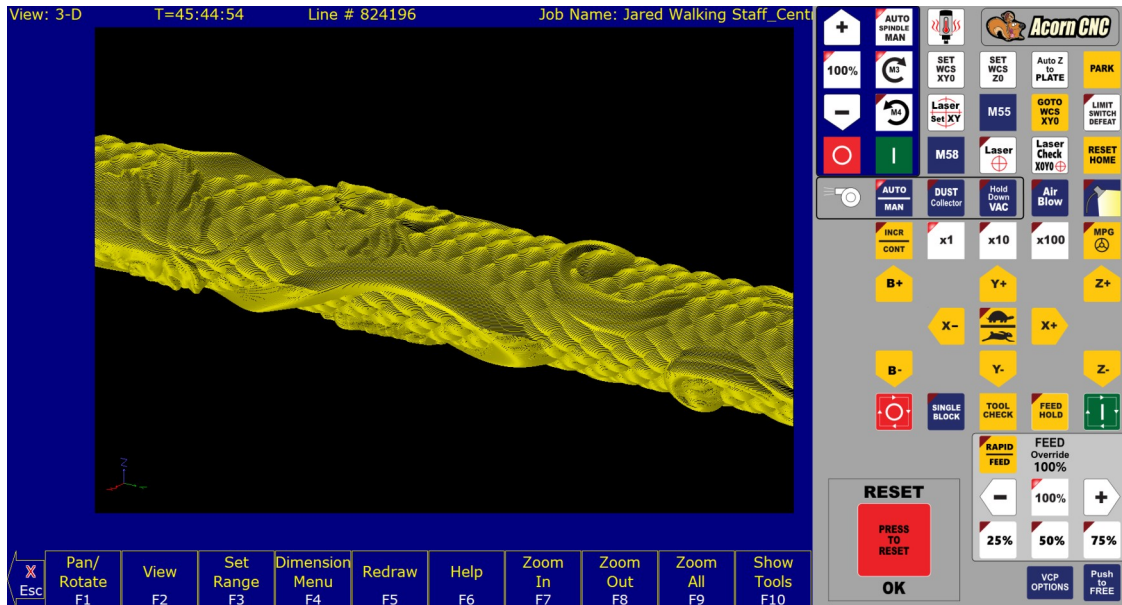


- 1. Rotary axis labels and G code now work in any combination parallel to the X or Y axis. Same with g code backplot graphing of Rotary moves in any combination of other axes.



- 2. Added a Rotary setup page added to Wizard

### Rotary Setup

|   |           |                |
|---|-----------|----------------|
| Rotary Axis   | A         |                |
| Axis Parallel To Rotary Center Line   | X         | ▼              |
| Center of Axis (Y) in machine coordinates   | 10        | (P116)         |
| Center of Axis (Z) in machine coordinates   | 12        | (P117)         |
| Rotary Axis Jog Increment   | 0.1       | degrees (P041) |
| Rotary DRO Display Type   | Rotations |                |
| Slave Rotary axis feedrate to a linear move feedrate on the same line*                  | No        | (P002)         |
| Rotary-only moves won't use a modal feedrate set by a prior rotary and non-rotary move* | No        | (P002)         |

*\*This feature has no effect for movement commands handled by Smoothing (P220=1)*

- M151 now also works with any rotary axis. (previously only available in CNC12 Lathe for C axes) M151 when used with a Rotary axis acts as a “DRO unwind”. M151/A will reset the DRO to a value between 0-360 degrees. So for instance if the Rotary axis has “wound up” past 11 turns and is at 11R121.462 (121.462 degrees)

A screenshot of a CNC control display. The background is dark blue with white text. On the left, the letter 'A' is displayed in a large font. To its right, the text '+11R121.462°' is displayed in a slightly smaller font. The display is framed by thin white lines.

Issue an M151/A and the DRO changes to:

A screenshot of a CNC control display, similar to the one above. The background is dark blue with white text. On the left, the letter 'A' is displayed in a large font. To its right, the text '+121.462°' is displayed in a slightly smaller font. The display is framed by thin white lines.

Effectively “un winding” the Rotary axis with NO MOTION. Resetting the DRO position at the same degree position but removing the number of full rotations.

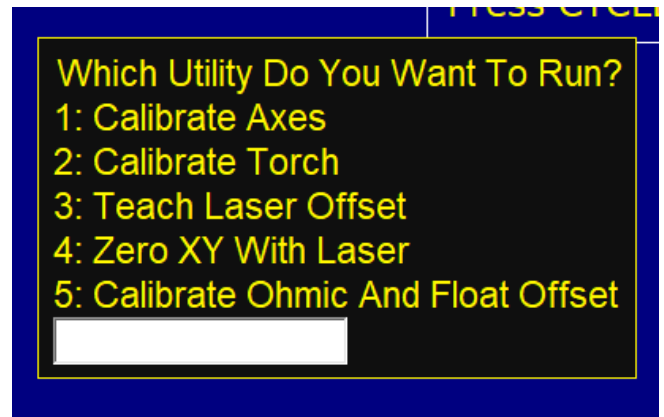
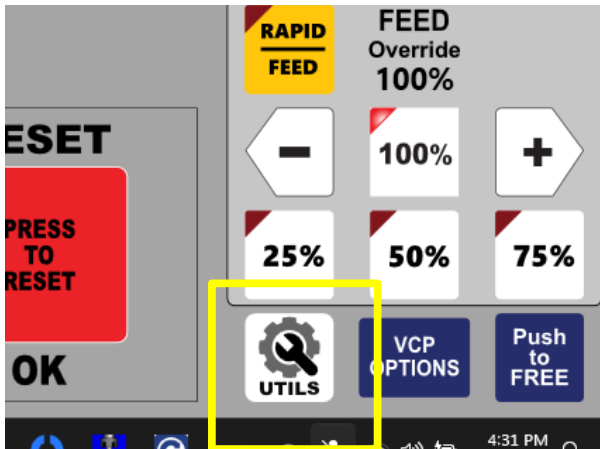
For example: Then a G0A0 would result in a 121.462 degree movement to get back to A0

Use MDI or put the M151 in your G code program or assign a VCP button if you find yourself doing this a lot.

M151 Demonstration Video. <https://youtu.be/hl4VmpGTG30>

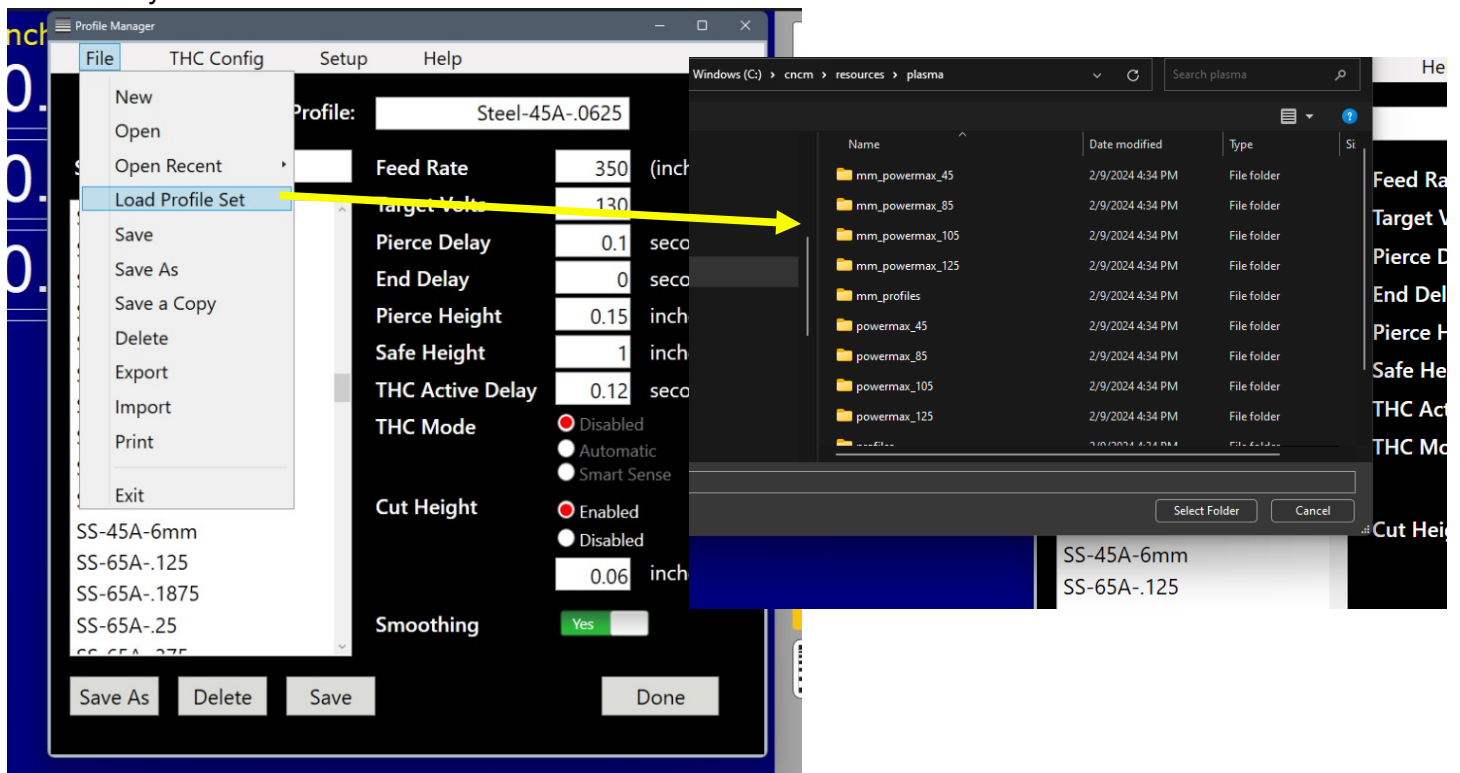
- Improved smoothing speed control for low step resolution per linear inch PLASMA machines. Turn on/off this new experimental feature on with Parameter 225.  
Set P225 = 1 to turn ON.  
Set P225 = 0 turn OFF (use the ‘old’ high resolution step smoothing)  
P225 is only available with Plasma CNC12.  
This experimental feature fixes the “feedrate going faster than profile manager set rate in certain conditions” on low resolution machines with short vector g code programs. ( low step resolution = turns ratio near or below 1 with 1600 steps per revolution drive settings. ) This is a software fix for machine mechanical short comings. When every possible mechanically design and build the machine kinematics with good motor step per inch resolution. [See this post for more info.](#)

- Added Utility button VCP with stock macro for common utility style functions (user editable and removable of course just like any other VCP button. Plasma Shown below.



- "Added additional probe protection based on the probe tool number for single input Touch Probes. Now a NC Probe will work without the additional Probe Detect Input as long as the loaded Tool Number matches the value of parameter 12 (The Touch Probe Tool Number). Said another way probe protection will be active when the tool number of the Probe is also active in CNC12. There are no changes to any previously supported probe trigger and detect input configurations."
- Added frequency tracking of ethernet packet communication send and receive errors with a Windows style pop-up warning to the operator when more than 100 send and receive errors have been logged within 1 hour.  
  
Parameter 267 controls the tracking frequency.  
P267 default value is 100.3 which is 100 messages per hour.  
100 messages per minute would be 100.2  
100 messages per second would be 100.1  
The number before the decimal represents the number of messages, the number after the decimal represents the unit, 1 = second, 2 = minute, 3 = hour  
Neglecting the decimal part would result in messages per hour (so just a value of 60 would be 60 messages per hour).
- On\_hover, on\_click VCP button graphical feature omission restored.
- Fixed screw comp axis labeling issue with AcornSix, now screw comp for an axis with a label will be applied to the paired axis labeled "N" as well. (in v5.08 to get screw comp to work the paired axis needs to be labeled (not "N")
- Plasma Restart Mode now has an "ignore list" that is used to tell Plasma Restart mode to ignore certain advanced g code techniques that are not compatible with Plasma Restart Mode. G code jobs with any thing that is on the ignore list will not have a restart file created nor will the job be recovered upon an interrupt (collision). This is list used to stop Restart mode from failing on advanced parametric style g code that has been variabilized.
- Plasma Restart Mode received a graphics engine upgrade and is now controllable with the mouse in addition to the traverse buttons, Zoom while in restart mode, Select Lead in point, Select pickup point, Select Pierce Point with mouse. All existing restart features are retain along with the new features.  
Joey's [Video link showing new restart features. https://photos.app.goo.gl/6oxqww51MPv1b9h37](https://photos.app.goo.gl/6oxqww51MPv1b9h37)

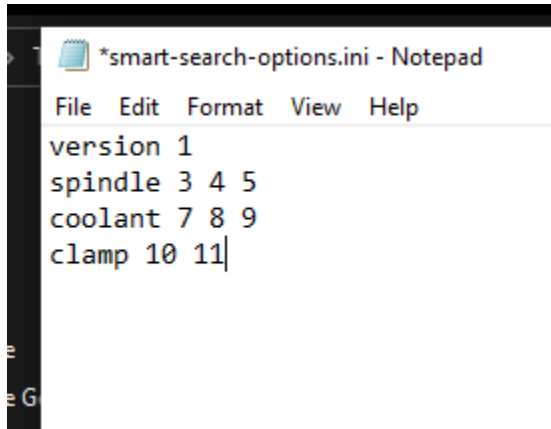
12. Added the ability to easily change profile sets in the plasma profile manager (between different amp rating units Powermax 45, 85,105 and sync, etc.). No need to copy/rename profile folders in windows anymore.



13. Fixed issue where the G10 command was limiting the R word value to the min/max parse distance settings (999999.9999).
14. G65 will now accept L0 as a valid repeat option, resulting in the G65 call being skipped
15. Added CentroidAPI calls GetSerialNumber and ImportLicense
16. Moved startup options from the properties in the shortcut to a file called startupOptions.ini. This allows for startup options to be defined when launching the program directly from the executable
17. Deprecated Centroid API calls: CNCSkinning.dll and CNCSkinningUserControls.dll. And replaced them with CentroidAPI.dll and CentroidAPIUserControls.dll, all previously deprecated functions have been removed. A 32 bit copy of CNCSkinning.dll will remain in the cncm/t directory for support of user created applications but will no longer receive updates.
18. Fixed API Call "GetCurrentHeightOffsetNumber" in Acorn
19. Added usb dongle status to the offline intercon title bar to make it very obvious that the USB key is found and working or not.

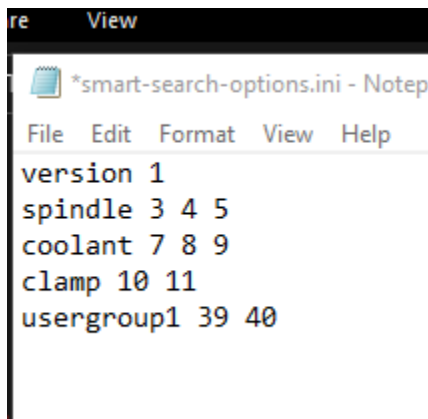
20. Added Smart Search Groups, controlled by a file called smart-search-options.ini, which allows for user controlled smart search groups

Upon startup, cnc12 will create a file called smart-search-options.ini preloaded with version, spindle modal macros, coolant modal macros, and clamp modal macros.



```
*smart-search-options.ini - Notepad
File Edit Format View Help
version 1
spindle 3 4 5
coolant 7 8 9
clamp 10 11
```

Up to 9 additional groups can be added by a user in the form of “usergroupX” where X is the user group number, 1-9



```
re View
*smart-search-options.ini - Notep
File Edit Format View Help
version 1
spindle 3 4 5
coolant 7 8 9
clamp 10 11
usergroup1 39 40
```

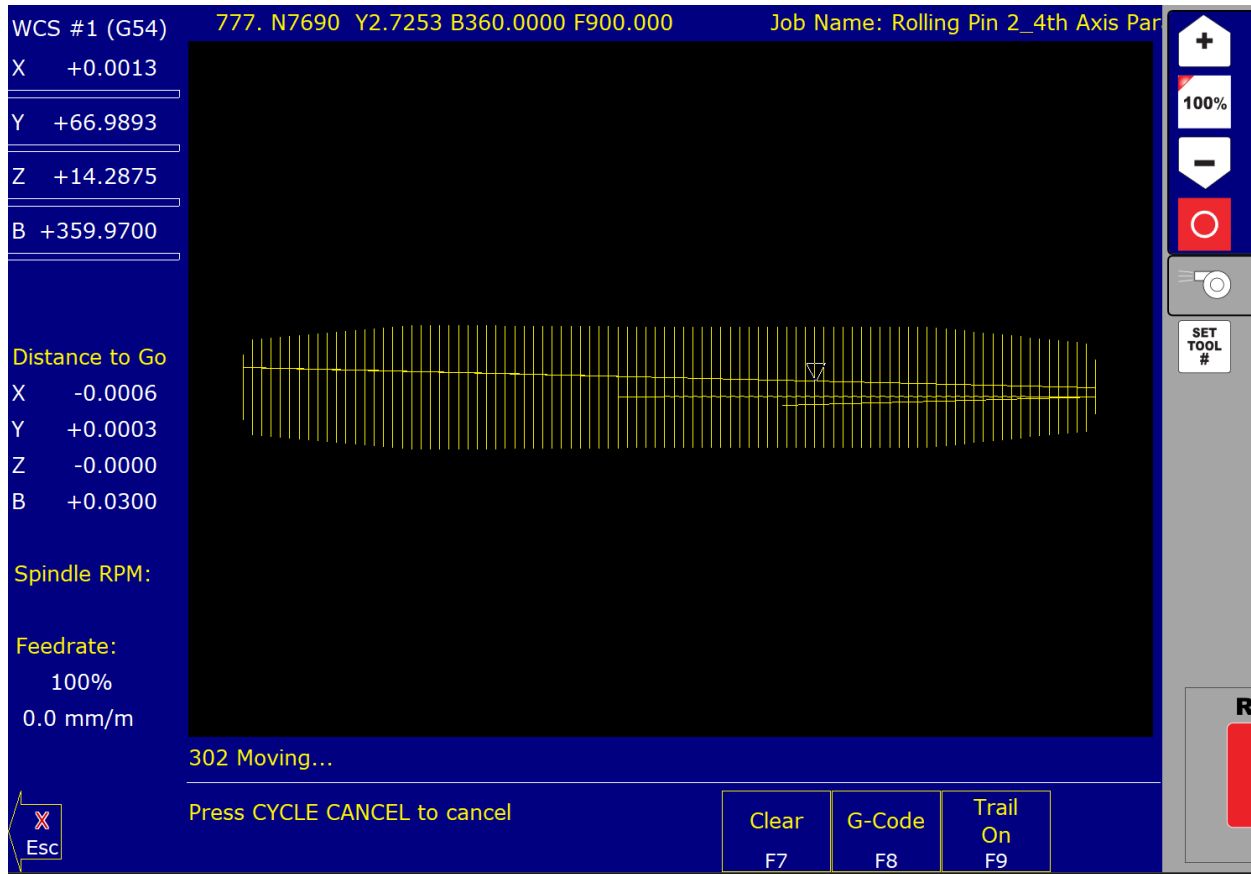
Then when run-search is used, this file will enable the last known macro in each group. For example, consider the following G-code

```
N104 ; TOOL - 01 DIA. - .2500 2D-SWEPT..
N106 M25 G49 ; Goto Z home, cancel tool length offset
N108 G17 G40 ; Setup for XY plane, no cutter comp,
N110 G20 ; inch measurements
N112 G80 ; cancel canned cycles,
N114 G90 ; absolute positioning,
N116 T1 M06
N118 S2000 M39
N120 G00 G54 X1.125 Y0.
N122 G43 H1 Z1.
N121 G1 X1.125 Y0 Z-.75
N124 G1 Y.75 Z-.75 F24.
N126 G2 X1.25 Y.875 R.125
...
N316 X-.1192
N318 Z-.6649 F12.
N320 Y.75 F24.
N322 G2 X1.25 Y2.1192 R1.3692
N324 G1 X2.
N326 Y2.125
N328 Z-.75 F12.
N330 X1.25 F24.
N332 G3 X-.125 Y.75 R1.375
N334 G1 Y0.
N336 G0 Z1.
N338 G40
N340 M25 G49 H0
N342 G00 X0. Y0.
N344 M40
; End of program
```

Searching for line 124 will turn on M39 as it is the last known macro in that group, as it is written in line 118.

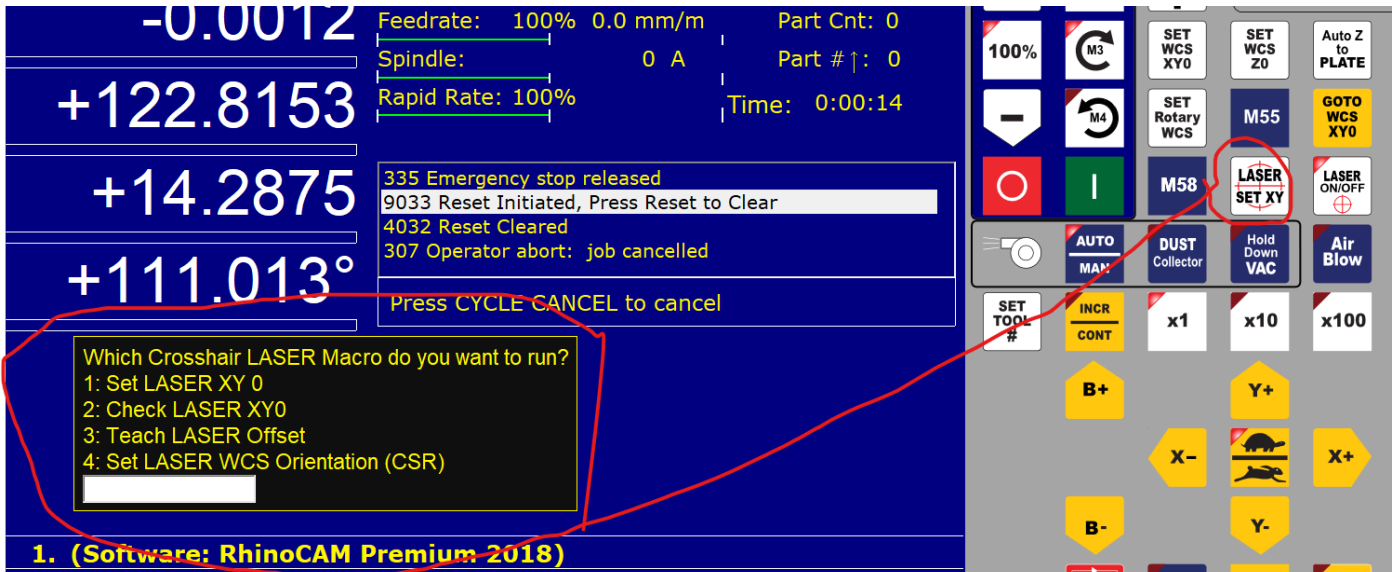
21. Fix a random ghost character graphics anomaly that would present itself in MDI mode
22. Switch to using the Feedrate Override % to override the Jogging speeds from the Rapid Override.
23. Added Centroid API call "SetToolLibrary" which allows the entire tool library to be changed in a single call
24. Simplified PLC Diagnostic App: Added Ctrl+alt+r to the simple plc diagnostic App to auto restore any user inverted/forced buttons back to the initial state when starting the App. Hopefully this will help users that get themselves mixed up since they forgot to turn an output back to "normal" or forgot to uninvert an input.

- 25. Fixed bug where when using the Simple PLC Diagnostic tool would crash when selecting the help or options buttons
- 26. Added command "-getversion" to the CNC12 installer to print the installed version to a file.
- 27. Rotary graphics are now applied to run-time graphics. This is temporarily limited to a top-down view only. This also includes a triangle tool indicator for rotary jobs

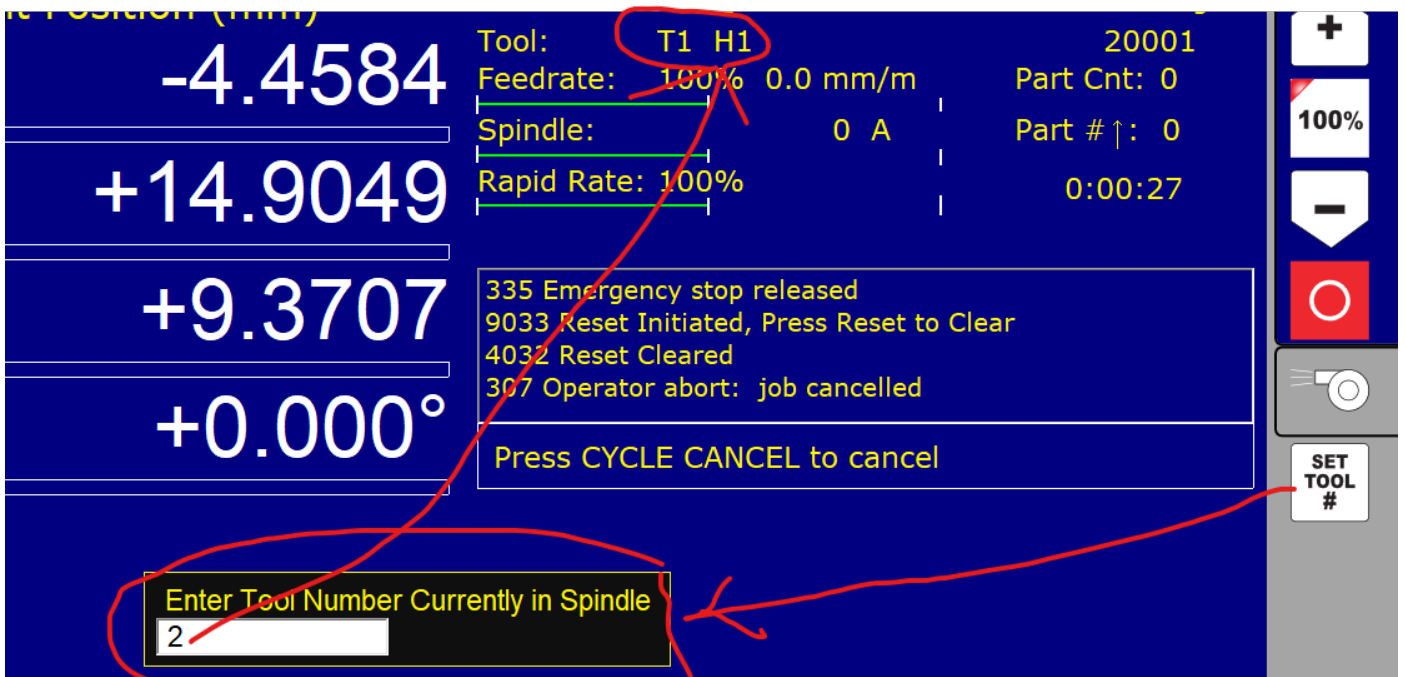


- 28. Jogging speed override now uses Feed Override percentage for speed calculation instead of Rapid Override speed percentage.

29. CNC12 Router and Plasma have improved Cross Hair Laser VCP buttons and macros. Now one button does as many Cross Hair laser functions as you want. This is a simple user editable macro.



30. Router CNC12 VCP now has a "Set Tool #" button to make it easy to set the current tool number for router users that don't use the "mill style" part Z zero menu.





31. added plc logic for "Spindle up to Speed" input for use with M3 and M4 so now user does not have to edit the M3 or M4 macro, the M3 or M4 will automatically wait for the Spindle up to Speed input before continuing on with the G code program. A message will be displayed in the message box letting user know that CNC12 is waiting for the spindle to come up to speed before continuing on with the job.
32. Added PWM Velocity Modulation to Hickory.
33. Added "--image [IMAGE\_PATH]" as a command line argument to the VCP. This create a .png of the vcp and save it to the specified image path. The vcp will then exit. This does not require connection to CNC12 or the Centroid API.
34. Added a temporary measure to withhold the System Test requirement message if home/limit switches are not setup or a paired axis exists
35. Fixed M98 skipping a call if an L command was not specified. Related to G65 accepting L0
36. Fixed VCP text feature not parsing text for buttons
37. Added Warning message when closing the Simple PLC diagnostic tool after having forced on any output(s)
38. Consolidated Cross Hair Laser Buttons to free up a Router and Mill VCP button space for Set Rotary Zero.
39. Fixed graphic anomaly where homing direction indicator in the Wizard for the 4<sup>th</sup> axis would not stick on the selected M91/2 setting. This was a graphic only bug and did not affect the actual homing process.
40. Fixed RTG DRO distance to go position Graphic overlay anomaly for Lathes that have Turret axes being displayed as an axis.
41. Fixed G code Search bug where two tool changes would occur when searched for a line that had a macro on it which would result in double tool changes and other unexpected results.
42. Increased the CNC12 Bitmap graphics resolution to maximum, now all bitmaps (.bmp's, jpegs and .pngs) look better in CNC12 than before.
43. Added Soft Limit control to Plasma and Laser THC position so now THC will not exceed a soft travel limit.
44. Added API call "RefreshGraph" which will refresh either the rtg or regular graph screens

45. Plasma Intercon: Fixed bugs where a lead-in/out from a Circle Frame/Arc would be incorrect

46. All Centroid CNC controller boards now require a Power Cycle after a Hardware Firmware change has been made (for instance when installing a new version of CNC12 or switching from Mill to Lathe CNC12). Messaging has been added to let the user know to power cycle the Acorn/AcornSix/Hickory board after a Firmware Change and then restart CNC12.

**CNC12 firmware update complete!**  
**A hardware power cycle is required to ensure proper operation.**  
**Press any key to exit, power cycle the CNC Control board and restart CNC12**

47. Added logic to the Default M3 and M4 to support the canned Wizard definition "VFDUpToSpeed". When this input definition is used the M3 will wait for the input to be made before continuing on with the G code program.

48. Added to the Wizard Tool Touch Off device setup menu choices between the two main methods of measuring tool height offsets. Reference Tool method OR Z ref = Z home.

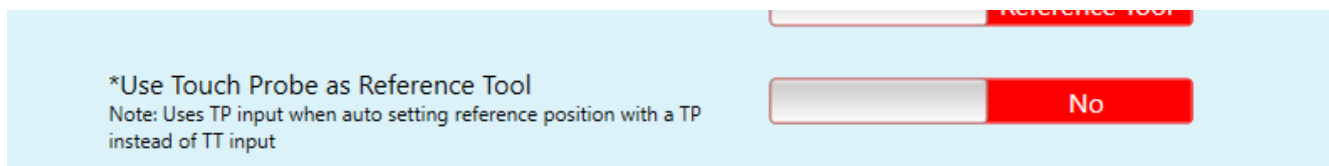
|  |                |        |
|--|----------------|--------|
| Tool touch Off type  | Mechanical     | (P405) |
| Input state when triggered<br><a href="#">*Probe Setup Documentation (PDF)</a>   | Closed*        | (P407) |
| Tool Measurement Reference Method  | Z Home = Z Ref | (P003) |
| *Use Touch Probe as Reference Tool<br>Note: Uses TP input when auto setting reference position with a TP instead of TT input | No             | (P043) |

49. Added to the Wizard Touch Probe setup menu the ability to enable/disable the new Probe Protection feature based on Tool number.

|  |     |        |
|--|-----|--------|
| Maximum probing distance   | 10  | (P016) |
| Probe Protection Enabled   | Yes | (P153) |
| Probe Protection based on Tool Number                              | No  | (P003) |
| Inhibit Spindle when Detect is on (Green)                          | Yes | (P416) |
| Display Warning to Verify that Probe or TT is functioning properly | Yes | (P410) |

## CNC12 v5.1 Known Issues:

1. 452 Ethernet Communication Error message has been inadvertently suppressed and will not display in the CNC12 message box or logs files. Even if the board ethernet cable is completely disconnected from the CNC12. Not to worry though, there are other CNC12 ethernet communication error messages that will appear even though this main error message has been inadvertently disabled. Does not affect CNC12 operation and will be corrected in the next version release.
2. Hickory Installer Option "Copy Manuals to Desktop" does not work. Workaround: Navigate to C:\cncx\ manual to find manuals. (X ofcourse is "m" or "t" for mill/lathe) Does not affect operation and will be corrected in the next version release.
3. G-Code Display shows up on the Setup Menu when if a bad password was entered for Config Menu or ATC Setup is run. Just Graphical Issue Does not affect operation and will be corrected in the next version release.
4. First Install of (Hickory Lathe) will produce a pop up message with "Invalid command line argument". Currently investigating cause, it appears to only happen on our fresh installs. Workaround : Install Hickory Lathe a second time, will correct the issue seemingly indefinitely.
5. behind the scenes Wizard logic is reversed on this selection so YES is NO..  
work around for example set slider to NO to have CNC12 to use a Touch Probe as the reference tool



## 6. Plasma only

The VCP button "Laser Set XY" macro 53 has 2 words flipped.

How to Fix it: Edit the file "mfunc53.mac" and change line 27 to read: G65 "\cncm\system\set\_laser\_XY.cnc"

(note the problem file looks like this G65 "\cncm\system\laser\_set\_XY.cnc")  
and has the words laser and set flipped.

## 7. Plasma only

The output selections in the Wizard under the "Custom" drop down are broken and cause the Wizard to crash.  
Work around , don't use them.

and fyi... These New tags soon will be shipping attached to every Centroid CNC control main board.

