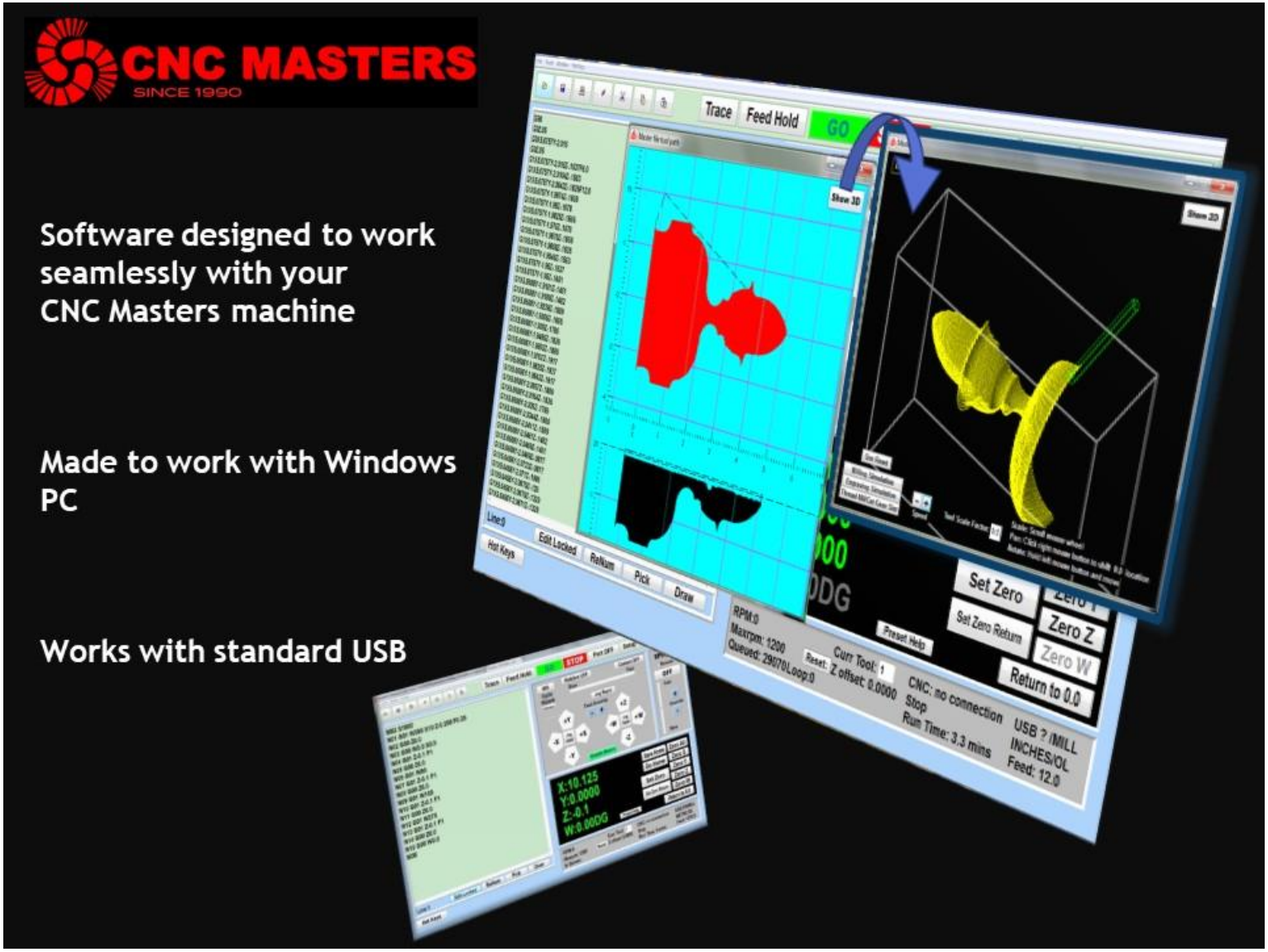
MX Software – Easy to Use, Easy to Learn – Included with your machine purchase

The MX software is designed to work seamlessly with your CNC Masters machine. It is made to work with Windows PC – desktop, laptop, or an all in one – on standard USB. Use it on Windows 64-bit operating systems. No internal conversion printer/serial port to USB software or additional conversion hardware is used with the MX.

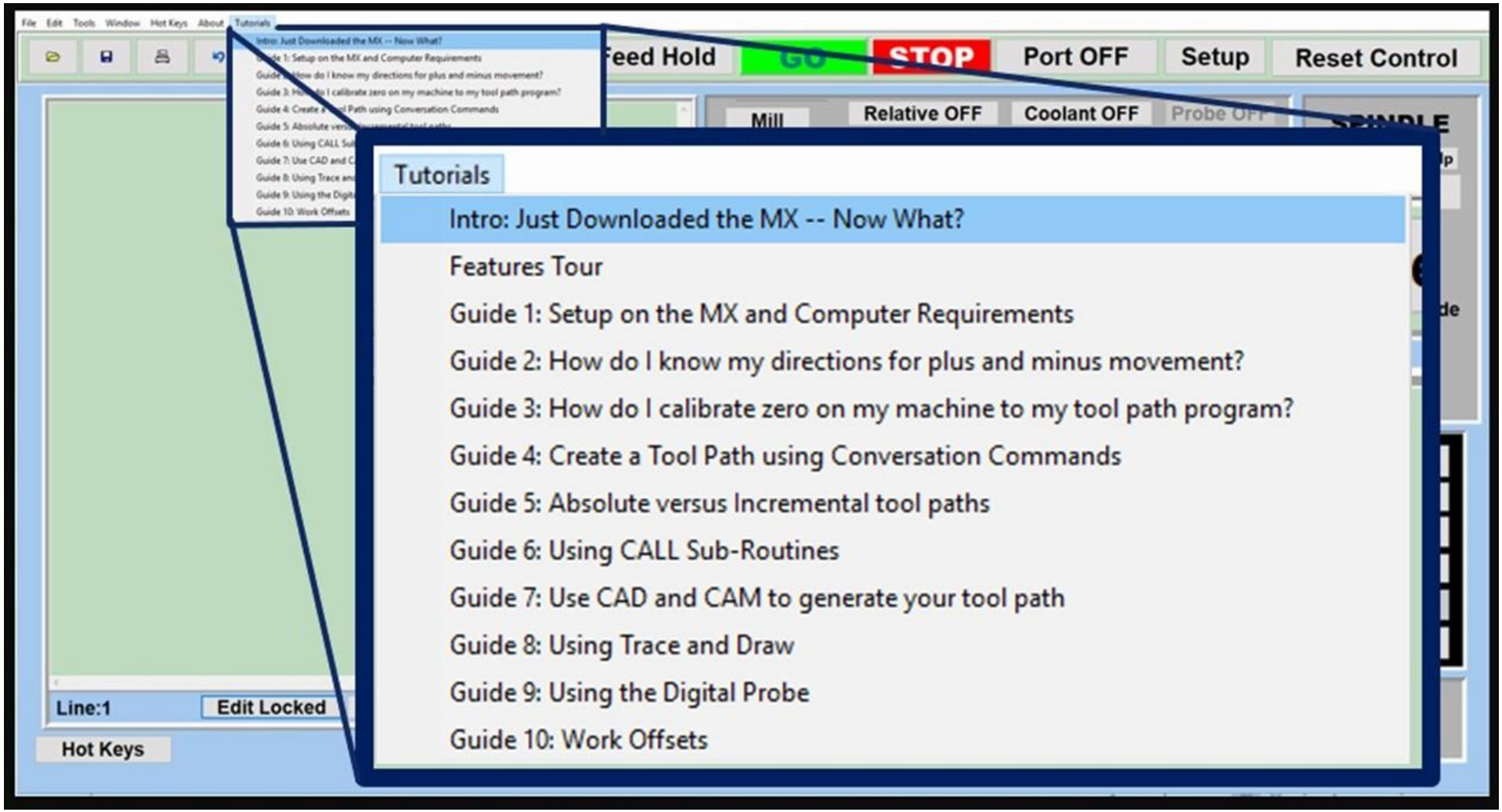


Clutter Free Interface

The MX is engineered for the CNC MASTERS machine so you do not have to fiddle with a detailed complicated configuration that can be overwhelming. Just load in the MX and start machining!

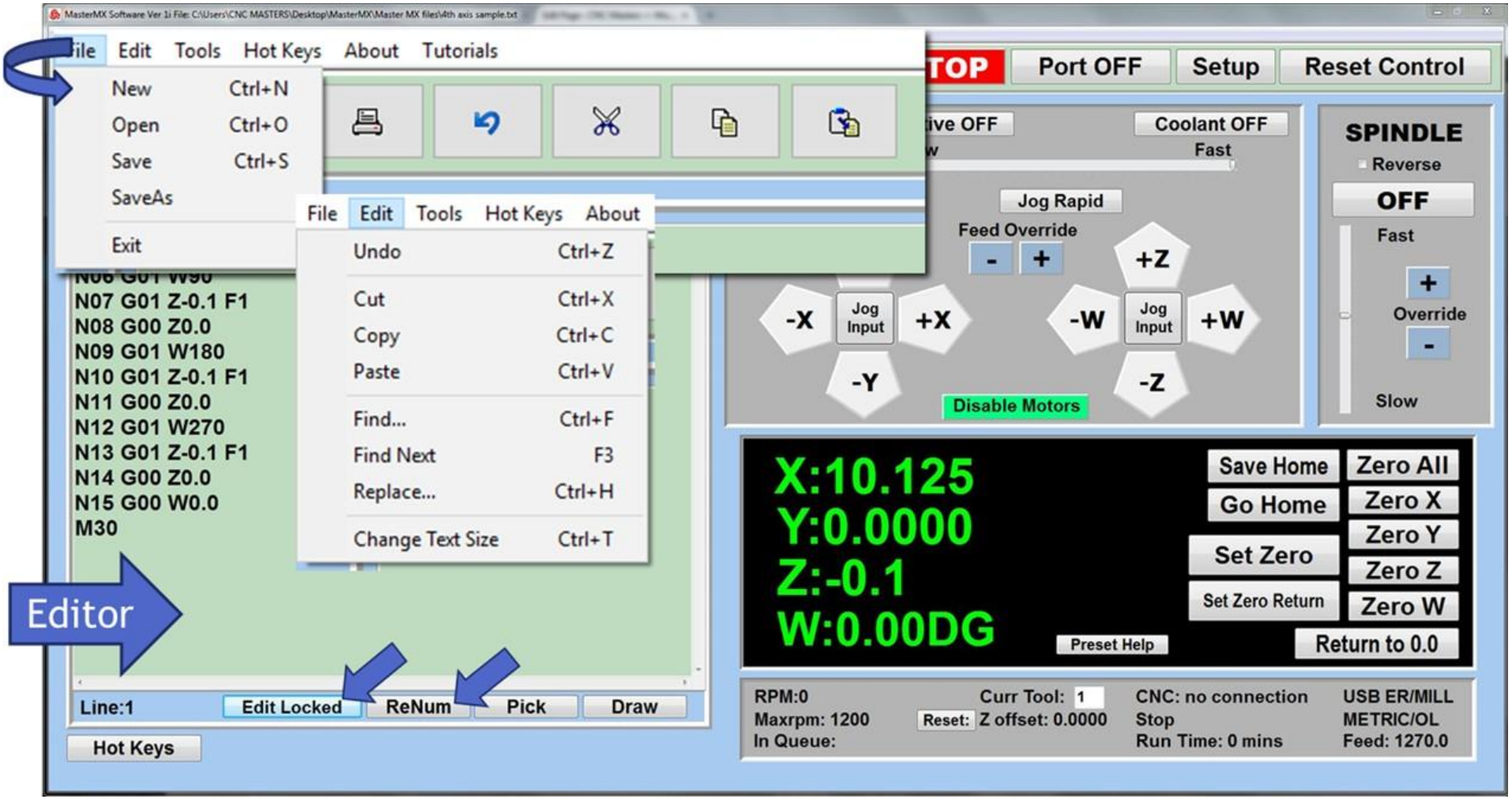
Features Tour and Tutorials Included

The Features Tour will give you a quick run-down on all the features the MX can do for you. The Tutorials are easy to follow even for the first time CNC machinist. Feel free to download the MX on any of your computers. We recommend downloading the MX along with your CAD and CAM software there at the comfort of your office computer to generate your tool path programs. You don't need to be hooked up to the machine either to test your program in simulation mode.



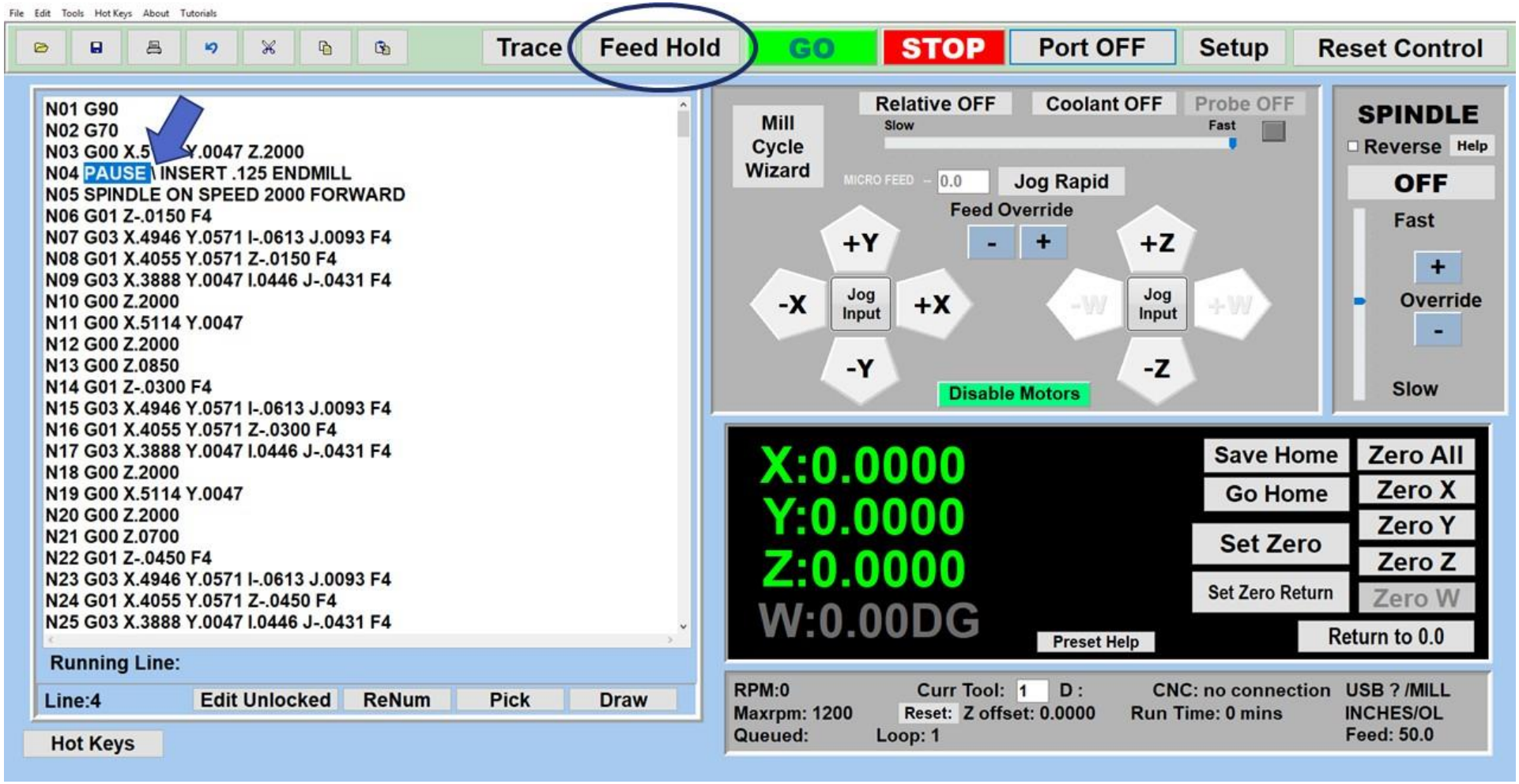
Navigate and Edit Your Program through the MX interface with Ease

With a few clicks of the mouse or using touch screen technology, you can easily navigate through the MX interface importing saved programs into the Editor from the File drop down menu. Using standard windows features to edit your program you can then lock the Editor Screen to avoid accidental editing, and if you need to insert a line in the middle of a program, just click on [ReNum] to re-number your tool path list. You can create a program or import CAM generated G-code tool paths into the Editor. The X Y and Z W arrow jog buttons are displayed from the point of view of the cutter to avoid confusion when the table and saddle are moving. You can also adjust your spindle speed and coolant control while jogging each axis.



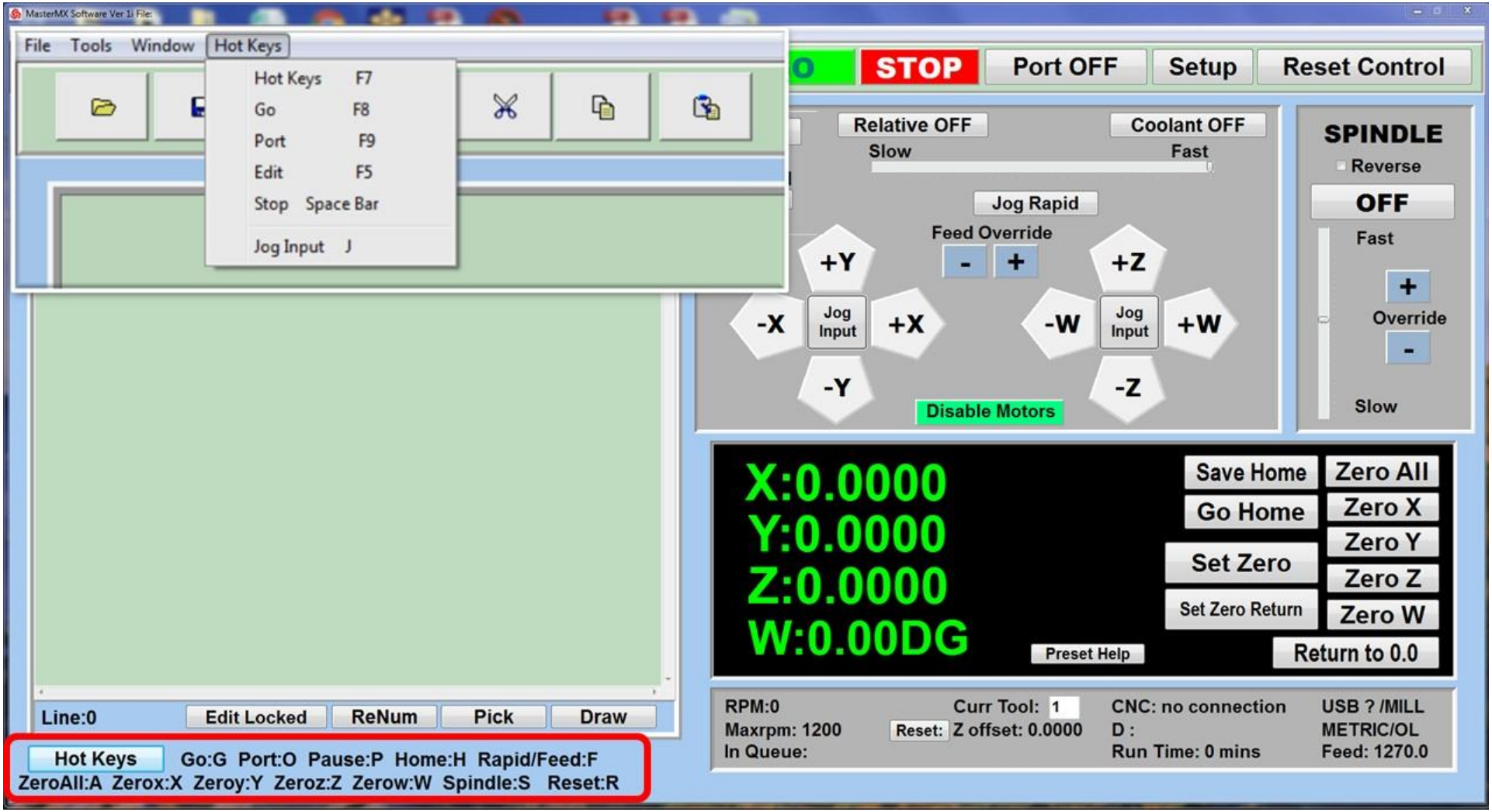
Feed Hold – Pause in the Middle of your Program

Feed Hold lets you pause in the middle of a program. From there you can step through your program one line at time while opting to shut the spindle off and then resume your program. You can also write PAUSE in the middle of your program and jog each axis independently while your program is in pause mode.



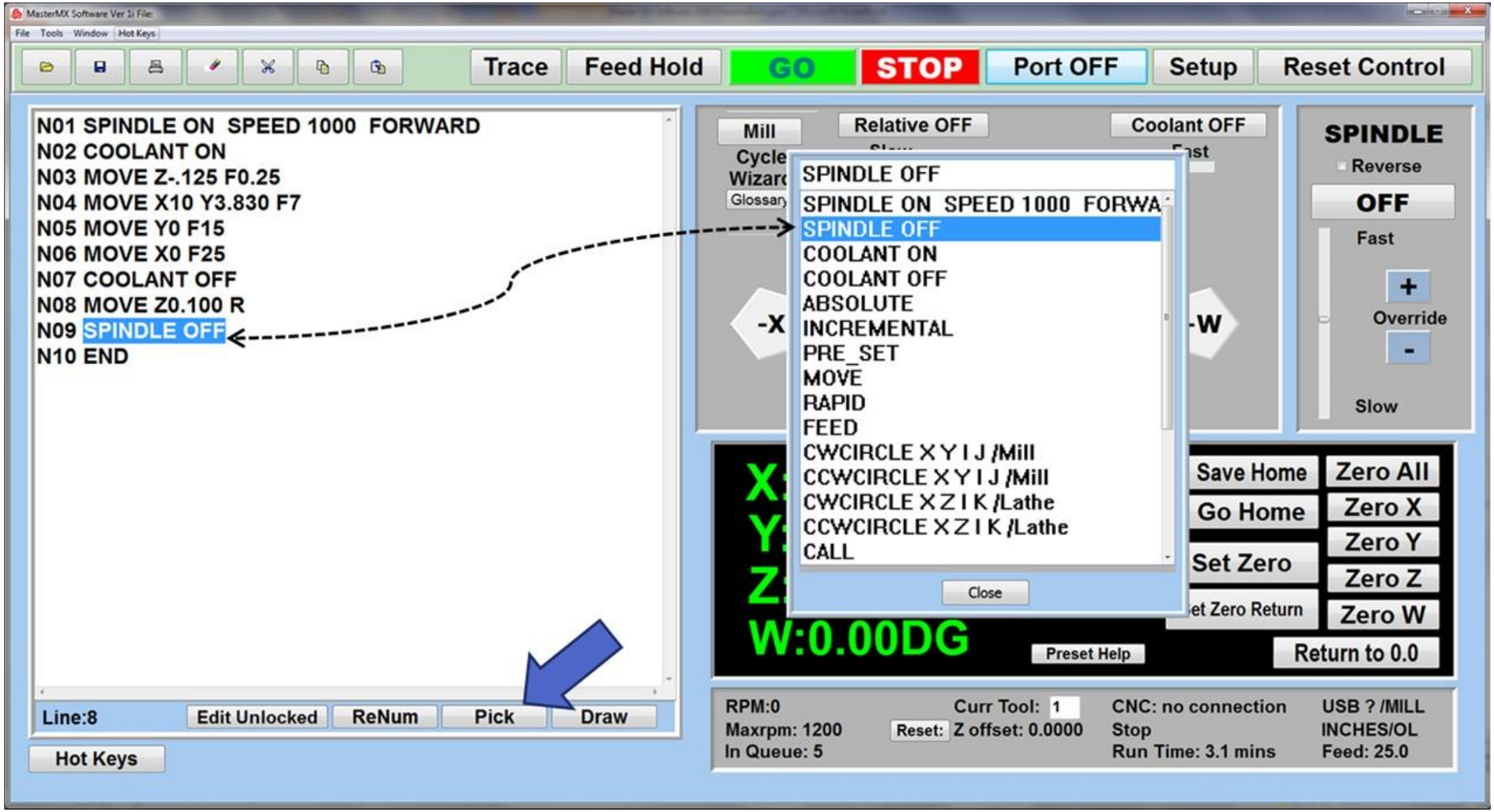
Hot Keys

Hot Keys is an alternative method to easily control your machine using your hard or touch screen keyboard. One can 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 in teach mode.



Pick Menu – for conversational mode programming

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.



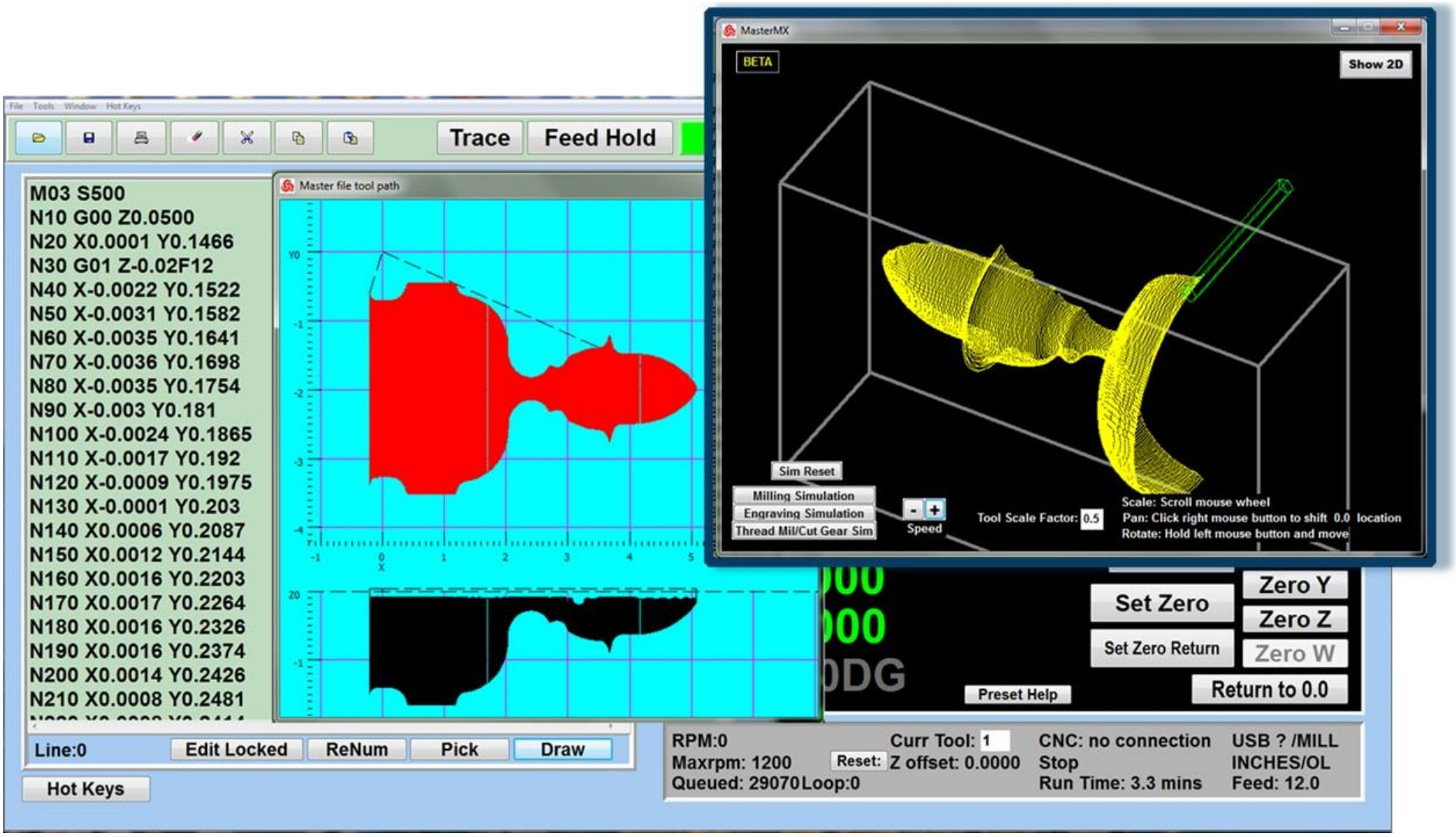
Pick Menu List of Options

Use commands such as MOVE, SPINDLE ON/OFF, COOLANT ON/OFF, PAUSE, DELAY, GO HOME…. to write your tool path programs in conversational mode.

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.

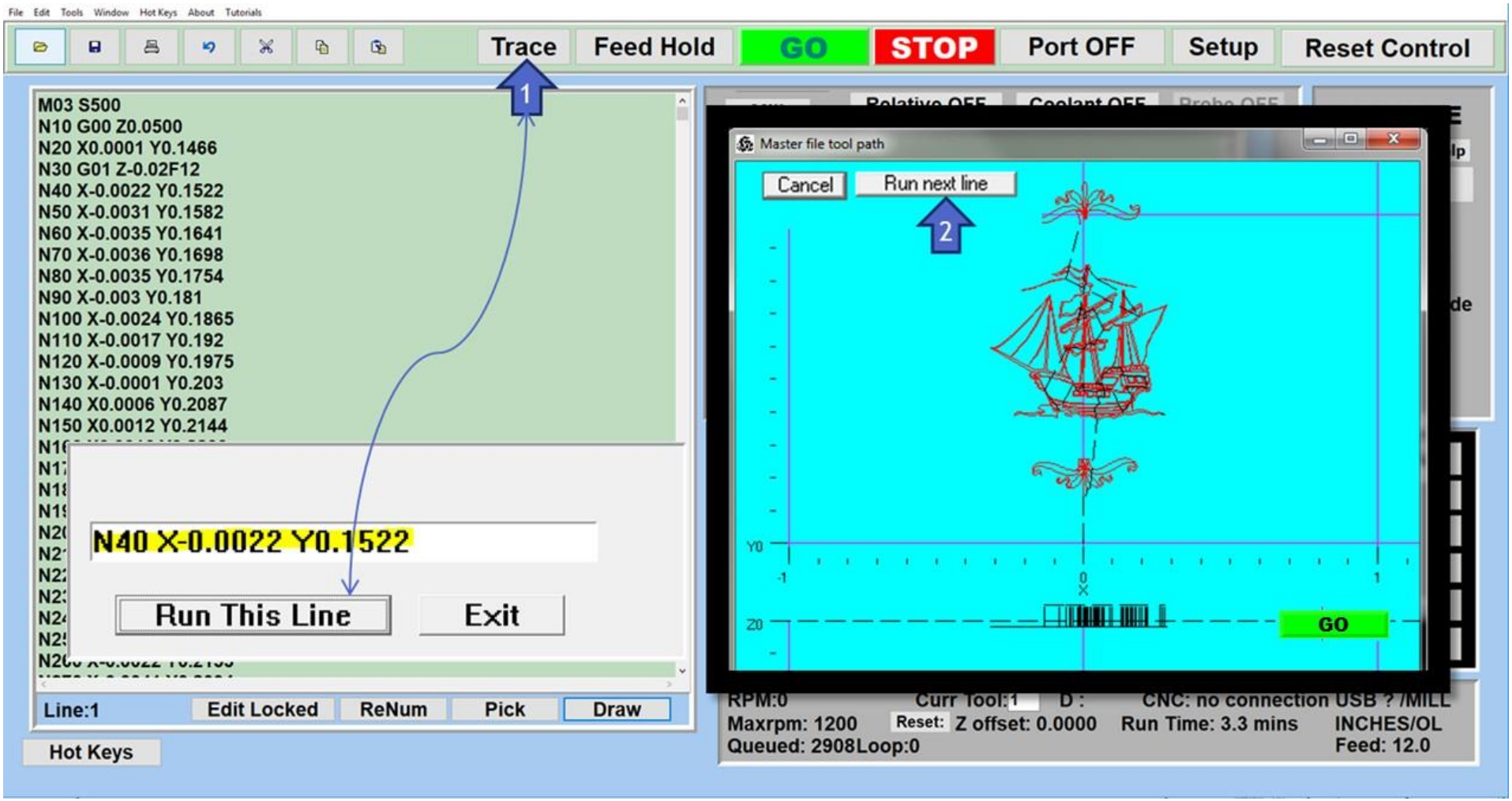
Draw the Tool Path to verify it before pressing Go

Hit Draw to view your tool path program drawing, check out its run time, 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. You can also hit Go within the Draw Window itself to verify the cutter's position on the machine. The current tool path will be highlighted and simultaneously draw out the next path so you can verify what the cutter will be doing next on the program.



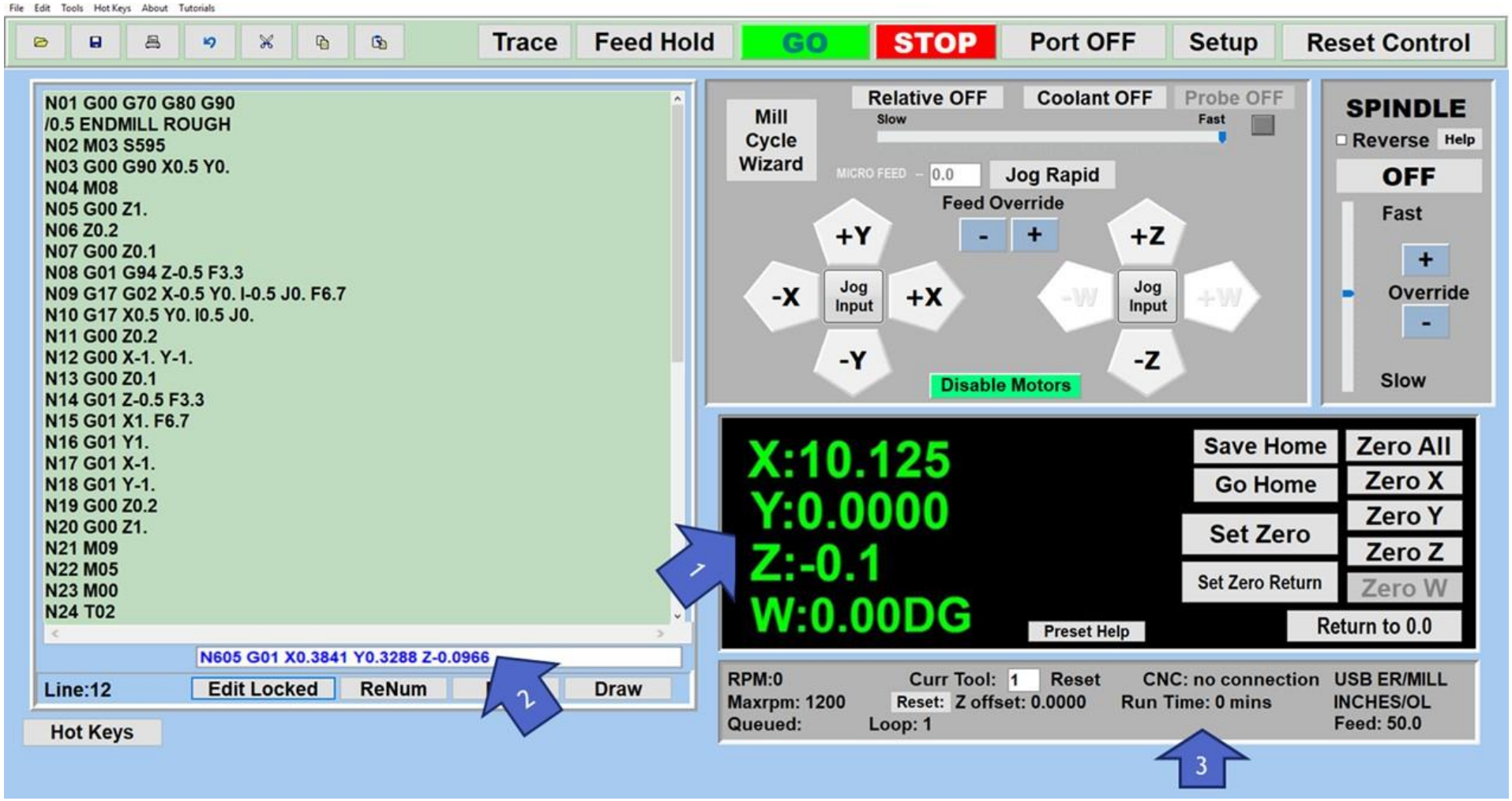
Run each tool path independently to study its movement

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 one line at a time in the Draw Window.



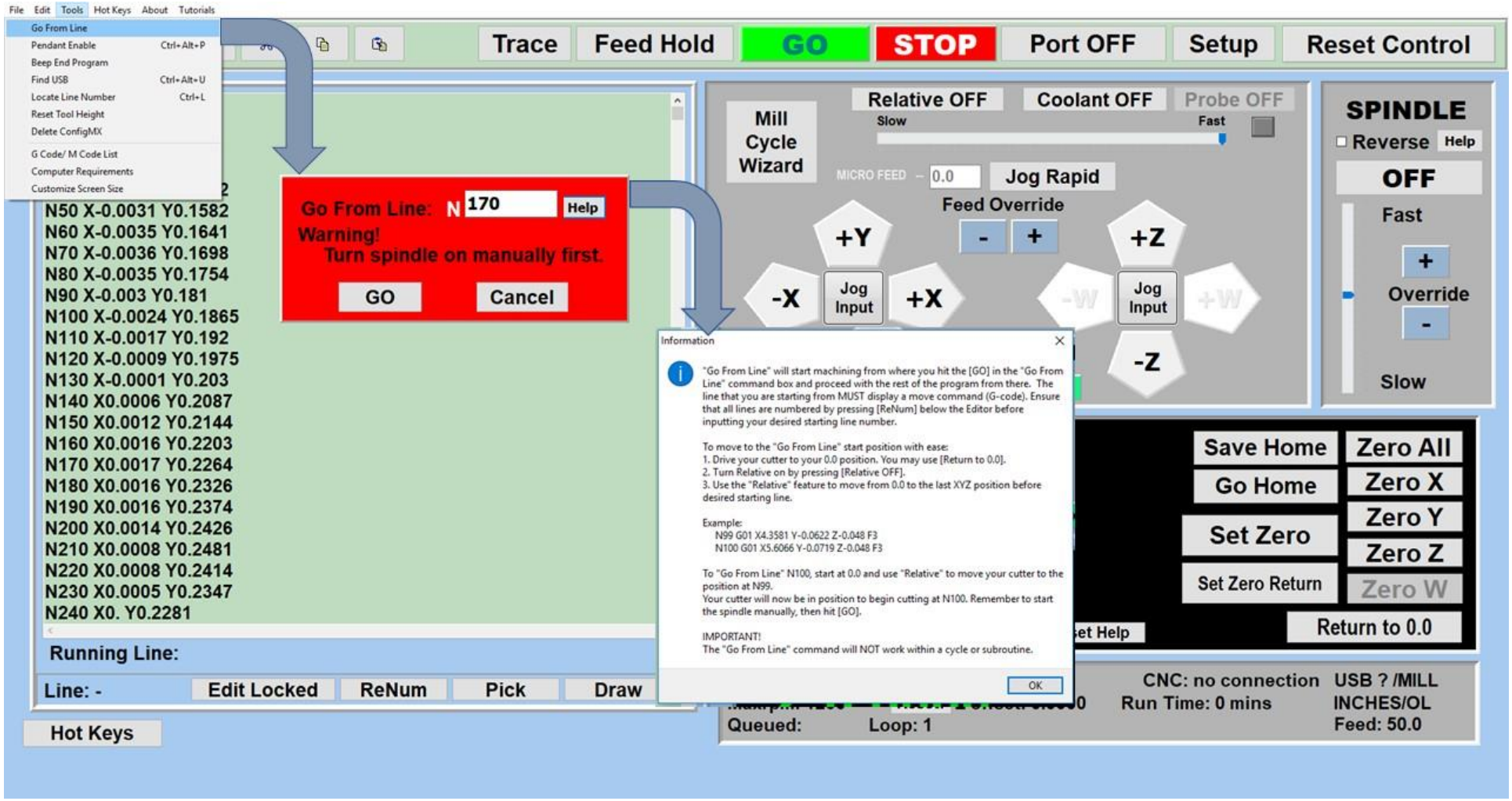
Counters display in Inches or Millimeters – Continuous Feed

1. When running a program, the counters will display a “real-time” readout while the machine is in CNC operation without counting ahead of the movement.
2. The current tool path is highlighted while the machine is in operation without causing slight interruptions/pauses as the software feeds the tool path to the machine. The MX internally interprets a program ten lines ahead to allow for “continuous machining” avoiding slight interruptions as the machine waits for its next tool path command.
3. “Run Time” tells you how long it takes to run your tool path program.



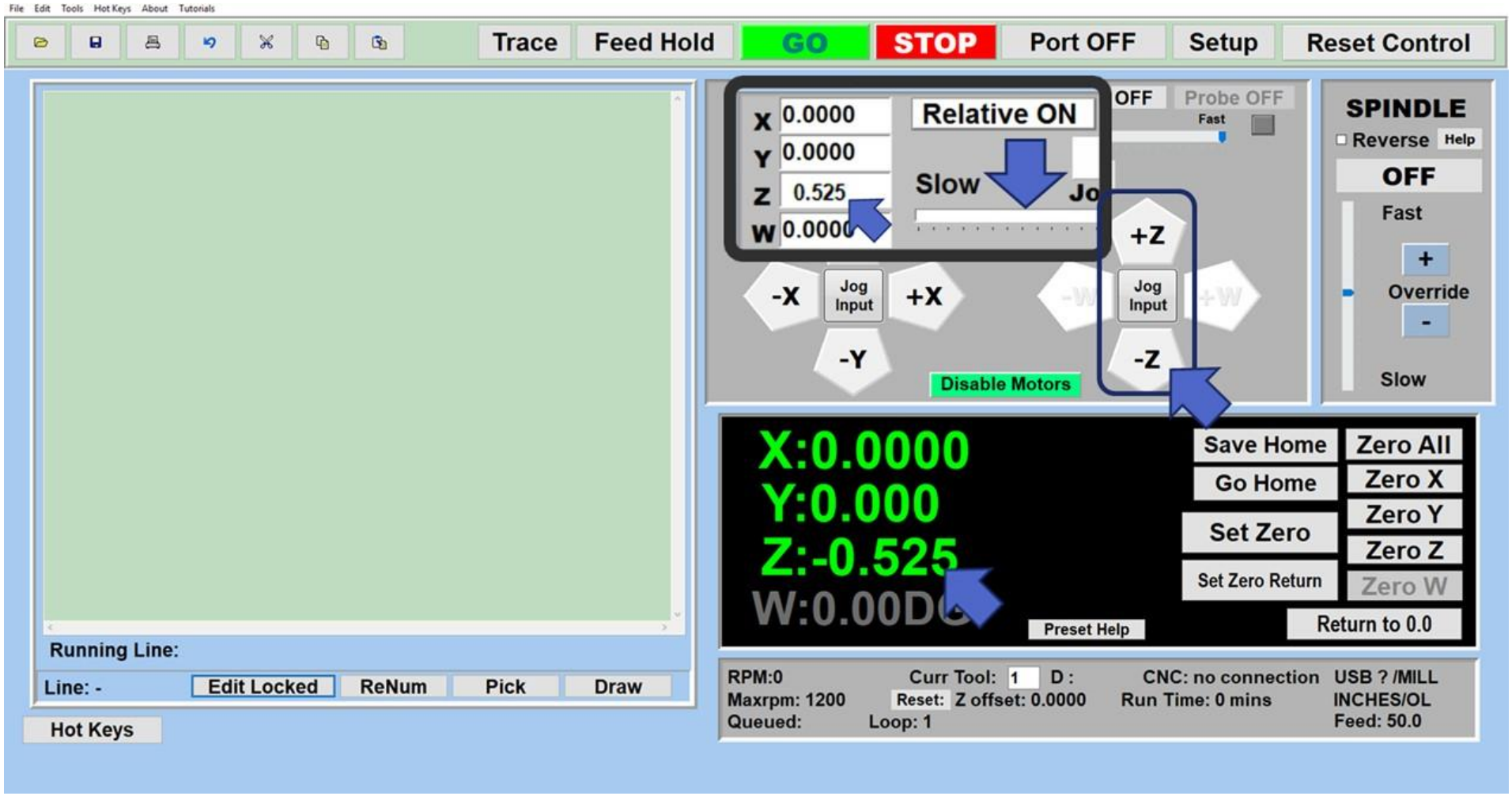
Use the “Go From Line” command to start in the middle of your program

If you ever need to begin your program from somewhere in the middle of it, use [Go From Line] which you can find under Tools. The Help guide will walk you through how to position the cutter without losing its position on the machine.



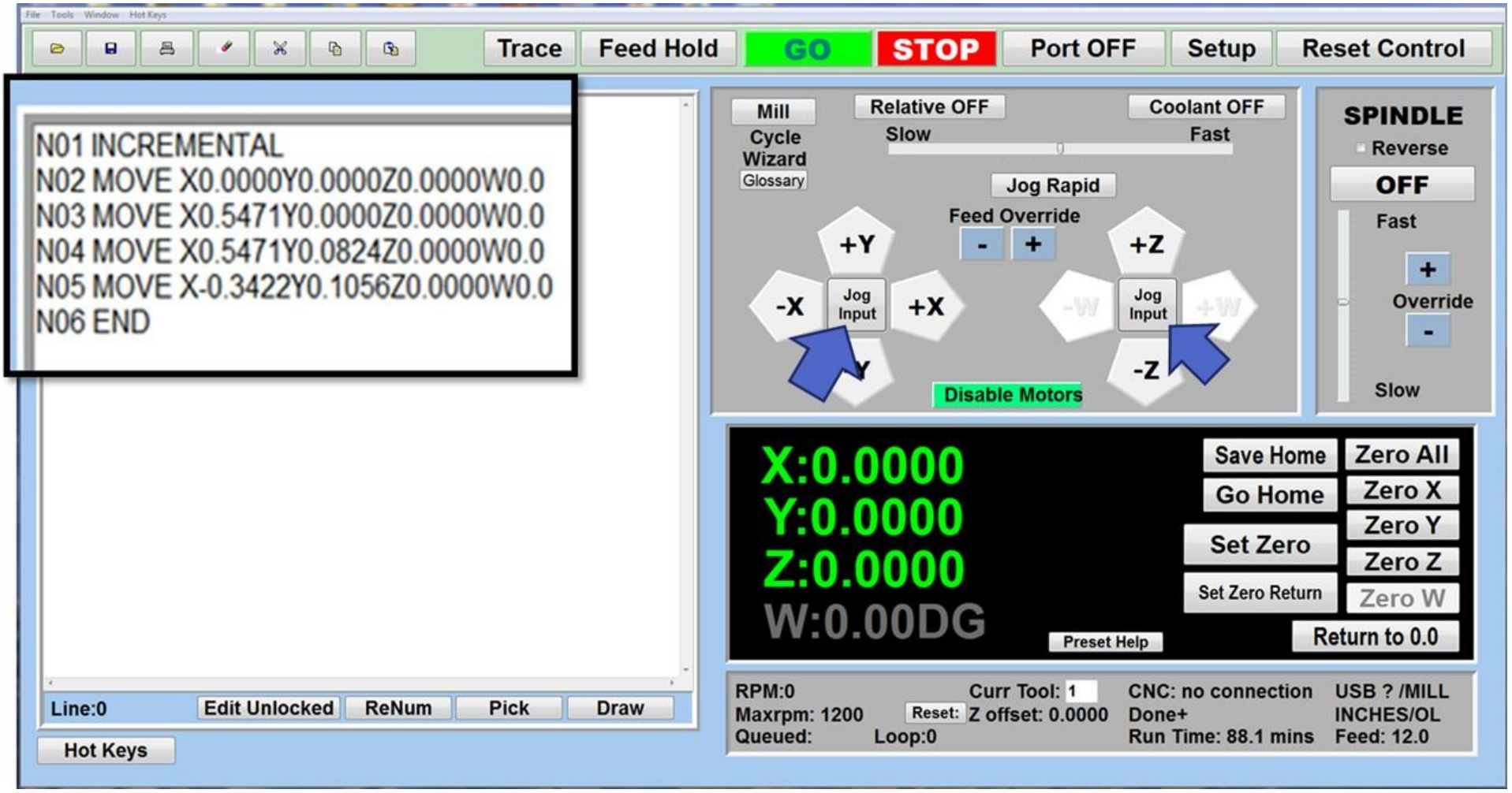
Exact Motion Distance without over-stepping on an axis while jogging

Use “Relative ON” to enter a specific coordinate to jog any of your axes to an exact location without having to write a program. It’s like using “power feed” but easier. You can jog an exact distance 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 jog button. 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”.



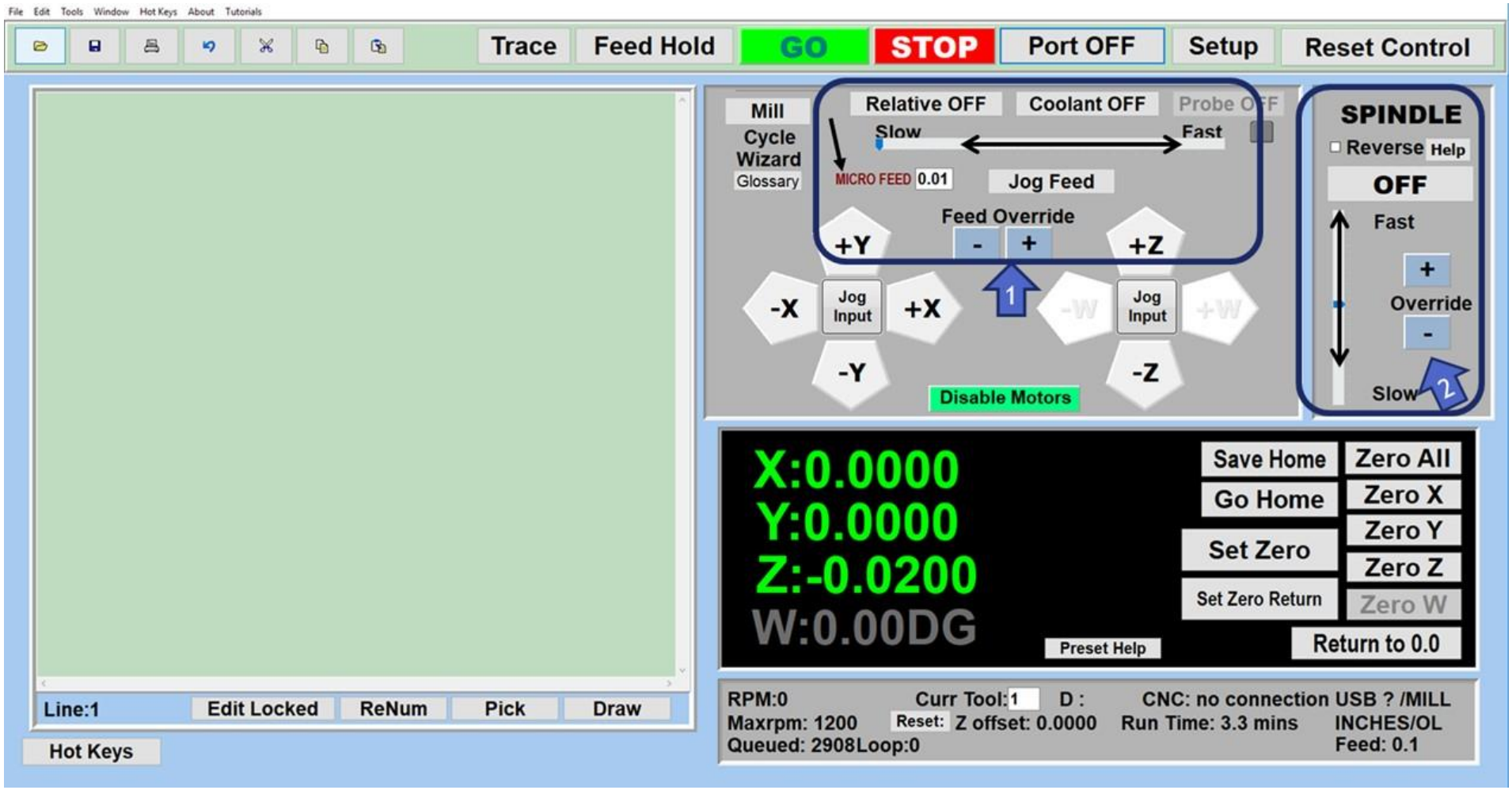
Teach Mode – Jog Input

You can create a tool path program by storing each point-to-point movement by simply jogging an axis one at a time. 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.



Override on the fly to adjust the Jog Feed to Rapid or the Spindle Speed during the middle of a program

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. You can even micro-step your jog as low as 0.01”/min. The [-][+] buttons allow you to fine tune feeds in 5% increments while the program is in motion.
2. Spindle Speed with Override: You can adjust speeds using the slider from a slow minimum RPM to the max RPM according to the machine setup. The [-][+] buttons allow you to fine tune feeds in 5% increments while the program is in motion.



Adjust Counters using Pre-Set if you cannot begin the program from 0.00

In a situation where you cannot begin your cutter at it's 0.00 location, you can "Pre-Set" directly into the counters by typing in your beginning coordinate. You can press Go from here to run your program. You can also "zero all" or "zero" your counters independently. With one click of the [Return to 0.0] button, all axes will travel back to its respective 0.0 on the machine.

Set and Save your 0.00 position for future runs

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 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.

Create a “Home” position to clear your application and run multiple times

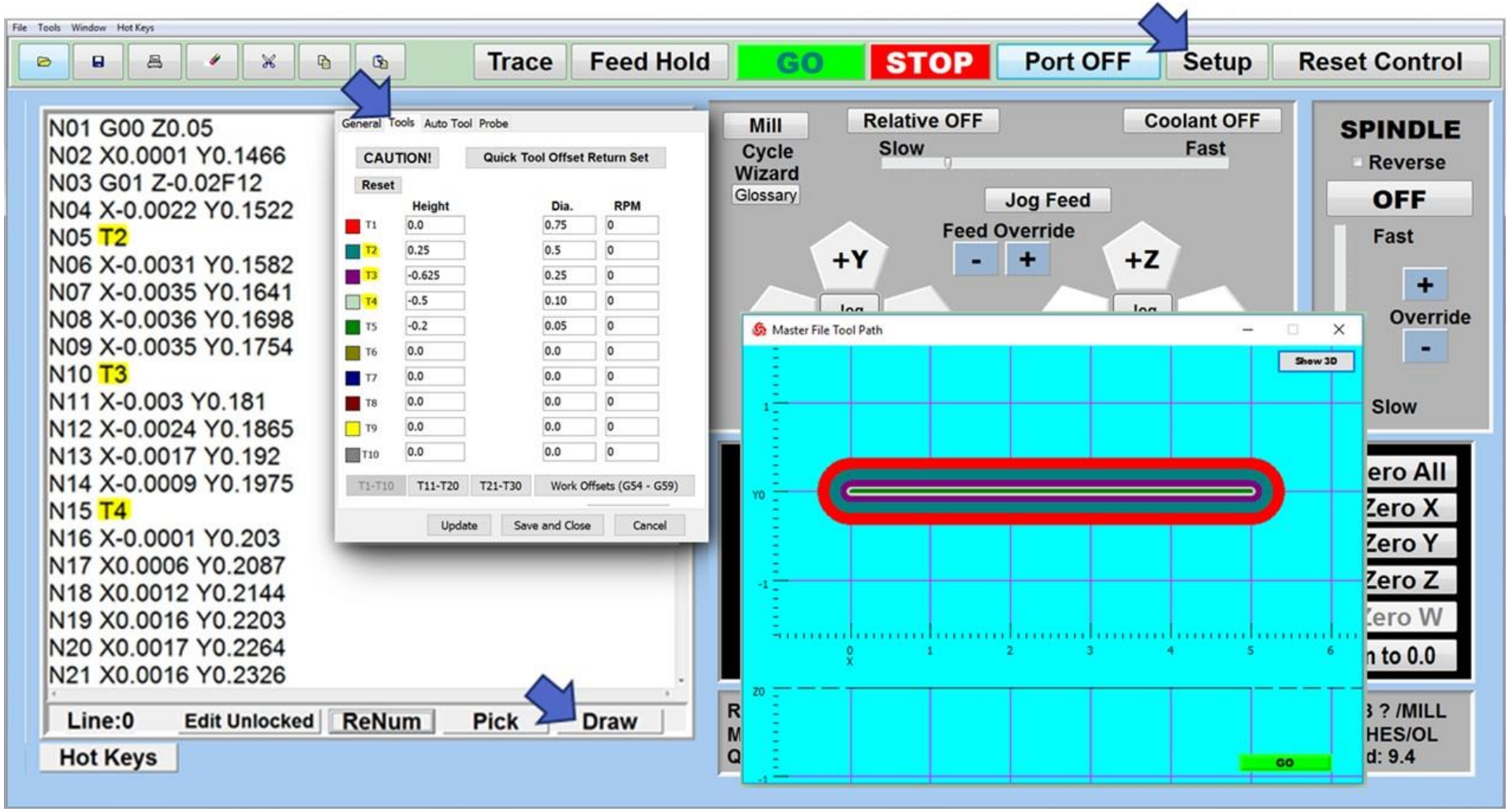
Let’s say you need to machine one application times 100 pieces. This usually requires a jig to retain that physical 0.00 position. But in this case, you want the program to end with a clearance of the axes to easily switch out the next piece of stock and start again. With Save Home, you have the ability 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. At the completion of your program the axes will end at your Home position. Replace your part, then press [Return to 0.0] button to allow the axes to return to its zero position, and press Go to start your next run.

Disable the axis motors to manually hand crank each axis into place

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.

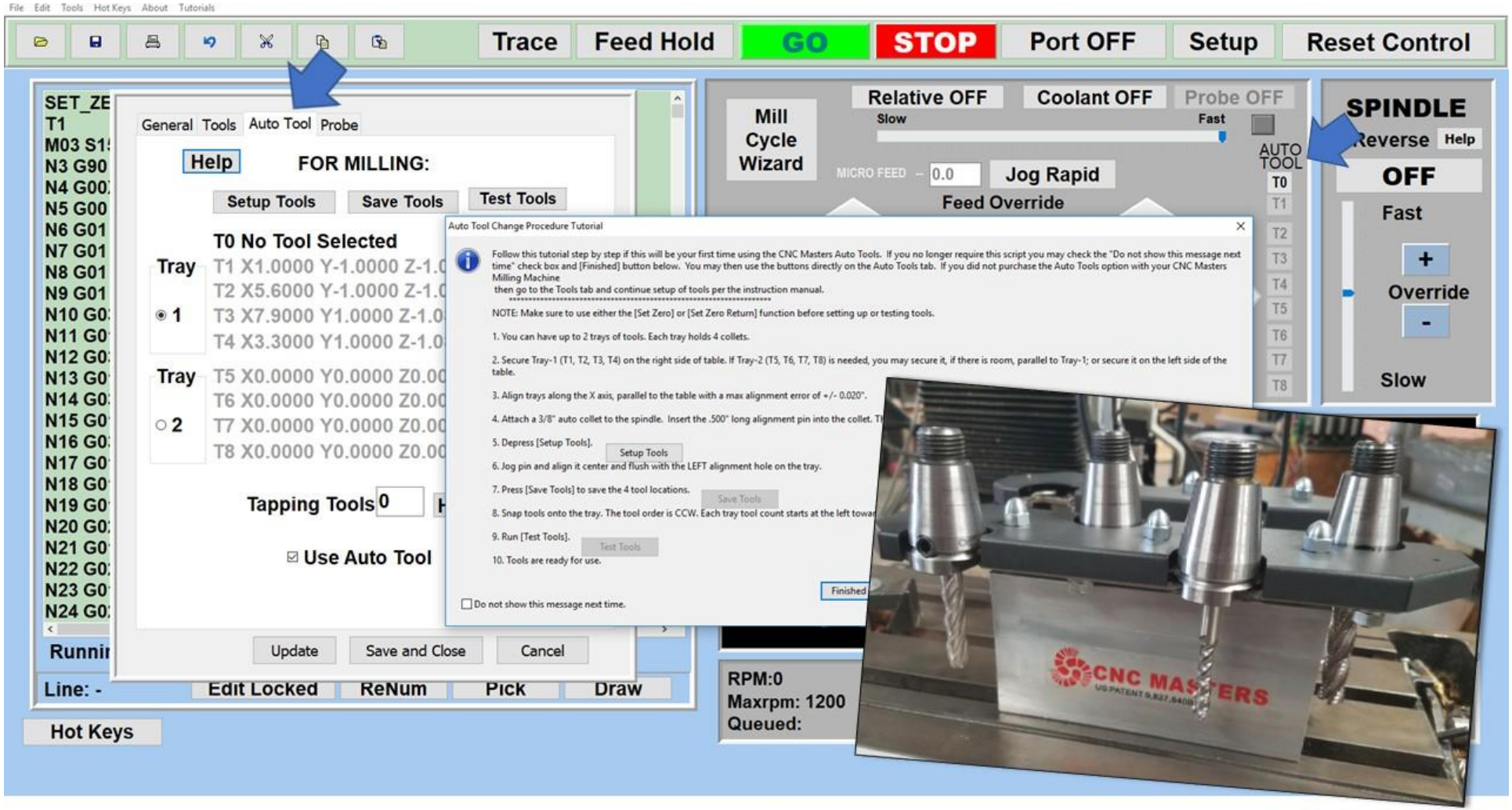
Change up to 30 tools with compensation, and store your tool offsets for other programs

The MX supports... Tool Height Compensation allows for accurate height offsets when making a tool change using quick change tools within a program. Up to 30 tool changes can be made. This feature can be very effective for improved productivity if your application requires several tool changes. Store a library of tool offsets in the Setup > Tools window. You can choose any tool 1 – 30 by writing a T# command on its own line in a program. With a T command, the spindle will automatically shut off and retract up to exchange tools without needing to write extra lines of code. 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 denoted by a different color in the Draw window.



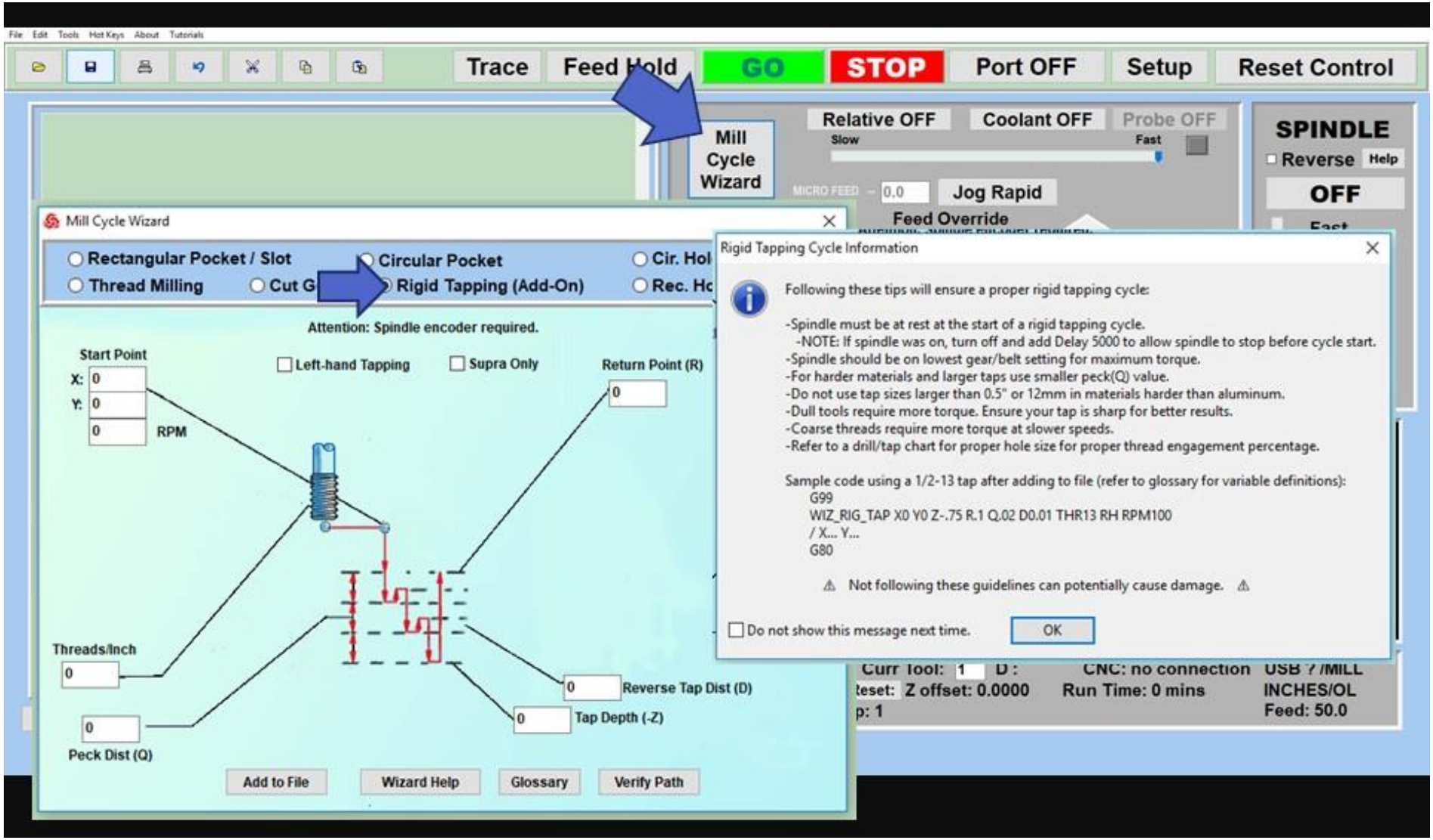
Use the optional ATC rack up to 8 tools for milling, drilling, and rigid tapping applications

The CNC Masters Automatic Tool Changer Rack and Tools (US Patent 9,827,640B2) can be added to any CNC Masters Milling Machine built with the rigid tapping encoder option. The tutorial will guide you through the set-up procedure using the ATC tools.



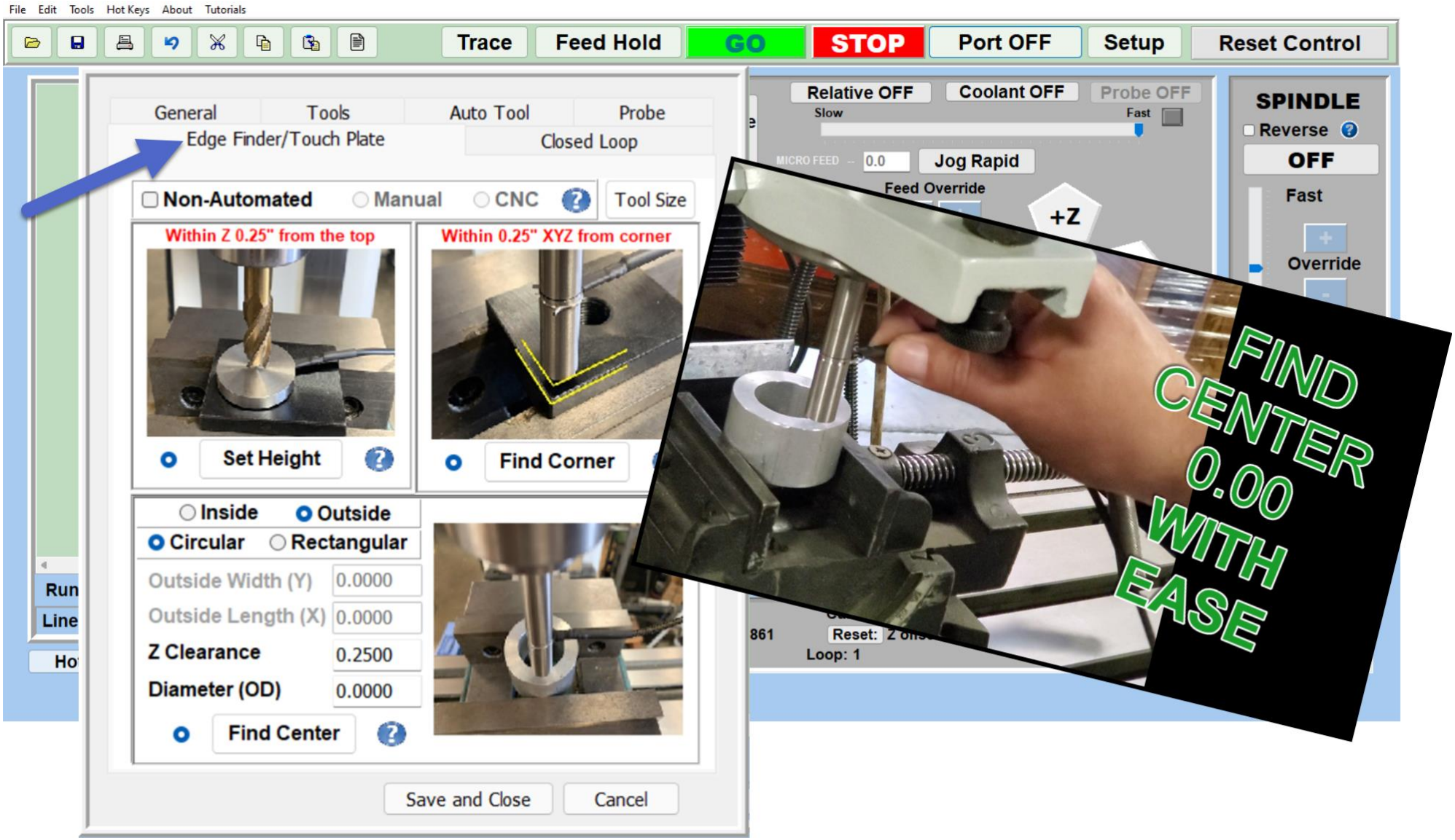
Use the optional Rigid Tapping Wizard without the need for tapping head attachments

When you order your CNC Masters machine, have it built with the optional rigid tapping encoder. You can take any drill cycle program and replace the top line with a tapping code created by the wizard to tap your series of holes up to 1/2” in diameter.



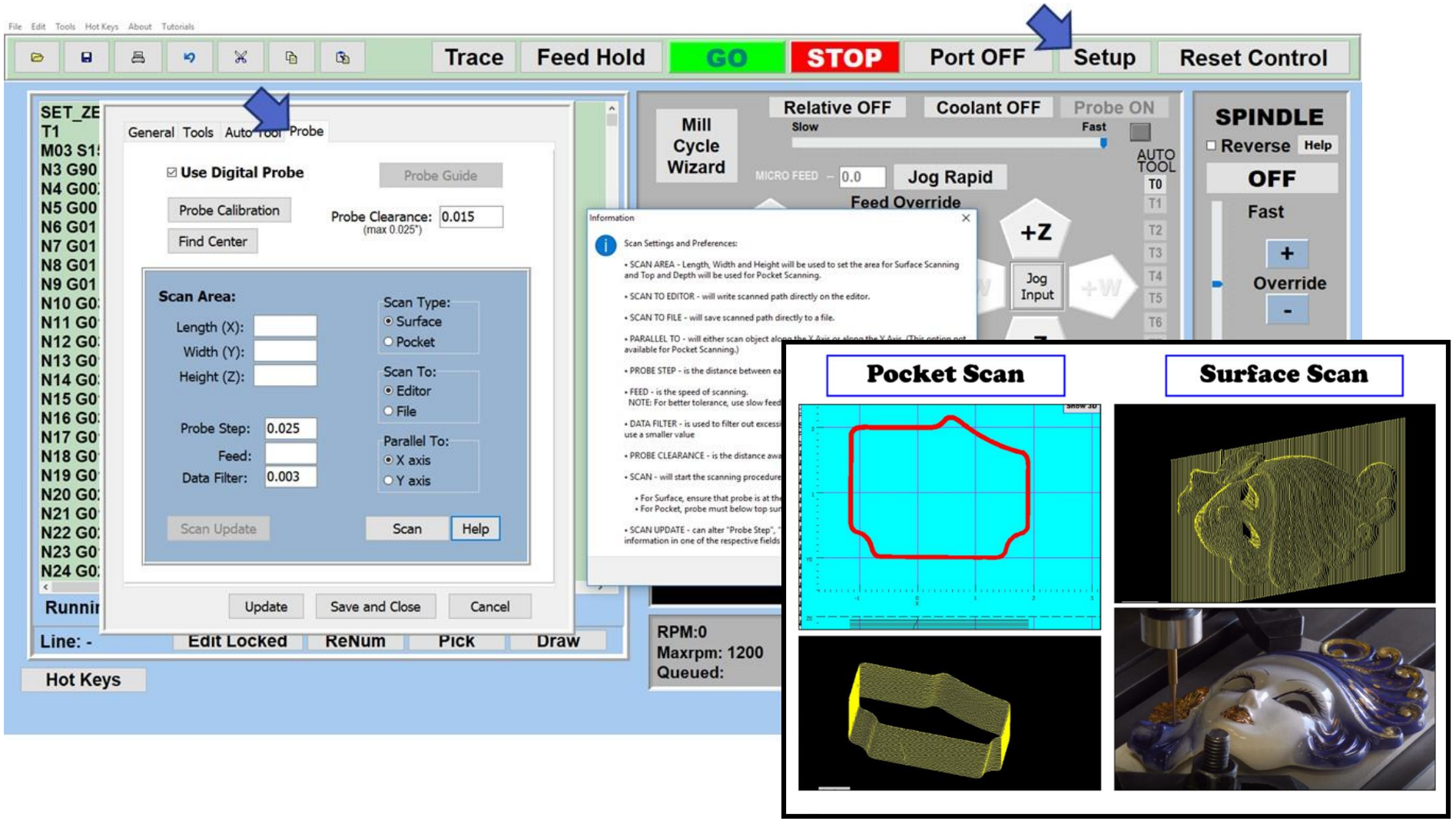
Use the optional Electric Edge Finder and Touch Plate to quickly find zero on your part

Order your CNC Masters machine with this kit and you will discover how simple it is to find the corner of your part, find the center of your part, and get your cutter to kiss the top of the part through a couple of commands on the software. If you have several tool changes, you can easily record the height of each tool using the touch plate. Using our edge finder will help you find that perfect center line of your cutter every time without having to manually rotate your cutter to find the edge.



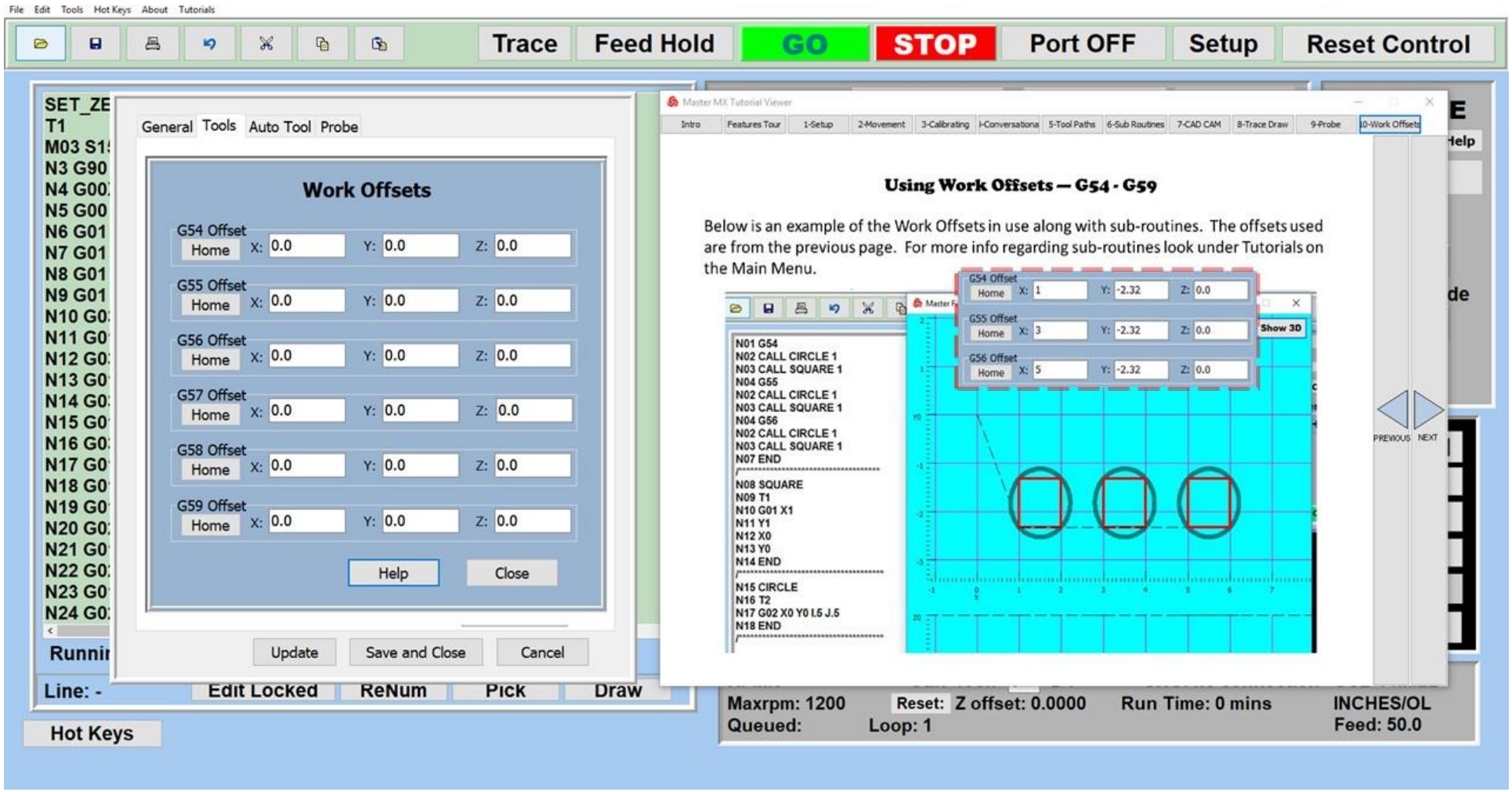
Use the optional Digital Probe to scan the profile and/or pockets of your fun/hobby type designs to write your tool path program and machine out a duplicate of your original design

To “surface” scan an object, you can program the probe along the X or Y plane. The stylus will travel over the part starting on the left side front corner of the object and work its way to the end of the part on the right side. Depending on how the stylus moves, it will record linear and interpolated movements along the X, Y, and Z planes directly on the MX Editor. To “pocket” scan an object containing a closed pocket such as circles or squares, the scan will start from the top front, work its way inside of the pocket, and scan the entire perimeter of the pocket. Under the Setup of the MX software you will find the Probe Tab which will allow you to calibrate and program your probe. Your “Probe Step”, “Feed”, and “Data Filter” can also be changed on the fly while the probe is in the middle of scanning your object.



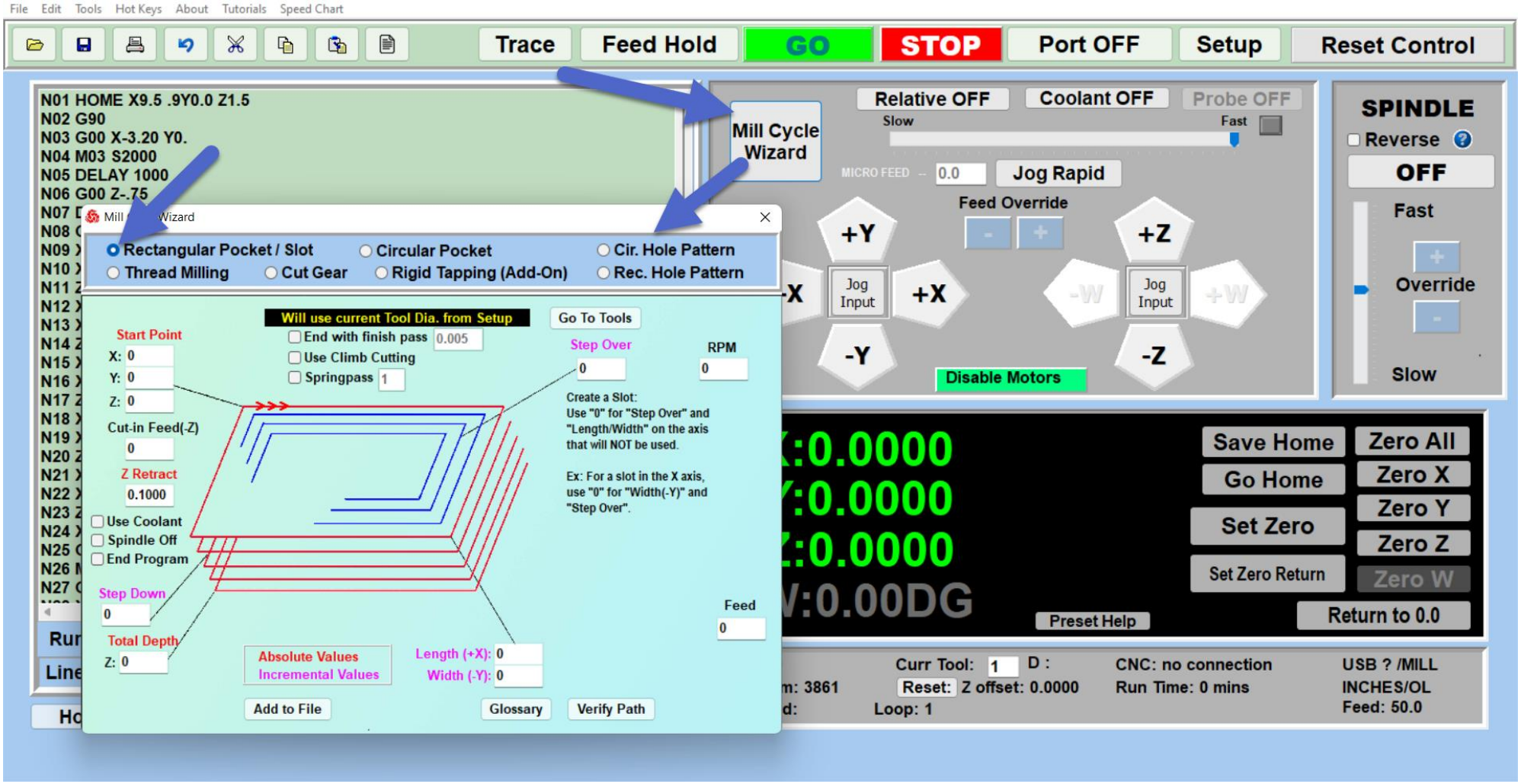
Use work offsets G54-G59 for nesting applications

The work offsets offer you a way to program up to six different machining locations. It's like having multiple 0.0 locations for different parts. This is very useful especially when using sub-routines/nesting applications.



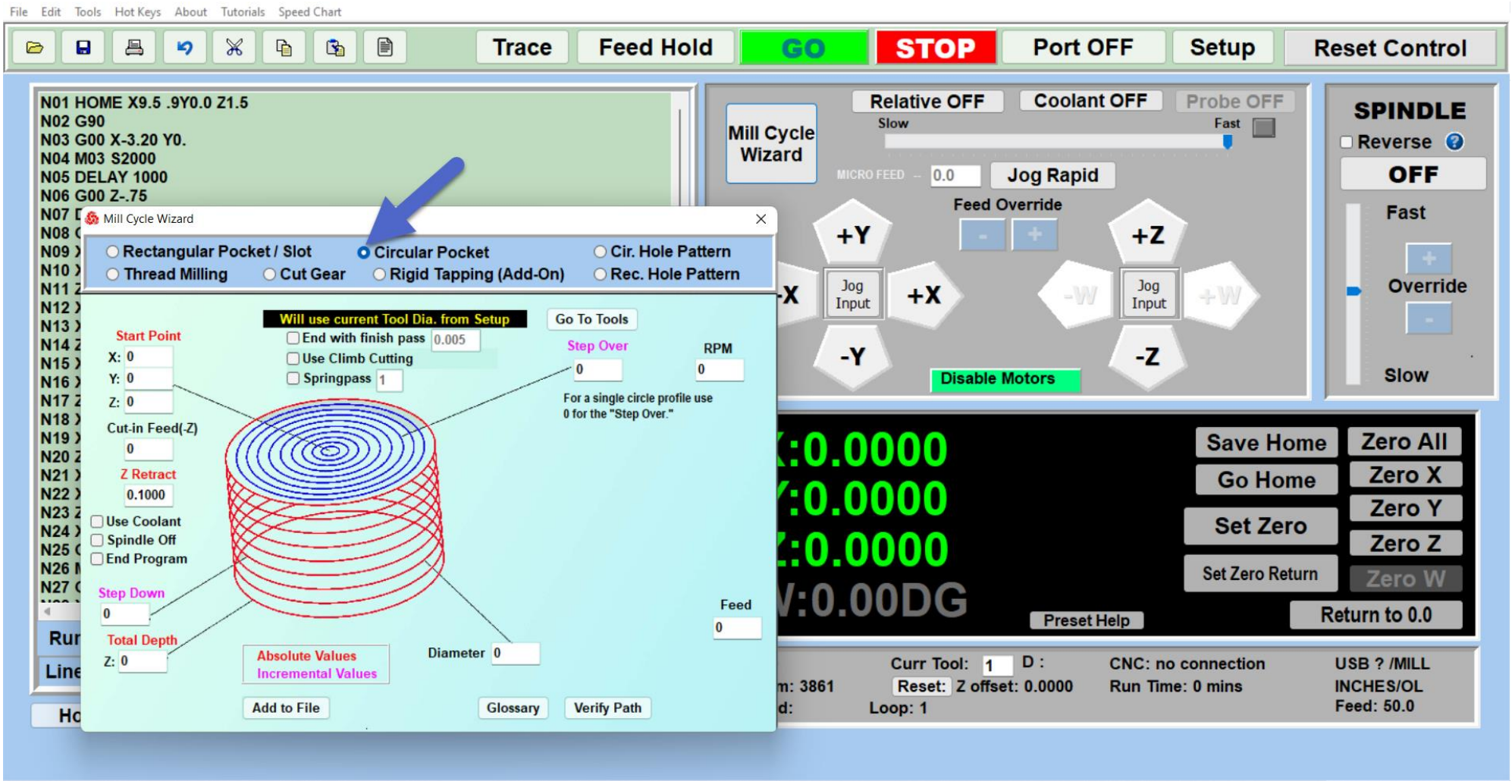
Create a Rectangular Pocket / Slot with our selection of Wizards to help you build a tool path program

The Cycle Wizards for the mill or lathe makes it easy to create a simple tool path without needing to use a CAD and CAM software. On this Wizard, the Rectangular Pocket / Slots, can be used to form a deep rectangular pocket into your material or machine a slot duplicating as many passes needed to its total depth.

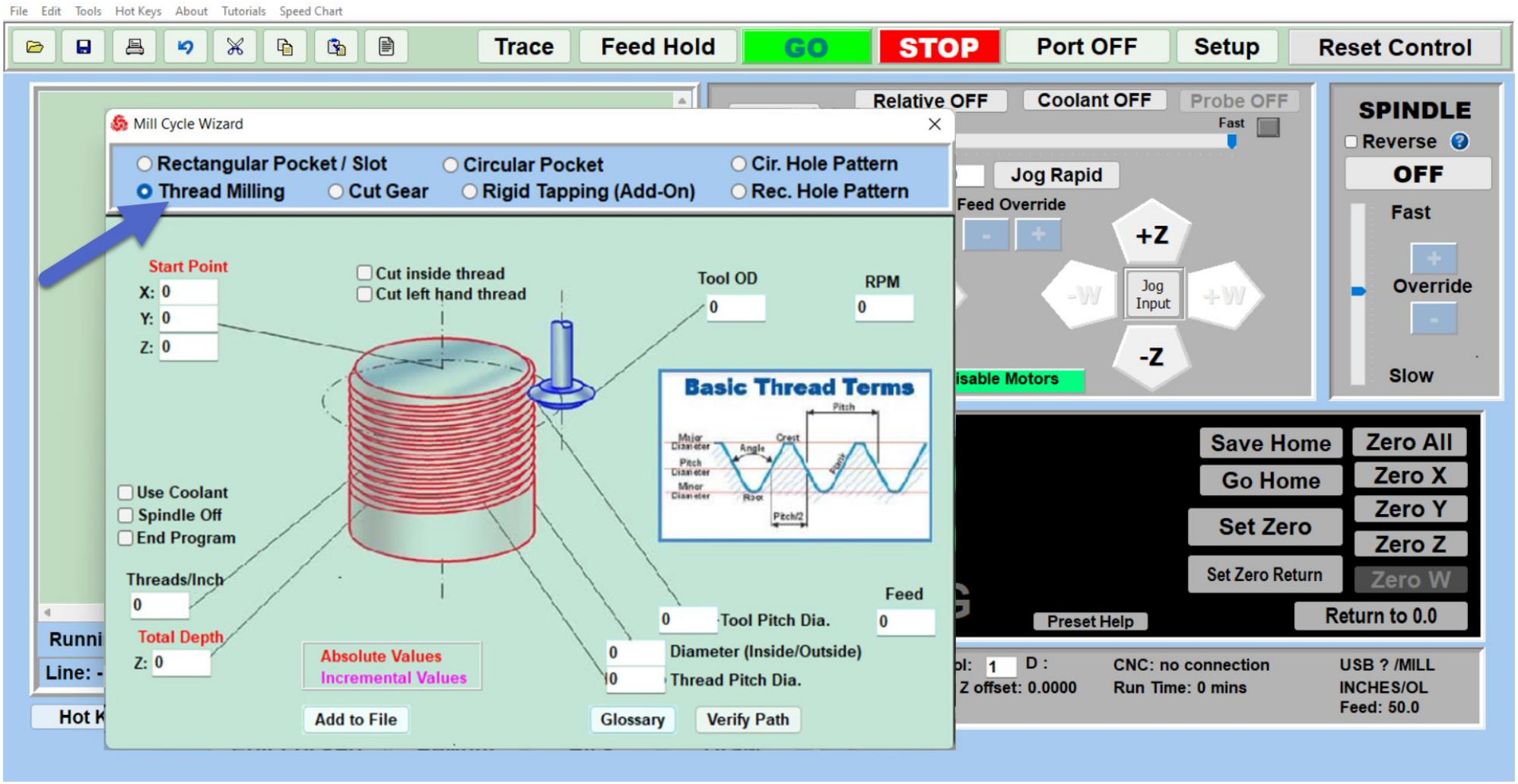


Create a Circular Pocket Wizard

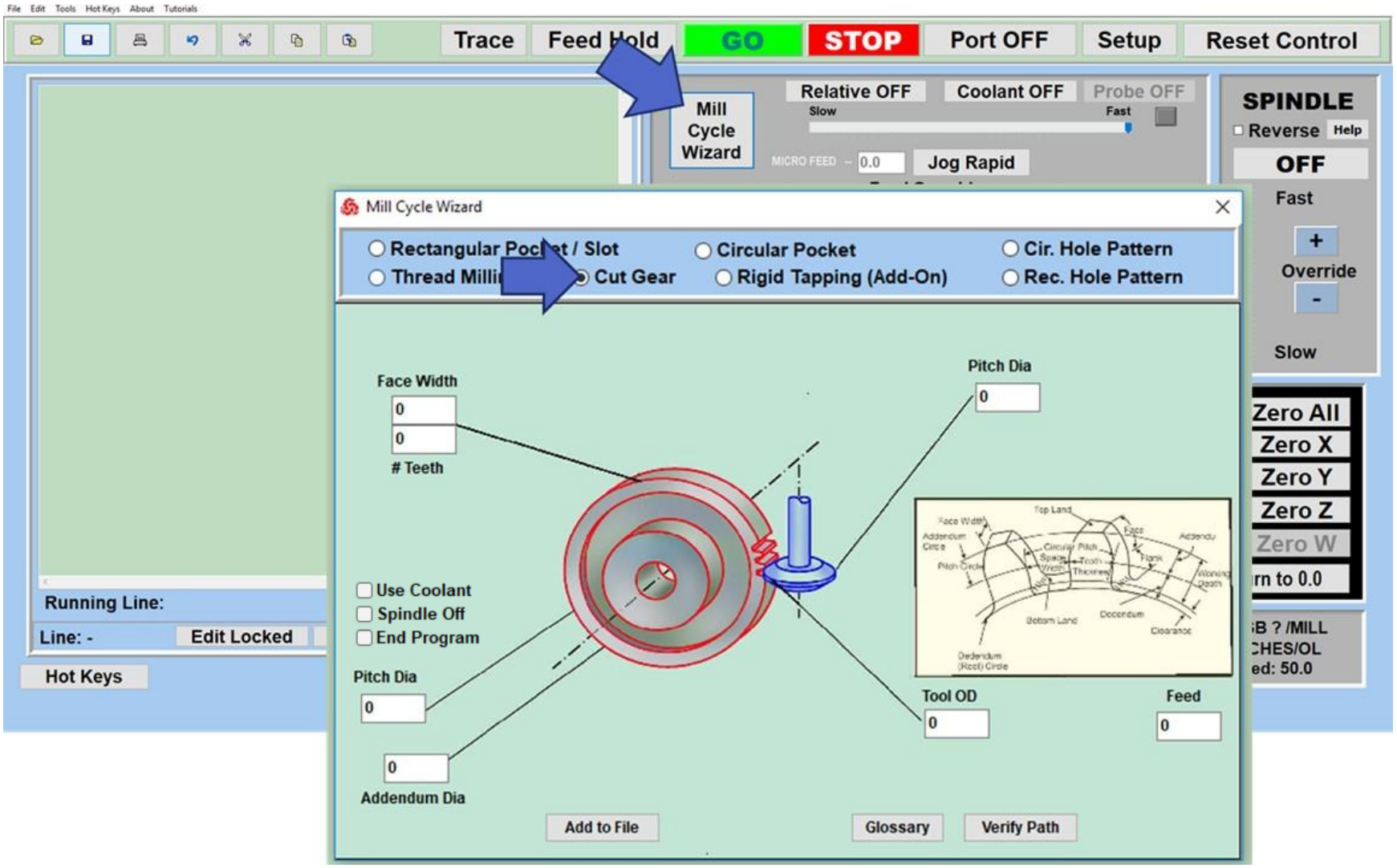
Input the total diameter, the step down, and total depth and the code will be generated.



Do Thread Milling using a single point cutter Wizard

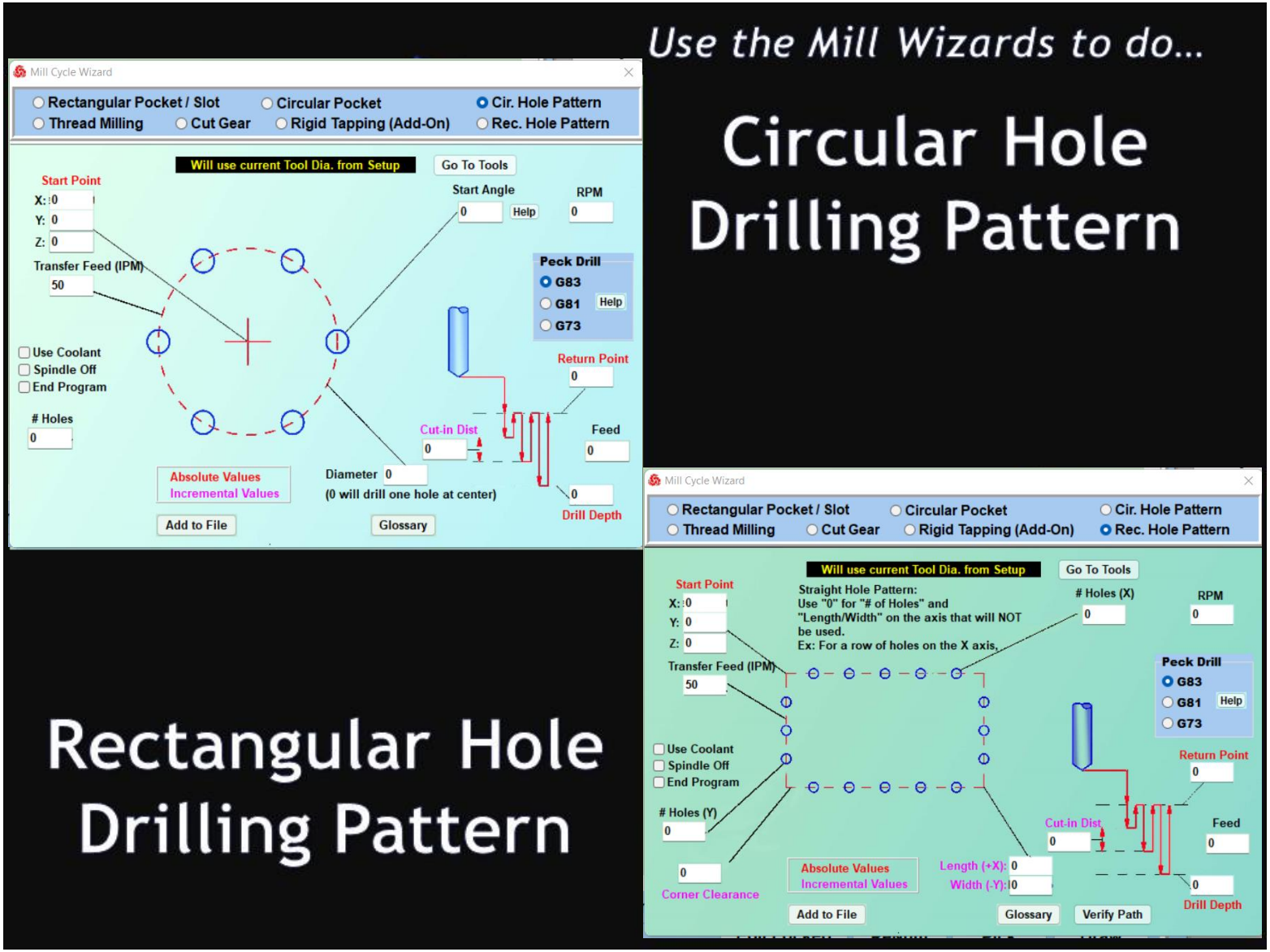


Cut a gear out using the Cut Gear Wizard with the optional Fourth Axis



Create a Peck Drilling Program in Circular or Rectangular Patterns

Using the Circular or Rectangular Drilling Wizards, you can program the machine to drill an un-limited series of holes along the X and Y planes. Program it to drill straight through to your total depth, use a high-speed pecking cycle, or deep hole pecking cycle. You can program the cut-in depth and return point for a controlled peck drill application to maximize chip clearance.

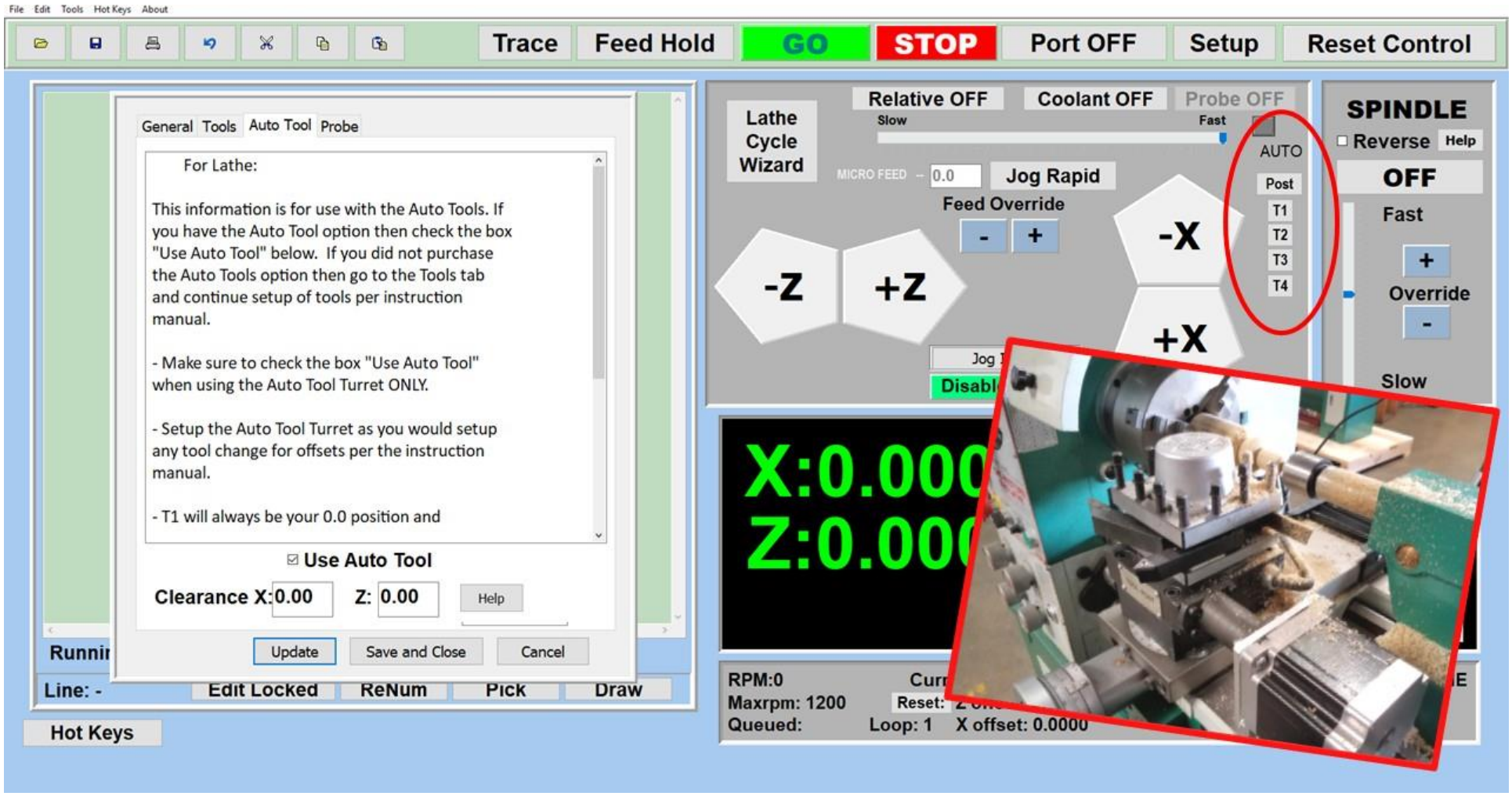


The MX interface can easily be interchanged from Mill Mode to Lathe Mode

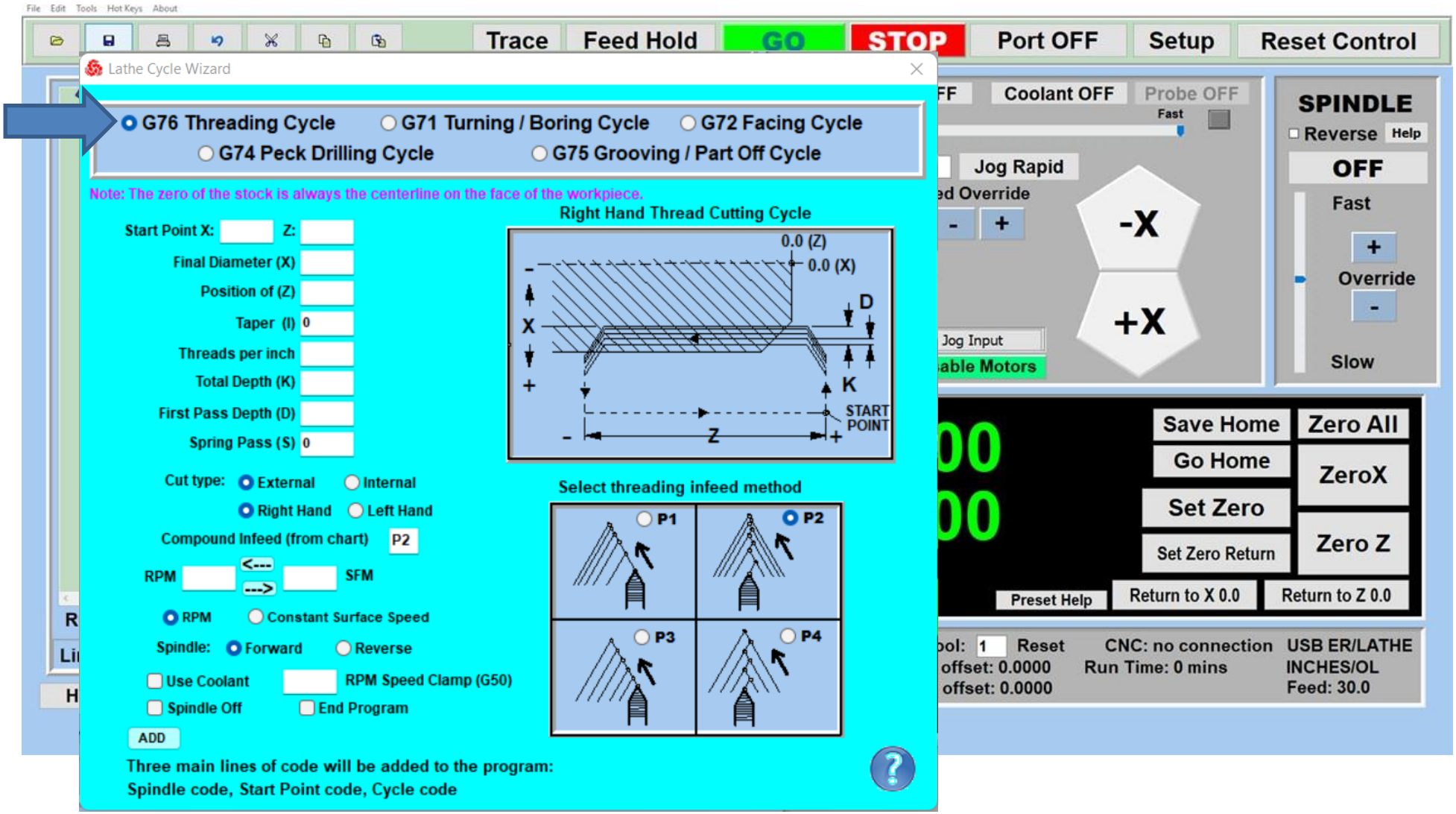
Use this interface for your CNC Masters Lathe. It contains all the same user-friendly features and functions that comes in Mill Mode. Simply go to the Setup page and change the interface.

Use Tool Change Compensation or the optional Auto Tool Changer Turret if your application requires more than one tool in a single program

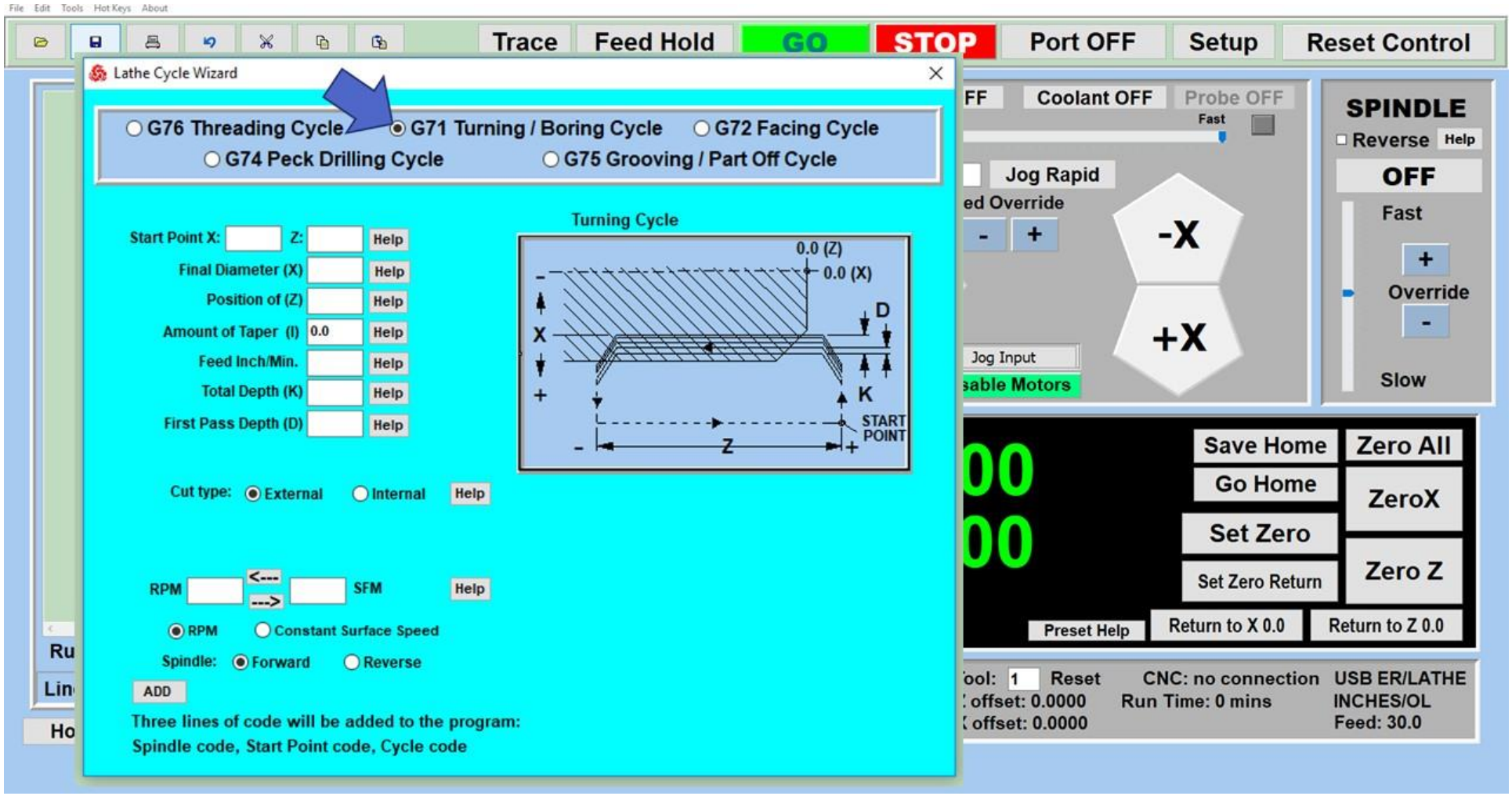
You can offset the length and angle of each tool and record it under Tools in your Setup. The program will automatically pause the lathe's movement and spindle allowing you to change out your tool, or allowing the optional ATC Turret to quickly turn to its next tool and continue machining. On the MX interface, you also have four Tool Position buttons. Select your desired T position, and the auto tool post will quickly turn and lock itself to that position.



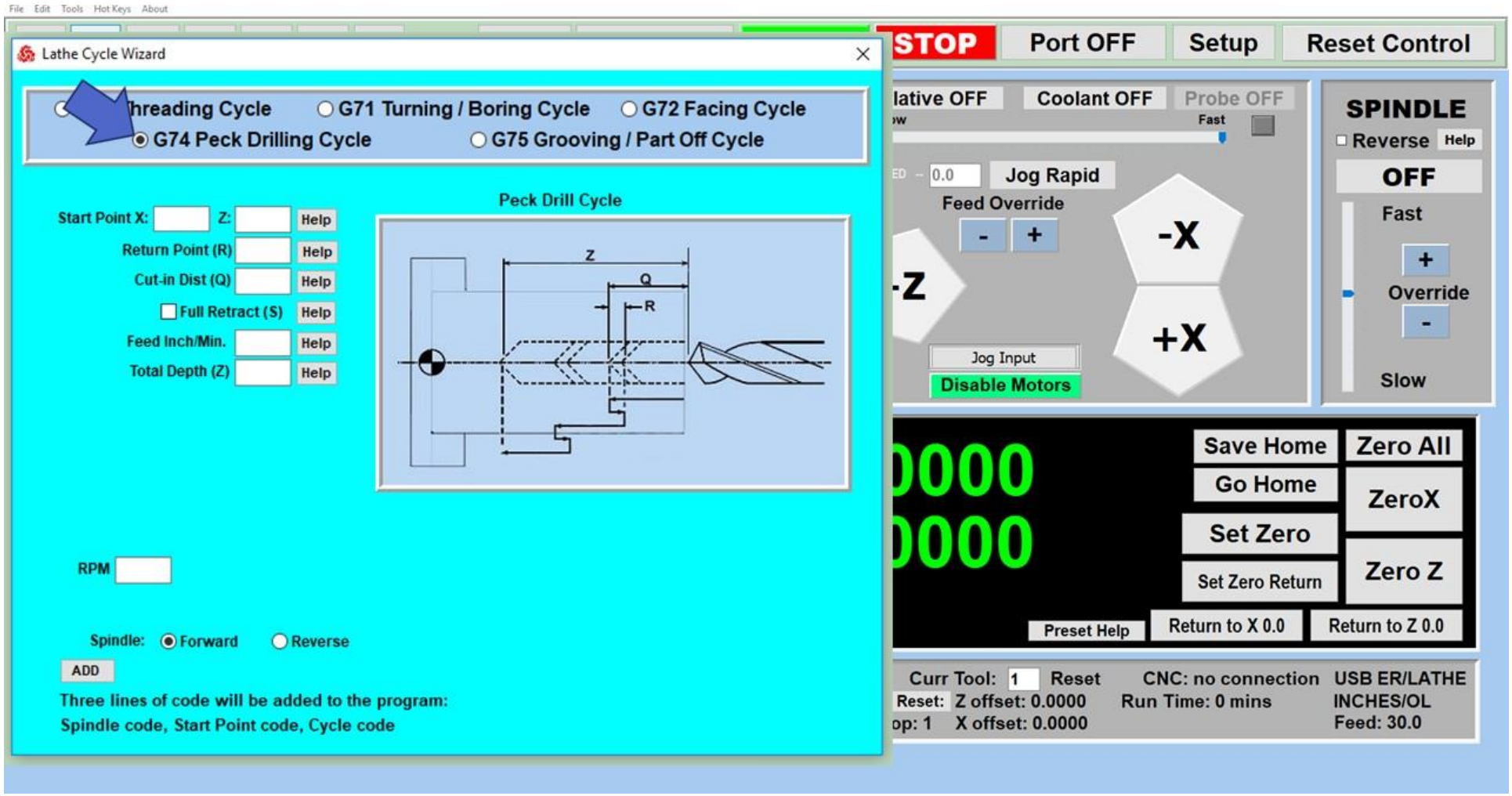
Use the Lathe Wizard Threading Cycle to help you program your lathe's internal or external threads in inches or metric



Use the Lathe Wizard Turning / Boring Cycle to help you program simple turning and boring cycles without having to go through a CAM or writing a long program with multiple passes

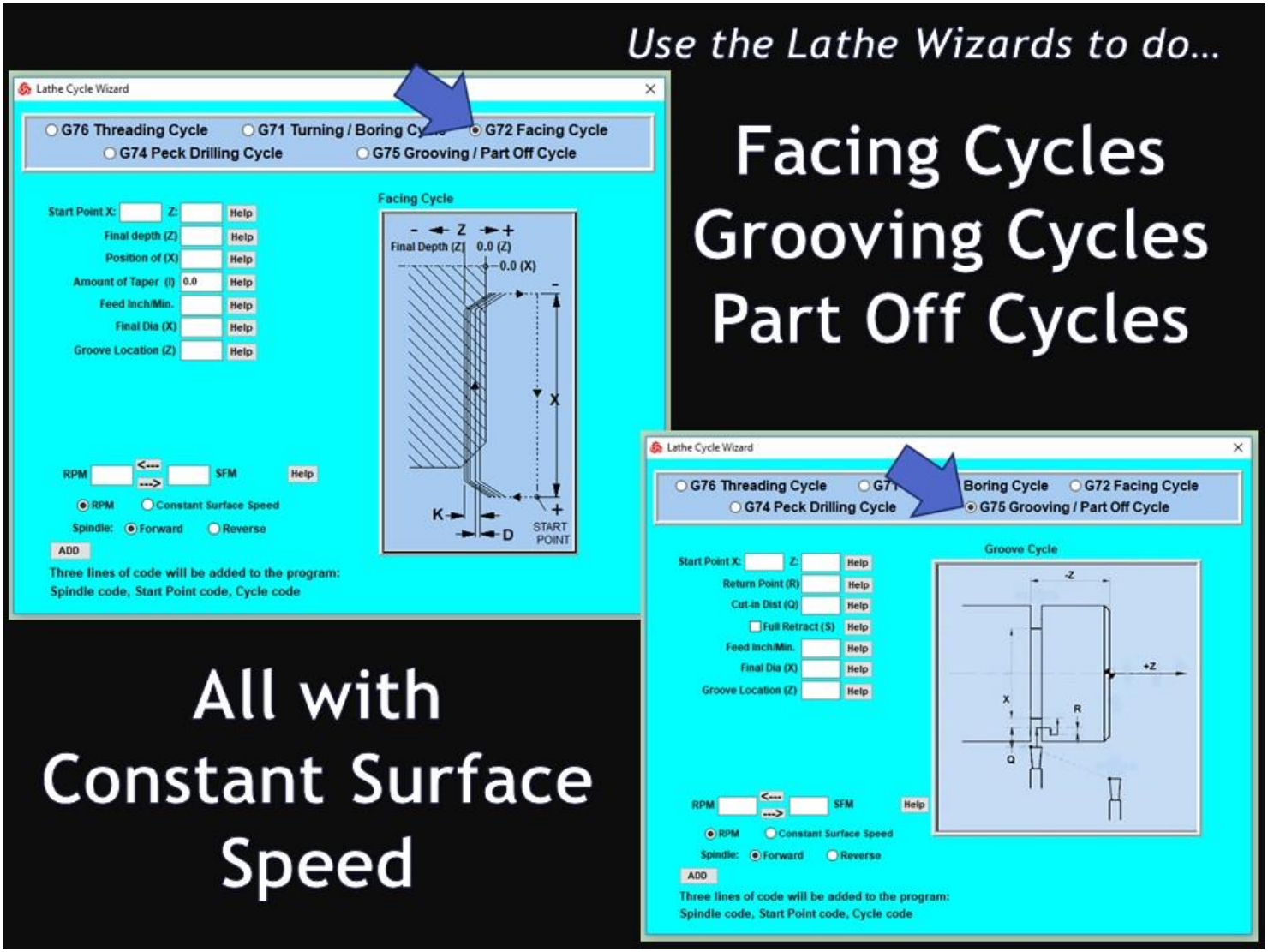


Use the Lathe Wizard Peck Drilling Cycle to help you program your drill applications or for face grooving



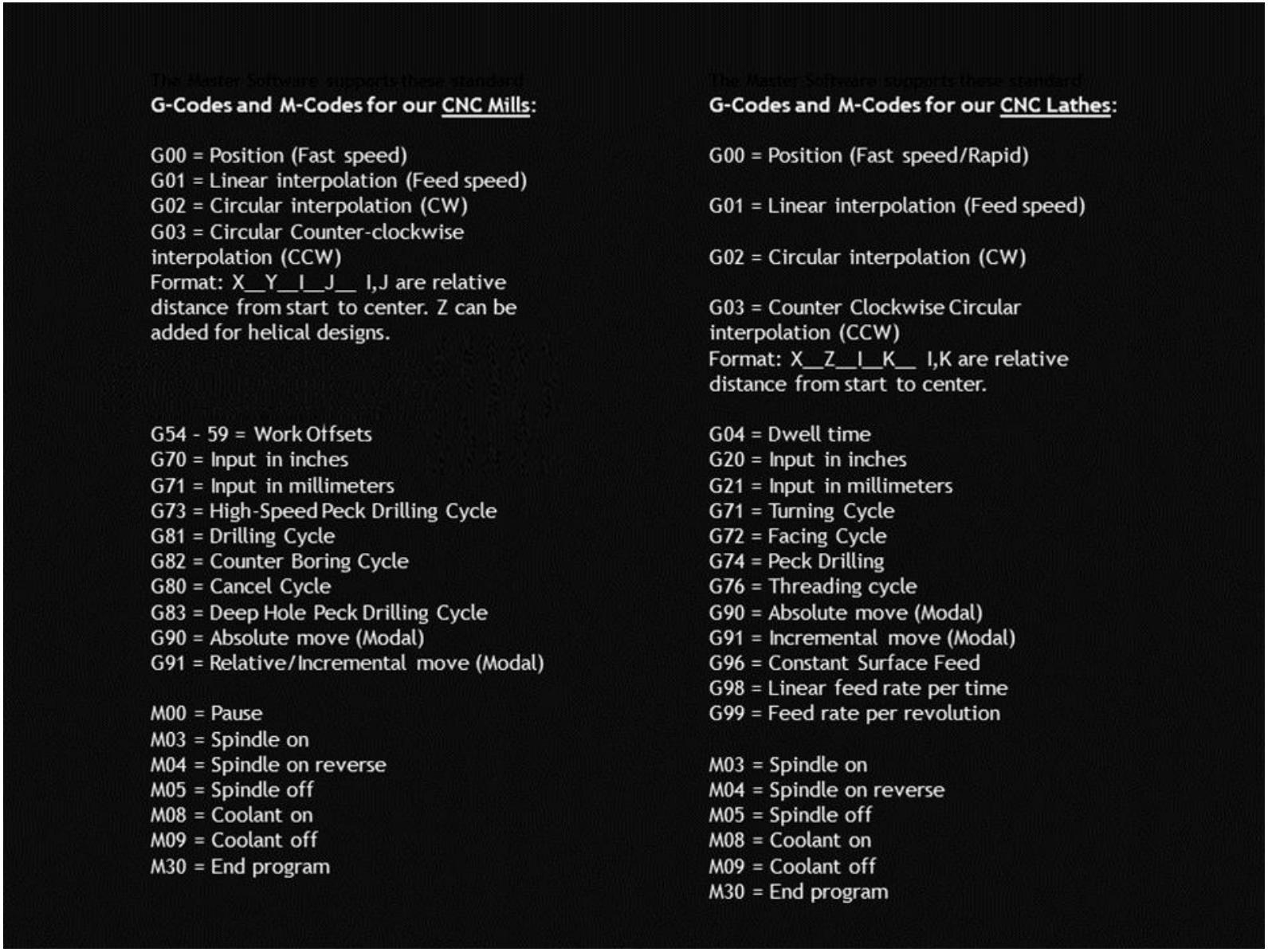
Facing / Grooving / Part Off Cycle Wizards – with Constant Surface Speed

These cycles can be used with Constant Surface Speed allowing the spindle speed to increase automatically as the diameter of the part decreases giving your application a consistent workpiece finish. With CSS built into the wizard, there is no need to break down the cycle into multiple paths and multiple spindle speed changes.



This is our list of supported G and M codes which can be found under Tools > G Code/ M Code List in the MX

If you plan to use a third-party CAM software to generate your tool path program, use a generic FANUC post processor and edit it to match our list of codes. As an option, we also sell Visual mill/turn CAM software which comes with a guaranteed post processor for our machines to easily generate your tool path programs based on your CAD drawings.



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.

G54 - 59 = Work Offsets
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)
G96 = Constant Surface Feed
G98 = Linear feed rate per time
G99 = Feed rate per revolution

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

Our pledge to you...

CNC MASTERS
SINCE 1990

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 first hand 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 mechanics, electronics, and explain the functions on the MX 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.

MX Software updates will also be made available to you for as long as the hardware in the control unit can sustain it.

Email us with any questions you may have at sales@cncmasters.com.