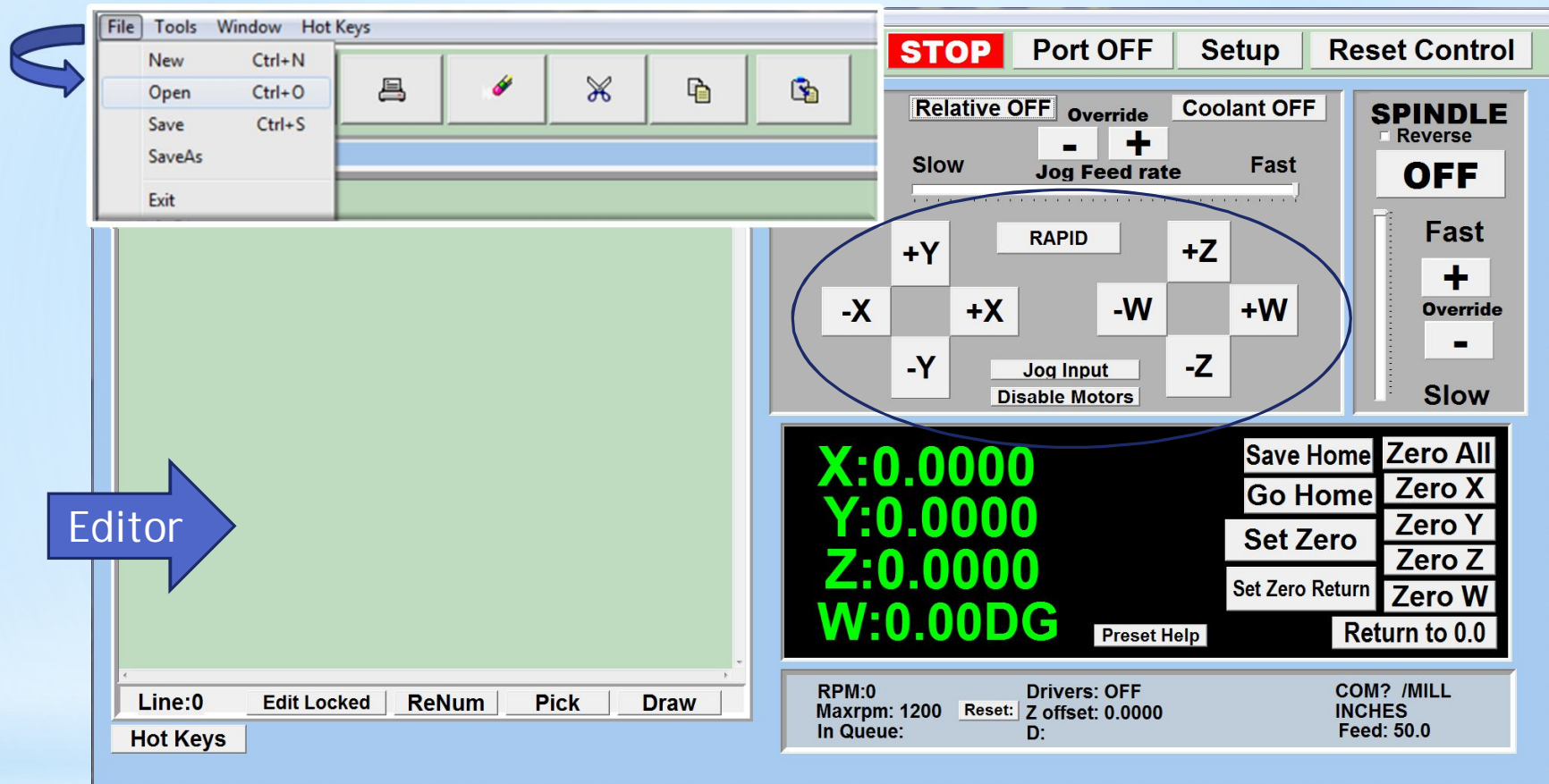
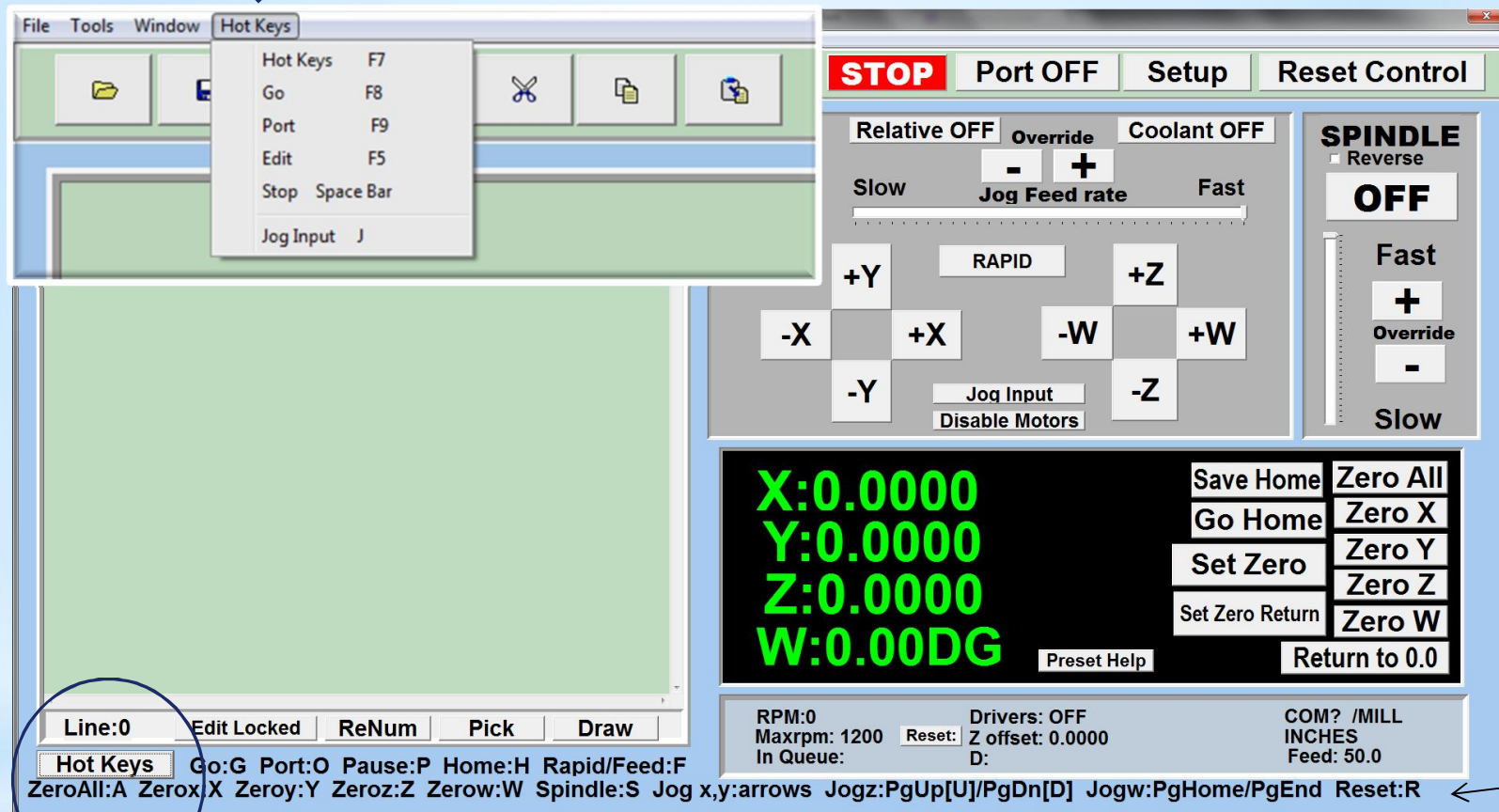


The Master Software is a PC based application system designed to work with 32 or 64 bit operating systems: Windows 8, 7, or Windows XP Pro with SP3 through your standard USB port interface on lap tops or desk tops.

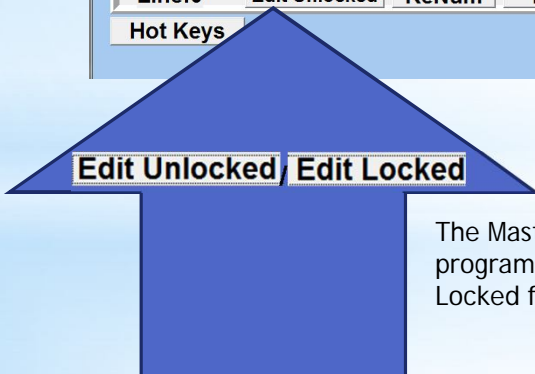
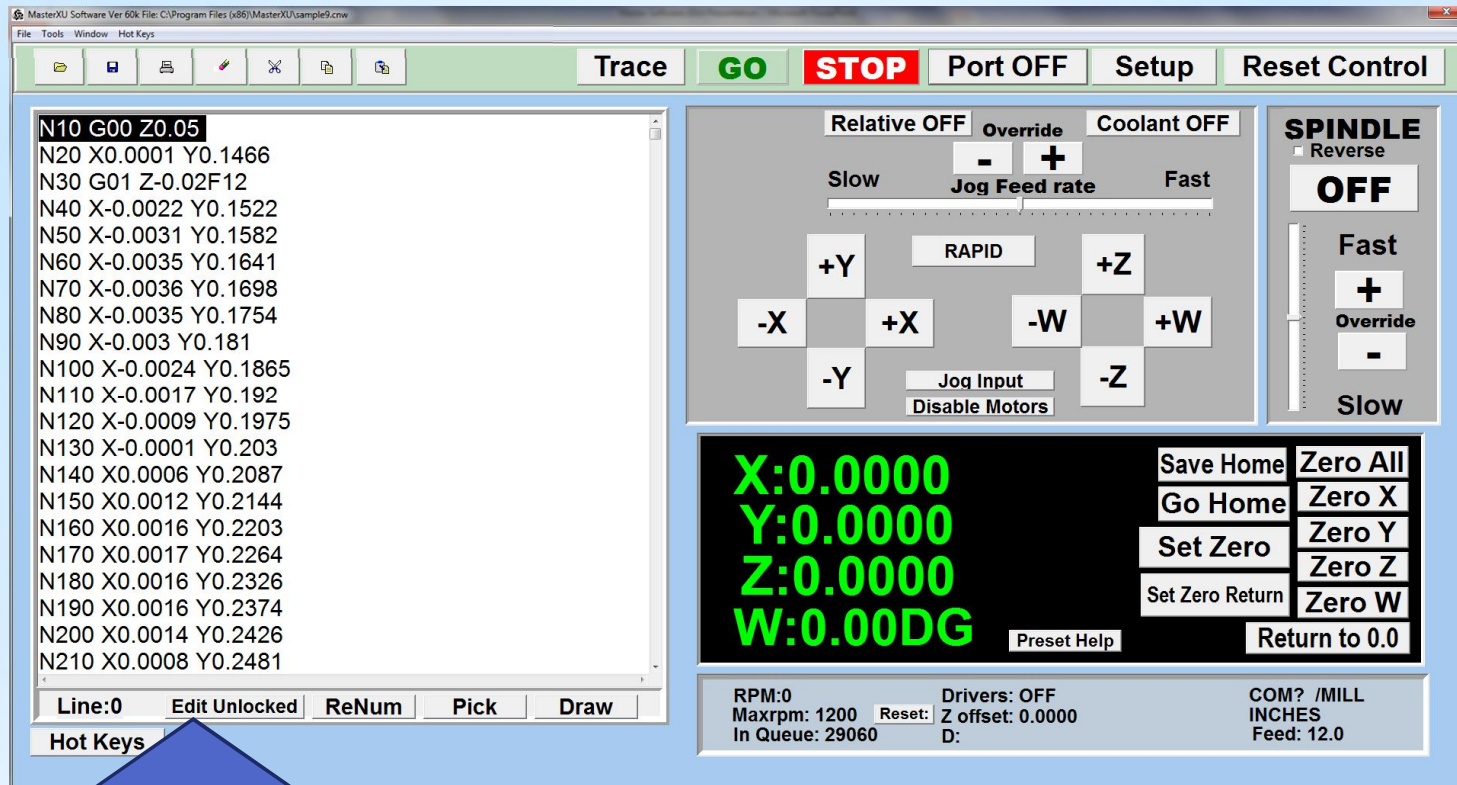




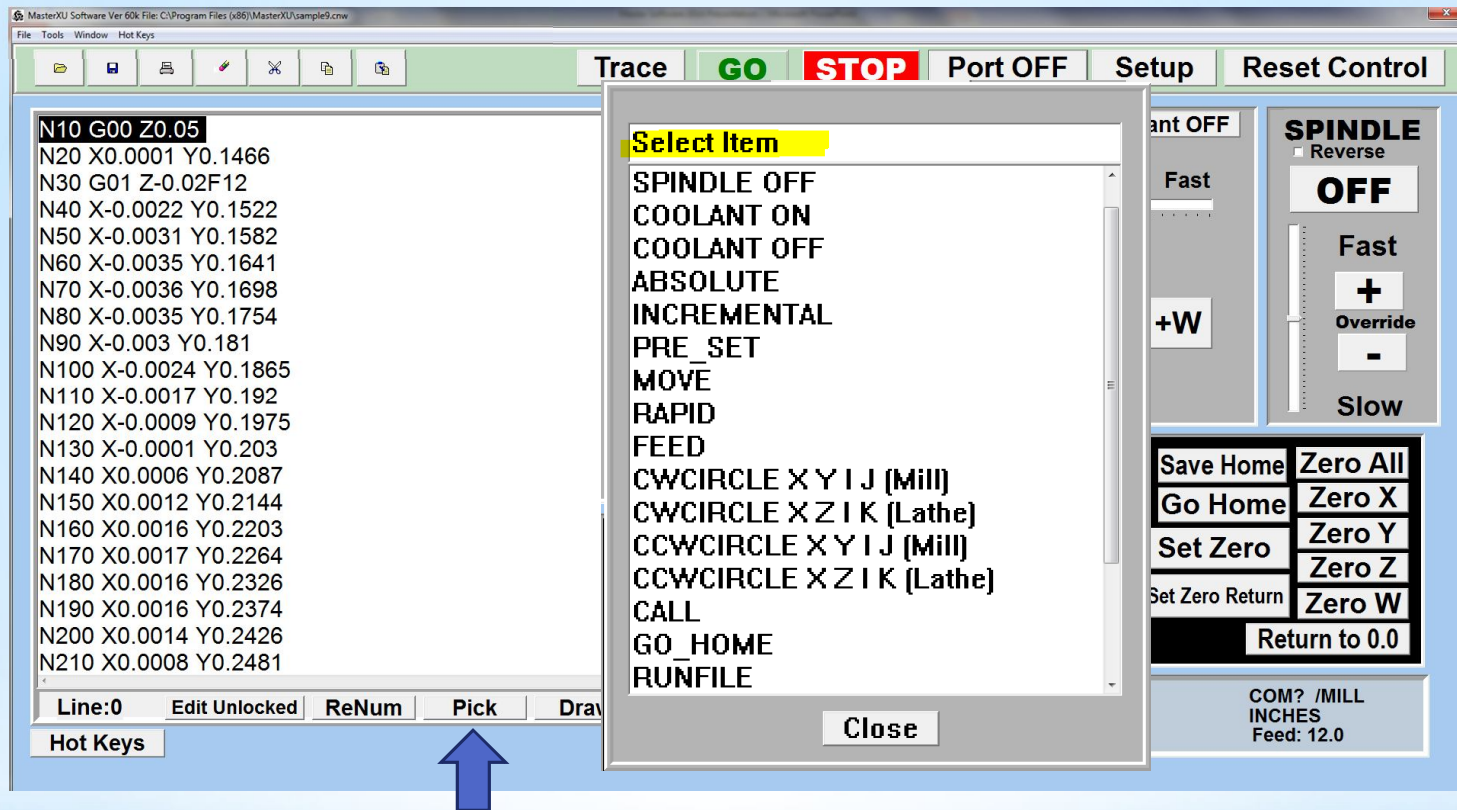
With a few clicks of the mouse or using touch screen technology, you can easily navigate through the Master interface importing saved programs into the Editor from the File drop down menu using standard windows features. You can type in a program or import CAM generated G-code tool paths into the Editor. You can jog any of the axes using your mouse, touch screen technology, or when the Hot Keys feature is activated, use the arrow keys on your keyboard. The X Y Z axes are displayed from the point of view of the cutter to avoid confusion. You also have the optional coolant and spindle speed controls.



Hot Keys is an alternative method to easily control your machine using your hard or on-screen keyboard. You can jog the axes with the arrow keys, press P to pause a program, press S to turn Spindle On, G to run a program, Space Bar to Stop, and/or J to record your individual movements one line at a time to create a program by teach mode. You also have options to zero your axes and reset your control as well.



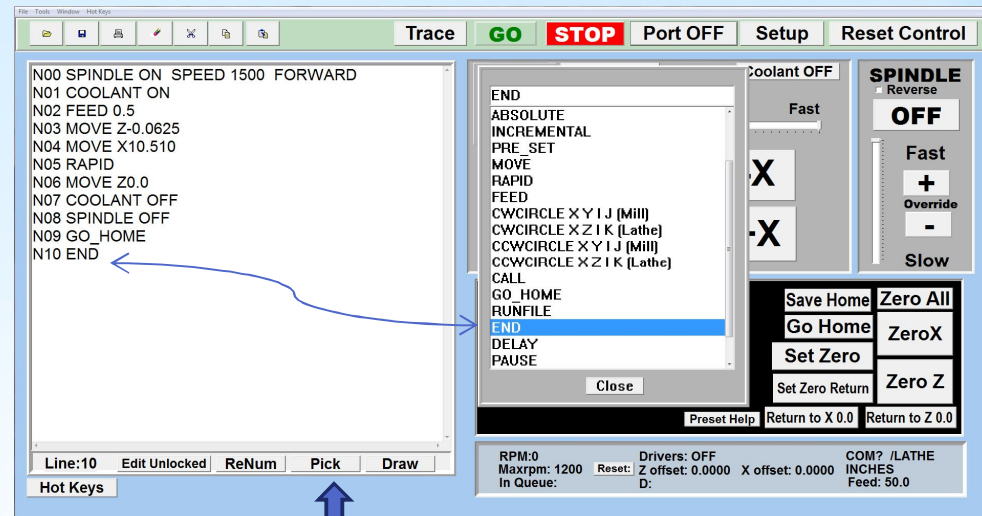
The Master defaults to an "Edit Locked" green editor screen. To create or edit your tool path program, you will have to "Edit Unlock" first which will turn the screen to a white color. This Edit Locked feature keeps your tool path program from being accidentally changed.



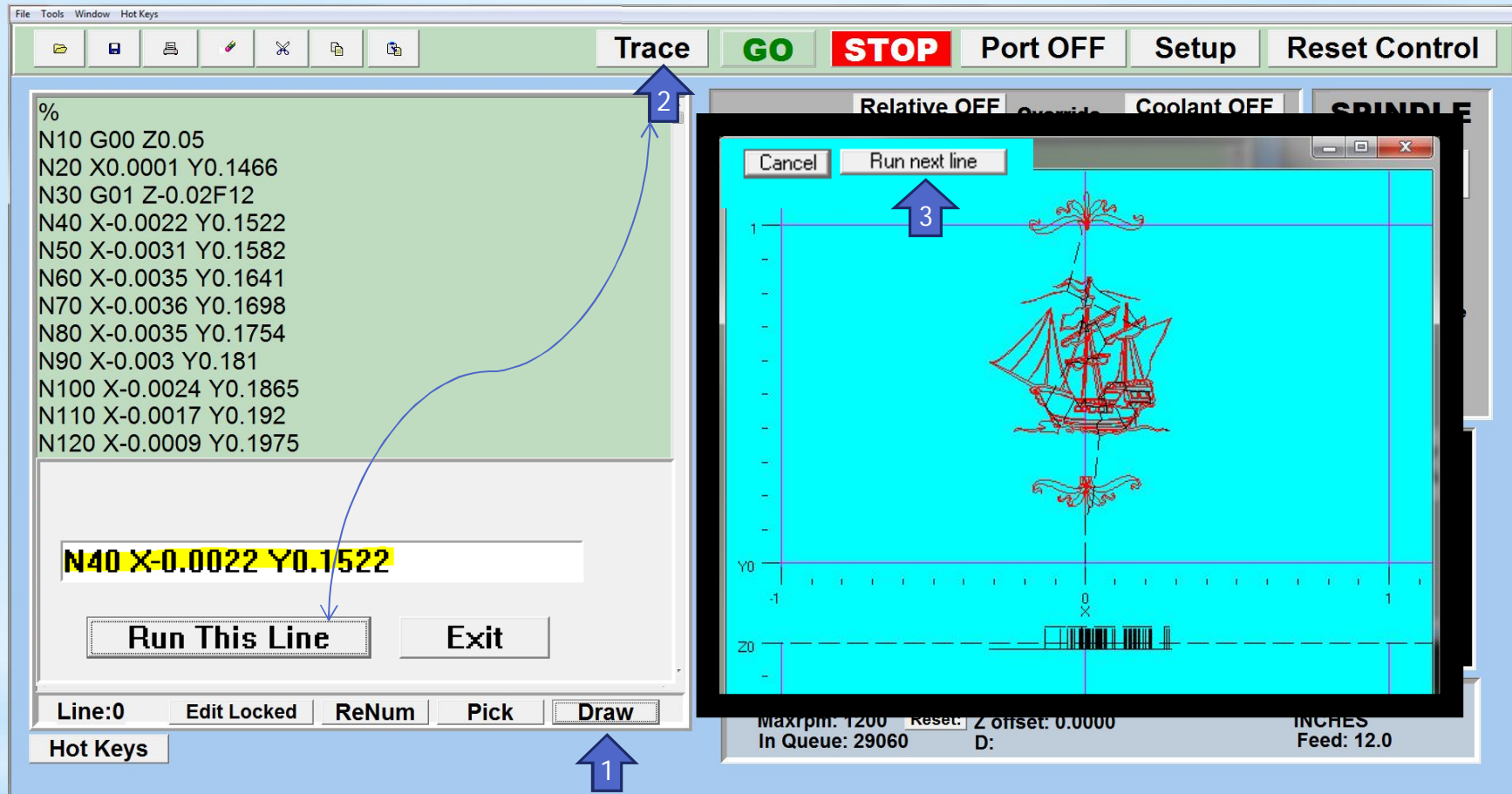
While in "Edit Unlock" mode, you can write FANUC style G-codes directly into the Editor, or select commands off the "Pick" menu and write your tool path program in conversational mode.



With the Pick Menu, you can easily write a simple program without G-codes. You can also interchange between codes and conversational commands in the same program. Using a CAM is always best for complicated applications and the Master has no problem receiving an imported FANUC based program with the correct post processor.



Pick Menu Features	Description
ABSOLUTE	Same as G90. The Master defaults to Absolute mode.
INCREMENTAL	Same as G91.
SPINDLE ON	Same as M03 for computer variable spindle control.
MOVE	Same as G00, G01. Move three or four axes simultaneously on the same tool path.
PRE_SET	Can't start from a 0.00 position? With Pre-Set, you can enter the coordinate you want to begin the tool path program. For example, Z axis is 0.5" above the material. Pre-Set will automatically start the Z coordinate at 0.5" on the counter. You can also pre-set a coordinate directly into the DRO counters, by clicking the cursor over the counter.
RAPID	Quickly retract or relocate the tool after an operation above the application. Rapid each individual axis up to 100 inches per minute of travel.
FEED	Slowly feed your axes with a minimum of 0.1 inches per minute of travel.
CWCIRCLE X Y I J	Same as G02. Enter in the coordinates for X Y I J and let the Master machine your arcs for you. Z movement can be added for helical applications.
CCWCIRCLE X Y I J	Same as G03.
CALL	CALL allows you to repeat as needed a sub-program with one line instead of having to re-write the sub-program several times within the larger program.
GO_HOME	GO HOME allows you to create an offset position on the machine. End your program with GO HOME on production runs to give you clearance of the axes to exchange parts on the machine.
DELAY	Stop the program at a particular point and delay further execution for the time shown in milliseconds up to 40,000.
PAUSE	Same as M00, or type in PAUSE in the middle of a program. Hit Enter to resume the program. You can also press P on the keyboard to pause the program.
RUNFILE	If the program is very long, using the RUNFILE command will read the program directly from your C drive.
\ (NOTE)	Using a \ in front of the notation, the Master will simply ignore this line in the program. This can be useful to place remarks within your program.
SPINDLE OFF	Same as M05.
COOLANT ON	Same as M08. Control the coolant pump directly into your program.
COOLANT OFF	Same as M09.
END	Same as M30.



1. Hit Draw to view your tool path program drawing. This can be helpful to quickly verify your program before running it. You also can slow down the drawing process with Draw Delay.
2. Run the machine on Trace mode. You can run each tool path independently, one line at a time to study the tool path movement to verify the position of the application and if any fixture/vise is in the way of the cutter's path.
3. Use Trace and Draw together to run each tool path independently in the Draw Window.



The screenshot shows the CNC MASTERS software interface. At the top left is the CNC MASTERS logo. Below it is a menu bar with 'File', 'Tools', 'Window', and 'Hot Keys'. A 'Tools' dropdown menu is open, showing options: 'Go From Line', 'Beep End Prog.', 'Delete Config.cn3', 'Find USB', and 'Change Text Size'. The main window is divided into several sections:

- Line Editor:** A list of G-code lines (N20 to N150). Line N20 is highlighted in blue. Below the list are buttons: 'Line:0', 'Edit Locked', 'ReNum', 'Pick', and 'Draw'. A 'Hot Keys' button is at the bottom left.
- Control Panel:** At the top right, buttons for 'Trace', 'GO' (green), 'STOP' (red), 'Port OFF', 'Setup', and 'Reset Control'. Below these are 'Relative OFF', 'Override' (with '-' and '+' buttons), and 'Coolant OFF'. A 'Jog Feed rate' section has 'Slow', 'Fast', and 'Jog Input' buttons. A 'Disable Motors' button is at the bottom of this section.
- SPINDLE Control:** A section on the right with 'Reverse' checkbox, 'OFF' status, 'Fast', 'Slow', and 'Override' buttons.
- Axis Position Display:** A large black area showing 'X:0.0000', 'Y:0.1466', 'Z:-0.0200', and 'W:0.00DG' in green. To the right are buttons: 'Save Home', 'Go Home', 'Set Zero', 'Set Zero Return', 'Zero All', 'Zero X', 'Zero Y', 'Zero Z', 'Zero W', 'Preset Help', and 'Return to 0.0'.
- Status Bar:** At the bottom, showing 'RPM:0', 'Maxrpm: 1200', 'In Queue: 30', 'Drivers: OFF', 'Reset:', 'Z offset: 0.0000', 'Stop', 'COM? /MILL', 'INCHES', and 'Feed: 12.0'.

Numbered callouts point to specific features:

- 1:** Points to the highlighted line N20 in the line editor.
- 2:** Points to the 'Tools' menu.
- 3:** Points to the 'GO' and 'STOP' buttons.
- 4:** Points to the 'Disable Motors' button.

1. When running a program, the In Queue window will display the tool paths fed into the machine control unit. The master feeds the control unit a pattern of three tool paths at a time for "continuous machining" to avoid slight interruptions as the machine waits for it's next tool path command.
2. Under Tools, run a "Go From Line" command if you need to begin the program from anywhere other than the first line.
3. Press Go to run your program or Stop to kill all CNC movement on the machine.
4. Click Disable Motors to crank each axis by hand, and then press Reset Control to re-energize the axis motors.



File Tools Window Hot Keys

Trace **GO** **STOP** Port OFF Setup Reset Control

%
N10 G00 Z0.05
N20 X0.0001 Y0.1466
N30 G01 Z-0.02F12
N40 X-0.0022 Y0.1522
N50 X-0.0031 Y0.1582
N60 X-0.0035 Y0.1641
N70 X-0.0036 Y0.1698
N80 X-0.0035 Y0.1754
N90 X-0.003 Y0.181
N100 X-0.0024 Y0.1865
N110 X-0.0017 Y0.192
N120 X-0.0009 Y0.1975
N130 X-0.0001 Y0.203
N140 X0.0006 Y0.2087
N150 X0.0012 Y0.2144
N160 X0.0016 Y0.2203
N170 X0.0017 Y0.2264
N180 X0.0016 Y0.2326
N190 X0.0016 Y0.2374
N200 X0.0014 Y0.2426

Line:0 Edit Locked ReNum Pick Draw

Hot Keys

X 0.0000 Y 0.0000 Z 0.525 W 0.0000

Relative ON

Coolant OFF

SPINDLE
Reverse
OFF
Fast
+
Override
-
Slow

-X +X -Y +Y -W +W -Z +Z

Jog Input
Disable Motors

X:0.0000
Y:0.0000
Z:0.0000
W:0.00DG

Save Home
Go Home
Set Zero
Set Zero Return
Preset Help

Zero All
Zero X
Zero Y
Zero Z
Zero W
Return to 0.0

RPM:0
Maxrpm: 1200
In Queue:

Reset:
Drivers: OFF
Z offset: 0.0000
D:

COM? /MILL
INCHES
Feed: 50.0

Clicking the Relative ON feature will allow you to enter a specific value to jog any of your axes to an exact location. Without having to write a program, you can jog an exact amount on any of the axes without needing to keep the key pressed down and mistakenly over-step the movement releasing your finger too slowly off the jogging button. For example, let's say you need to drill a hole exactly 0.525" using the Z. So you enter 0.525 in the Z box. Next, adjust the JOG FEED RATE slider for the desired feed rate. Then "click once" on the +Z or -Z button to activate the travel. In this case you click the -Z button first to drill the hole, and then click the +Z button to drive the axis back up.



File Tools Window Hot Keys

Trace **GO** **STOP** Port OFF Setup Reset Control

N01 INCREMENTAL
 N02 MOVE X0.0000Y0.0000Z0.0000W0.0
 N03 MOVE X0.5471Y0.0000Z0.0000W0.0
 N04 MOVE X0.5471Y0.0824Z0.0000W0.0
 N05 MOVE X-0.3422Y0.1056Z0.0000W0.0
 N06 END

Relative OFF Override Coolant OFF

Slow **Jog Feed rate** Fast

+Y RAPID +Z

-X +X -W +W

-Y Jog Input -Z

Disable Mot

SPINDLE
☐ Reverse
OFF

Fast
 +
 Override
 -
 Slow

X:-0.3422
 Y:0.1056
 Z:0.0000
 W:0.00DG

Save Home Zero All
 Go Home Zero X
 Set Zero Zero Y
 Set Zero Return Zero Z
 Zero W

Preset Help Return to 0.0

Line:0 Edit Locked ReNum Pick Draw

Hot Keys

RPM:0 Maxrpm: 1200 In Queue: Drivers: OFF Z offset: 0.0000 D: COM? /MILL INCHES Feed: 50.0

Jog Input - Teach Mode: You can create a tool path program by storing each point-to-point movement by simple jogging an axis. Click on the Jog Input button to store each movement on the Editor screen. You can then add Spindle ON, feed commands, and press GO to run the new program as needed. This is a great feature to help you learn to create a program by the movements you make on the machine without necessarily writing out an entire program first.



File Tools Window Hot Keys

Trace **GO** **STOP** Port OFF Setup Reset Control

Relative OFF Override Coolant OFF

Slow **-** **+** Fast

Jog Feed rate

+Y FEED +Z

-X +X -W +W

-Y Jog Input -Z

Disable Motors

SPINDLE

☐ Reverse

OFF

Fast

+

Override

-

Slow

X:0.0000

Y:0.0000

Z:0.0000

W:0.00DG

Save Home Go Home Set Zero Set Zero Return

To All Zero X Zero Y Zero Z Zero W

Preset Help Return to 0.0

Line:0 Edit Unlocked ReNum Pick Draw

Hot Keys

RPM: 1055 Maxrpm: 1200 In Queue: Reset: Drivers: OFF Z offset: 0.0000 D: COM? /MILL INCHES Feed: 29.7

1. Jog Feed and Rapid with Override: You can adjust feeds using the slider from slow minimum 0.1" per minute to a rapid of 100" per minute of travel. The **-** **+** allows you to fine tune feeds in 10% increments while the program is in motion.
2. Spindle Speed with Override: You can adjust speeds using the slider from a slow minimum 0 RPMs to the max RPMs according to the machine. The **-** **+** allows you to fine tune RPMs in 10% increments while the program is in motion.



File Tools Window Hot Keys

Trace GO STOP Port OFF Setup Reset Control

Relative OFF Override Coolant OFF

Slow Jog Feed rate Fast

+Y FEED +Z

-X +X -W +W

-Y Jog Input -Z

Disable Motors

SPINDLE

Reverse

OFF

Fast

+

Override

-

Slow

X:0.0000

Y:0.0000

Z:0.0000

W:0.00DG

Save Home Zero All

Go Home Zero X

Set Zero Zero Y

Set Zero Return Zero Z

Zero W

Preset Help

Return to 0.0

Line:0 Edit Unlocked ReNum Pick Draw

Hot Keys

RPM: 1055 Maxrpm: 1200 In Queue: Drivers: OFF Z offset: 0.0000 D: COM? /MILL INCHES Feed: 29.7

Reset:

1. Pre-set directly into the counters by typing in your beginning coordinate if you cannot start your program with 0.00.
2. The counters X Y Z can display in inches or mm. The optional W rotary table axis displays in degrees.
3. Zero all counters or zero each counter independently. With one click of the Return to 0.0 button, all axes will travel back to 0.0 on the machine.



FileToolsWindowHot Keys

TraceGOSTOPPort OFFSetupReset Control

SET_ZERO X-7.2261 Y-4.0418 Z-1.2415 W0.0000
HOME X15.25 Y0.00 Z4.0

Relative OFFOverrideCoolant OFF

SlowJog Feed rateFast

+Y

RAPID

+Z

-X

+X

-W

+W

-Y

Jog Input

-Z

Disable Motors

SPINDLE

Reverse

OFF

Fast

+

Override

-

Slow

X:0.0000
Y:0.0000
Z:0.0000
W:0.00DG

Save HomeZero All

Go HomeZero X

Set ZeroZero Y

Set Zero ReturnZero Z

Zero W

Return to 0.0

Preset Help

Line:2Edit UnlockedReNumPickDraw

Hot Keys

RPM:0Maxrpm: 1200In Queue:

Reset:Drivers: OFFZ offset: 0.0000D:

COM? /MILLINCHESFeed: 50.0

Set Zero

Set Zero Return

Set and save your 0.00 position on the machine. These coordinates will be recorded as the first line of the program in the Editor screen. Should you desire to return to this program at a later date, you will only have to click on the Set Zero Return button. This will command the machine to automatically jog each axis to the saved "set" 0.00 position according to the recorded coordinates at the first line of the program.



File Tools Window Hot Keys

Trace GO STOP Port OFF Setup Reset Control

SET_ZERO X-7.2261 Y-4.0418 Z-1.2415 W0.0000
HOME X15.25 Y0.00 Z4.0
MOVE Z-0.625 F0.25
MOVE X5.0 Y-1.325 F1.5
MOVE Z0 R
MOVE X0 Y0 R
MOVE Z-1.25 F0.25
MOVE X5.0 Y-1.325 F1.5
GO_HOME
END

Relative OFF Override Coolant OFF

Slow - + Fast

Jog Feed rate

+Y RAPID +Z

-X +X -W +W

-Y Jog Input -Z

Disable Motors

SPINDLE

Reverse

OFF

Fast

+ Override -

Slow

X:0.0000
Y:0.0000
Z:0.0000
W:0.00DG

Save Home Zero All
Go Home Zero X
Set Zero Zero Y
Set Zero Return Zero Z
Return to 0.0

Line:9 Edit Locked ReNum Pick Draw

Hot Keys

RPM:0 Drivers: OFF COM? /MILL
Maxrpm: 1200 In Queue: Reset: Z offset: 0.0000 INCHES
Feed: 50.0

A "Home" and a "Zero" position will help immensely when running the same program multiple times. For example: Let's say you need to machine one application times 100 pieces. This usually requires a jig to retain that physical 0.00 position. But in this case, you want the program to end with clearance of the axes to easily switch out the next piece of stock and start again. With Save Home, you have the ability to save this home position while still retaining your Set Zero position where the machine will mill your part out. Pressing Save Home will record this new position under the Set Zero line in your program.

Pressing Go Home will jog your axes back to your "saved home" position where you originally pressed the Save Home command. You can also input GO_HOME from the Pick Menu as its own tool path in your program. So at the completion of your program it will end the axes at your Home position. Replace your piece of stock, and then press the Return to 0.0 to start your next run.



Trace **GO** **STOP** **Port OFF** **Setup** **Reset Control**

N01 G00 Z0.05
 N02 X0.0001 Y0.1466
 N03 G01 Z-0.02F12
 N04 X-0.0022 Y0.1522
 N05 **T2**
 N06 X-0.0031 Y0.1582
 N07 X-0.0035 Y0.1641
 N08 X-0.0036 Y0.1698
 N09 X-0.0035 Y0.1754
 N10 **T3**
 N11 X-0.003 Y0.181
 N12 X-0.0024 Y0.1865
 N13 X-0.0017 Y0.192
 N14 X-0.0009 Y0.1975
 N15 **T4**
 N16 X-0.0001 Y0.203
 N17 X0.0006 Y0.2087
 N18 X0.0012 Y0.2144
 N19 X0.0016 Y0.2203
 N20 X0.0017 Y0.2264
 N21 X0.0016 Y0.2326

Tools

CAUTION! Quick Tool Offset Return Set

Reset	Height	Dia.	RPM
T1	0.0	0.0	0
T2	0.25	0.0	0
T3	-0.625	0.0	0
T4	-0.5	0.0	0
T5	0.0	0.0	0
T6	0.0	0.0	0
T7	0.0	0.0	0
T8	0.0	0.0	0
T9	0.0	0.0	0
T10	0.0	0.0	0

Cancel Save

Line:0 Edit Unlocked **ReNum** Pick Draw

Hot Keys

Relative OFF **Override** Coolant OFF

Slow **Jog Feed rate** Fast

+Y RAPID +Z

-X +X -W +W

-Y Jog Input -Z

Disable Motors

SPINDLE

Reverse

OFF

Fast

+ Override -

Slow

X:0.0000 Y:0.0000 Z:0.0000 W:0.00DG

Save Home Zero All

Go Home Zero X

Set Zero Zero Y

Set Zero Return Zero Z

Return to 0.0

RPM:0 Drivers: OFF

Maxrpm: 1200 Reset: Z offset: 0.0000

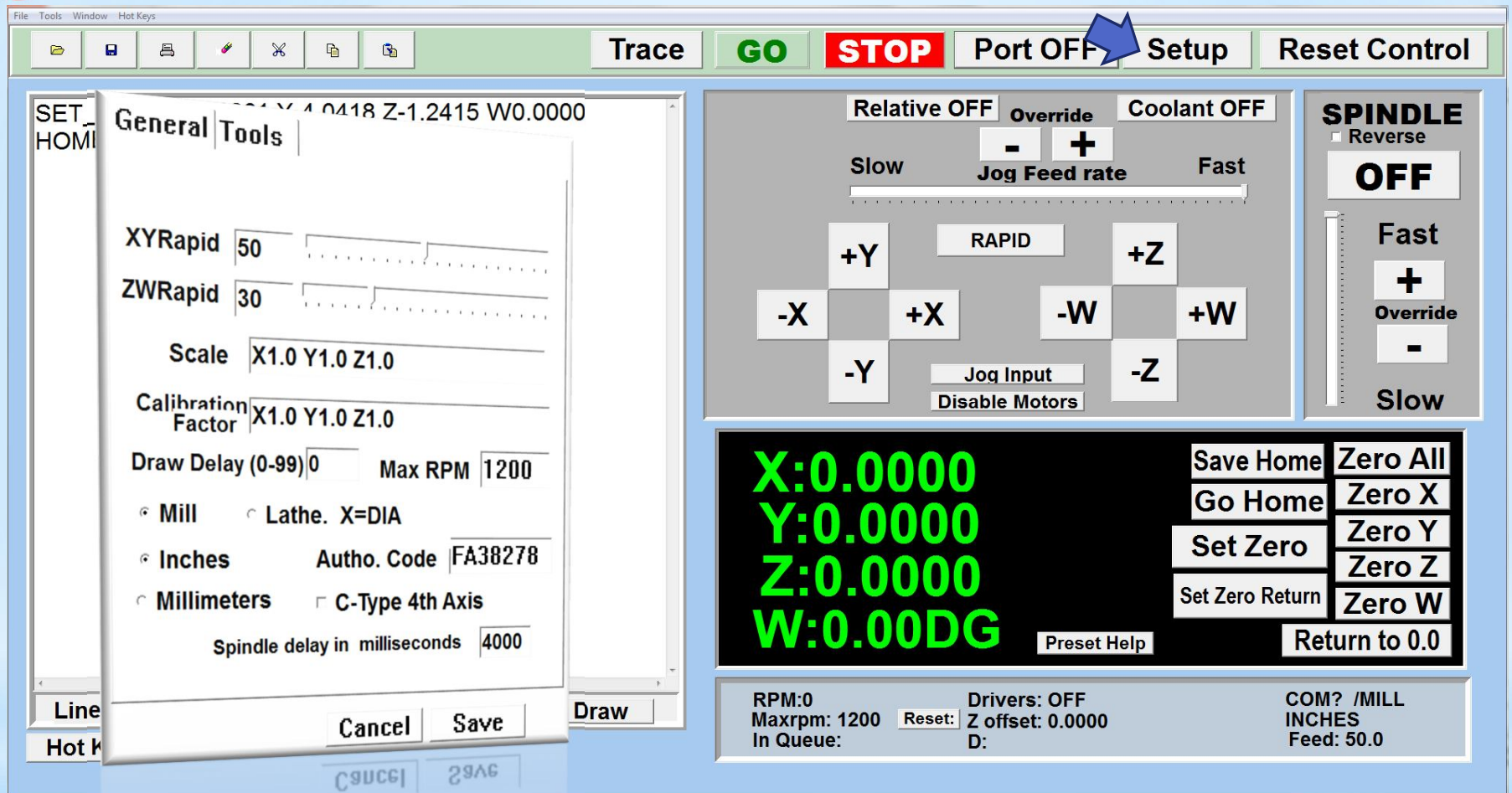
In Queue: D:

COM? /MILL INCHES Feed: 50.0

Tool Height Compensation allows for accurate height offsets when making a tool change with quick change tools within a program. Up to 10 tool changes can be made. This feature can be very effective for improved productivity if your application requires several tool changes.

Tool Radius Offsets can also be done. If you choose to use a G41/G42 for a radius tool offset, you can enter the diameter in the Tools Window under Setup, and the machine will offset the radius of the tool.

Diameter of Tool: By entering the size of the cutter in the Setup > Tools Window, you can also view the tool paths according to cutter size in the Draw window.



In the Setup window, you will find other useful features such as:

Adjust Rapid on the axes from .1" to 100"/minute of travel.

Scale feature allows you to quickly double the size or miniaturize the size of the application.

Calibration Factor allows you to fine tune an axis for enhanced repeatability.

Inch/mm Display allows you to change from inch to mm mode.

Max RPM allows you to synchronize the spindle motor belt or gear positions for computer variable spindle control.

Change the Face of the Master to Lathe Mode or Mill Mode for any of our CNC machines.



The Master Software supports these standard G-Codes and M-Codes for our CNC Mills:

G00 = Position (Fast speed)
G01 = Linear interpolation (Feed speed)
G02 = Circular interpolation (CW)
G03 = Circular Counter-clockwise interpolation (CCW)
Format: X__Y__I__J__ I,J are relative distance from start to center. Z can be added for helical designs.
G17 = Cancels G60 Command
G40 = Cancels G41 and/or G42
G41 = Tool Radius compensation left
G42 = Tool Radius compensation right
G60 = Switch data from Y to W * radius factor
G70 = Input in inches
G71 = Input in millimeters
G73 = High-Speed Peck Drilling Cycle
G81 = Drilling Cycle
G82 = Counter Boring Cycle
G80 = Cancel Cycle
G83 = Deep Hole Peck Drilling Cycle
G90 = Absolute move (Modal)
G91 = Relative/Incremental move (Modal)

M00 = Pause
M03 = Spindle on
M04 = Spindle on reverse
M05 = Spindle off
M08 = Coolant on
M09 = Coolant off
M30 = End program

The Master Software supports these standard G-Codes and M-Codes for our CNC Lathes:

G00 = Position (Fast speed/Rapid)
G01 = Linear interpolation (Feed speed)
G02 = Circular interpolation (CW)
G03 = Counter Clockwise Circular interpolation (CCW)
Format: X__Z__I__K__ I,K are relative distance from start to center.

G04 = Dwell time
G20 = Input in inches
G21 = Input in millimeters
G71 = Turning Cycle
G72 = Facing Cycle
G74 = Peck Drilling
G76 = Threading cycle
G90 = Absolute move (Modal)
G91 = Incremental move (Modal)

M03 = Spindle on
M04 = Spindle on reverse
M05 = Spindle off
M08 = Coolant on
M09 = Coolant off
M30 = End program



Our Pledge to you...

When you do business with CNC Masters, you will be doing business with a company who cares about its customers. We take our after sales tech support very seriously by phone or email for as long as your company owns a CNC Masters machine purchased directly from us. We understand that your company has invested in CNC Masters to help you along the way. It does not matter how many years have passed, we will be happy to assist you from guidance, trouble shooting on the mechanics, electronics, to explaining the functions on the Master Software. We are honored you have chosen CNC Masters over other brands in the market, so it will remain our goal to help you as needed so you can put your product out in the market as quickly as possible.

Master Software updates will also be made available to you for as long as the hardware in the control unit can sustain it. Thank you for watching, and be sure to email us with any questions you may have at sales@cncmasters.com.