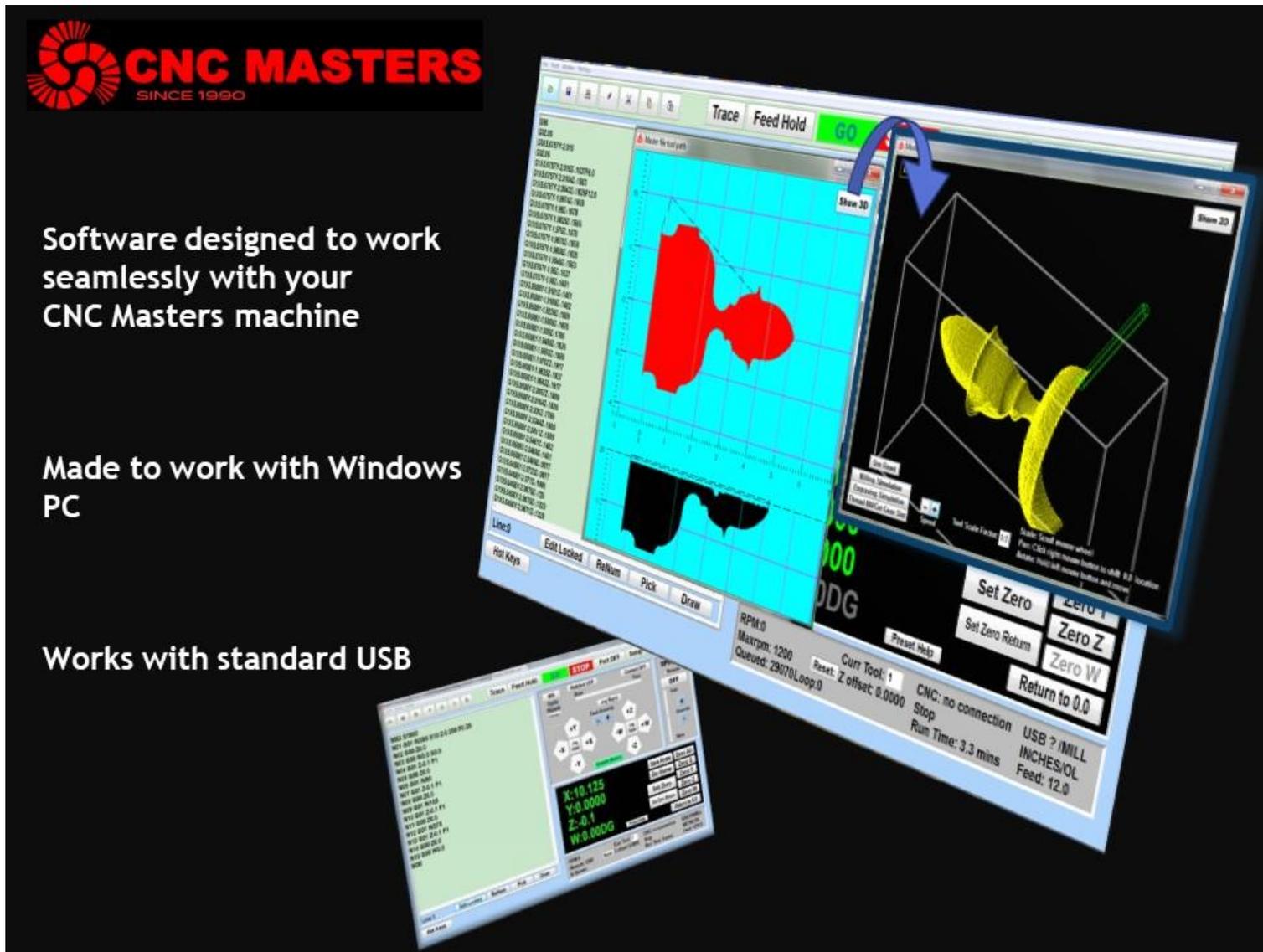


MX Software – Easy to Use, Easy to Learn – Included with your machine purchase

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.



CNC MASTERS
SINCE 1990

Software designed to work seamlessly with your CNC Masters machine

Made to work with Windows PC

Works with standard USB

The screenshot displays the MX software interface. On the left, a 2D view shows a red part on a cyan grid. On the right, a 3D view shows a yellow part with a green tool bit. The interface includes a 'Trace Feed Hold GO' control bar at the top, a 'Show 3D' button, and a 'Set Zero' panel with buttons for 'Zero X', 'Zero Y', 'Zero Z', and 'Zero W'. A status panel at the bottom right shows 'RPM: 0', 'Maxrpm: 1200', 'Queued: 29170(Loop:0)', 'Curr Tool: 1', 'Reset: Z offset: 0.0000', 'CNC: no connection', 'Run Time: 3.3 mins', and 'USB ? /MILL INCHES/OL Feed: 12.0'. A smaller window in the bottom left shows a list of lines and a coordinate display: 'X:10.125', 'Y:0.0000', 'Z:-0.1', 'W:0.0000'.

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!

Clutter free interface....

The software is engineered for the machine so you don't have to fiddle with detailed complicated configuration that can be overwhelming.



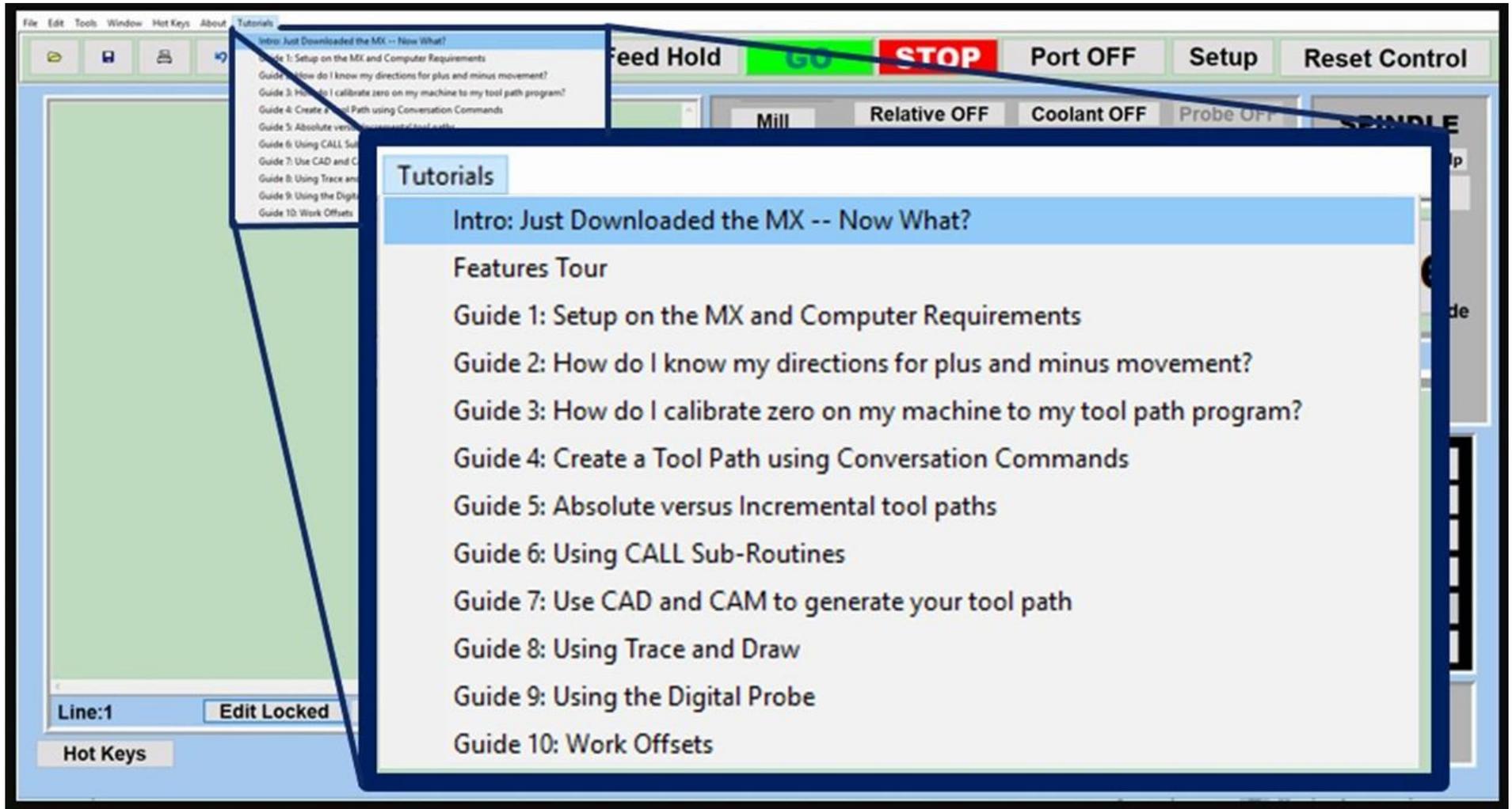
Baud Rate?
Timeout?
Port Num?
Kernel Speed?
Step and Direction?
Debounce Interval?
Enhanced Pulsing?
Slave Axis?

Calibration?

***Load the software...
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.

The screenshot displays the MasterMX software interface. The top menu bar includes File, Edit, Tools, Hot Keys, About, and Tutorials. The File menu is open, showing options like New, Open, Save, SaveAs, and Exit. The Edit menu is also open, showing options like Undo, Cut, Copy, Paste, Find..., Find Next, Replace..., and Change Text Size. The main editor area shows a G-code program with lines N06 through M30. The bottom status bar includes buttons for Edit Locked, ReNum, Pick, and Draw. The right side of the interface features a control panel with buttons for TOP, Port OFF, Setup, and Reset Control. It also includes jog controls for X, Y, Z, and W axes, a Feed Override section, and a SPINDLE control section. The bottom right corner displays real-time coordinates (X:10.125, Y:0.0000, Z:-0.1, W:0.00DG) and various system status indicators.

File Edit Tools Hot Keys About Tutorials

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

Edit Tools Hot Keys About

- Undo Ctrl+Z
- Cut Ctrl+X
- Copy Ctrl+C
- Paste Ctrl+V
- Find... Ctrl+F
- Find Next F3
- Replace... Ctrl+H
- Change Text Size Ctrl+T

N06 G01 W50
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

TOP Port OFF Setup Reset Control

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

Disable Motors

SPINDLE Reverse OFF Fast + Override - 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
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

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 a 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.

The screenshot displays a CNC control software interface. At the top, a status bar includes buttons for 'Trace', 'Feed Hold' (circled in red), 'GO', 'STOP', 'Port OFF', 'Setup', and 'Reset Control'. The main window is divided into several sections:

- Program Editor:** A list of G-code lines (N01 to N25). Line N04 is highlighted in blue and labeled 'PAUSE'. A blue arrow points to this line.
- Machine Controls:** Includes 'Mill Cycle Wizard', 'Relative OFF', 'Coolant OFF', 'Probe OFF', 'MICRO FEED' (0.0), 'Jog Rapid', and 'Feed Override' buttons. It also features directional jog buttons (+Y, -Y, +X, -X, +Z, -Z, +W, -W) and a 'Disable Motors' button.
- SPINDLE Control:** A vertical slider with 'OFF', 'Fast', 'Override', and 'Slow' positions. A 'Reverse' checkbox is also present.
- Position Display:** Shows current coordinates: X:0.0000, Y:0.0000, Z:0.0000, and W:0.00DG.
- Zeroing Buttons:** A grid of buttons for 'Save Home', 'Go Home', 'Set Zero', 'Set Zero Return', 'Zero All', 'Zero X', 'Zero Y', 'Zero Z', and 'Zero W'.
- System Status:** Displays 'RPM:0', 'Maxrpm: 1200', 'Queued:', 'Curr Tool: 1', 'D:', 'Reset: Z offset: 0.0000', 'Loop: 1', 'CNC: no connection', 'Run Time: 0 mins', and 'USB ? /MILL INCHES/OL Feed: 50.0'.

At the bottom, there are buttons for 'Line:4', 'Edit Unlocked', 'ReNum', 'Pick', 'Draw', and 'Hot Keys'.

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.

The screenshot displays the MasterMX Software Ver 11 File interface. A 'Hot Keys' menu is open, listing the following key assignments:

- Hot Keys: F7
- Go: F8
- Port: F9
- Edit: F5
- Stop: Space Bar
- Jog Input: J

The main interface includes several control panels:

- STOP** (red button), **Port OFF**, **Setup**, **Reset Control**
- Relative OFF** (Slow to Fast slider), **Coolant OFF** (Fast slider)
- Jog Rapid** button
- Feed Override** (-, + buttons)
- Jog Input** buttons: +Y, -X, +X, -Y, +Z, -W, +W, -Z
- Disable Motors** (green button)
- SPINDLE** section: Reverse, **OFF** button, Fast/Slow slider, **+** Override, **-** Override, Slow
- Position Display** (green text): X:0.0000, Y:0.0000, Z:0.0000, W:0.00DG
- Buttons:** 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
- Status Bar:** RPM:0, Maxrpm: 1200, In Queue: , Curr Tool: 1, Reset: Z offset: 0.0000, CNC: no connection, D: , Run Time: 0 mins, USB ? /MILL, METRIC/OL, Feed: 1270.0
- Bottom Panel:** Line:0, Edit Locked, ReNum, Pick, Draw, **Hot Keys** (highlighted), Go:G, Port:O, Pause:P, Home:H, Rapid/Feed:F, ZeroAll:A, Zerox:X, Zeroy:Y, Zeroz:Z, Zerow:W, Spindle:S, Reset:R

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.

The screenshot displays the MasterMX Software Ver 1.1 File interface. The main window is divided into several sections:

- Top Panel:** Contains control buttons: Trace, Feed Hold, GO (green), STOP (red), Port OFF, Setup, and Reset Control.
- Editor Box (Left):** Displays a list of G-codes:

```
N01 SPINDLE ON SPEED 1000 FORWARD
N02 COOLANT ON
N03 MOVE Z-.125 F0.25
N04 MOVE X10 Y3.830 F7
N05 MOVE Y0 F15
N06 MOVE X0 F25
N07 COOLANT OFF
N08 MOVE Z0.100 R
N09 SPINDLE OFF
N10 END
```

Line 9, "SPINDLE OFF", is highlighted in blue. A dashed arrow points from this line to the Pick menu.
- Pick Menu (Center):** A dropdown menu is open, listing various commands:
 - SPINDLE OFF
 - SPINDLE ON SPEED 1000 FORWA
 - SPINDLE OFF (highlighted)
 - COOLANT ON
 - COOLANT OFF
 - ABSOLUTE
 - INCREMENTAL
 - PRE_SET
 - MOVE
 - RAPID
 - FEED
 - CWCIRCLE X Y I J /Mill
 - CCWCIRCLE X Y I J /Mill
 - CWCIRCLE X Z I K /Lathe
 - CCWCIRCLE X Z I K /Lathe
 - CALLA "Close" button is at the bottom of the menu.
- Right Panel:** Contains machine status controls:
 - Buttons: Mill, Relative OFF, Coolant OFF.
 - SPINDLE section: Reverse checkbox, OFF button, Fast/Slow slider, Override (+/-) buttons.
 - Home/Zero buttons: Save Home, Go Home, Set Zero, Set Zero Return, Zero All, Zero X, Zero Y, Zero Z, Zero W.
 - Return to 0.0 button.
- Bottom Panel:** Contains status information:
 - Line:8, Edit Unlocked, ReNum, Pick, Draw buttons.
 - Hot Keys button.
 - Machine status: RPM:0, Maxrpm: 1200, In Queue: 5, Curr Tool: 1, Reset: Z offset: 0.0000, CNC: no connection, Stop, Run Time: 3.1 mins, USB ? /MILL INCHES/OL, Feed: 25.0.

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 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.

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.

The screenshot displays the MasterMX software interface. On the left, a list of G-code commands is shown, starting with M03 S500 and ending with N210 X0.0008 Y0.2481. The main window shows a 2D tool path drawing on a grid, with the path highlighted in red. Below the drawing, there are buttons for 'Line:0', 'Edit Locked', 'ReNum', 'Pick', and 'Draw'. A 'Hot Keys' button is also present. On the right, a 3D simulation window titled 'MasterMX' is open, showing a yellow 3D model of a part being milled by a green tool. The simulation window includes a 'BETA' label, a 'Show 2D' button, and various controls like 'Sim Reset', 'Milling Simulation', 'Engraving Simulation', and 'Thread Mill/Cut Gear Sim'. It also features a 'Speed' control with minus and plus buttons, a 'Tool Scale Factor' of 0.5, and instructions for scaling, panning, and rotating. At the bottom right, there are buttons for 'Set Zero', 'Zero Y', 'Zero Z', 'Set Zero Return', 'Zero W', 'Return to 0.0', and 'Preset Help'. The bottom status bar shows 'RPM:0', 'Maxrpm: 1200', 'Queued: 29070Loop:0', 'Curr Tool: 1', 'Reset: Z offset: 0.0000', 'CNC: no connection', 'USB ? /MILL', 'Stop', 'Run Time: 3.3 mins', and 'Feed: 12.0'.

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.

The screenshot displays a CNC control interface with a menu bar (File, Edit, Tools, Window, Hot Keys, About, Tutorials) and a toolbar containing buttons for Trace, Feed Hold, GO, STOP, Port OFF, Setup, and Reset Control. The main window shows a G-code program with the following lines:

```
M03 S500
N10 G00 Z0.0500
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 X-0.0022 Y0.2199
```

The line **N40 X-0.0022 Y0.1522** is highlighted in yellow. Below the program, there are buttons for "Run This Line" and "Exit". A blue arrow labeled "1" points from the "Trace" button in the toolbar to the "Run This Line" button.

The right side of the interface shows a "Master file tool path" window with a cyan background. It displays a red wireframe model of a ship's hull. The window has a "Cancel" button and a "Run next line" button. A blue arrow labeled "2" points from the "Run next line" button to the "GO" button in the main toolbar. The "GO" button is highlighted in green. The window also shows a coordinate system with X and Y axes and a "GO" button at the bottom right.

At the bottom of the interface, there is a status bar with the following information:

- Line:1
- Edit Locked
- ReNum
- Pick
- Draw
- Hot Keys
- RPM:0
- Maxrpm: 1200
- Queued: 2908Loop:0
- Curr Tool:1
- Reset: Z offset: 0.0000
- D :
- CNC: no connection USB ? /MILL
- Run Time: 3.3 mins
- INCHES/OL
- Feed: 12.0

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.

The screenshot displays a CNC control software interface with the following components:

- Top Bar:** Includes menu items (File, Edit, Tools, HotKeys, About, Tutorials) and control buttons: Trace, Feed Hold, GO (green), STOP (red), Port OFF, Setup, and Reset Control.
- Program Editor:** A text area containing G-code. Line 12 is highlighted in blue: `N605 G01 X0.3841 Y0.3288 Z-0.0966`. Below the editor are buttons for Edit Locked, ReNum, and Draw.
- Machine Control Panel:** Features a Mill Cycle Wizard, Relative OFF, Coolant OFF, and Probe OFF indicators. It includes a speed control slider (Slow to Fast), MICRO FEED (0.0), Jog Rapid, and Feed Override (-, +) buttons. Directional jog buttons (+Y, -Y, +X, -X, +Z, -Z, +W, -W) are arranged around a central Jog Input button. A Disable Motors button is at the bottom.
- SPINDLE Control:** Includes a Reverse checkbox, a Help button, and a speed control slider (Fast to Slow) with + and - Override buttons.
- Readout (R0) Panel:** Displays real-time coordinates in green: X:10.125, Y:0.0000, Z:-0.1, and W:0.00DG. It also contains buttons for 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:** Shows RPM:0, Maxrpm: 1200, Queued: Loop: 1, Curr Tool: 1, Reset, Z offset: 0.0000, CNC: no connection, Run Time: 0 mins, and USB ER/MILL INCHES/OL Feed: 50.0.

Three blue arrows with numbers 1, 2, and 3 point to the Readout panel, the Line:12 status bar, and the Run Time status bar, respectively.

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.

The screenshot displays a CNC control interface with a menu bar (File, Edit, Tools, Hot Keys, About, Tutorials) and a toolbar (Trace, Feed Hold, GO, STOP, Port OFF, Setup, Reset Control). The main window shows a G-code program with lines N50 through N240. A red dialog box titled "Go From Line: N 170" is open, with a warning: "Warning! Turn spindle on manually first." and buttons for "GO" and "Cancel". An information window is also open, providing instructions:

Information

"Go From Line" will start machining from where you hit the [GO] in the "Go From Line" command box and proceed with the rest of the program from there. The line that you are starting from MUST display a move command (G-code). Ensure that all lines are numbered by pressing [ReNum] below the Editor before inputting your desired starting line number.

To move to the "Go From Line" start position with ease:

1. Drive your cutter to your 0.0 position. You may use [Return to 0.0].
2. Turn Relative on by pressing [Relative OFF].
3. Use the "Relative" feature to move from 0.0 to the last XYZ position before desired starting line.

Example:
N99 G01 X4.3581 Y-0.0622 Z-0.048 F3
N100 G01 X5.6066 Y-0.0719 Z-0.048 F3

To "Go From Line" N100, start at 0.0 and use "Relative" to move your cutter to the position at N99.
Your cutter will now be in position to begin cutting at N100. Remember to start the spindle manually, then hit [GO].

IMPORTANT!
The "Go From Line" command will NOT work within a cycle or subroutine.

OK

The interface also features a "Mill Cycle Wizard" section with "Relative OFF", "Coolant OFF", and "Probe OFF" indicators, along with "Jog Rapid" and "Feed Override" controls. A "SPINDLE" section includes "Reverse" and "OFF" buttons, and a speed control slider. A bottom panel contains 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". The status bar at the bottom shows "Queued: Loop: 1" and "CNC: no connection", "Run Time: 0 mins", "USB ? /MILL INCHES/OL", and "Feed: 50.0".

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”.

The screenshot shows a CNC control software interface with the following elements:

- Top Bar:** File, Edit, Tools, Hot Keys, About, Tutorials. Buttons for Trace, Feed Hold, GO (green), STOP (red), Port OFF, Setup, and Reset Control.
- Coordinate Display:** X: 0.0000, Y: 0.0000, Z: 0.525, W: 0.0000. A blue arrow points to the Z value.
- Relative Mode:** Relative ON (checked), OFF, Probe OFF, Fast. A blue arrow points to the Relative ON button.
- Jog Feed Rate:** Slow. A blue arrow points to the Slow button.
- Axis Jog Buttons:** -X, +X, -Y, +Y, -Z, +Z. A blue arrow points to the -Z button.
- SPINDLE:** OFF, Reverse, Fast, Slow, Override (+, -).
- Coordinate Display (Bottom):** X:0.0000, Y:0.0000, Z:-0.525, W:0.0000. A blue arrow points to the Z value.
- Buttons (Bottom):** Save Home, Go Home, Set Zero, Set Zero Return, Zero All, Zero X, Zero Y, Zero Z, Zero W, Return to 0.0, Preset Help.
- Status Bar:** RPM:0, Maxrpm: 1200, Queued: , Curr Tool: 1, D: , Loop: 1, CNC: no connection, Run Time: 0 mins, USB ? /MILL, INCHES/OL, Feed: 50.0.
- Running Line:** Line: - Edit Locked ReNum Pick Draw.
- Hot Keys:** Hot Keys button.

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.

The screenshot displays a CNC control interface with the following components:

- Menu Bar:** File, Tools, Window, Hot Keys
- Toolbar:** Trace, Feed Hold, GO (green), STOP (red), Port OFF, Setup, Reset Control
- Program Editor:** A window containing the following G-code:

```
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
```
- Jog Control Panel:** Features buttons for Mill, Cycle Wizard, Glossary, Relative OFF, Coolant OFF, Slow, Fast, Jog Rapid, Feed Override (-, +), Jog Input (+Y, +X, -X, -Y, +Z, -Z, -W, +W), and Disable Motors (green).
- SPINDLE Control Panel:** Includes Reverse, OFF, Fast, +, Override, -, Slow.
- Coordinate Display:** Shows X:0.0000, Y:0.0000, Z:0.0000, W:0.00DG.
- Control Buttons:** 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.
- Status Bar:** 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.
- Bottom Bar:** Line:0, Edit Unlocked, ReNum, Pick, Draw, Hot Keys.

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.

The screenshot displays a CNC control software interface with the following components:

- Top Bar:** File, Edit, Tools, Window, Hot Keys, About, Tutorials. Buttons: Trace, Feed Hold, GO (green), STOP (red), Port OFF, Setup, Reset Control.
- Main Control Area:**
 - Left: Mill Cycle Wizard, Glossary.
 - Center: Relative OFF, Coolant OFF, Probe OFF. Slider: Slow to Fast. MICRO FEED 0.01, Jog Feed, Feed Override (-, +). Jog Input buttons: +Y, -X, +X, -Y, -Z, +Z, -W, +W. Disable Motors (green).
 - Right: SPINDLE OFF, Reverse Help, Fast, Override (+, -), Slow.
- Bottom Right Panel:**

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
- Bottom Status Bar:**

Line:1 Edit Locked ReNum Pick Draw

Hot Keys

RPM:0 Curr Tool:1 D : CNC: no connection USB ? /MILL
Maxrpm: 1200 Reset: Z offset: 0.0000 Run Time: 3.3 mins INCHES/OL
Queued: 2908Loop:0 Feed: 0.1

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

In a situation where you cannot begin your cutter at its 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.

The screenshot displays a CNC control interface with the following elements:

- Top Bar:** File, Tools, Window, Hot Keys. Buttons: Trace, Feed Hold, **GO** (green), **STOP** (red), Port OFF, Setup, Reset Control.
- Left Panel:** Mill Cycle Wizard, Glossary.
- Center Panel:** Relative OFF (Slow to Fast slider), Coolant OFF (Fast), Jog Feed, Feed Override (-, +), Jog Input (+Y, -X, -Y, +X, -Z, +Z, -W, +W), Disable Motors (green).
- Right Panel:** SPINDLE Reverse, OFF, Fast, +, Override, -, Slow.
- Main Display:** X:0.0000, Y:-0.325, Z:0.0000, W:0.00DG. Pre-set: Y-0.325. Buttons: Save Home, Go Home, Set Zero, Set Zero Return, Zero All, Zero X, Zero Y, Zero Z, Zero W, Return to 0.0.
- Bottom Bar:** Line:0, 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 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.

The screenshot displays a CNC control interface with the following components:

- Top Bar:** File, Tools, Window, Hot Keys. Buttons: Trace, Feed Hold, GO (green), STOP (red), Port OFF, Setup, Reset Control.
- Program Editor:** A text area containing:

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

A blue arrow points to the first line of the program.
- Jog Control Panel:** Includes Mill Cycle Wizard, Glossary, Relative OFF, Coolant OFF, Jog Feed, Feed Override (-, +), Jog Input (+X, -X, +Y, -Y, +Z, -Z, +W, -W), and Disable Motors (green button).
- SPINDLE Control:** Reverse checkbox, OFF button, Fast/Slow speed indicators, and Override (+, -) buttons.
- Status Display:** Shows current coordinates in green:

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

A blue arrow points to the Set Zero button in the adjacent panel.
- Control Buttons:** Save Home, Go Home, Set Zero, Set Zero Return, Zero All, Zero X, Zero Y, Zero Z, Zero W, Return to 0.0, Preset Help.
- Bottom Bar:** Line:2, Edit Unlocked, ReNum, Pick, Draw, Hot Keys.
- Machine Status:** 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.

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.

The screenshot displays a CNC control software interface. The main window shows a G-code program with the following lines:

```
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
```

The 'HOME X15.25 Y0.00 Z4.0' and 'GO_HOME' lines are highlighted in yellow. Blue arrows point to these lines. The interface includes a top status bar with buttons for 'Trace', 'Feed Hold', 'GO', 'STOP', 'Port OFF', 'Setup', and 'Reset Control'. Below this is a jog input panel with directional keys (+X, -X, +Y, -Y, +Z, -Z, +W, -W) and a 'Disable Motors' button. To the right is a spindle control panel with 'SPINDLE' status, 'Reverse' checkbox, and speed controls. At the bottom, a coordinate display shows 'X:0.0000', 'Y:0.0000', 'Z:0.0000', and 'W:0.00DG'. A 'Return to 0.0' button is located at the bottom right. The bottom status bar shows 'Line:9', 'Edit Locked', 'ReNum', 'Pick', 'Draw', and 'Hot Keys'.

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.

The screenshot shows a CNC control software interface. At the top, there is a menu bar with 'File', 'Tools', 'Window', and 'Hot Keys'. Below the menu bar is a toolbar with icons for file operations and a row of buttons: 'Trace', 'Feed Hold', 'GO' (green), 'STOP' (red), and 'Por'. The main interface is divided into several sections:

- Left Panel:** A large green rectangular area, likely a coordinate display or workspace.
- Top Right Panel:** Contains buttons for 'Mill', 'Cycle Wizard', and 'Glossary'. It also has a 'Relative OFF' indicator and a 'Slow' speed selector.
- Center Panel:** Features a 'Jog Input' section with four directional buttons: '+Y' (top), '-X' (left), '+X' (right), and '-Y' (bottom). To the right of these are 'Jog Rate' and 'Feed Override' controls with '-' and '+' buttons.
- Right Panel:** An inset photograph shows a person's hand cranking a mill. A blue arrow points from the 'Disable Motors' button in the software to the hand in the photo.
- Bottom Right Panel:** A black area with green text showing coordinates: 'X:0.0000', 'Y:0.0000', 'Z:0.0000', and 'W:0.00DG'. To the right are buttons for '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'.
- Bottom Left Panel:** A 'Running Line:' section with 'Line: -' and buttons for 'Edit Locked', 'ReNum', 'Pick', and 'Draw'. Below this is a 'Hot Keys' button.
- Bottom Center Panel:** Displays machine status: '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', and 'Feed: 12.0'.

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.

The screenshot displays a CNC control interface with several key components:

- Top Panel:** Includes buttons for Trace, Feed Hold, GO (green), STOP (red), Port OFF, Setup, and Reset Control.
- Program Editor:** Shows a G-code program with lines N01 through N21. Lines N05, N10, and N15 are highlighted with yellow tool change commands (T2, T3, T4).
- Tools Window:** A "Quick Tool Offset Return Set" dialog box is open, showing a table of tool offsets for tools T1 through T10.
- Machine Controls:** Features buttons for Mill, Cycle Wizard, Glossary, Relative OFF, Coolant OFF, SPINDLE OFF, Jog Feed, and Feed Override (+, -).
- Draw Window:** A "Master File Tool Path" window showing a 2D plot of a tool path. The path is highlighted in red, and the tool's diameter is indicated by a red oval.
- Bottom Panel:** Includes buttons for Line:0, Edit Unlocked, ReNum, Pick, and Draw.

Tool	Height	Dia.	RPM
T1	0.0	0.75	0
T2	0.25	0.5	0
T3	-0.625	0.25	0
T4	-0.5	0.10	0
T5	-0.2	0.05	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

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.

The screenshot displays the CNC Masters software interface. At the top, a menu bar includes 'File', 'Edit', 'Tools', 'Hot Keys', 'About', and 'Tutorials'. Below the menu is a toolbar with icons for file operations and a row of control buttons: 'Trace', 'Feed Hold', 'GO' (green), 'STOP' (red), 'Port OFF', 'Setup', and 'Reset Control'. The main interface is divided into several sections:

- Left Panel:** A list of tool numbers (T1 to T24) and their corresponding G-code commands (e.g., M03 S1, N3 G90, N4 G00, etc.).
- Top Center:** 'Mill Cycle Wizard' section with 'Relative OFF', 'Coolant OFF', and 'Probe OFF' indicators. Below these are 'Slow', 'Fast', and 'Jog Rapid' speed controls, and a 'MICRO FEED - 0.0' input field.
- Right Panel:** 'SPINDLE' control section with 'Reverse' and 'Help' buttons, a large 'OFF' indicator, 'Fast' and 'Slow' speed indicators, and '+' and '-' override buttons.
- Center:** 'Auto Tool Change Procedure Tutorial' dialog box. It contains an information icon, a title bar, and a list of 10 numbered steps for setting up the tools. A 'Finished' button is at the bottom right of the dialog.
- Bottom:** A 'Run' button, 'Update', 'Save and Close', and 'Cancel' buttons. Below these are 'Edit Locked', 'ReNum', 'Pick', and 'Draw' buttons. A status bar shows 'RPM:0', 'Maxrpm: 1200', and 'Queued:'.

Blue arrows point to the 'Auto Tool' button in the top toolbar and the 'SPINDLE' control panel.

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.

File Edit Tools Hot Keys About Tutorials

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

Relative OFF Coolant OFF Probe OFF

Slow Fast

MICRO FEED - 0.0 Jog Rapid

Feed Override

SPINDLE

Reverse Help

OFF

Fast

Mill Cycle Wizard

Rectangular Pocket / Slot Circular Pocket Cir. Hol

Thread Milling Cut G Rigid Tapping (Add-On) Rec. Hc

Attention: Spindle encoder required.

Start Point X: 0 Y: 0 Z: 0 RPM

Left-hand Tapping Supra Only Return Point (R) 0

Threads/Inch 0

Peck Dist (Q) 0

Reverse Tap Dist (D) 0

Tap Depth (-Z) 0

Add to File Wizard Help Glossary Verify Path

Rigid Tapping Cycle Information

Following these tips will ensure a proper rigid tapping cycle:

- Spindle must be at rest at the start of a rigid tapping cycle.
- NOTE: If spindle was on, turn off and add Delay 5000 to allow spindle to stop before cycle start.
- Spindle should be on lowest gear/belt setting for maximum torque.
- For harder materials and larger taps use smaller peck(Q) value.
- Do not use tap sizes larger than 0.5" or 12mm in materials harder than aluminum.
- Dull tools require more torque. Ensure your tap is sharp for better results.
- Coarse threads require more torque at slower speeds.
- Refer to a drill/tap chart for proper hole size for proper thread engagement percentage.

Sample code using a 1/2-13 tap after adding to file (refer to glossary for variable definitions):

```
G99  
WIZ_RIG_TAP X0 Y0 Z-.75 R.1 Q.02 D0.01 THR13 RH RPM100  
/ X... Y...  
G80
```

⚠ Not following these guidelines can potentially cause damage. ⚠

Do not show this message next time. OK

Curr Tool: 1 D: CNC: no connection USB ? /MILL

Reset: Z offset: 0.0000 Run Time: 0 mins INCHES/OL

p: 1 Feed: 50.0

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.

The screenshot displays the CNC software interface with the 'Edge Finder/Touch Plate' settings panel open. The panel is divided into several sections:

- General:** Includes tabs for 'General', 'Tools', 'Auto Tool', and 'Probe'. The 'Edge Finder/Touch Plate' tab is selected, and the 'Closed Loop' checkbox is checked.
- Mode Selection:** Radio buttons for 'Non-Automated', 'Manual', and 'CNC'. The 'CNC' mode is selected.
- Set Height:** A sub-section titled 'Within Z 0.25" from the top' with an image of a drill bit and a 'Set Height' button.
- Find Corner:** A sub-section titled 'Within 0.25" XYZ from corner' with an image of a corner and a 'Find Corner' button.
- Find Center:** A sub-section with radio buttons for 'Inside' and 'Outside' (selected), and 'Circular' and 'Rectangular' (selected). It includes input fields for 'Outside Width (Y)' (0.0000), 'Outside Length (X)' (0.0000), 'Z Clearance' (0.2500), and 'Diameter (OD)' (0.0000). A 'Find Center' button is at the bottom.

At the bottom of the panel are 'Save and Close' and 'Cancel' buttons. A blue arrow points to the 'Edge Finder/Touch Plate' tab. An inset image shows a hand using the device on a workpiece, with a black callout box containing the text 'FIND CENTER 0.00 WITH EASE'.

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.

The screenshot displays the MX software interface with several key components:

- Top Bar:** Includes buttons for Trace, Feed Hold, GO (green), STOP (red), Port OFF, Setup (highlighted with a blue arrow), and Reset Control.
- Left Panel:** Shows a G-code list (SET_ZE, T1, M03 S1, N3 G90, N4 G00, N5 G00, N6 G01, N7 G01, N8 G01, N9 G01, N10 G0, N11 G0, N12 G0, N13 G0, N14 G0, N15 G0, N16 G0, N17 G0, N18 G0, N19 G0, N20 G0, N21 G0, N22 G0, N23 G0, N24 G0) and a 'Runnig' status.
- Auto Tool Probe Window:** A sub-window titled 'Auto Tool Probe' with a 'General Tools' tab. It features a 'Use Digital Probe' checkbox (checked), 'Probe Calibration', 'Find Center', and 'Probe Guide' buttons. The 'Probe Clearance' is set to 0.015 (max 0.025"). The 'Scan Area' section includes fields for Length (X), Width (Y), and Height (Z). The 'Scan Type' section has radio buttons for Surface (selected), Pocket, Editor, and File. The 'Scan To' section has radio buttons for Editor (selected) and File. The 'Parallel To' section has radio buttons for X axis (selected) and Y axis. Other fields include 'Probe Step' (0.025), 'Feed', and 'Data Filter' (0.003). Buttons for 'Scan Update', 'Scan', and 'Help' are at the bottom.
- Mill Cycle Wizard:** A panel with 'Relative OFF', 'Coolant OFF', and 'Probe ON' indicators. It includes a 'MICRO FEED' slider at 0.0, 'Jog Rapid', and 'Feed Override' buttons. A 'Jog Input' panel with directional arrows (+Z, +W, -Z, -W) is also present.
- SPINDLE Control:** A panel with 'Reverse' and 'Help' checkboxes, a 'Fast' indicator, and 'OFF', '+', 'Override', and '-' buttons.
- Information Window:** A pop-up window titled 'Scan Settings and Preferences' with a list of settings and their descriptions:
 - SCAN AREA - Length, Width and Height will be used to set the area for Surface Scanning and Top and Depth will be used for Pocket Scanning.
 - SCAN TO EDITOR - will write scanned path directly on the editor.
 - SCAN TO FILE - will save scanned path directly to a file.
 - PARALLEL TO - will either scan object along the X axis or along the Y axis. (This option not available for Pocket Scanning.)
 - PROBE STEP - is the distance between ea
 - FEED - is the speed of scanning. NOTE: For better tolerance, use slow feed
 - DATA FILTER - is used to filter out excess use a smaller value
 - PROBE CLEARANCE - is the distance awa
 - SCAN - will start the scanning procedure
 - For Surface, ensure that probe is at the
 - For Pocket, probe must below top sur
 - SCAN UPDATE - can alter "Probe Step", information in one of the respective fields
- Bottom Bar:** Includes 'Line: -', 'Edit Locked', 'ReNum', 'Pick', 'Draw', and 'Hot Keys' buttons.
- Status Panel:** Shows 'RPM:0', 'Maxrpm: 1200', and 'Queued:'.

Two inset images are included in the bottom right:

- Pocket Scan:** A 2D grid showing a red outline of a pocket.
- Surface Scan:** A 3D wireframe model of a mask being scanned by a probe.

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.

The screenshot displays the Master MX software interface. On the left, a vertical list of program lines (N01 to N24) is visible. The main window is divided into several sections:

- Control Panel:** Located at the top, it includes buttons for Trace, Feed Hold, GO (green), STOP (red), Port OFF, Setup, and Reset Control.
- Work Offsets Dialog:** A dialog box titled "Work Offsets" is open, showing settings for G54 through G59. Each offset has a "Home" button and input fields for X, Y, and Z coordinates. All X, Y, and Z values are currently set to 0.0.
- Tutorial Window:** A window titled "Using Work Offsets - G54 - G59" is open, displaying a 2D coordinate grid. Three squares are drawn on the grid at different X positions, representing the locations defined by G54, G55, and G56. The G54 offset is set to X: 1, Y: -2.32, Z: 0.0. The G55 offset is set to X: 3, Y: -2.32, Z: 0.0. The G56 offset is set to X: 5, Y: -2.32, Z: 0.0.
- Program Editor:** A window showing the following G-code program:

```
N01 G54
N02 CALL CIRCLE 1
N03 CALL SQUARE 1
N04 G55
N02 CALL CIRCLE 1
N03 CALL SQUARE 1
N04 G56
N02 CALL CIRCLE 1
N03 CALL SQUARE 1
N07 END
-----
N08 SQUARE
N09 T1
N10 G01 X1
N11 Y1
N12 X0
N13 Y0
N14 END
-----
N15 CIRCLE
N16 T2
N17 G02 X0 Y0 I.5 J.5
N18 END
```
- Status Bar:** At the bottom, it displays "Maxrpm: 1200", "Reset: Z offset: 0.0000", "Run Time: 0 mins", "INCHES/OL", "Queued:", "Loop: 1", and "Feed: 50.0".

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.

The screenshot displays a CNC control interface with a 'Mill Cycle Wizard' window open. The wizard is configured for a 'Rectangular Pocket / Slot' operation. The background shows a G-code program list on the left and a control panel with buttons for 'Trace', 'Feed Hold', 'GO', 'STOP', 'Port OFF', 'Setup', and 'Reset Control'. The wizard window includes the following elements:

- Start Point:** X: 0, Y: 0, Z: 0
- End Point:** X: 0, Y: 0, Z: 0
- Step Over:** 0
- Step Down:** 0
- Total Depth:** Z: 0
- Options:** End with finish pass (0.005), Use Climb Cutting, Springpass (1)
- Buttons:** 'Go To Tools', 'Add to File', 'Glossary', 'Verify Path'
- Text:** 'Will use current Tool Dia. from Setup', 'Create a Slot: Use "0" for "Step Over" and "Length/Width" on the axis that will NOT be used. Ex: For a slot in the X axis, use "0" for "Width(-Y)" and "Step Over".'

The control panel also features a 'SPINDLE' section with 'Reverse', 'OFF', 'Fast', 'Override', and 'Slow' settings, and a 'Jog' section with '+Y', '+X', '+Z', '-Y', '-X', '-Z', '-W', '+W' buttons and 'Jog Input' fields. A 'Disable Motors' button is also present.

Create a Circular Pocket Wizard

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

The screenshot displays a CNC control interface with a 'Mill Cycle Wizard' dialog box open. The dialog box is titled 'Mill Cycle Wizard' and has a close button (X) in the top right corner. A blue arrow points to the 'Circular Pocket' radio button, which is selected. Other options include 'Rectangular Pocket / Slot', 'Cir. Hole Pattern', 'Thread Milling', 'Cut Gear', 'Rigid Tapping (Add-On)', and 'Rec. Hole Pattern'. The 'Will use current Tool Dia. from Setup' checkbox is checked. Below this, there are input fields for 'End with finish pass' (0.005), 'Use Climb Cutting', and 'Springpass' (1). A 'Go To Tools' button is also present. The 'Step Over' field is set to 0, and the 'RPM' field is set to 0. A note states: 'For a single circle profile use 0 for the "Step Over."' The 'Cut-in Feed(-Z)' field is set to 0. The 'Z Retract' field is set to 0.1000. There are checkboxes for 'Use Coolant', 'Spindle Off', and 'End Program'. The 'Step Down' field is set to 0. The 'Total Depth' field is set to 0. The 'Diameter' field is set to 0. There are radio buttons for 'Absolute Values' and 'Incremental Values'. At the bottom of the dialog, there are buttons for 'Add to File', 'Glossary', and 'Verify Path'. The background interface shows a menu bar (File, Edit, Tools, Hot Keys, About, Tutorials, Speed Chart), a toolbar with icons, and a control panel with buttons for 'Trace', 'Feed Hold', 'GO', 'STOP', 'Port OFF', 'Setup', and 'Reset Control'. The 'SPINDLE' section has a 'Reverse' checkbox, a 'Jog Rapid' button, and a 'Disable Motors' button. The 'Jog Input' section has buttons for '+Y', '+X', '-Y', '-X', '+Z', and '-Z'. The status display shows 'X: 0.0000', 'Y: 0.0000', 'Z: 0.0000', and 'W: 0.00DG'. The bottom status bar shows 'Curr Tool: 1', 'D: ', 'CNC: no connection', 'Run Time: 0 mins', 'USB ? /MILL INCHES/OL', 'Feed: 50.0', 'm: 3861', 'Reset: Z offset: 0.0000', and 'Loop: 1'.

Do Thread Milling using a single point cutter Wizard

File Edit Tools Hot Keys About Tutorials Speed Chart

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

Relative OFF Coolant OFF Probe OFF

SPINDLE
 Reverse ?
OFF
Fast
+
Override
-
Slow

Jog Rapid
Feed Override
- +
+Z
-W Jog Input +W
-Z
Disable Motors

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

pl: 1 D : CNC: no connection USB ? /MILL
Z offset: 0.000 Run Time: 0 mins INCHES/OL
Feed: 50.0

Mill Cycle Wizard

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

Start Point
X: 0
Y: 0
Z: 0

Cut inside thread
 Cut left hand thread

Tool OD: 0 RPM: 0

Use Coolant
 Spindle Off
 End Program

Threads/Inch: 0

Total Depth
Z: 0

Basic Thread Terms
Major Diameter, Pitch Diameter, Minor Diameter, Angle, Crest, Pitch, Root, Pitch/2, Flank

0 Tool Pitch Dia. Feed: 0
0 Diameter (Inside/Outside)
10 Thread Pitch Dia.

Absolute Values
Incremental Values

Add to File Glossary Verify Path

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

The image shows a CNC control interface with a 'Mill Cycle Wizard' dialog box open. The wizard is configured for 'Cut Gear' and includes the following fields and options:

- Options:** Rectangular Pocket / Slot, Circular Pocket, Cir. Hole Pattern, Thread Milling, **Cut Gear** (selected), Rigid Tapping (Add-On), Rec. Hole Pattern.
- Face Width:** 0
- # Teeth:** 0
- Pitch Dia:** 0
- Tool OD:** 0
- Feed:** 0
- Addendum Dia:** 0
- Checkboxes:** Use Coolant, Spindle Off, End Program.

A 3D diagram of a gear being cut by a tool is shown in the center. A technical drawing of a gear tooth profile is also visible, with labels for: Face Width, Addendum Circle, Pitch Circle, Top Land, Circular Pitch, Spoke, Tooth Thickness, Bottom Land, Dedendum, Clearance, Addendum, and Fillet Radius.

The background interface includes a status bar with 'Trace', 'Feed Hold', 'GO', 'STOP', 'Port OFF', 'Setup', and 'Reset Control'. A 'SPINDLE' control panel on the right shows 'OFF' status and speed controls. A 'Zero All' button is also visible.

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.

Use the Mill Wizards to do...

Circular Hole Drilling Pattern

Rectangular Hole Drilling Pattern

Rectangular Hole Drilling Pattern

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.

File Edit Tools Hot Keys About

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

Lathe Cycle Wizard

Relative OFF Coolant OFF Probe OFF

Slow Fast

MICRO FEED -- 0.0 Jog Rapid

Feed Override - +

-Z +Z -X +X

Jog Input

Disable Motors

SPINDLE

Reverse Help

OFF

Fast + Override - Slow

X:0.0000
Z:0.0000

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

Preset Help Return to X 0.0 Return to Z 0.0

Running Line:
Line: - Edit Locked ReNum Pick Draw

Hot Keys

RPM:0 Maxrpm: 1200 Queued: Curr Tool: 1 Reset Loop: 1 X offset: 0.0000 CNC: no connection Run Time: 0 mins USB ER/LATHE INCHES/OL Feed: 30.0

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.

The screenshot displays the MX software interface. On the left, a dialog box titled "Auto Tool Wizard" is open, showing instructions for using the Auto Tool option. The dialog includes a "Use Auto Tool" checkbox which is checked, and fields for "Clearance X: 0.00" and "Z: 0.00". The main control panel on the right features a "Lathe Cycle Wizard" section with buttons for "Relative OFF", "Coolant OFF", and "Probe OFF". Below these are "Slow" and "Fast" speed indicators, a "MICRO FEED" field set to 0.0, and "Jog Rapid" and "Feed Override" controls. A central area contains directional buttons for "-Z", "+Z", "-X", and "+X". To the right of these buttons is a "Post" selection menu with options "T1", "T2", "T3", and "T4", which is circled in red. Further right is the "SPINDLE" control section, including a "Reverse" checkbox, a "Help" button, and "OFF", "Fast", "Override", and "Slow" modes. At the bottom, a digital readout (DRO) displays "X:0.000" and "Z:0.000" in green. A red-bordered inset image shows a close-up of the lathe's tool turret mechanism. The top of the interface has a menu bar (File, Edit, Tools, Hot Keys, About) and a toolbar with icons for file operations. The top status bar includes buttons for "Trace", "Feed Hold", "GO", "STOP", "Port OFF", "Setup", and "Reset Control". The bottom status bar shows "RPM:0", "Maxrpm: 1200", "Queued:", "Loop: 1", and "X offset: 0.0000".

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

Lathe Cycle Wizard

G76 Threading Cycle G71 Turning / Boring Cycle G72 Facing Cycle
 G74 Peck Drilling Cycle G75 Grooving / Part Off Cycle

Note: The zero of the stock is always the centerline on the face of the workpiece.

Start Point X: Z:
Final Diameter (X)
Position of (Z)
Taper (I) 0
Threads per inch
Total Depth (K)
First Pass Depth (D)
Spring Pass (S) 0

Cut type: External Internal
 Right Hand Left Hand
Compound Infeed (from chart) P2

RPM SFM
 RPM Constant Surface Speed
Spindle: Forward Reverse
 Use Coolant RPM Speed Clamp (G50)
 Spindle Off End Program

Three main 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

P1 P2
 P3 P4

Background CNC controls: Trace, Feed Hold, GO, STOP, Port OFF, Setup, Reset Control, Jog Rapid, Jog Input, SPINDLE (OFF, Fast, Slow), Save Home, Go Home, Set Zero, Set Zero Return, Zero All, Zero X, Zero Z, Preset Help, Return to X 0.0, Return to Z 0.0, Cool: 1, Reset, CNC: no connection, Run Time: 0 mins, USB ER/LATHE, INCHES/OL, Feed: 30.0

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

The screenshot displays a CNC control interface with a 'Lathe Cycle Wizard' window open. The wizard window has a cyan background and contains the following elements:

- Navigation:** Trace, Feed Hold, GO (green), STOP (red), Port OFF, Setup, Reset Control.
- Wizard Options:**
 - G76 Threading Cycle
 - G71 Turning / Boring Cycle (indicated by a blue arrow)
 - G72 Facing Cycle
 - G74 Peck Drilling Cycle
 - G75 Grooving / Part Off Cycle
- Parameters:**
 - Start Point X: [] Z: [] Help
 - Final Diameter (X) [] Help
 - Position of (Z) [] Help
 - Amount of Taper (I) 0.0 Help
 - Feed Inch/Min. [] Help
 - Total Depth (K) [] Help
 - First Pass Depth (D) [] Help
- Diagram:** A 'Turning Cycle' diagram showing a lathe part with dimensions X, Z, D, K, and START POINT.
- Cut type:** External Internal Help
- Speeds:** RPM [] SFM [] Help. Options: RPM, Constant Surface Speed.
- Spindle:** Forward Reverse
- Buttons:** ADD
- Summary:** Three lines of code will be added to the program: Spindle code, Start Point code, Cycle code.

The main control panel on the right includes:

- FF** (Feed Forward) and **Probe OFF** indicators.
- SPINDLE** section with Reverse Help, **OFF** status, and speed controls (Fast, +, Override, -, Slow).
- Jog** section with Jog Rapid, Jog Input, and Jog Override (+, -) buttons.
- Axis Controls:** Large -X and +X buttons.
- Function Buttons:** Save Home, Go Home, Set Zero, Set Zero Return, Zero All, Zero X, Zero Z, Preset Help, Return to X 0.0, Return to Z 0.0.
- Status Bar:** Coolant: 1, CNC: no connection, Run Time: 0 mins, USB ER/LATHE, INCHES/OL, Feed: 30.0.

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

File Edit Tools Hot Keys About
STOP Port OFF Setup Reset Control

Threading Cycle
 G71 Turning / Boring Cycle
 G72 Facing Cycle
 G74 Peck Drilling Cycle
 G75 Grooving / Part Off Cycle

Start Point X: Z: Help

Return Point (R) Help

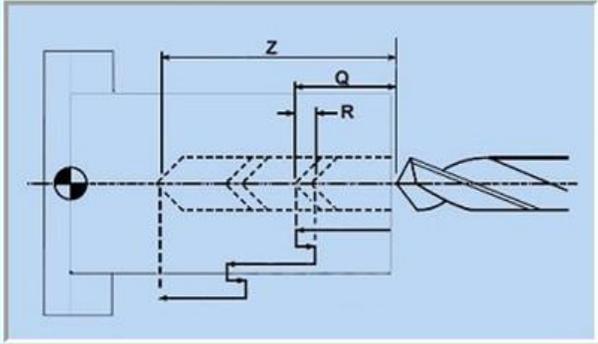
Cut-in Dist (Q) Help

Full Retract (S) Help

Feed Inch/Min. Help

Total Depth (Z) Help

Peck Drill Cycle



Relative OFF
Coolant OFF
Probe OFF

ED - 0.0
Jog Rapid

Feed Override
-X

-Z
+X

Jog Input
Disable Motors

SPINDLE
Reverse Help

OFF
Fast

+
Override

Slow

0000
Save Home
Zero All

0000
Go Home
Zero X

Set Zero
Zero Z

Set Zero Return

Preset Help
Return to X 0.0
Return to Z 0.0

Curr Tool: 1
Reset
CNC: no connection
USB ER/LATHE

Reset: Z offset: 0.0000
Run Time: 0 mins
INCHES/OL

op: 1 X offset: 0.0000
Feed: 30.0

RPM

Spindle: Forward Reverse

ADD

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

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.

Use the Lathe Wizards to do...

Facing Cycles Grooving Cycles Part Off Cycles

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

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

All with Constant Surface Speed

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.

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

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...



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.