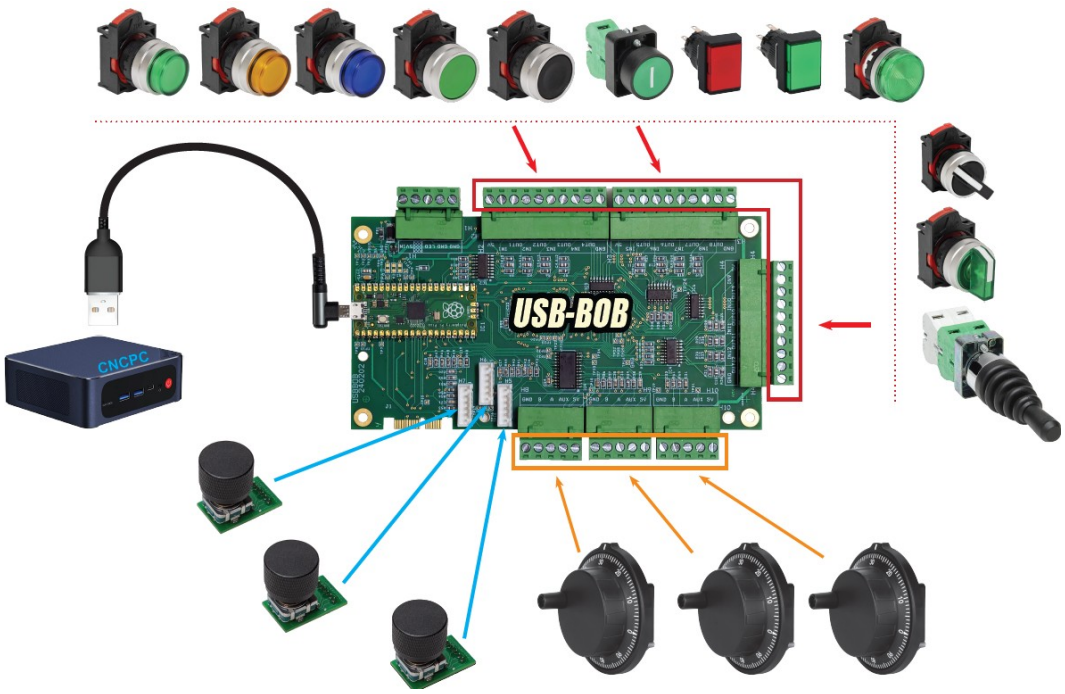


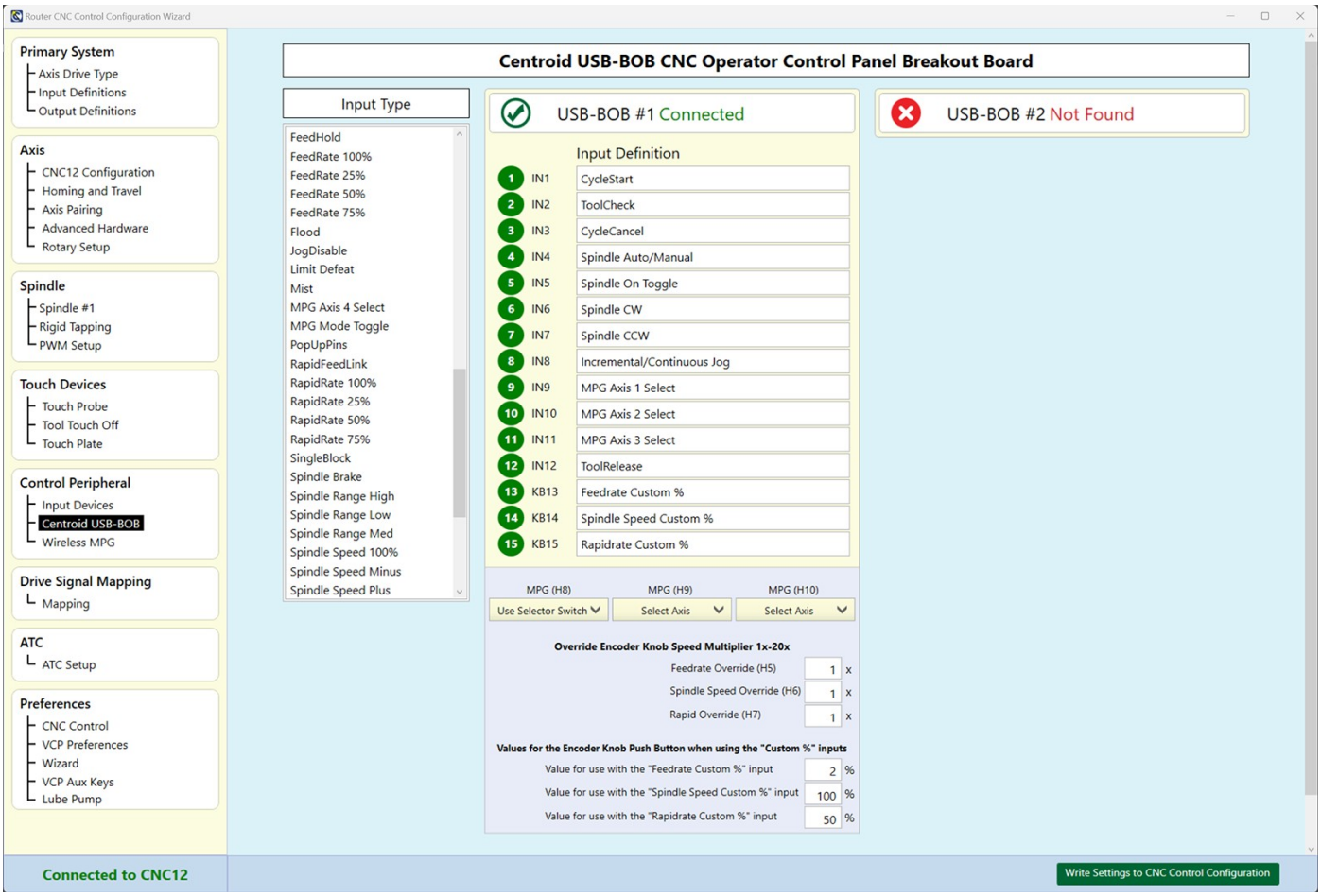
Now you can control a CNC Plasma Machine with the AcornSix or Hickory CNC Controllers in addition to the Acorn controller.

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!

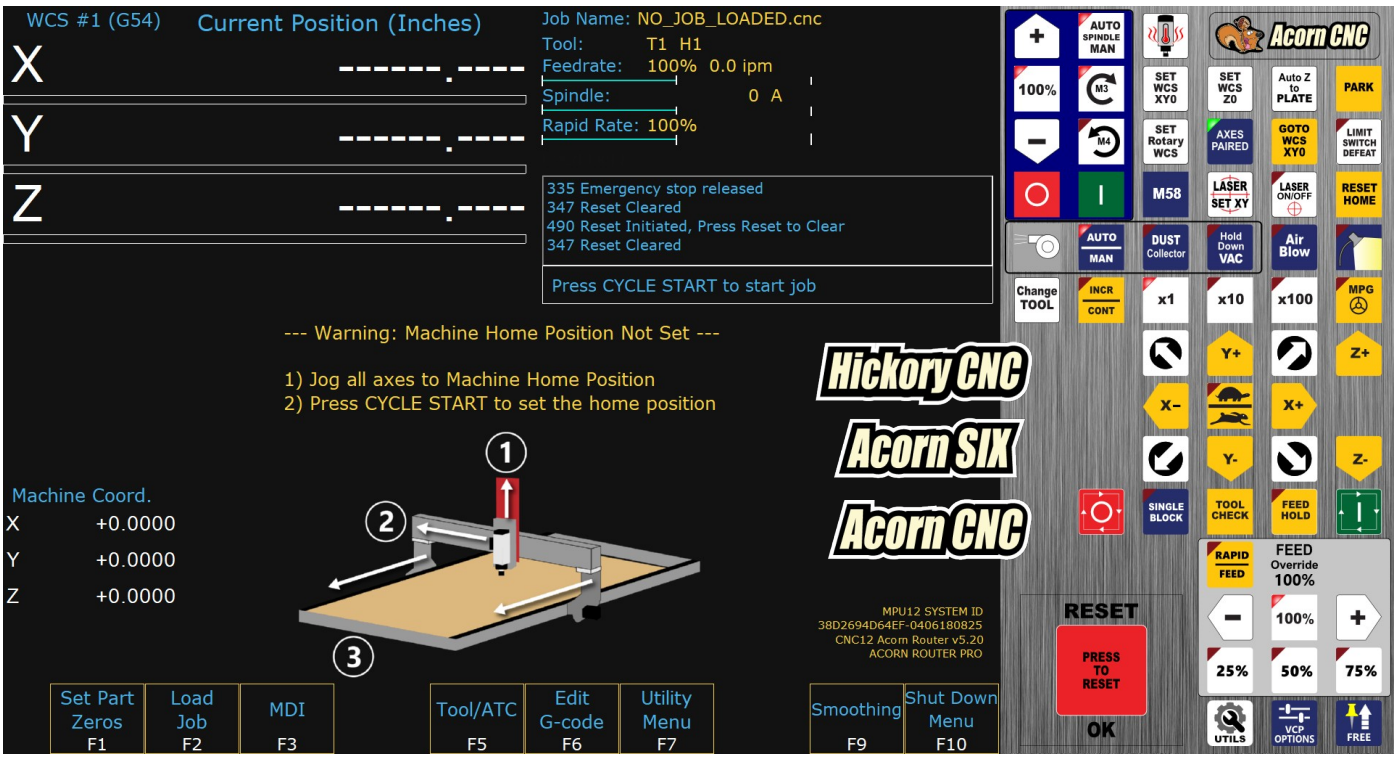




More information on USB-BOB is here: https://shopcentroidcnc.com/shop/cnc-accessories/centroid_usb-bob_operator_control_panel/

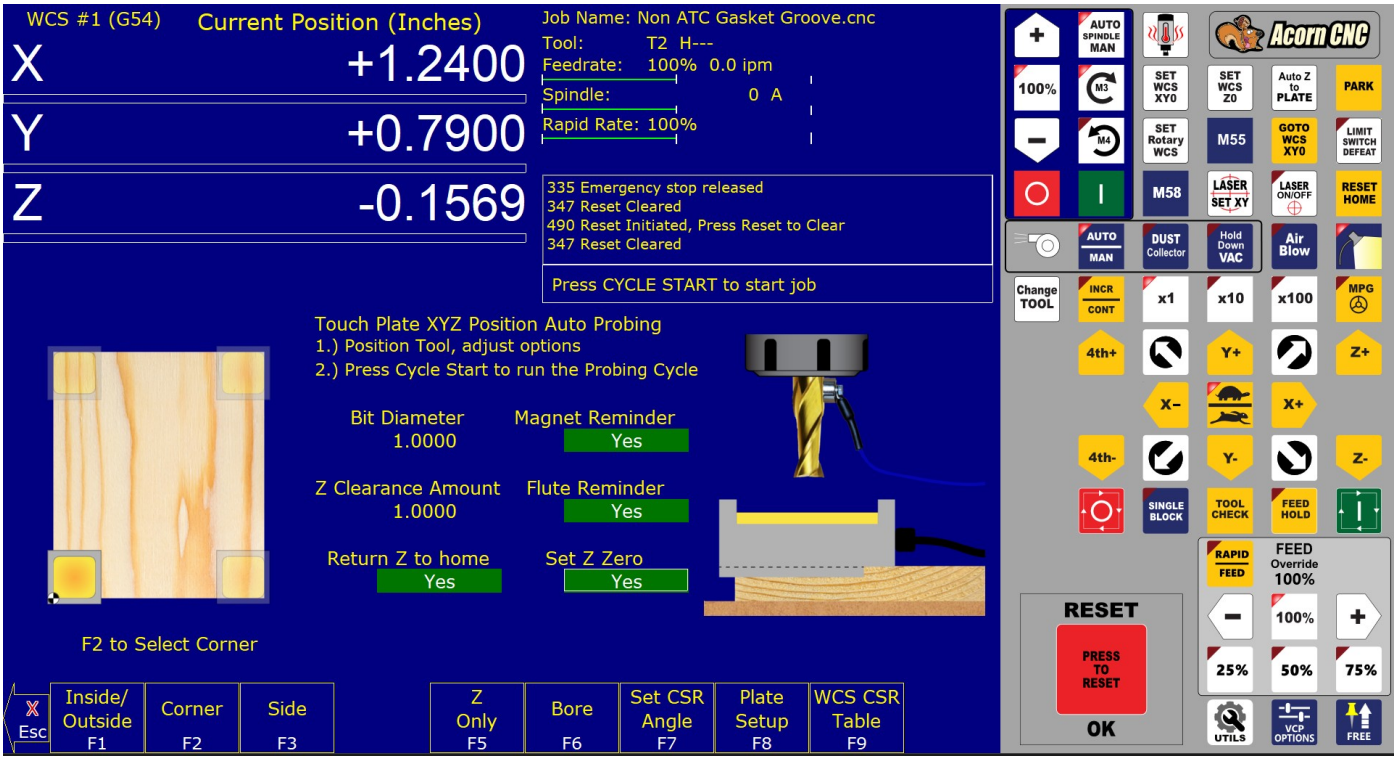
4.) New CNC12 Router specific CNC12 software for Acorn, AcornSix and Hickory!

- 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)

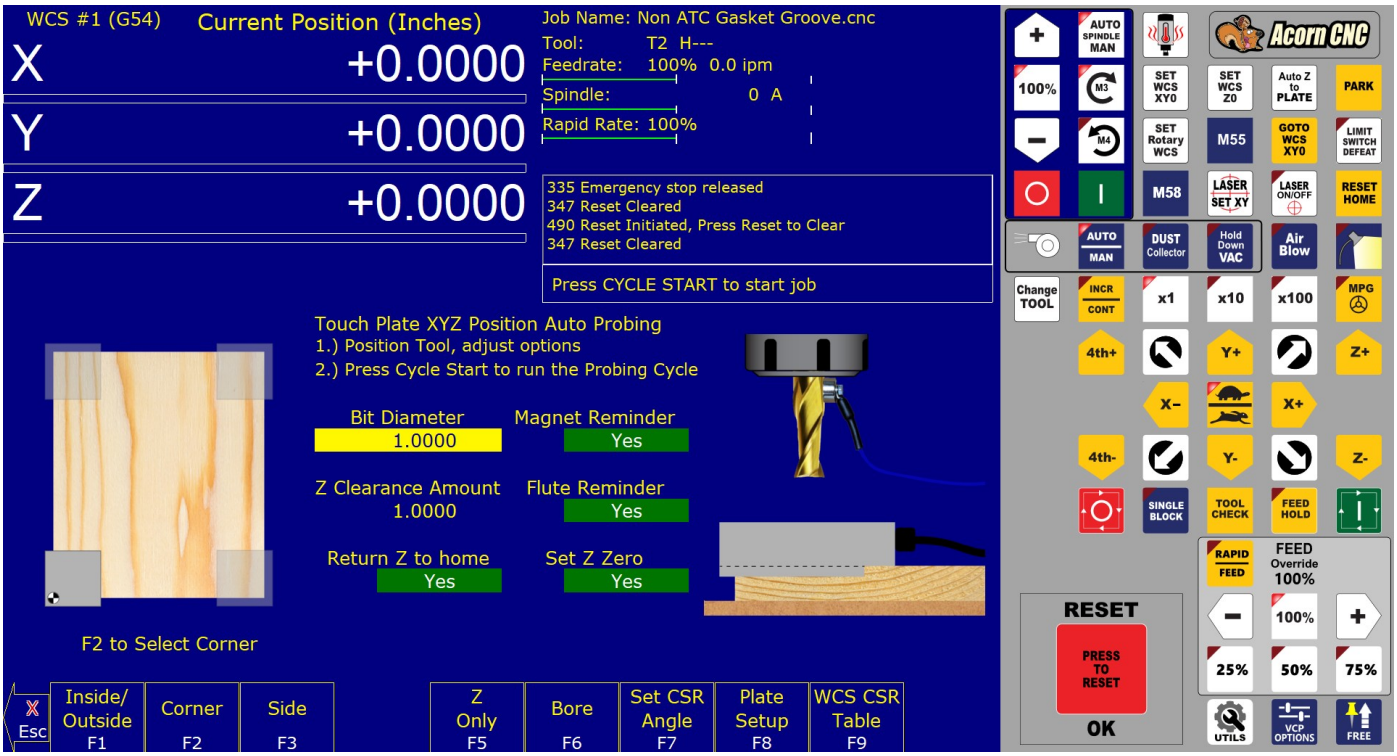


- All new Router Part Setup Menu with menu driven support for Part Setups for all common Router specific methods such as but not limited to: Touch Plate, Cross Hair Laser, Touch Probe, and various Manual methods all in one quick to navigate menu. Here are a few samples of the new part setup menus.

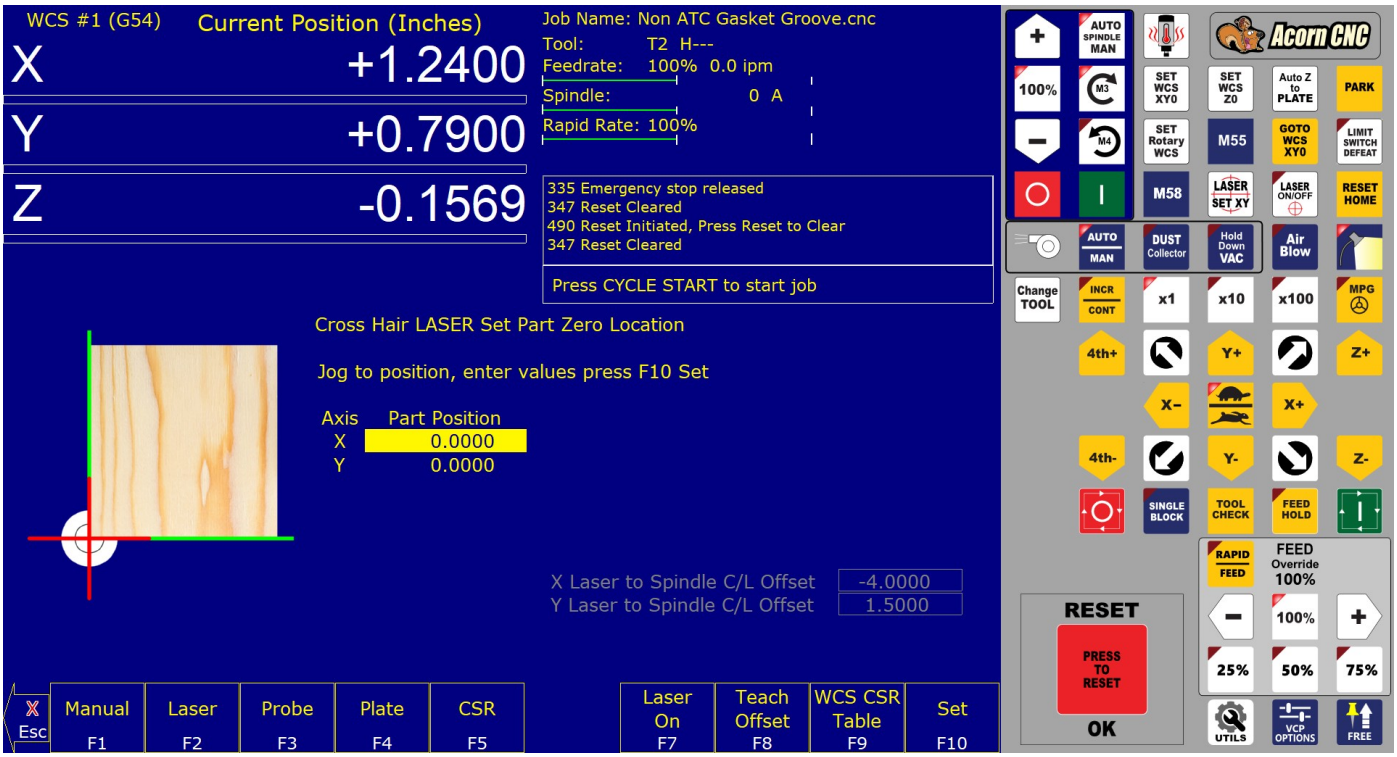
A few select Router Part Setup Menu Screen Shots



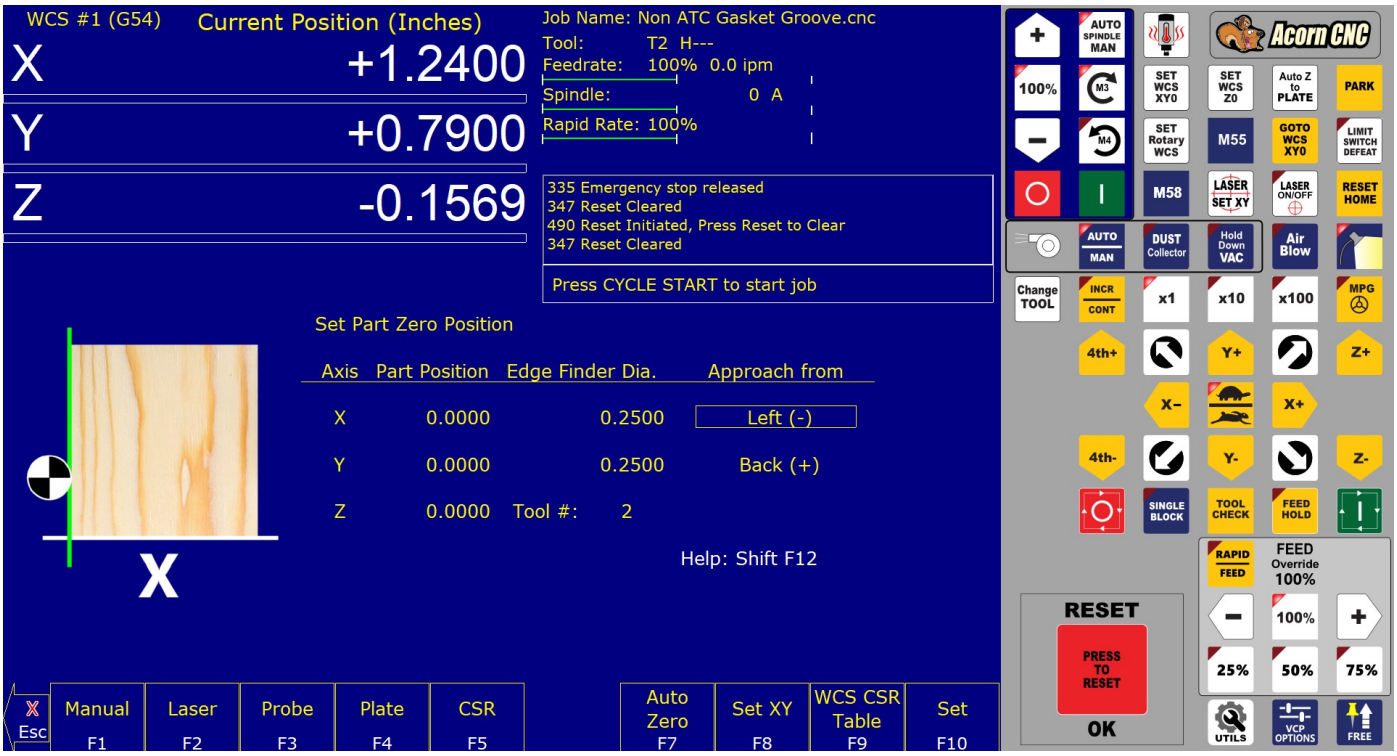
Avid CNC Touch Plate Part Zero setup menu.



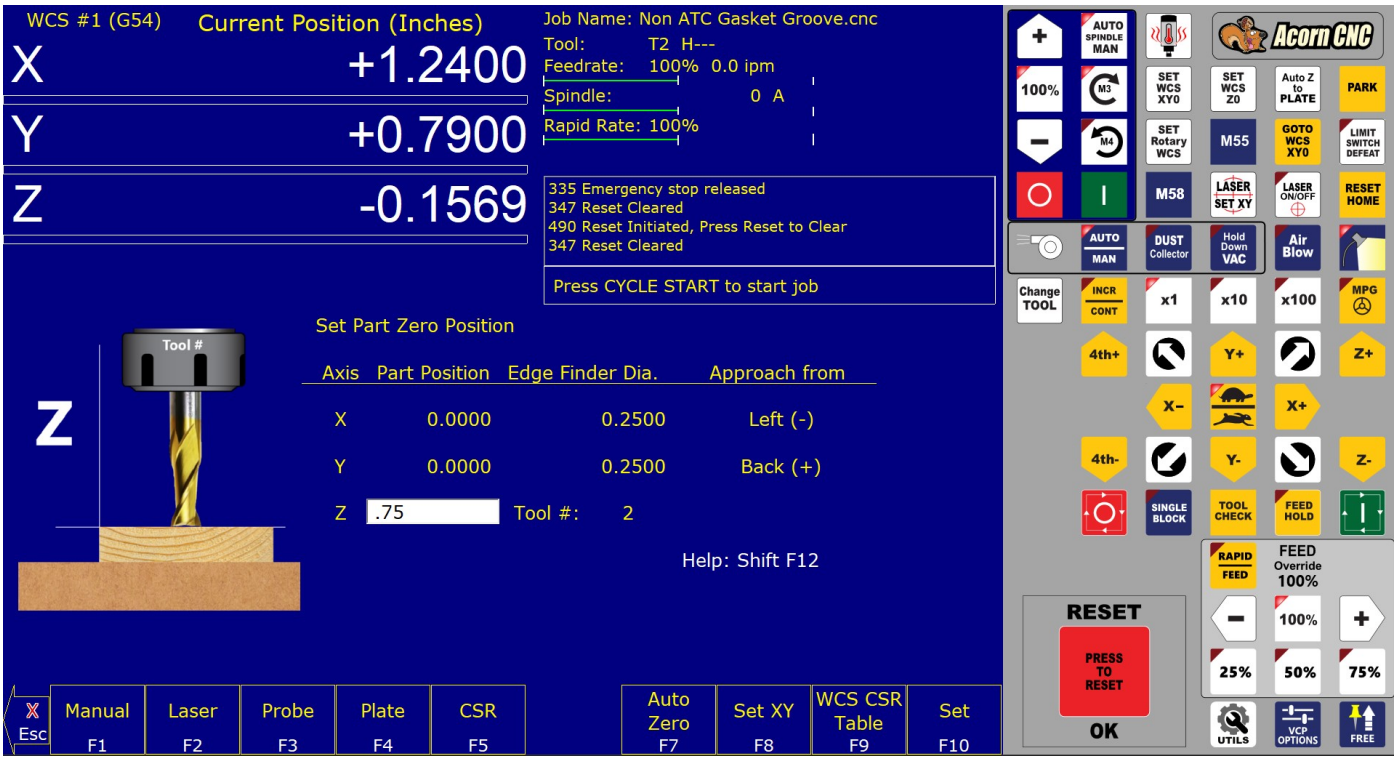
Touch Plate menus



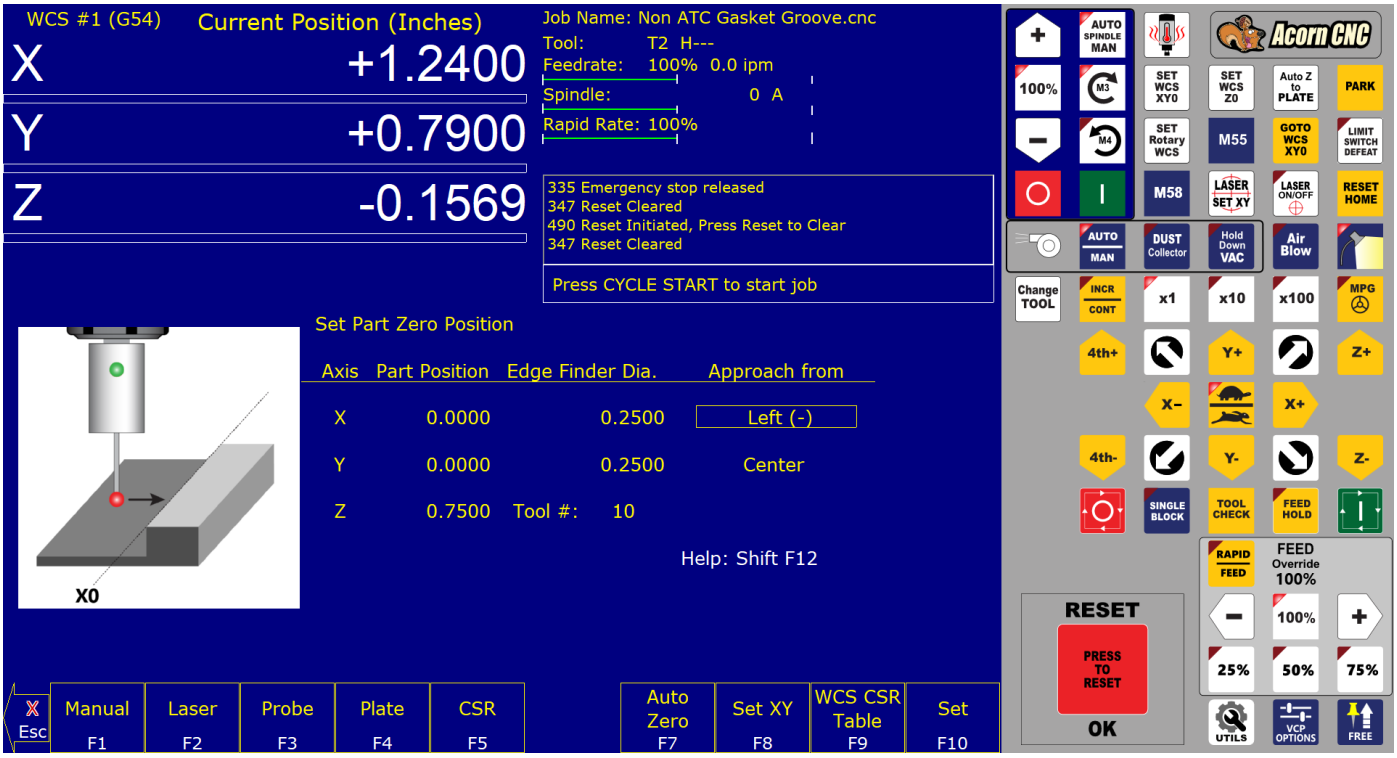
Cross Hair Laser to set X and Y zero position



Manually touch part edge and set X zero position

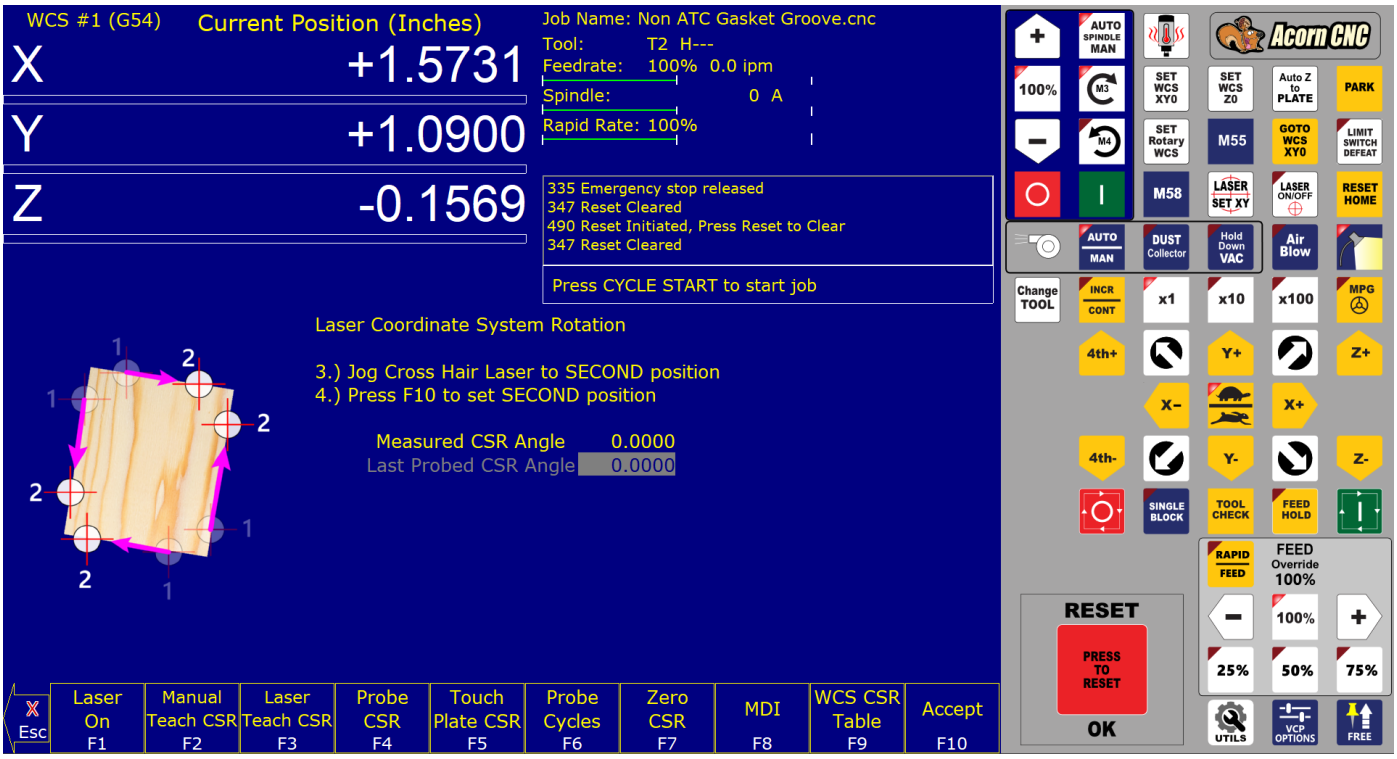


Manually set Z part zero position

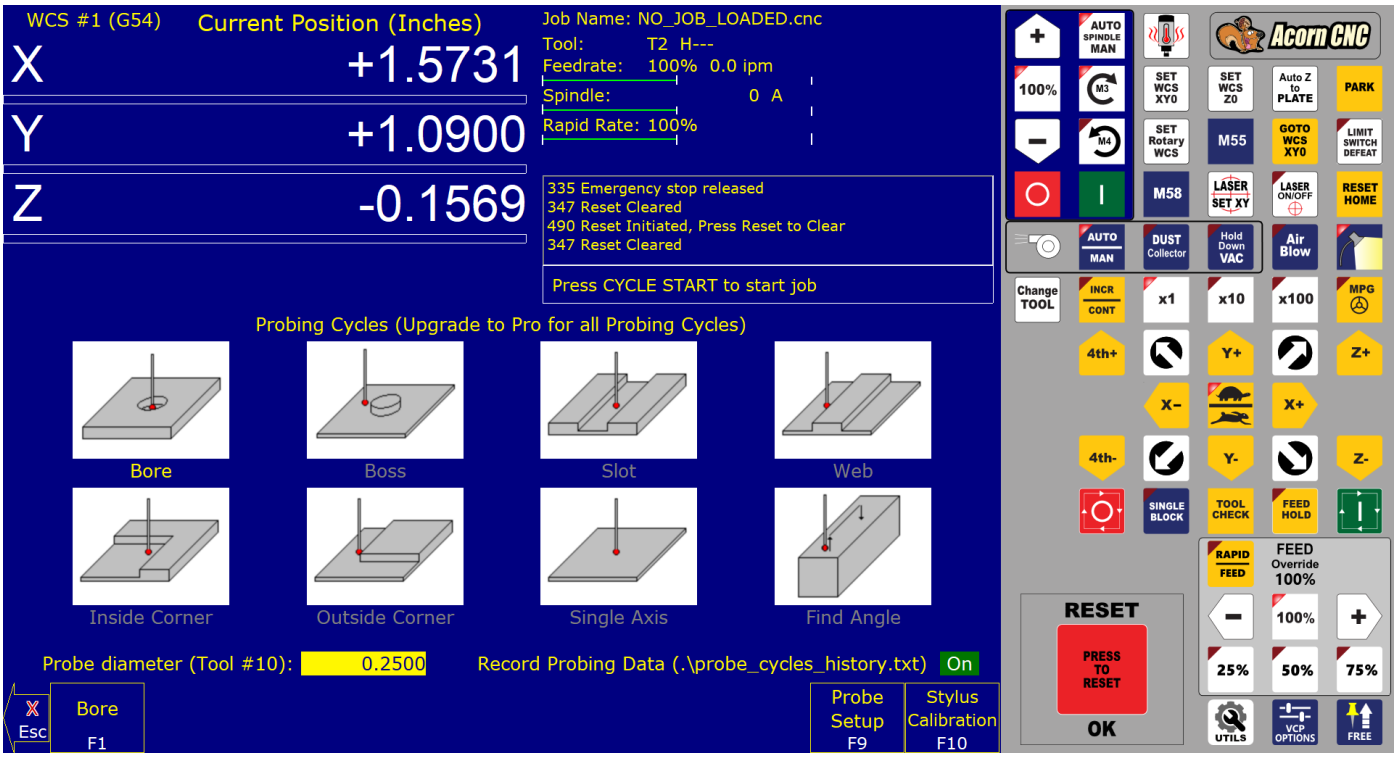


Set Part Edge X0 with Touch Probe

Manual “jog to position to set Coordinate System Rotation (CSR)”

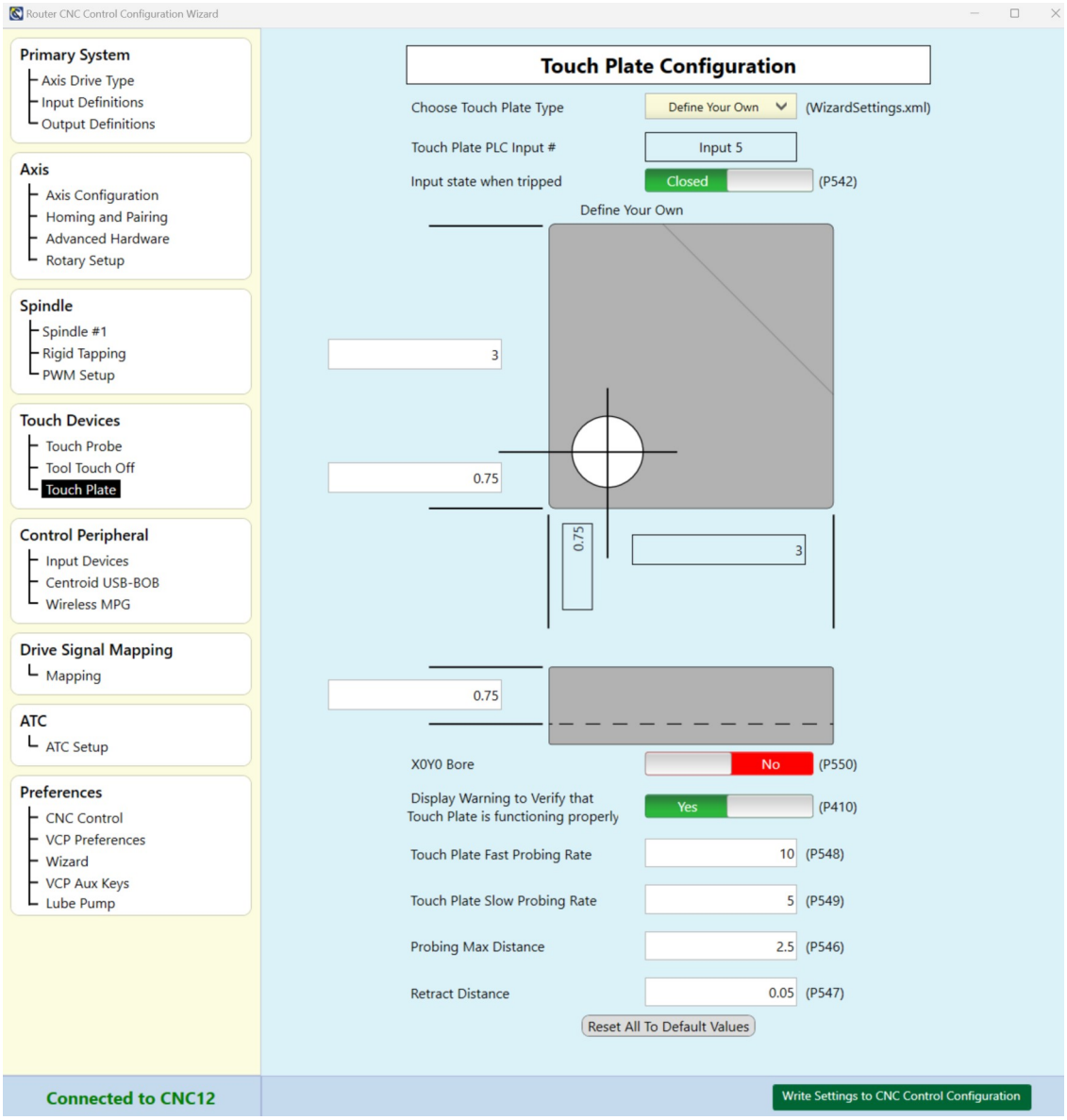


Cross Hair Laser set Coordinate System Rotation (CSR)

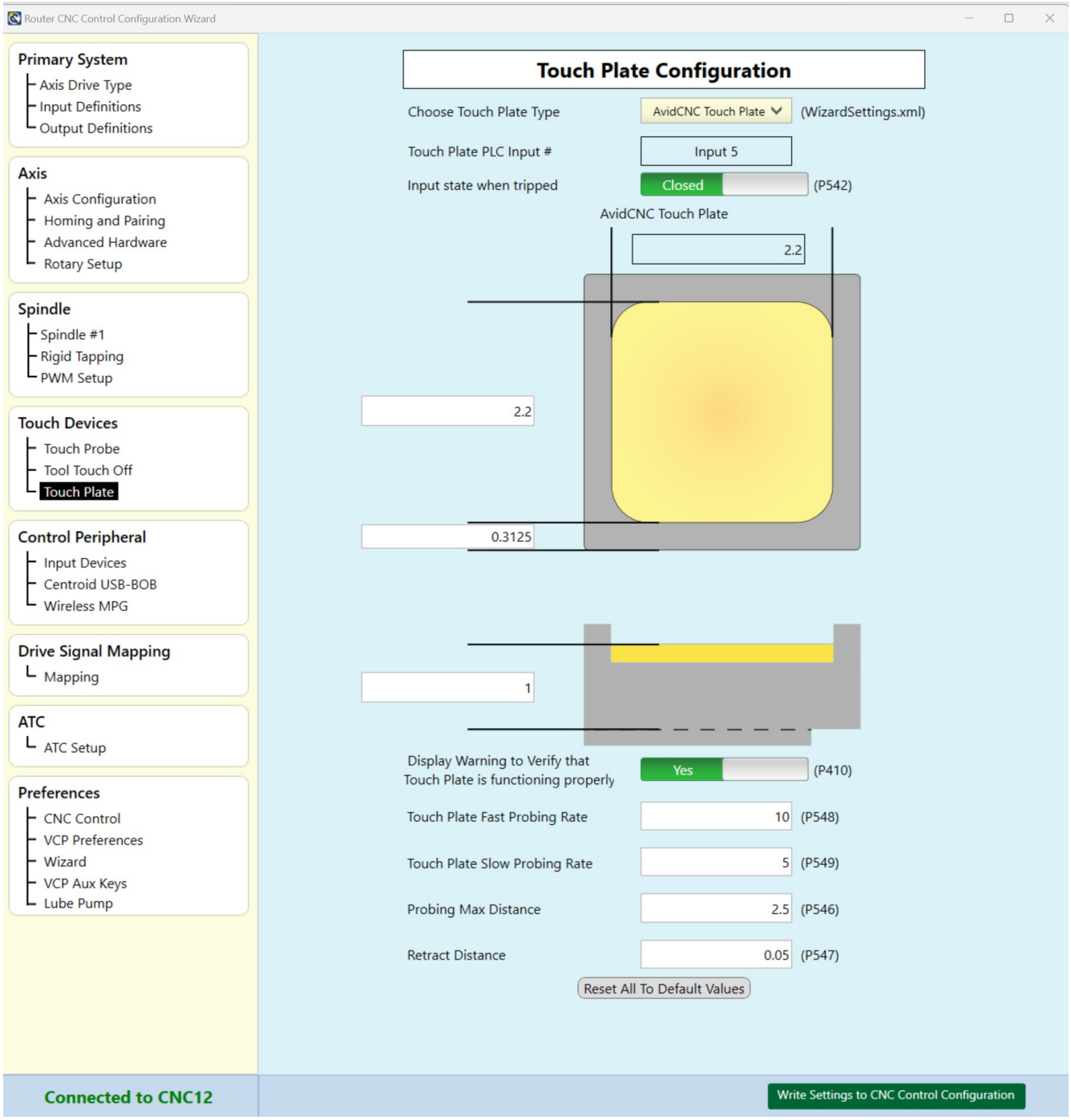


Touch Probe Menu, find positions, set part zeros, calibrate probe pre-travel automatically, record probed data to a text file.

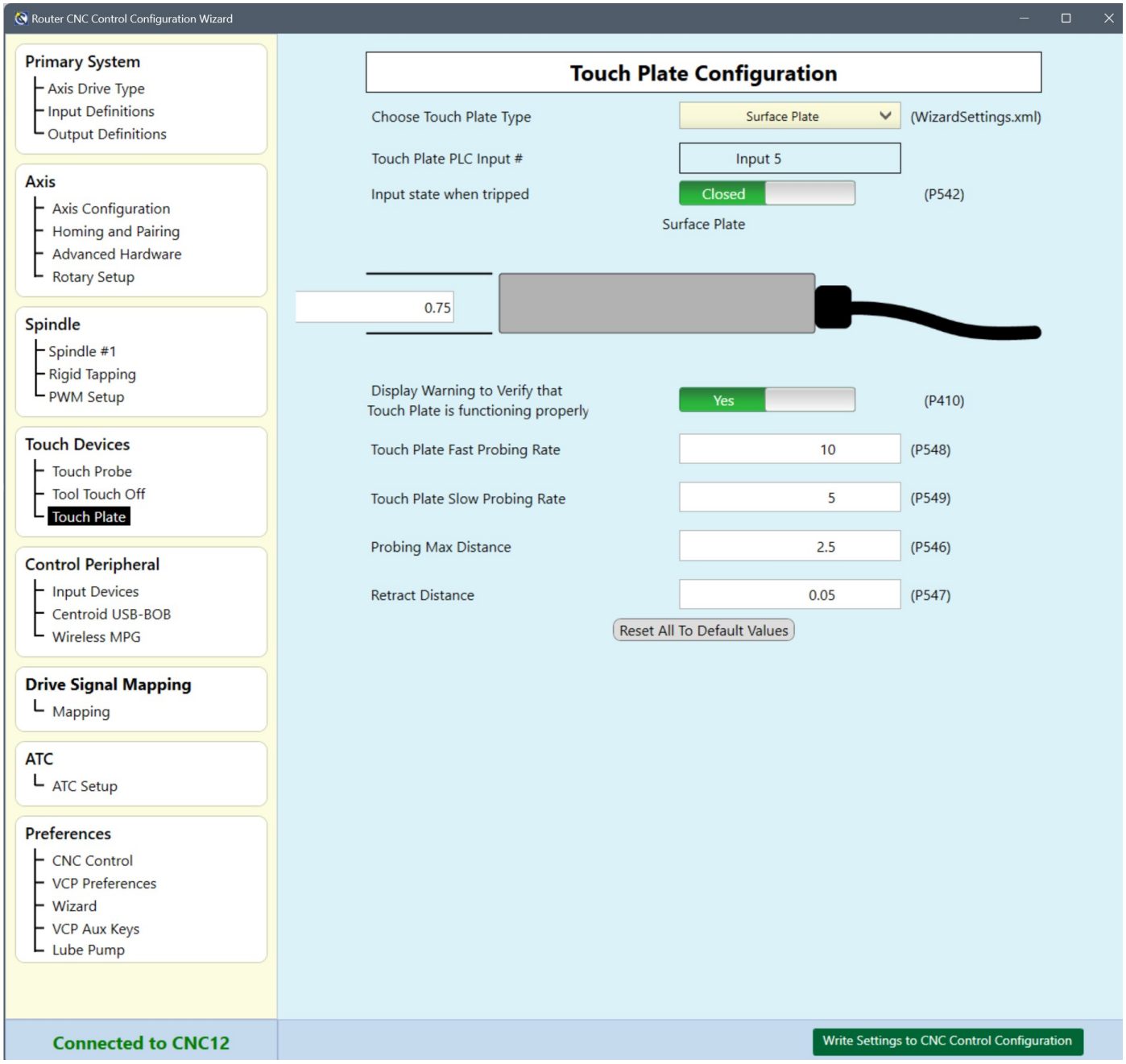
- Router Touch Plate setup menus added to the Router Wizard, Preset AvidCNC Touch Plate selection along with “Define your own Touch Plate Geometry” for simple touch plate setup.



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

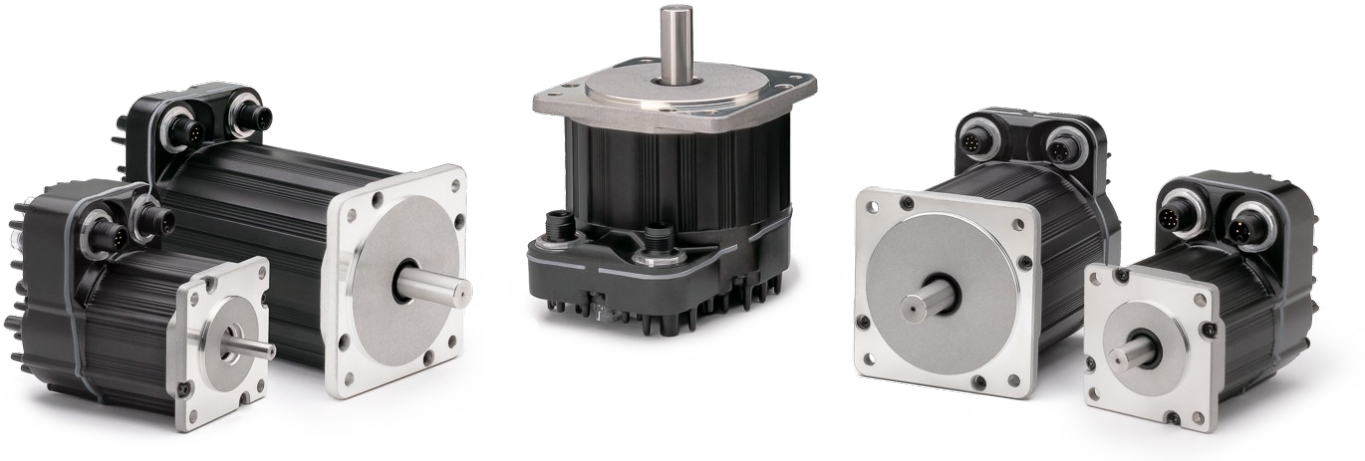


Avid CNC Touch Plate is pre-filled in with the correct data, just select it and press “Write Settings to CNC control configuration”



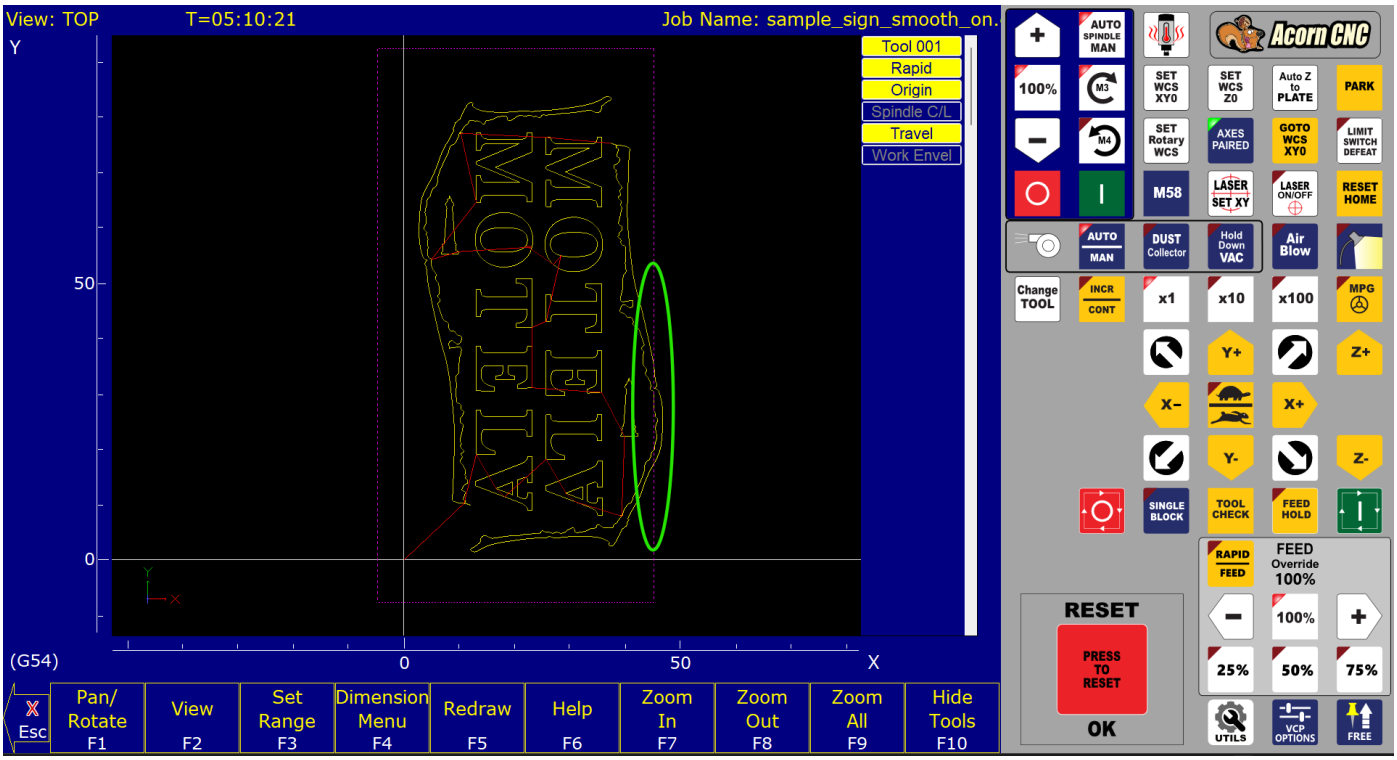
and for those using a simple piece of metal as a touch off to set part Z zeros rather than a XYZ touch plate.

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

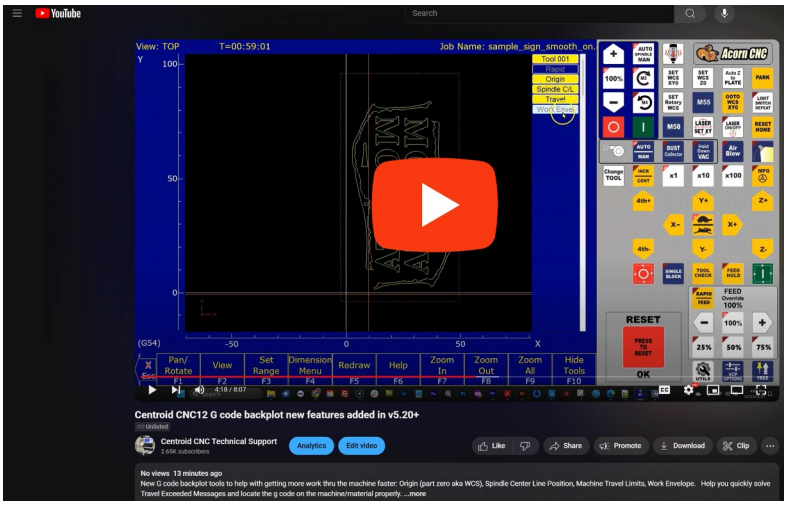


Link to Teknic web page.
https://teknic.com/products/clearpath-brushless-dc-servo-motors/ethercat-servo/

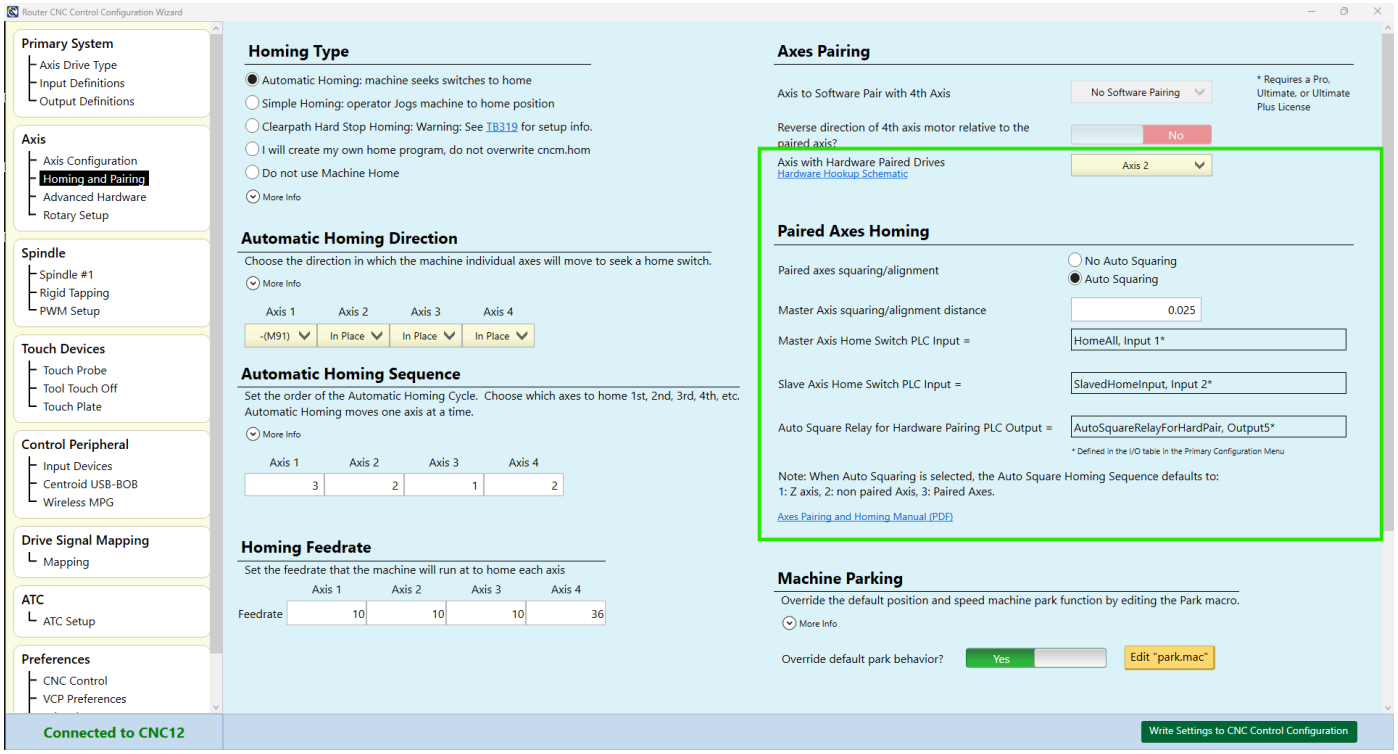
6.) Useful new G-code backplot features such as Travel Limit, Work Envelope, Dynamic Spindle Position indicator, WCS position. Example below shows the G code job exceeding the machine travel limits with the current Part zero location. Available in Mill, Router and Plasma. (Lathe will get these new features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



Video demonstration is here https://youtu.be/dhe_YG7Fsc4

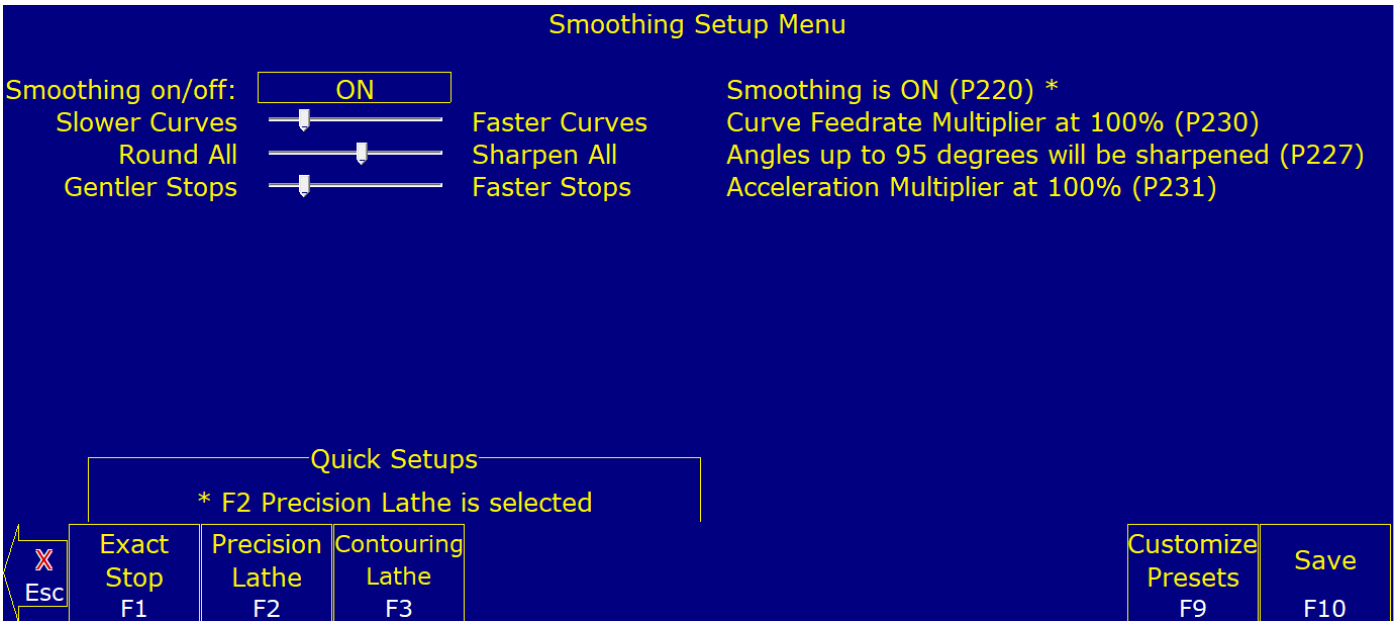


7.) Hardware Pairing with Autosquaring is now a “canned” feature and has been added to the Acorn and AcornSix Wizard. Now you can hardware pair an axis (two axis motors hardwired together on one axis, typically a gantry) and the Wizard will write the homing program for you that will home and auto square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

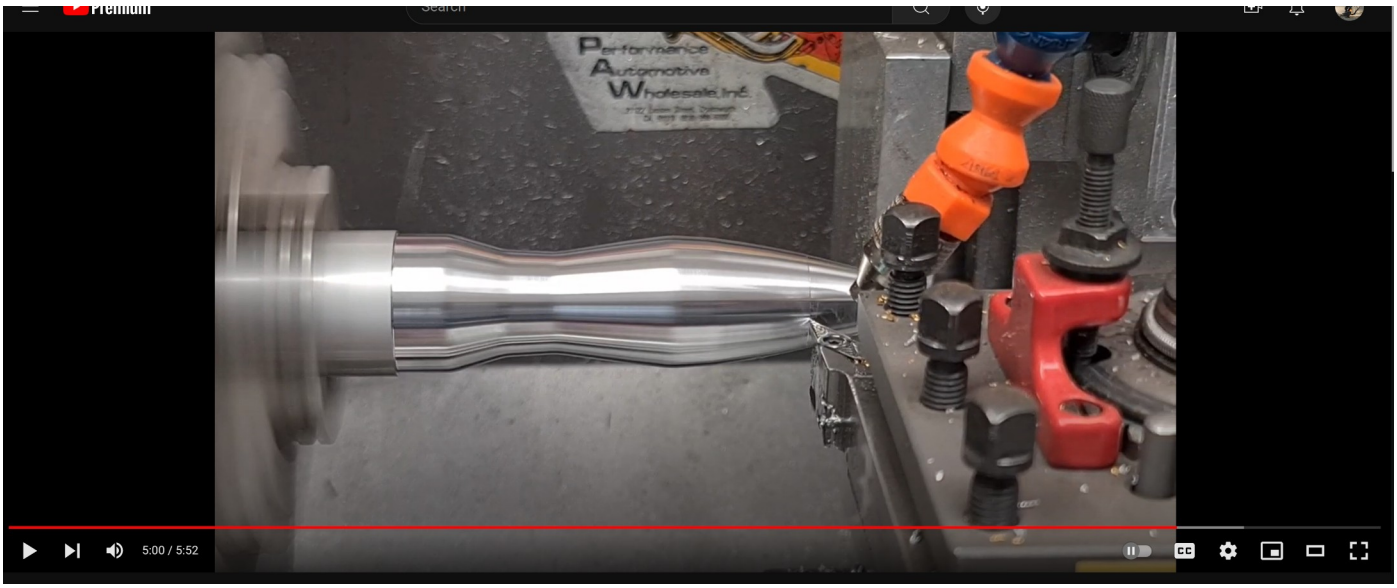


Link to typical Hardware Paired CNC Router Gantry machine schematic with Autosquaring. (S15200)
https://www.centroidcnc.com/centroid_diy/schematics/pbrowse.php?term=hardware

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

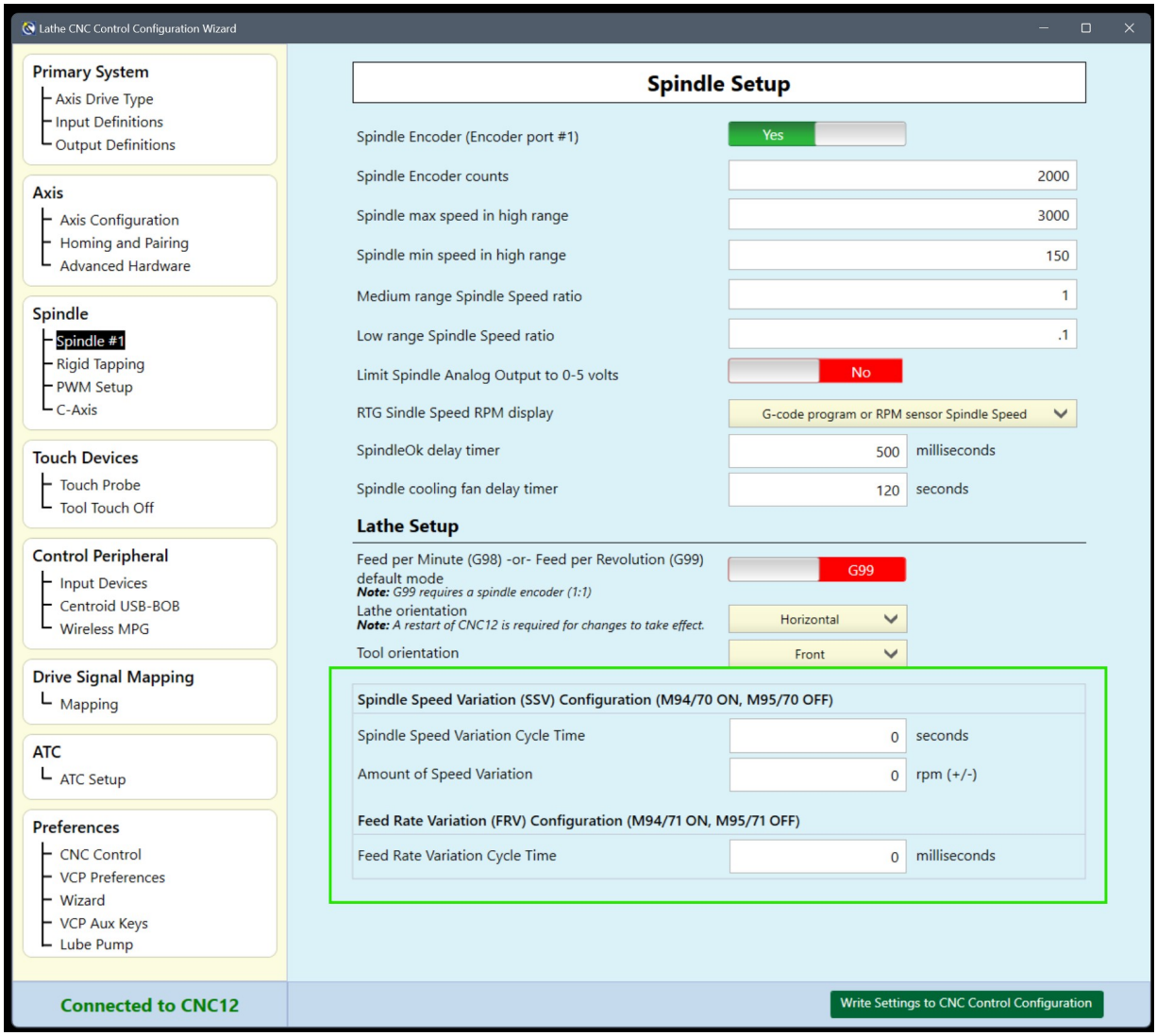


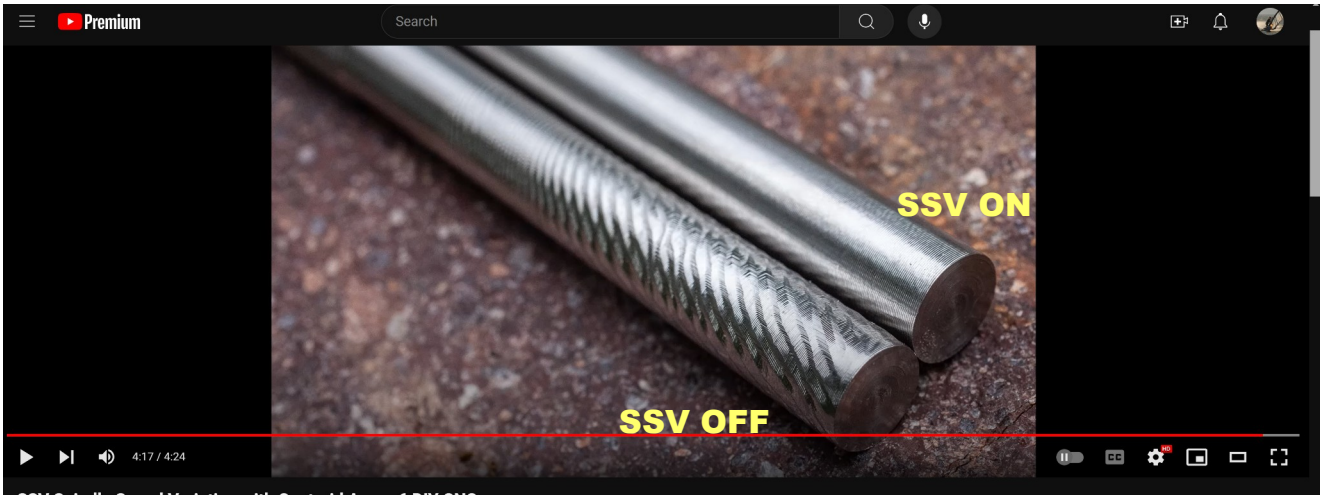
Uwe Mattern made a video showing his unique way of using G64 on a Lathe Part.
https://youtu.be/HNRWMMaluq4?si=lFqPM2RwDXPVJ3tu



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.





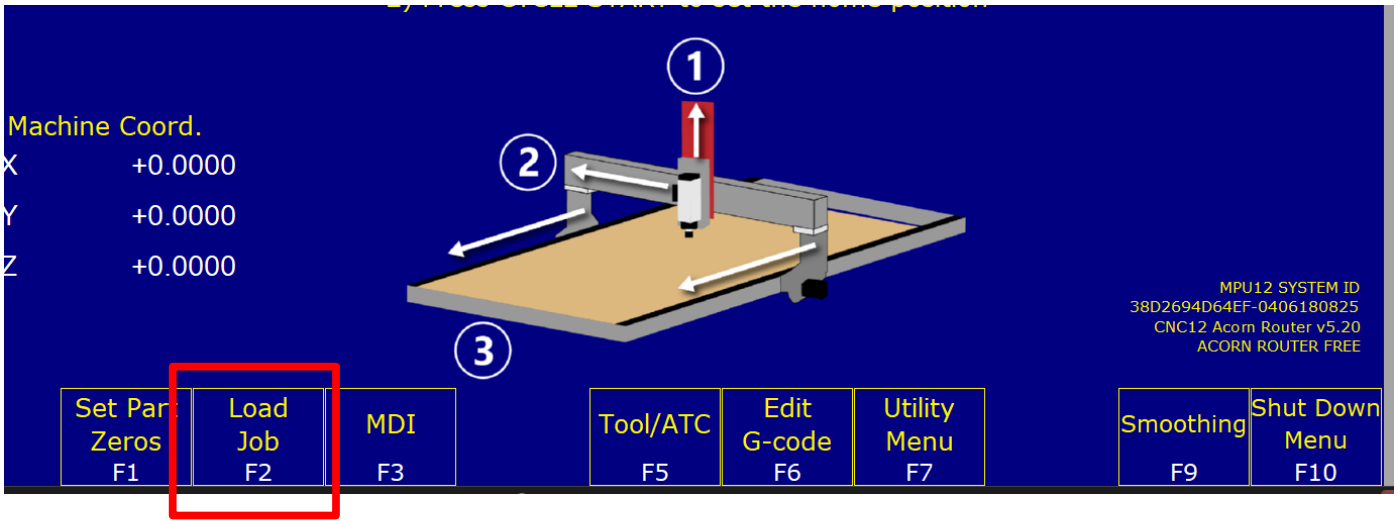
Video showing SSV OFF vs. SSV ON https://youtu.be/o6YsrY0MDoA

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"

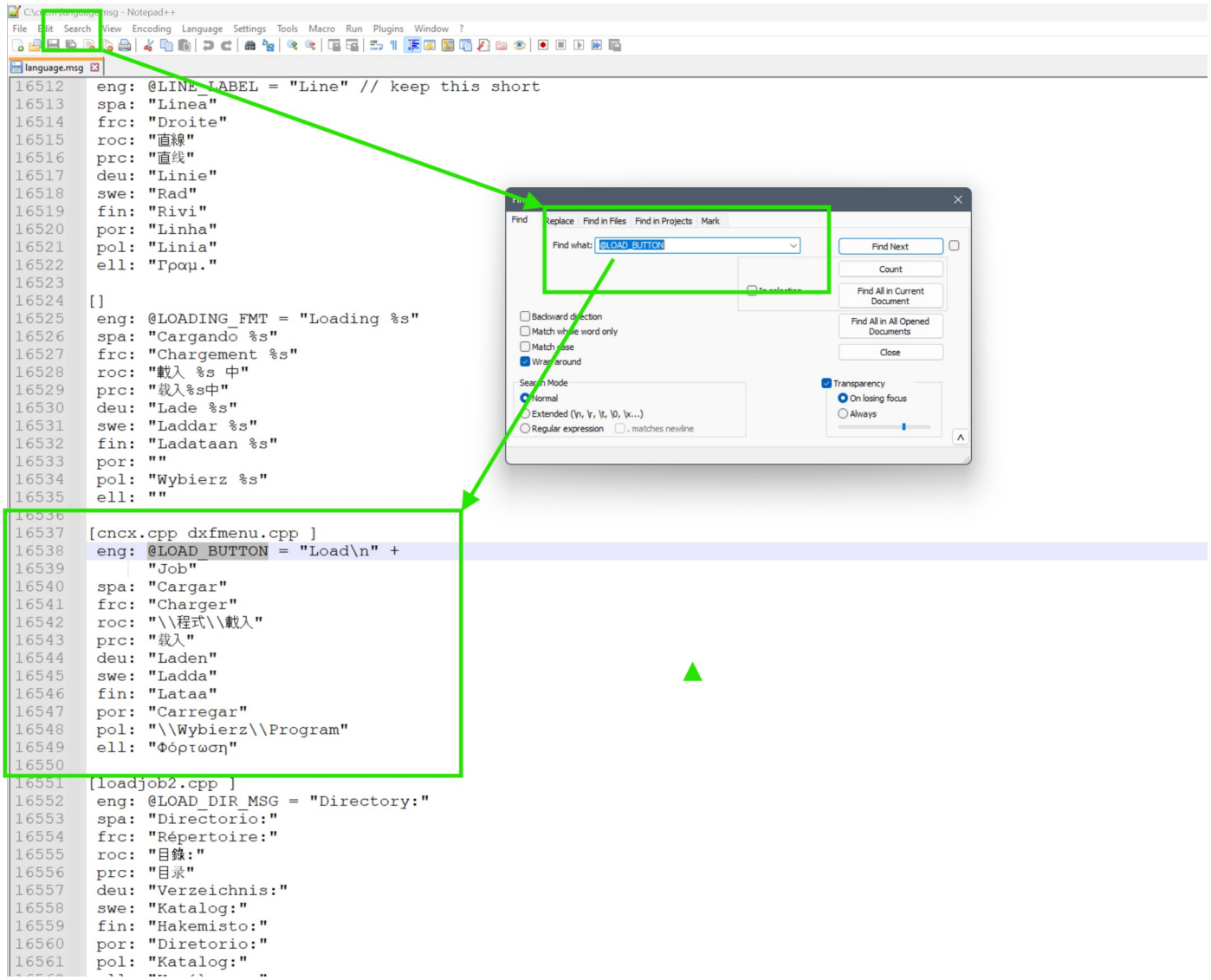


a.) Open Notepad ++ and create a new file. " user_language.msg " and save it in the root CNC12 directory (cncm or cnct etc)

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

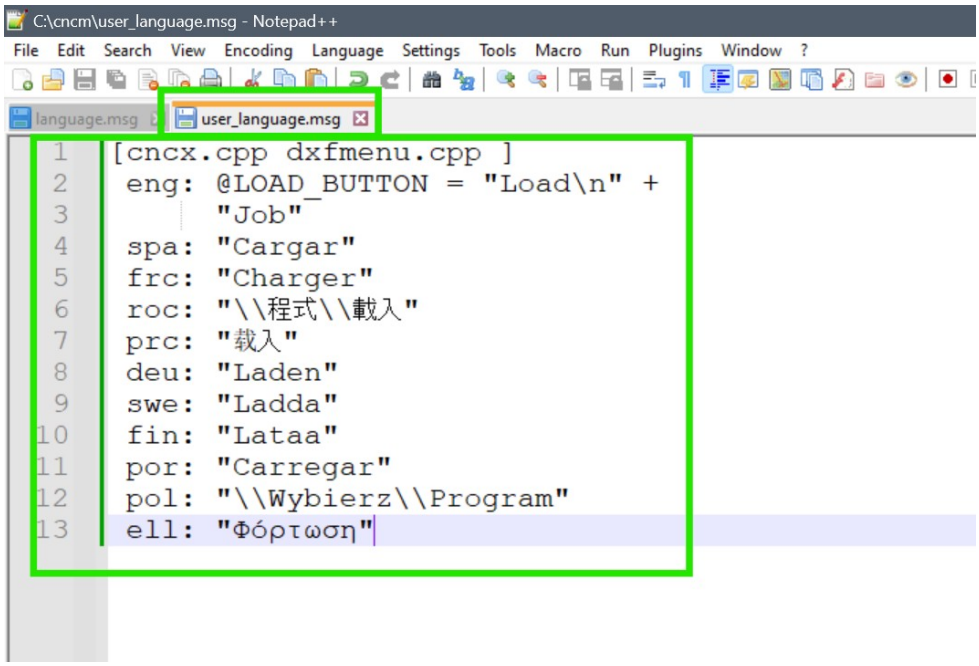
Use the search feature of Notepad ++ find the words you want to change.

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




```
[cncx.cpp dxfmenu.cpp ]
 eng: @LOAD_BUTTON = "Load\n" +
      "Job"
 spa: "Cargar"
 frc: "Charger"
 roc: "\\程式\\載入"
 prc: "载入"
 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

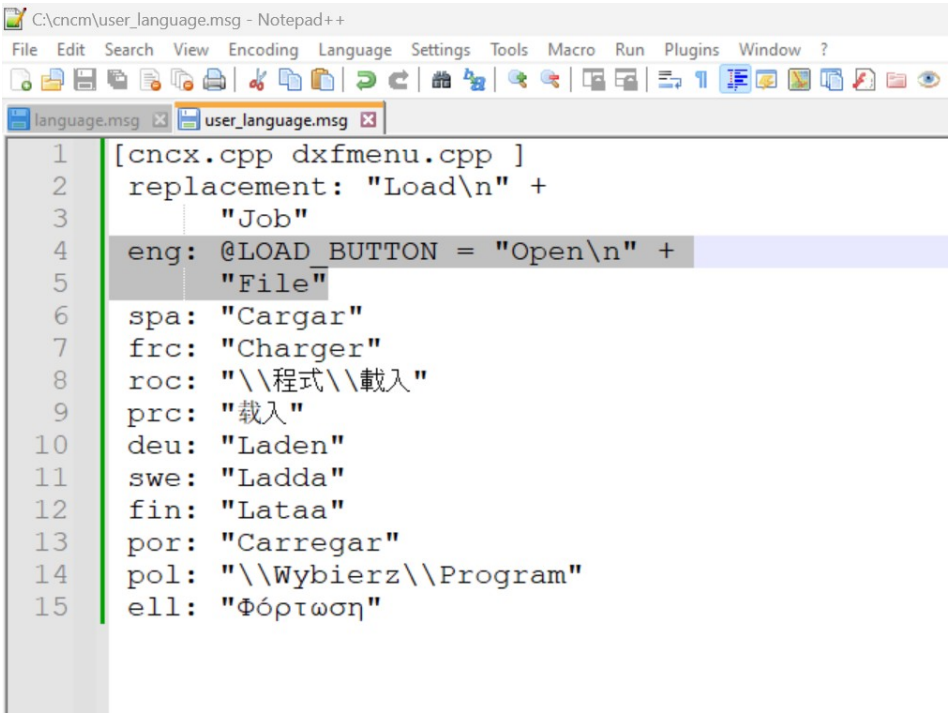
```
[cncx.cpp dxfmenu.cpp ]
 replacement: "Load\n" +
      "Job"
eng: @LOAD_BUTTON = "Load\n" +
      "Job"
 spa: "Cargar"
 frc: "Charger"
……..
```

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.

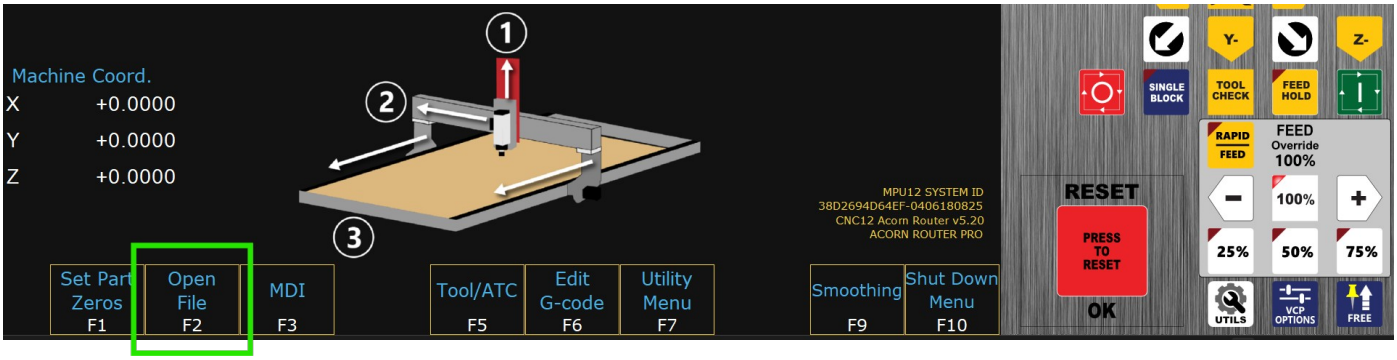
In this example, the English words for the F2 Load Job Function key are changed from “Load Job” to “Open File” on the eng: line highlighted in yellow below

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

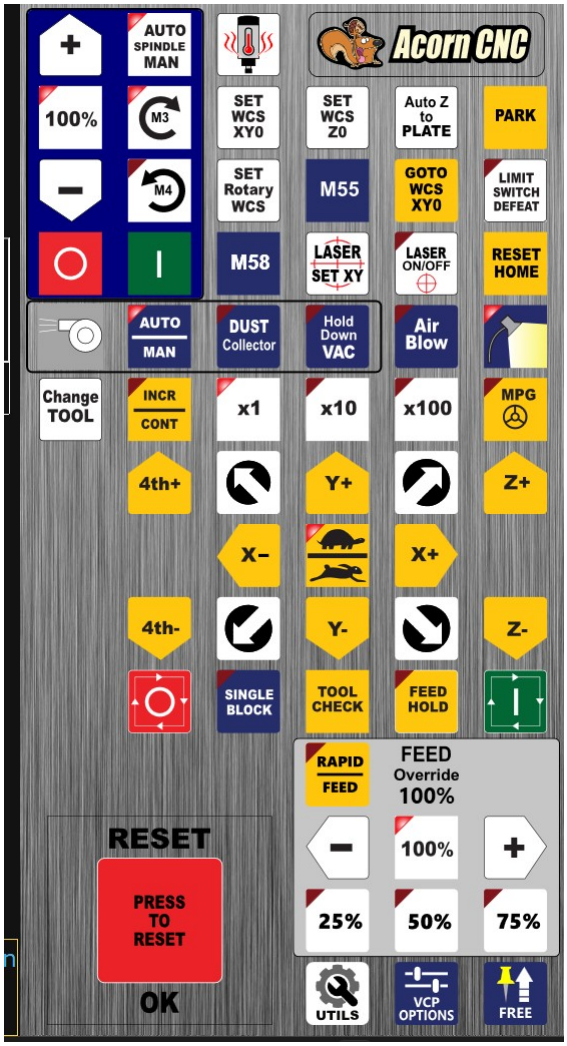
- Makes updating to a new version of CNC12 with your customizations easy. Simply copy your existing “user_language.msg” file into the root CNC12 directory after a new CNC12 installation to have your changes take effect.

- 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.

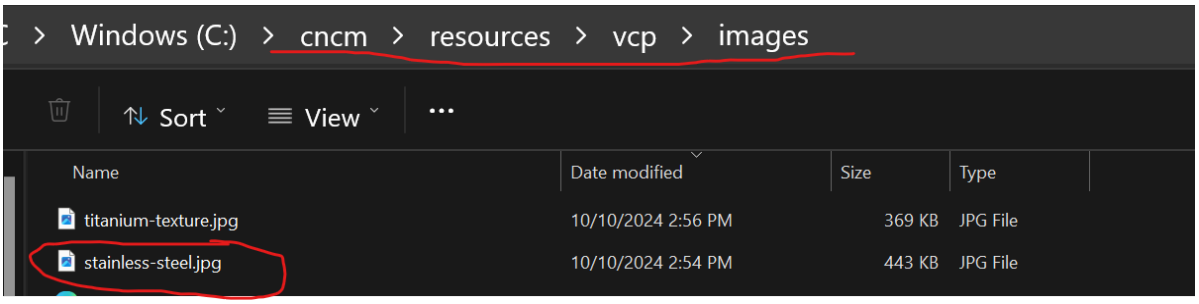
- You can share your “user_language.msg” with others using version v5.20+ of CNC12

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:\cncm\resources\vcp\images folder



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

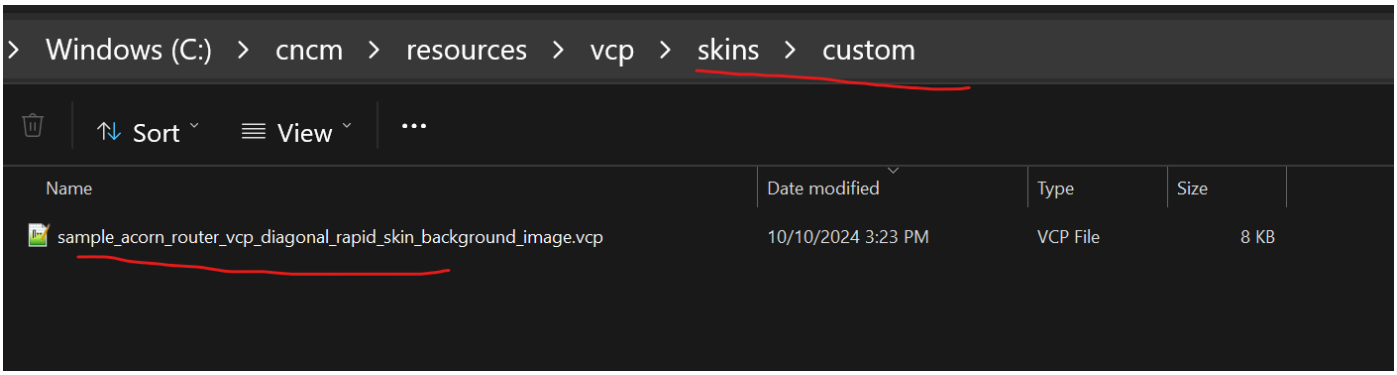
Edit the VCP skin and add the new background node <background>

under <vcp_skin> with the path to the image.

Highlighted in yellow below.

```
<vcp_skin>
        <background>
                c:\cncm\resources\vcp\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>
```

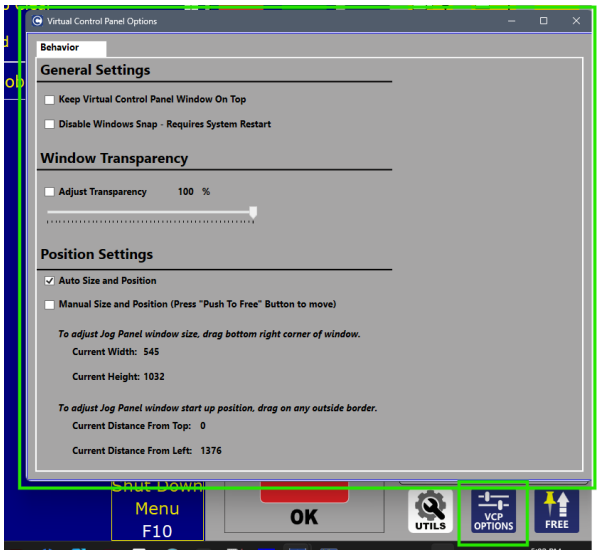
--- OR ---

```
<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.

- 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\Plcmsg.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

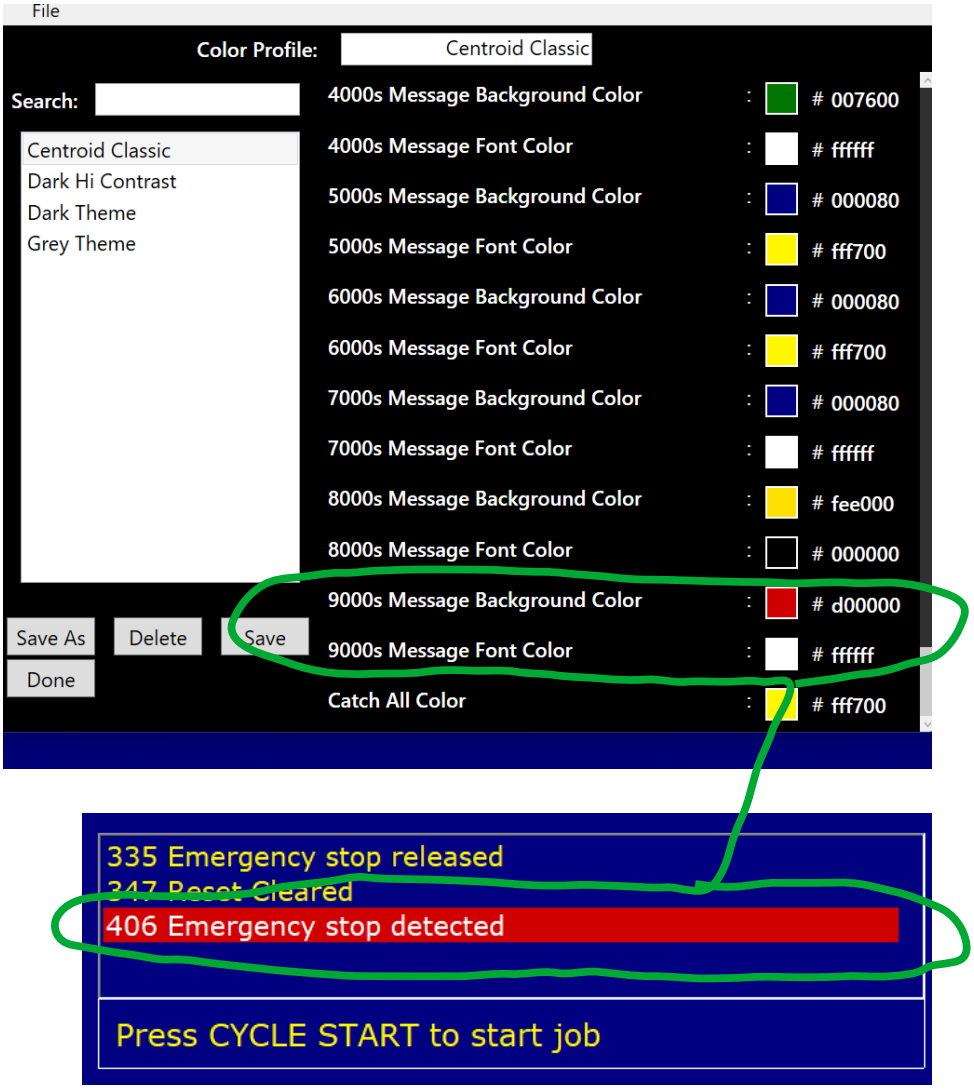
Warning messages use the 7000's, default colors: Blue and White

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.

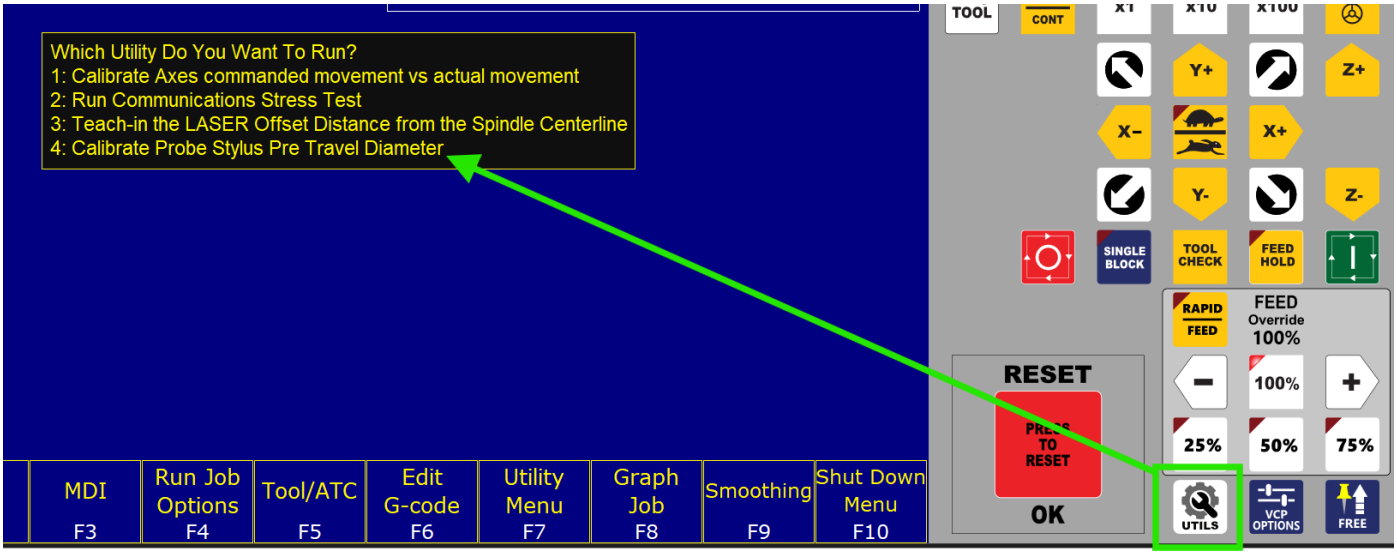
(Refer to CNC12 Operator Manuals for more information on all types of messages.)

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

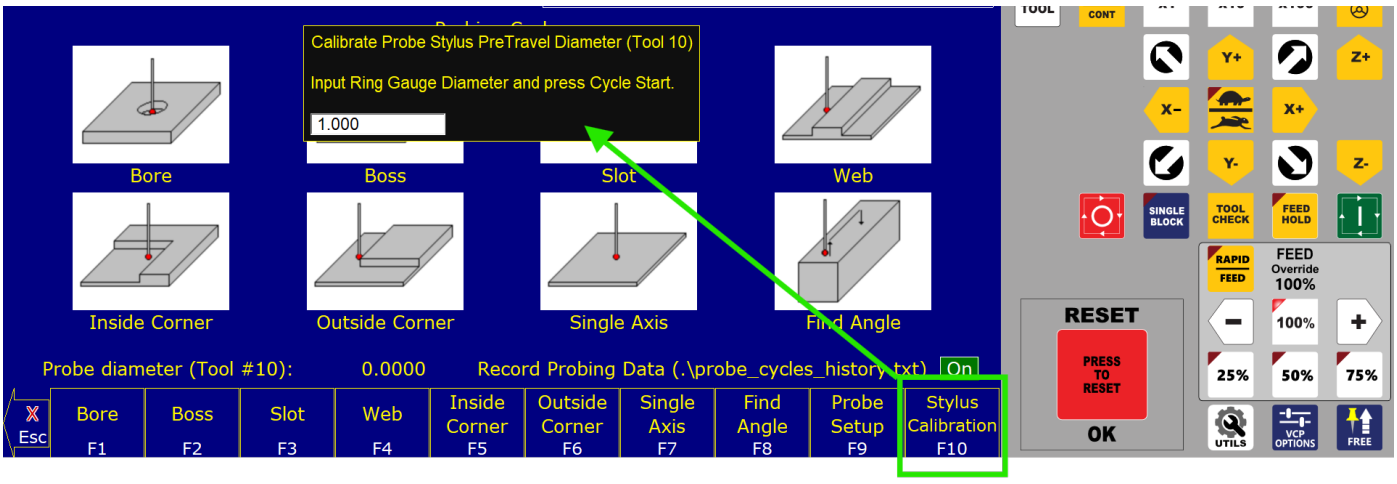


13.) Probing

- 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.



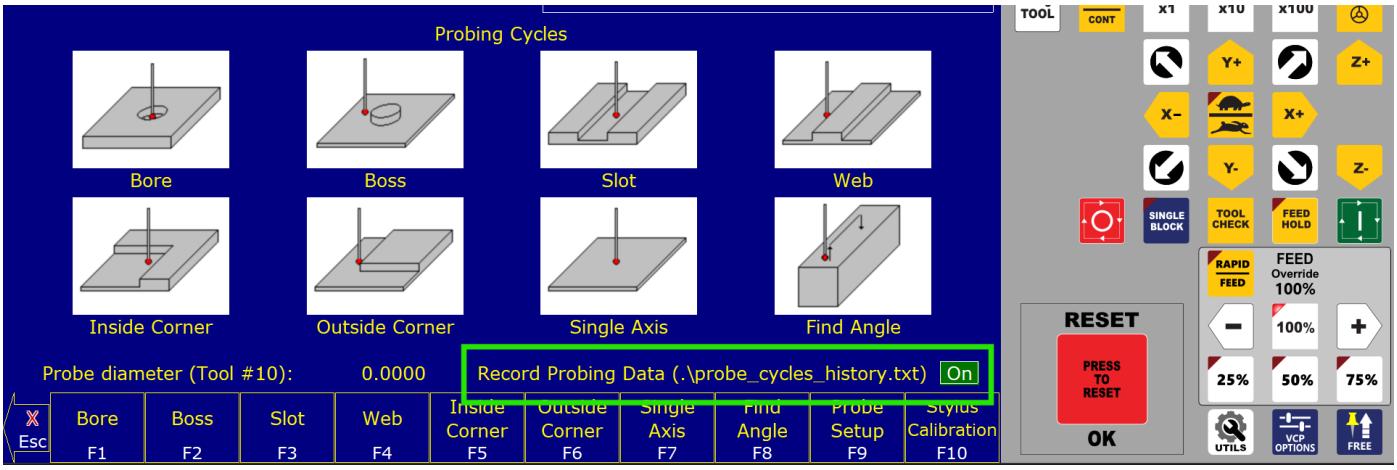
and directly in the probing menu as well.



- 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

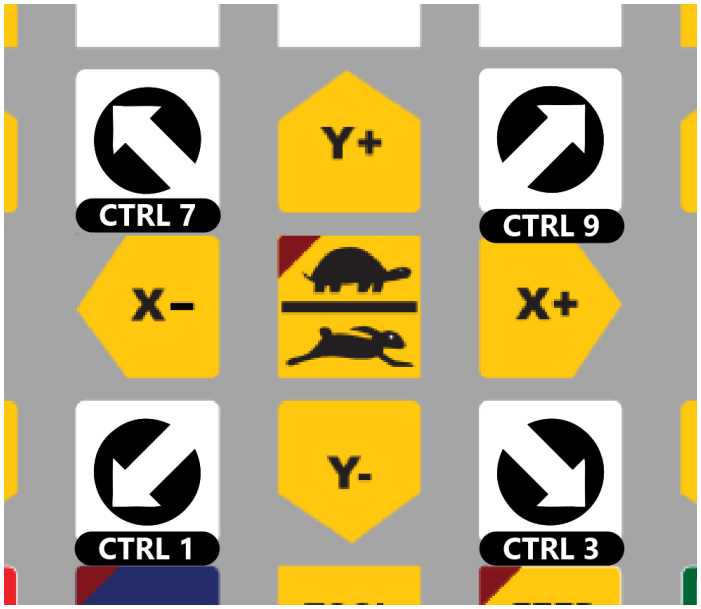
- Added "Return Z to Home" option in the router touch plate menu

- 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 willl 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

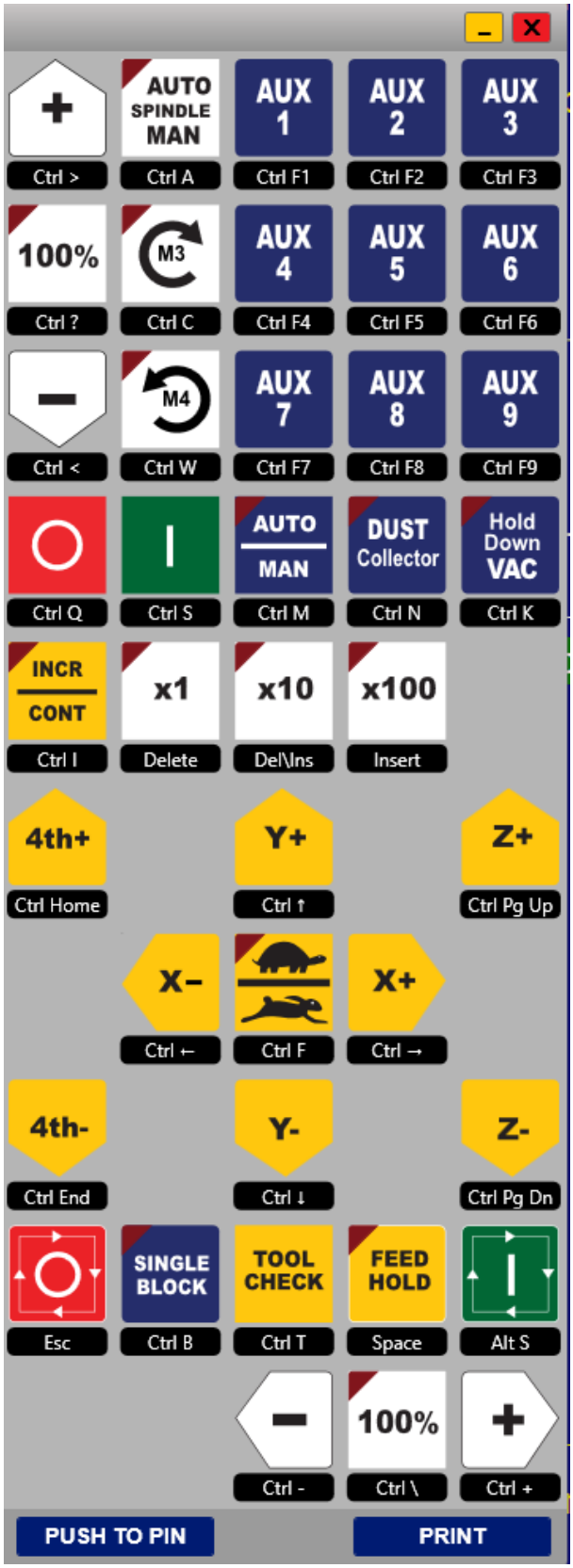
CTRL 1 = X - , Y – at the same time

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.



- 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 scrooling 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

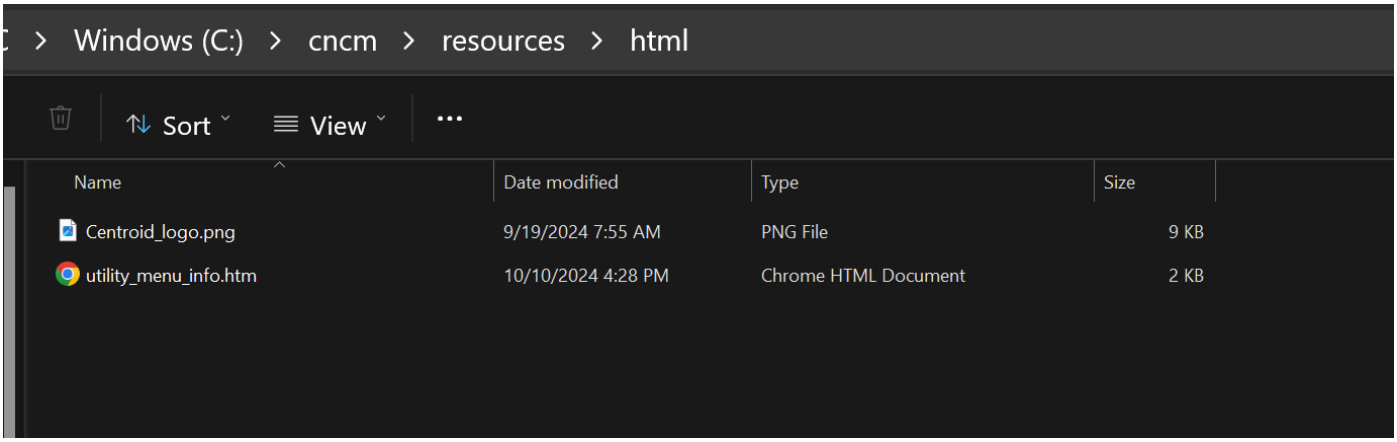
you can now use the html a href tag directly within the message m code.

M221 #100 "This will take you to the best website ever"

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.

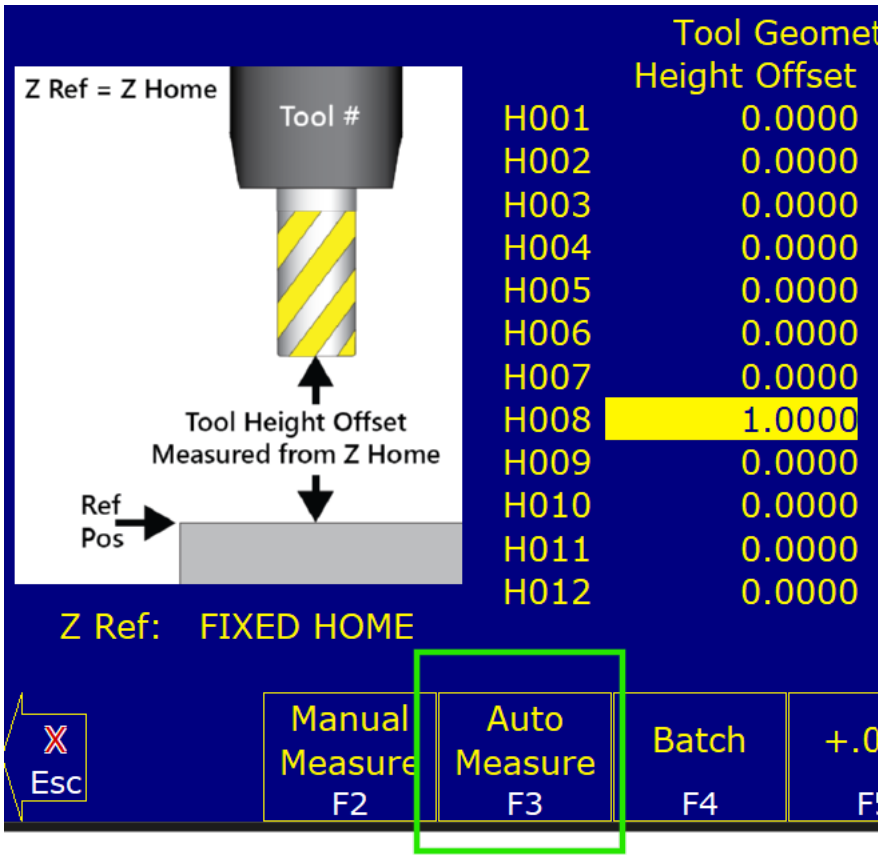


- Added Support for the Applied Motion EtherCat open loop stepper drive STF06-EC to Hickory CNC12 This drive is only recommended use is for accessory motion (think ATC Carousel or Rack in/out type applications) this drive is NOT for cutting motion.

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

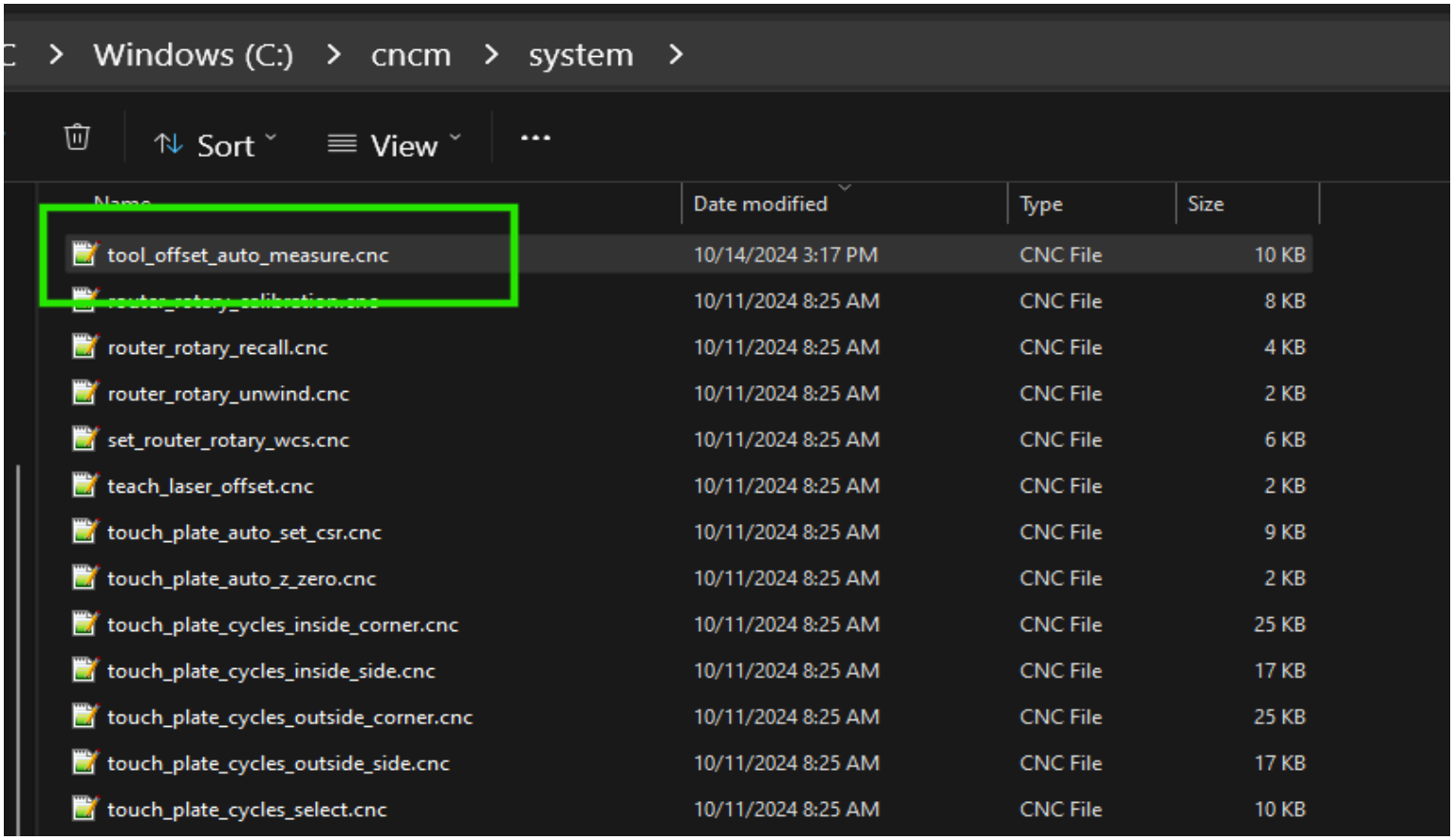
- Improved the Wizards Print (CRTL+P) feature so the Wizard screens print nearly a full page width of 8.5x11 piece of paper

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

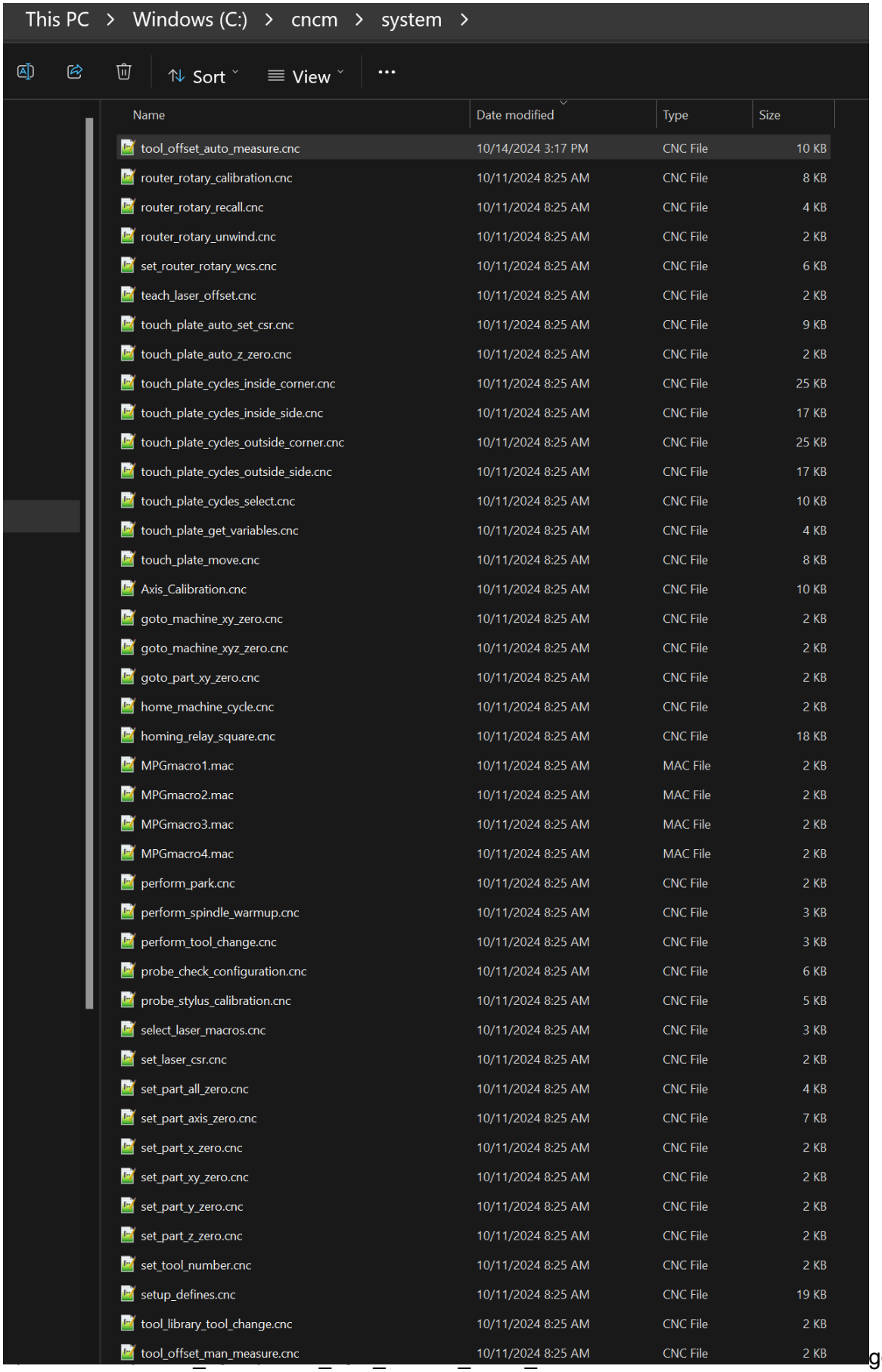


The CNC12 Router Version of this macro ask the operator if they would like to Automatically go to the Tool Touch Off fixed position or Manually Jog the Tool over the TT. Auto is used for measuring tools that can come straight down on the TT's location. Manually jog the tool over the TT position and then 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

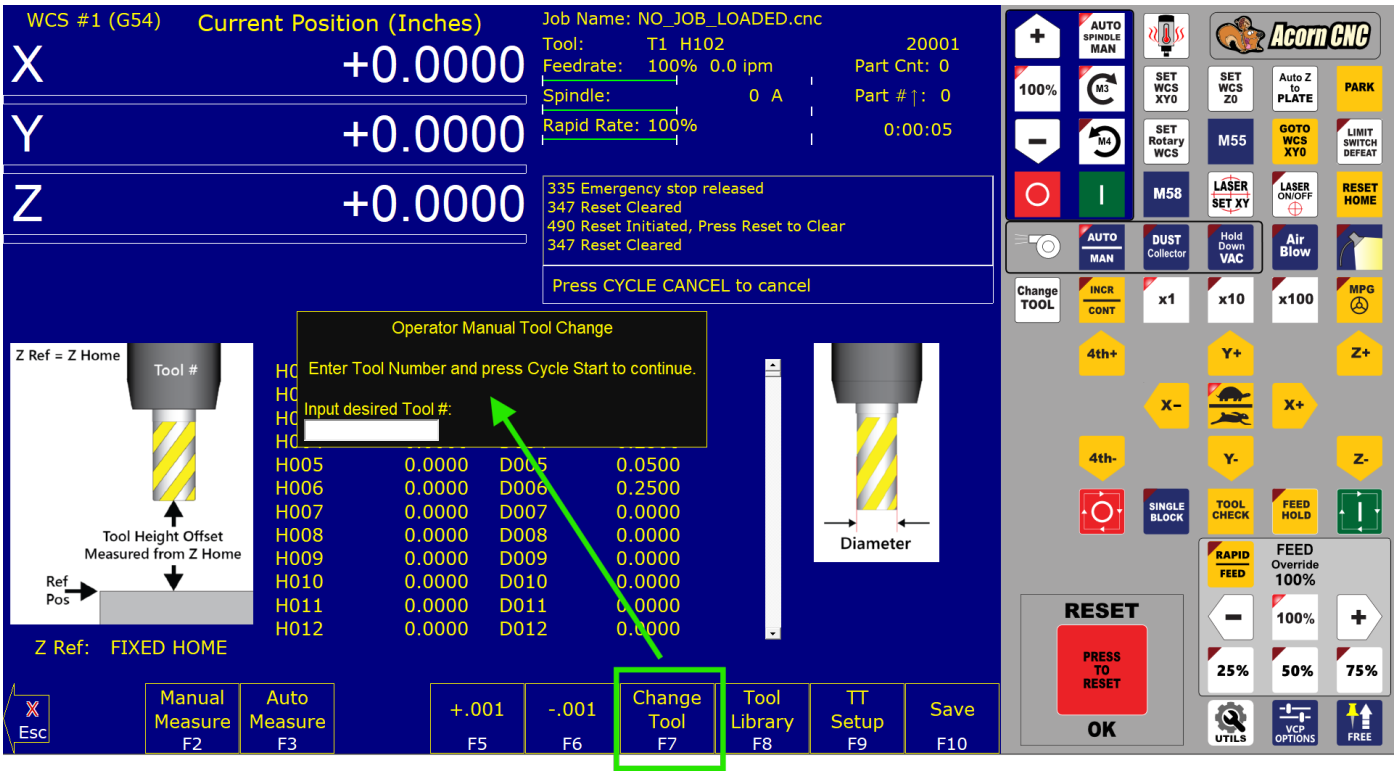


Related to that, we have been working through as many Function keys inside of CNC12 to break them out to user editable macros so they can be customized. These new macros can be found here. C:\cncm\system

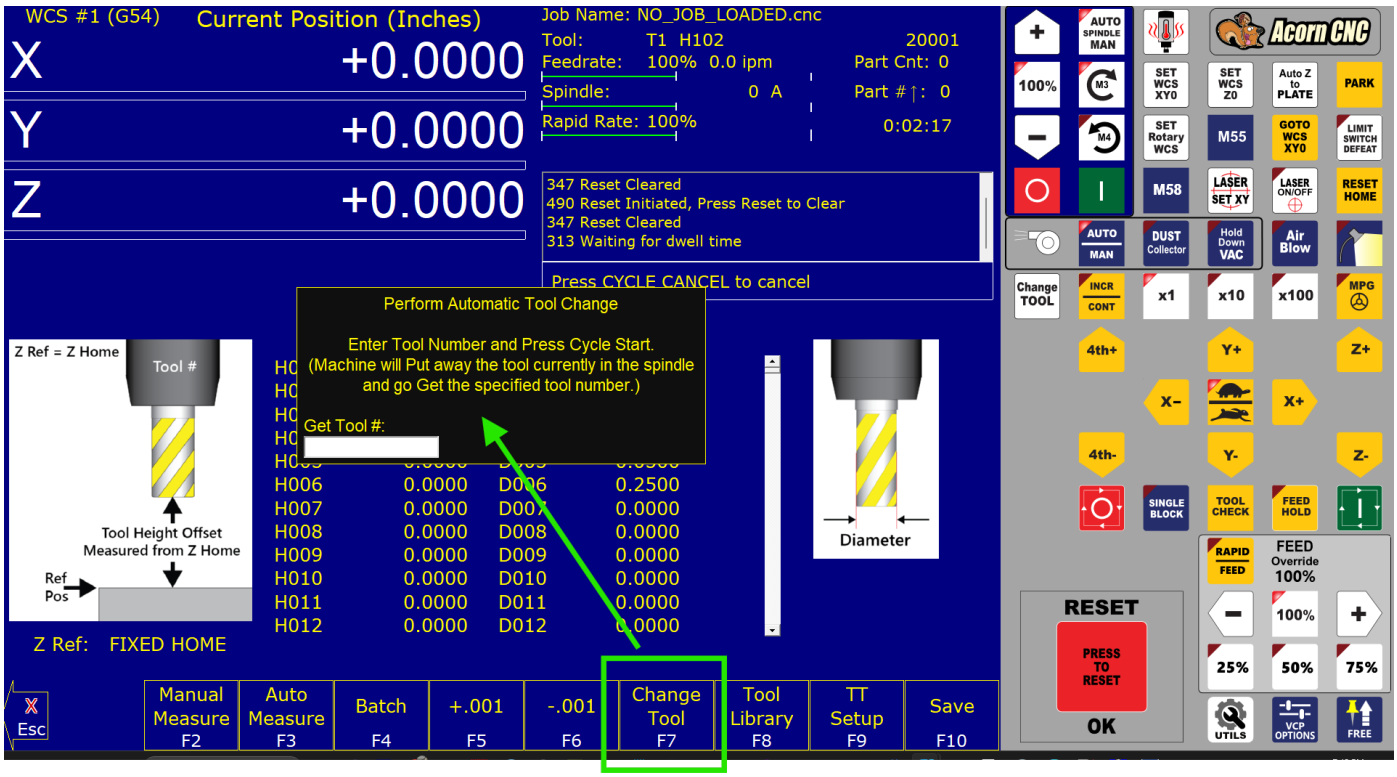


- If no ATC F7 = Operator Manual Tool Change

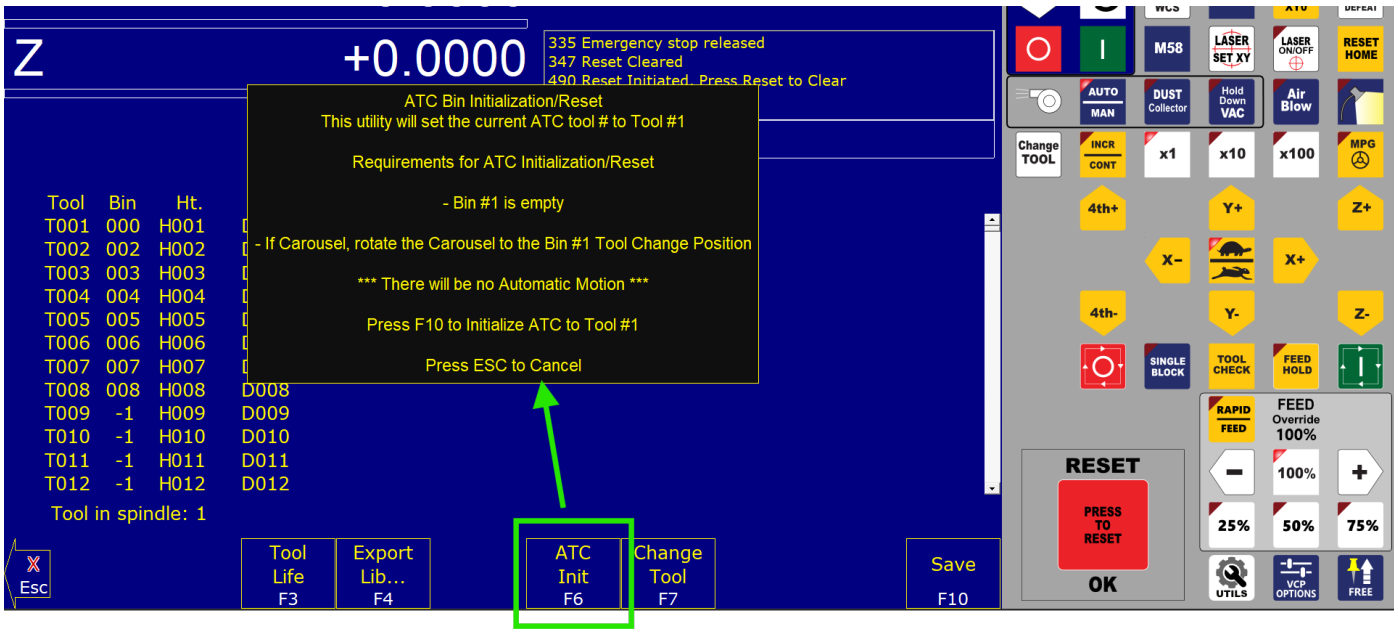
Put the tool you want in the spindle and let CNC12 know which tool you loaded in the spindle manually.



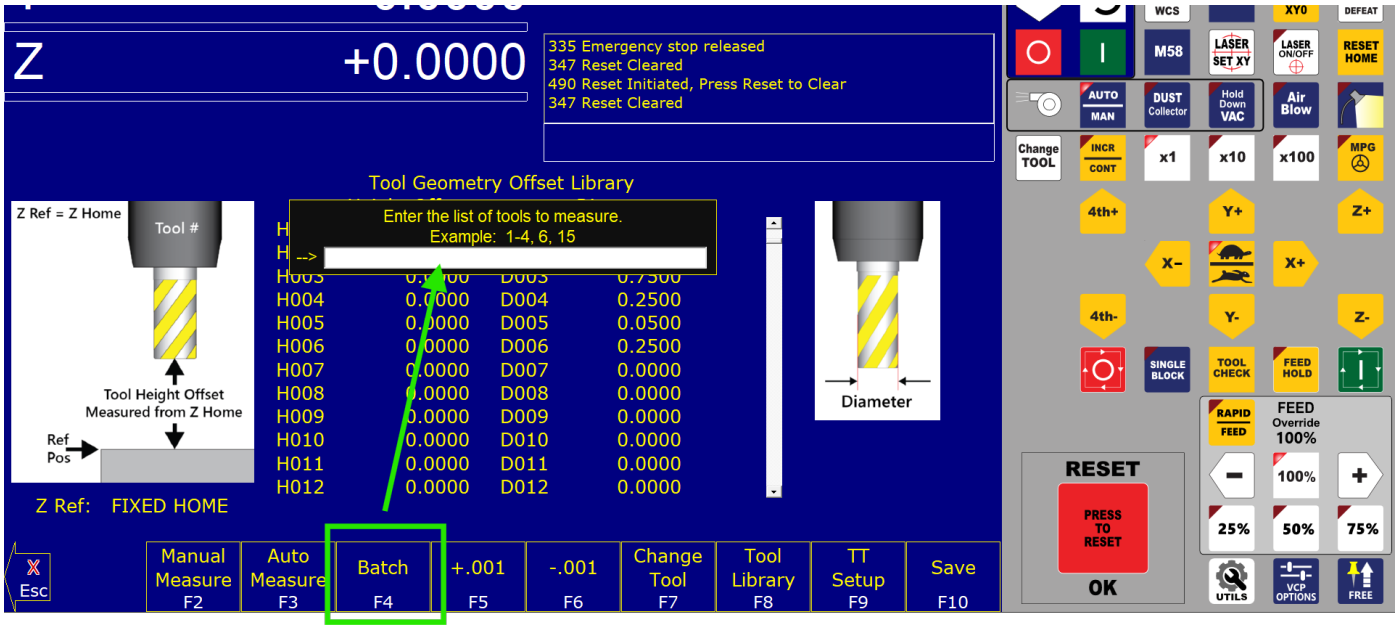
If ATC present F7 Auto Tool Change will put and get tools automatically



- 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:

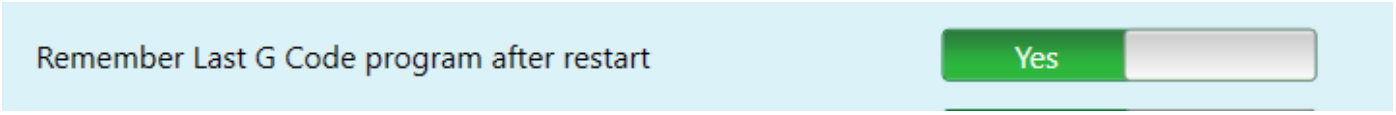
- Enter “1” and press cycle start and machine will put current Tool away , pick up Tool #1 and go touch it off the fixed TT.

- 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).



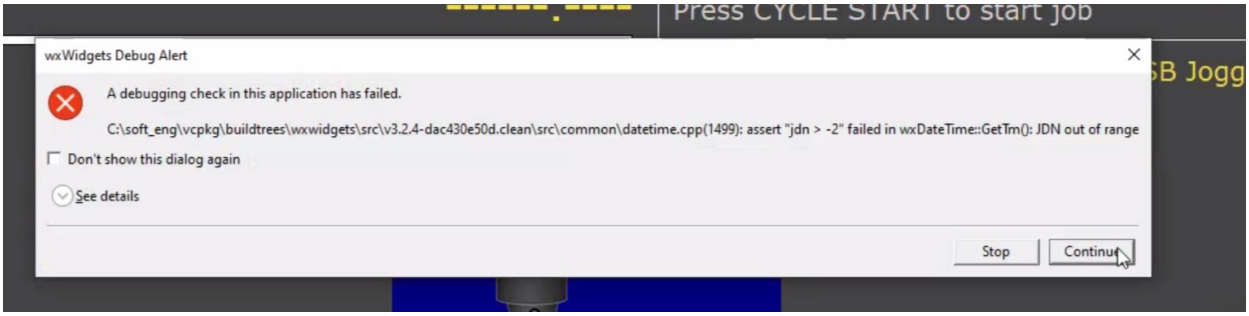
Added Default Jobs (g code programs) that will be loaded upon the very first start up of CNC12, instead of defaulting to “no_job_loaded.cnc” Any subsequent start/restart of CNC12 will of course remember what ever job you had loaded upon the next start/restart of CNC12.

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.

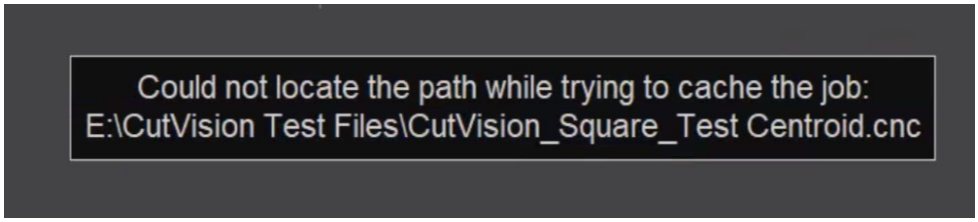
- Fixed a rare but annoying windows wxwidgets message related to loading a g code file remotely.



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.

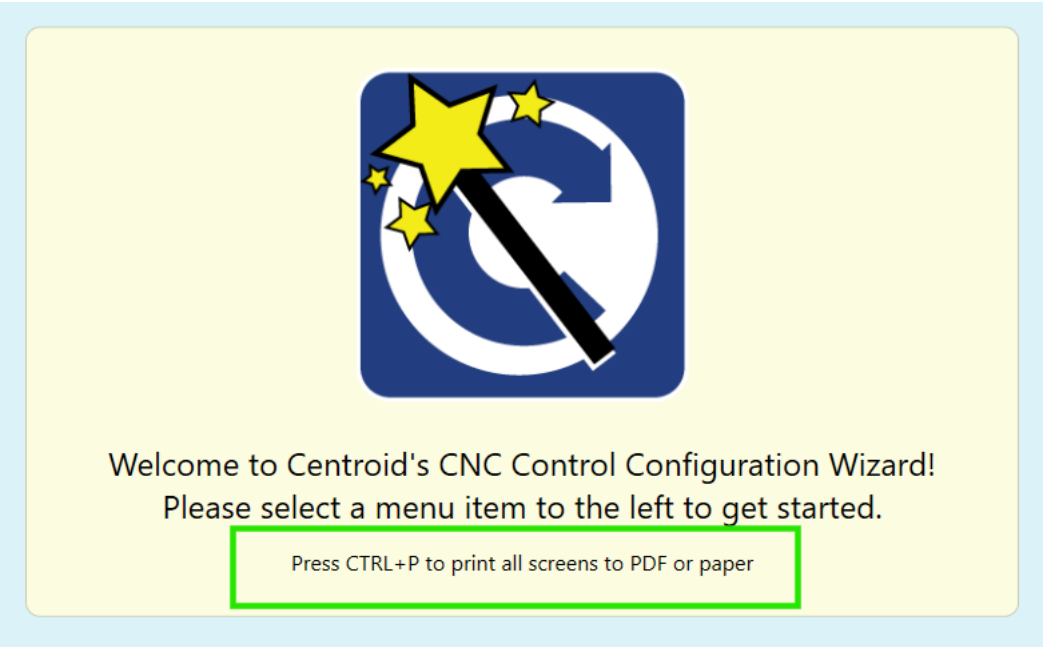
When using P4=26, CNC12 caches a copy of a g-code program locally from a remote drive, this feature was useful back in the day of small hard drive space, not so much anymore. (Recommended setting for Parameter 4 these days is 25 and the default is P4=25.

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

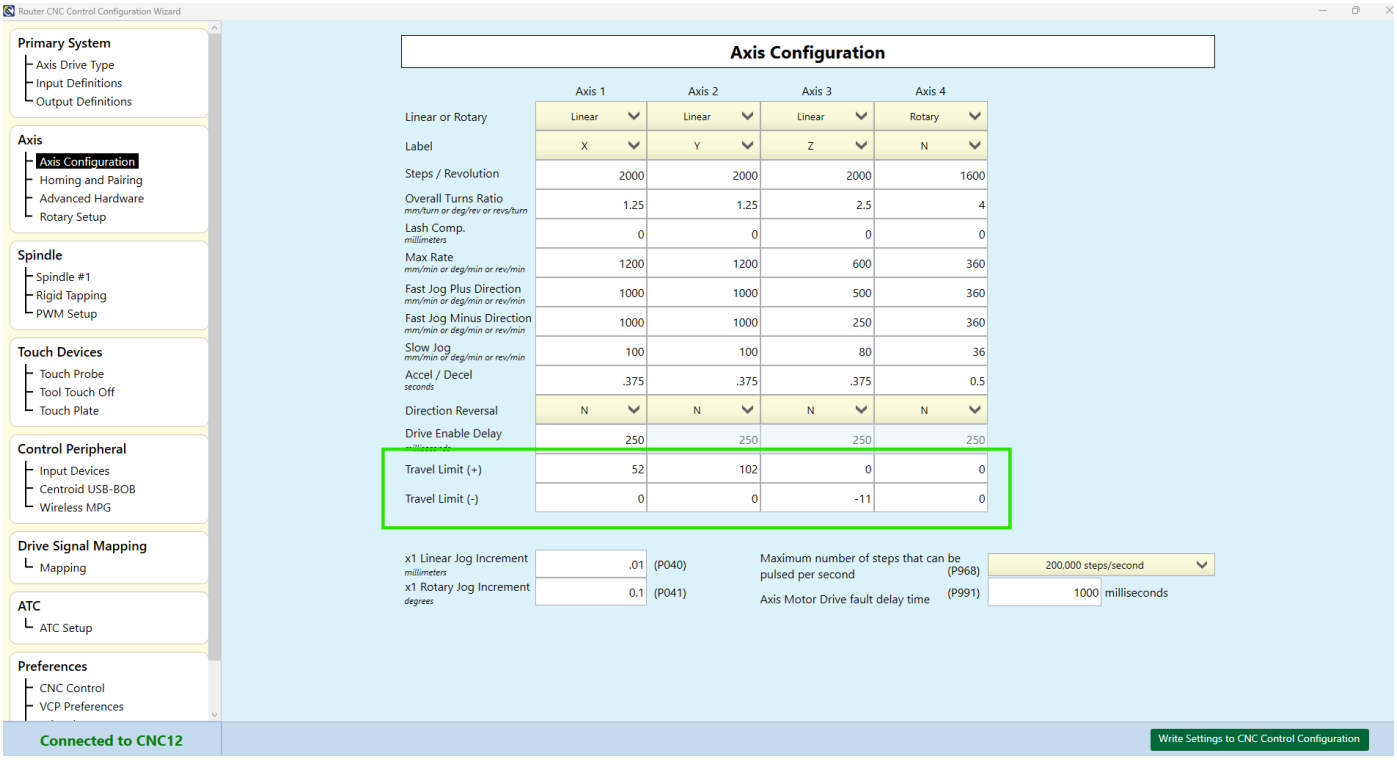


More information on Parameter 4 can be found in the CNC12 Operator manuals.

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

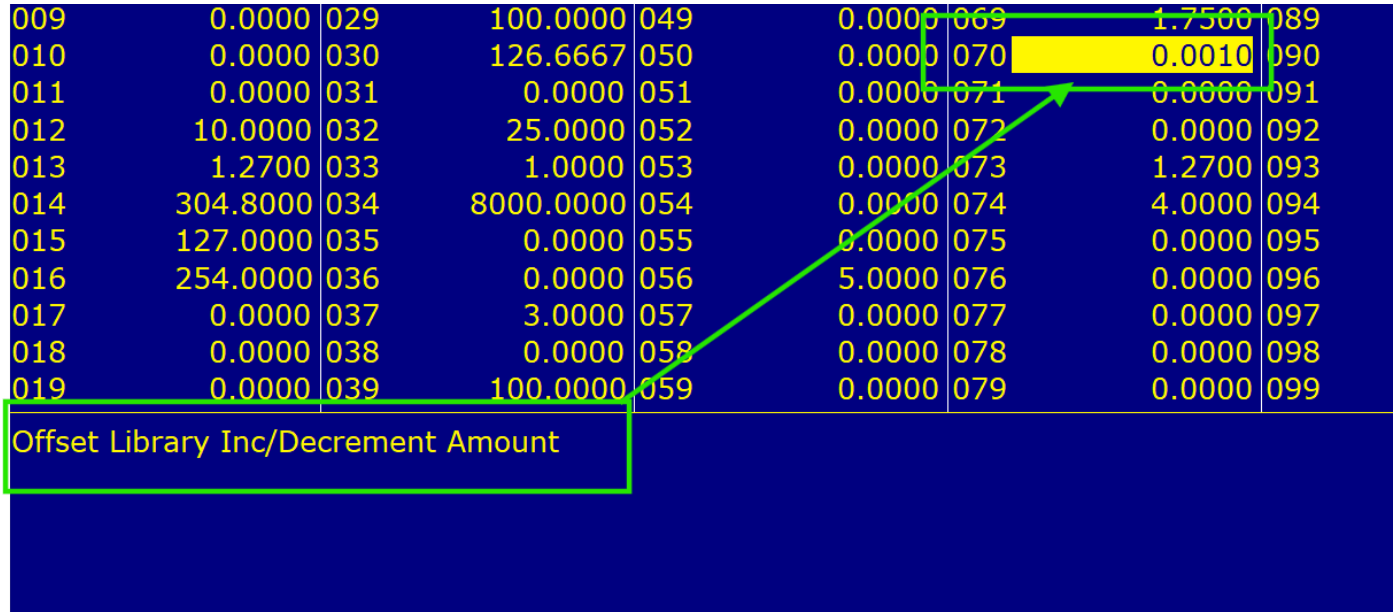


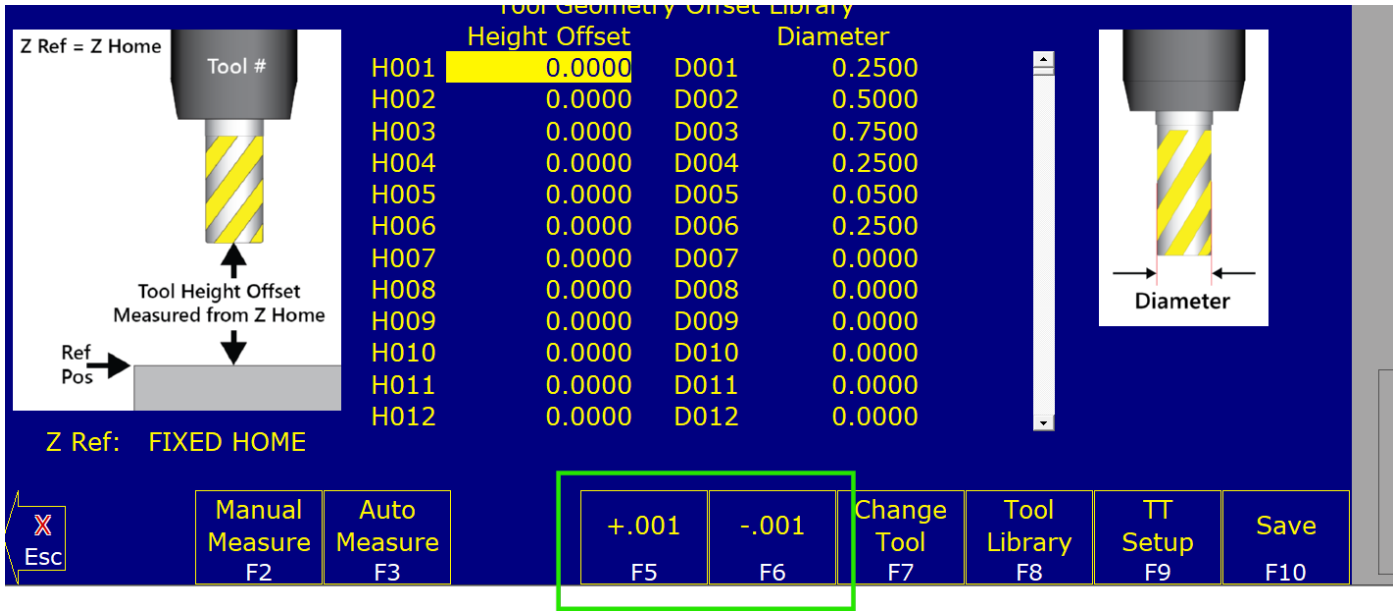
- Moved the Wizard Axis travel limits into the Axis Configuration menu



- 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.

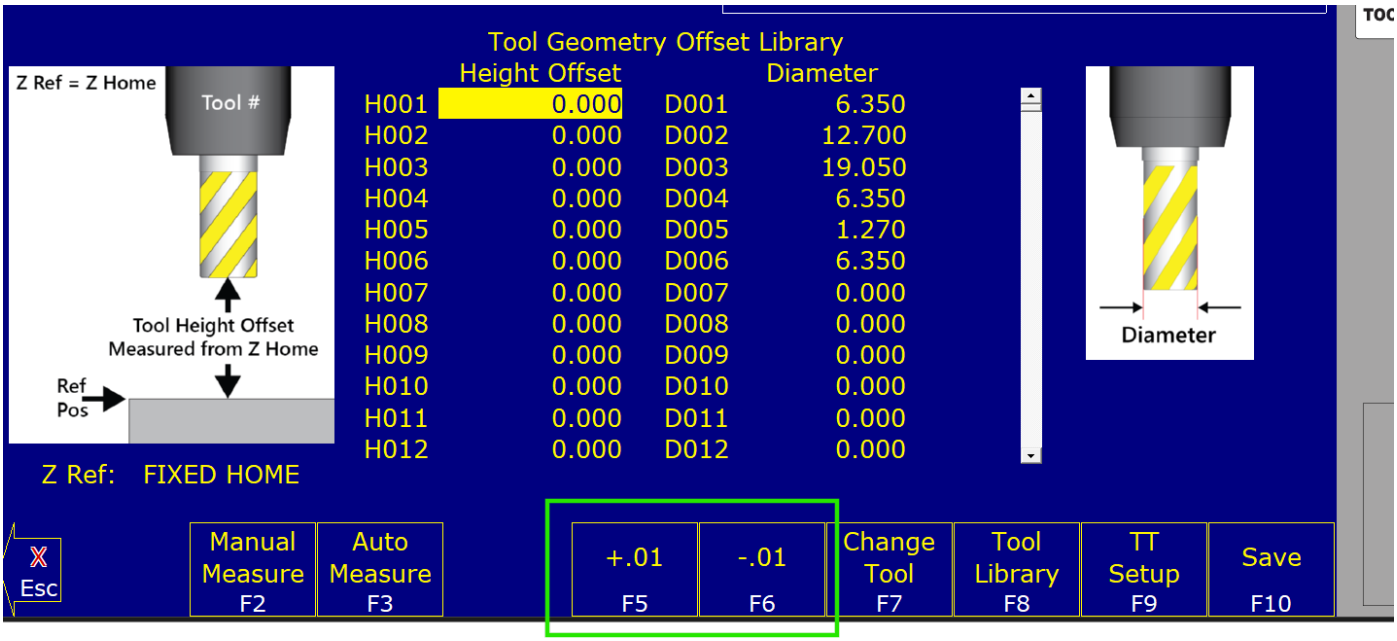
Default value for P70 is .001, (when in inches this is .001” when in mm this is .001 mm.)





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.



- 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

1.) Oak, Allin1DC and MPU11 users should be aware: running the CNC12 v5.20 on top of the existing cncm\t 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.