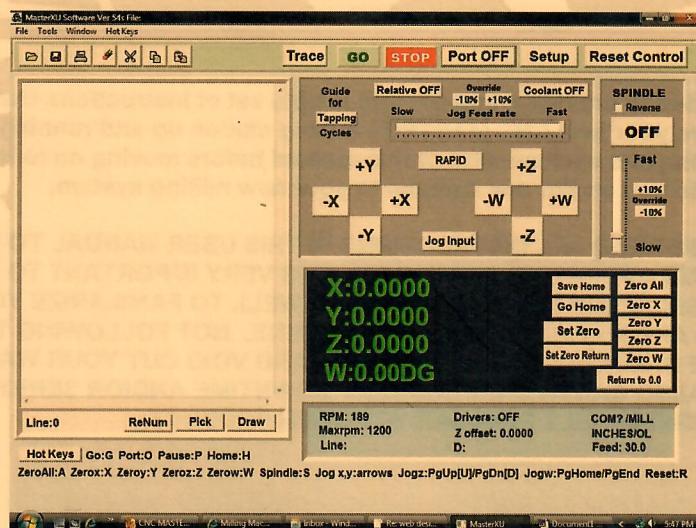


Master MX

CNC MASTERS

CNC MASTER SOFTWARE

USER INSTALLATION AND OPERATIONAL MANUAL



CNC MASTERS, Inc.
16031 Arrow Hwy. Ste. J&K
Irwindale, CA 91706 USA
Tel: (626)962-9300
sales@cncmasters.com
www.cncmasters.com

READ BEFORE INSTALLING SOFTWARE

CNC MASTERS, Inc.

The Master MX Operating Software User's Manual

For use on the CNC MASTERS brand machines only:

**CNC Jr.
CNC Jr. Plus
CNC Baron
CNC Max
CNC SUPRA**

**CNC MASTERS, INC.
16031 ARROW HWY, STE. J
IRWINDALE, CA 91706
PH: 626-962-9300
sales@cncmasters.com
www.cncmasters.com**

Preface

This manual is intended to provide a smooth and simple set of instructions that, if followed correctly, should have your new CNC MASTERS milling station up and running quickly, reliably and safely. Please complete each section of this manual before moving on to the next. This will prevent any injuries to you and/or any damage to your new milling system.

IMPORTANT: PLEASE READ ALL INSTRUCTIONS IN THIS USER MANUAL TO BECOME FAMILIAR WITH THE MASTER MX OPERATING SOFTWARE. IT IS VERY IMPORTANT TO READ THE MACHINE'S CORRESPONDING USER'S MANUAL AS WELL TO FAMILIARIZE YOURSELF WITH THE PROPER HANDLING AND MECHANICS OF THE MACHINE. NOT FOLLOWING THE INSTRUCTIONS MAY RESULT IN THE DAMAGE OF YOUR PRODUCT AND VOID OUT YOUR WARRANTY WITH CNC MASTERS. IT MAY ALSO CAUSE UNNECESSARY DOWNTIME AND/OR SERIOUS INJURY SINCE YOU MAY NOT BE COMPLETELY FAMILIAR WITH THIS PRODUCT.

01.23

Remove this page and post on the machine or by the operator station.

IMPORTANT:

**ON THE MASTER SOFTWARE, BEFORE
YOU PRESS GO:**

- 1. POSITION CUTTER AT PHYSICAL X,Y,
Z, (W) 0.00 LOCATION ON YOUR
MACHINE.**
- 2. PRESS RESET CONTROL.**
- 3. PRESS ZERO ALL.**
- 4. VISUALLY CHECK TO SEE
COUNTERS ARE 0.00.**
- 5. PRESS GO.**

****Oil your machine daily.** Run all axes in the air to allow the oil to spread throughout the ways. Grease the X, Y, and Z ball screws every two months.**

IMPORTANT

ON THE MASTER SOFTWARE, BEFORE YOU PRESS GO:

1. POSITION CUTTER AT PHYSICAL X,Y,
Z,(W) 0.0 LOCATION ON YOUR
MACHINE.

2. PRESS RESET CONTROL.

3. PRESS ZERO ALL.

4. VISUALITY CHECK TO SEE
COUNTERS ARE 0.00.

5. PRESS GO.

**ON YOUR MACHINE DAILY. RUN ALL AXES IN THE SILENT MODE.
GEAR SWING TO OIL SEATS THROUGHOUT THE WORKS. GEARS
ARE TWO MOTORS. X, Y, AND Z PELL SCREWS EVELY TWO MONTHS.

TABLE OF CONTENTS

SEC 1. PRE-INSTALLATION

- STEP 1. REQUIRED ITEMS FOR A SUCCESSFUL INSTALLATION**
- STEP 2. THE MILLING STATION**
- STEP 3. THE PC COMPUTER AND CNC MILLING STATION CONTROL UNIT**
- STEP 4. THE MASTER SOFTWARE INSTALLATION**
- STEP 5. THE MASTER SOFTWARE CONFIGURATION**
- STEP 6. CONNECTING THE PC TO THE CNC CONTROL UNIT**

SEC 2. ABOUT YOUR NEW MASTER MX SOFTWARE SCREEN

- ITEM 1. UPPER LEFT SCREEN (BASIC WINDOWS COMMANDS)**
- ITEM 2. UPPER MIDDLE AND RIGHT SCREEN**
- ITEM 3. LOWER LEFT SCREEN (PROGRAM CREATION/EDITING WINDOW)**
- ITEM 4. LOWER RIGHT SCREEN (MANUAL FEEDING AREA)**
- ITEM 5. CNC MASTERS MILL: Accessories and Options**

SEC 3. Computer Control Variable Spindle Speed Drive/Manual Control of Axis Motors (Option)

SEC 4. PROGRAMMING YOUR CNC MILL WITH THE MASTER SOFTWARE – IMPORTANT SECTION.

- Exercise 1: The Absolute Mode**
- Exercise 2: The Incremental Mode**
- Exercise 3: The Circle**
- Exercise 4: The Drill using Nest/Call commands**

SEC 5. MASTER Command List found under your PICK Button

SEC 6. G-Code File Interpreter In Master Software

SEC 7. Master Software Additional Commands

SEC 8. Saving Your Zero & Home Position; Limit Sensor Switches

SEC 9. Addendum: Touch Screen Special Features

SEC. 1 PRE-INSTALLATION

STEP 1 REQUIRED ITEMS FOR A SUCCESSFUL INSTALLATION

It is highly recommended you have all of the following items available for use before beginning this project. The following instructions will call for each of the items at least once.

- Utility Power, Volt Meter, Milling Station, CNC Controller Unit, Mill Tramming tool,
- PC computer, 600-watt minimum Battery Backup, Master Control Software, USB Cable.

INSTALLATION

STEP 2 THE MILLING STATION

- 1. Go to PAGE 3 of this manual. It is VERY IMPORTANT to remove page 3 from this manual and affix in a location on or near the machine operator's station. These five steps will help ensure proper results when operating your machine.
- 2. Make sure all electrical power required for your milling station is installed. This will be based on the model of CNC MILLING STATION purchased from CNC MASTERS. PLEASE refer to your CNC MILLING MACHINE'S owner's manual for specific power requirements if not already installed. Verify proper voltage with your Volt Meter.
- 3. The CNC MILLING STATION setup: This system should have been received, unpacked and properly assembled in its final location. This should include all required electrical receptacles and power cables. The stepper motors should also be installed. PLEASE refer to your CNC MILLING MACHINE'S owner's manual for specific assembly procedures if not already installed.
- 4. Make sure limit switches are open. Check all three axes (6 switches) to ensure none of the limit switches are activated. A limit switch can be found on all extreme ends of travel on the X, Y, and Z axes. For the Z-axis, rotate the small hand-wheel on the motor until the spindle looks to be at the center of the machine head. Do the same for the X and Y axes until the table and saddle look centered at the machine.
- 5. Make sure Fluid Levels are full. If your Milling System came with coolant control, verify proper coolant levels. Lack of proper coolant will damage the submersible pump.
- 6. Make sure the CNC MILLING STATION has been trammed properly. Depending on your style of machine, it can be tilted from side to side and from front to back. This allows for versatility of the machine, but these adjustments can drift. Before milling, one should check and adjust the head so that the spindle will be square to the plane of the table. Your milling table and spindle must be aligned properly for accurate milling to occur. If you notice a front/back tilt on a table top model, contact us and we will walk you through to square the head to the table.



Tramming Tool

STEP 3 THE PC COMPUTER AND CNC MILLING STATION CONTROL UNIT

THE PC COMPUTER

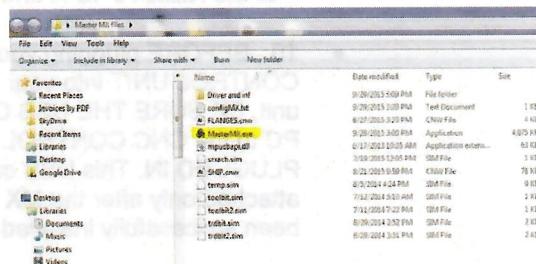
- 8. Make sure all electrical power required for your PC computer and CNC MILLING STATION CONTROL UNIT is installed. You will need one 110VAC single-phase dedicated circuit using a NEMA 5-15R with a true ground. Please test the circuit with your Volt Meter.
- 9. Install your Battery backup unit (600 watts or stronger) as the only item attached to the NEMA 5-15 R and allow it to charge.
- 10. BEFORE plugging your CNC CONTROL UNIT into your battery backup unit, BE SURE THE USB CABLE from the PC to the CNC CONTROL UNIT is NOT PLUGGED IN. This USB cable will be attached only after the MX Software has been successfully installed on your PC.
- 11. The computer can be a laptop, desktop, or all in one. No tablet type computers. It must be a 64-bit operating system Windows 10. It is recommended that the Mouse and Keyboard be cabled.
- 12. Keep this computer dedicated to the machine. It should be part of the machine tool. The computer should not be used for anything else except to run the machine.
- 13. Run a series of Windows Updates on the computer. Re-boot the computer and check to ensure that the computer is fully updated in your Settings under the Windows Update page.
- 14. Disable all anti-virus and firewall settings. Disable any network settings and any automatic sequences that may take place periodically on your computer, such as Windows Updates. An auto-sequenced program could interrupt the running of your tool path program, resulting in machining errors. Disable Wi-fi and Blue-tooth under Settings.
- 15. Disable Power Options under Settings. Hibernation mode and Sleep mode should be on "never."
- 16. Close any open programs.
- 17. The PC is now dedicated to ONLY running the CNC MX SOFTWARE while performing milling operations. Again **keep the computer dedicated. Do not use this computer for other functions while running the Master MX Software program.** A loss of communication between the PC and the CNC controller may result in machining errors.
- 18. The PC is now ready.



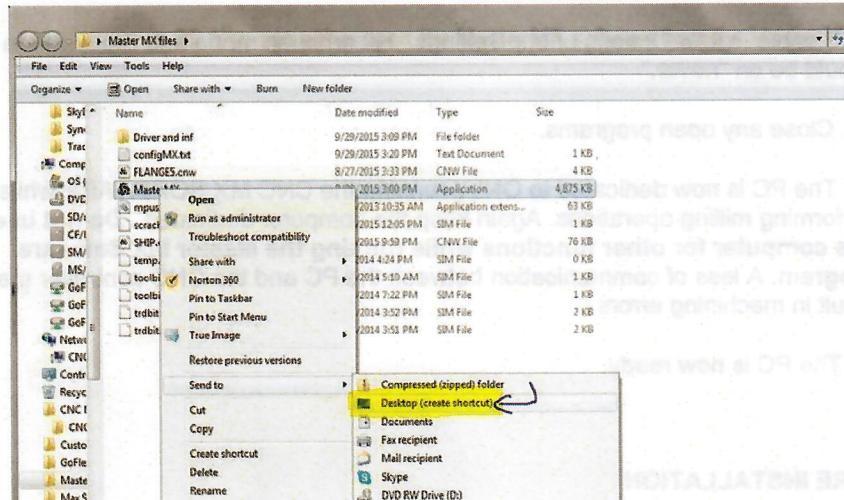
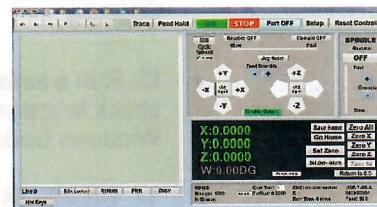
**USB Cable
Do not exceed 8 feet**

STEP 4 THE MASTER SOFTWARE INSTALLATION

- 19. Install the MX software from your flash-drive USB stick if you received the MX software zipped folder from your email. If you are the original buyer of this machine, you may email us at sales@cncmasters.com for a copy of the MX Software.
- 20. Do not open or double click on the zipped folder. Begin by right clicking over the zipped folder; choose copy. Then paste onto an open area on your Desktop screen, or at another desired location such as C: Hard Drive in Program Files.
- 21. Right click over the zipped folder. Choose Extract All...
- 22. A new un-zipped folder will appear containing the Master MX Files. Open this folder which will lead you to the Master MX files. Open this folder.
- 23. Double click on **MasterMX.exe** to begin installation.



- 24. The Master MX will now open up.
- 25. The Master MX Software will ask you for the type of machine you are using; Mill or Lathe. Please click the appropriate answer.
- 26. The Master MX Software will ask you for the type of units you wish to work in; Inches or Millimeters. Please click the appropriate answer. This is your new Master Software screen.
- 27. You can either open the Master MX software from the Master MX Files Folder by simply double clicking on the MasterMX.exe each time. If you prefer, you can also create a shortcut on your desktop by right clicking over the MasterMX.exe. Then choose Send to and click on Desktop (create shortcut).



- 28. The Master Software is now installed. Please note that if your computer contains anti-virus protection, it may red-flag the Master MX. You will need to allow the anti-virus protection to trust the MasterMX.exe.

MASTER MX SOFTWARE UPDATES:

To replace a previously installed Master Software program, simply delete the zipped folder and the un-zipped Master MX Files folders. Updates are made available by email. There is no charge for updates as long as the hardware on the control unit is compatible and your company is the original buyer of this machine. The USB driver installation will not need to be repeated when downloading a newer version. For operators who purchase a CNC MASTERS machine second-hand, there will be a nominal charge for Master updates or support from CNC MASTERS by email or telephone.

STEP 5.

THE MASTER SOFTWARE CONFIGURATION

- 29. To verify the settings just made in your software, please locate and push the "SETUP" button on the upper right of your software screen.
- 30. Verify the settings in your software. This is your opportunity to make changes to any of the values shown. You may return to this screen at anytime in the future. Once satisfied with the values, please click the "SAVE" button.
- 31. To close the Master Software, click "File" and then "Exit" on the top left of the monitor screen; OR click the red "X" on the top right of the monitor screen. NOTE: There is no "minimize" option available on the Master Software. This is intentional to keep the computer dedicated when operating your machine. Adjust screen resolution through your control panel if any part of the Master software appears to be cut off.
- 32. The Master Software is now configured. If you wish to continue to Step 6 below to install the USB drivers, shut down the Master Software by clicking the Red X top right of the screen.

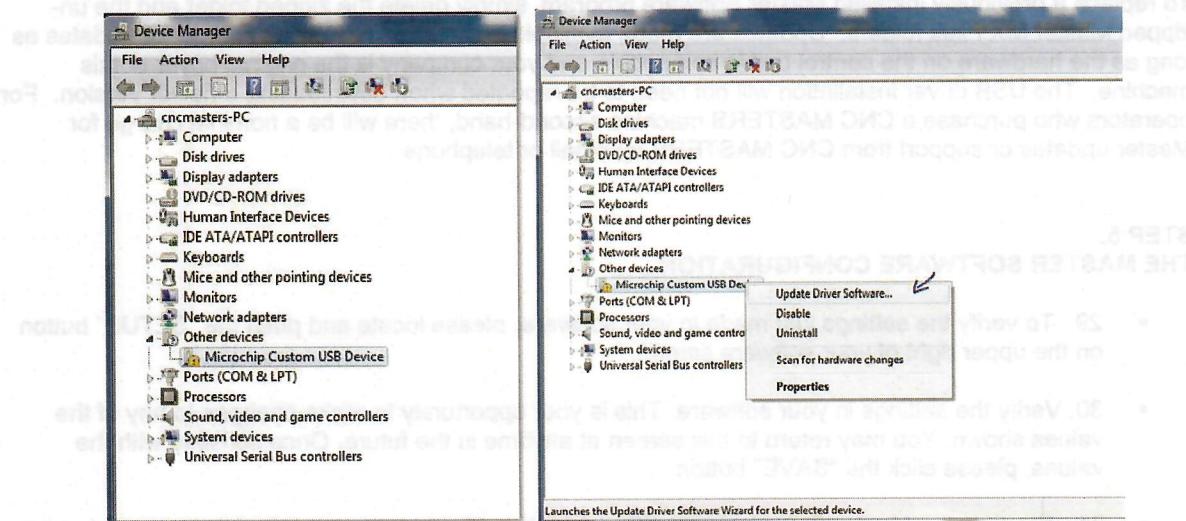
STEP 6.

CONNECTING THE PC TO THE CNC CONTROL UNIT – USB PORT CONNECTION

- 33. Make sure the CNC Control Unit and Master software is shut down. The CNC Control Unit must be off before attaching the USB cable. The premature installation of drives could result in an installation error. Should this happen, a System Restore on your PC is required.
- 34. **Unplug the Spindle Motor's 220vac plug from its power source. Do not hardwire in.** You can opt to hardwire in the plug after all completing X Y Z tests to see that all axes are in operation. With the exception of the Jr. Mill, the spindle motor plug for all CNC MASTERS machines comes off the VFD in the smaller rectangular black box.
- 35. Now connect the USB cable from the Control Unit to the PC computer. **Do not use an extension on this USB cable.** If a USB cable replacement is ever required, the cable must not exceed 8' feet in length and must be well shielded or else communication errors may occur. **Do not place the cable next to, or tie wrap it to the Control Unit or spindle motor power cord. This can cause electrical noise to go into the Control Unit.**
- 36. After verifying the electrical requirements for your Control Unit, power up the Control Unit. You will find a large red mountain rocker switch located on the front of the machine head. **IMPORTANT:** If a "Found New Hardware" wizard pops up, close this wizard. The correct drives are in the MasterMX Files Folder and you will install them as part of the next instruction.

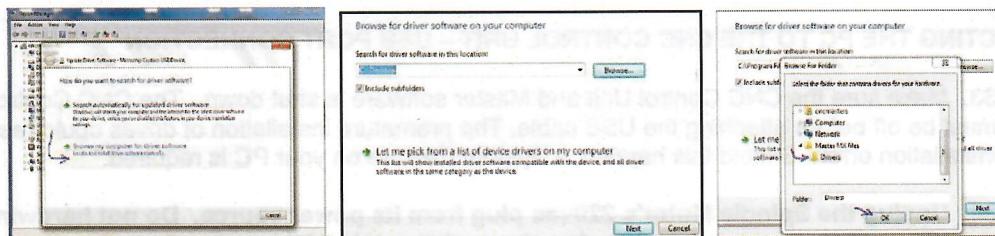
In your Windows Search box which you can find after pressing the Start left corner button on your Desktop, look for Device Manager, or locate Device Manager by going to your Windows Control Panel.

- 37. You will then see under Other Devices > Microchip Custom USB Device. Right click over Microchip Custom USB Device, and left click over Update Driver Software.

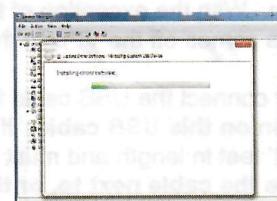


- 38. In the next window, Update Driver Software – Microchip Custom USB Device, click on Browse my computer for driver software. Click Browse and search for the MasterMX Files Folder. At the beginning of the Master MX installation, it was recommended that you either place the MasterMX Files Folder on your Desktop or in your C: Drive under Program Files. So as you browse for the MasterMX Files Folder, keep in mind where you originally pasted the folder.

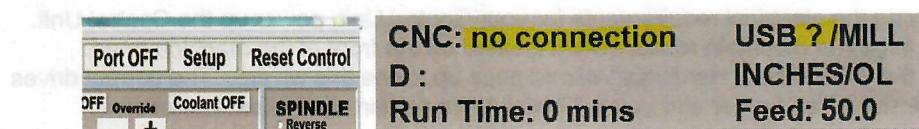
Once you locate the Master MX files in the Browse for Folder window, click on Drivers and then OK.



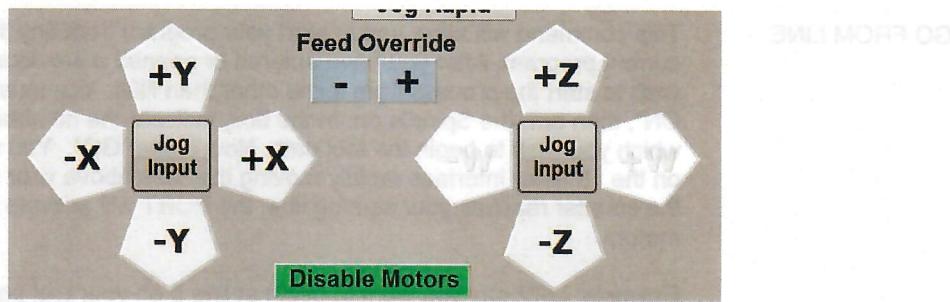
- 39. The MX drivers should now properly install in your system.
- 40. Open the Master Software program.
- 41. Activating your new drivers: Locate the "Reset" at the upper right of the Master Software screen. Press "CNC: no connection" to "CNC: OK" to appear toward right of the screen.



Control" button it and wait for the bottom



- 42. Activating Port communications: By clicking "Reset Control," this will automatically change the "Port OFF" button to "Port ON." Clicking the "Port ON" to "Port OFF" button will cease all other commands from running the machine. This is important should you wish to make a change in your tool path program in the Editor while being safe from accidentally pressing a movement command. Click back to "Port ON" or "Reset Control" when you are ready to drive the machine once again.
- 43. Testing CNC controlled movement: Click the +X, -X, +Y and -Y axes with your PC mouse. You should see movement on all of your mill's axes with the use of your PC mouse or touch screen monitor. **DO NOT rapidly click the same axis or you could lose control of an axis's movement.** Please note that your axes buttons are from the point of view of the cutter on the machine. If you are using a touch screen monitor, you may need to adjust its sensitivity at your windows control panel.



- 44. Keep all three axes in the center of the machine away from their extreme positions of travel. Plug in the spindle motor to its power source. With the exception of the Jr Milling Machine, follow Sec .3 of this manual, "Computer Control Variable Spindle Speed Drive," to learn how to operate the spindle.

SEC 2.

ABOUT YOUR NEW MASTER MX SOFTWARE

The Master Software uses many of the Windows commands you are familiar with and some that will be new to you. These commands will be explained in this section.

ITEM 1. Upper Left Screen (basic windows commands)

File Edit Tools Hot Keys About Tutorials

In the upper left screen you will see the typical Windows tool bar:
File, Edit, Tools, Hot Keys, About, Tutorials

"File" has the basic functionality you will use on a daily basis: NEW, OPEN, SAVE, SAVE AS and EXIT.

NEW	Will be used to create new programs and code for your CNC machine to use immediately or in the future. Clicking NEW will also clear the editor screen.
OPEN	Will allow you to recall programs previously created or programs that may be modified to work for a new application.
SAVE	Will allow you to save the current program; but be aware, this WILL REPLACE the original program of the same name. DO NOT SAVE your programs in the Master MX Files Folder. Choose a directory outside the Master MX Files Folder.

SAVE AS	Works just like the Save button but allows you to give your current program a new name under a separate saved document while preserving the original program.
RECENT FILES	Easily open the previous saved file.
EXIT	Will end the MASTER SOFTWARE program immediately. Be aware, any changes not saved to the open program will be lost without warning.

"Tools" has the operational tools and user configuration options: "GO FROM LINE", "BEEP END OF PROGRAM", "DELETE CONFIG.CN3", "FIND USB", and "CHANGE TEXT SIZE".

GO FROM LINE This command will allow you to start your program from any line within your current program. After you have entered or opened a previous program, you may wish to start the process from a line other than N01. You must first have "PORT ON", then turn the Spindle on. In the box, indicate the number of the line from which you wish to begin the tool path. Now press "GO". You will see the counters on the Master Interface rapidly moving the lines above your starting point. Once the counter reaches your starting line, the PORT will activate and drive the axis motors.

Example: Let's say you want to begin at line 5 on your tool path. Enter the value 05 and press GO. You will need to enter the same value as numerically represented for that particular line. **IMPORTANT:** If you begin on line 5, your cutter must be positioned at the end of its travel on line 4. If you cannot verify and confirm this mechanical position on the machine, then do not attempt to run this command or you may damage your part. Jumping lines on a tool path that contain CALL or subroutines will not be calculated, so it is recommended to jump to a line in your program with an Absolute position, not an Incremental position.

BEEP END OF PROGRAM This is a flag that, if set, will send the operator an audible beeping sound at the completion of a program as long as the computer supports sound. Enable or disable this function only before or at the conclusion of any program. Attempting to change this flag while the CNC program is producing a product will cause the machine to pause. This may result in milling errors. This feature may be useful for users who are in a different room. The beeping sound will continue until the user clicks OK. By default, once the Master Software is closed, the BEEP END OF PROGRAM will be disabled the next time the Master is opened.

DELETE CONFIG.CN3 The DELETE CONFIG.CN3 command can be found under the "Tools" pull-down Menu at the top of the Master Interface on the left hand side. Use this command if an unusual error or pop-up message comes up and cannot be closed, or seems to be caught in a loop every time you open the Master Software. This can be a useful command to use for experimental programs you are writing that may contain incorrect G-code usage, wrong math, or any programming that may leave the Master Software in a confused state as it tries to correct itself. The "Config." File will delete all settings in your Master Software and will shut down the interface. When you click on the icon to open the software, it will open as if you just downloaded the software for the first time. If the message keeps popping up, or the interface does not shut down after clicking on this command, delete the ConfigMX.txt file located in the Master MX files folder. Now go to your C Drive, go to Program Files [or Program Files (x86)], and look for the Master MX folder. There you will find only one ConfigMX file. Delete this file and close all windows. Shut down the control unit, and re-boot the computer. The Master should then

open as if you downloaded it for the first time, which is a successful internal debugging of the software.

FIND USB

In very rare cases, the USB device may not be detected. FIND USB allows you to force a USB connection.

CHANGE TEXT SIZE

This command allows for ease of code reading. The text size can be changed only before or at the conclusion of any program. Attempting to change this flag while the CNC program is producing a product will cause the machine to pause. This may result in milling errors.

"Hot Keys" is a group of the most commonly used tools in the Master Software: "HOT KEYS" (F7), "GO" (F8), "PORT" (F9), "STOP" (Space Bar) and "JOG INPUT" (J).

HOT KEYS (F7)

When clicked, a list of all available CNC machine functions will be displayed at the bottom of the screen for your reference. The Hot Keys cannot be triggered from the keyboard unless the Hot Key list is visible on the bottom of the screen. You may use any of the function keys (F7, F8, F9, Space Bar or J) at any time.

GO (F8)

The GO (F8) key will start the machine and is the same as the green GO button on the top middle of the screen.

PORT (F9)

The PORT (F9) key will toggle the communications port on or off and is the same as the gray PORT button on the top middle/right of the screen.

STOP (Space Bar)

The STOP (Space Bar) key will stop the machine at any time within the program. It is the same as the RED STOP button on the top middle of your screen. We recommend placing a piece of red tape along the space bar to remind of this feature.

JOG INPUT (J)

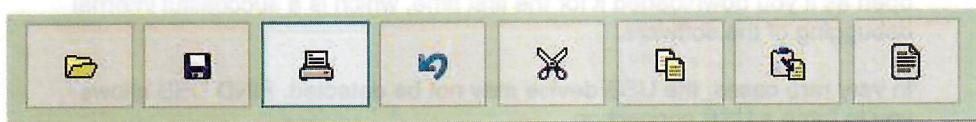
JOG-INPUT feature or (TEACH MODE):

This command will input each line of the jogging you make with your PC mouse, keyboard, or hand held remote. Example: You click +X with your mouse and you drive to a new location, then press the JOG-INPUT button on the screen. The Master will then incrementally write that line on the editor. You then Jog -Y and press the JOG-INPUT button; the Master will record this action as the second line on the editor. This feature can be useful to build a program to use again in the future by driving one axis at a time. Keep in mind that the program is being written in INCREMENTAL mode, so be sure you write "INCREMENTAL" as the first line in your program before pressing GO to run this newly written program.

NOTE: The image below consists of two different screen shots. Specifically look for the additional line of N05 and how the X, Y, and Z values match the information on right.



These next buttons below are the most popular quick action buttons: "OPEN", "SAVE", "PRINT", "UNDO", "CUT", "COPY" and "PASTE". You may run your mouse over each button and the function of the button will be displayed.



	OPEN	Will allow you to recall programs previously created or programs that may be modified to work on a new application.
	SAVE	Will allow you to save the current program, but be aware, this WILL REPLACE the original program of the same name.
	PRINT	Will allow you to print your entire program. Use this function only before or at the conclusion of any program. Attempting to print while the CNC program is producing a product will cause the machine to pause. This may result in milling errors.
	UNDO	The UNDO command will "reverse" the last command or change you made to your program. If you deleted a line, it will replace it; if you added a line, it will remove it.
	CUT	The CUT command is used to remove groups of lines, a single line, or a piece of a line from your program. After cutting a portion of your program, it will remain in your computer's memory where it can be pasted somewhere else within the current program or placed in a new or different program.
	COPY	The COPY command is used to duplicate groups of lines, a single line, or a piece of a line from your program. Once copied, this portion of your program will remain in your computer's memory where it can be pasted somewhere else within the current program or be placed in a new or different program.
	PASTE	The PASTE command will allow you to insert previously cut or copied information anywhere within your current program. It may also be used in a different program or a new program.
	NEW	The NEW command simply clears the Editor Screen.

ITEM 2. Upper Middle to Right Screen

The last items on the top of the Master Software program: "TRACE", "GO", "STOP", "Port OFF/ON", "SETUP", and "RESET CONTROL".



Trace

The TRACE allows you to execute your program manually one line at a time (allowing you to monitor your results one line at a time). Another way to verify each line of your program is to use the TRACE command in combination with the DRAW feature. Press TRACE first, and then press the DRAW button. Now press the RUN NEXT LINE button and you will see the drawing take place one line at a time. This will not operate the machine one line at a time though as in the TRACE command by itself. Once the program is verified, close the windows.

Feed Hold

Feed Hold is to pause your tool path program while on the fly. This opens up commands to step through the program line by line.

GO

The GO button starts the execution of your program from line one (1).

STOP

The STOP button stops your program immediately at whatever line it is currently on. It is NOT a pause button. You should consider your program terminated after using STOP and you MUST reposition your cutter at the 0.0 position. Pushing the GO button without repositioning your cutter will result in the program restarting from Line one (1), but with your cutter beginning from the wrong location.

Port OFF

The controller and PC communicate through a USB port. The Port OFF/ON button allows you to enable or disable this communication. The PORT must be ON for the system to work. This button provides the same function as manually pressing the (F5) key to toggle the PORT Off or On. The PORT button will display ON after pressing this button.

Setup

The SETUP button opens up to the "General" tab and the "Tools" tab. The "General" tab allows for user specific information to be recorded: inches or millimeter, lathe or mill, RAPID settings, RPM settings, etc... The "Tools" tab is used strictly for tool height compensation.

Reset Control

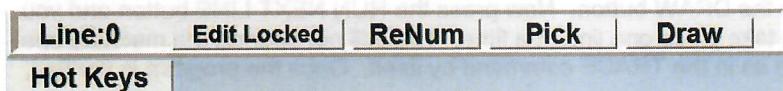
The RESET CONTROL is a critical step in starting your Master Software. It clears and resets the communication link between the controller and the PC. It also verifies the drivers to be in good working order to the user by changing the CNC: no connection to OK. Reset Control must always be pressed prior to pressing GO.

ITEM 3. Lower Left Screen (program creation/editing window)

```
SET HOME X7.2201 Y-4.0418 Z1.2415 W0.0000  
N01 THIS SAMPLE CUTS THE "CAGE.DGN" DONE.WI  
N02 THE FEED SPEEDS ARE FAST AND DESIGNED 1500  
N03 SPINDLE ON FORWARD SPEED 1500  
N04 MOVE X0 Z0  
N05 CALL OUTSIDE 1  
N06 CALL STRAIGHT 1  
N10 MOVE X1.197 Y4.97  
N11 CALL HOLE 1  
N12 MOVE X1.197 Y2.867  
N13 CALL HOLE 1  
N14 SPINDLE OFF  
N15 MOVE X0 Y0 Z0  
N17 END  
N18 *****  
N19 POCKET  
N20 THE FOLLOWING G-CODE FILE WAS CREATED  
N21 THIS PROGRAM WITH THE "I" COMMAND. TO DE
```

Line:0 Edit Locked ReNum Pick Draw

The lower left portion of the screen is the program creation/editing window. When the Master Software is started, it recalls and loads the last program loaded into memory.



The specific features on the bottom of this window are: "Line:0", "Edit Locked/Unlocked", "ReNum", "Pick", "Draw" and "Hot Keys".

Line:0 LINE:0 shows the location of your cursor in the program or the line the program is about to execute.

Edit Unlocked / Edit Locked These are the Master defaults to an Edit Locked green editor screen. To create or edit your tool path program, you will need to Edit Unlock first, which will turn the screen to a white color.

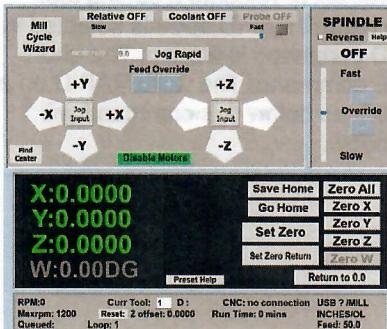
ReNum ReNum is used after editing or adding to an existing program. ReNum only makes each line sequential; it does not move or edit your instructions.

Pick The Pick button is an exclusive feature of the CNC MASTERS Software program. It provides a list of plain language tools that may be used within your program.

Draw The DRAW window will give you a top-side view drawing of your written program. What you see in the drawing is the actual tool path of the cutter along with Z-axis movement at the bottom of the screen. The cancel button on the top left side of the window is only used for lengthy programs (i.e.: The DRAW feature is in the middle of interpreting and you wish to cancel the remainder of the drawing process). **Always use the DRAW feature prior to running a new program.** Let's say you used CAM software to view the computerized tool path simulation, and the simulation came out successfully. It is still wise to run a DRAW on the Master itself to verify that the Master is interpreting the G-code tool paths in the same way as your simulator. Example: Your simulator shows the part being machined successfully, but the DRAW on the Master shows an erratic tool path. This could mean you may have accidentally used a different post processor. In making this discovery, you avoid damaging your part. Running the program in the air is also a good idea to study the tool path.

Hot Keys The Hot Keys button will display all keyboard commands at the bottom of the screen. The keyboard only becomes functional when the Hot Keys commands are displayed on the Master Interface. The Hot Keys may be used at any time, provided a tool path program is not running.

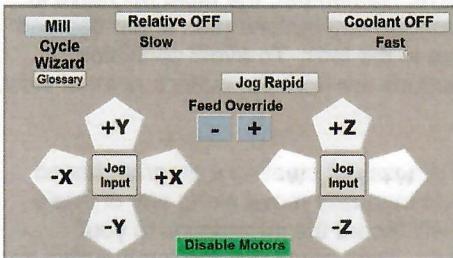
ITEM 4. Lower Right Screen (Manual feeding area)



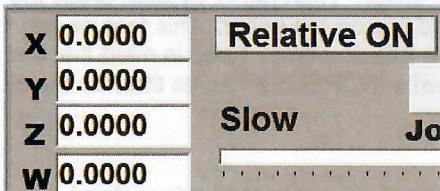
This area of the MASTER SOFTWARE is here to provide you with the controls and digital feedback your machinist will need when producing product or creating new programs. This area can be broken down into four functional areas:

1. Manual Table and Head Adjustment (or JOG) Area
2. Spindle Speed and Reverse Area
3. Digital Readout and Axis Manual Zero Functions
4. Machine Status Window.

1. Manual Table and Head Adjustment (or JOG) Area



This is designed to provide the primary manual controls on your machine. At the top you should find two buttons; Relative OFF/ON and Coolant OFF/ON. Coolant OFF/ON starts the coolant pump on your CNC machine to cool your cutter. This function is only available if your machine comes with coolant control.



RELATIVE ON/OFF INTERFACE feature (SIMILAR TO POWER FEED CONTROL)

The Relative ON feature will allow you to enter a specific value to jog any of your axes. The Relative ON feature will allow you to jog to an exact specification on each axis without needing to keep the key pressed down or the button of the axis pressed down with the mouse. Example: You need to travel exactly 2"

on the X. So you enter 2.000 in the X box. Next, using your mouse, click once on the +X or -X button for the desired direction in which you need the X table to travel. The RELATIVE ON feature will then drive the X table 2" in the specified direction. Be sure to adjust the JOG FEED RATE first on the interface should you need to be on a certain feed, or on RAPID, before pressing the axis button to execute the relative move (For further details on the Jog buttons, refer to the "Jog Feed Rate Override" paragraph below). Only use the mouse to activate the drives on each axis, not the keyboard Hot Keys. To jog your three or four axes with your mouse or keyboard without the use of the RELATIVE feature, be sure the RELATIVE button is in the OFF position. Note: The counters will automatically display the desired value instead of counting to the desired value when using this relative feature.

Coolant OFF

The COOLANT ON and COOLANT OFF commands will turn the pump On and Off on your Master Software display and can also be controlled with M-code commands.

Feed Override

Jog Feed Rate Override

The + and – buttons located over the top slider are used to adjust the feed rate by PC mouse on the Master Interface. These two buttons can be used to "fine tune" feed during the running of a program after you press GO. Once you make your adjustment, you will see the feed increase or decrease after the line that the machine is on.

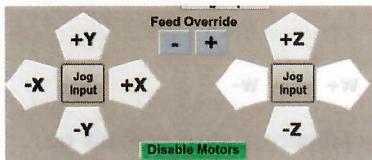
RAPID

The RAPID command is typically used to quickly retract or relocate a cutting tool after an operation above the part. The RAPID command will activate the default RAPID speeds selected in the SETUP window. The RAPID speeds will remain in effect until a FEED command is written. The short form is the letter "R" that can be added as the last parameter on a MOVE command. The word RAPID on a line by itself must not be followed by any other parameters or values. All lines below this command will default to the Rapid feed of the axis motors as entered in the SETUP window.

Ex: RAPID

MOVE X1.25 Y.870 or

Ex: MOVE X1.25 Y.870 R



The JOG buttons resemble a pair of crosses on the left and right of this area. You may push any of the axis buttons (Y+/-, X+/-, Z+/-, or W+/-) at any time, provided a program is not running and a communication link to the CNC controller is active. Remember, the table will move in the opposite direction when using the Jog buttons (I.e.: If you depress the +X button the table will move to the left). To avoid confusion, always remember that the jog buttons are from the cutter's point of view.

JOG-INPUT feature or (TEACH MODE)

The Jog Input button, located in the center of each cross, function is a way of creating a program based on single movements. You can use the X, Y, and/or Z jog buttons to manually move along any of these axes. To record your location of all axes, click the Jog Input and it will record the information to your program. Example: You click +X with your mouse and you drive it 5". Then press the Jog Input button on the screen. The Master will then incrementally write that line on the editor. Then jog -Y at .032, press the Jog Input button, and the Master will record this action as the second line on the editor. This feature can be useful to build a program to use again in the future by driving one axis at a time. Keep in mind that the program is being written in INCREMENTAL mode, so be sure you write INCREMENTAL as the first line in your program before pressing GO to run this newly written program.

Disable Motors

To machine a part manually (without the use of the automated axis motors) simply make sure the port is "On" and press "Disable Motors". This will de-energize all motors. To re-enable the axis motors, press the Reset Control. You may disengage the motors for manual control at any time except during the running of a tool path program. The user can still maintain variable spindle computer control while the motors are disengaged. The DRO on the computer screen will not count the steps in the motors while disengaged unless your machine came with the closed loop accessory add-on.

2. Spindle Speed and Reverse Area



Spindle RPM with Override Use the OFF/ON button to enable or disable your spindle in manual mode. You may click the "Reverse" box at any time; however it will only take effect at the beginning of a spindle start. You may vary the speed by using the slider to the left in the window. The + and - buttons are use to adjust the spindle speed through the PC on the Master Interface while the program is running. These two buttons are used to fine tune speed during the running of a program after you press GO. Once you make your adjustment, you will see the speed increase or decrease after after the line that the machine is on.

3. Digital Readout and Axis Manual Zero Functions



Almost half of this window is used to provide digital feedback to the operator as to the physical location of the machine's X, Y, Z, and W axes. The second half has the Zero All, Set Zero, and other functions.

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

This window is used to provide digital feedback to the operator as to the physical location of the machines X, Y, Z, and W axes. As you operate the CNC machine manually or as a program executes, these counters will change.

Set Zero and Set Zero Return (These functions work together.)

Set Zero

and **Set Zero Return** move each axis of the machine to the desired physical zero position. Make sure that the counters for each axis are zeroed at this position by pressing ZERO ALL, and then press the SET ZERO button on the interface. Each axis of the machine will automatically move to the zero limit switches located at each motor bracket and then return back to the actual zero position while recording the distance of movement from each limit switch. **CAUTION: Make sure that there is nothing obstructing the path of the machine on the way to the zero switches, and be sure to clean any debris from the zero switch tips and their points of contact.** A command line will then show at the top of the program with the new saved zero position and list as SET ZERO. Now save the tool path program. Note: If there is any existing SET ZERO command line in the program, it will be overridden with this operation.

Once a SET ZERO position has been saved on the program, the machine can go to that position at any time and from whatever position it is currently at by pressing the SET ZERO RETURN button on the interface. Each axis of the machine will then automatically travel to the zero switches and then return to the saved SET ZERO command line at the top of your program. **CAUTION: If you override your SET ZERO position with a new SET ZERO line, you must also reset your Home line, or Tool Height comps if you are using these commands in your program as well.** In certain cases, if you require a different zero, or more specifically, a home position to change out your parts, or allow clearance to change out your tools (other than the saved SET SERO position), jog the axes to your desired home position. This leads us to the SAVE HOME and GO HOME commands:

Save Home and Go Home (These functions work together.)

Save Home and Go Home

Having two positions, a home position and a zero position, can help with changing out of parts or changing of tools, especially on a program where you have to machine several pieces using the same program. 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. You write the program's last tool paths ending with the axes returning to 0.00. You then press GO, and the machine mills out your part, and the axes end at the 0.00 positions for the next part. The problem: Due to the nature and size of the part, you have to move the table all the way to the left to have clearance to change the part out. This would be considered your home position, but your program's SET ZERO position is off to the right. The solution: with SAVE HOME, you have the ability to save this home position while still retaining your SET ZERO program position where the machine will mill your part out. Pressing SAVE HOME will record this new position outside of your program's set zero position on its own command line, which will be written underneath your SET ZERO line in your program. **It is important to only use this command after you have established and saved your SET ZERO.**

Pressing GO HOME will jog your three 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. This feature provides you many options as to how to machine your application in a production run. **It is important to only use the GO Home command from the machine's SET ZERO position. If you change the SET ZERO, then SAVE HOME must be changed as well.**

CAUTION: SAVE HOME and GO HOME will ONLY WORK IN REFERENCE TO YOUR SET ZERO position or SET ZERO RETURN position. Be sure to use the command SET ZERO so the SAVE HOME and GO HOME have a reference to your SET ZERO position. If you press the ZERO ALL, or ZERO X, Y, Z, W buttons, your SAVE HOME and GO HOME will be relative to that new 0.00 position on the displayed counters.

Return to 0.0

This command is simply a re-counting of steps back to the "last 0.00" position displayed on the counters. Use the command from the GO HOME position to bring your axes back to its 0.00 position on the machine. Please note that if your Z axis is displayed at 0.00, it will still move +0.002" up to clear the other axes and then drive back to 0.00.

If you press Zero ALL, or Zero X, Y, Z, or W buttons, jog the axes or run a program and stop it. Then press the RETURN TO 0.00 button and the axes will re-count their steps to this 0.00 position according to the displayed counters. This command is useful to quickly bring the axes back to the 0.00 position instead of having to press SET ZERO RETURN and wait for the axes to touch their limit switches and return to your set zero position.

Zero All

The Zero All button will clear all the counters and reset them to 0.00.

Zero X

The Zero X, Y, Z and W work just like the Zero All Button, but they clear individual counters and reset them to zero (0).

Zero Y

Zero Z

Zero W

4. Machine Status Window (bottom right)

RPM:0	0DG	Curr Tool: 1	CNC: no connection	USB ? /MILL
Maxrpm: 1200	Reset:	Z offset: 0.0000	D :	INCLES/OL
In Queue:			Run Time: 0 mins	Feed: 50.0

This window provides additional machine status information.

RPM:0 The RPM display shows the current cutter head speed in Revolutions per Minute (RPM). This number will increase as you manually (or through your program) change its value.

Maxrpm:

The Max RPM for your spindle motor can be found under your SETUP window in the Master Software. The default setting is at 1200 RPM. This value may need to be changed depending on your mill. Refer to the RPM Speed Chart and make the speed adjustment in the Max RPM box under your SETUP window. Be sure to save it. This is important to synchronize the machine speed belt or gear to the Master Software computer control. (Supra Mill Only: the Speed Chart is located in the Supra machine manual.)

Queued:

All tool path programs are line-oriented programs. They execute one line at a time; line one (1), then line two (2) then line three (3), and so on. The "Queued:" displays the line that was just fed into the machine's control unit.

Z offset: This displays the Z axis offset if you decide to use the Tool Height Compensation Feature. Be sure the display reads 0.0000 if you are not using Tool Height Comp. Hitting the [Reset:] button next to the Z offset display will bring this offset back to 0.0000 for the next run.

USB 4 This shows the USB port assigned to your CNC Control Unit. IMPORTANT: If the computer is used for other functions aside from running your Masters Software, be sure to plug in the USB cable to the same port you originally loaded the drivers into to avoid communication errors.

INCHES or **METRIC** shows the units of measurements in which the machine is configured. You can change it by pressing the Setup button at the top of the screen. Look under the "General" tab and select Inches or Millimeters. OL stands for Open Loop. CL stands for Closed Loop which is an optional add-on accessory.

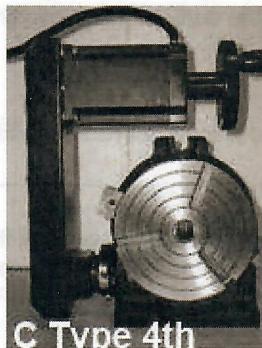
Feed: The Feed displays how fast the machine will attempt to cut your material.

ITEM 5. CNC MASTERS MILL: Accessories and Options

A. Fourth Axis Rotary Table Option

The CNC MASTERS MILL has true 4 axis capabilities using the rotary table option. Mount the rotary table to the table. The rotary table can be controlled by the Master Software using "W" commands. Go to the Setup button of the Master and check off your setting for fourth axis control. Save. If you have an older C-Type 4th Axis as shown below, place an additional check next to the C-Type 4th Axis and save.

General Tools Auto Tool Probe		
DRO Scale Calibration	X 0.5000	Y 0.5000
XY Rapid max feed	50	
Z Rapid max feed	30	
W Rapid deg/sec	125	
Scale	X1.0 Y1.0 Z1.0 W1.0	
Calibration Factor	X1.0 Y1.0 Z1.0	
Draw Delay (0-99)	0	Max RPM 1200
<input checked="" type="radio"/> Mill	<input type="radio"/> Lathe X=DIA	<input type="checkbox"/> 3D Printing
<input checked="" type="radio"/> Inches	<input type="radio"/> Millimeters	<input type="checkbox"/> 4th Axis Help
Return to 0.0		<input type="checkbox"/> C-Type 4th Axis
<input type="checkbox"/> Move XY First		<input type="checkbox"/> Pendant Enable
0.020 Z Return Clearance		Spindle Delay 2000
<input type="checkbox"/> Closed-loop	Autho. Code HX9-2915	



Rapid on the W: Although the W axis is displayed in degrees on the counters, the W axis internally changes the degrees into inches for the Master to properly drive it in combination with the other axes. For example, 9 inches of travel equals 360 degrees—one full revolution of the table for the C type 4th. The W feed can only be entered on the RAPID setting under your SETUP window in the Master Software. If you were to place a 9" value in the RAPID box, that would be interpreted as 9 inches per minute. So the rotary table will turn one complete revolution in one minute.

CAUTION: Do not lift the fourth axis from the motor and bracket. Carry the fourth axis from the rotary table only.

B. CAD-CAM Software

The Master Software works with FANUC G-code. If you plan to use your own CAM separate from what CNC MASTERS sells, use a FANUC based post processor and re-write it to be compatible with our list of G-code. The list of codes can be found by going to the top left of the MX screen under Tools > G Code / M Code List.

The setup depends on your software, but for the most part the only thing that is necessary for your CAM to work with the CNC MASTERS Machine is the post processor. The post processor can be obtained from the software manufacturer of your CAD-CAM or you will have to write one based on the G code list provided in the MX. As another option, CNC MASTERS sells a CAM with a post processor built for our machines.

Once your file is written, double check that your G-codes are supported in the MX. Another important factor to note when generating your tool path program, be sure adjust the max resolution to 0.0002" in your CAM preferences. Be advised, if you do not require a high resolution on your part, adjust the max resolution and the CAM post processor to point three decimal places back. This will allow for a faster tool path. Keep in mind the higher the resolution your tool path program, the slower the feed rate you must have to accommodate for these small movements. Do not run a fast feed rate on small movements or it could lead to inaccuracy on the part you are machining. Following this procedure will prevent the CAM from generating values smaller than what the CNC MASTERS Machine can support. If you are working in mm mode, the minimum resolution setting in mm is 0.005 -- only if the CAM can generate multiples of .005 mm. If your system cannot handle multiples, then it is best to put your resolution setting at .01 mm so your system can generate the correct values to run the program.

SEC 3.

Computer Control Variable Spindle Speed Drive/Manual Control of Axis Motors

Variable speed drive can be controlled by the Master Software and the keypad pendant optional accessory. This only applies to machines with computer variable spindle control. The CNC MASTERS Mill does not have a spindle motor power switch. Powering the spindle is done through either the Master software, or the optional hand held remote control. (The power switch on the Supra Mill only adjusts your hi or low settings.)

Speeds and Torque

A Speed RPM chart is needed for your particular CNC MASTERS Mill:

Jr. Plus Mill:

At the top of the machine head pulley housing cover, you will find a large sliding tab at the right center. There you will find the chart on the inside of the tab. Having the belt on the highest settings will give you the ability to have control of the spindle motor from 0 to 3000 RPM. However, if a high torque is needed, it will be necessary to adjust the belt to the proper setting which you will be machining your part at. Example: If you are working at a speed of 1100 RPM and need a high torque, you will need to adjust the belts so that the speed will be in the 1100 range. Working on the higher belt range will still give you 1100 RPM, but the torque may not be as strong as you need it to be. Much will depend on the material being used and the tool path you are creating.

Baron Mill:

You will find the chart at the front face of the machine head. Having the gear on the highest setting will give you the ability to have control of the spindle motor from 0 to 2000 RPM. However, if a high torque is needed, it will be necessary to adjust the gear to the proper setting which you will be machining your part at. Much will depend on the material being used and the tool path you are creating.

CAUTION: NEVER ADJUST GEARS WHILE THE SPINDLE IS ON. ROTATE THE SPINDLE BY HAND WHEN ADJUSTING GEARS TO PROPERLY ENGAGE THE GEARS INSIDE THE HEAD.

Max Mill:

You will find the chart at the front face of the machine head. Having the belt on the highest settings will give you the ability to have control of the spindle motor from 0 to 4500 RPM. However, if a high torque is needed, it will be necessary to adjust the belt to the proper setting which you will be machining your part at. Example: If you require less than 1680 RPM, adjust the belt position for a stronger torque. Much will depend on the material being used and the tool path you are creating.

CAUTION: IF YOU HAVE THE BELT ON THE HIGHEST RPM SETTING (4500), DO NOT DRIVE THE SPINDLE IMMEDIATELY TO THE MAX 4500 RPM WHILE THE SPINDLE IS "COLD." Allow the bearings inside the spindle to warm up, especially in a cold shop environment. The sealed bearings at the bottom of the front pulley need to warm up the bearing grease first, especially in a cold shop environment when you are powering up the machine for the first time in your day.

Supra Mill:

You will find the chart in your CNC Supra Machine Manual under Sec. 4, along with your settings for the high and low gears. Having the belt on the highest setting will give you the ability to have control of the spindle motor from 0 to approximately 5600 RPM. However, if a stronger torque is needed, it will be necessary to adjust the belt to the proper setting which you will be machining your part at. Example: If you are working at a speed of 1100 RPM and need a stronger torque, you will need to adjust the belts so that the speed will be in the 1100 RPM range. Working on the higher belt range will still give you 1100 RPM, but the torque may not be as strong as you need it to be. Much will depend on the material being used and tool path you are creating. To machine at higher or lower RPM, you will need to refer to the chart as well as to the machine's manual.

CAUTION: Rapidly increasing or decreasing the RPM with the movement of the mouse may cause the spindle to automatically shut-off. Alternating the RPM quickly back and forth may cause an automatic shut down as well. Move the PC mouse gently when increasing or decreasing the RPM. Allow the motor's RPM to catch up to the movement of your spindle with your PC mouse.

SEC 4.**PROGRAMMING YOUR CNC MILL WITH THE MASTER SOFTWARE—IMPORTANT SECTION.**

OPERATORS: PLEASE READ THIS IMPORTANT SECTION TO UNDERSTAND THE FUNCTIONALITY OF HOW THE MASTER SOFTWARE WILL COMMUNICATE TO THE CONTROL UNIT OF YOUR CNC MASTERS MILL.

Before beginning, it is important to close down any networks and/or other programs that your computer may be operating as this may cause a disruption during the operation of the Master Software. If you happen to leave the computer idle between jobs, it is important to shut down the hibernation/sleep mode on your computer. Keeping the control unit powered on and allowing the computer to go into hibernation mode can cause communications errors, which can only be removed by re-booting the computer and powering off/on the control unit.

There are two specific things to keep in mind as we go forward. Firstly, the Master Software comes pre-installed with user-friendly conversational commands under the PICK button that will help you create programs immediately. A complete list of commands is located towards the end of this section. Secondly, the basic programs listed below will give you an understanding of how the software expects to see your program. Please remember that these commands can do a lot, but there are things you will simply not be able to do without a quality CAD and/or CAM system since the object you are thinking of creating may be too complicated.

Exercise 1: ABSOLUTE Mode

For the first exercise you will make a 1" square using an ABSOLUTE set of instructions. The ABSOLUTE command sets the Absolute mode on. All moves after this command will be relative to the machine's zero position. The Master Software is defaulted to Absolute mode.

Exercise Overview: Let's do a brief run-through of the machining of the 1" square in Absolute mode before we go into detailed instructions. First, we will position the cutter so that the top left corner of the square is the 0 point for the X and Y axes. Next we will type in X1.0; this will cause the cutter to cut 1 inch in the X direction (All movement of the axes is from the viewpoint of the cutter). Next, we will type in Y1.0, which will cause the cutter to cut 1 inch in the Y direction. Now, since all cuts in Absolute Mode are relative to the Zero position, the objective of the next two (2) commands will be to return to the initial Zero position. We will now type in X0.0. This will cause the cutter to move back 1 inch in the X direction to the zero (0) X position (the bottom left corner of the square). Finally, we will type in Y0.0, which will move the cutter back 1 inch in the Y direction, hence completing the square in Absolute Mode. We will now go into further detail of this process below.

On the File menu, select NEW. This will clear the white editor screen and the memory buffers.

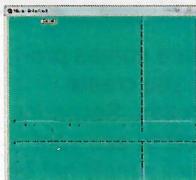
Jog the two axes using your PC mouse so that they line up at the center of the machine. Be sure your Z Quill is elevated enough and out of the way. **To avoid confusion between POS and NEG directions, always look at your axis movement from the point of view of the cutter.** For example X POS is the cutter cutting in the direction to the right. However, the X table is moving in the left direction. If you are new to machining, always look in the direction in which the cutter is cutting. The Master Software interface and the optional Hand Held Remote Control are also designed in the viewpoint of the cutter.

From the PICK menu, which is found on the Editor Screen, select ABSOLUTE. The ABSOLUTE command will read each tool path line relative to the machine's zero position. The selection will be copied to the editor. Position the cursor on the next line. After each line, be sure to reposition your cursor when inputting commands manually. From the PICK menu, select MOVE. The word MOVE will appear on this new command line with a line number to properly identify this line. It is important when writing a program on the editor that the first line be positioned at the top of the editor screen and that there are no line spaces.

Always use a space between key words and data. Type X1.0 after MOVE to complete the command line. Be sure not to confuse the number zero 0 for the letter o.

For the next command, pick MOVE and add Y1.0. Repeat the above procedures for X0.0 on its own line and Y0.0 on its own line. Finish by inserting an END command line, also on its own line. All programs must terminate with an END command line. **Important: Do not leave spaces between tool path lines.** This will cause the Master Software to freeze when it reads the space after the tool path line above. Also, be sure that the first line begins at the top left of the editor screen when you write your program.

```
N00 ABSOLUTE  
N01 MOVE X1.0  
N02 MOVE Y1.0  
N03 MOVE X0.0  
N04 MOVE Y0.0  
N05 END
```



Verify your data graphically by pressing the DRAW button at the bottom of the editor screen. The CANCEL button at the top left side of the DRAW window is only used for very large files where you wish to stop the command from generating and drawing the remainder of your program. Use the X button at the top right to close the DRAW window. Your program is now ready to run. Save your program and name it (example name - TEST). You may also verify your program by using the TRACE command.

The TRACE command allows you to easily verify each line of your program independently. If the control unit is on, the TRACE will run each line independently as well. Another way to verify each line of your program is to use the TRACE command in combination with the DRAW command. Press TRACE first, then press the DRAW button. Now press the RUN NEXT LINE button and you will see the drawing take place one line at a time. This will not operate the machine one line at a time though as in the TRACE command by itself.

NOTE: The dashes in the DRAW window represent the tool path in RAPID mode. Solid lines represent tool paths in FEED mode. You can also enter an F0.25 on the N01 Line above: N01 MOVE X1.0 F0.25. All lines underneath it will now travel at a feed of 0.25"/minute. The square will also change to solid lines in the draw window.

Once the program is verified, close the windows. Press RESET CONTROL and verify that the CNC: is OK at the bottom right of the screen. **Always get in the habit of pressing RESET CONTROL prior to the start of any job.**

Zero all counters by pressing the ZERO ALL button. **PRESSING THE RESET CONTROL BUTTON ON THE MASTER PROGRAM IS RECOMMENDED ALWAYS PRIOR TO THE START OF A NEW JOB, REPEAT JOBS, OR WHEN POWERING UP THE CONTROL UNIT. PRESSING THE ZERO ALL BUTTON IS ALSO RECOMMENDED EACH TIME YOU PRESS THE RESET CONTROL BUTTON TO ENSURE THAT ALL COUNTERS ARE AT 0.00.** From time to time upon pressing RESET CONTROL, the counters may jump a few digits. This is normal and it will not cause the axes to physically move. This is why we have you press ZERO ALL after pressing RESET CONTROL. If you happen to press GO and there are values on the counters, these values can distort the tool path program.

Press the GO button to run the program. The cutter will move in the direction of a 1" square.

Congratulations! You have successfully completed your first CNC program.

RECAP: BEFORE YOU PRESS GO:

- 1. PRESS RESET CONTROL**
- 2. PRESS ZERO ALL**
- 3. VISUALLY CHECK TO SEE COUNTERS ARE 0.00**
- 4. PRESS GO.**

MOTOR ENGAGEMENT/DISABLE

If you decide to manually hand crank the axes to your zero position, and then power on the control unit, keep in mind that powering on the control unit will also energize/engage the axis motors. As the motors engage this may cause the motor shafts to slightly shift. Double-check your zero position before running your program. It may also be best to energize the axis motors and jog them into your zero position with the use of the PC mouse, keyboard, or optional hand held remote. However, you can disable the axis motors at anytime when not running a program by either powering off the control unit, or clicking on the Disable Motors button located above the X, Y, and Z counters display. First, be sure the Port is On to disable the motors. Your motors will now be disabled, allowing you to manually hand crank the motors.

Exercise 2: INCREMENTAL Mode

For the second exercise, you will make a 1" square like in the first exercise, but you will use the INCREMENTAL mode. The INCREMENTAL command cancels the Absolute mode, which is the default, and sets the Incremental mode. All moves after this command will be relative to the ending point of the previous line. In other words, it adds from one line to the next.

Exercise Overview: Let's do a brief run-through of the machining of the one-inch square in Incremental mode before we go into detailed instructions. First, we will position the cutter so that the top left corner of the square is the 0 point for the X and Y axes. We will first type in X1.0, which will cause the cutter to cut 1

inch in the X direction. Next, we will type in Y1.0, which will cause the cutter to cut 1 inch in the Y direction. Now the objective of the next 2 commands will be to complete the square. Any commands given in Incremental mode will only have reference to the last given command. So in this case, we are at the X1.0 and Y1.0 position currently, and we need to type in commands that will get us back to the initial zero (0) position to complete the square. We cannot, however, simply type in X0.0 as we did in Absolute mode to return to the zero (0) X position because, in Incremental mode, the command X0.0 would simply tell the machine to "move zero inches in the X direction", which basically tells the machine not to move at all. Instead, we must type in X-1.0, which tells the cutter to move 1 inch in the negative X direction (back one inch in the X direction). This command has now just drawn the bottom left corner of the square. Now to complete the square, we must type in Y-1.0, which will cause the cutter to cut 1 inch in the negative Y direction. Similar to in the previous case, we cannot simply type in Y0.0 to go back to the starting point because, in Incremental mode, the machine would interpret this as "move zero (0) inches in the Y direction". Now, after typing in Y-1.0, the cutter will cut 1 inch in the negative Y direction, hence completing the square in Incremental mode. We will now go into further detail of this process below.

Like your first exercise, create a new file and enter the program below. When you hit the draw command, you will see it is the same square written Incrementally.

```
N00 INCREMENTAL
N01 MOVE X1.0 F0.25
N02 MOVE Y1.0
N03 MOVE X-1.0
N04 MOVE Y-1.0
N05 END
```

Exercise 3: THE CIRCLE

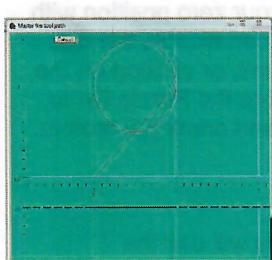
This is a circle with a 1" diameter.

```
N01 MOVE X1 Y1 F2
N02 CCWCIRCLE X1 Y1 I-0.5 J0 F0.5
N03 END
```

The MOVE line will move the tool to 45°. The CCWCIRCLE line will make the circle move counter clockwise.

Select NEW on the "File" menu. This will clear the white editor screen and the memory buffers. Jog the two axes using your PC mouse so that they line up at the center of the machine. Be sure your Z Quill is elevated enough and out of the way. Line (1) will move the cutter out 1 inch in the X and Y direction. Line (2) will move the cutter using the CCW (Counter Clockwise) circle.

Copy the above program, click the "Draw" button, and you should see the image below.



Exercise 4: THE DRILL USING A CALL SUB ROUTINE

```
N00 ABSOLUTE
N01 MOVE X1.0
```

```
N02 CALL DRILL 1  
N03 MOVE Y1.0  
N04 CALL DRILL 1  
N05 MOVE X0.0  
N06 CALL DRILL 1  
N07 MOVE Y0.0  
N08 CALL DRILL 1  
N09 END  
N10 DRILL  
N11 MOVE Z-.250F5  
N12 MOVE Z.0R  
N13 END
```

The CALL command can be a very useful tool for nesting type applications and can simplify drilling applications. You can machine the same small part a few times on one program.

Continuing with the first exercise, let's extend the TEST program and add a drilled hole at each corner of the square. First you will write a subprogram or subroutine to move the Z-axis 0.0250 down and back. Place your cursor on the next line after the END line in your square program. Type in "DRILL" as the name of the routine. The name "DRILL" is arbitrary. You can pick any word you want that will help you name your subprogram. Use alphabet letters only. Do not use characters other than alphabet letters to name your subroutine.

Pick MOVE and enter Z-.250. Now we want to select a feed speed because if we're cutting into steel, for example, we don't want to drill out a hole at 50" per minute. For the sake of this exercise, we will enter 5, which means the machine at this point will now travel at 5 inches per minute. Feed is denoted by the letter F, so it will look like: Z-.250F5.

The square program written was simply defaulted to your RAPID feed settings since no FEED was added into the initial program. The RAPID setting can be changed in the SETUP. Next, pick MOVE and enter Z0.R. Now since we are moving the cutter back up from the work piece we can select RAPID, which will be denoted by the letter "R" after your Z movement, written as: Z.0R. Lastly, pick END to end the call routine. Go to the main program (not the routine) and pick CALL after each MOVE line. Each call line should read CALL DRILL 1. A number after a call label (DRILL) tells the Master Software to repeat the routine (only once in this case). If you are only going to run the subroutine one time, it is still necessary to place a value after the call label. Save this program and name it CCWTEST1.

Now you may run your program. Use the TRACE command in combination with the DRAW command to understand how a CALL routine works.

USEFUL TIP: Always check the DRAW feature to ensure your drawing is read correctly on the Master program. Run the program in the air first. Using machine-able wax or wood to mill a part is always a good idea if you need to experiment with your tool path first. This will avoid breaking end mills or wasting metals if you are still in a trial and error stage of your tool path program.

SAMPLE FILES LOCATION

If you are interested in taking a look at some of our sample files, simply go to the top left of the MX screen. Go to File > Open. In the Open window, drop to MasterMX in the "Look In:" drop down box. Inside the MasterMX, you will find the Sample Files Folder.

RECOMMENDATION: Power off the Control Unit at the stop of each job instead of letting it idle. This will allow the control unit and motors to cool off in between jobs if you are running for several hours in a day.

SEC 5.

MASTER Command List (found under your PICK Button)

The following is a description of all CNC MASTERS MILL Commands listed in alphabetical order, which can be used instead of using a CAD-CAM software to generate your G-code program.

ABSOLUTE (G90)

The ABSOLUTE command sets the Absolute mode on. All moves after this command will be relative to the program home/zero position. The Master Software is defaulted to Absolute mode as well.

INCREMENTAL (G91)

The INCREMENTAL command cancels the Absolute mode, which is the default, and sets the Incremental mode. All moves after this command will be relative to the ending point of the previous line. In other words, it adds from one line to the next. Placing the ABSOLUTE command after programming lines in the Incremental mode will default the lines underneath to Absolute.

CWCIRCLE X Y I J (G02)

This command will move the tool along a circular arch in a clockwise direction. X and Y indicate the ending point of the arch. I and J indicate the center of the arch relative to the tool position at beginning of the arch. I and J must be assigned according to the direction. I is relative to the position of X. J is relative to the position of Y. When adding this line from the PICK Menu, be sure to delete the note "mill". When using G02/G03 commands, only go on a slow feed.

If you are using a CAM software to generate your G-codes, be sure a Z coordinate does not appear on this line unless you require a helical movement to take place. See **Helical Movement** below.

CCWCIRCLE X Y I J (G03)

This command will move the tool along a circular arch in a counter-clockwise direction.

EX. This is a circle with a 1" diameter.

```
N01 MOVE X1 Y1  
N02 CCWCIRCLE X1 Y1 I-0.5 J0 F0.5  
N03 END
```

Helical Movement: When choosing to do **helical movement** with the Z-axis on either of the two circle (G02 and G03) commands, the Z value can be read in Incremental or Absolute mode. If you are NOT doing a helical movement, do not place a Z value in the G02 or G03 tool path. Any Z value placed in a G02/G03 line will command the Master to read the Z value in your program.

EX: The following is an example program of a square with rounded corners. This example will give you an indication of how to use the circle commands from the PICK menu. You can check the program's drawing by pressing the DRAW button on the bottom of the screen.

```
N00 ABSOLUTE  
N01 MOVE X1.0 F5  
N02 CCWCIRCLE X1.5 Y0.5 I0 J0.5 F0.5  
N03 MOVE X1.5 Y1.5  
N04 CCWCIRCLE X1.0 Y2.0 I-0.5 J0  
N05 MOVE X0 Y2.0  
N06 CCWCIRCLE X-0.5 Y1.5 I0 J-0.5  
N07 MOVE Y0.5  
N08 CCWCIRCLE X0 Y0 I0.5 J0  
N09 END
```

CALL

A CALL command allows you to write an incremental subroutine outside the main program. After the key word CALL, you must enter the name of the subroutine and an optional repeat factor. The running program will jump into the subroutine of the corresponding name and will repeat it as indicated by the

repeat factor. You may have any number of CALL commands in a program, but the subroutines must be written after the END command of the main program. Each subroutine starts with the subroutine name and must terminate with an END command. Subroutines may contain CALL commands to other subroutines.

EXAMPLE:

```
N00 INCREMENTAL
N01 SPINDLE ON SPEED 1000 FORWARD
N02 CALL DRILL 10
N03 MOVE Y1
N04 CALL HOLE 10
N05 MOVE Y1
N06 CALL DRILL 10
N07 MOVE Y1
N08 CALL HOLE 10
N09 MOVE Y-3
N10 SPINDLE OFF
N12 END
DRILL
N14 MOVE Z -.2F30
N15 MOVE Z.2
N16 MOVE X-1R
N17 END
HOLE
N19 MOVE Z-.2F30
N20 MOVE Z.2
N21 MOVE X1R
N22 END
```

Notes on above CALL sample program:

N00: All CALL commands require the program to be written in INCREMENTAL mode.
N02: This program begins with a CALL DRILL subroutine. This subroutine is written on N14 through N17 and is programmed to run ten times. You will see that the DRILL subroutine also contains MOVE X-1R on N16. Writing the subroutine this way allows the user to drill several holes without having to write several individual tool paths. N03, N05, N07, and N09 are tool paths that jog the drill to the next position on the part that will require another 10 holes to be drilled. This sample program above has two subroutines to indicate that several subroutines can be placed within a given program.

COOLANT (M08, M09)

This command turns the system coolant on/off. It may be inserted into any part of your program.

Ex: COOLANT ON
COOLANT OFF

DELAY

The Delay command allows you to stop the program at a particular point and delay further execution for the time shown in milliseconds. After time runs out the program will resume. So let's say you enter DELAY 40000 on its own line; this will give you a delay/pause of 40 seconds. Forty (40) seconds is the maximum delay time.

Ex: DELAY 1000

PAUSE / FEED HOLD

The program will pause at the conclusion of its current tool path line, and will only resume upon pressing the OK/Resume button on the prompt window. The PAUSE command can also be done using the hot keys button on the Master interface. Press "P" on your keyboard when you need to pause. The pause command will take place once it finishes the current tool path it is on. In the prompt window, you will also have the option of also pausing the spindle. Caution: If you jog the axes during pause mode, you must drive your axes back to that last counter position or this can throw off your program.

The PAUSE command can also be found in your PICK menu and can be entered in your tool path program. Please note it will not work if you place it between two RUNFILE commands. In this case, if you must place two RUNFILE commands on the same program, and have to PAUSE in between the two commands, then place PAUSE as the first line on the second RUNFILE program.

END

This command marks the end of your main program or subprogram like the M30 code. There is a slight difference between the END and M30 commands. END is primarily used to define the ending of a subroutine within a CALL command. If you choose to use END instead of M30 at the end of a program, you must use a SPINDLE OFF or M5 before END to shut the computerized spindle if applicable. M30 will automatically shut everything off at the end of the program.

RAPID

The RAPID command is typically used to quickly retract or relocate a cutting tool after an operation such as a simple plunge or complex set of instructions. The RAPID command will activate the default RAPID speeds selected in the SETUP window. The RAPID speeds will remain in effect until overwritten by a FEED command. The short form is the R letter that can be added as the last parameter on a move command. The word RAPID on a line by itself must not be followed by any other parameters or values. All lines below this command will default to the rapid feed of the axis motors as entered in the SETUP window.

Ex: RAPID

MOVE X1.25 Y.870 or

Ex: MOVE X1.25 Y.870 R

In the SETUP window, the RAPID command must be followed by X, Y, Z and W parameters. They set the rapid speed for your machine. The maximum feed is 100" per minute of travel. Selecting faster speeds than what your system allows may cause the axis motors to stall. The Master software defaults the RAPID setting at 50" per minute at each axis. It is advised to leave the Z axis at 30" especially if your applications do not require faster feeds. This will avoid accidentally plunging of the quill. The RAPID command in the PICK menu will activate the default RAPID feeds written in the SETUP window.

CAUTION: If you would like to increase the RAPID, we recommend increasing it to 50" max. Practice safe rapids on the machine especially in a small shop environment where other people are close to the machine. Take pre-cautions for safety around the machine if you decide to run rapid higher than 50" a minute. Remove the handles off the X and Y hand wheels as well to avoid getting caught on clothing.

FEED

The FEED command will activate the default feed rate selected in the SETUP window. You may also enter a new feed rate immediately after the word FEED. The feed command will remain effective until overwritten by another feed command or rapid command. The short form of FEED is <F> followed by the feed value. It may be added at the last value of a move command line.

Ex: FEED 3

MOVE X1.25 Y.870 or

Ex: MOVE X1.25 Y.870 F3

FEED OVERRIDE

Do not use the + and - buttons, located over the top slider, to adjust your feed before pressing GO. These two buttons can be used to adjust +/- 10% feed during the running of a program after you press GO. Once you make your adjustment, you will see the feed increase or decrease after approximately three lines of tool paths have finished executing on the machine.

MOVE (G00, G01)

Here, you may enter your X, Y, Z or W coordinates. You may enter one or more axes on the same command. All axes will interpolate. The speed will be according to the last selected RAPID or FEED

command. You can also write G00 or G01 in lieu of the word MOVE. G00 simply jogs the axes at the RAPID setting stored in your Setup window. G01 jogs the axes according to the feed rate you place at the end of the tool path; denoted by the letter <F> followed by inches per minute or mm per minute value.

PRE_SET

Pre-Set will help you if you cannot start from a 0.00 position. With Pre-Set, you can enter the coordinate from which you want to begin the tool path program. Example: The Z axis is 0.5" above the material. Pre-Set will automatically write in the Z coordinate at 0.5" on the counter display.

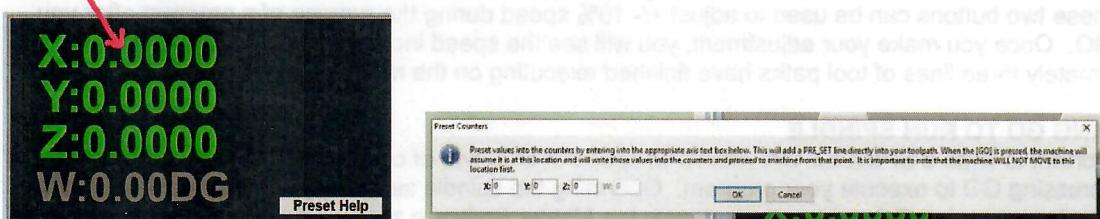
You can also pre-set a coordinate directly into the counters by clicking the cursor over the counter to the right. This will highlight the counters and you can then enter a pre-determined coordinate on the counter itself. This way you can still follow the operational procedure of hitting Reset Control, and making sure all counters read 0.00 before pressing GO. **Each axis must have its own pre-set coordinate.**

Example:

```
N01 PRE_SET X0.125  
N02 PRE_SET Z0.25  
N03 SPINDLE ON FORWARD SPEED 1500  
N04 MOVE X0.00 F1  
N05 MOVE Z0.00
```

Upon pressing GO, if the first line in your program is PRE_SET X.0125, the X counter will then display 0.125. Then the Z counter will change to 0.25. This will not cause the machine axes to move. The pre-set is simply telling the Master that you are beginning from a position other than 0.00.

You can also pre-set a coordinate directly into the counters by clicking over the counters. This will prompt a dialog box so you can enter a pre-determined coordinate on the counter itself.



Now before you enter this pre-set coordinate, hit Reset Control, and Zero All. Make sure all counters are at 0.00, then enter your preset coordinates and press GO. The Master will then begin the program from these pre-set coordinates. You need to make sure that the first tool path in your program will correspond to the pre-set values.

CAUTION: Be sure to enter the pre-set coordinates according to your program written in Absolute mode only. As always, play with this feature in the air first to make sure the logic makes sense before cutting something out.

PROGRAM NOTES or REMARKS

The backslash (\) indicates a comment line. Up to 70 characters may be used to add comments or notes to the program. The MASTER Software will ignore all text following the backslash during a program run.

IMPORTANT: Note lines in your program will break the modal sequence of your program. CAUTION: If you insert a NOTE into a modal sequence, make sure to add the proper G-code on the following line. Example: Let's say you have 10 lines in a program. Line 1 is written as a G01, and G01 is the modal all the way up to line 6. Now, if you want to place a note at line 5, be sure to write G01 on line 6 as well. **If you write a note into your program, keep the note on its own line. Do not share the note with another command.**

RUNFILE

With the RUNFILE command you may run an external G-Code file or another Master Software file. If the file is not in the directory, then the correct DOS path must be included with the file name. The RUNFILE command may be necessary for large tool path files. If you are having difficulty cutting and pasting a file on the interface, it is because the file is too long and it will need to be opened through the RUNFILE command. Using the RUNFILE command only requires the user to enter this command on the first line of the Master interface. Press GO and the Master will read the tool path directly from the directory where your tool path was originally saved.

Ex: RUNFILE C:\MY DOCUMENTS\MY FOLDER\MY DRILLING FILE.TXT

There is a restriction; the RUNFILE command cannot run an external Master Software file that contains CALLS or other RUNFILE commands.

SPINDLE (M03 = ON, M04 = REV, M05 = OFF)

This command allows you to control the spindle on/off, direction, and spindle speed. After the key word SPINDLE, you can enter one or more of the following key words: ON, OFF, FORWARD, REVERSE, and RPM. Make sure that the words are separated with a space character, and that the word "RPM" is followed by a number from 0 to 2000 corresponding to the RPM selected. If you prefer to write in your spindle command through the M code, Place <S> followed by the RPM value.

Ex: SPINDLE ON FORWARD RPM 2000

M03 S1000

SPINDLE OFF

SPINDLE REVERSE

SPINDLE SPEED OVERRIDE

Do not use the + and – buttons located to the right side of the slider to adjust the speed before pressing GO. These two buttons can be used to adjust +/- 10% speed during the running of a program after you press GO. Once you make your adjustment, you will see the speed increase or decrease after approximately three lines of tool paths have finished executing on the machine.

PRESSING GO TO RUN SPINDLE

It is important to write a SPINDLE ON with RPM speed or M03 line of command in your tool path program before pressing GO to execute your program. Operating the spindle separately before pressing GO will not work since pressing GO automatically resets the Master Software and will shut down the spindle if started outside of the program. Pressing GO automatically resets the Master program intended for users doing production runs of the same program.

SEC 6.

G-Code File Interpreter in the Master Software

What is a G-code? A G-Code is the common name for the most widely used computer numerical control (CNC) programming language, which has many implementations. Used mainly in automation, it is part of computer-aided engineering. G-code is sometimes called **G programming language**.

In fundamental terms, G-code is a language in which people tell computerized machine tools what to make and how to make it. The "what" and "how" are mostly defined by instructions on where to move to, how fast to move, and through what path to move. The most common situation is that a cutting tool is moved according to these instructions, and cuts away excess material to leave only the finished work piece.

G-codes in your file that are not listed below will not be read by the Master program and will cause inaccuracy in the machining of the part, or wrong-construction error flags to appear.

Resolution

Keep in mind the resolution if you type in coordinates directly into the editor or use a CAM software to generate your programs. The minimum resolution setting in inches is 0.0002". The minimum resolution setting in mm is .005 -- **only if the CAM can generate multiples of the .005 mm.** However, if your system cannot handle multiples, then it is best to put your resolution setting at .01 mm so your system can generate the correct values to run the program.

G-code and M-codes supported:

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. Adding a Z coordinate on this line can be done for helical movements. When using G02/G03 commands, go on a slow feed only. Fast feeds on interpolated movements can destroy the part you are machining.

G17 = Cancels G60 Command

G70 = Input in inches

G71 = Input in millimeters

G73 = High-Speed Peck Drilling Cycle

G81 = Drilling Cycle

G82 = Counter Boring Cycle

G80 = Cancel G73, G81, G82, or G83 Cycle

G83 = Deep Hole Standard Peck Drilling Cycle

G90 = Absolute move (Modal)

G91 = Incremental, relative move (Modal)

M00 = Pause

M03 = Spindle on

M04 = Spindle on reverse

M05 = Spindle off

M08 = Coolant on

M09 = Coolant off

M30 = End program

With the G-CODES FILE INTERPRETER, G-Codes produced from other CAD-CAM programs can be read and used by the Master software. If the G-code file is large, then it is recommended to insert a "RUNFILE" command in your program. This is only necessary if you see your file getting cut off on the editor screen. See RUNFILE command in SEC 5 MASTER COMMAND LIST.

ALSO BE ADVISED: CHECK your numbers on your G-code file carefully. A common user error is to write in the letter "O" in place of the number "0". Double-check that the codes are WRITTEN IN THE SAME FASHION as in your sample files, and also that the codes are supported in the list above to prevent downtime in trying to determine the error. If an error ever occurs, try using the TRACE command, which helps you to see line by line where your error may be.

Peck, Drilling, Boring Cycles:

G73 = High-Speed Peck Drilling Cycle: X__Y__Z__R__Q__P__F__

G81 = Drilling Cycle: X__Y__Z__R__F__

G82 = Counter Boring Cycle: X__Y__Z__R__P__F__

G83 = Deep Hole Peck Drilling Cycle: X__Y__Z__R__Q__P__F__

G80 = Cancels the above cycles.

Coordinate Definitions for Cycles above:

X, Y: Specifies the hole position by an Incremental or Absolute value.

Z: Specifies the total depth of the hole.

R: Specifies the return point. After each cut-in value the drill will return to the R point. The R point is above the work piece for G83. For G73, the R point retracts a small pre-determined distance to break the

chips and then feeds to the next peck, allowing the tool to go deeper. (FYI: In G83, the R point is read as an Absolute Value. The R point in G73 is read as an Incremental Value.)

Q: specifies each cut-in value with a negative Incremental value. The peck will not work if Q is left at 0.00.

P (optional): Specifies dwell time in milliseconds at the bottom of drilled hole.

F (optional): Specifies feed speed.

Peck-drilling cycle example:

G91

G83 X.5 Y.7 Z-.5 R-.1 Q-.05 P500 F5

X.5Y.8 (These next three lines represent the X and Y coordinates where the peck drill, drill, or bore will continue.)

X-.3Y1.

X.2Y1

G80 (This line cancels the cycle.)

Note: All cycles can be written either in Absolute or Incremental mode.

All coordinates X, Y, Z, R, and Q MUST be written after G73 or G83 for the peck drill cycle to work correctly.

SEC 7.

Master Software Additional Commands

TOOL CHANGE and HEIGHT COMPENSATION with AUTOMATIC QUILL RETRACTION

When using the Quick Tool System, setup of the tool height compensation in the Master program can be done by going to SETUP, then TOOLS. Place your first tool at zero on your part, then place your second tool at zero as well and see how much compensation is needed. Enter the +/- figure in T2 for each tool and save. T1 is defaulted at 0.0 to indicate your first tool. When you need to change out a tool, write T2 in your program on its own line. Once your program is running, it will pause when it reaches T2, and a window will pop up with a note to change out the tool and the diameter size you need. The compensation will happen automatically. This feature is also useful to pause the program to change out a tool. If you do not require height compensation, just leave it at 0.0. The Diameter and RPM boxes are reminders for you to know which tool to use, or if you plan to use the tool offsets G41/G42. (Movement of Z, including height compensation, can only be downward outside of a G41/G42 cycle.) To properly use the automatic quill retraction, it is important to make sure that your program's first line is the line written by the SET ZERO command. The quill will automatically retract up and stop before making contact with the Z home switch when using the Tool Height commands. It is NOT recommended to use the Tool Height Feature on standard collets. Only use collets that allow you to secure and set the cutter to keep from sliding in the collet. Please see our selection of Quick Change tools that we offer as an option. Also, do not jog an axis, or change the counters by zeroing back to 0.0 while the Master is in tool change mode. This will cause you to lose the positioning of the Master if you are in the middle of running a program.

After you run the program, you will need to reset using the "Reset" button under the Tools tab in the Setup window. The Master will continue to offset all Z movement (including any movement using the Relative On command or any jogging on the Z) if you do not reset it. Note that when an offset is active, it will appear at the bottom right of the screen where it says "Z Offset" under the counters. The Z Offset will show a value other than zero to show it is active based on the last offset you did on the program. To default the Z-axis back to 0.00, be sure to hit Reset in the Tools window to delete the offset.

Be sure to either write down the individual height compensation coordinates as a note using a "/" after a T command line. (ie: T2 / -.125 offset from T1) This way, you can quickly enter the values at the Setup > Tools window should you decide to come back to this program another day.

Production Run Hint using T commands and Resetting Z-axis:

Now if you are doing a production run on one program using several tool changes, add a T1 toward the end of the program. In doing so, this will bring your cutter back to its Z0.0 position (T1 position) for your

next part without having to press the RESET button on the Tools section every time. By doing this, you will not need to bring the Z up separately for the same program.

If you would like to receive a "detailed step by step" tutorial on using Tool Height Comp using Quick Change Tools, send us an email at customerservice@cncmasters.com.

SCALE

The scale command, found in the SETUP window, allows either one or all of the axes to be scaled up or down. This feature can be used to miniaturize or double the size of your part without having to re-write the program. If an axis does not require a scale factor, use a scale of 1.0 to indicate a scale of 100%. If you place 2.0 on the X Y Z, your part will be machined at double the size of the written program. The scale command can also be used in the situation where there is absolute "repeatable error" on a given axis. You can use the scale to +/- compensate for the error. 5 turns equal one inch of travel on the pitch of the ball screws of the machine. The Master Software is designed to direct drive the axis motor at 5 turns per inch of travel as well. The scale command can be setup anyway to offset this pitch.

Ex: SCALE X2.0 Y2.0 Z1.0

CALIBRATION FACTOR (X, Y, and Z)

This command, found in the SETUP window, can be used in the situation where there is an absolute repeatable limitation on the mechanics of the X, Y, or Z-axis travel. You can use this formula to +/- compensate for the error. Let's say you get .004" repeatable mechanical error using a one-inch dial indicator and you would like to obtain a tighter tolerance. Based on this movement, let's say you drive the Z-axis down 1" using the Relative Command feature (Feed it slow for testing on an indicator.) According to your dial indicator, it reads an actual 0.996". Now using the Relative Command, run +Z back up one inch. The dial indicator shows the needle at 0.00 every time you do this test. Do this test a few times. With successful repeatability, you will get an accurate Z movement. However, in the case that you need a tighter tolerance on the Z, you will need to calibrate it. Based on the above test, we have determined that the Z is off 0.004" from reaching an actual -1 inch of travel. So what is the factor? Take 1" divide by the actual movement 0.996, which equals 1.004; so enter 1.004 in the Z CALIBRATION FACTOR BOX and press Save. Be sure to write this figure down separately in the event you do a Delete Config command or get a Master Software update.

Factor Formula: (Relative Movement on Master Using -Z like above) / (Actual Movement as Indicated Mechanically) = Offset (round to nearest ten thousandth). Place this offset value in the Factor box, Save, and test again to be sure.

Caution: 0.005" is the max that can be calibrated and any figure entered above 1.005" or below 0.995" will "default" to this max and min position. If you are working in mm, 0.127 is the max that can be calibrated and any figure entered above 25.527 mm or below 25.273 mm will "default" to this max and min position.

DRAW DELAY

You can adjust the speed of the tool path being drawn out on the DRAW screen. Go to the SETUP window. There you will find the Draw Delay command that allows you to enter a 0 (no delay) to 99 (slow delay.)

SEC 8.

Saving Your Zero & Home Position; Limit Sensor Switches

The CNC MASTERS Mill is equipped with zero limit switches on the X, Y, and Z-axis. The machine will use these switches as reference points for saving or going to a saved Set Zero position on a program, and will stop when driving an axis to an extreme at either end. The zero position of all axes is the starting point of the program, which is designated by the user. Saving a set zero position will enable the user to quickly return to the zero position on any program without having to manually jog each axis and set it again. In order for these limit switches to work properly, the machine's spindle motor and control unit must share the same ground/earth line at the electrical panel service box.

LIMIT SWITCHES

At each extreme point of each axis there is a limit switch. So if you jog an axis too far to one end, it will make contact with the limit switch and instantly disable all three axes. **Hand-crank the axis motor away from whichever axis is making contact with its switch. Then press Reset Control to resume.** If you are running a program, and the axis goes to the extreme, it will also disable all three motors. You can do the same to re-enable the motors, however, you will lose your axes positioning. It will be necessary to re-establish your zero position on all three axes to avoid the axes from crashing at either end again.

CAUTION 1: Avoid metal chip build-up between the limit switches and axes. Plenty of build up can trigger the limit switch and shut down your program. **CAUTION 2:** Follow the instructions in the Limit Switch Error Box. If they are not followed correctly, the error box may continue to open repeatedly. Should this happen, click Port-On after closing the error box to correct this error, and then continue with a Reset Control.

Servicing Switches: Should the program ever stop, or if the motors are continuously disabled even after hand cranking an axis away from the limit switch, and there is no indication of the limit making contact on any of the three axes, then it is possible that a limit switch needs adjustment or repairs. To figure out which axis has the faulty switch, begin by powering off the computer and the controller. Unplug the controller and open it. Unplug only two axis motor cables and leave one plugged in. Next, close the controller, plug it in, power up the controller, and re-boot the computer. Open the Master Software and jog the axis that is plugged in. If it works, then we know it is not this axis that has the switch problem. Now, shut down the Master Software. Power off the controller and re-open it. Plug in the next axis motor connector. Power it on and open the Master Software. Re-test as before. If nothing drives after pressing Reset Control, then we know that this axis is the one with the driving problem. Repeat the same process for the third axis if the second axis is working properly.

Once you have determined which axis is causing the problem, shut down the Master Software. Then, power off controller and go to the limit switch on that axis. Unplug the switch at its connection using needle-nose pliers. Re-test like above to determine which switch is faulty. Internally, the switch may be touching ground, so the controller believes the switch is activated. The switch may need to be adjusted, repaired, or replaced so that it is not making contact to ground at the initial power up of the controller. Lastly, make sure the controller is also isolated from the machine and not making any contact to it through a metal connection. Contact CNC MASTERS for support if necessary.

SEC 9.

Addendum: Touch Screen Special Features

When jogging an axis using a touch screen computer, press the jog button with your finger and then roll finger slightly to activate the jog. Release your finger off the button when the axis reaches its desired location. All other buttons is a tap and release.

