Centroid CNC12 v5.40.04 Mill, Lathe, Router, Plasma, Laser Release Notes: 1-16-26

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.

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.) Bug fixes are free.

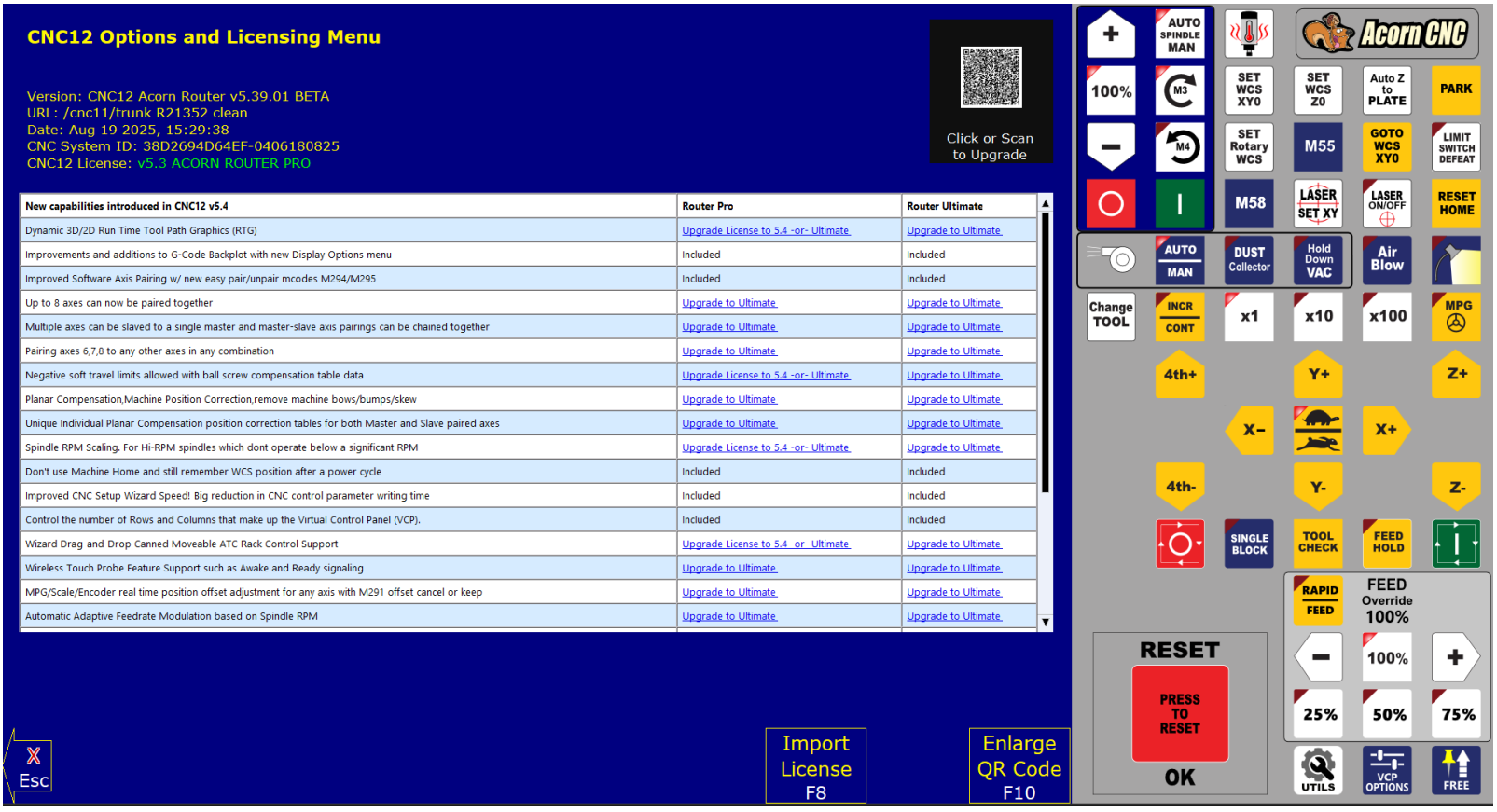
v5.40.01, v5.40.02 and v5.40.04 "point release" release notes start on page 32

V5.4 Release Notes

1. Starting with v5.40.00 Centroid has rolled out an optional software version feature upgrade charge for existing License holders. All bug fixes and some features and improvements are free. Your existing license file will work with this new version. Pay only for new features only if you want them for a modest upgrade dollar amount which will enable reinvestment in our team, ensuring the continued development of our CNC software that provides enhanced value for our customers (your controller keeps learning new tricks!) while maintaining the strong Centroid CNC support services that come along with each new version of CNC12. Read the details and FAQ here.

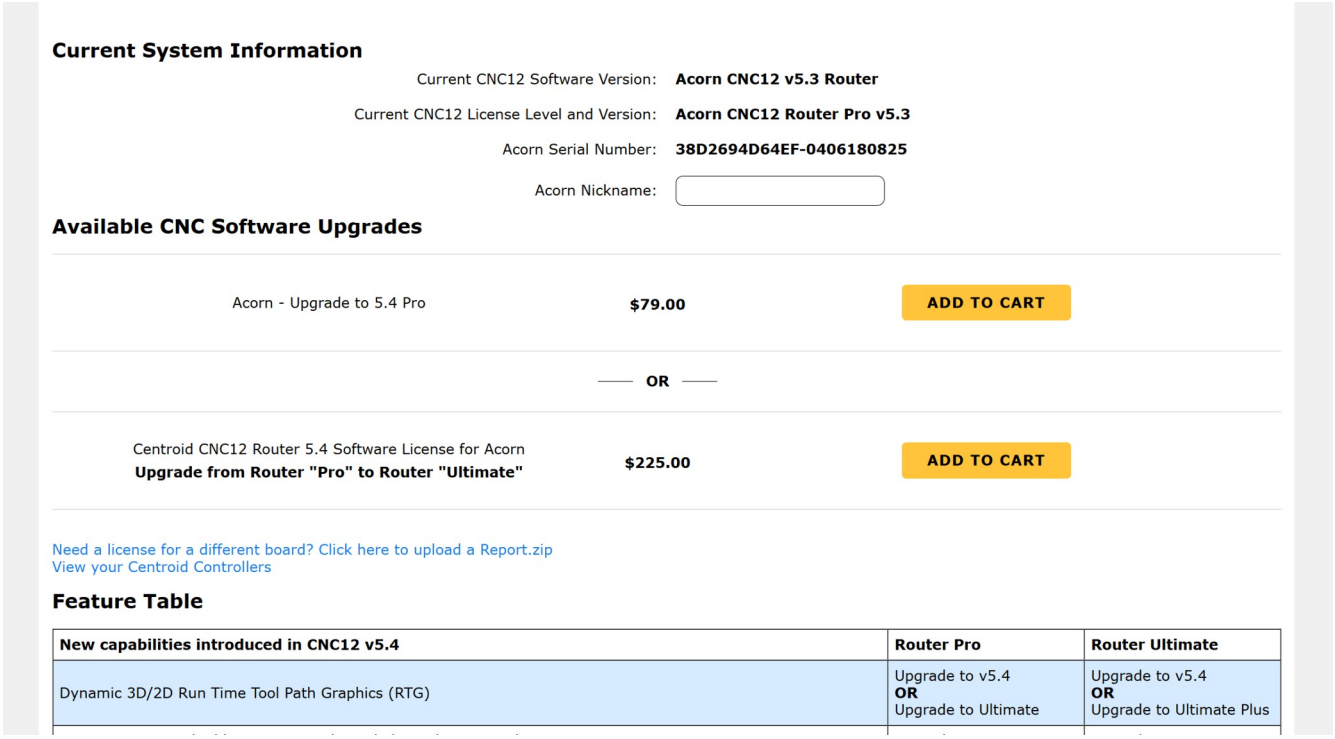
   https://www.centroidcnc.com/centroid_diy/centroid_cnc_software_licensing.html

   CNC12 Options and Licensing Menu. F7 Utility → F8 CNC12 License



If the CNC control has an internet connection you can click on any link to purchase a version upgrade or a new License level, Pro/Ultimate/Ultimate + . , if no internet connection at the CNCPC then use your phone to scan the QR code.

The Centroid License web page purchase portal will present you with all available options for your system. Both the link and the QR code contain the version and current license information of the system and the new web page will automatically recognize it and present you with the system and license information and all available upgrade options.



Current System Information

Current CNC12 Software Version: **Acorn CNC12 v5.3 Router**

Current CNC12 License Level and Version: **Acorn CNC12 Router Pro v5.3**

Acorn Serial Number: **38D2694D64EF-0406180825**

Acorn Nickname:

Available CNC Software Upgrades

Acorn - Upgrade to 5.4 Pro **$79.00** ADD TO CART

OR

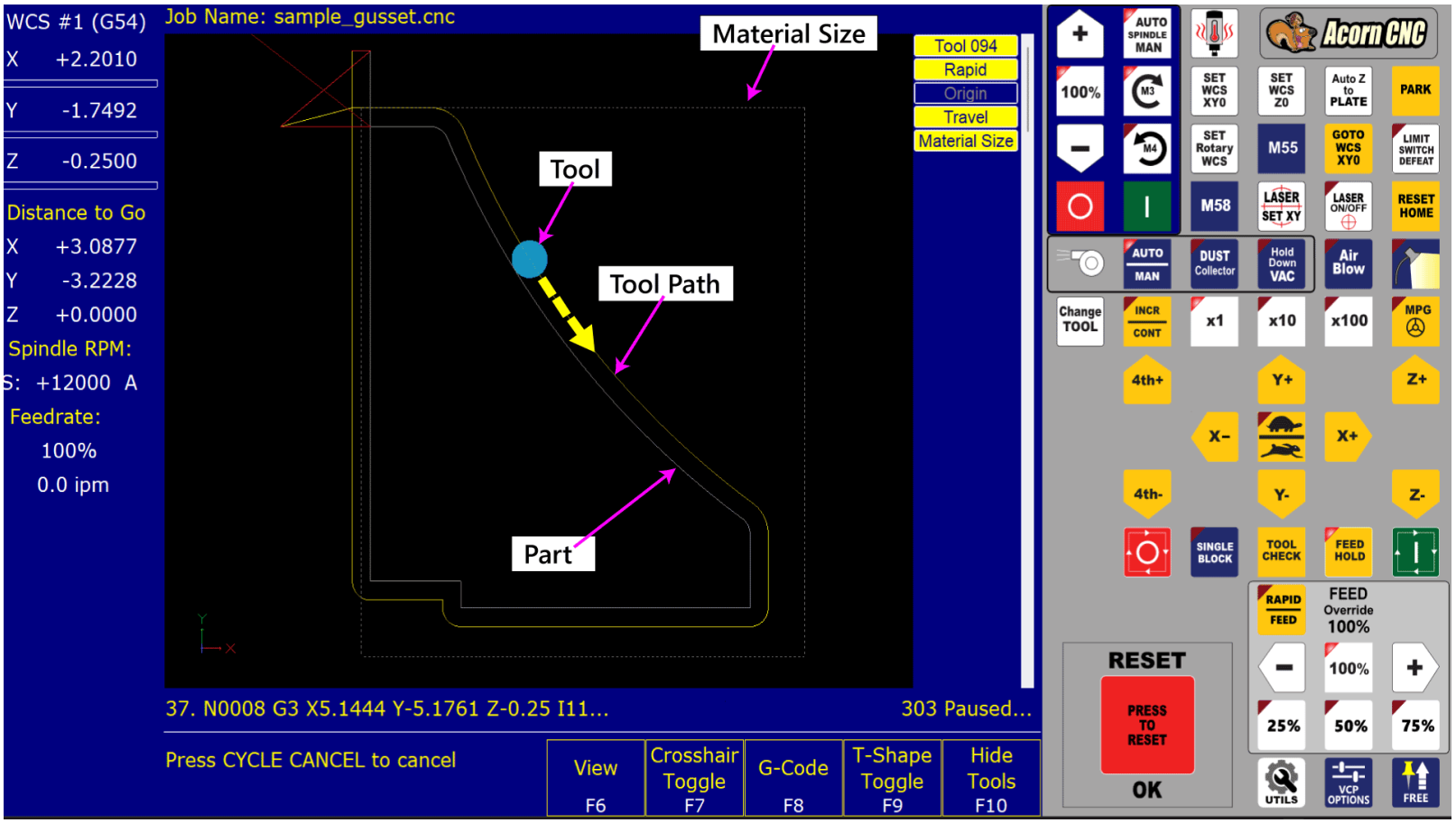
Centroid CNC12 Router 5.4 Software License for Acorn
Upgrade from Router "Pro" to Router "Ultimate" **$225.00** ADD TO CART

Need a license for a different board? Click here to upload a Report.zip
View your Centroid Controllers

Feature Table

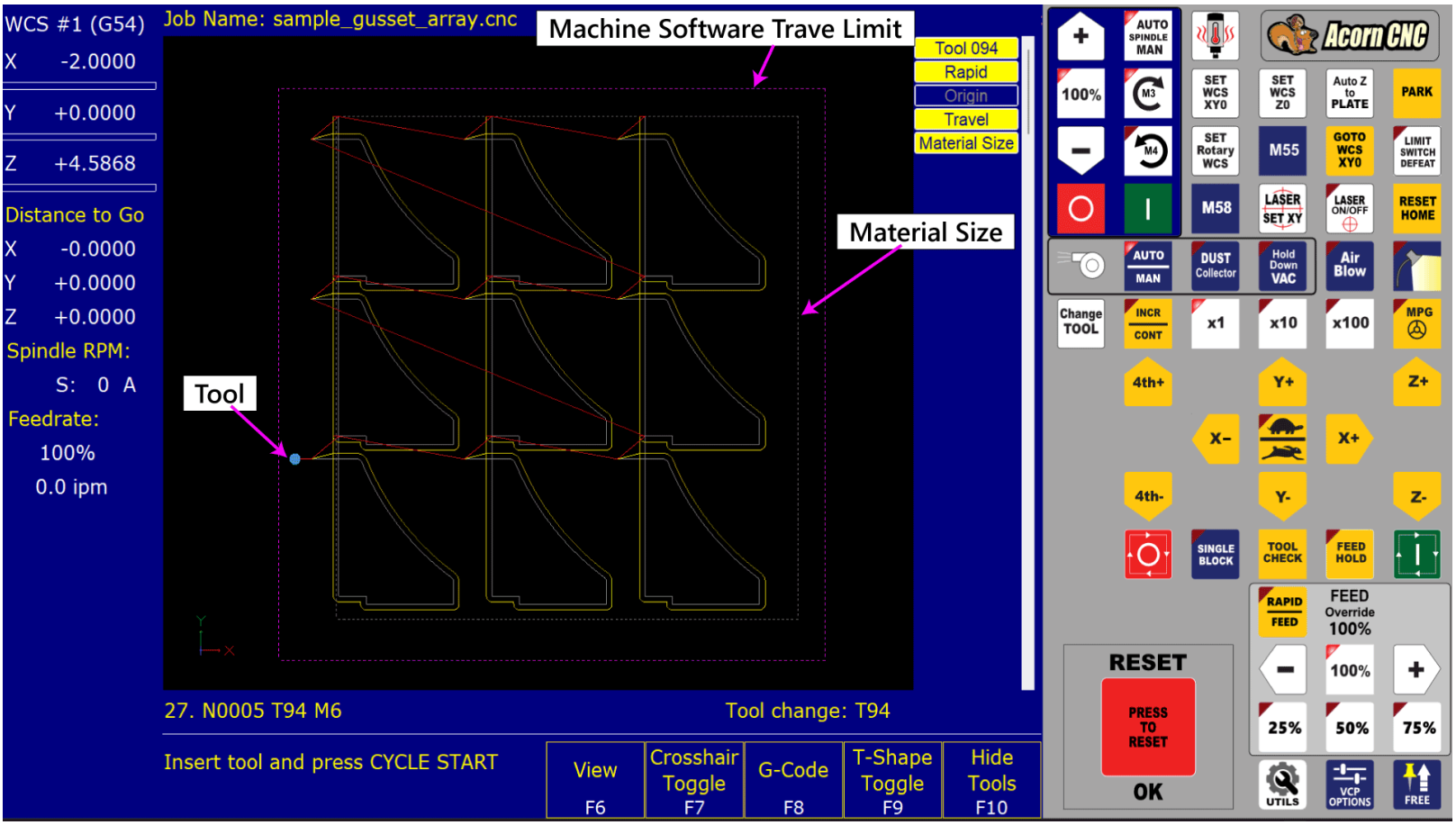
New capabilities introduced in CNC12 v5.4	Router Pro	Router Ultimate
Dynamic 3D/2D Run Time Tool Path Graphics (RTG)	Upgrade to v5.4 **OR** Upgrade to Ultimate	Upgrade to v5.4 **OR** Upgrade to Ultimate Plus

2. Mill, Lathe and Router. New real time Tool Path Backplotting Graphics while running a job. (“RTG” Run Time Graphics = Live Tool Position with G code program backplot. Dynamic 3D rotation/views (mill and router) with user control over display of additional information such as WCS position, Current tool position, Work envelope, Software Travel limits, Material Size, Tool indicator actual diameter while running a part program live, a solid plane with opacity option for the work envelope, toggled through the show tools menu. A v5.4 Pro or Ultimate License is required for this feature.

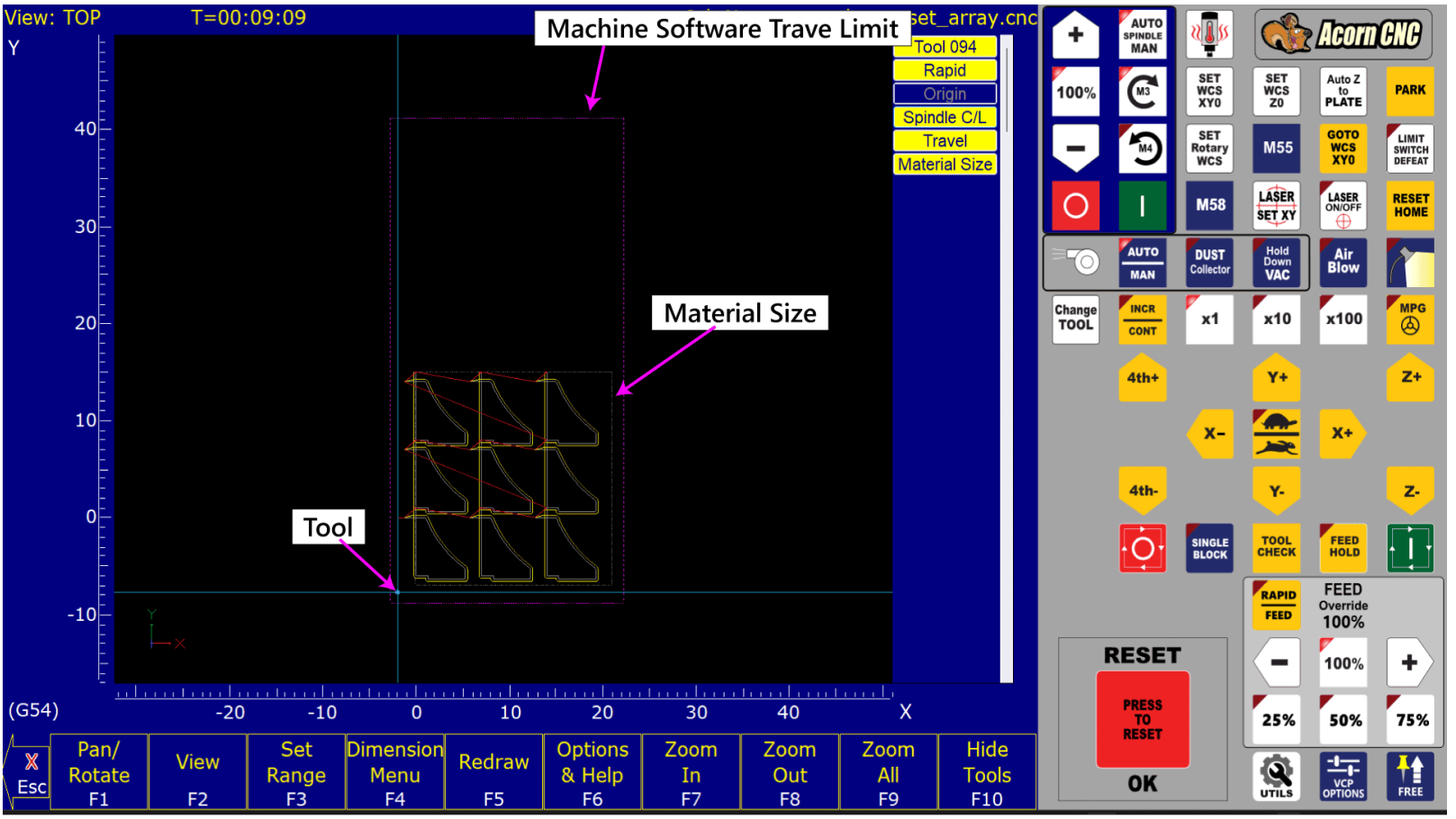


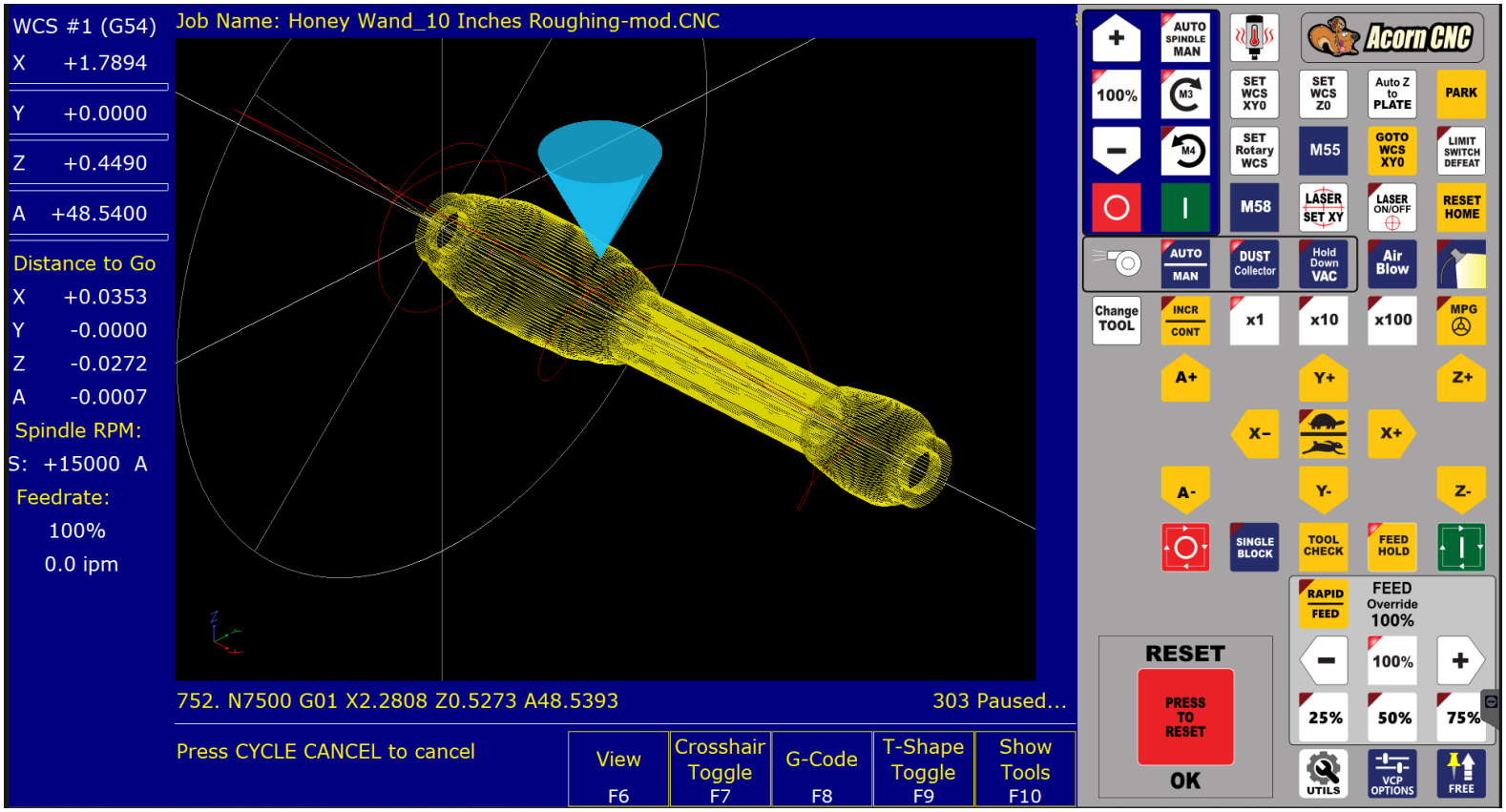
See Uwe Mattern's Video for live demonstration of the new run time graphics.

https://youtu.be/LPvx8mNNLC4?si=TLs2vAeQ_J8yehgj



25” x 50” Machine Total Travel example.




WCS #1 (G54)
X +1.7894
Y +0.0000
Z +0.4490
A +48.5400
Distance to Go
X +0.0353
Y -0.0000
Z -0.0272
A -0.0007
Spindle RPM:
S: +15000 A
Feedrate:
100%
0.0 ipm
Job Name: Honey Wand_10 Inches Roughing-mod.CNC
752. N7500 G01 X2.2808 Z0.5273 A48.5393
303 Paused...
Press CYCLE CANCEL to cancel
View F6
Crosshair Toggle F7
G-Code F8
T-Shape Toggle F9
Show Tools F10
AUTO SPINDLE MAN
100%
M3
M4
SET WCS XY0
SET WCS Z0
Auto Z to PLATE
PARK
SET Rotary WCS
M55
GOTO WCS XY0
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
A+
Y+
Z+
X-
X+
A-
Y-
Z-
SINGLE BLOCK
TOOL CHECK
FEED HOLD
RAPID FEED
FEED Override 100%
100%
25%
50%
75%
RESET
PRESS TO RESET
OK
UTILS
VCP OPTIONS
FREE

Material Size: **Light Grey Dashed line. ----------**

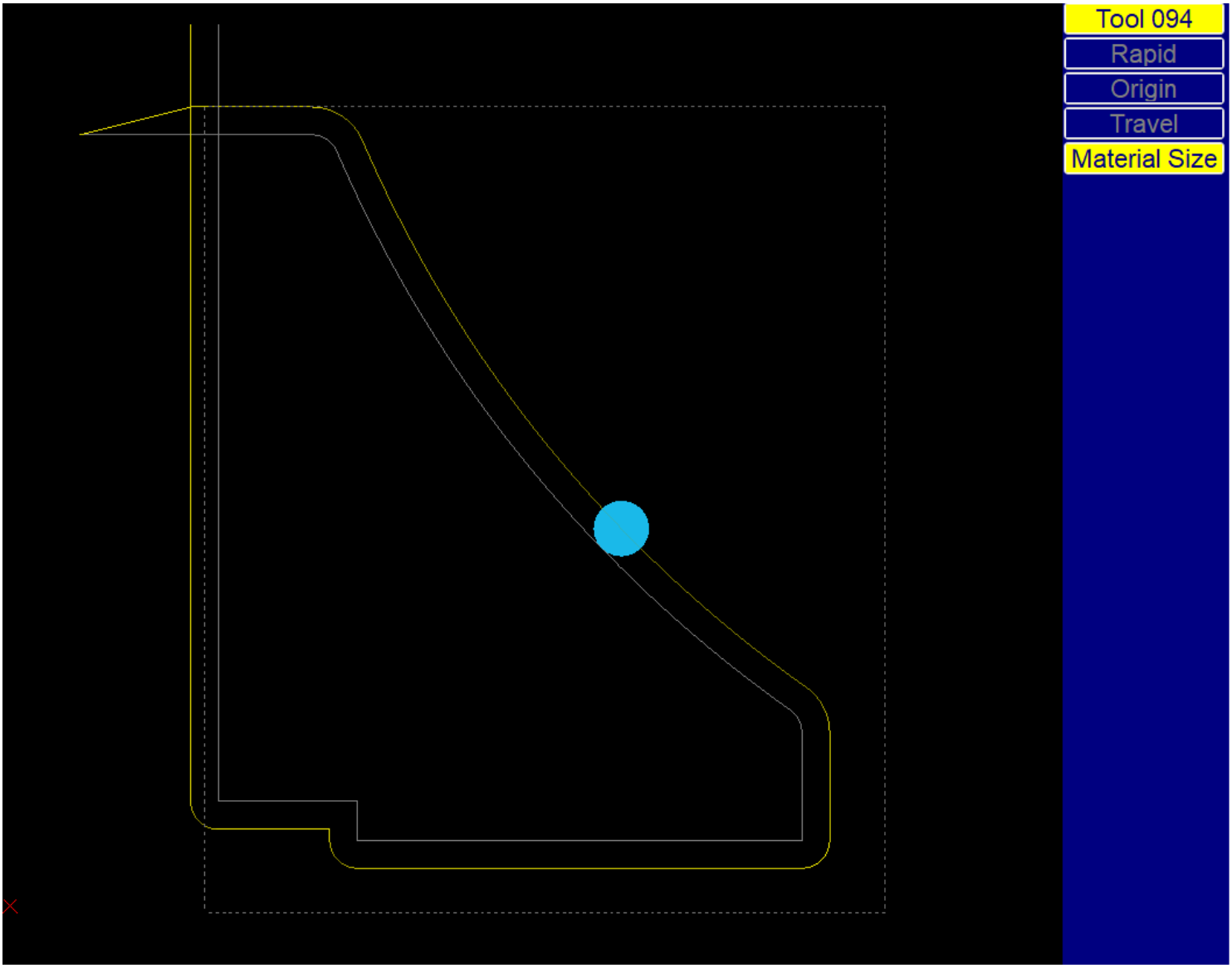
Can now be displayed via G code! Add the desired material size to your g code program here is the format.

:::: X- = -.125, X+ = 6

:::: Y- = -7, Y+ = .25

:::: Z- = -.75, Z+ = 0.0

Example graphics using the Material Size data above.



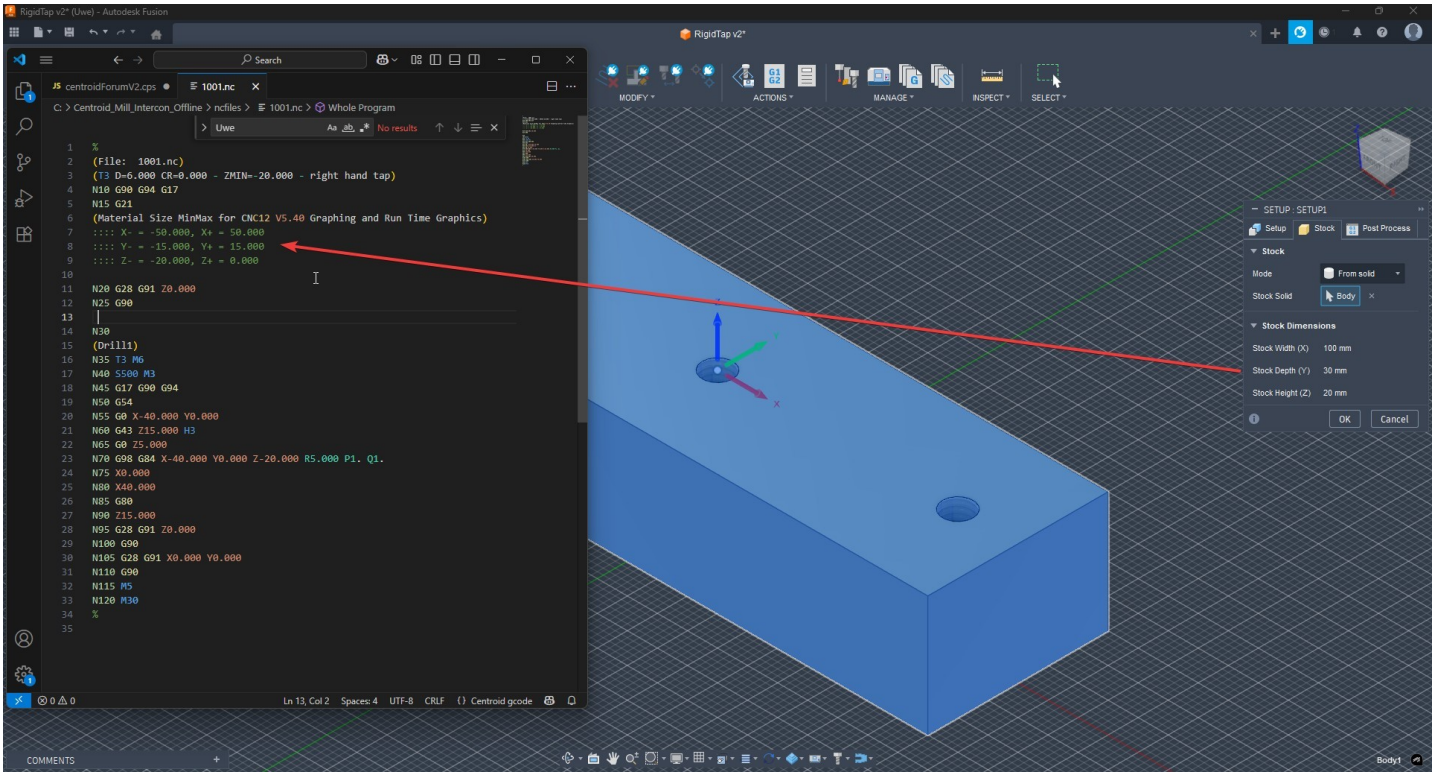
For Vectric Users Scott shows us how to setup the Vectric post processor.

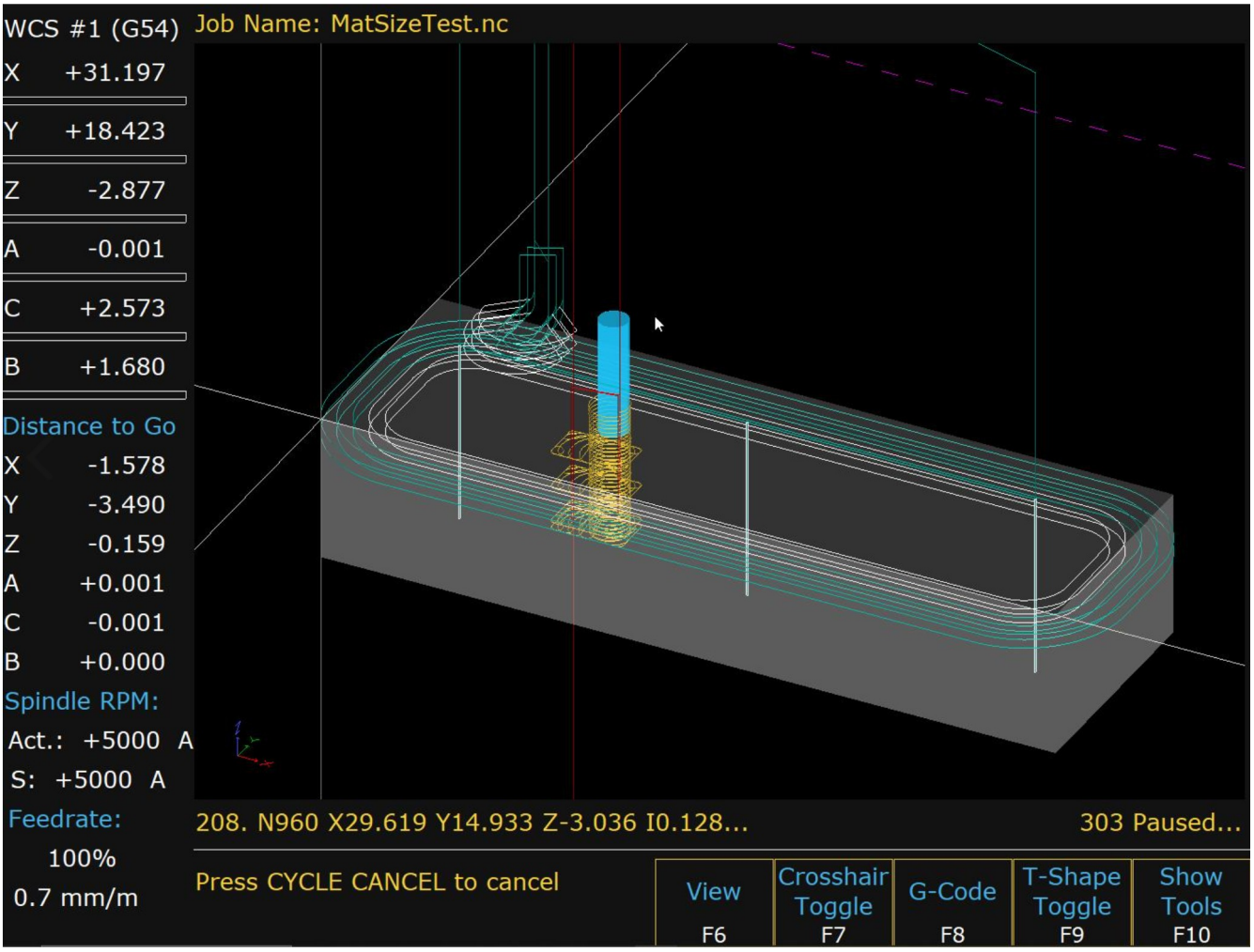
```
" ; Material Size: X [XLENGTH][34]  Y [YLENGTH][34]  Z [ZLENGTH][34]"
" ; X_Y origin: [XY_ORIGIN]"
" ; Origin offset: [X_ORIGIN_POS] [Y_ORIGIN_POS]"
" ; Z zero location: [Z_ORIGIN]"
""
" ; Material Size Min/Max for CNC12 V5.40+ Graphing and Run Time
Graphics"
""
"    :::: X- = [XMIN], X+ = [XMAX]"
"    :::: Y- = [YMIN], Y+ = [YMAX]"
"    :::: Z- = [ZMIN], Z+ = [ZMAX]"
```

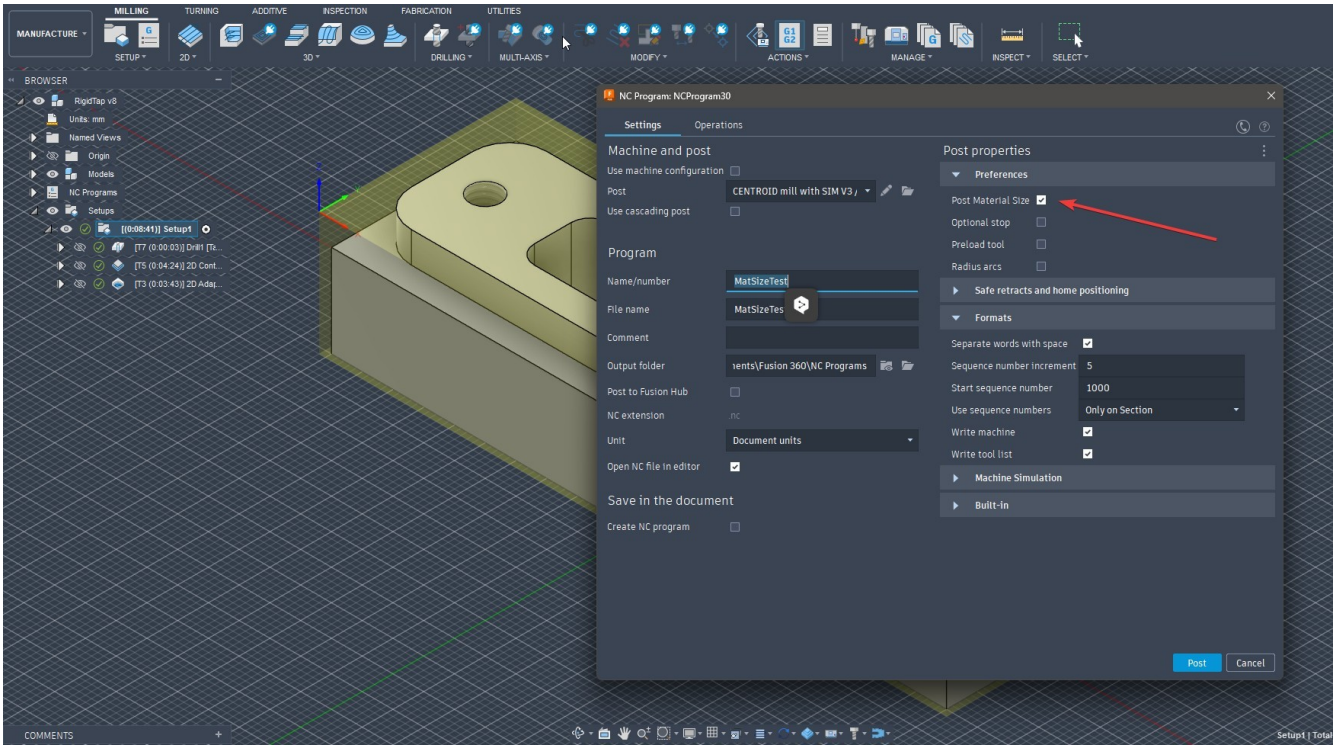
and below is an example of the Vectric generated G code using the material size post additon above.

```
; Material Size: X 12.000"  Y 12.000"  Z 0.750"
 ; X_Y origin: Bottom Left Corner
 ; Origin offset: 0.000 0.000
 ; Z zero location: Material Surface

 ; Material Size Min/Max for CNC12 V5.40+ Graphing and Run Time
Graphics

   :::: X- = 0.000, X+ = 12.000
   :::: Y- = 0.000, Y+ = 12.000
   :::: Z- = -0.750, Z+ = 0.000
```

For Fusion users Uwe has already updated the Fusion 360 post to include this new feature so G code program generated by Fusion will automatically carry this data along. See this post for more information. https://centroidcncforum.com/viewtopic.php?p=103245#p103245




WCS #1 (G54)
Job Name: MatSizeTest.nc
X +31.197
Y +18.423
Z -2.877
A -0.001
C +2.573
B +1.680
Distance to Go
X -1.578
Y -3.490
Z -0.159
A +0.001
C -0.001
B +0.000
Spindle RPM:
Act.: +5000 A
S: +5000 A
Feedrate:
100%
0.7 mm/m
208. N960 X29.619 Y14.933 Z-3.036 I0.128...
303 Paused...
Press CYCLE CANCEL to cancel
View F6
Crosshair Toggle F7
G-Code F8
T-Shape Toggle F9
Show Tools F10

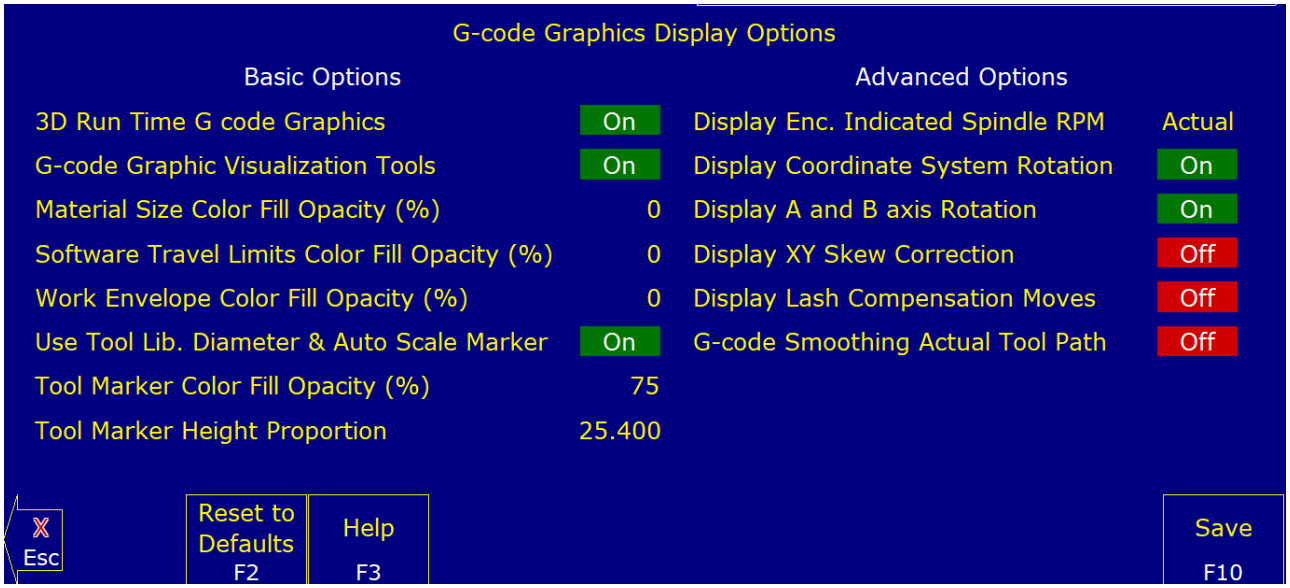

MANUFACTURE
MILLING
TURNING
ADDITIVE
INSPECTION
FABRICATION
UTILITIES
NC Program: NCProgram30
Settings
Operations
Machine and post
Use machine configuration
Post
CENTROID mill with SIM V3
Use cascading post
Program
Name/number
MatSizeTest
File name
Comment
Output folder
Post to Fusion Hub
NC extension
Unit
Document units
Open NC file in editor
Save in the document
Create NC program
Post properties
Preferences
Post Material Size
Optional stop
Preload tool
Radius arcs
Safe retracts and home positioning
Formats
Separate words with space
Sequence number increment
Start sequence number
1000
Use sequence numbers
Only on Section
Write machine
Write tool list
Machine Simulation
Built-in
Post
Cancel

Machine Software Travel Limits. Fushsia dashed line

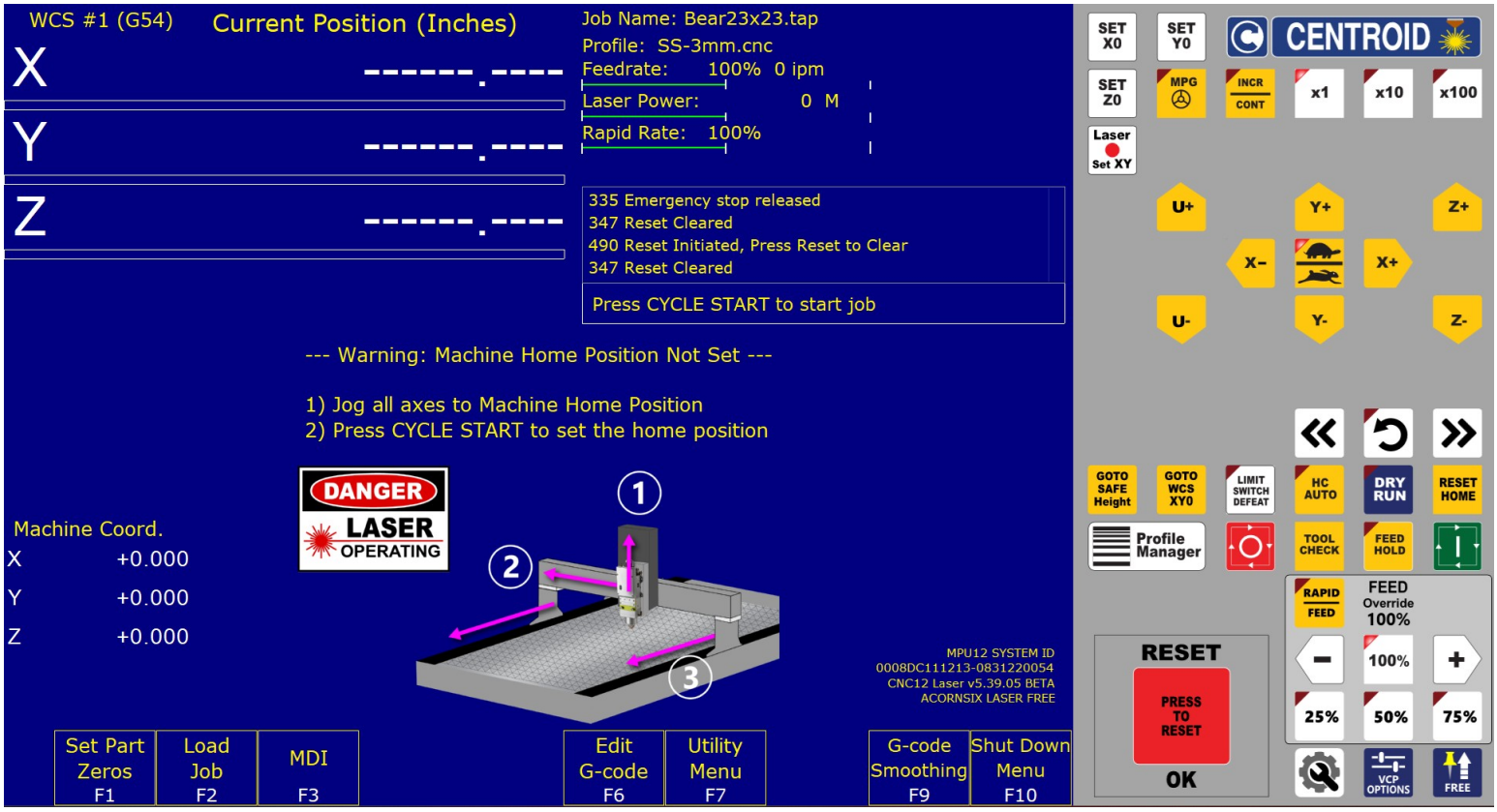
Software Travel Limits in the Wizard Axis configuration Menu.

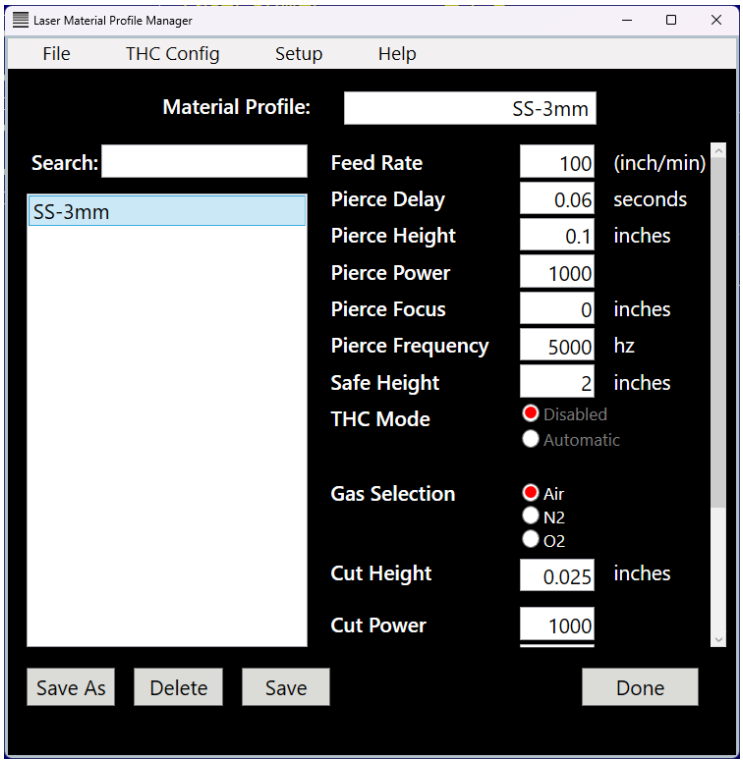
milliseconds	250	250	250
Travel Limit (+)	24	24	0
Travel Limit (-)	0	0	-6

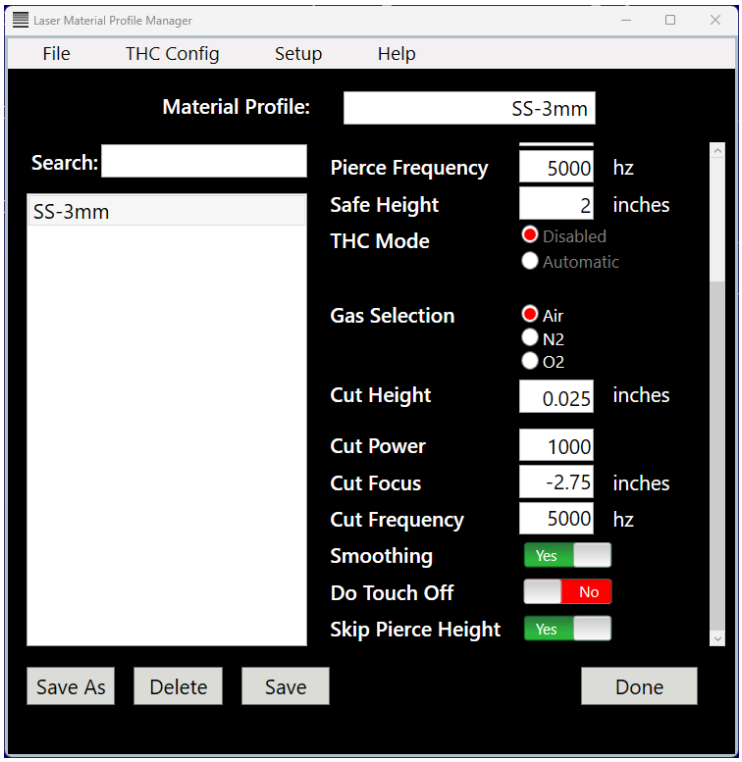
3. Added a CNC12 menu for g code graphing options. Requires a v5.4 Pro or Ultimate license.



4. **Available as a BETA within v5.4**. Contact joey@centroidcnc.com 570-927-2516 for more information about Centroid's Fiber Laser CNC controller. CNC12 variant for Fiber Laser CNC Controller. (AcornSix hardware only for this release!). Fiber Laser feature set: Auto Focusing, Non contact Z position touch off, High speed closed loop capacitive Laser Head Z Height control, Restart Mode, start anywhere, restart anywhere, auto lead in creation ,backup along cut path, jog forward or backward along cut path, jog off cut path and start with auto lead in, Run with either the built in Material Profile manager or G-code material parameter methods both are supported, Programmable Laser Power and Frequency Level control, Gas control, note: manual focus head and contact touch off also supported. Note: Fiber Laser CNC12 be used with or without the Centroid THC.







5. The CNC12 software version number now is in line with industry standards and has additional decimal place. v5.40.00

   v 5. 40. 00 = Major. Minor. Patch

   so in the future you will start to see cnc12 versions like v5.40.01 or v5.40.03 both of these would be a revision patches of 5.40 typically would be just bug fixes.
   All future version contain all features and fixes of past versions.

   After v5.40.00 will be v5.50.00 and then v5.60.00. While these are considered a 'minor' new version within the major v5. They will have significant features added to them.

   Note: if you see an odd number that indicates BETA software. For instance, the public beta of (yet to be released v5.40.00) for Beta testing version numbers would look like this v5.39.02 or v5.39.04 etc.

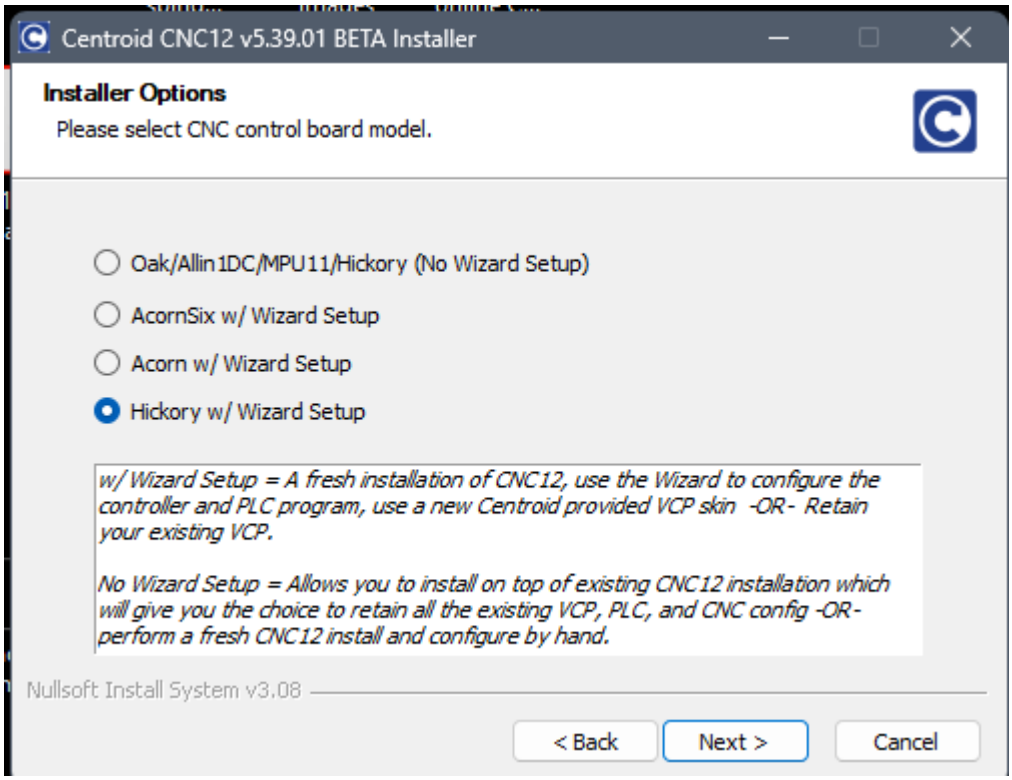
6. Each CNC12 variant now installs in it own dedicated directory. Mill =c:\cncm, Lathe = c:\cnct, Router = c:\cncr, Plasma = c:\cncp, Fiber Laser = c:\cncl

7. The CNC12 Installer now gives Hickory users two choices on how to install and configure a Hickory based CNC control system.

   a.) Hickory not using the Wizard (only recommended for OEM/Advanced retrofitters)

   -or-

   b.) Hickory using the Centroid setup Wizard. Recommended Method (even if you are going to customize)



Choose Hickory w/Wizard Setup if this is your first time installing CNC12 on this CNC PC.
Even if you no you are going to customize the CNC12 installation, we recommend initial setup using the Wizard as it sets a wide variety of parameters and files based on your selections and input in the Wizard for proper machine function. You can then always go modify any parameters, macros, plc programs if needed to customize the installation.

Choose Hickory (No Wizard Setup) if you want to manually setup the CNC control configuration, use a stock Centroid provided base PLC program -or- use you own customized PLC program. This selection can be used for both a fresh new installation -OR- most commonly used to “install on top” of an existing older CNC12 version (do this if you have a configured running Hickory based machine and you just want to update CNC12 and leave everything else alone).

8. Support for LS Electric Ethercat Servo drives (sold by Automationdirect.com) added for use with Hickory.

   https://www.automationdirect.com/selectors/ls-servo?_gl=1*1i1q5yh*_up*MQ..*_gs*MQ..&gclid=CjwKCAjw_fnFBhB0EiwAH_MfZn2bYJnX6YW80nwB6tu2ckjkZ9yZj9WDM5ozKDp_nbJ-Ya1Db4ubOxoC1qsQAvD_BwE&gbraid=0AAAAAD_dnO1admGr6xSkYwMpMq3O5lT7t

9. Improved Software Axis Pairing. New software axis pairing. M294 and M295. Requires Pro or Ultimate License, while not necessary we strongly suggest purchases the v5.4 upgrade when using software axis pairing on a machine tool.

   Axis Pairing Parameters:

   - Slave Axis definition parameters are: P551-558 (replacing the old P64 method, P64 is no longer used)

   551 = axis 1, 552 = axis 2, 553 = axis 3, etc.

   Assign a slave axis to a master

   Examples:
   Setting P554 = 2 will slave the 4th Axis to the 2nd Axis.
   Setting P551 = 2 will slave the 1st axis to the 2nd axis
   Setting P552 = 4 will slave the 2nd axis to the 4th axis

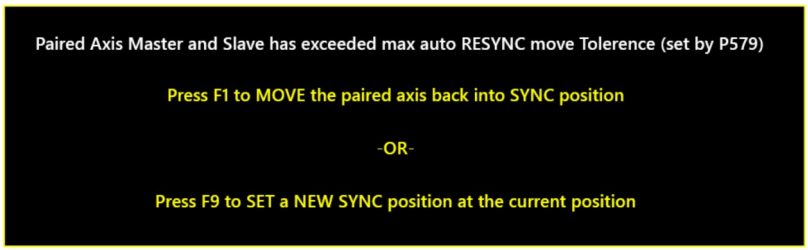
   - P559 controls whether the slaved axis is position is visible on the DRO while paired, useful for troubleshooting. This will display the slave axis label as “N” on the DRO.

   Example for Acorn users set P559 = 8 to display the 4th axis which is used as a software slave.

   Display DRO for slave axis 1 Add 1
   Display DRO for axis 2 Add 2
   Display axis 3 Add 4
   Display axis 4 Add 8
   Display axis 5 Add 16
   Display axis 6 Add 32
   Display axis 7 Add 64
   Display axis 8 8Add 128

   If you have more than one sets of paired axes you can display all the slave axes by adding up the numbers. For example

   - P579 is the minimum AUTO paired axis resync tolerance, this is the max deviation the axis can automatically re-square the axis after an Estop release without a pop-up message to the operator asking to press cycle start to resync. Typically this number is adjusted for the type of machine, machines with lots of spring in the paired axis can use a larger number so the controller just automatically resyncs the paired axes after an estop event and is ready to run. Some other types of machine may want to use a small number so any unexpected (by the operator) motion to resync is minimized. Operator is presented with a message to press cycle start to move to resync when the two motors are out of position by more than the tolerance specified. For Example on a typical CNC Router Gantry. When P579 = .050”, after an Estop release CNC12 looks at the two encoders on the master and slave motors and determines if the two sides of the gantry are within .050” if yes then the machine will automatically move the master and slave back to the SYNC position at the slow jog rate upon an Estop release.



   If the controller determines the gantry axis is out of sync by more than .050” the operator is prompted with a message letting him know that the gantry is out of square by more than the P579 value and to press cycle start

- M-Codes: Two new M-Codes are introduced to eliminate the need to modify the parameters directly to pair and unpair servo motor drives.

M294 Unpair and **M295** Re-Pair and Sync, and **M295 with P1** Move to Sync position and Re-Pair.

Examples:

M294 / * (Where * is the master axis Label) will un-pair the paired axes.

This will allow the two axis motors to be commanded separately if desired. Useful in auto squaring macros.

Y axis Examples:

M294 /Y will UnPair what ever axis has been paired with the Y axis.

M295 / Y will Re-Pair and **RESETS** the SYNC position at the current location between the two axes.

M295 / Y P1 will **MOVE** the Slave motor back independently to the last known SYNC position previously set between the two axes and then Re-Pair the two motors.

M295 /* P2 - Tells CNC12 to ignore re-sync, and does not update the sync position. This is for cases where the user only wants to re-pair the axes in their current location without re-syncing or setting a new sync position.

New M26 and M26/Q1 functionality:

M26 /* - Homes the master axis and any axes slaved to that master, but does not pair the axes. This will also set the sync position between all homed pairs.

M26 /* Q1 - Re-syncs the slaved axes, homes the master axis and any slaved axes, and sets the sync position between all homed pairs. This does not pair the axes.

Other notable action when compared to previous versions. With Hickory Ethercat controller an E-Stop and a Powercycle is no longer required to set the Square and Paired Axis and Home Positions.

10. Up to 8 axes can now be paired together using new parameters P551-558. Requires a v5.4 Ultimate License.

11. Multiple axes can now also be slaved to a single master and master-slave axis pairings and can now be chained together. Requires a v5.4 Ultimate License.
12. Pairing axes 6,7,8 to any other axes in any combination. Requires a v5.4 Ultimate License.

    Negative travel limits with screw compensation are now allowed when using ballscrew pitch compensation (requires a v5.4 Pro or Ultimate License)

13. Planar Compensation. (formerly known as 2.5D VC) This is Machine Position correction compensation tables that use information from 2 axes. With this feature you can take a bow, bump, angle or skew out of an a set of axes. Very useful for CNC router XY and Bed Mill XZ or YZ or Lathe XZ machine position correction, this feature allows you to correct for mechanical position issues with the machine tool. Requires a v5.4 Ultimate License. See this post for instructions with video demonstration. https://centroidcncforum.com/viewtopic.php?t=9266

14. Unique Planar Compensation correction tables for Master and Slave paired axes. Requires a v5.4 Ultimate License. CNC12 can now use two different position correction data tables, one for each of the paired axis motors. For example, A two axis motors paired on a moving gantry: Y1 and Y2, previously used the Y1 correction table for both Y1 and Y2 movements, Now Y1 and Y2 can have their own

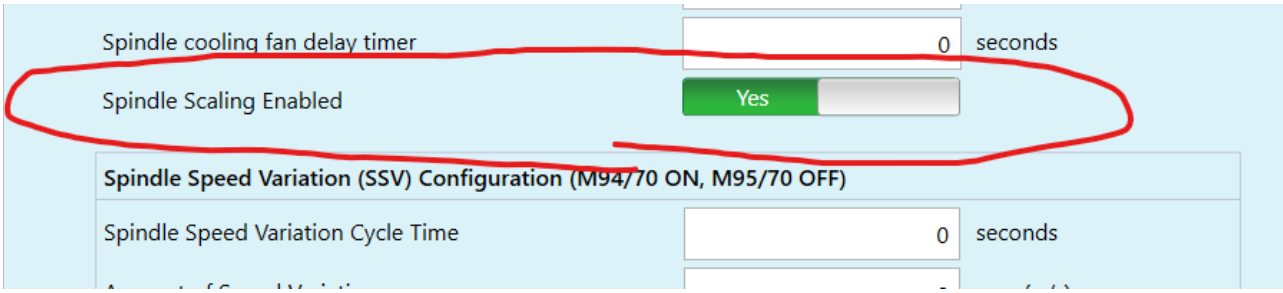
dedicated correction lookup tables to correct for pitch errors (screw compensation) between the master and slave while also correcting for bows/bumps/skew (planar compensation) in the machine tool.

15. Spindle RPM Scaling. Useful for hi RPM spindle machines like CNC Routers that don't operate below a significant RPM. Available across all platform and included in the Hickory, Acorn, AcornSix Wizard and stock PLC programs. Feature requires a v5.4x.x Pro or Ultimate License .

    A new slider in the Wizard spindle menu enables the feature.

    When slider is set to No, the 0-10V output voltage is scaled normally from 0 to max spindle speed.

    When slider is Yes, the 0-10V output voltage is scaled from minimum to max spindle speed.



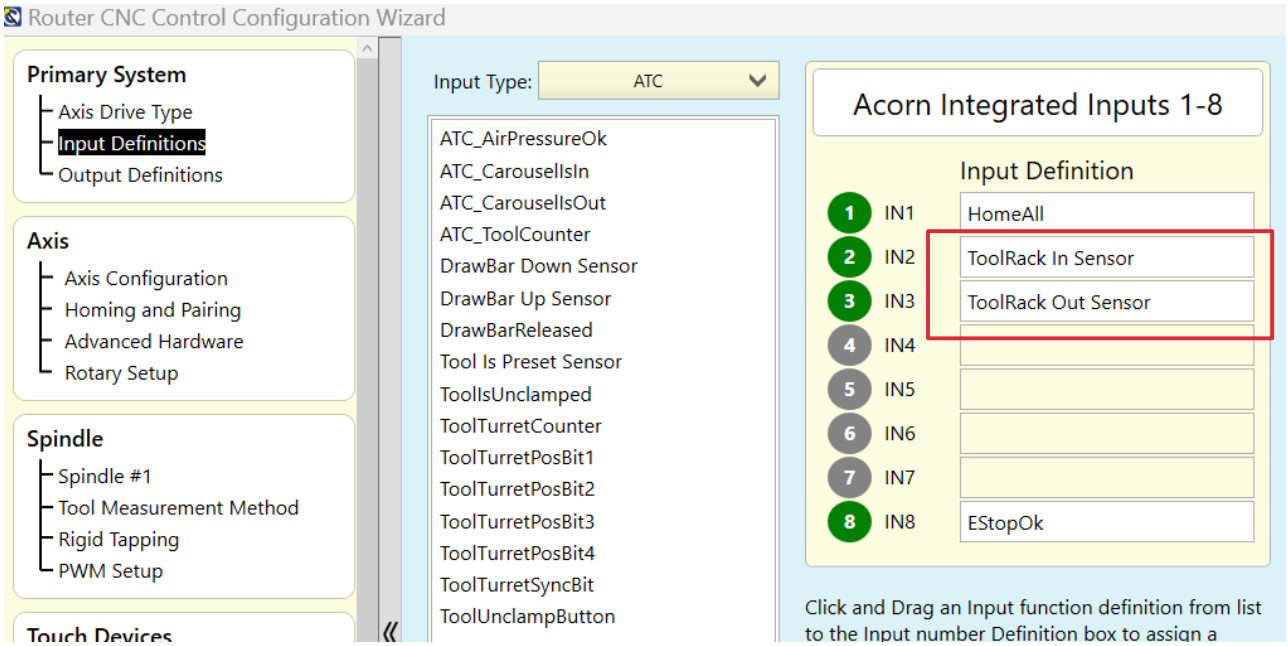
So for Example,
when Max Spindle Speed = 24000
and Min Spindle Speed = 6000

Without Scaling,
S0 = 0V
S6000 = 2.5V
S12000 = 5V
S18000 = 7.5V
S24000 = 10V

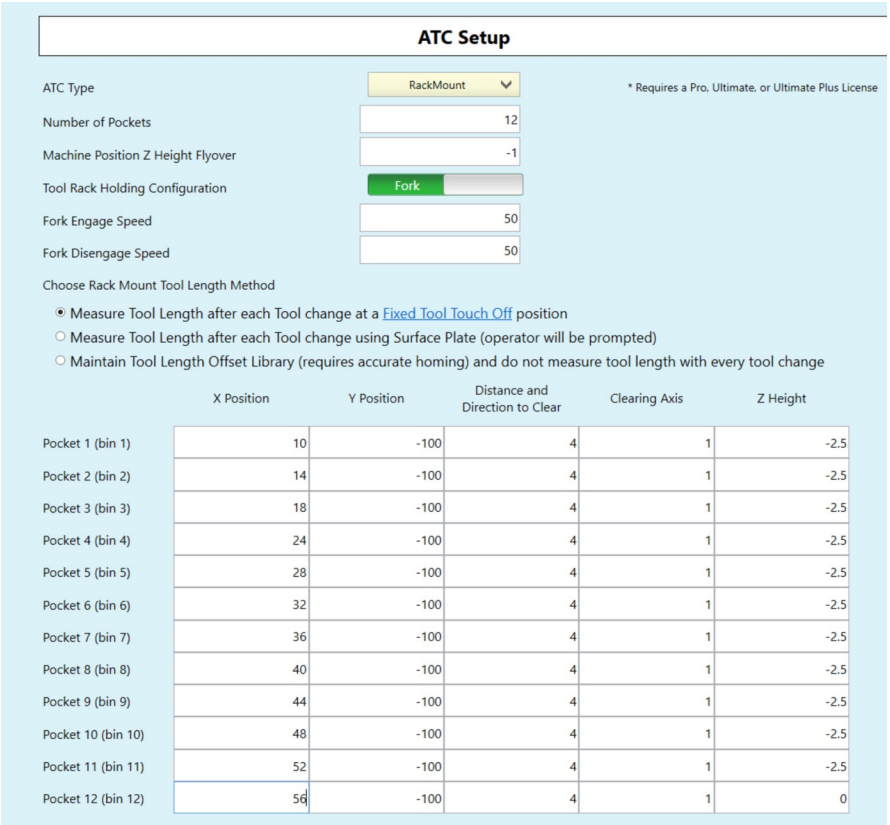
With Scaling
S0 = 0V
S6000 = 0V
S10500 = 2.5V
S15000 = 5V
S19500 = 7.5V
S24000 = 10V

For non Wizard users this feature is with Parameter 78 Bit 4 (add 16) and the supporting PLC program logic is included with all the Centroid provided programs across all platform (Oak, Allin1DC, Hickory, Acorn, AcornSix)

16. Wizard Drag and Drop Canned Moveable ATC Rack Support. New Inputs added to the ATC section of the Wizard. “ToolRack In Sensor” and “ToolRack Out Sensor” requires a v5.4x.x Pro or Ultimate License



Works automatically with your Rack Mount Setup page.



	X Position	Y Position	Distance and Direction to Clear	Clearing Axis	Z Height
Pocket 1 (bin 1)	10	-100	4	1	-2.5
Pocket 2 (bin 2)	14	-100	4	1	-2.5
Pocket 3 (bin 3)	18	-100	4	1	-2.5
Pocket 4 (bin 4)	24	-100	4	1	-2.5
Pocket 5 (bin 5)	28	-100	4	1	-2.5
Pocket 6 (bin 6)	32	-100	4	1	-2.5
Pocket 7 (bin 7)	36	-100	4	1	-2.5
Pocket 8 (bin 8)	40	-100	4	1	-2.5
Pocket 9 (bin 9)	44	-100	4	1	-2.5
Pocket 10 (bin 10)	48	-100	4	1	-2.5
Pocket 11 (bin 11)	52	-100	4	1	-2.5
Pocket 12 (bin 12)	56	-100	4	1	0

Note: The Wizard automatically programs the PLC program with this logic:
Tool Rack In (Towards Spindle) = SV_M94_M95_80
Tool Rack Out (Away from Spindle) = SV_M94_M95_81
Logic is under the "ToolRackInOutStage" in PLC.
Wizard also sets Parameter 830 to 7. to let the PLC program know that the ATC type is Rack Mount.

17. A new “Don't use Machine Home” selection will remember WCS position after a power cycle/ Remember part zero after power cycle without homing the machine. The Wizard feature: “Don’t use Machine Home” will now remember last known WCS position and reinstate that DRO position after a power cycle. This feature is for Open/Hybrid loop systems that don’t want to use home to switch and want to mimic absolute encoder functionality. While not for everyone, this feature works well for some applications as long as the machine does not move while being powered down. requires a v5.4x.x Pro or Ultimate License

18. Significant Wizard speed increase reduces the time that it takes the Wizard to write CNC12 files and parameters. Its fast.

19. Improved Plasma Backup Torch along path responsiveness, quicker and more natural. Better UX in the popular Plasma restart mode feature.

20. Control the number of Rows and Columns that make up the Virtual Control Panel (VCP)

VCP: Two new tags have been added

<column_count> </column_count>

and

<row_count> </row_count>

These tags are use to define the number of rows and columns that the VCP uses.

Place the tags right under the <vcp_skin> tag.

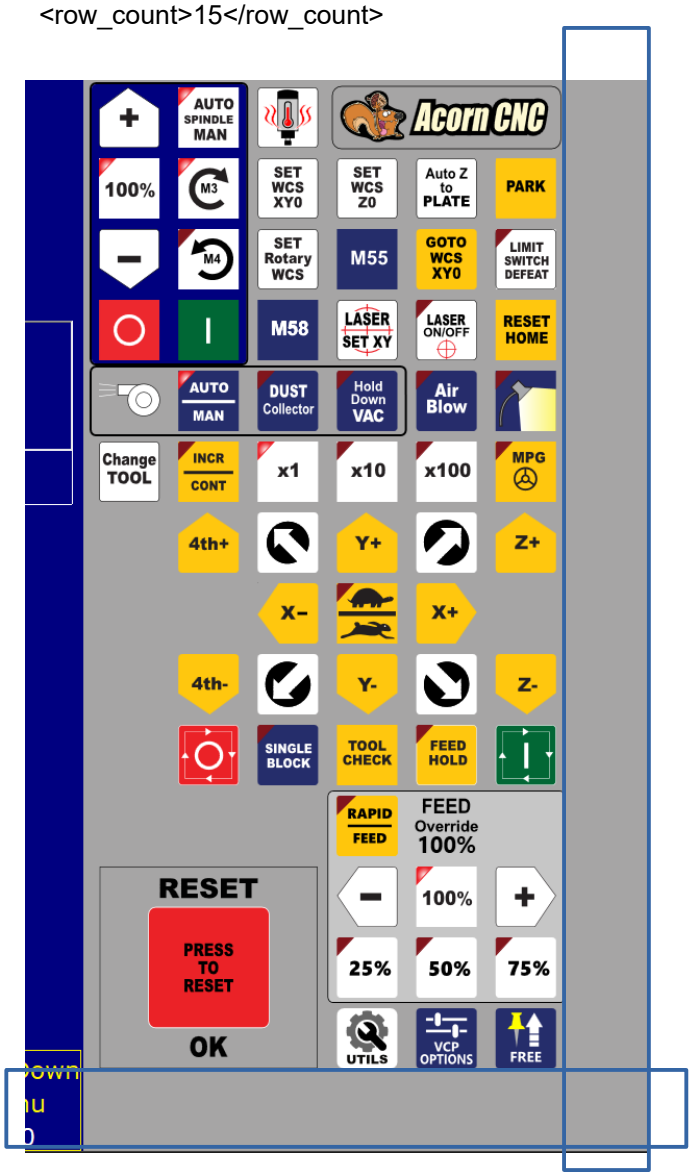
For example lets add a row and a column to the VCP base layout.

To increase the number of columns from 6 to 7 and the number of rows from 14 to 15, add the following lines to the skin xml file. This additional area can be use just like any other row and column on the VCP. More info and other examples can be found in the VCP Users manual.

<vcp_skin>

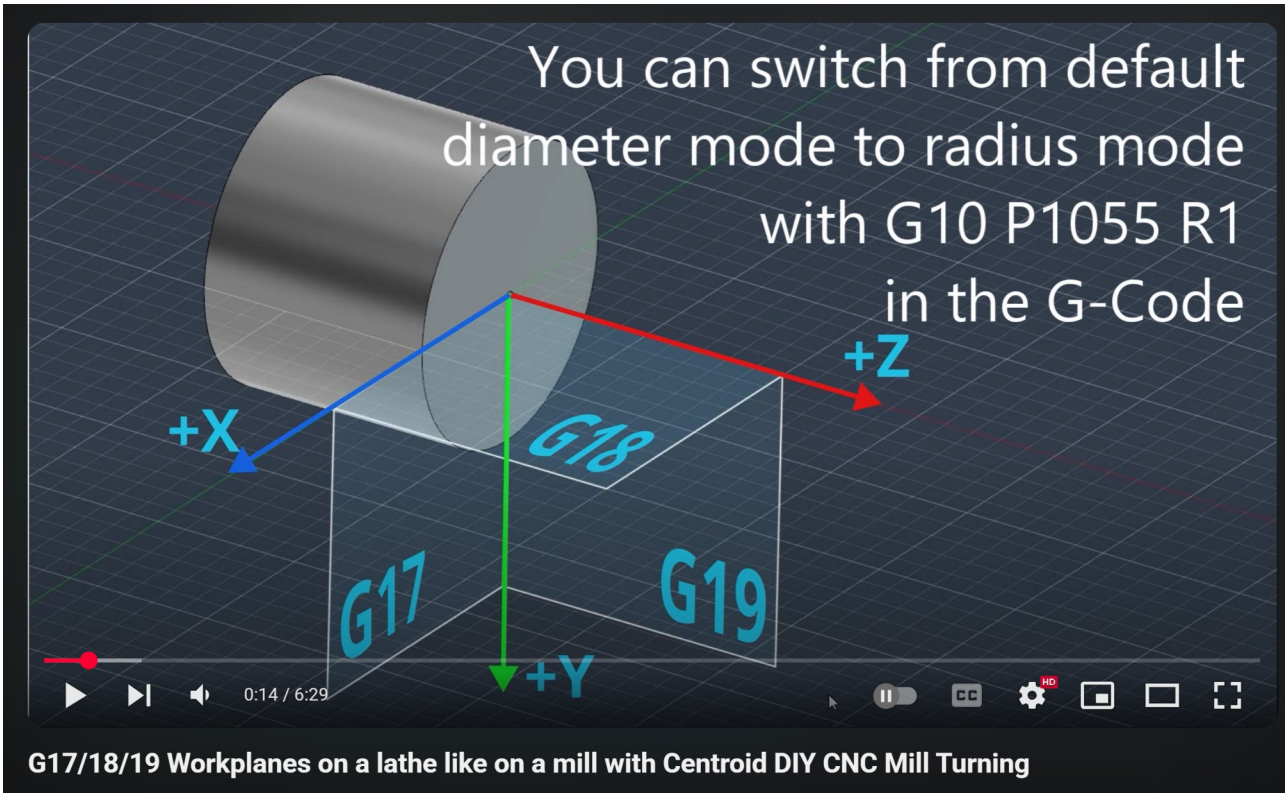
<column_count>7</column_count>

<row_count>15</row_count>

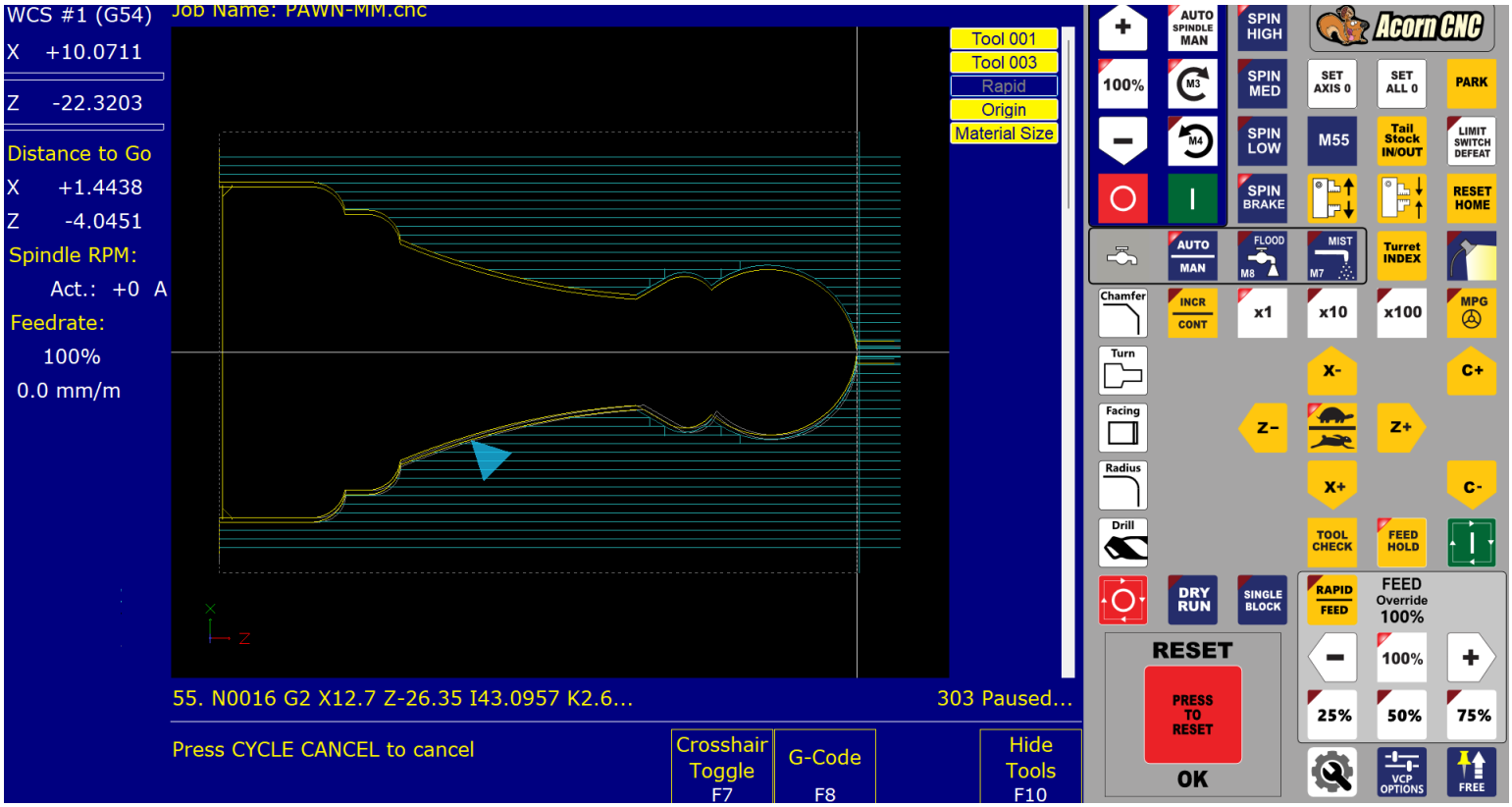


21. Lathe: G codes for XY ,and YZ arcs have been added to CNC12 Lathe, G17 and G19. requires a v5.4 Pro or Ultimate License

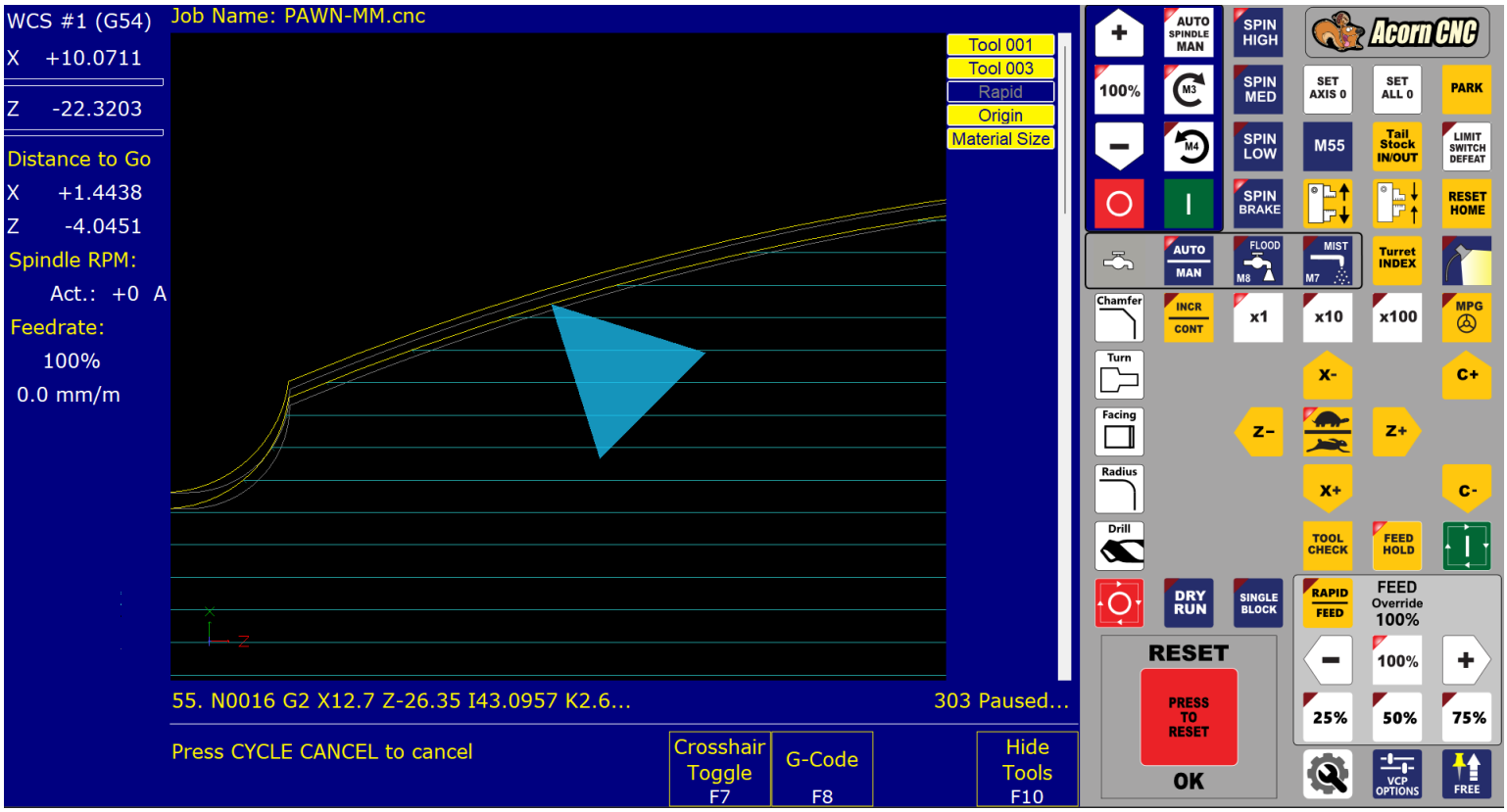
    See Uwe Mattern's Video https://youtu.be/X76VosJb924?si=QaETSZhqcCGB_sqz



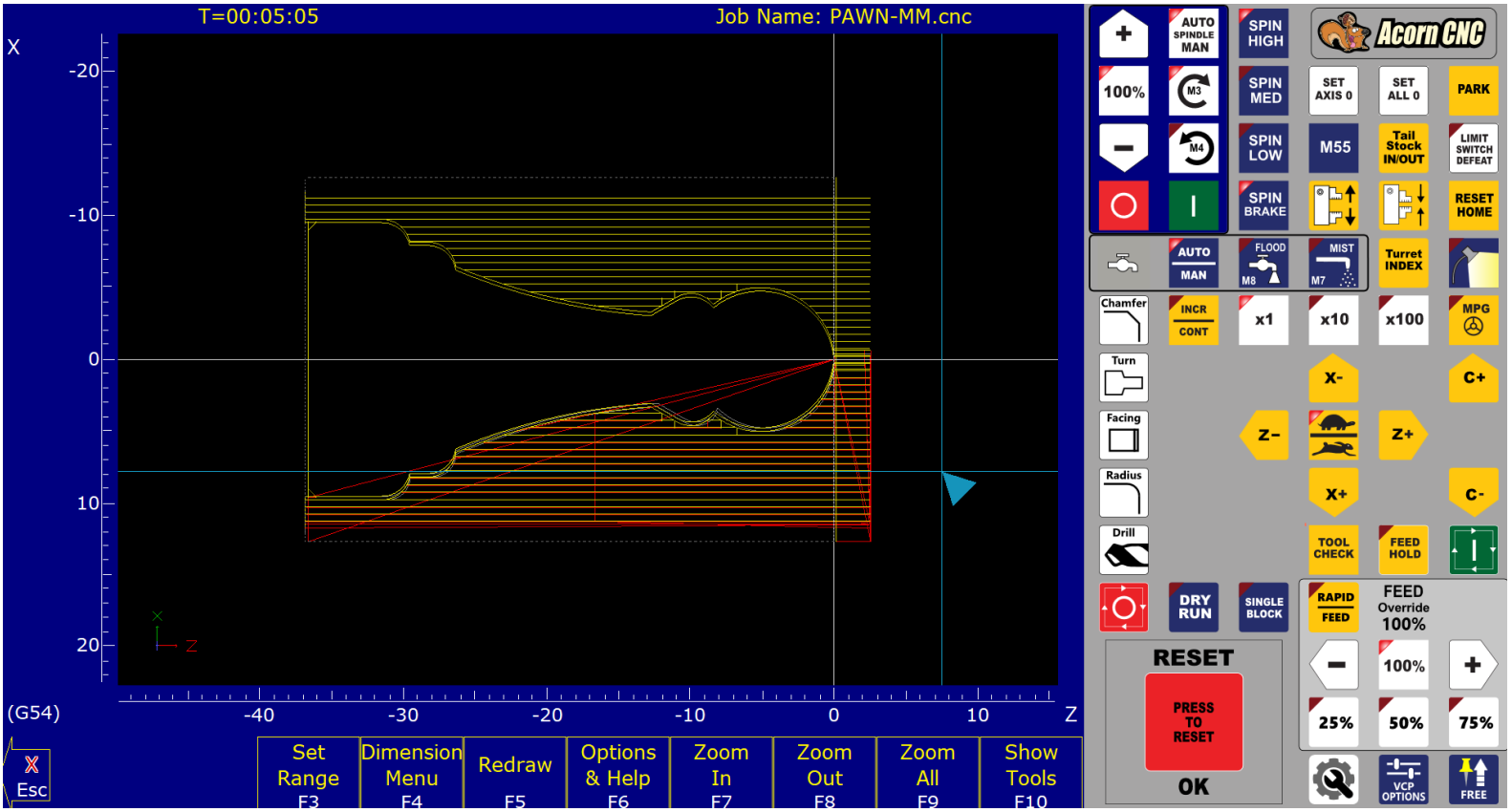
22. CNC12 Lathe Graphics has been upgraded and now use the same v5.4 g code backplot graphics engine as the other v5.4 variants of CNC12, but using the 2D top down XZ view. The new Lathe part graphics now read tool vector orientation from the tool library as well and you can now zoom in and out and pan while the machine is running and use the graphics tools to show/hide material size, travel limits, work envelope, rapids etc. while the job is running. Requires a v5.4 Pro or Ultimate license



F4 Run, F2 Search → line 54, F10 Accept, cycle start. Rapids set to do not display, Origin set to display (white cross hair), Material Size set to display (dashed line).
Zoom in and Pan while running the job.



Tool Position is live in the G code backplot so you can see current tool position relative to part program. Jogging in the graphics menu will reflect the cutter position live.



23. Wireless Touch Probe Support, P11, P18, P44, P257, P540, and P541 now have an output or memory bit value (60000+). requires a v5.4 Pro or Ultimate License. This new wireless touch probe and tool setter support allows the existing CNC12 probe parameters to have an output or memory bit associated with them so that information can be used by the PLC program to do “wireless probe stuff” like figuring out if the wireless Probe and Tool setter is currently awake and ready to go before running a probing cycle.

24. In-process axis position adjustment from external source. Use a MPG/Scale/Encoder for real time position offset adjustment for any axis with M291 offset cancel or keep. This features allows on the fly axis position adjustment/ correction by the operator, typically used for Z height adjustments such as Flame Torch height tweaks while cutting but, also has other applications such as following a pattern or template with the pattern/template data being feed into the Centroid Encoder port via/ encoder or scale. Feature requires a v5.4 Pro or Ultimate License

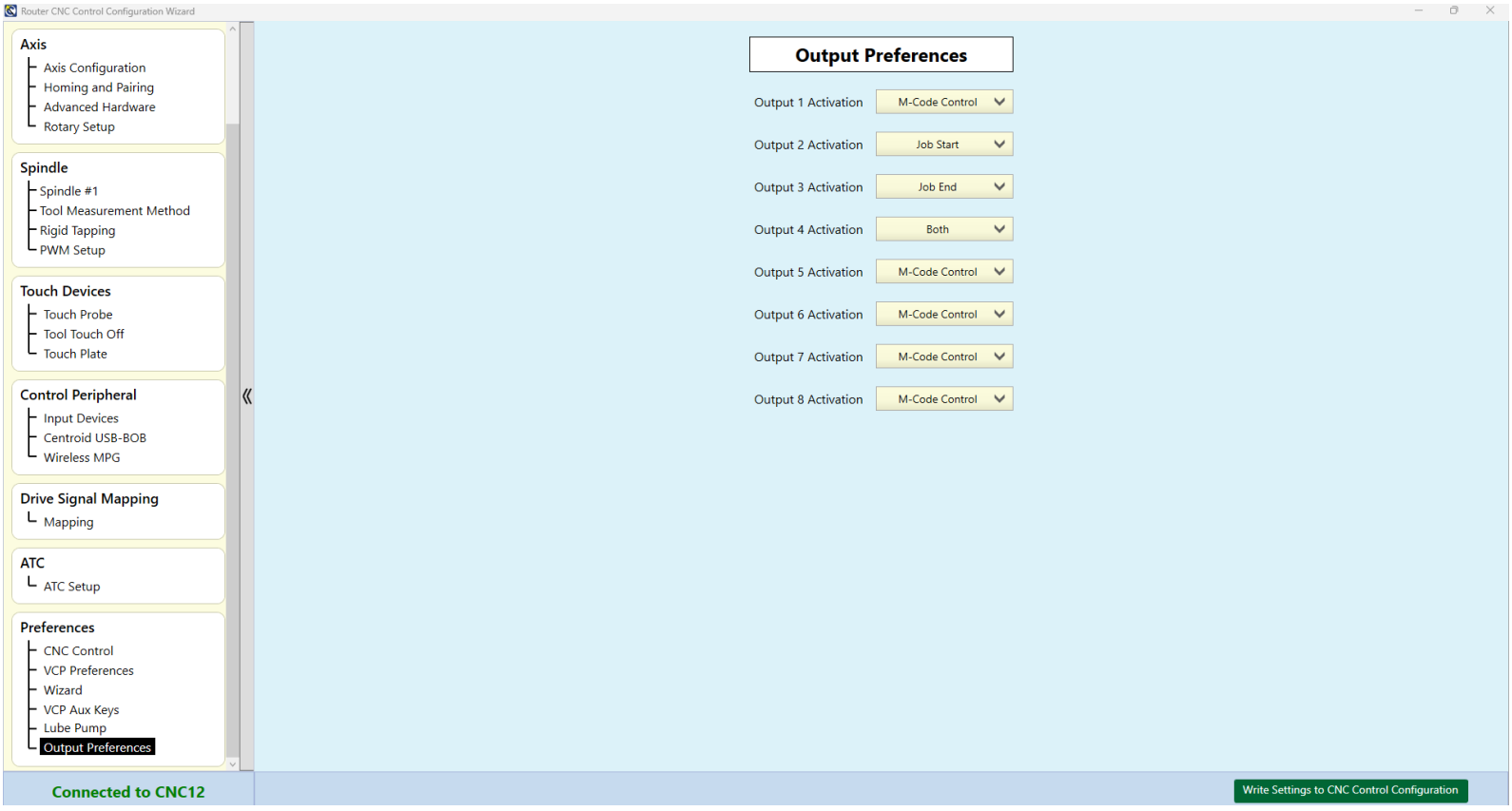
25. Adaptive Feedrate Modulation based on Spindle RPM. As the spindle RPM varies the machine tool Cutting Feedrate will slow down or speed up to keep the chip load the same. Feature requires a v5.4 Pro or Ultimate License. P078 bit 1 (add 2) will turn the feature on. (Note: a G code to turn AFM on and off will be added in v5.5 to make this feature more convenient to use, for now you turn it on and off witth the parameter).

26. Added Wizard canned logic for Hickory Lathe Electric ATC Turrets. Requires a v5.4Pro or Ultimate License

27. Improvements and additions to G-Code Backplot, new Display Options menu.

    graphic goes here.

28. Added 'Output Preferences' page to wizard which allows a global command for each output to be issued automatically. Choices for Automatically Activate Output upon Job Start or Job End or Both or M code control which is the default mode.



29. Centroid API improvements and changes to facilitate complete CNC12 UI replacement as well as better performance for all Apps using the Centroid API. Enhanced Centroid APi speed/responsiveness improvements. This base architecture change improves any app or feature that uses the Centroid API. Such as: “CHIPS”, Paper Tools, Centroid VCP, Centroid CNC setup Wizard, Centroid Plasma Restart Mode, Centroid input/output diagnostic screen. Plasma/Laser Material profile manager and more.

Calls Added

– State.GetConsoleType

– State.SetConsoleType

– WCS.SetWorkpieceLocation

– Job.SetSystemVariable

– Axis.GetPower

– System.GetHighestLicenseVersion

– Tool.SetToolHeightOffsetAmount (fixed typo)

– Tool.SetToolDiameterOffsetAmount (fixed typo)

– Tool.Save

– InboundComm.ChangeDroType

Calls Deprecated

– State.SetPlasmaInterruptPoint. Restart mode graph no longer displayed within CNC12

– Job.Load(string path, string cnc12_working_directory). Function created a file as a “dead drop” for CNC12 to load. This causes a race condition with the improvements introduced in R19862-R20434

– Tool.SetToolHeightOffsetAmout. Typo in function name

– Tool.SetToolDiameterOffsetAmout. Typo in function name

– State.IsPCPoweringOff. Preferably handled via the outgoing pipe

Improvements

- Job.LoadJob and Job.RunCommand have overloads that do not require the working directory as an argument

- Fixed bug where Tool.GetHeightOffsetAmount would always return H1 rather than the active H value if no H value was specified

– Job.CancelExecution will issue the “Escape” key for the emulator/offline intercon install

– Added dedicated support for Visual Basic via VB_[class] classes to resolve ambiguity between class and class instance

– Fixed bug where WCS.SetWorkpieceOrigin was not working

– Added ERROR_LICENSE_LOCKED error code

- Restructured pipeline/packets between CNC12 and the API to reduce time complexity. Total Wizard write time reduced from 10.77 seconds to 1.11 seconds

- Added outbound data pipe from CNC12 to the API with queue and message handlers which facilitate real time data with no request necessary. (you can pipe data directly to an element without having to ask for it)

Communication Types are

DRO_UPDATE

Sends the coordinates of the dro as a double array (8)

CNC12_SHUT_DOWN

Sends whether CNC12 is shutting down gracefully. A crash of CNC12 will probably not send this message

PC_SHUT_DOWN

Sends whether CNC12 has triggered a shutdown of the PC. A half second sleep is enacted in CNC12 after this message has been sent

MESSAGE_WINDOW_MESSAGE

Sends a message as it's printed in the message window

An event handler needs to be attached to CNCPipe.MessageRecieved and CNCPipe.StartListening called to begin to receive messages from the outbound pipe.

CNCPipe.StopListening can be called to no longer receive messages.

CNCPipe.ClearUnhandledMessages and CNCPipe.TryPopUnhandledMessage can be used to access the last 1000 unhandled messages (messages where e.Handled == false). It is estimated that 1000 messages is roughly the last 30 seconds of dro updates/CNC12 message window messages.

Speed tests show that updating the dro of a test app via requesting the information through Dro.GetDro resulted in an updated value every 4 ms whereas getting the same information through the outward pipe from the DRO_UPDATE communication type resulted in an updated value every < 1 ms!

CNC12 Outbound communication will now report current job info: Current line number, stack level, and running job

Added API call Job.GetCurrentlyRunningJob(out string current_job) which will return the current running job, subprogram, api command, mdi, or no job running.

On going Centroid API support and user development discussions are here.

https://centroidcncforum.com/viewforum.php?f=72

30. Added a new App SystemVariableViewer.exe to the CNC12 installer. Provides real time system variable viewing.

31. CNC12 will now use Windows Clipboard for its copy/paste operations

32. Reconfigured the Color Picker to be more useful search so it searches for color description names (instead of profiles).

33. Removed Plasma-Router Swap from installer, as it is no longer relevant since cnc12 now installs in it own directory for each variant.
34. Plasma Profile Manager will now prompt to select a profile set on first launch.
35. Added parameter 523 bit 8 (Add 256) to be able to invert the thc touch input (SV_THC_TORCH_TOUCH)

36. Dimension Menu - F3 button "Goto line" will now highlight, center on, and zoom into the selected line
37. Dimension Menu - F4 button "Measure" will now train the starting point cross hair on the mouse until the mouse is clicked (Left button down). At this point, the starting point will be anchored at that location and the ending point cross hair will be trained on the mouse.
38. Misc. improvements to the CNC12 G code program Backplotter. Such as Fixed bug where Next and Prev Line buttons in the dimension menu were not working, improved
39. Rearranged information on the RTG screen to better accommodate longer file names

40. Added the ability to control the attributes and dimensions of parameters 700-799 (OEM feature). This is done through the new file parm_attribs.xml in the form of:

```
<parameters>
  <parameter>
    <number>700</number>
    <dimension>1</dimension>
    <attribute>0</attribute>
    <axis>-1</axis>
  </parameter>
</parameters>
```

41. You can choose whether you want the old backplot for RTG or the new dynamic graphic engine. P150 bit 7 (add 128) will now allow the backplot (F8 Graph) to be used for RTG. Note: Old license = old graphics engine, a v5.4 license = new graphics engine or old your choice.

42. Acorn: Added a timer to Unclamping tool that will fault the system, more inline with the Hickory/Oak PLC logic
43. Acorn: Added Check in M6 Tool Rack Macros to check for if Tool is in spindle
44. Acorn: Added Drawbar Up Output (Unclamp the Tool), Drawbar Up/Down Inputs for compatibility with certain types of ATC spindles. When both Drawbar up/Down are off, it is considered to have a Tool in spindle (Tool prohibits drawbar from reaching down position). Drawbar Up is unclamp, Drawbar down is Clamp without tool

45. Added string variable #401 to allow for custom error messages. Setting the variable to a non-empty string will display the string as an error message and cancel a job.
46. Added #25033 to give license levels, Free = 0, Pro = 1, Ultimate = 2, Ultimate Plus = 3
47. Fixed bug where "Do not use Machine Home" wouldn't stick in the wizard
48. Fixed a bug with P424 that caused the new PWM delay feature to only be applied when changing from a zero to non-zero spindle speed
49. Fixed a bug that would cause a crash when trying to backplot a job using M293

50. Fix a bug in CNC12 that would cause the license file not to register when it takes a long time for the control board to communicate with CNC12
51. Added Checks to Axis_Calibration macro for when the Distance Moved is near zero or if the input measurement is zero, both of which would result in errors in pitch calculations.
52. Updated the part_zero_auto_z macro to allow the Centroid DP-7 Probe to work when setting Z Part Zero
53. Updated parameter descriptions for P406 and P407
54. Added a fix for Ethercat Drive Leadshine EL7 “failed start up” (symptom: high load meter, no motion)
55. Added Checks to ensure set_laser_csr macro sets the angle properly. Ensures the angle is between 90 and -90 degrees. Macro now also assumes the "longer distance" side is the side your measuring, so if measuring the side in Y-Axis 90 degrees is factored in properly for the CSR Value
56. Fixed a bug where attempting to command a move to the current position would cause paired axes to

get out of sync when used in conjunction with M221-225

57. The CNC12 installer will now backup the intercon and icn_lath directories as well.

58. CNC12 Router: Corrected a typo in touch_plate_cycles_select for the Bore Cycle that used the first axis position for the second axis, resulting in unwanted movement
59. Fixed a bug where jobs would pause when transitioning from a linear (G1) move to a rapid (G0) move when using a feedrate override greater than 100% when there was no commanded movement between the two lines.
60. Added Checks to Axis_Calibration macro for when the Distance Moved is near zero or if the input measurement is zero, both of which will result in errors in pitch calculations
61. System variable #25014 will now return a 3 for Router and a 4 for Plasma
62. System variable #400 will return a string of the current install directory (C:\cncm, C:\cnct, C:\cncr, etc.)

63. Removed Parameter 219 as the vcp is now controlled via the control configuration menu

64. In the CNC12 control config menu, we separated the feature "Press Cycle Start twice to start a job" from “Jog Panel Required” so you can choose them individually.

65. Fixed bug where wizard would display a fatal error if outputs were not defined and jtech laser configuration presets were used

66. Removed Parameter 5 bits 0 and 1 as Home In Place is now an option in the CNC12 control configuration

67. probe_cycles_history.txt will now record length of slot/web probe move in both X and Y directions if applicable

68. Added a reassurance message to the startup screen of CNC12 to indicate that the USB-BOB system is being set up
69. Fixed a bug where the disconnect warning message for USB-BOB would not show up in the message window properly.
70. Fixed a bug where inverted USB-BOB inputs would trigger when hot swapping the USB-BOB
71. Fixed a bug that prevented the simplified PLC Diagnostic Screen from maintaining focus when swapping between board (Acorn/AcornSix/Hickory) and USB-BOB IO
72. Improved setup process and messaging during the USB-BOB setup for better UX.

73. Added Paper Tools App Support. More info here. https://centroidcncforum.com/viewtopic.php?p=98634#p98634

74. PWM Power Floor % and P815 (Add4), adds yes/no toggle for “Only Apply Floor During PWM Velocity Modulation Moves”

75. Updated Hickory PLC to include C-Axis Enable output for C-Axis style that uses a clutch or gear to engage the motor to spindle. mfunc51 corrected for hickory memory bits.

76. Changed auto_z VCP button to directly call the touch_plate_z_zero.cnc macro. This button was the predecessor to the auto_z_plate button.

77. Added "Material Size" to Mill/Router Intercon so user can specify material size to be displayed in the G

code backplot graphics and RTG screens.

78. Added in version v5.4 we introduced a new parameter that helps the WMPG client run on slower/older computers.

    Parameter 414 "Wireless WMPG Sleep Rate" to control how often the wireless mpg client sends data (in milliseconds) to CNC12 which will reduce the cpu usage of the wireless mpg client. Only intended or certain PC's that exhibit excessive CPU usage with the Wireless MPG client. It is not recommended that this value exceed 300 milliseconds (a number this high will cause a delay in wheel movement to actual machine movement) Default value = 1 millisecond. Our in house testing with a 'good' computer showed that we could go up to 100 without seeing any noticeable lag in the wireless WMPG performance from a user standpoint. If you have to set this number higher than 150 to get the CPU useage down the issue is most likely a PC /Windows issue. Lower number = fastest update rate for best WMPG performance. Use a good computer.

    If you have an older slow computer and the MpgClient.exe is using a lot of CPU resources try P414 = 50 or 75 this range is a good compromise between WMPG responsiveness and reducing the CPU usage. If you are still having issues Follow the WMPG trouble shoot guidance here. https://centroidcncforum.com/viewtopic.php?t=10632

79. Fixed a WCS lockout bug that was not locking out WCS origin values.

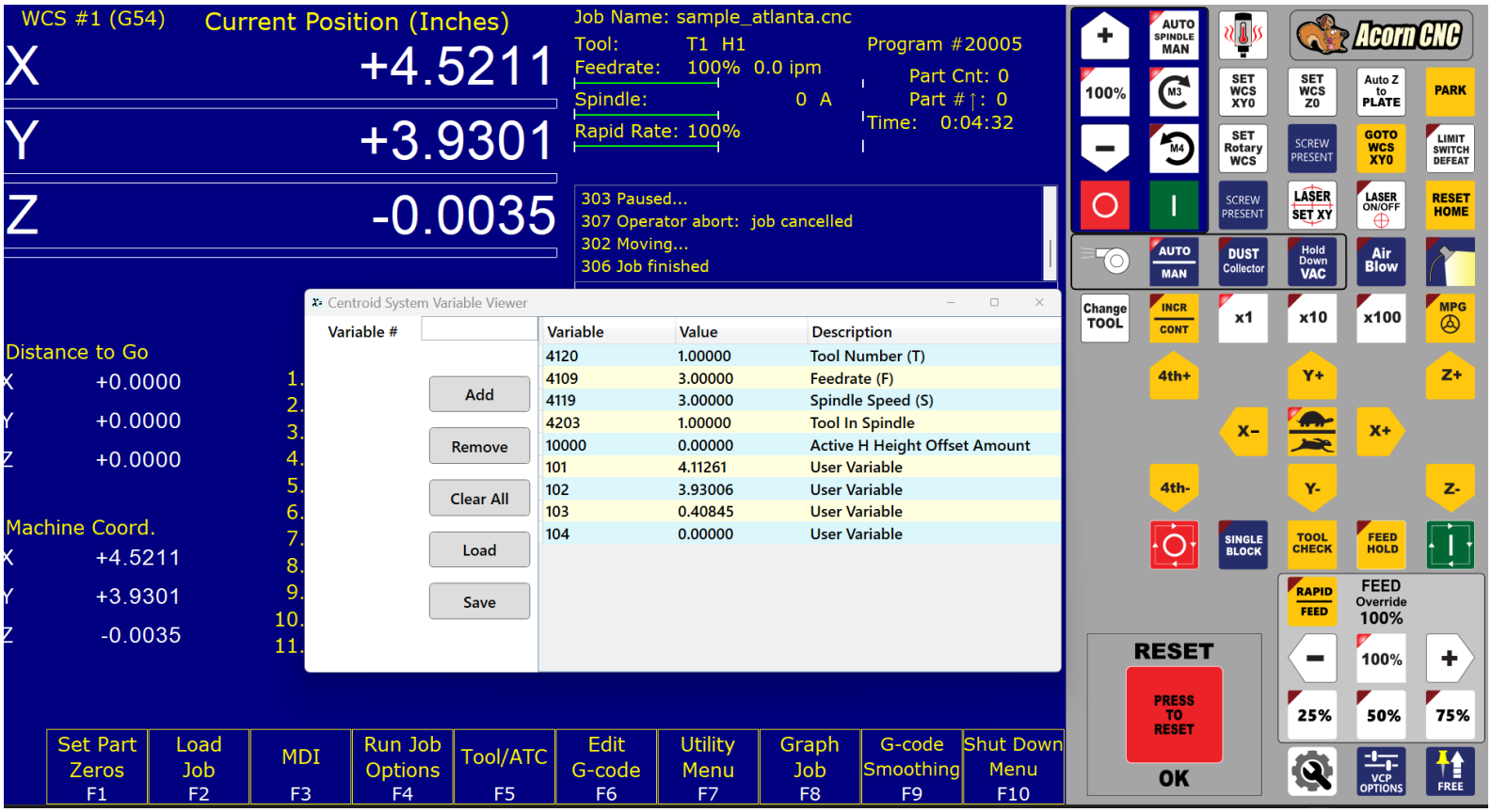
80. Better DRO Decimal place control. Parameter 263 for number of decimal resolution is now used in place of the bit that used to control this in Parameter 143. With P263 you can now you can just type in how many decimal places you want on the DRO display resolution.

81. Updated the CNC12 Mill and Router Part Menu seek Z zero with touch probe macro to retract the probe off the probed Z zero surface by the value of parameter 13 "Probing Recovery Distance" (default value is .050"/1.27mm) from the surface if the probe is still tripped after the initial probe clearing move. Note: The macro will keep track of the probed surface location even in the case of the additional clearing move, so when operator defines the touched surface location using the menu the surface WCS is set properly to the probed surface even though it is above that surface.

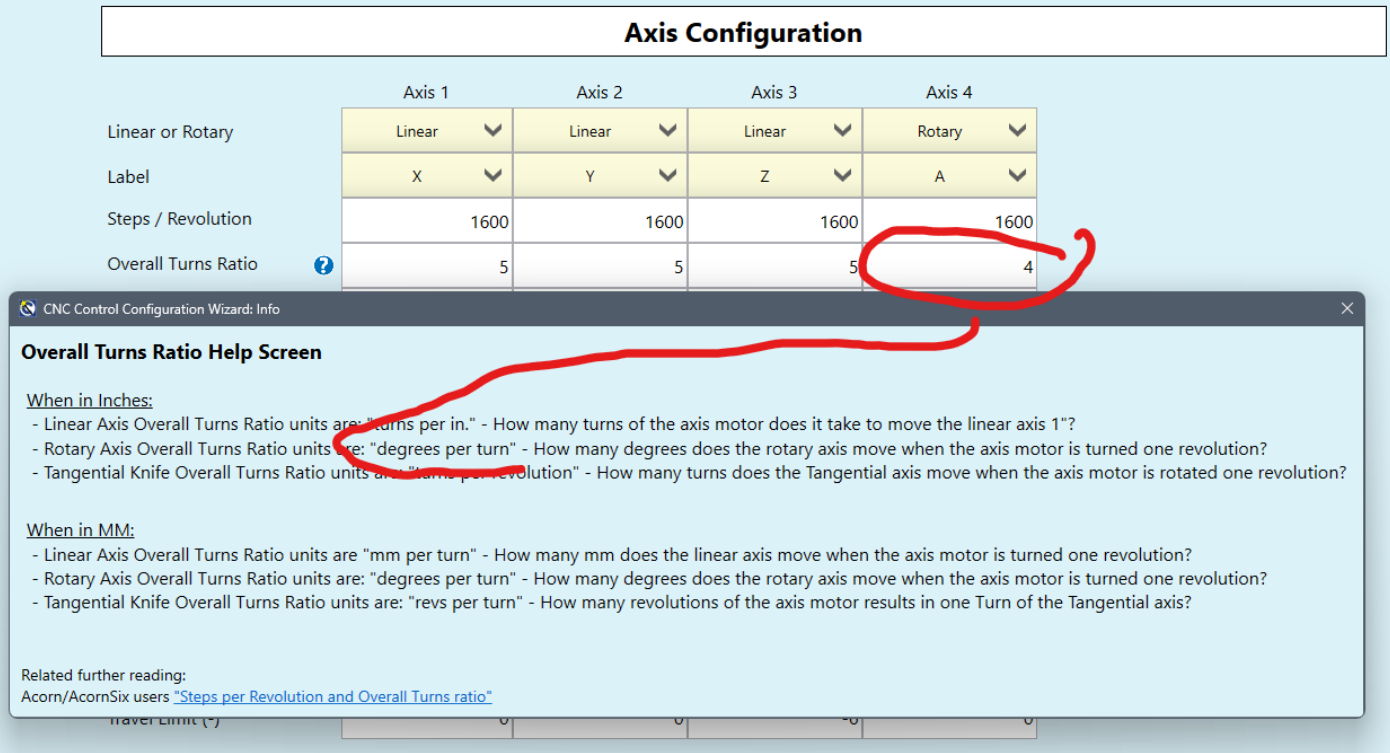
82. Centroid System Variable Viewer Tool. Great for debugging custom macros.

Press <CTRL W> to launch

Alternative, launch from c:\cnc?\SystemVariableViewer.exe



83. Wizard Rotary Overall turns ratio Units are in “degrees per turn” whether you are in mm or inch mode. got a 10:1 table? then your degrees per rev is = 36, got a 360:1 table? then the degrees per rev = 1 . 90:1 the degrees per rev = 4 etc..



Note: When upgrading from older version of CNC12, DO NOT just type in the old inch number that was in the units of “turns per degree”. Please use the new unit of “degrees per turn”.

1. Fixed bug where CNC12 would crash due to the outpipe sending job information if the stack level increased while running an internal macro
2. RTG with Rotary Axis and multiple WCS locations in same G code: Fixed bug where a rotary axis WCS value would appear incorrect in 3D RTG when WCS is changed in Job via G54, G55, etc.
3. Fixed License version information reporting in the Report.txt file included in the report.zip file.
4. Windows 11 CNC12 Start Up improved due to Microsoft change: Selecting "Start CNC12 at Startup" will now schedule a task to launch cnc12 on user login instead of setting a start up application in the registry keys. This is done to get around Microsoft's decision to add a 90 second delay to startup applications in Windows 11
5. WMPG Performance Mode has been deprecated. Removed Parameter 855. WMPG performance mode has been set to "Quick Response" permanently as the other MPG profiles are no longer needed and have been deprecated.
6. Keyboard Shortcuts: ProbeCyclesHistory.exe won't launch via keyboard shortcut if the spindle is in manual mode or if the spindle is running. SystemVariableViewer.exe won't launch via keyboard shortcut if the spindle is in manual mode. This is done to solve keyboard shortcut conflicts between "Keyboard Jogging" and CNC12 utility applications
7. Lathe: Fixed bug hitting a tool check during a lathe job, going into the tool offset menu (F2), saving (F10), a crash would occur
8. CNC12 Menu display: Fixed bug where a free license would append "v5.3" token
9. CNC12 Menu display: Fixed bug where only one digit of the patch version (v5.40.X) appeared
10. CNC12 APi: Fixed an issue where the api call PLC.SetWatchList would fail after a number of inputs/outputs/memory bits were added to the watch list. The number of bits that would cause failure varies board to board. The call now allows up to 255 bits to be added without issue. This number can be increased in the future if needed.
11. Hickory: Fixed case where a Rotary Job only performs 51R (2^31 in motor counts) and would stop. https://centroidcncforum.com/viewtopic.php?t=12108

1. Added validation to P263. Values must be between 1 and 12. Default is 4. This fixes issues with double formatting (seen in the dro and mini coord displays)
2. Fixed bug where status window would not update if machine was not homed
3. Added a splash screen for starting CNC12 on startup so users know CNC12 is waiting on OS to fully boot before starting.

v5.40.03 Build Notes

Note: - there was no public release of v5.40.03.
- all these items below are included with the release of v5.40.04

1. PLC Detective now stays on top of other applications
2. Reports will now include all .mac files located within the system folder
3. Eliminated the start up splash screen for OS boot up finish and added new technique "Focus WatchDog"
4. Added "Focus WatchDog" that will keyboard maintain focus on CNC12 if "--exclusivefocus" is specified. This will remain on until an application is launched from CNC12. The Windows Key will toggle this on and off
5. Added WMPG power off reminder to when CNC12 triggers a shut down of the PC
6. Fixed a bug where the automatic resync or resync prompt (see P579) would not resync paired axes after restoring power to the motors
7. 3D RTG graphing will ignore the feedrate override percentage while drawing the part
8. Fixed bug where using arrow keys would crash the utility menu
9. Added "--fullscreen" as a startup argument. Should be used for 4:3 monitors only
10. Added antivirus detection and identification to the report.txt file for antivirus software that registers in the Windows Security Center
11. Added a PipeDisconnected Event Handler to the API
12. Added a timestamp to all outbound CNC12 API packets
13. Fixed bug where Lathe status window part count would overrun the program number
14. Added "Job Concluded" packet to the outgoing CNC12 communication
15. Fixed bug where AcornSix wizard would always overwrite the plc, regardless if custom plc was selected
16. Alt+M MDI will now work in the PID Encoder menu
17. Fixed bug where Centroid API would continue to propagate messages after CNC12 shutdown
18. Fixed bug where the graphics tool marker would appear in a different plane if the tool height offset changed during the job (H value, not H number)
19. Fixed bug where not having a tool loaded on a metric machine and using the tool library for graphing the tool marker would result in a very thin tool marker.
20. Fixed bug where toggling the advanced i/o display on and off while the g-code display is running would not redisplay the g-code
21. Fixed bug where measurement amount (F8 >> F4 >> F4) would be cutoff if the taskbar was hidden

- Fixed bug where the CNC12 PLC Diagnostic tool would not update with the color picker colors for Router and Plasma

- P404 will now return Z to G28 position after Tool Change during batch tool measurements

- P4 Bit 4 (add 16) which remembers the last open directory will now only apply to loading job files (F2 from the main screen)

- Fixed a bug where pressing "F8" while a MDI command was in progress would crash CNC12

- Fixed a bug where using Centroid API call Job.RunCommand and then pressing "F8" would crash CNC12

- Added overload to Centroid API call Dro.GetDro which will return axis positions as a double rather than tuple strings if the machine is homed. The return code for an unhomed machine is "ERROR_NOT_HOMED"

- In the wizard, changing the Max Rate value for a given axis to a number lower than the Fast or Slow Jog rates will clamp those rates to the new Max Rate

- Updated names of the CNC12 Color Picker. Graph Spindle Location and Tool Indicator color updated to #00FF00 for the Centroid Classic color profile

- Fixed some issues with the "Watchdog Enabled" pop up message box displaying on a Windows 11 machine

- Allin1DC/Oak systems: When P44 and P257 are zero, If a probe is not actively detected then the Tool Auto Measure will default to a TT using the probe input (P11)