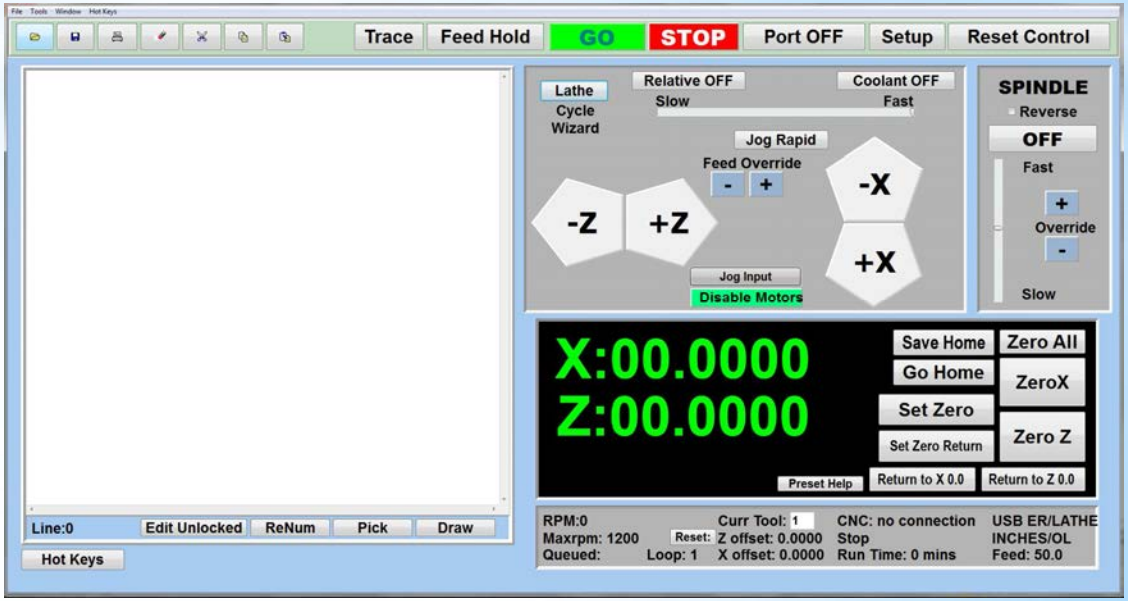
CNC MASTERS

The MX Software

Use it on your CNC Mill...

Designed to seamlessly work with your CNC Masters machine.



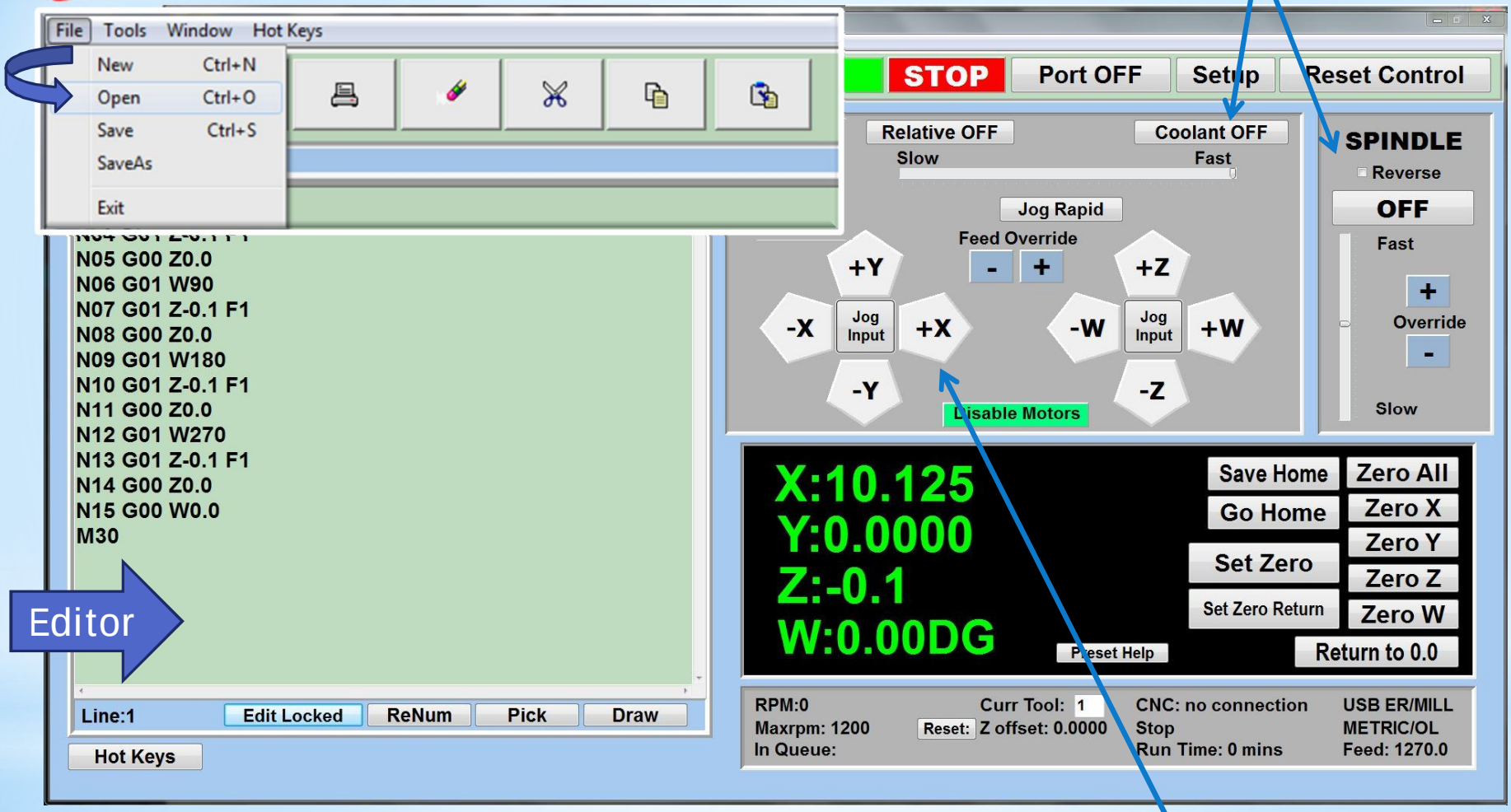
Use it on your CNC Lathe...

CNC MASTERS

* **The Master MX Software** is a PC based application system designed to work with 64 bit operating systems: Windows 10, 8, or 7 through your standard USB port interface on lap tops or desk tops. The interface is designed to used with touch screen monitors too.

CNC MASTERS

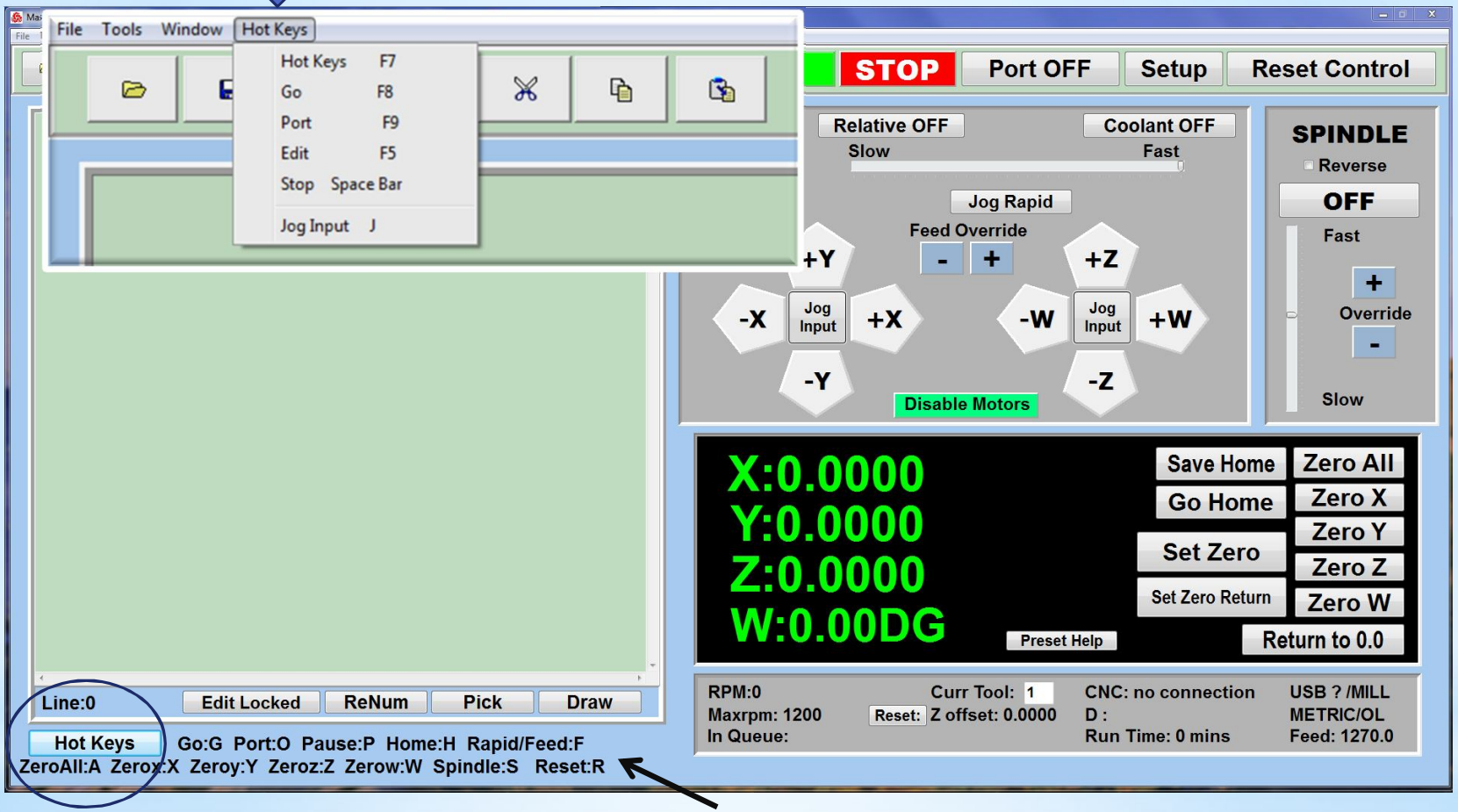
You also have the optional coolant and spindle speed controls.



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.

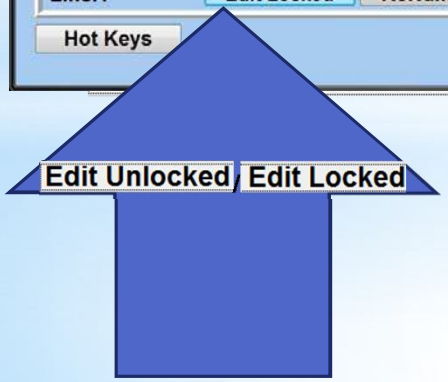
The X Y Z and W arrow driving buttons are displayed from the point of view of the cutter to avoid confusion when the table and saddle moving.

CNC MASTERS



Hot Keys is an alternative method to easily control your machine using your hard or touch screen keyboard. One 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, J to record your individual movements one line at a time to create a program by teach mode.

CNC MASTERS



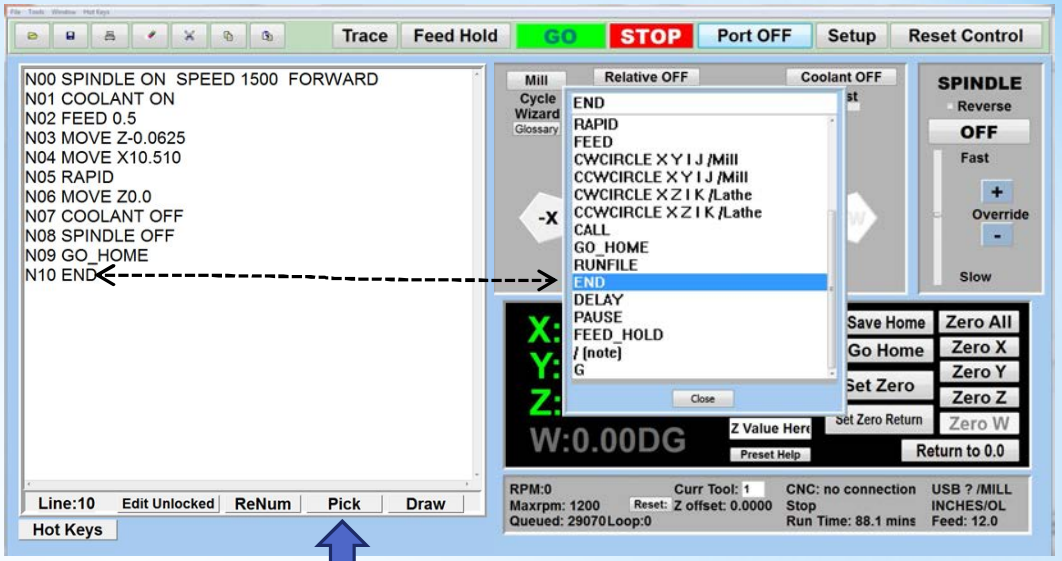
The Master defaults to an “Edit Locked” green editor screen. To create or edit your tool path program, you will have to “Edit Unlock” first which will turn the screen to a white color. This Edit Locked feature keeps your tool path program from being accidently changed.

CNC MASTERS

While in “Edit Unlocked” 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 such as what is written in the Editor box. You can even mix between conversation commands and G-codes in the same program.

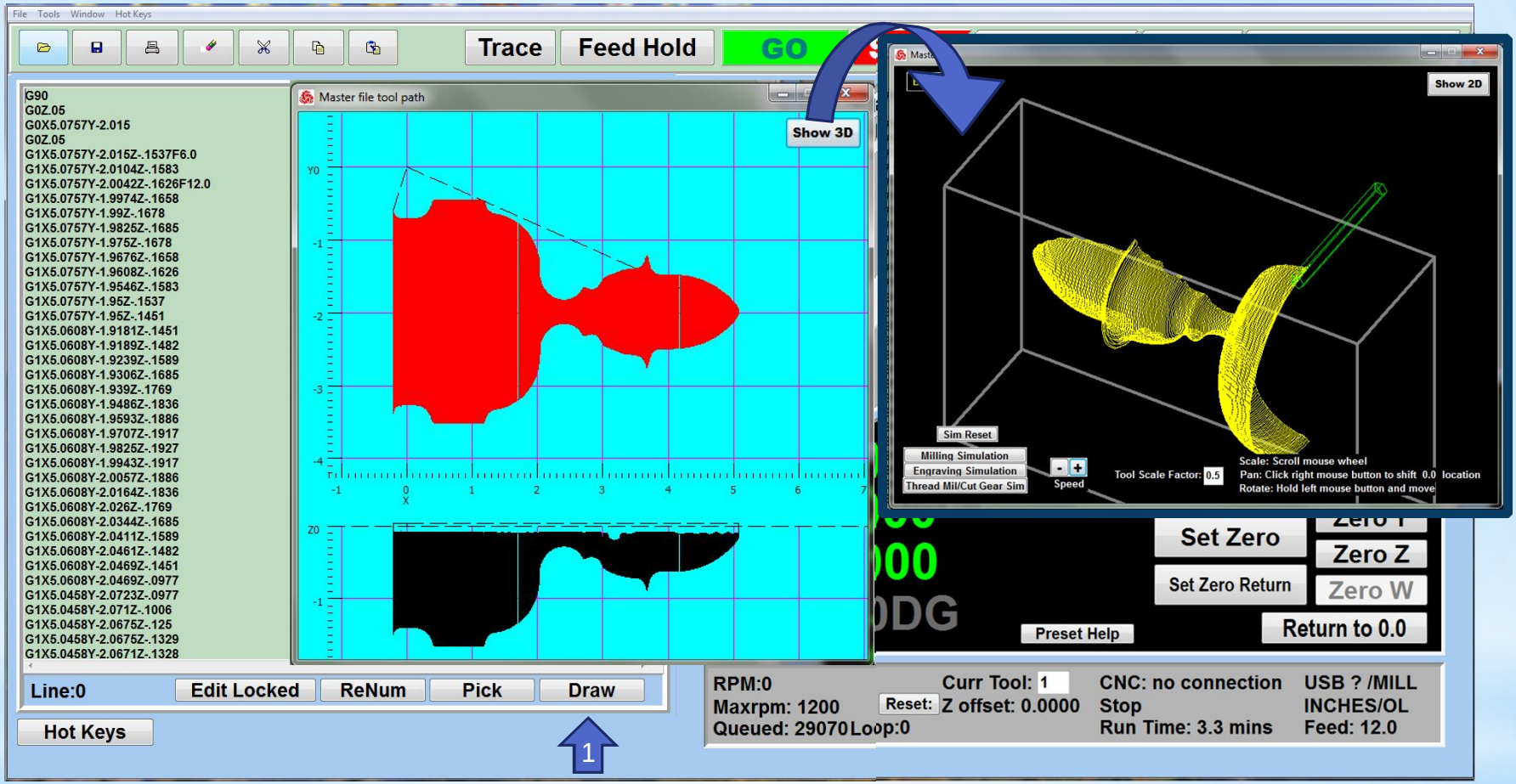
CNC MASTERS

With the Pick Menu, you can easily write a simple program without G-codes. You 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.



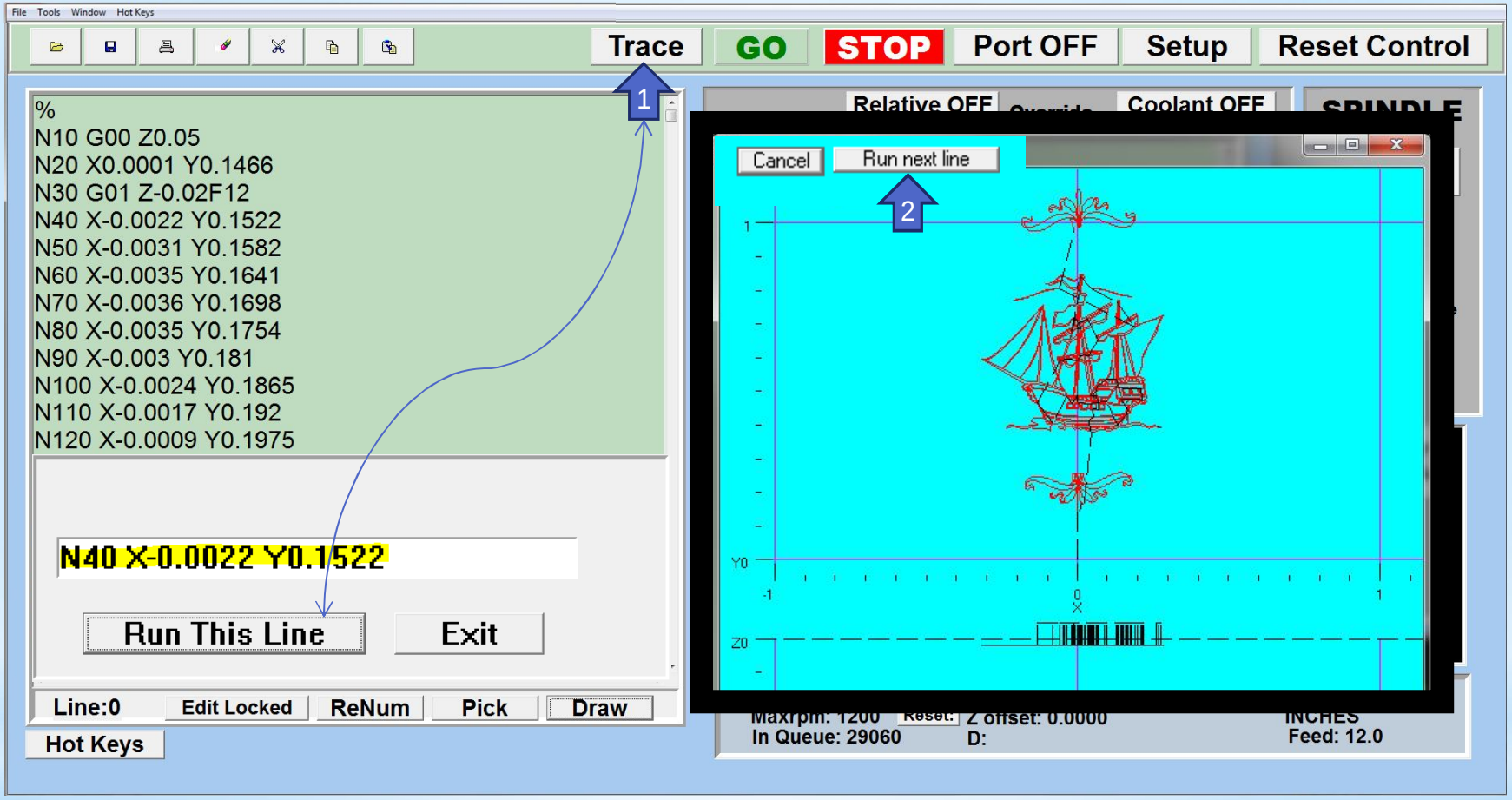
Pick Menu Features	Description
ABSOLUTE	Same as G90. The Master defaults to Absolute mode.
INCREMENTAL	Same as G91.
SPINDLE ON	Same as M03 for computer variable spindle control.
MOVE	Same as G00, G01. Move three or four axes simulataneoulsy on the same tool path.
PRE_SET	Can't start from a 0.00 position? 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.
RAPID	Quickly retract or relocate the tool after an operation above the application. Rapid 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 in the 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.

CNC MASTERS



Hit Draw to view your tool path program drawing, or even simulate the tool path in 3D mode. This can be helpful to quickly verify your program before running it. You can also slow down or speed up the drawing or simulation process.

CNC MASTERS



1. Run the machine on Trace mode. You can run each tool path independently, one line at a time to study the tool path movement on the machine to verify the position of the application and if any fixture/vise is in the way of the cutter's path.

2. You can also verify your program by clicking on the Trace and Draw buttons together. This will allow you to view each tool path independently in the Draw Window.

CNC MASTERS

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. The counters also display a "real-time" readout while the machine is in cnc operation without counting ahead of the movement.

2. Under Tools, run a "Go From Line" command if you need to begin in the middle program instead of starting from the first line.

3. Press Go to run your program or Stop to kill all CNC movement on the machine.

CNC MASTERS

1. Easily de-energize the axis motors by clicking Disable Motors to crank each axis by hand, and then press Reset Control to re-energize the axis motors.

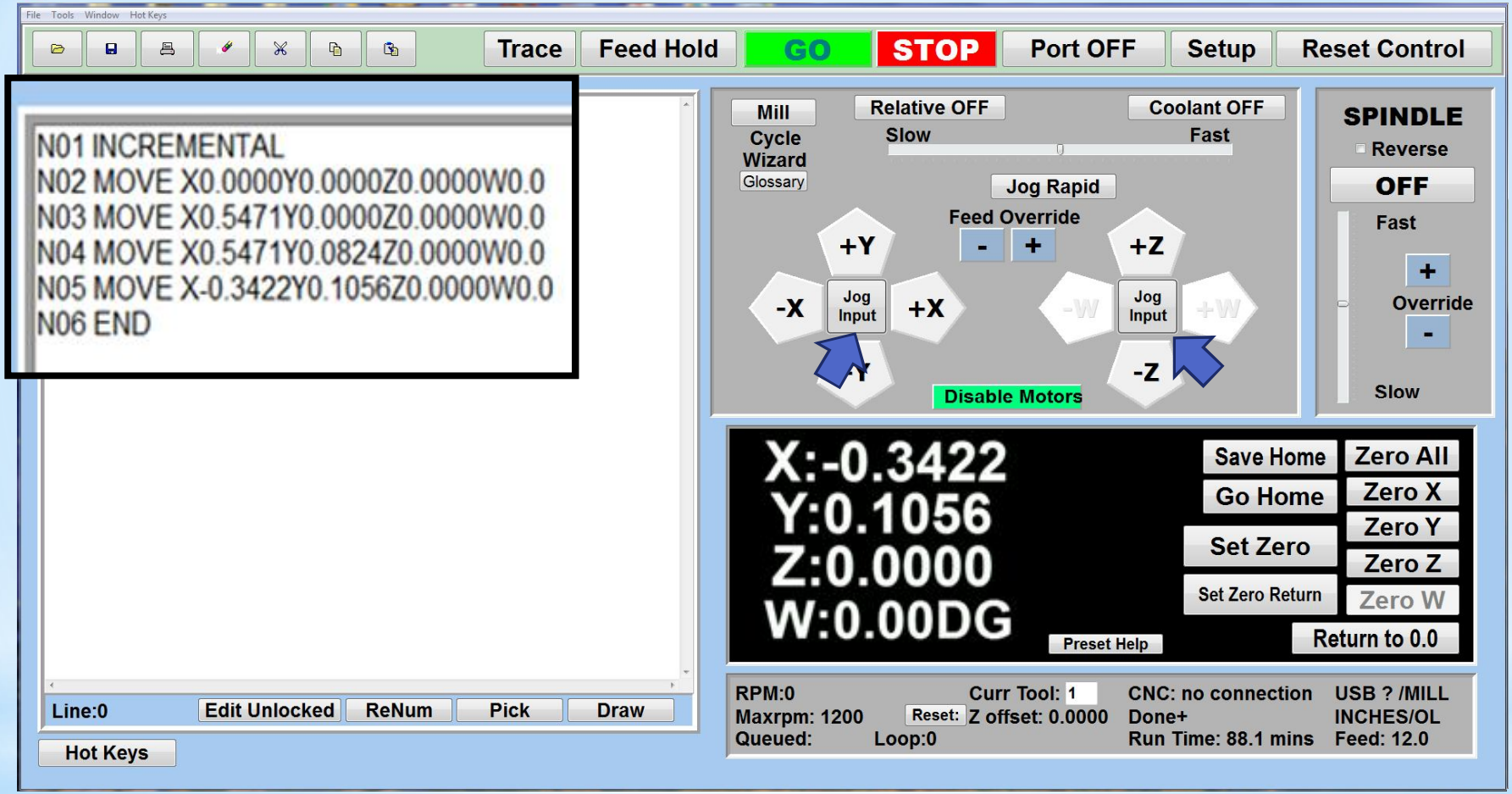
2. Feed Hold in the middle of a program, and step through the program one line at a time.

CNC MASTERS

Use Relative ON to enter a specific coordinate 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 once the –Z button first to drill the hole exactly 0.525". Then click once on the +Z button to drive the axis back up 0.525".

CNC MASTERS



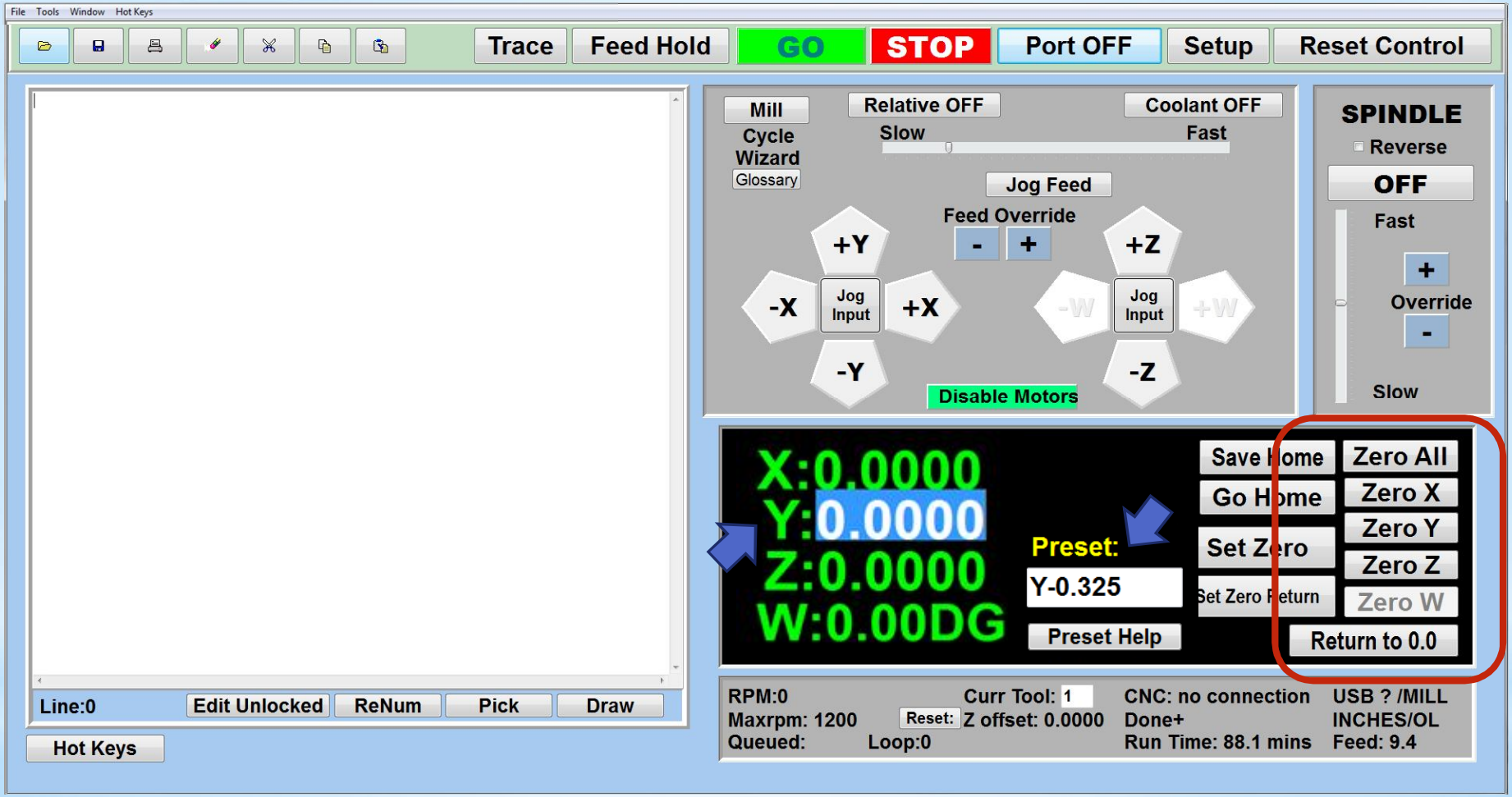
Jog Input – Teach Mode:
You can create a tool path program by storing each point-to-point movement by simple jogging an axis. Click on either of the Jog Input buttons to store each movement on the Editor Screen. You can then add Spindle ON, feed commands, and press GO to run the new program as needed. 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.

CNC MASTERS

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 RPM to the max RPM according to the machine. The [-] [+] allows you to fine tune RPMs in 10% increments while the program is in motion.

CNC MASTERS



1. Pre-set directly into the counters by typing in your beginning coordinate if you cannot start your program at 0.00.

2. The counters X Y Z can display in inches or mm. You can make this change in the Setup or by G-code. 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.

CNC MASTERS

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 its saved "set" 0.00 position according to the recorded coordinates at the first line of the program.

CNC MASTERS

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 offset 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.

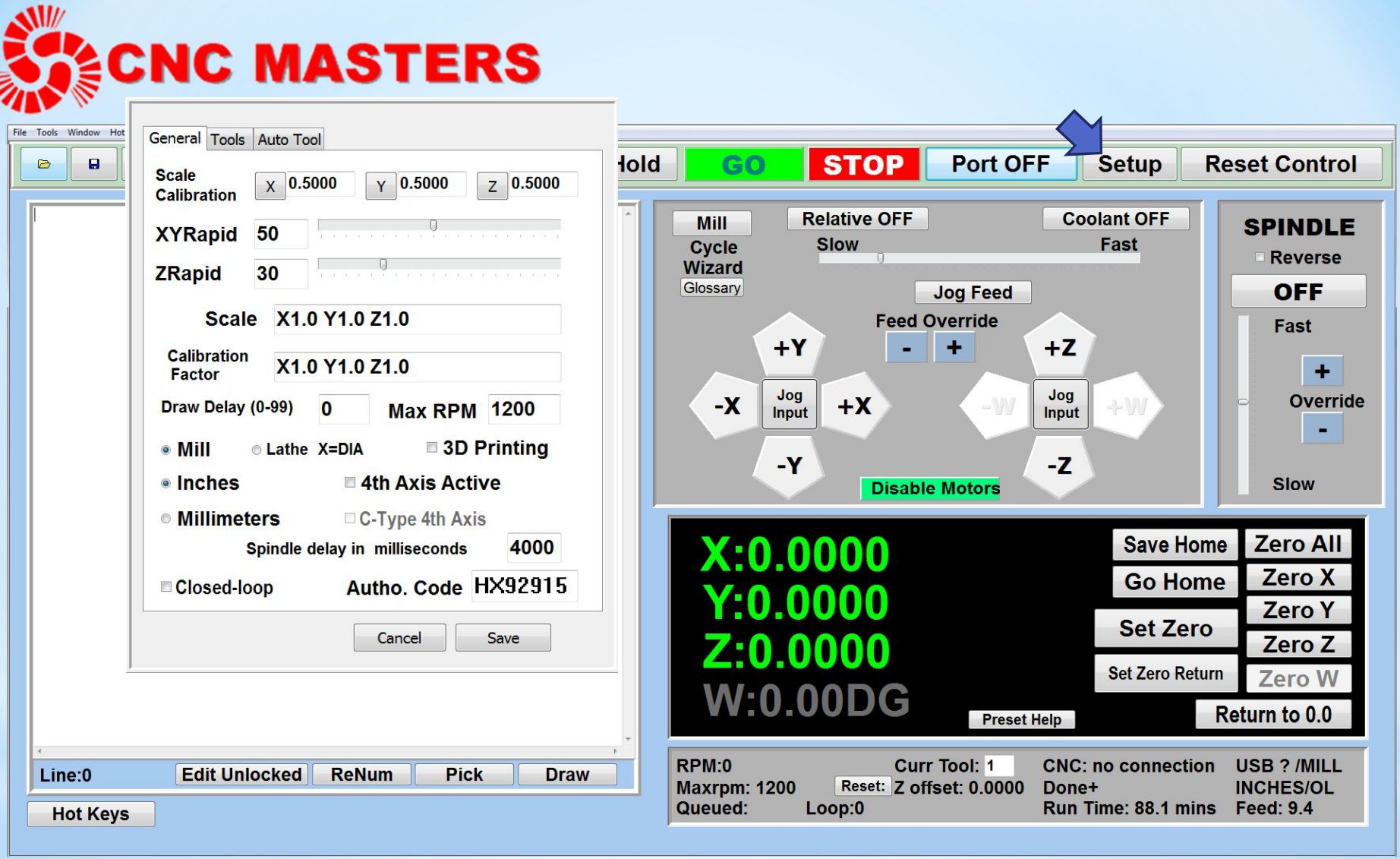
Pressing Go Home will jog your axes back to your “saved home” position where you originally pressed the Save Home command. You can also input GO_HOME from the Pick Menu as its own tool path in your program. So at the completion of your program it will end the axes at your Home position. Replace your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

CNC MASTERS

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.

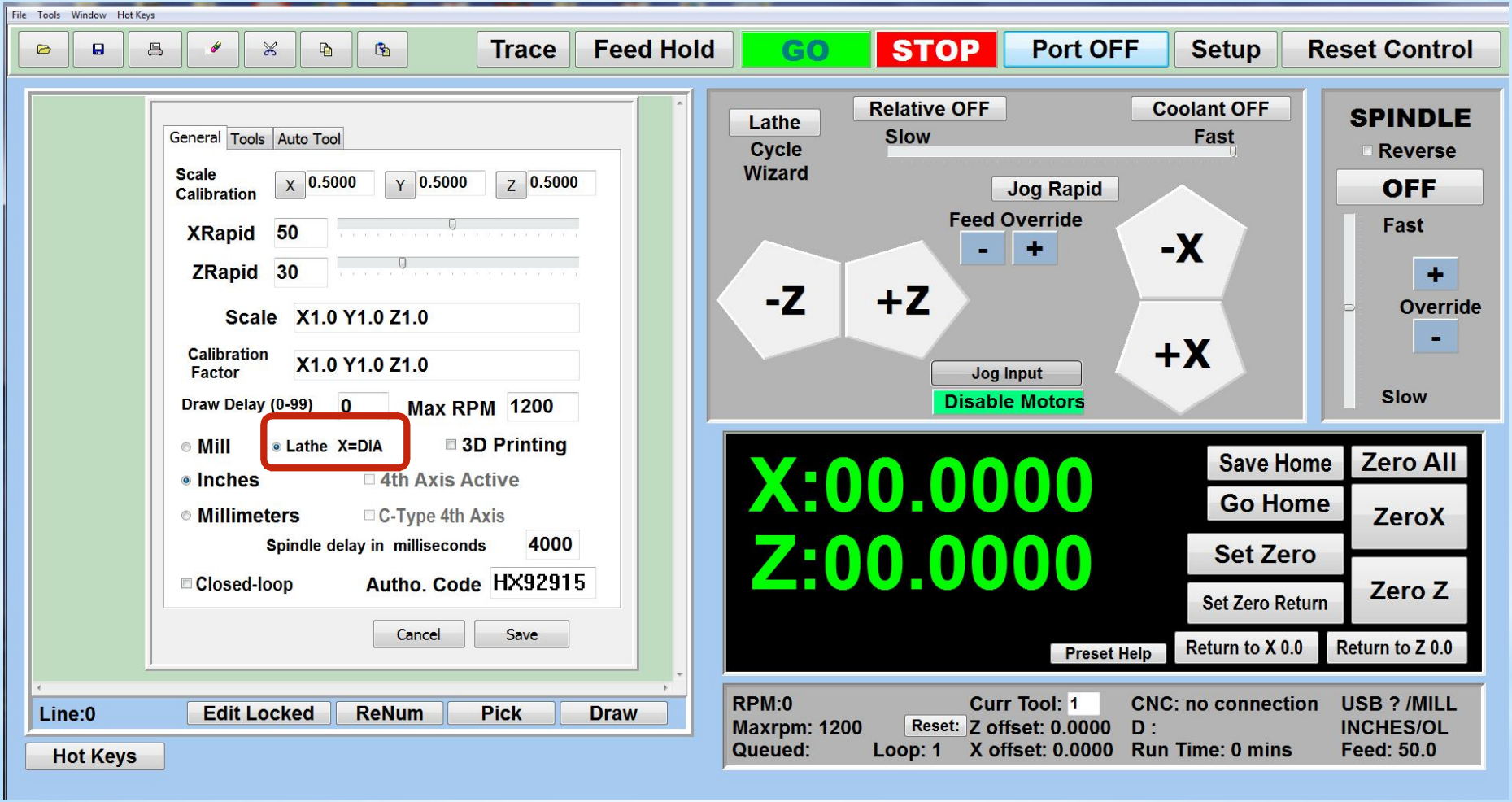
Diameter of Tool: By entering the size of the cutter in the Setup > Tools Window, you can also view the tool paths according to cutter size in the Draw window.



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

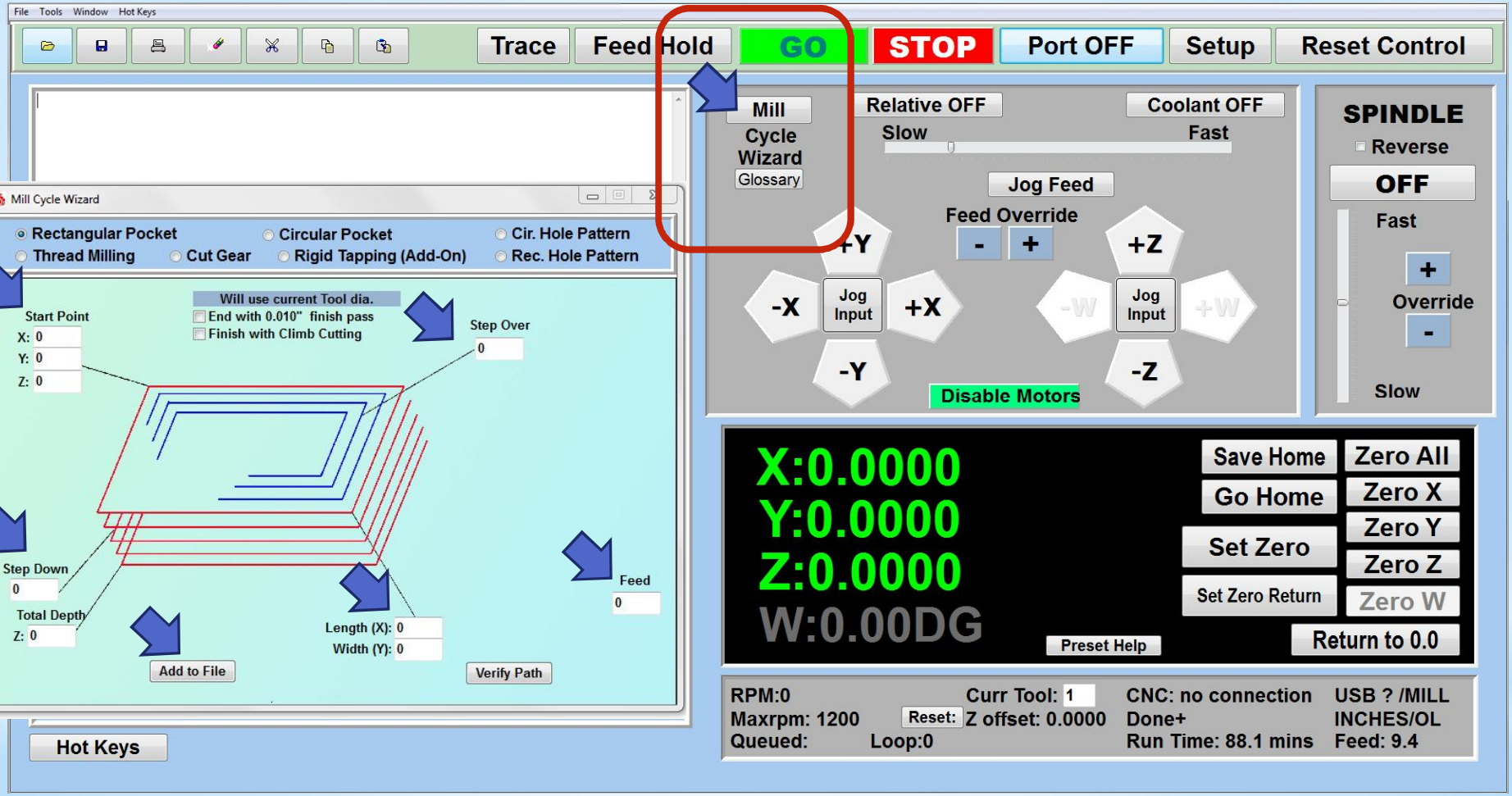
Adjust Rapid on the axes from .1" to 100"/minute of travel.
Scale feature allows you to quickly double the size or miniaturize the size of the application.
Calibration Factor allows you to fine tune an axis for enhanced repeatability.
Inch/mm Display allows you to change from inch to mm mode.
Max RPM 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.

CNC MASTERS



The Master MX can easily be switched from a mill to lathe operations interface.

CNC MASTERS

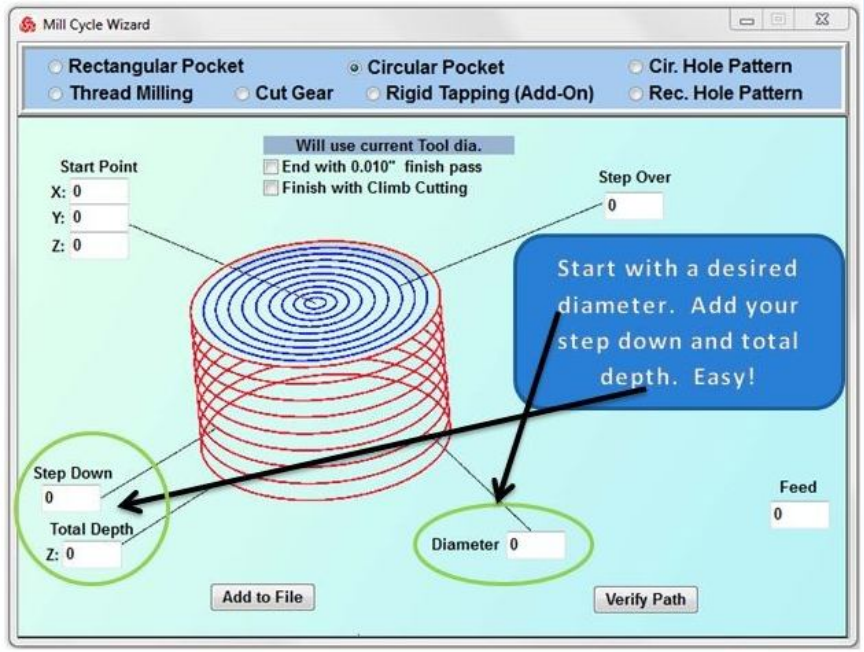


The Cycle Wizards makes it easy to create a simple tool path without needing to use a CAD and CAM software.

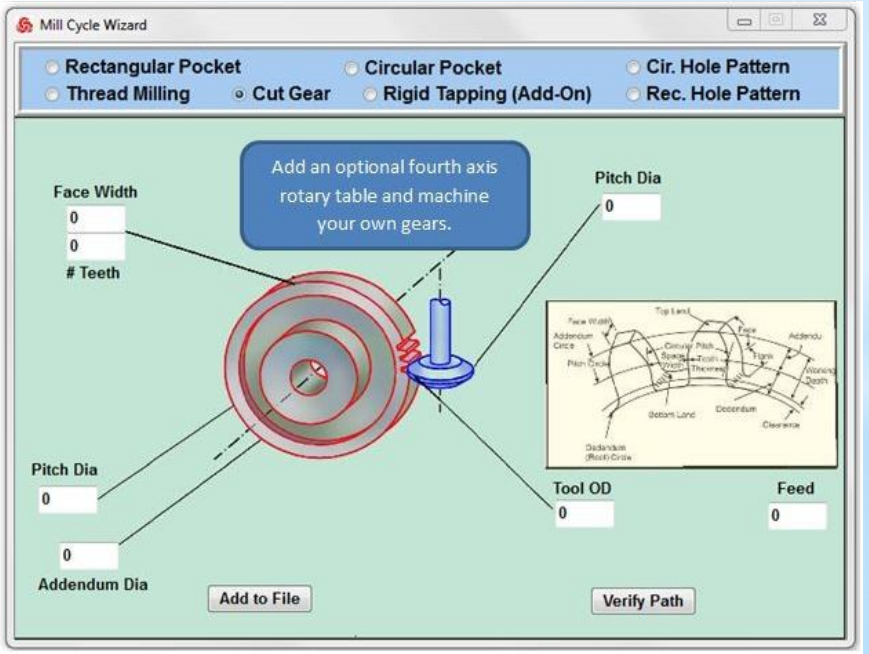
Create circle patterns and profiles, slots, rectangular pocketing, thread milling, gear cutting, rigid tapping (with optional encoder kit) and peck drilling applications programs with a few key entries.

CNC MASTERS

Use the Mill Wizards to do...



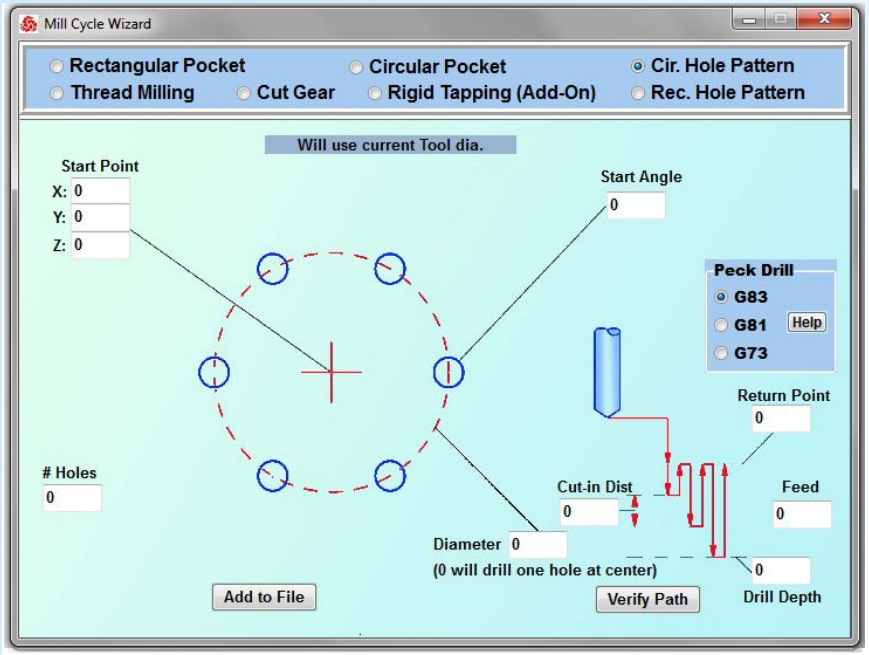
Circle Pocketing



Gear Cutting

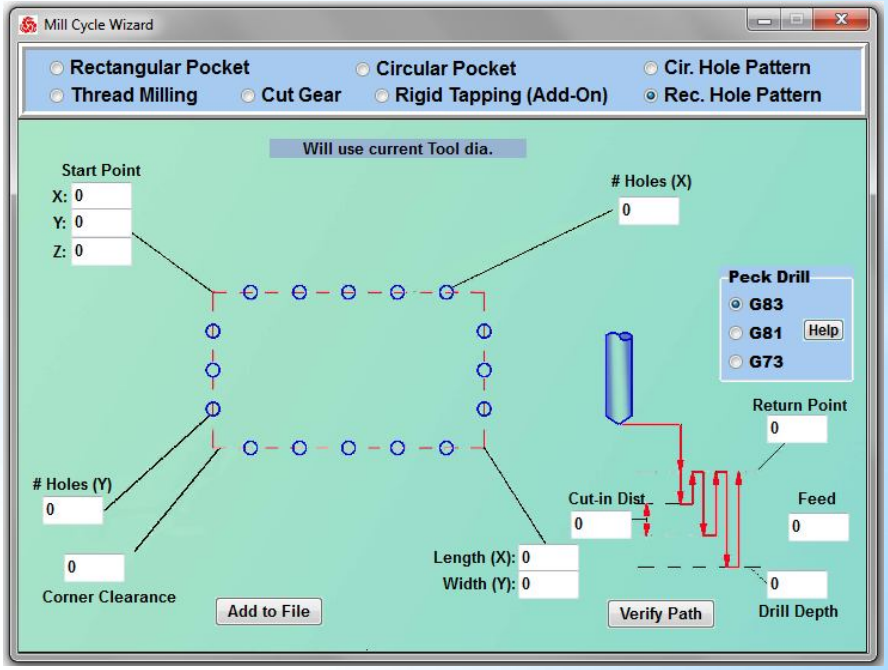
CNC MASTERS

Use the Mill Wizards to do...



Circular Hole Drilling Pattern

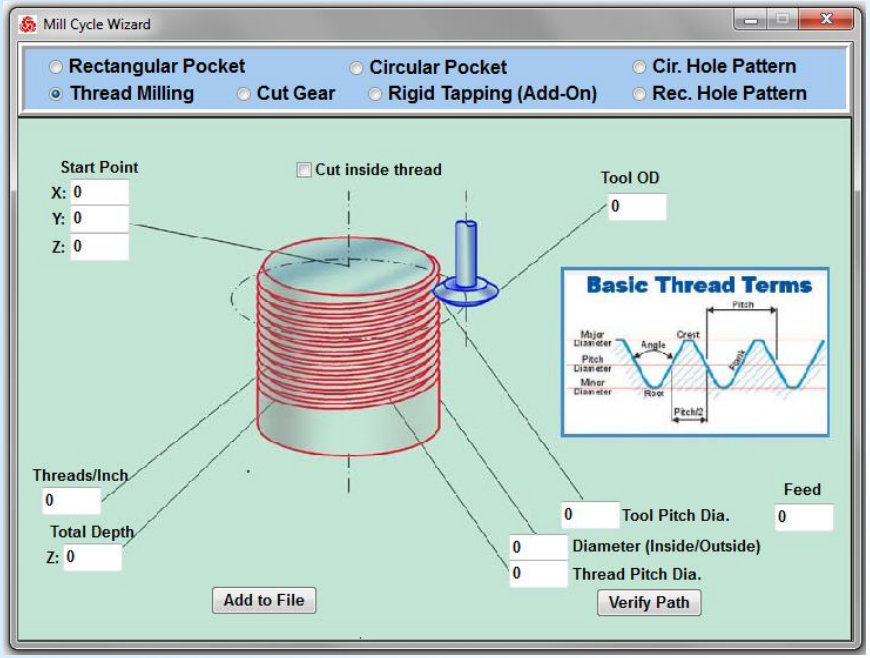
Rectangular Hole Drilling Pattern



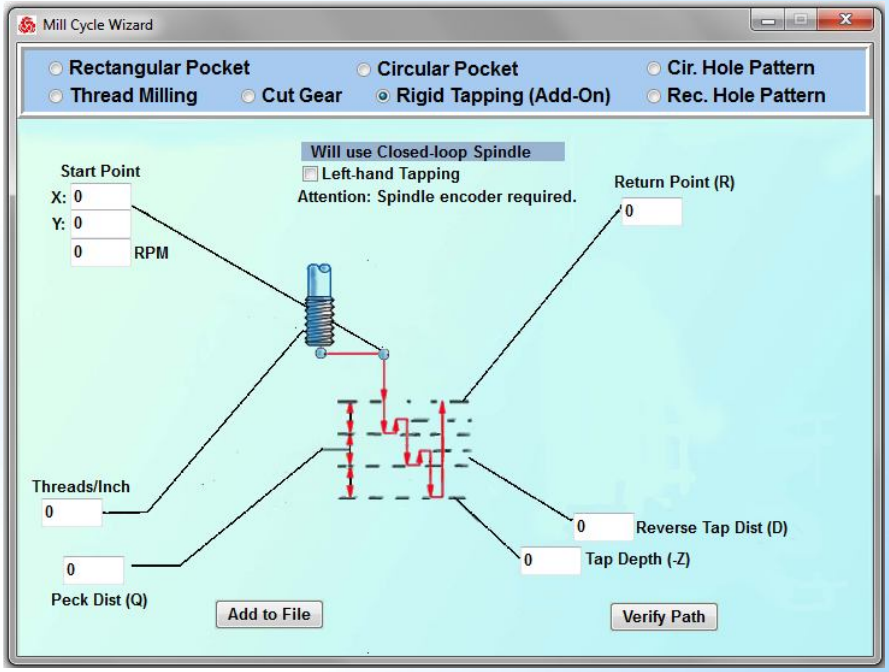
CNC MASTERS

Use the Mill Wizards to do...

Thread Milling

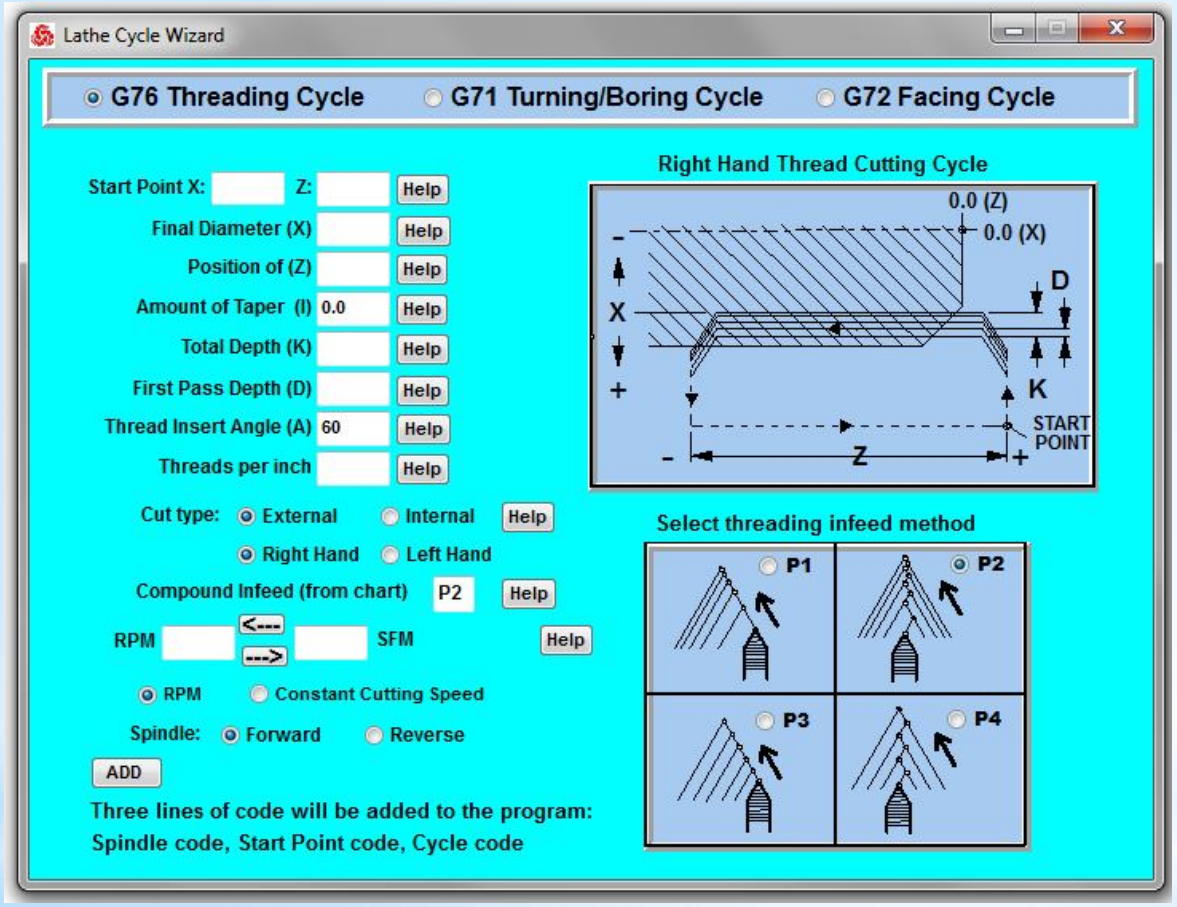


Rigid Tapping



CNC MASTERS

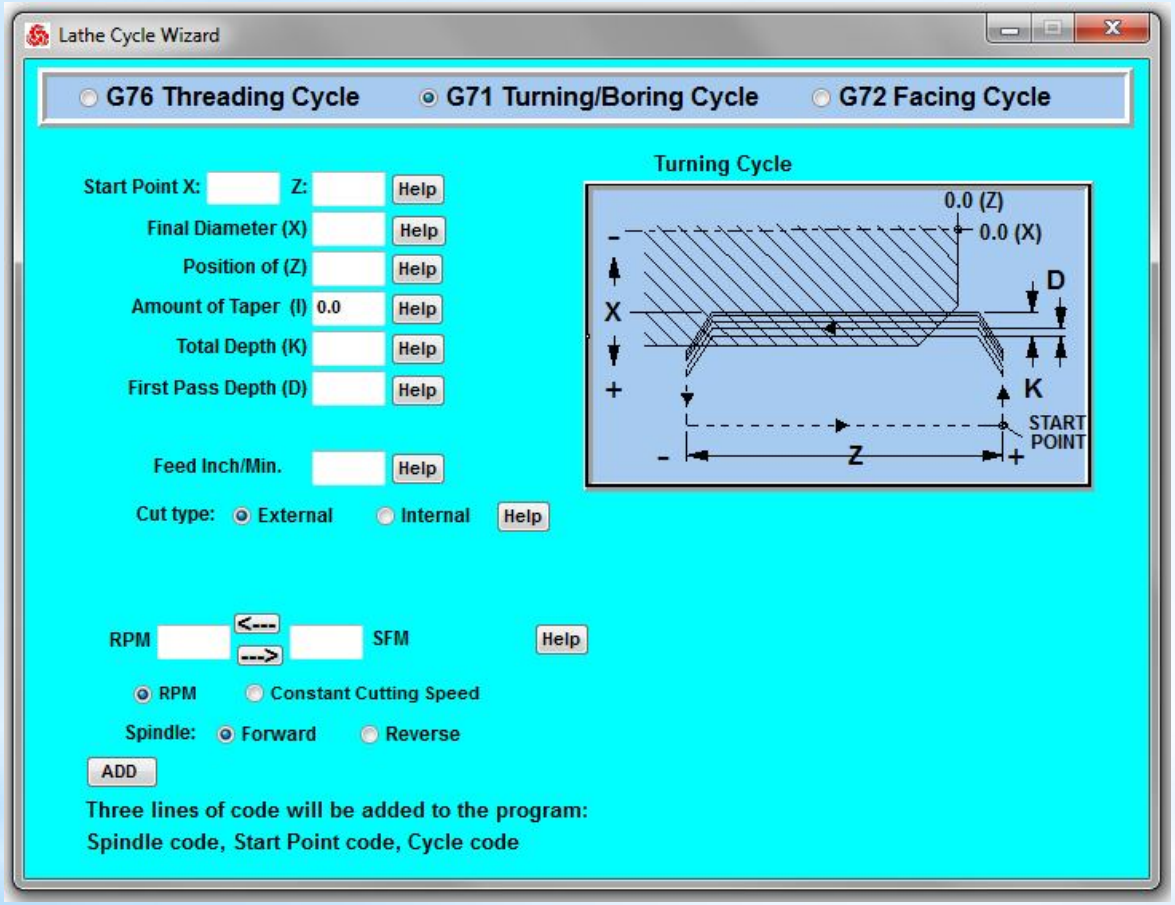
Use the Lathe Wizards to do…



Threading Cycles

CNC MASTERS

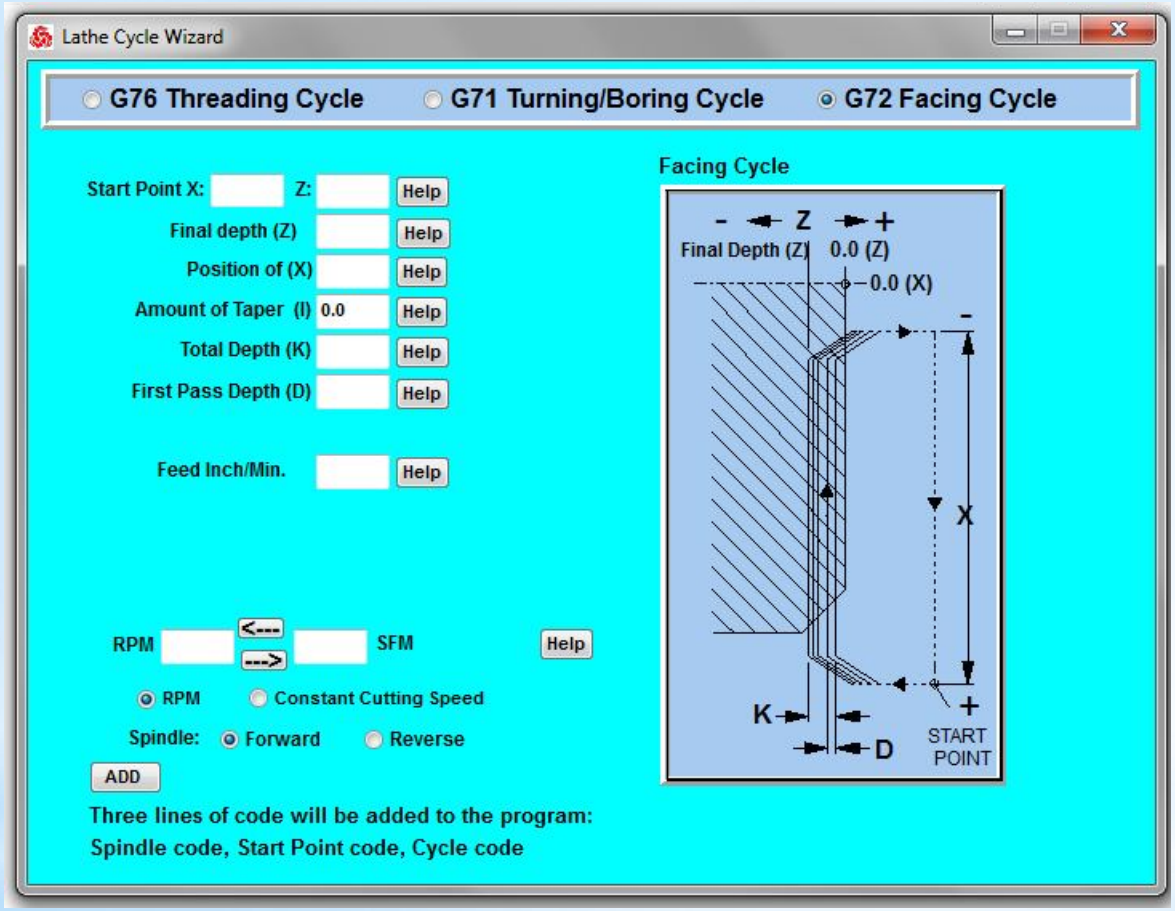
Use the Lathe Wizards to do…



Turning & Boring Cycles

CNC MASTERS

Use the Lathe Wizards to do…



Facing Cycles

CNC MASTERS

In summary, the MX supports….

•Tool Height/Length Compensation up to Ten Tools within one program
•Tool Radius Offsets
•Feed Hold – pause and step through your program while opting to shut the spindle off and then resume program
•Coolant Control – optional
•Variable Spindle Control from 0 to max – optional
•Spindle Encoder for rigid tapping – optional
•Jog Feed/Teach Mode – create a program simply by jogging your axes
•Feed and Spindle Speed Over-Rides on the fly
•Relative exact movement positioning without writing a program — type in one movement and one-click your jog +/- arrow to drive that exact movement
•Press Go from another location other than 0.00 – just preset the new coordinates directly into the counters
•Save 0.00 *and* a home offset position for future program production runs
•Live Counter Display during computer numerical control movement without jumping ahead
•Displays and runs in either inches or millimeters
•4th axis interpolation – simultaneous motion with the other axes – optional
•Rapids up to 100"/minute
•Trace Mode – Run one line independently at a time from beginning to end in your program to help you study the movement and establish your setup
•Editor Locked/Unlocked to easily write and edit your program and prevent accidental typing during a program run
•Displays Run Time
•Hot Keys – if your preference is to control your machine by keyboard such as the arrows, space bar, and letters simply open the Hot Keys command
•Run Sub-Routine programs using CALL for nesting applications or to mass produce the same part on a constant loop.
•Disable Motors — Easily disable motors to hand crank each axis. Re-engage the motors for cnc control in one click.
•Use a touch screen monitor or control your machine with your pc mouse all by true USB.

The MX is designed to be seamless….none of this having to pre-engineer or tune the software to work with your machine. Just plug in the machine to a pc and play!

CNC MASTERS

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.
G40 = Cancels G41 and/or G42
G41 = Tool Radius compensation left
G42 = Tool Radius compensation right
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

The MX is a free download. Try it today.

CNC MASTERS

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 by guiding you over the phone or by email step by step if needed. We will help you trouble shoot the the mechanics, electronics, and explain 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 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.