



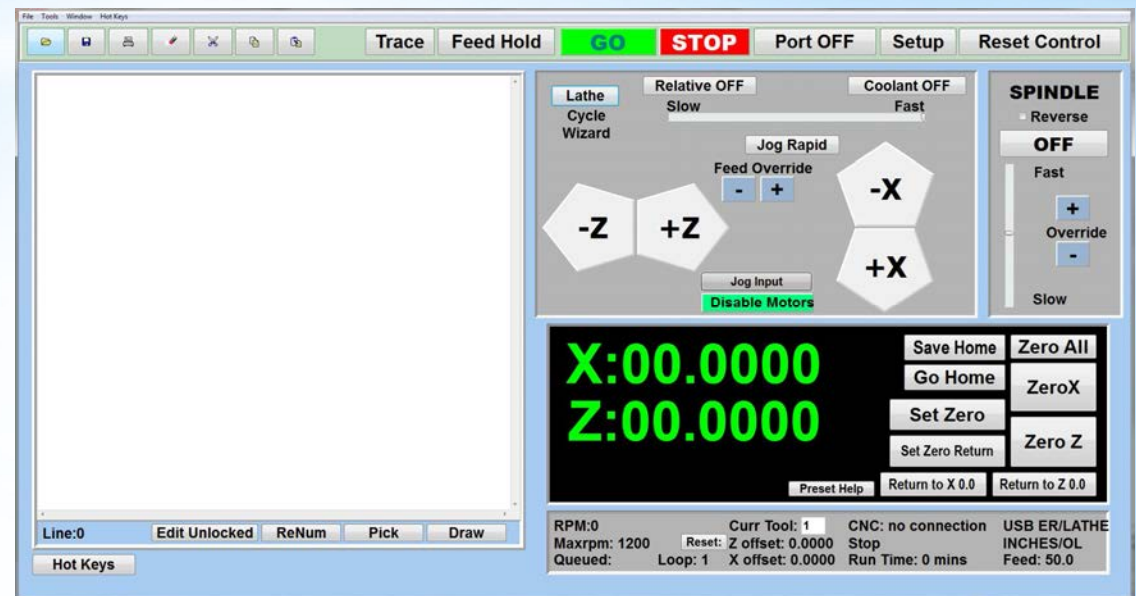
# The MX Software

*Use it on your CNC Mill...*



*Designed to seamlessly work with your CNC Masters machine.*

*Use it on your CNC Lathe...*





MasterMX Software Ver 1i File: C:\Users\CNC MASTERS\Desktop\MasterMX\Master MX files\4th axis sample.txt

File Tools Window Hot Keys

Trace Feed Hold **GO** **STOP**

M03 S1000  
 N01 G01 W360 X10 Z-0.258 F0.25  
 N02 G00 Z0.0  
 N03 G00 W0.0 X0.0  
 N04 G01 Z-0.1 F1  
 N05 G00 Z0.0  
 N06 G01 W90  
 N07 G01 Z-0.1 F1  
 N08 G00 Z0.0  
 N09 G01 W180  
 N10 G01 Z-0.1 F1  
 N11 G00 Z0.0  
 N12 G01 W270  
 N13 G01 Z-0.1 F1  
 N14 G00 Z0.0  
 N15 G00 W0.0  
 M30

Line:1 Edit Locked ReNum Pick Draw

Hot Keys

Mill Cycle Wizard Glossary

Relative OFF Slow

+Y +X -X -Y

Jog Input

+W +Z -W -Z

Jog Input

Disable Motors

Slow

X:10.125  
 Y:0.0000  
 Z:-0.1  
 W:0.00DG

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

Preset Help

RPM:0 Curr Tool: 1 CNC: no connection USB ER/MILL  
 Maxrpm: 1200 Reset: Z offset: 0.0000 Stop METRIC/OL  
 In Queue: Run Time: 0 mins Feed: 1270.0

\* 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.





You also have the optional coolant and spindle speed controls.

**File** **Tools** **Window** **Hot Keys**

New Ctrl+N  
Open Ctrl+O  
Save Ctrl+S  
SaveAs  
Exit

N04 G01 Z-0.1 F1  
N05 G00 Z0.0  
N06 G01 W90  
N07 G01 Z-0.1 F1  
N08 G00 Z0.0  
N09 G01 W180  
N10 G01 Z-0.1 F1  
N11 G00 Z0.0  
N12 G01 W270  
N13 G01 Z-0.1 F1  
N14 G00 Z0.0  
N15 G00 W0.0  
M30

**STOP** **Port OFF** **Setup** **Reset Control**

Relative OFF  
Slow

Coolant OFF  
Fast

Jog Rapid  
Feed Override  
- +

+Y  
-X  
Jog Input  
+X  
-Y

+Z  
-W  
Jog Input  
+W  
-Z

Disable Motors

**SPINDLE**  
☐ Reverse  
**OFF**  
Fast  
+  
Override  
-  
Slow

X:10.125  
Y:0.0000  
Z:-0.1  
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:

Curr Tool: 1  
Reset: Z offset: 0.0000

CNC: no connection  
Stop  
Run Time: 0 mins

USB ER/MILL  
METRIC/OL  
Feed: 1270.0

Line:1  
Edit Locked  
ReNum  
Pick  
Draw

Hot Keys

**Editor**

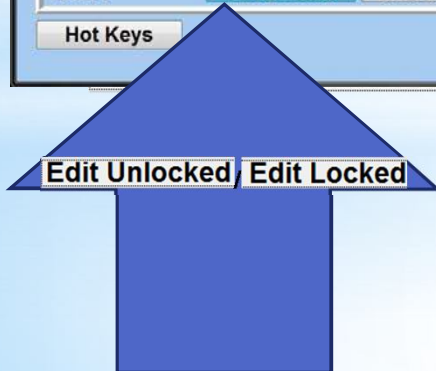
With a few clicks of the mouse or using touch screen technology, you can easily navigate through the Master interface importing saved programs into the Editor from the File drop down menu using standard windows features. You can type in a program or import CAM generated G-code tool paths into the Editor.

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.

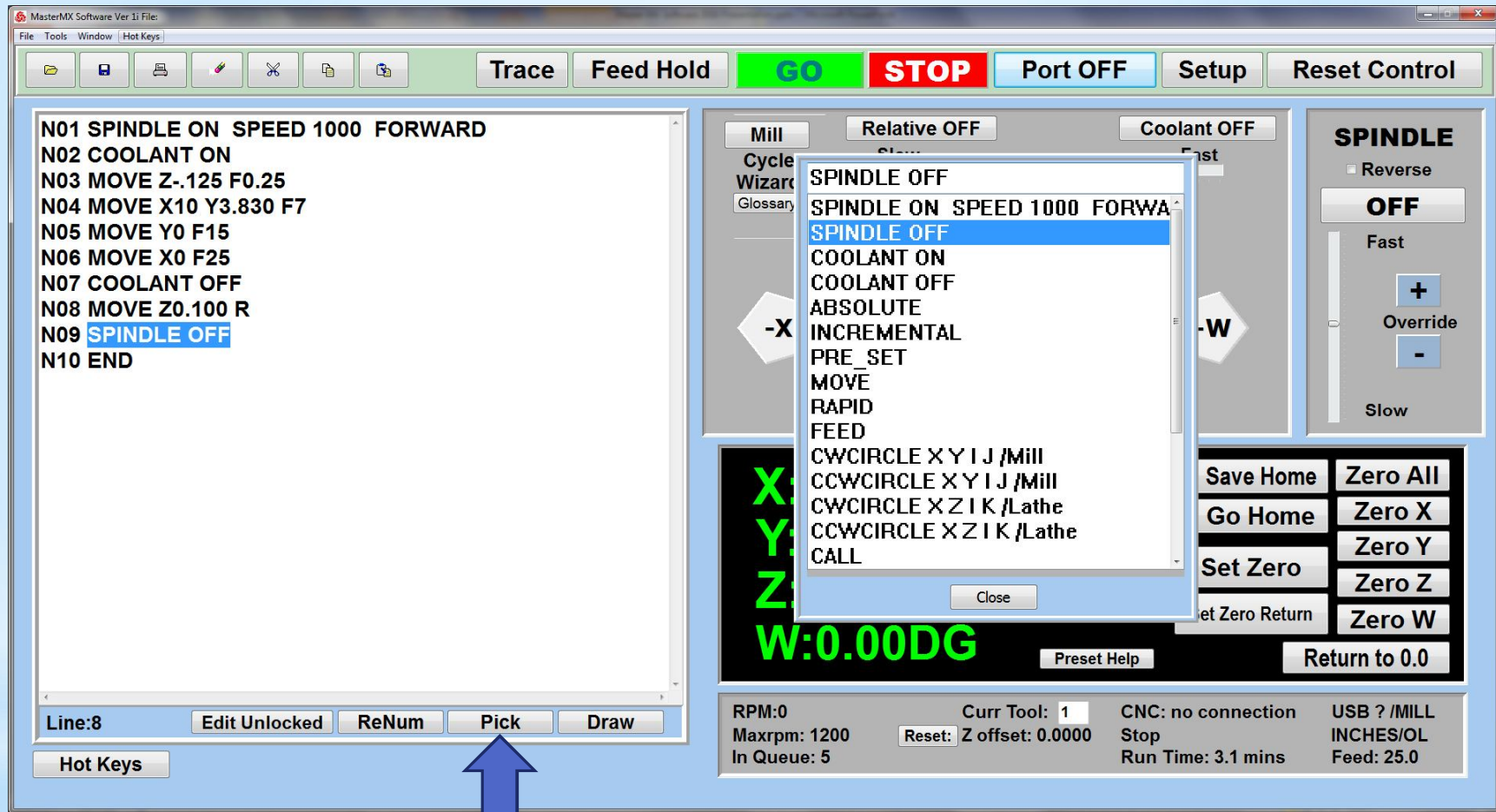




**Hot Keys** is an alternative method to easily control your machine using your hard or touch screen keyboard. One can jog the axes with the arrow keys, press P to pause a program, press S to turn Spindle On, G to run a program, Space Bar to Stop, J to record your individual movements one line at a time to create a program by teach mode.



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.

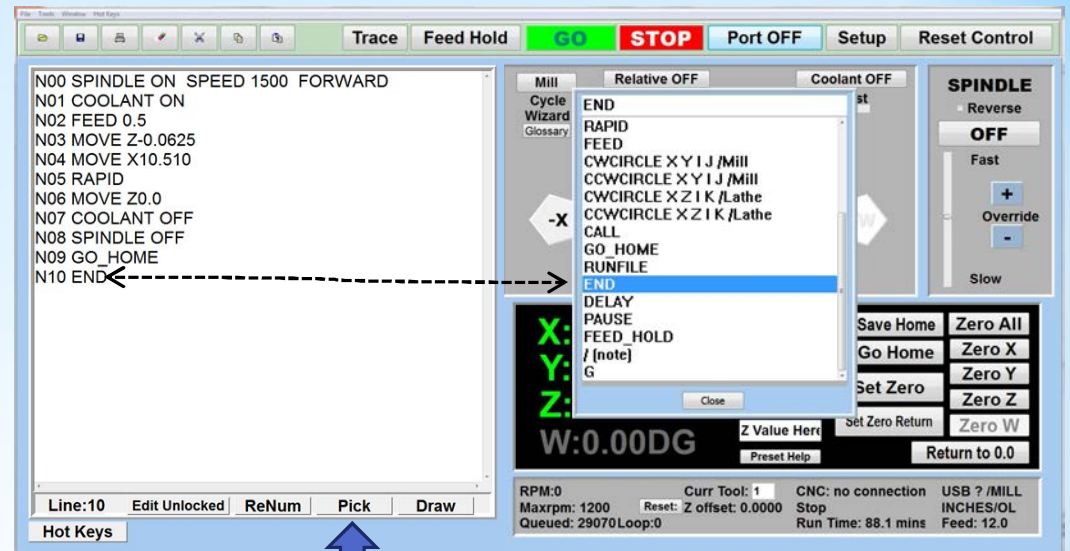


While in "Edit Unlocked" mode, you can write FANUC style G-codes directly into the Editor, or select commands off the "Pick" menu and write your tool path program in conversational mode such as what is written in the Editor box. You can even mix between conversation commands and G-codes in the same program.

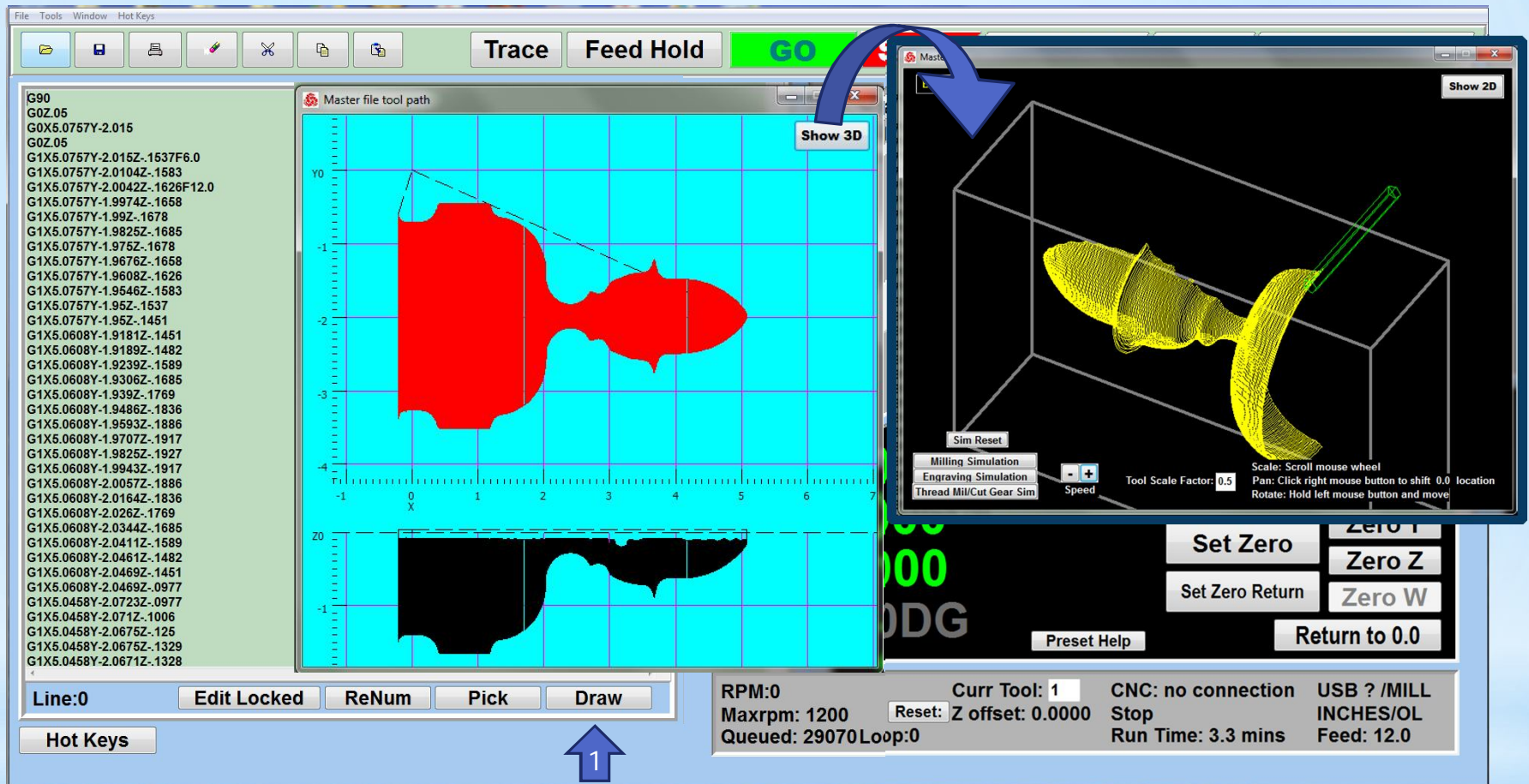




With the Pick Menu, you can easily write a simple program without G-codes. You also interchange between codes and conversational commands in the same program. Using a CAM is always best for complicated applications and the Master has no problem receiving an imported FANUC based program with the correct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

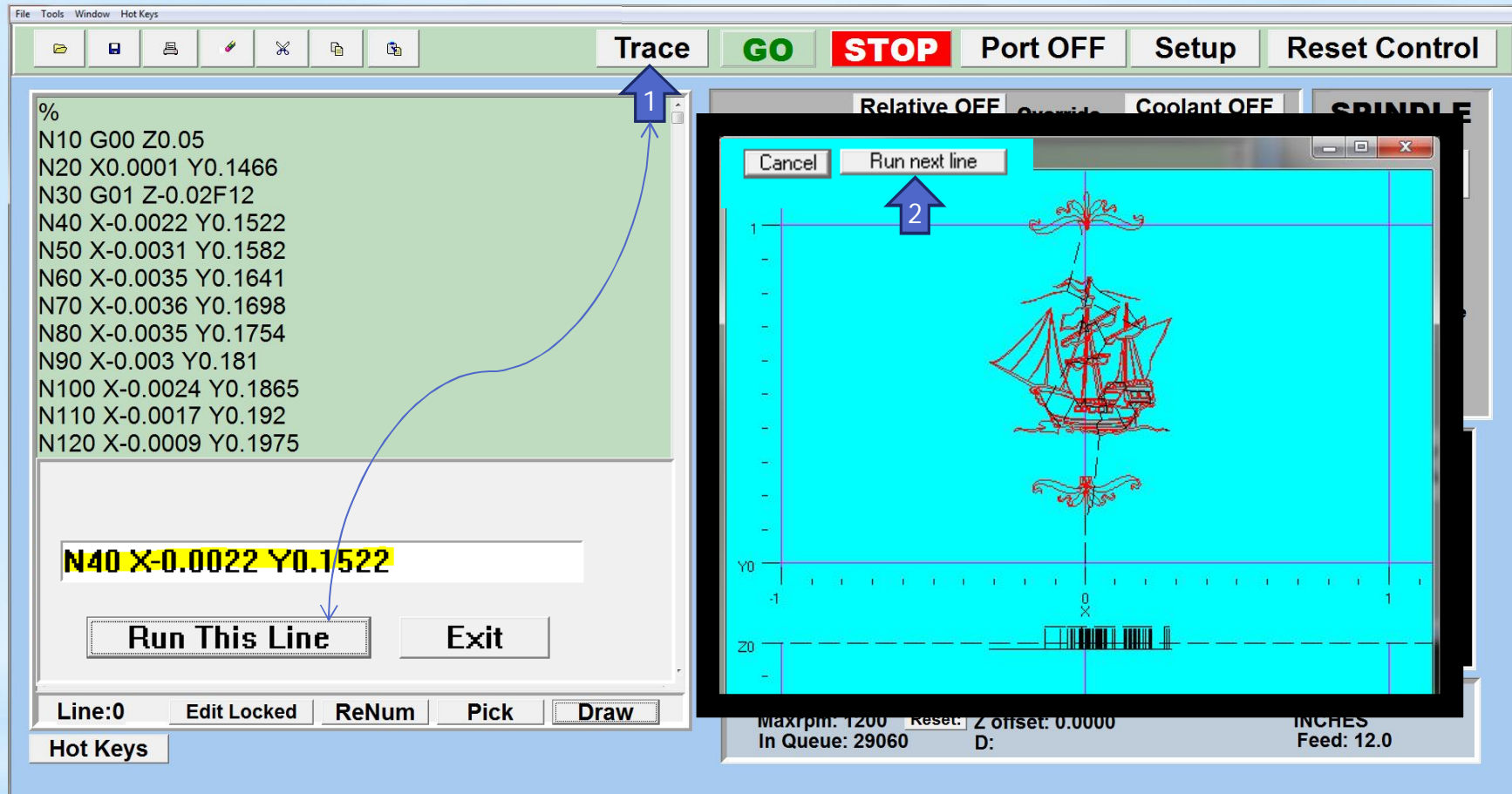


Pick Menu Features	Description
ABSOLUTE	Same as G90. The Master defaults to Absolute mode.
INCREMENTAL	Same as G91.
SPINDLE ON	Same as M03 for computer variable spindle control.
MOVE	Same as G00, G01. Move three or four axes simultaneously on the same tool path.
PRE_SET	Can't start from a 0.00 position? 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.
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.



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.





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.



**Tools** **Window** **Hot Keys**

Go From Line  
Beep End Prog.  
Delete Config.cn3  
Find USB  
Change Text Size

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

Mill Cycle Wizard Glossary

Relative OFF Slow Fast Coolant OFF

Jog Rapid

Feed Override - +

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

Jog Input

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 Preset Help

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

Line:0 Edit Locked ReNum Pick Draw

Hot Keys

RPM:0 Maxrpm: 1200 Queued: Curr Tool: 1 Reset: Z offset: 0.0000 Loop:0 CNC: no connection Done+ Run Time: 88.1 mins USB ? /MILL INCHES/OL Feed: 12.0

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.



The screenshot shows the CNC MASTERS software interface. The main window is divided into several sections. On the left, there's a list of G-code programs. The central area contains control buttons: 'Trace', 'Feed Hold', 'GO', and 'STOP'. To the right of these buttons is a 'Jog Input' section with directional buttons (+X, -X, +Y, -Y) and a 'Disable Motors' button. Below the 'Jog Input' section, a large display shows the current axis positions: X:0.0000, Y:0.0000, Z:0.0000, and W:0.00DG. To the right of this display, there are buttons for 'Save Home', 'Go Home', 'Set Zero', 'Set Zero Return', 'Zero All', 'Zero X', 'Zero Y', 'Zero Z', and 'Zero W'. At the bottom, a status bar displays various parameters: RPM:0, Maxrpm: 1200, Queued: Loop:0, Curr Tool: 1, Z offset: 0.0000, Done+, Run Time: 88.1 mins, and Feed: 12.0. A blue arrow labeled '1' points to the 'Disable Motors' button, and another blue arrow labeled '2' points to the 'Feed Hold' button.

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.





The screenshot displays the CNC MASTERS software interface. On the left, a G-code program is listed, with the current line highlighted in blue:

```

G90
G0Z.05
G0X5.0757Y-2.015
G0Z.05
G1X5.0757Y-2.015Z-.1537F6.0
G1X5.0757Y-2.0104Z-.1583
G1X5.0757Y-2.0042Z-.1626F12.0
G1X5.0757Y-1.9974Z-.1658
G1X5.0757Y-1.99Z-.1678
G1X5.0757Y-1.9825Z-.1685
G1X5.0757Y-1.975Z-.1678
G1X5.0757Y-1.9676Z-.1658
G1X5.0757Y-1.9608Z-.1626
G1X5.0757Y-1.9546Z-.1583
G1X5.0757Y-1.95Z-.1537
G1X5.0757Y-1.95Z-.1451
G1X5.0757Y-1.94Z-.1451
G1X-2.197Y-.5842Z-.0915
G0Z.05
G0X0Y0
END
  
```

Below the program list are buttons for "Line:0", "Edit Locked", "ReNum", "Pick", and "Draw".

At the top, there are control buttons: "Trace", "Feed Hold", "GO" (green), "STOP" (red), "Port OFF", "Setup", and "Reset Control".

The central jog control area includes a "Relative ON" toggle, a "Coolant OFF" button, and a "Slow" feed rate slider. Below these are directional jog buttons (-X, +X, -Y, +Y, -Z, +Z) and a "Disable Motors" button. A blue arrow points from the "Relative ON" toggle to the jog buttons.

On the right, the "SPINDLE" section shows a "Reverse" checkbox, a "Fast" speed selector, and an "OFF" button. Below this is an "Override" slider with "+" and "-" buttons.

At the bottom right, a status panel displays current coordinates in large green text:

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

Buttons for "Save Home", "Go Home", "Set Zero", "Set Zero Return", "Zero All", "Zero X", "Zero Y", "Zero Z", "Zero W", and "Return to 0.0" are also present.

The bottom status bar shows system information:

RPM:0 Maxrpm: 1200 Queued: Loop:0 Curr Tool: 1 Reset: Z offset: 0.0000 CNC: no connection Done+ Run Time: 88.1 mins USB ? /MILL INCHES/OL Feed: 12.0

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".



File Tools Window Hot Keys

Trace Feed Hold **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

Line:0 Edit Unlocked ReNum Pick Draw

Hot Keys

Mill

Cycle Wizard

Glossary

Relative OFF

Coolant OFF

Slow

Fast

Jog Rapid

Feed Override

-

+

+Y

-X

Jog Input

+X

-Z

Jog Input

+Z

-W

+W

Disable Motors

SPINDLE

☐ Reverse

OFF

Fast

+

Override

-

Slow

X:-0.3422

Y:0.1056

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

Queued:

Curr Tool: 1

Reset:

Z offset: 0.0000

Loop:0

CNC: no connection

Done+

Run Time: 88.1 mins

USB ? /MILL

INCHES/OL

Feed: 12.0

#### 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 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.



File Tools Window Hot Keys

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

Mill Cycle Wizard Glossary

Relative OFF Slow Fast Coolant OFF

Jog Feed Feed Override

+Y -X Jog Input +X -Y -W Jog Input +W -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 Preset Help

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

Line:0 Edit Unlocked ReNum Pick Draw

Hot Keys

RPM:0 Maxrpm: 1200 Queued: Loop:0 Curr Tool: 1 Z offset: 0.0000 Reset: CNC: no connection Done+ Run Time: 88.1 mins USB ? /MILL INCHES/OL Feed: 9.4

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.





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.



File Tools Window Hot Keys

Trace Feed Hold **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

Mill Cycle Wizard Glossary

Relative OFF Slow Fast

Coolant OFF

Jog Feed

Feed Override - +

+Y +X -X -Y

Jog Input

+Z +W -Z -W

Jog Input

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 Preset Help

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

Line:2 Edit Unlocked ReNum Pick Draw

Hot Keys

RPM:0 Maxrpm: 1200 Queued: Loop:0 Curr Tool: 1 Reset: Z offset: 0.0000 CNC: no connection Done+ Run Time: 88.1 mins USB ? /MILL INCHES/OL Feed: 9.4

**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.



File Tools Window Hot Keys

Trace Feed Hold **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

Mill Cycle Wizard Glossary

Relative OFF Slow Fast Coolant OFF

Jog Feed

Feed Override - +

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

Jog Input

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

Zero All Zero X Zero Y Zero Z Zero W

Return to 0.0

Line:9 Edit Locked ReNum Pick Draw

Hot Keys

RPM:0 Curr Tool: 1 CNC: no connection USB ? /MILL  
 Maxrpm: 1200 Z offset: 0.0000 Done+ INCHES/OL  
 Queued: Loop:0 Run Time: 88.1 mins Feed: 9.4

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.





The screenshot displays the CNC MASTERS software interface. The top menu bar includes File, Tools, Window, and Hot Keys. Below the menu is a toolbar with icons for file operations and machine control. The main interface is divided into several sections:

- Top Control Bar:** Contains buttons for Trace, Feed Hold, GO (green), STOP (red), Port OFF (blue), Setup, and Reset Control.
- Left Panel (Program Editor):** Shows a list of G-code lines (N01 to N21). Line N05 is highlighted with a yellow background and a blue arrow pointing to it. The code includes tool changes (T2, T3, T4) and coordinates (X, Y, Z).
- Tools Window:** A sub-window titled "Tools" with a "CAUTION!" label. It contains a table for tool offsets and a "Quick Tool Offset Return Set" section.
 

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
- Right Panel (Jog Feed):** Contains controls for Mill, Cycle Wizard, Glossary, Relative OFF, Slow, Fast, Coolant OFF, Jog Feed, Feed Override, Jog Input, and Jog Feed. It also has a "Disable Motors" button.
- Bottom Right (Status Bar):** Displays real-time data including RPM:0, Maxrpm: 1200, Curr Tool: 1, Z offset: 0.0000, CNC: no connection, USB ? /MILL, Run Time: 88.1 mins, and Feed: 9.4. It also includes buttons for Save Home, Go Home, Set Zero, Set Zero Return, Zero All, Zero X, Zero Y, Zero Z, Zero W, and Return to 0.0.

**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.



The screenshot displays the CNC MASTERS software interface. On the left, the 'Setup' window is open, showing various configuration options under the 'General' tab. The main control panel on the right includes a top bar with 'Hold', 'GO', 'STOP', 'Port OFF', 'Setup', and 'Reset Control' buttons. Below this is a central control area with 'Jog' buttons for X, Y, Z, and W axes, a 'Feed Override' slider, and a 'Disable Motors' button. To the right of the central area is a 'SPINDLE' control section with 'Reverse', 'OFF', 'Fast', 'Override', and 'Slow' buttons. At the bottom, a status bar displays real-time data: 'X:0.0000', 'Y:0.0000', 'Z:0.0000', 'W:0.00DG', 'RPM:0', 'Maxrpm: 1200', 'Queued:', 'Loop:0', 'Curr Tool: 1', 'Reset: Z offset: 0.0000', 'CNC: no connection', 'Done+', 'Run Time: 88.1 mins', 'USB ? /MILL', 'INCHES/OL', and 'Feed: 9.4'. A 'Preset Help' button is also visible in the status bar area.

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.



File Tools Window Hot Keys

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

General Tools Auto Tool

Scale Calibration X 0.5000 Y 0.5000 Z 0.5000

XRapid 50 ZRapid 30

Scale X1.0 Y1.0 Z1.0

Calibration Factor X1.0 Y1.0 Z1.0

Draw Delay (0-99) 0 Max RPM 1200

☐ Mill ☒ **Lathe X=DIA** ☐ 3D Printing

☒ Inches ☐ 4th Axis Active

☐ Millimeters ☐ C-Type 4th Axis

Spindle delay in milliseconds 4000

☐ Closed-loop Autho. Code HX92915

Cancel Save

Lathe Cycle Wizard

Relative OFF Slow Fast Coolant OFF

Jog Rapid Feed Override - + -X +X

Jog Input Disable Motors

**SPINDLE**

☐ Reverse

**OFF**

Fast + Override - Slow

**X:00.0000**  
**Z:00.0000**

Save Home Zero All  
Go Home ZeroX  
Set Zero Zero Z  
Set Zero Return  
Preset Help Return to X 0.0 Return to Z 0.0

Line:0 Edit Locked ReNum Pick Draw

Hot Keys

RPM:0 Curr Tool: 1 CNC: no connection USB ? /MILL  
Maxrpm: 1200 Reset: Z offset: 0.0000 D : INCHES/OL  
Queued: Loop: 1 X offset: 0.0000 Run Time: 0 mins Feed: 50.0

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





File Tools Window Hot Keys

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

Mill Cycle Wizard Glossary

Relative OFF Slow Fast Coolant OFF

Jog Feed Feed Override - + +Z -W Jog Input +W -Z -X Jog Input +X -Y

Disable Motors

**SPINDLE**  
☐ Reverse  
**OFF**  
Fast + Override - Slow

**Mill Cycle Wizard**

Rectangular Pocket Circular Pocket Cir. Hole Pattern  
Thread Milling Cut Gear Rigid Tapping (Add-On) Rec. Hole Pattern

Will use current Tool dia.  
☐ End with 0.010" finish pass  
☐ Finish with Climb Cutting

Start Point X: 0 Y: 0 Z: 0 Step Over 0

Step Down 0 Total Depth Z: 0 Length (X): 0 Width (Y): 0 Feed 0

Add to File Verify Path

Hot Keys

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 Queued: Curr Tool: 1 Z offset: 0.0000 Loop:0 CNC: no connection Done+ Run Time: 88.1 mins USB ? /MILL INCHES/OL Feed: 9.4

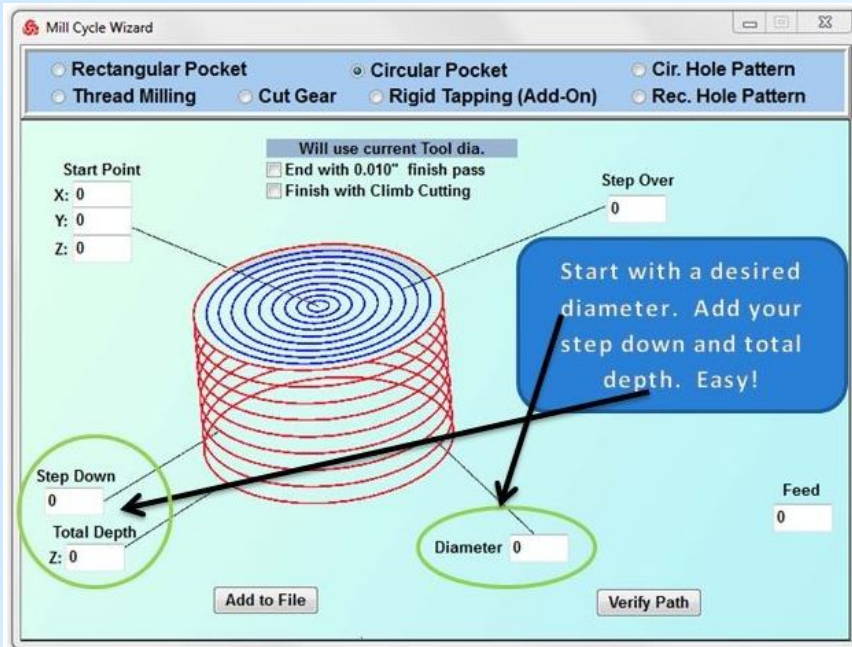
The Cycle Wizards makes it easy to create a simple tool path without needing to use a CAD and CAM software.

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.

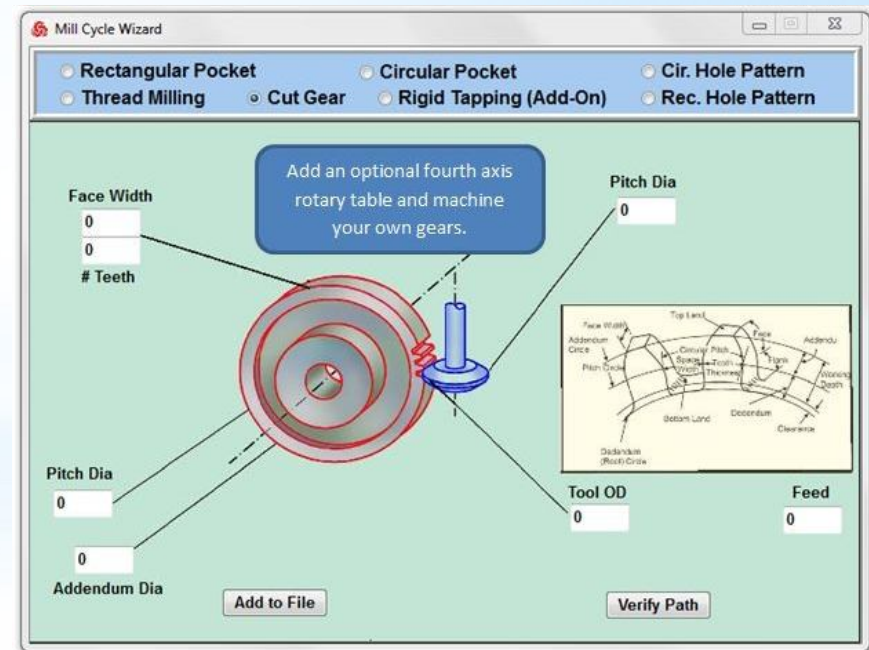


*Use the Mill Wizards to do...*

# Circle Pocketing

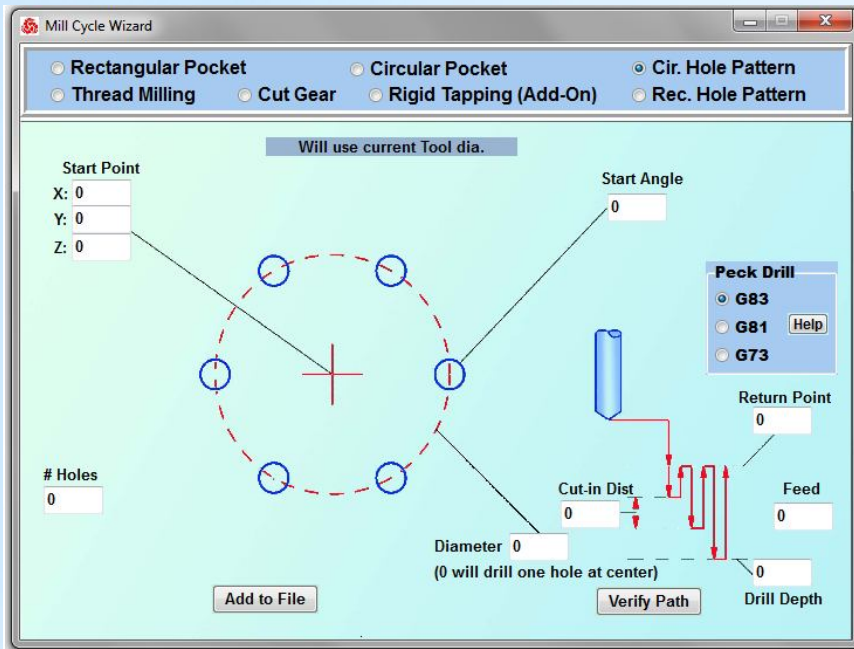


# Gear Cutting

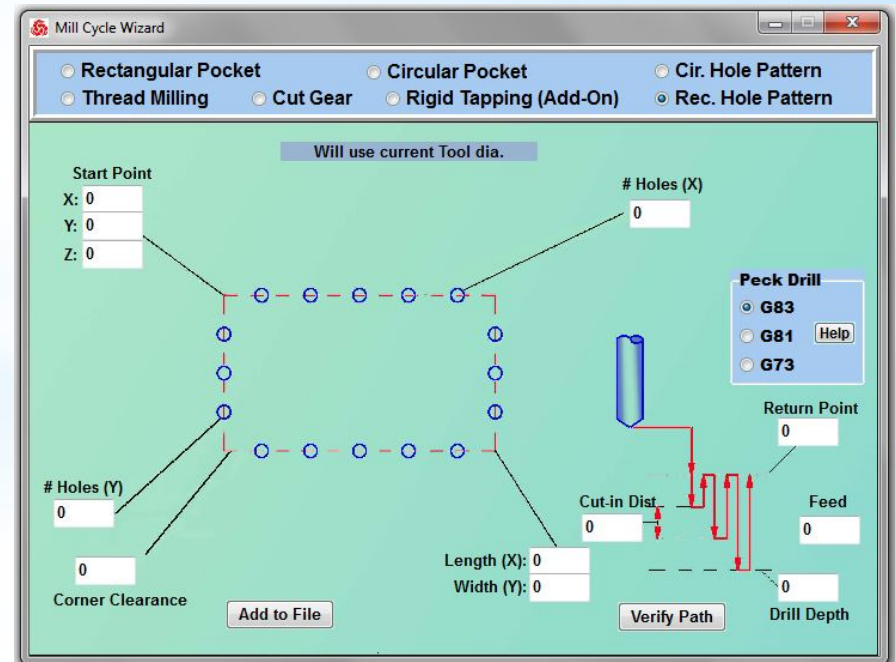


*Use the Mill Wizards to do...*

# Circular Hole Drilling Pattern



# Rectangular Hole Drilling Pattern

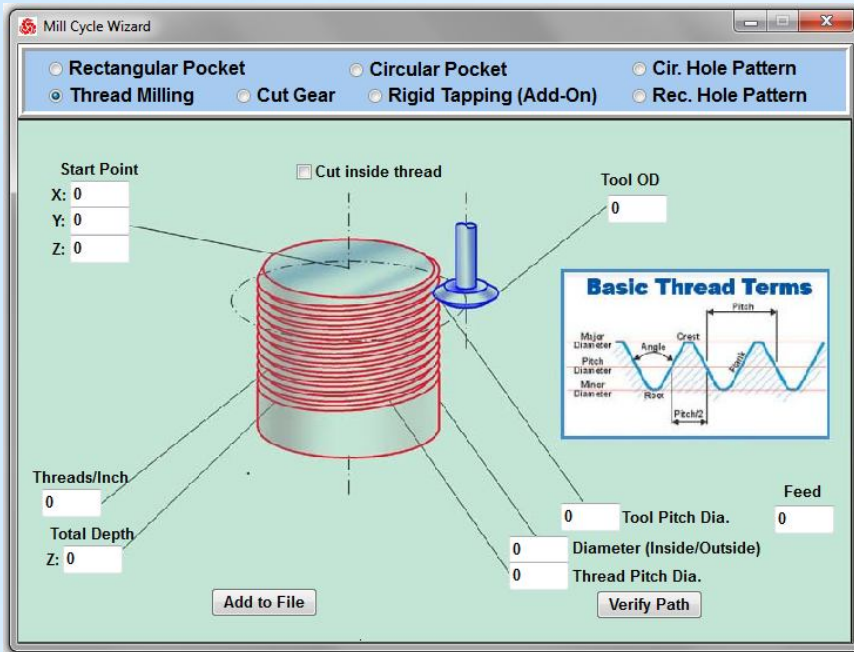




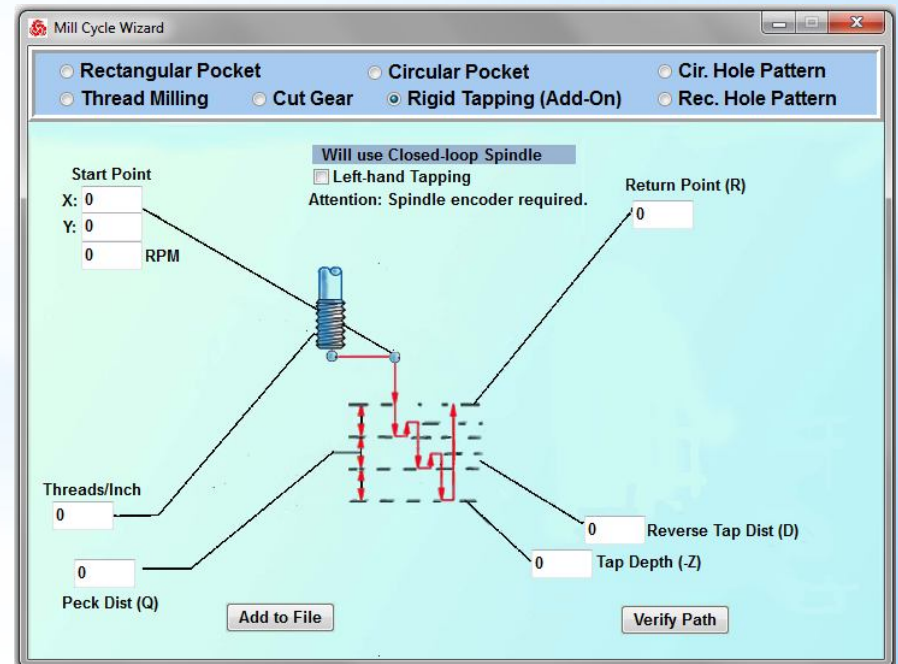


*Use the Mill Wizards to do...*

# Thread Milling



# Rigid Tapping





# CNC MASTERS

*Use the Lathe Wizards to do...*

Lathe Cycle Wizard

☒ G76 Threading Cycle   ☐ G71 Turning/Boring Cycle   ☐ G72 Facing Cycle

Start Point X:  Z:

Final Diameter (X)

Position of (Z)

Amount of Taper (I) 0.0

Total Depth (K)

First Pass Depth (D)

Thread Insert Angle (A) 60

Threads per inch

Cut type: ☒ External   ☐ Internal

☒ Right Hand   ☐ Left Hand

Compound Infeed (from chart) P2

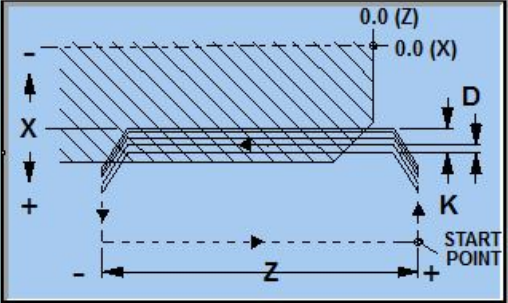
RPM    SFM

☒ RPM   ☐ Constant Cutting Speed

Spindle: ☒ Forward   ☐ Reverse

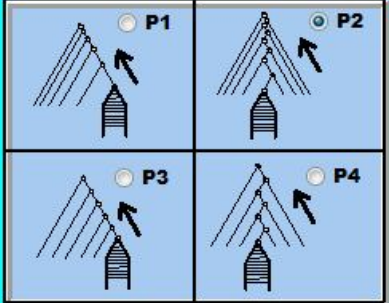
Three lines of code will be added to the program:  
Spindle code, Start Point code, Cycle code

**Right Hand Thread Cutting Cycle**



**Select threading infeed method**

<input type="radio"/> P1	<input checked="" type="radio"/> P2
<input type="radio"/> P3	<input type="radio"/> P4



# Threading Cycles



# CNC MASTERS

*Use the Lathe Wizards to do...*

Lathe Cycle Wizard

☐ G76 Threading Cycle ☒ G71 Turning/Boring Cycle ☐ G72 Facing Cycle

Start Point X:  Z:

Final Diameter (X)

Position of (Z)

Amount of Taper (I) 0.0

Total Depth (K)

First Pass Depth (D)

Feed Inch/Min.

Cut type: ☒ External ☐ Internal

RPM    SFM

☒ RPM ☐ Constant Cutting Speed

Spindle: ☒ Forward ☐ Reverse

Three lines of code will be added to the program:  
Spindle code, Start Point code, Cycle code

Turning Cycle

# Turning & Boring Cycles





# CNC MASTERS

*Use the Lathe Wizards to do...*

Lathe Cycle Wizard

☐ G76 Threading Cycle ☐ G71 Turning/Boring Cycle ☒ G72 Facing Cycle

Start Point X:  Z:

Final depth (Z)

Position of (X)

Amount of Taper (I) 0.0

Total Depth (K)

First Pass Depth (D)

Feed Inch/Min.

RPM    SFM

☒ RPM ☐ Constant Cutting Speed

Spindle: ☒ Forward ☐ Reverse

Three lines of code will be added to the program:  
Spindle code, Start Point code, Cycle code

Facing Cycle

## Facing Cycles



**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!***



**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.***





*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](mailto:sales@cncmasters.com).