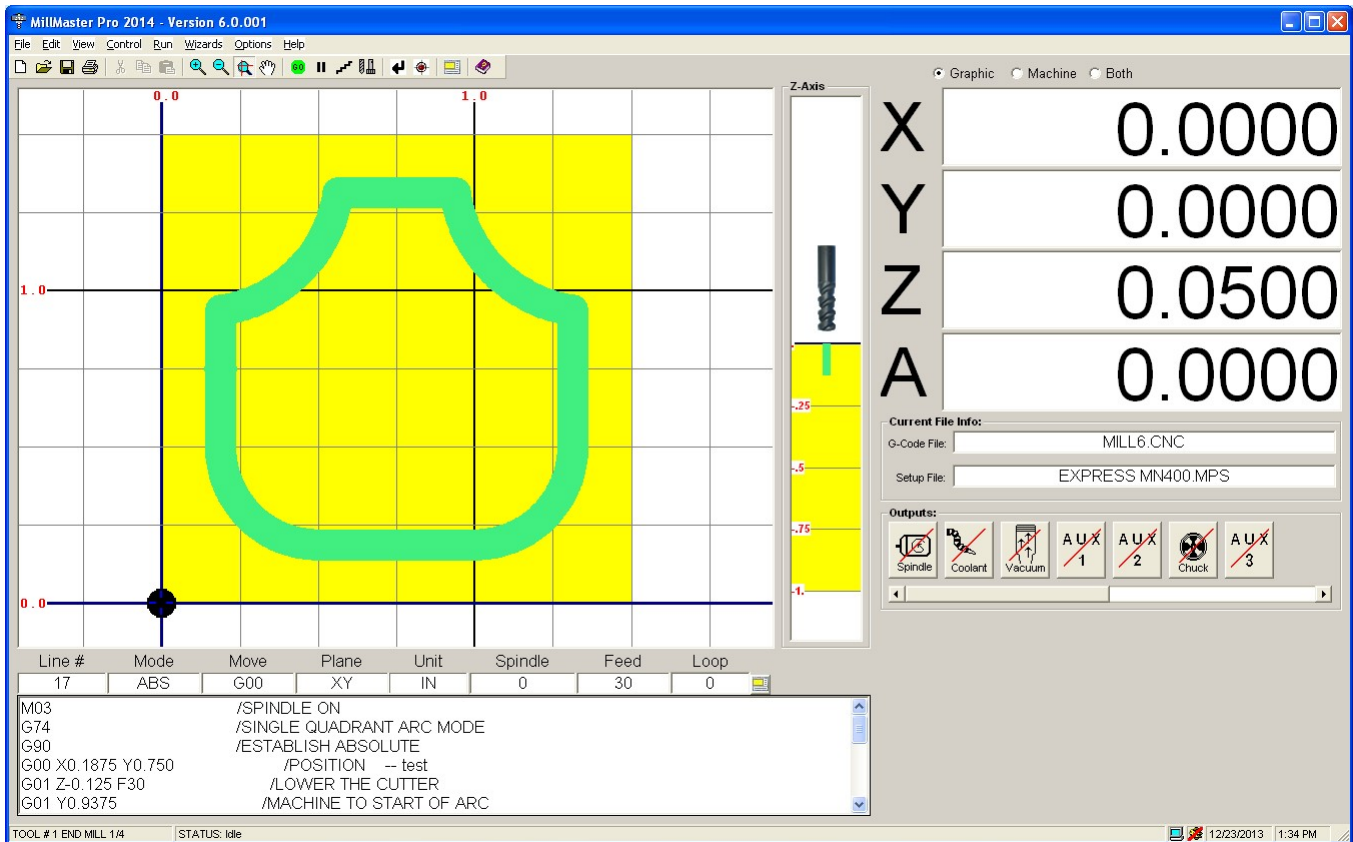


MillMaster Pro™ 2014

Milling Machine Control and CNC Program Verification
For Microsoft Windows™ Based PC's
Version 6.0.xxx



Contents Quick Jump

[Minimum System Requirements](#)

[Installing MillMaster Pro](#)

[Starting MillMaster Pro](#)

[Getting Started with MillMaster Pro](#)

[Configuration and Setup](#)

[Menu Reference](#)

[Toolbar and Icon related commands](#)

[Function Detail Reference](#)

[Function Quick Reference](#)

[Tutorial](#)

[Using the Math Formula Evaluation Feature](#)

[Customizing MillMaster Pro](#)

[MN400 Controller Info.](#)

[Appendix A - Keyboard Scan Codes](#)

[Appendix B - MetaCommand Reference](#)

[Appendix C - Wiring Diagram for External Cutting Speed Control](#)

[Appendix D - Using the Configuration File: HPMILL.MPS](#)

[Appendix E - Using Machine Parameters as Read-Only Variables](#)

Minimum System Requirements



- A Microsoft Windows PC
- Windows 98SE with Internet Explorer 5.5 or higher installed if using the OptiStep or Quickphase controllers, as well as an open ISA slot
- Antivirus software must be disabled if running with OptiStep
- Any version of the Windows OS if running with MN400 controller and one free serial or USB port
- 30 megs of free hard drive space
- Minimum 128 megs RAM with windows 98SE and 512 with XP and 1 Gig with Vista
- Minimum 300MHz Processor with windows 98SE, 500MHz with XP and 1 GHz with Vista

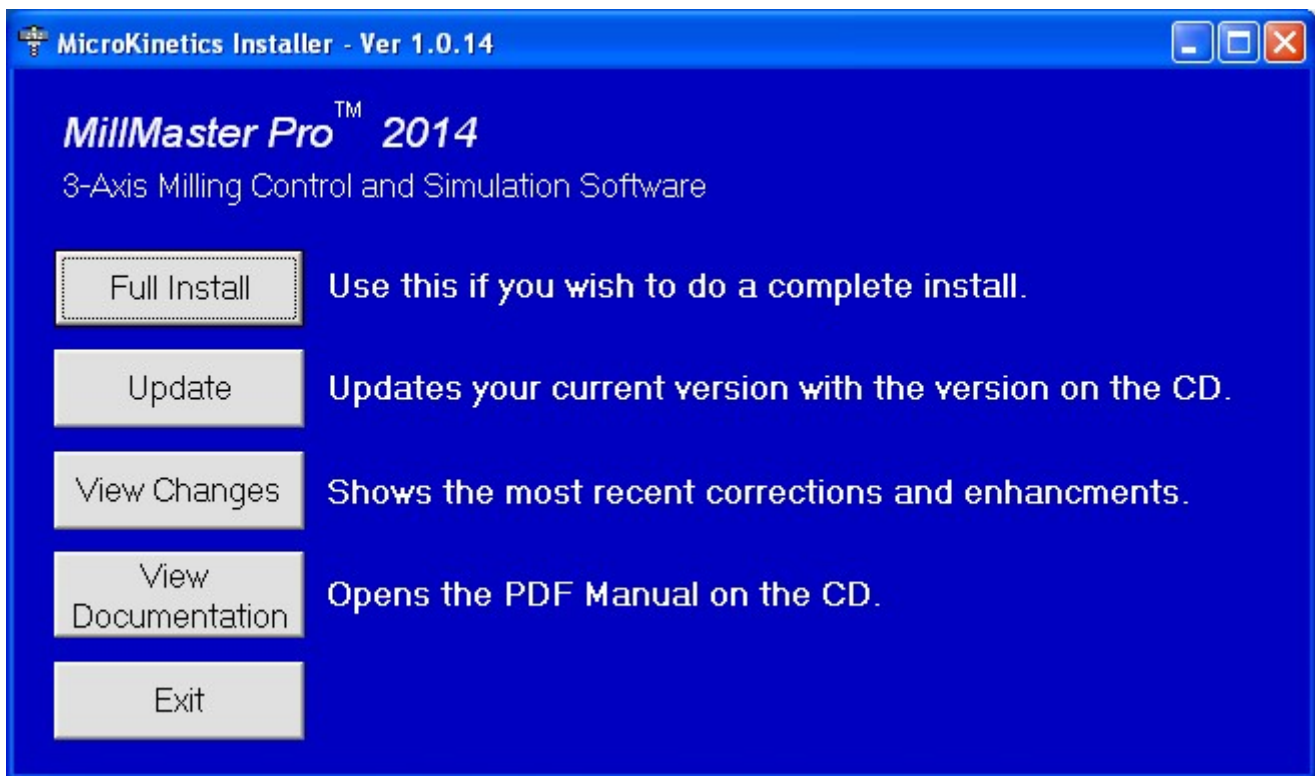
Note that **OptiStep** or **QuickPhase** control cards require a computer with the older ISA slot and **Windows 98SE**. For support of newer computers using USB, please choose the **MN400** controller or an intelligent **Driverack** system. If you would like to upgrade you legacy systems, **MicroKinetics** offers an upgrade path for your older system. Inquire at www.microkinetics.com

Installing MillMaster Pro 2014

**** Exit ALL programs and disable ALL Anti-Virus before installing ****

From the Windows Desktop:

- 1) For **Windows XP™**, **Vista™** and future **Windows** releases, log in as an **Administrator**. If you are logged in as a Standard User, some files may not copy correctly.
- 2) Insert the **MillMaster Pro 2014** installation CD. Depending on your PC's *AutoPlay* settings, the following screen may appear.



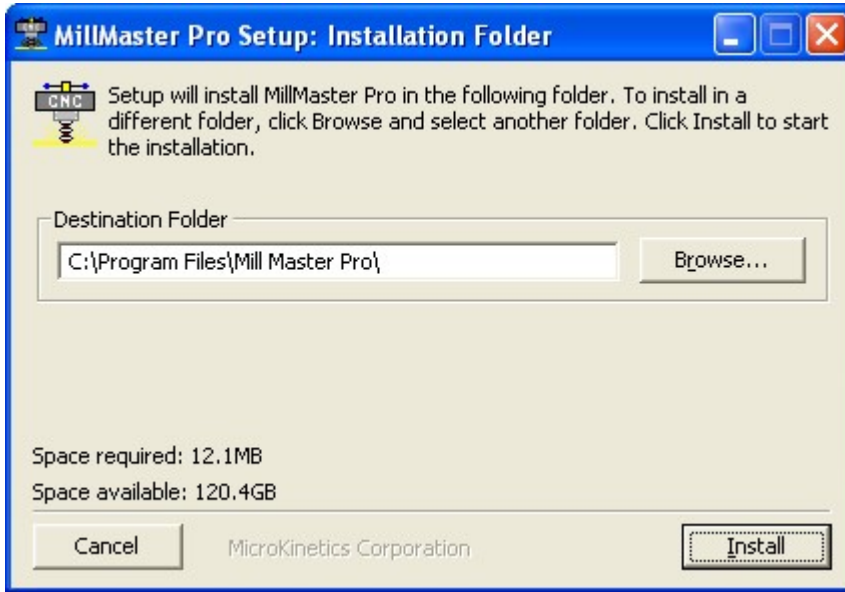
If you see an **AutoPlay** screen with a list of choices, choose "**Run mk_install.exe**" to see this screen.

If nothing happens when you insert the installation CD, open *Windows Explorer* manually and double-click **MK_Install.exe** to see the this screen. Alternatively, you can double-click **Mmpfw_Install.exe** and continue to **Step 4**.

Note: If you are installing the latest update from the **MicroKinetics** website, navigate to <http://www.microkinetics.com/tmpfw/instruct.htm> and follow the instructions.

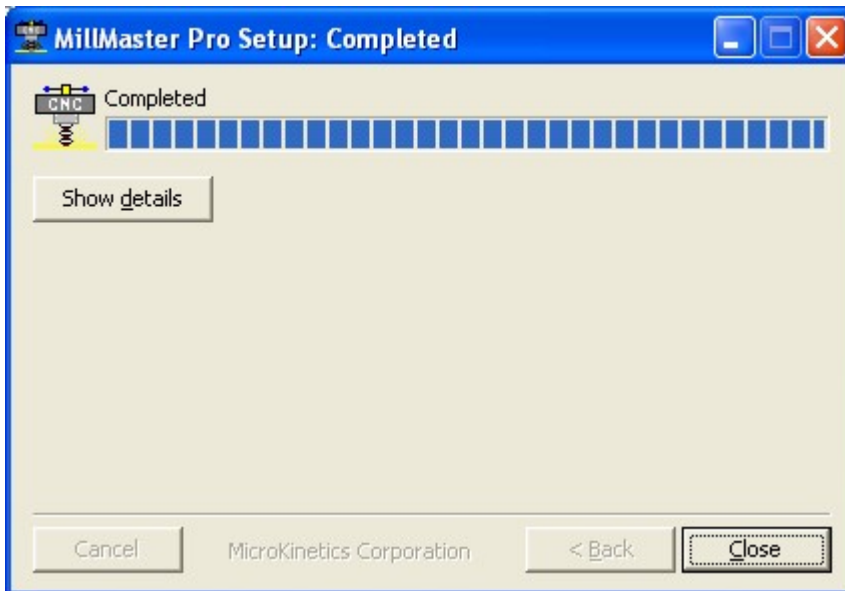
- 3) You may view documentation or install the software. If you choose **Full Install** or **Update**, continue on to **Step 4**. Otherwise press the **Exit** button to close the install program.

4) The following screen (or a similar screen if **Update** was selected) will appear:



You may accept the default Destination Folder, or select a different one. Click the **Install** button. This starts the Auto-Install process.

5) After the installation process is completed, the following screen will appear:



MillMaster Pro is now installed on your hard drive.

**** NOTE:** If you experience any difficulties or errors when installing, write down detailed information regarding the problem, including any cryptic error information, before calling Tech Support.**

** This will enable us to solve your problem as efficiently as possible.**

Starting MillMaster Pro 2014

IMPORTANT: Due to precise motor control timing you must disable/close **ALL** Instant Messaging and Virus protection software before running **MillMaster Pro**.

After Installation is complete, you must re-start your computer. Then you can run the program by selecting '**START**' ; '**PROGRAMS**' and then clicking on the **MillMaster Pro** icon in the program list.

TIP: If you right click the icon in the program list, a menu will appear with an option to "**Send To -->**". Put your mouse pointer over this item and another menu will pop out with an option that says "**Desktop as Shortcut**". Put your mouse over this option and left click.

Windows has now placed a **MillMaster Pro shortcut** Icon onto your desktop for easy access.

When you first start **MillMaster Pro** you will see the following screen:



You will notice the evaluation message displayed on this screen until you 'Unlock' the software. This can be done by using the File Menu Bar at the top of the **MillMaster Pro** Main Screen; clicking the word **HELP**, this will drop down a small sub-menu; click the option that says 'Unlock Software'

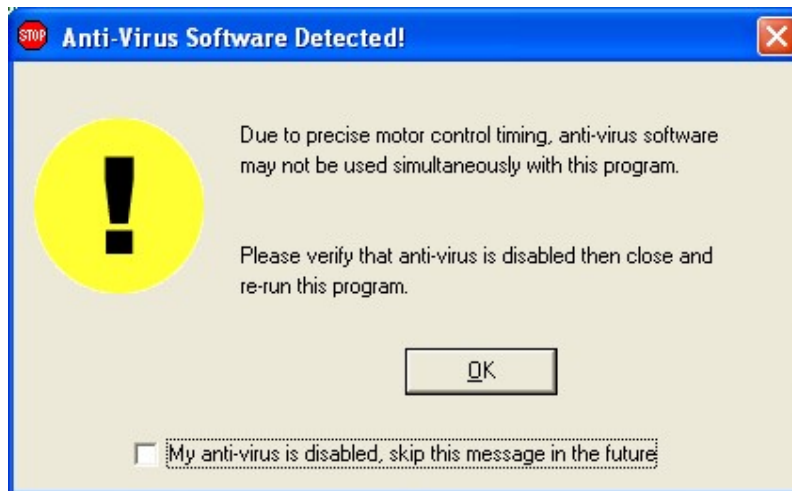
A small window will appear and contain a **System ID #** and have a blank space to enter an **Unlock Code**. To obtain an Unlock Code, call our sales department @ (770) 422-7845 or via email to sales@microkinetics.com.

Once you receive your Unlock Code, you enter it into the space provided and click the button labeled 'Unlock Software'.

Now when you start **MillMaster Pro 2014** the initial screen will look like this:



After the initial splash screen shown above, if you have Norton Antivirus or McAfee Virus Shield installed on your system, you will see the following screen:



This screen will appear each time you start **MillMaster Pro**, unless you check the option at the bottom of the screen that says 'My Anti-Virus is disabled, skip this message in the future'. This screen is just a reminder, so if you do check the option to skip the message in the future and do not disable your Anti-Virus software, you will not be able to perform machining functions.

MillMaster Pro only detects Norton Anti-Virus or McAfee Virus Shield. If you are running another brand of Anti-Virus, this message will not be displayed. You will need to disable ANY virus protection, regardless of manufacturer before machining.

Also note that for Windows versions later than Windows 98, **MillMaster Pro** will not check for anti-virus software, and thus this screen will not be displayed.

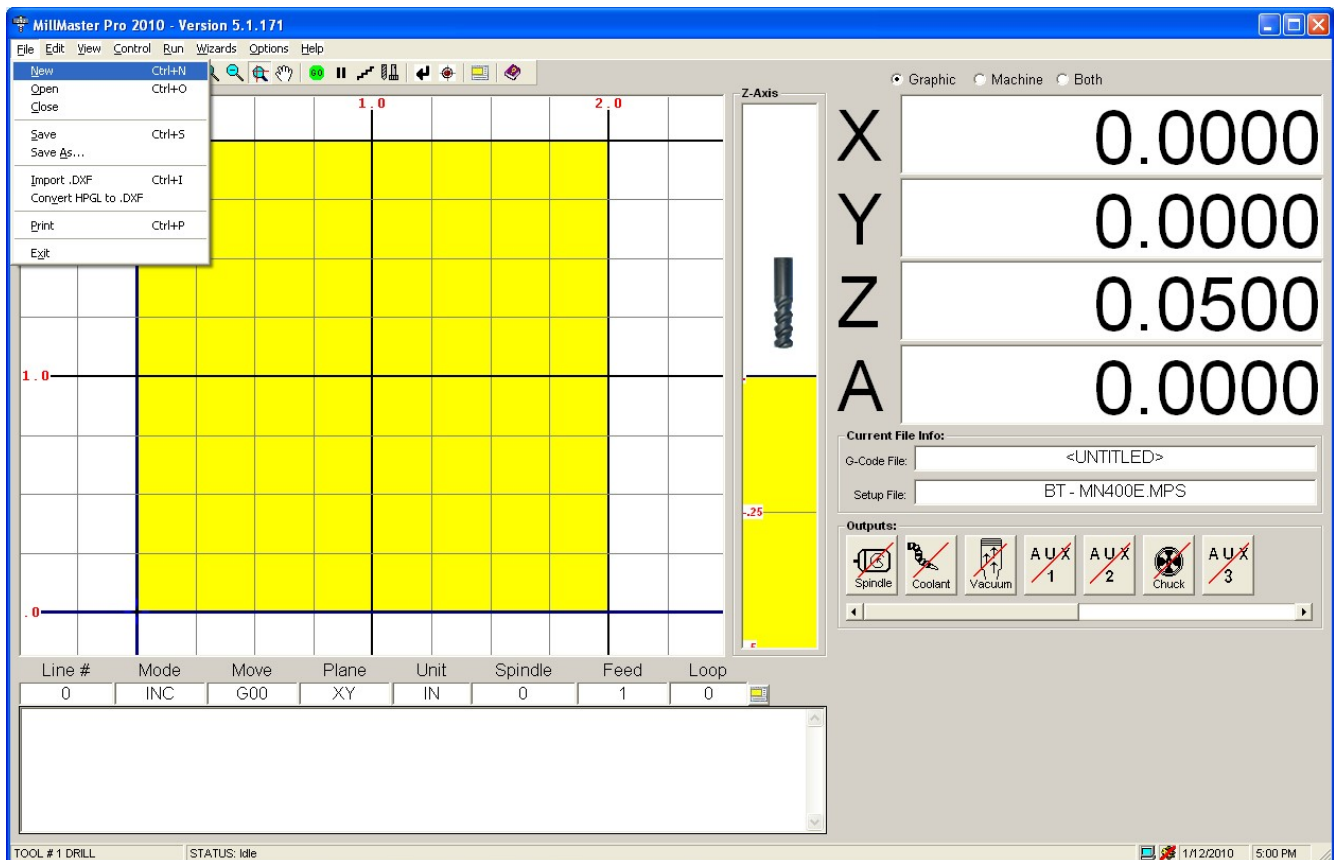
Norton Antivirus and McAfee Virus Shield are trademarks of their respective companies.

Getting Started with MillMaster Pro 2014

Thank You for purchasing **MillMaster Pro 2014** G-code interpreting Software. This is an installation and command reference manual for **MillMaster Pro** that will help you with all aspects of the program. Observe the top line of the MillMaster Window. Each word is a heading for a pull-down menu.

The pull-down menu will appear when the heading is selected by the mouse or keyboard. To access a pull-down menu, use the mouse to position the arrow on top of a pull-down menu heading and click the left mouse button. Alternatively, you can use the keyboard to access a pull-down menu by pressing <ALT> and the first letter of a heading.

Once the menu is in view, you can select any of its options by positioning the arrow on the option and clicking the left mouse button.



Some of the selections have **Hot Keys** associated with them. Hot keys allow quick access to a command through the keyboard and they are indicated to the right of the selection. For example, the hot key used to open a part program is **<CTRL>-O** which means holding down the **Ctrl key** and pressing **O** simultaneously.

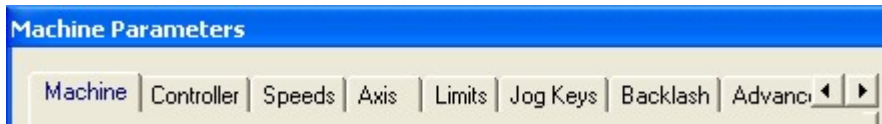
The underlined letters are called **QuickSelect** keys. These allow access to a command only while the pull-down menu is displayed. Some of the selections in the menu have corresponding icons on the toolbar below the menu heading. These allow the user to rapidly execute a command without using the pull-down menus. Clicking these icons executes the same command as using the pull-down menus.

In the **Machine Parameters Window**, the menu is setup with **Tabs**, just like a filing cabinet. When the mouse moves over these tabs, they become highlighted. This is called **Mouse-over Hot Keys**. Click on a tab to show its relevant information.

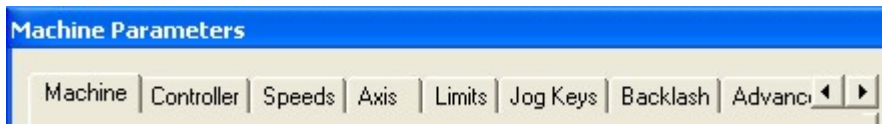
The following graphics show the Machine Parameter Tabs. The graphic on top demonstrates what the tab looks like when the **Mouse-over Hot Key** is **active**.

The graphic on the bottom demonstrates what the tab looks like when the **Mouse-over Hot Key** is **inactive**.

Active:



Inactive:



Some menus can be accessed rapidly with a **Mouse QuickSelect** click. These allow access to the **Open File Dialog Window** and the **Machine Parameters menu**, by clicking in the white space of the G-code File or the Setup File boxes in the Current File Info frame. **(Shown below)**



The **Zoom-In**, **Zoom-Out**, and **Pan** functions are also accessed by left clicking and dragging the mouse over an area in the cutting window.

If the mouse looks like a crosshair when over the XY cutting window, you are in **Zoom** mode. You may drag the mouse over a an area, and **MillMaster Pro** will re-zoom the XY cutting window to most closely match the area specified.

If the mouse looks like a hand when over the XY cutting window, you are in **Pan** mode. Clicking and dragging the mouse will pan the XY cutting window in the direction and distance specified.

Configuration and Setup

This section will instruct you on how to configure **MillMaster Pro** to work with your milling machine. This section will only cover the procedures that are **necessary** to successfully machine a part. Enter the values as indicated below.

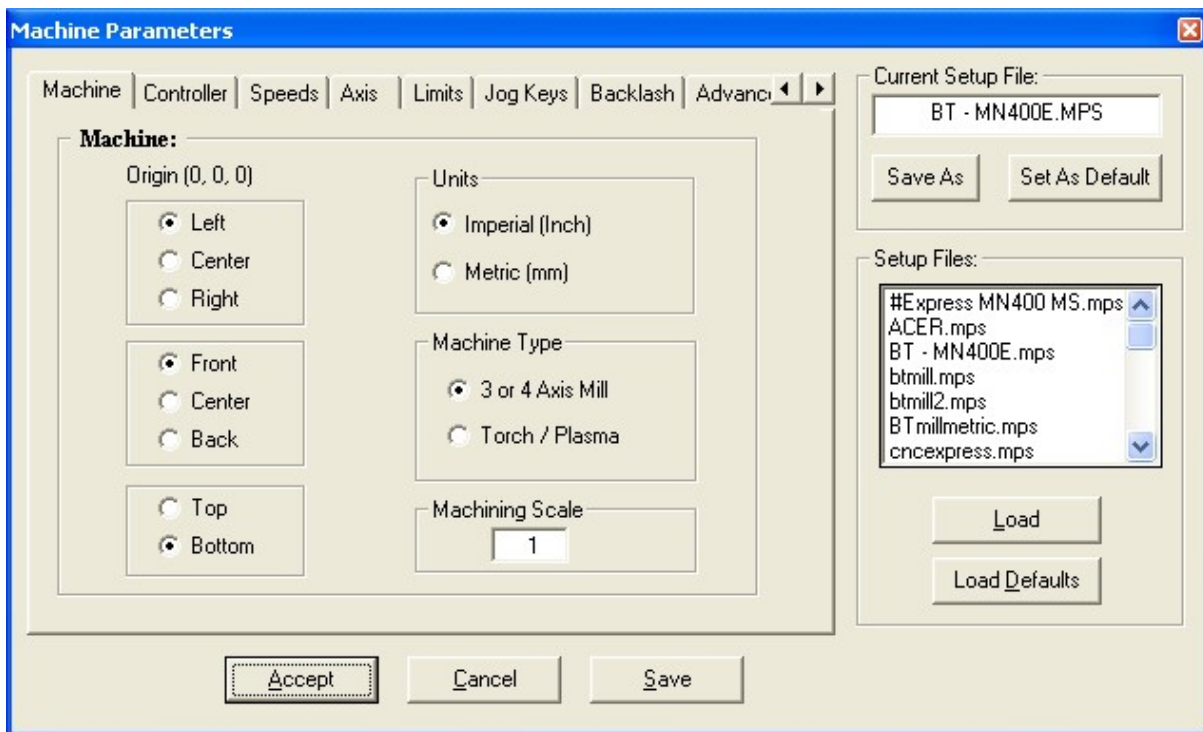
4.1 Entering Part Program Information

It is necessary to enter information about your part program into **MillMaster Pro** so that it can accurately simulate part production. Enter the values shown if you want to run the tutorial in Section 5.

1. Select the type of **Units (inch)**, the **Machine Type**, and the **Location of the Origin (left, front, top)** under **Options -> Machine Parameters -> Machine tab**.
2. Enter the **Machining Scale**. Entering **1** means "**TO SCALE**" (See **Option Menu Section for Further Information**). Enter **1**.
3. Choose **[Accept]**
4. Under **Options -> Material Size**, enter the **Material Length** in the specified **Units**. Enter **3.0**.
5. Enter the **Stock Width** in the specified **Units**. Enter **3.0**.
6. Enter the stock height in the specified units. Enter **0.5**.
7. Choose **[Accept]**.

4.2 Setting the Machine Control Speed

The following graphic shows the **Machine Parameters Window** in **MillMaster Pro**. This is where **ALL** machine and program settings should be made and/or adjusted.



The **MillMaster Pro** software has to know the speeds your machine can move at **reliably**. Choose the **Options-Machine Parameters-Speeds** option with the mouse or type **<ALT>-O M** and select the **Speeds Tab**.

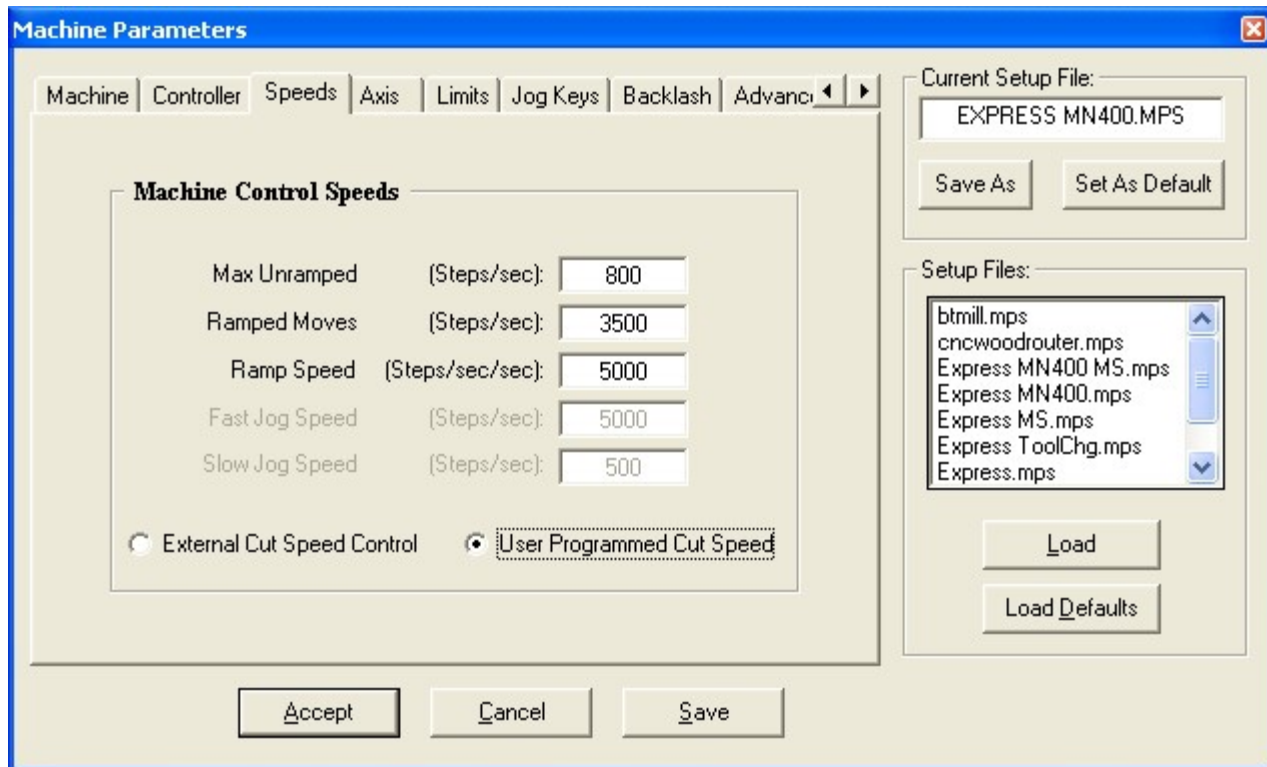
1. Set the **Max Unramped** to the **Maximum Instantaneous Speed** at which the machine can be moved **without** acceleration. This speed is in steps per sec and can easily be converted from inches per minute by using the following formula:

$$\text{Steps/Sec} = \frac{\text{Screw Pitch} \times \text{\#Steps/Rev}}{60}$$

The default setting is **800 Steps/Sec** for a half-stepping driver with a 20 TPI leadscrew and is reasonable for **Soft Metal** and **Plastic**.

2. Set the **Ramped Moves** to the maximum speed at which your machine can move **reliably**. The default is set to between 2500 and 3500 Steps/Sec depending on the machine you are running. You may be able to set this at a much higher rate if you are using ballscrews or other low friction mechanical drives.
3. The **Ramp Speed** is an acceleration parameter and is usually set for 5,000 Steps/sec/sec. You may adjust this up or down depending on your machine but it is practically set between 5,000 and 20,000.
4. Set the **Fast Jog Speed** to the same speed as the **Max Unramped** speed.
5. Set the **Slow Jog Speed** to a speed that can be used for fine positioning. A good value to start with is 200. Adjust the value as needed.

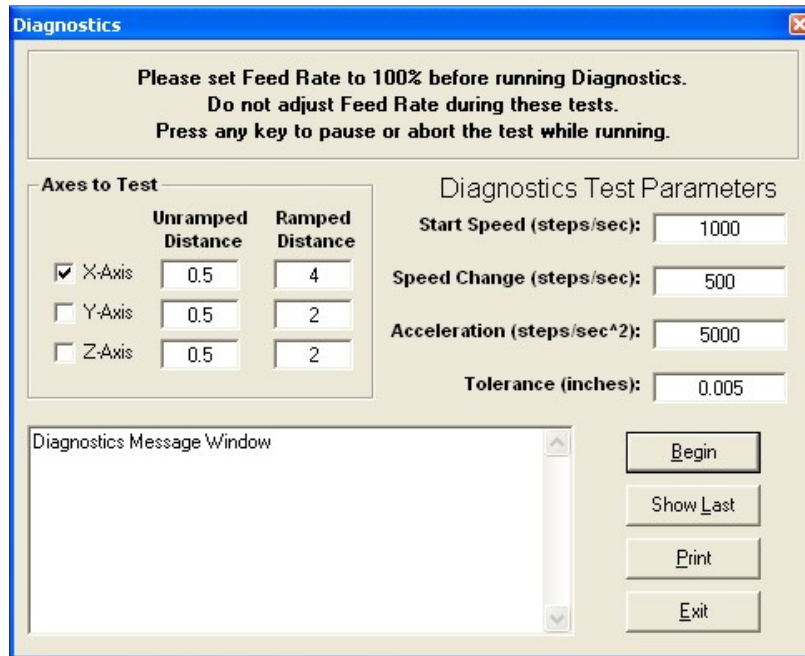
The **Fast Jog Speed** and **Slow Jog Speed** are only enabled for the **OptiStep Plus** and **QuickPhase** controllers, and disabled for the **MN400** controller.



** The **Control-Diagnostics** option can be used to automatically find the **maximum unramped** and **ramped speeds** of your machine. Press **<ALT>C D** on the keyboard (or **Control -> Diagnostics** from the menu), select the axes to be tested, and enter the test parameters. Choose **[Accept]** to start the test.

You will want to have a minimum value of 4" in the distance field to allow the machine to reach higher ramping speeds. Make sure that your feedrate override knob is set to 100% (where applicable). Keep in mind that you may have problems reaching the 130% that the feedrate knob provides for if you accept the values after testing, so a slight adjustment in the **Max Unramped** speed may be necessary.

The following graphic shows the **Diagnostics Window**.



4.3 Axis Control Parameters

Choose Options-Machine Parameters-Axis Control with the mouse or type <ALT>-O M. and then select Axis Control. If you purchased a machine from us or are retrofitting using one of our kits, then we have default setup files for you to choose like **CNCExpress.mps** or **BTMill.mps** (for benchtop and desktop mills) which can be selected. Make sure you load the correct file and then select **Set as Default**. By pressing **Save** after this you will keep this file as your default every time you load the program.

1. Set the **Lead Screw Pitch** for both axes to the proper value for your machine. The default is 20 threads per inch (TPI) for a Sherline sized machine and 5 for the larger ballscrews like the **CNC Express** and Bridgeport retrofits come with.. Some typical values for ballscrews and ACME threads are 5 and 10 TPI. Enter this value according to your screw.
2. Set the **Logical Steps Per Revolution** for both axes to the proper value for your machine. All MicroKinetics motors have 200 full steps per revolution of the motor. To calculate steps/rev, multiply 200 times the inverse of your step rate.

Example: Using DM8010 in 1/10 step mode.

$$2000 \text{ Steps/Rev} = 200 \times \frac{1}{10}$$

3. Set the **Polarity** of each axis. A **NORMAL Polarity** is defined as: if the axis turns counterclockwise as viewed from the motor end, a positive cutting move will result. Set the axis to **NORMAL** if the above rule is true and **REVERSE** if it is false. **Note:** If **REVERSE** is selected for any axis, then the limit switch connections must also be reversed (i.e. **connect a positive limit switch to the Negative Input**).

4.4 Setting the Limit Sensors

Choose **Options-Machine Parameters-Limit Sensors** with the mouse or type **<ALT>-O M**.

****NOTE: This is to be done only if the machine you are using has limit sensors.**

Choose the **Limit Sensor** for each axis that is to be considered home. These will be the three sensors that the machine goes to in homing moves. The X and Y axes will move to the negative direction limit sensor (X will move the table to the right, and Y will move the table away from you) during a home, reposition, or reprogram move, while the Z axis will move to its positive limit (up). If your machine does not move in the proper direction stop it immediately and check that you have the correct polarity setting for your axes.

4.4.1 Locating the Limit Sensors

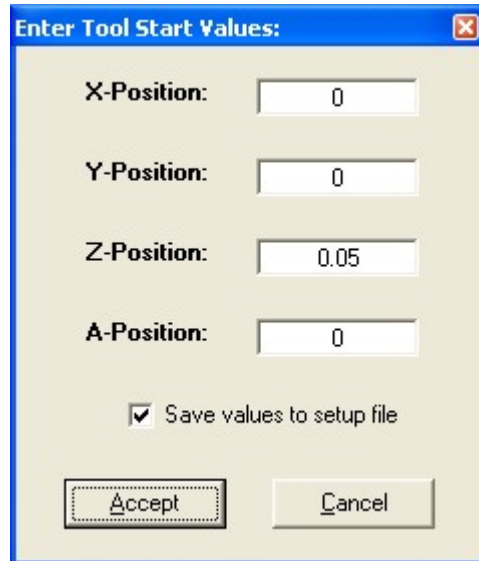
1. Locate where the **Absolute Tool Starting Position** is on the machine.
2. Jog the tool so that it is centered over the **X & Y** starting position and is raised to **Z Starting Position Height**. Exit **Jog Mode**.
3. Choose **<ALT>-C I (Control-Initialize as Start Position)**.
4. Choose **<ALT>-C P (Control-Reprogram Limit Sensors)**. The machine will automatically move to all the limit sensors and return to the **Tool Start Position**. The limit sensor locations have been automatically entered into the **Options-Machine Parameters-Limit Sensors** dialog box.
5. **Save parameters as default**. This will store all your specific parameters on disk so that they automatically load at start-up.

4.5 Setting the Tool Start Position

Choose Tools - **Start Position**. Enter the absolute coordinates of where the tool will start from at the beginning of the machining process. This value should match the **G92** code in the CNC program file that is to be run. Typically, the tool is centered over a corner of the stock and the Z-axis is above the surface 0.050".

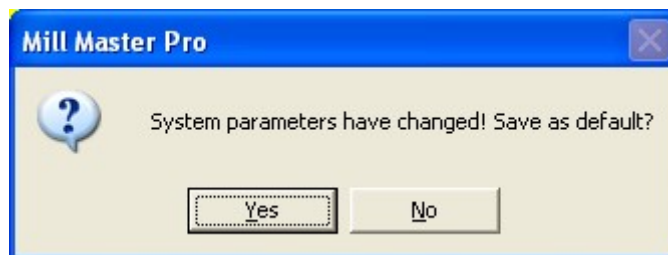
To work with the tutorial, set the following values:

X axis = 0"
Y axis = 0"
Z axis = 0.050"
A axis = 0"



Choose **[ACCEPT]** when complete. These values may be changed to suit your specific needs.

If the box "**Save values to setup file**" is checked, the values will be saved to the setup file immediately upon clicking **Accept**. If unchecked, you will receive notice when exiting **MillMaster Pro** that you system parameters have changed. This is described below.



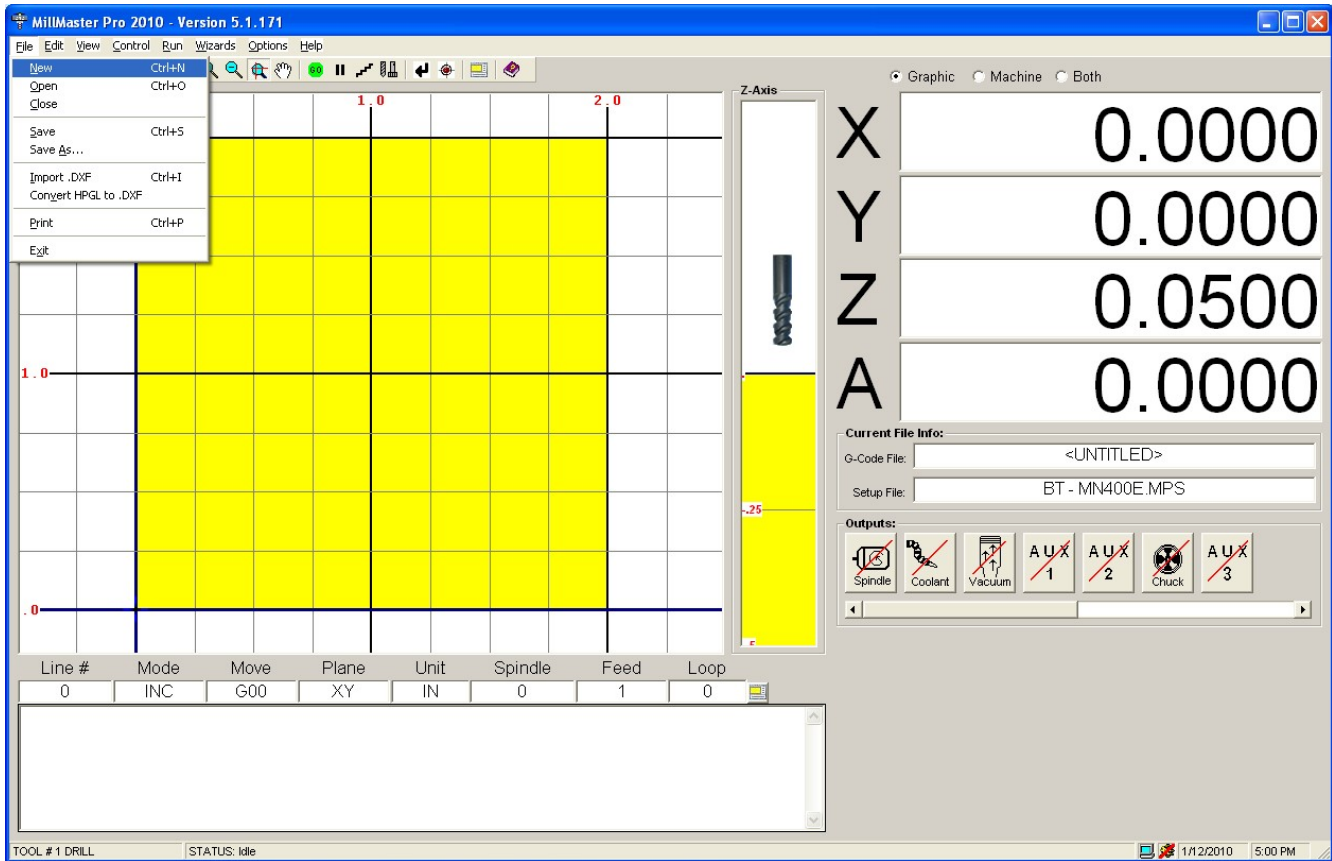
Save Parameters as Default?

When exiting **MillMaster Pro**, you may receive the above message. Click "**Yes**" to store all your specific parameters on disk so that they automatically load at start-up.

Congratulations, the software is now configured to your specific machine.

Menu Reference

The graphic below represents the Menu Bar with the "File" pull down menu accessed.



File- New

Clears the part program in memory and establishes the standard defaults for a new part program. The first icon in the Toolbar also corresponds to this menu command.

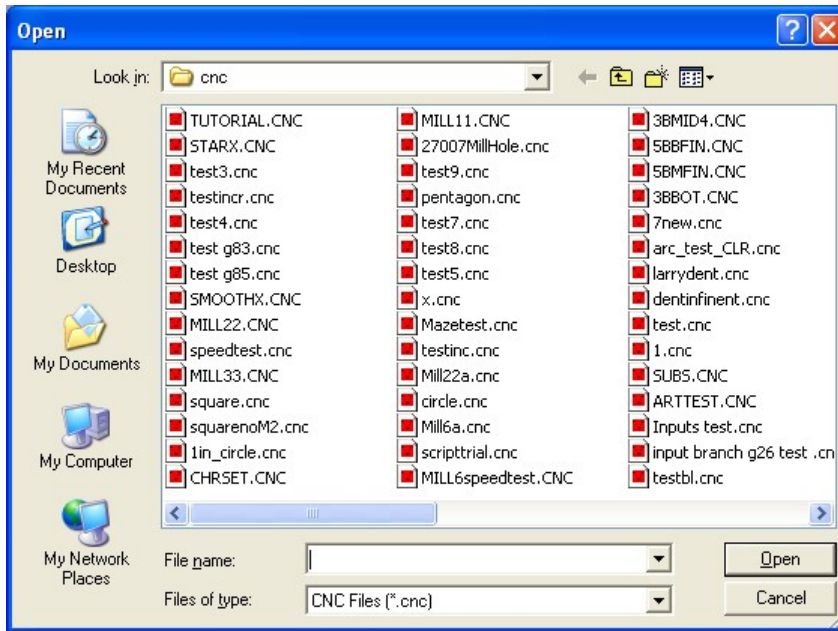
File- Open

Displays the Open File Dialog Window which displays all part programs in the current directory. The second icon in the Toolbar also corresponds to this menu command. The graphic below represents the Open File Dialog Window.

You can easily load your favorite part program by Dragging and Dropping the .CNC file onto the **MillMaster Pro** icon that is used to start the program. **MillMaster Pro** must not be running when this feature is used. After dragging and dropping the file onto the **MillMaster Pro** start icon, **MillMaster Pro** will startup normally and the file that was dropped onto the icon will be automatically loaded.

TIP:

Just click in the whitespace of the G-code file box in the **Current File Info:** frame for rapid access to the Open File Dialog Window



File- Close

Closes the current .CNC file in memory, but does not restore the counters or the cutting screen.

File- Save

Quickly saves the current part program to disk. The extension **CNC** is automatically assigned. The third icon in the Toolbar also corresponds to this menu command.

File- Save As...

Displays the Save File Dialog Window, which Saves the current part program to disk allowing the user to rename the file. The file extension **.CNC** is automatically assigned.

File- Import .DXF

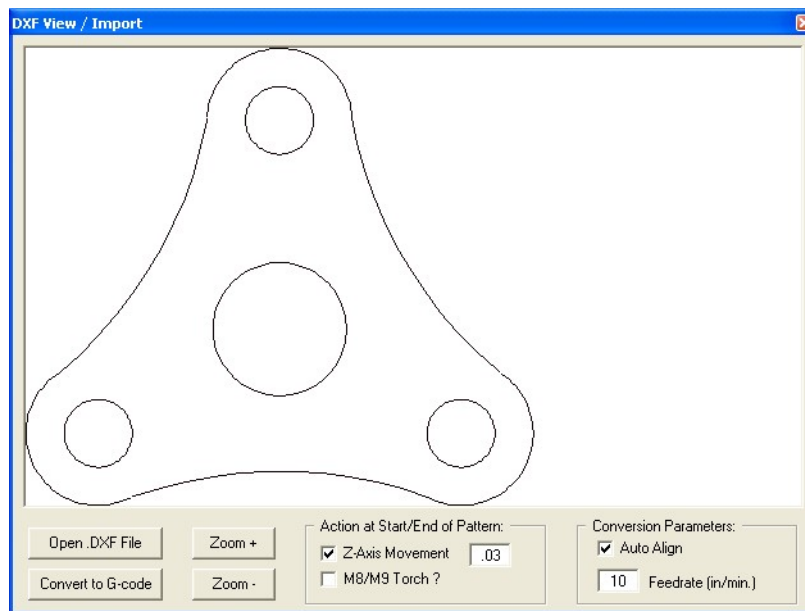
This menu command allows you to import **industry standard .DXF files up to version 15 (AutoCAD 2000 format)**. This is the latest **AutoDesk** release as of the date of release of this software..

.DXF files that are saved with Tables or that use 3D modeling will not show up in the DXF window, (shown below), and will not convert to G-code. The following entities are supported for conversion to G-code:

- Polyline
- LWPolyline
- Line
- Circle
- Arc

****NOTE:** AutoCAD does true circular interpolation whereas Corel Draw breaks circles and arcs into coordinates and therefore the motion generated is not as smooth and continuous. Contained within the **MillMaster Pro** program directory, there is a **DXF** folder, which contains some sample **.DXF** files with various designs. To get familiar with this feature, import one of these DXF files, convert it to G-code and run it in Graphics mode. You will see the exact shape that appeared in the DXF window, appear in the part material in the main Viewport.

When selected, an **Open File** dialog box will appear. Select the **.DXF** file of your choice and click 'Open'. A new window will appear with a large viewport. (shown below) This is the **DXF viewing conversion window**. At the bottom of the window you will notice two command buttons, and a group of conversion parameters. Once the file is opened, it will appear in the viewport. Input your conversion parameters into the areas provided and select "**convert to G-code.**" A **Save File** dialog box will appear next. Select a drive, directory and name to save your file under and click 'Save'. You have just created a **.CNC** file for use with **MillMaster Pro**.



****NOTE:** MillMaster will automatically name your G-code file the same as your **.DXF** file unless you specify otherwise.

TIP: Remember to test your program Graphically before machining to prevent potential damage to your machine.

Conversion Parameter details:

Z-axis depth:

This parameter specifies the **Z-axis** depth for the conversion utility. These are only initial values, and may be changed with the **CNC Editor**.

This parameter is only available when the machine is **setup** for a **three** or **four axis Mill**.

Feedrate:

This parameter specifies the feedrate for the **conversion utility**. This is just an **initial value**, and may be changed with the **CNC Editor**.

Auto-align:

This parameter specifies the auto alignment feature for cutting which brings the drawing into view. If the values contained within your **.DXF file** are some distance off the viewable screen, the conversion utility will add a **G92** specifying the X and Y reference point for all moves such that the lower left corner of the design is at the lower left corner of the part.

Auto-Adjust Material Size:

This parameter specifies whether **MillMaster** should automatically change the current material size needed to cut the current imported **.DXF file**. If selected, when you return to the **main screen**, the material size will be adjusted automatically.

Auto-Load Program:

This parameter specifies whether **MillMaster** should automatically load the program into memory. If selected, the program will be loaded when you return to the **main screen**,

File- Convert HPGL to DXF

ATTENTION: This converter works 100% with HPGL files, but complications arise when trying to convert HPGL/2 files, since HPGL/2 includes commands not recognized by HPGL. If you have an HPGL/2 file, try doing a "Save As" and select HPGL format before using the converter.

This function uses a 3rd party utility called HPGL2DXF.EXE, written by Quantum Diagnostics, Inc. The following 2 files are the converter package in whole, and are included in our distribution setup program free of charge:

HPGL2DXF.EXE - Converter program

HPGL2DXF.TXT - Text file containing information on the program and licensing (**READ THIS FILE**)

AutoCad features supported:

- Lines
- Arcs
- Circles
- Text (we suggest defining MONOSPACE as the default font for text in your drawing as start, then you can experiment with other text styles).

Converter USAGE:

- 1) Copy the PLT file you wish to convert into your **MillMaster Pro** directory.
- 2) Select 'Convert HPGL to DXF' from the file menu.
- 3) Browse your **MillMaster Pro** directory for your PLT file.
- 4) Select the file and click Open.
- 5) You should see a small black window appear for about 1-3 seconds and then disappear.
- 6) Now select Import DXF from the File menu and select your **MillMaster Pro** directory.
- 7) You should see the name of your original PLT file appear, but this time with the extension DXF
- 8) Select this file and click Open.
- 9) Follow the procedures for importing a DXF file into **MillMaster Pro**

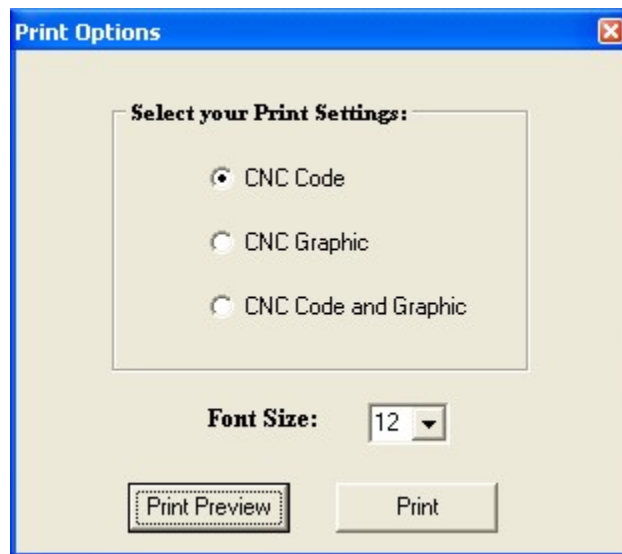
TIP: The conversion from HPGL to DXF is very generic to prevent unwanted alterations of your original .PLT file design in the newly created .DXF file, but the .DXF files created by the converter default to 110" by 110" in length and width. Once the part program is loaded use Options --> Material Size and specify the correct length and width for your specific part.

File- Print

Displays the Print Dialog Window, which allows the user to choose Print Options, and utilize the built-in Print Preview Function.

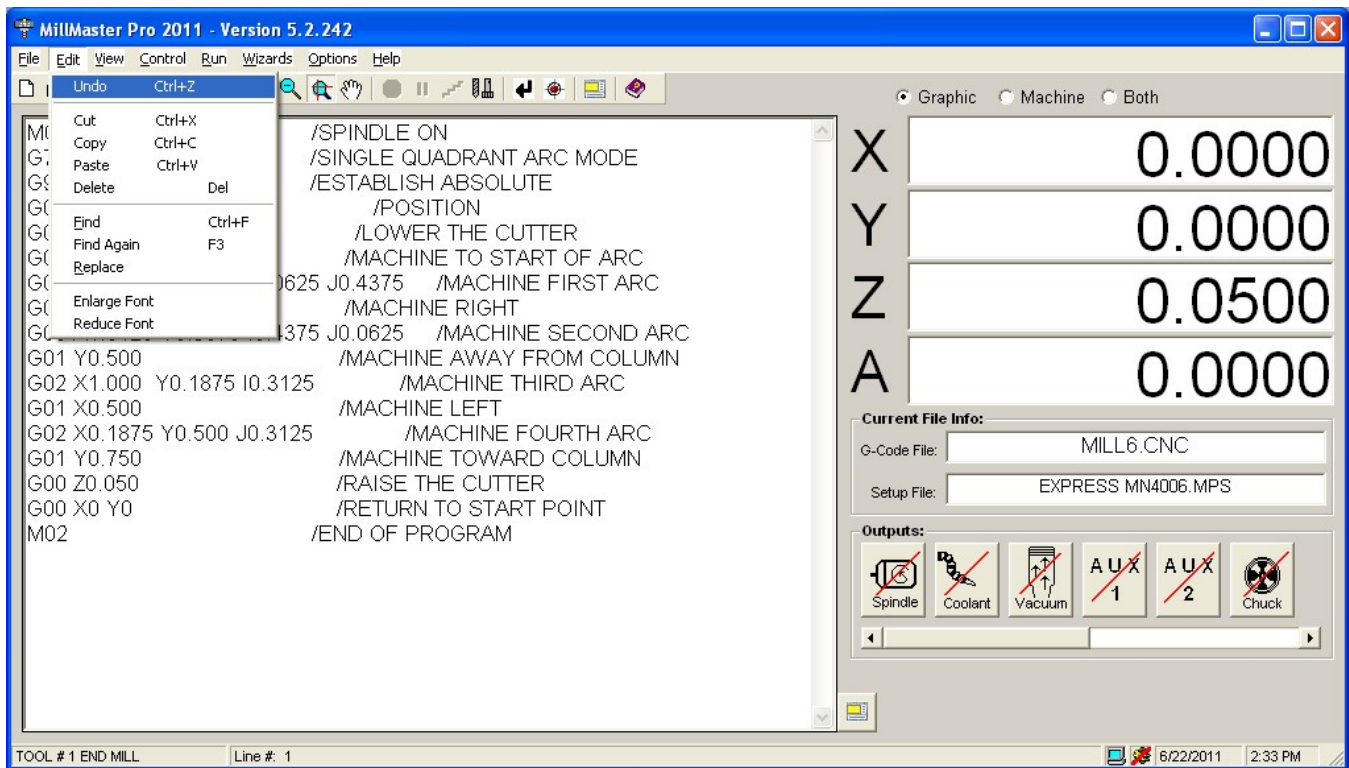
The fourth icon in the Toolbar also corresponds to this menu command.

The graphic below represents the Print Dialog box.



File- Exit

Exits **MillMaster Pro** and returns to the Windows desktop.



Selecting the **Edit Menu** produces the list of Editing selections. The commands on this menu are Standard Text Editing commands seen in today's popular text editors. Some of the selections have **Hot Keys** associated with them. **Hot Keys** allow access to a command quickly through the keyboard and they are indicated to the right of the selection.

The underlined letters are called **QuickSelect** keys. These allow access to a command only while the pull-down menu is displayed.

**** NOTE:** The Edit Menu is only enabled when in full-screen **CNC Edit Mode**. See the View Menu for more information on **CNC Edit Mode**.

All the Edit Menu commands are grayed out and disabled unless **MillMaster Pro** is in CNC Edit Mode (Full-Screen Editor). ******

Edit- Undo

Undoes the last edit (insert, delete, or change). Only the last edit can be undone.

Edit- Cut

Cuts the currently highlighted text out of the page and onto the clipboard. The fifth icon in the Toolbar also corresponds to this menu command.

Edit- Copy

Copies the currently highlighted text onto the clipboard. The sixth icon in the Toolbar also corresponds to this menu command.

Edit- Paste

Pastes the current data from the clipboard to the current cursor location. The seventh icon in the Toolbar also corresponds to this menu command.

Edit- Delete

Deletes the currently highlighted text permanently.

Edit- Find

Finds a specified sequence of characters in the CNC Edit window.

Edit- Find Again

Finds the next iteration of the specified sequence of characters you previously looked for with the FIND command.

Edit - Replace

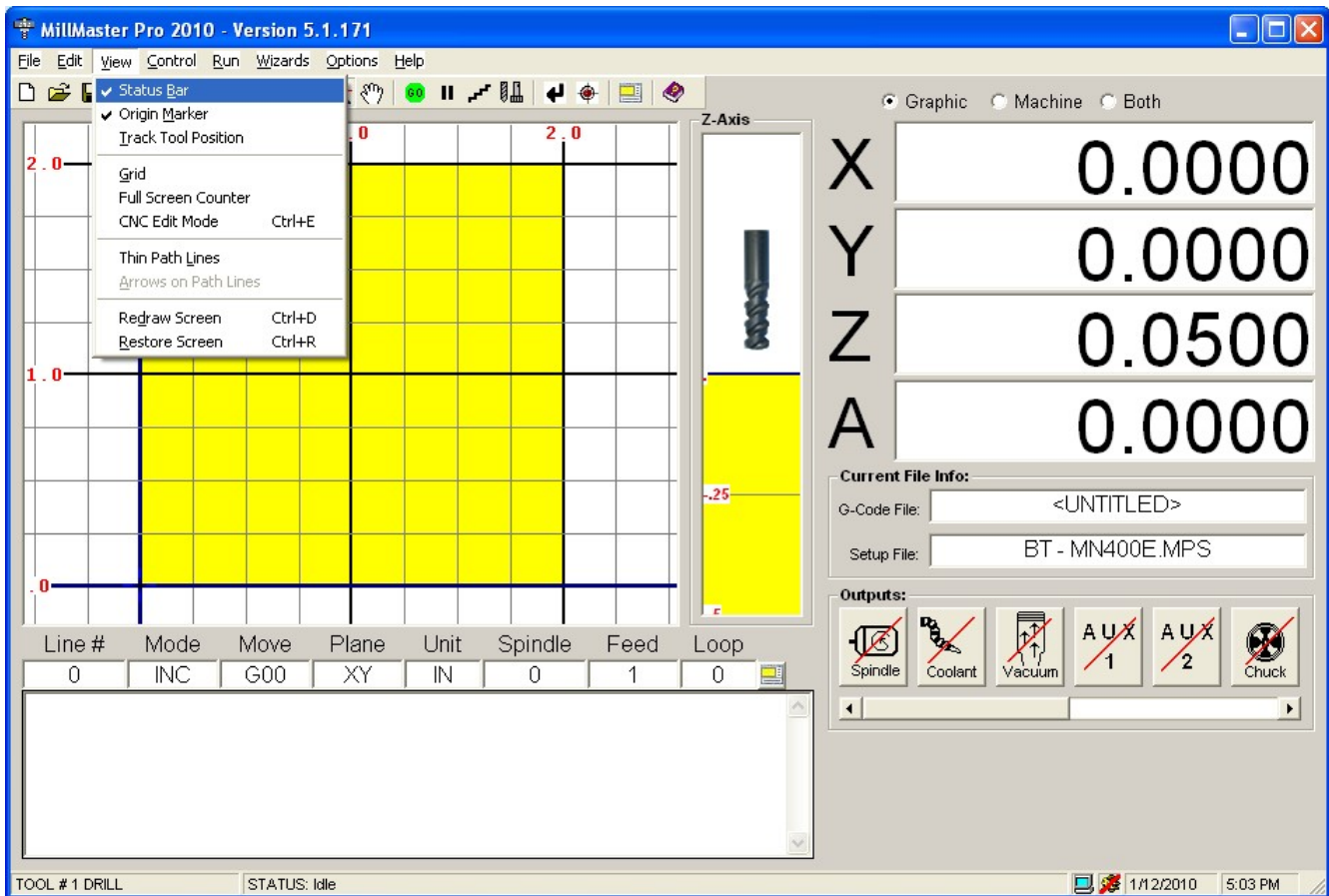
Finds a specified string of characters in the CNC Edit window and replaces that string with a different string. The user will have the option to replace only the first occurrence of the original string or all occurrences.

Edit- Enlarge Font

Enlarges the font on the CNC Edit window. Each time this option is selected the font size is increased by 1.5 pt, up to a maximum size of 16 pt.

Edit- Reduce Font

Reduces the font on the CNC Edit window. Each time this option is selected the font size is decreased by 1.5 pt, down to a minimum size of 8 pt.



Selecting the **View Menu** produces this list of visual selections. Some of the selections have **Hot Keys** associated with them. Hot keys allow access to a command quickly through the keyboard and they are indicated to the right of the selection.

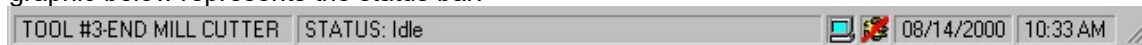
For example, the Hot key used to restore the screen is **<CTRL>-R**. This is done by holding down the **<CTRL>** key and pressing **R** simultaneously.

The underlined letters are called **QuickSelect** keys. These allow access to a command only while the pull-down menu is displayed.

View - Status Bar

Allows the user to either enable or disable displaying a status bar at the bottom of the window with current tool selection, current program status, current program mode, and current system date/time. When a check mark appears next to this function, it is enabled.

The graphic below represents the status bar.



View - Origin Marker

Allows the user to either enable or disable displaying the blue, crosshair Origin Marker over the tool in the XY viewport. When a check mark appears next to this function, it is enabled.

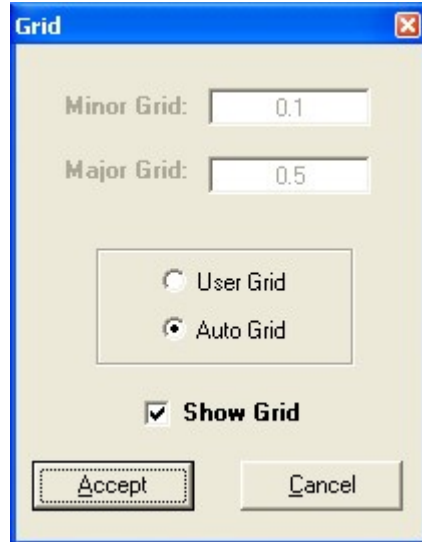
View - Track Tool Position

This option is only applicable in **Both** (Graphics and Machine) mode. If **Track Tool Position** is checked, then the tool position on the graphics screen moves in step with the actual tool. If not checked, then the graphical tool moves immediately to the end position of the current move before the machine move is executed.

View - Grid

Displays the **Grid Window** which allows the user to select the settings for the Minor and Major grid lines, use the **Auto-Grid** function, or turn **On/Off** the Grid. When a check mark appears next to this function, it is enabled.

The graphic below represents the **Grid menu**.



View - Full Screen Counter

Displays the **XYZA** Counters in **Full Screen Format**, while temporarily hiding all other controls.

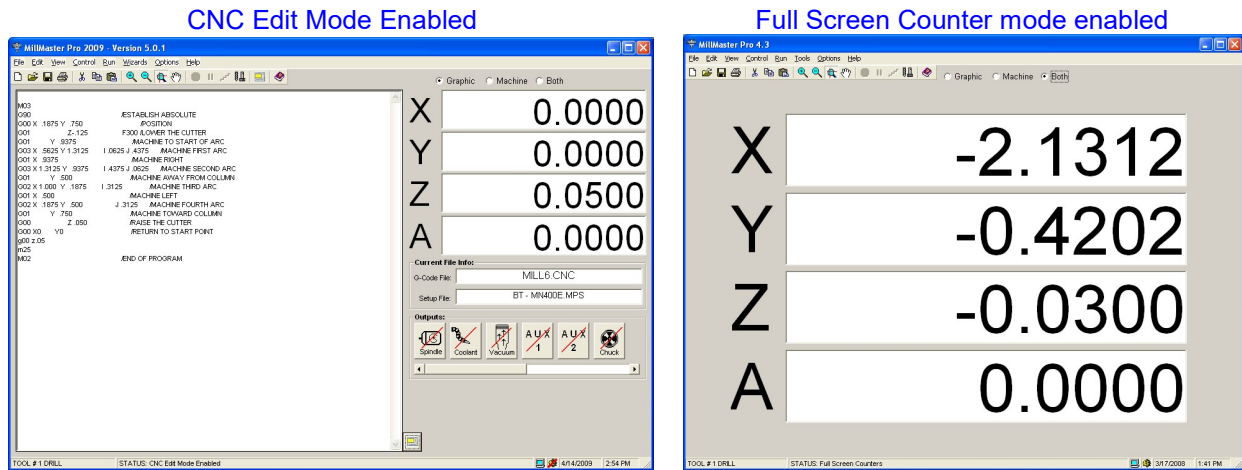
When a check mark appears next to this function, it is enabled. The graphic below represents the **Main screen in CNC Edit mode (left)** and the **Full Screen Count mode (right)**

View- CNC Edit Mode

Toggles CNC text edit mode. When on, **MillMaster** displays the CNC text editor while temporarily hiding the cutting window. This allows the user to Edit any currently loaded program, or enter a part program manually. This works like any standard Text editor. When a check mark appears next to this function, it is enabled.

Note that the **CNC Edit Mode** can also be toggled via the **<Ctrl> E** hotkey, or the small image of the application on the right side of the CNC textbox.

The graphic below represents the Main screen in **CNC Edit mode (left)** and the **Full Screen Count mode (right)**.



(These graphics have been reduced in size for spacing purposes)

View - Thin Lines

If this option is checked and a program is run in **Graphics** mode, the tool path will be thin (1 pixel). If this option is not checked, the tool path will be the actual width of the tool.

View - Arrows on Path Lines

This option is only available if **Thin Lines** is checked. If checked, arrows are drawn at the end of tool path lines to indicate the direction of the tool path.

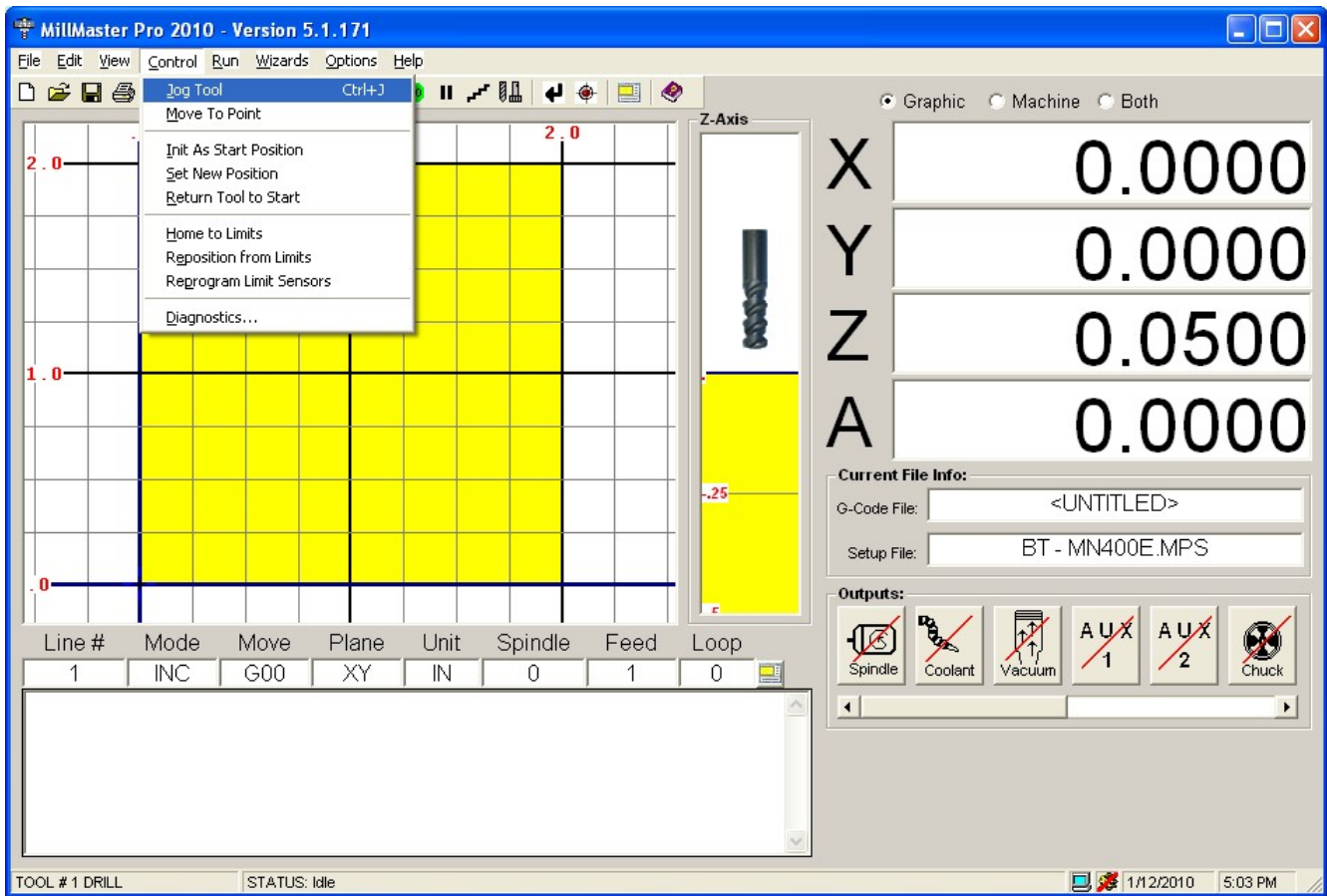
View - Redraw Screen

Redraws the screen. Any previous graphical run will be cleared, but the size and location of the viewport will be retained.

View - Restore Screen

Restores the screen to show the entire uncut raw stock. Choosing this option when in 'Graphics' mode resets the graphical XYZA counters. When in 'Machine' mode, only the machine counters are reset and when in 'Both' mode, both the graphical and machine counters are reset.

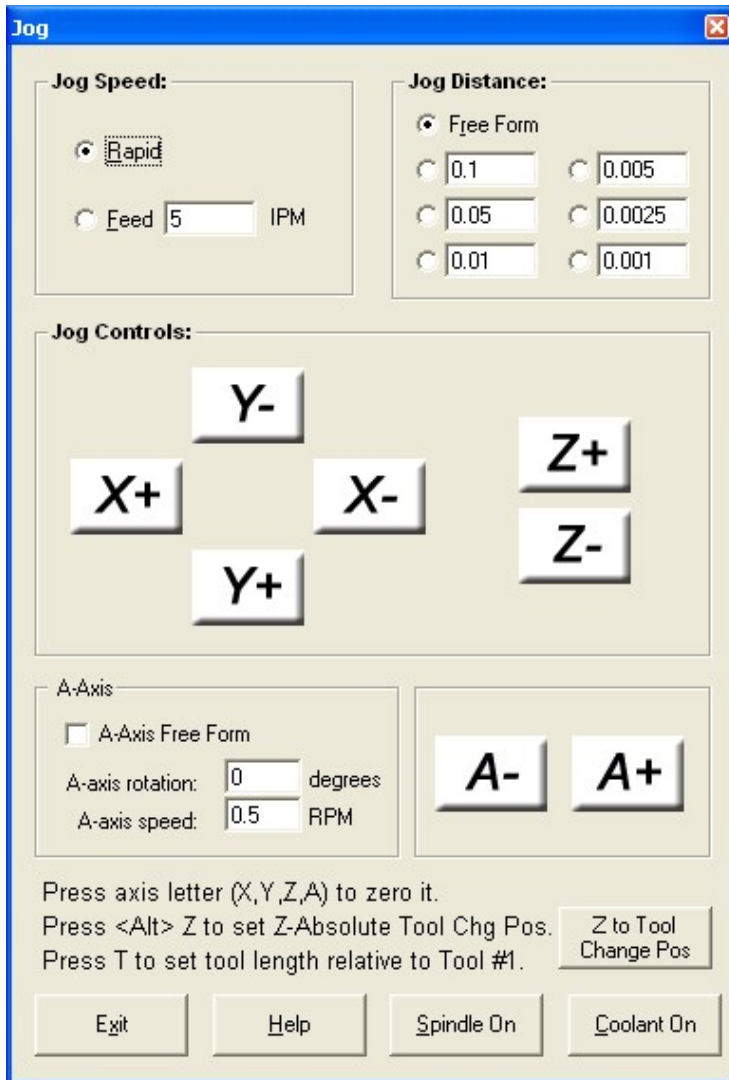
The values in the Material Size menu are applied here for the redrawing of the uncut stock material.



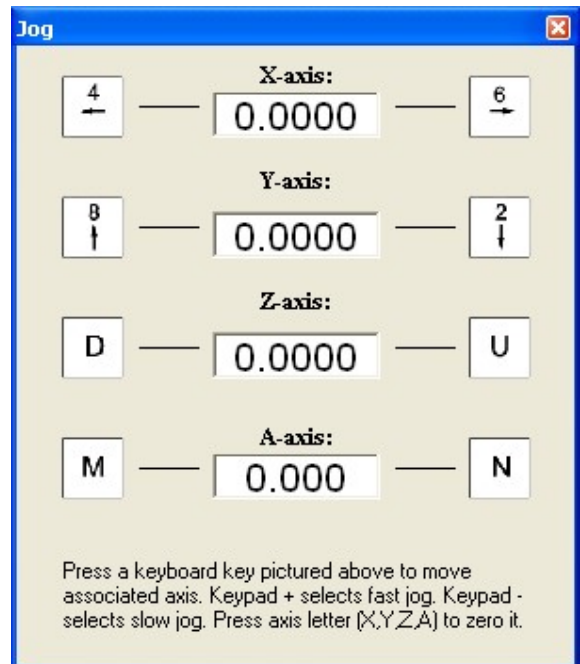
The **Control Menu** is used to jog the tool of the machine, set up the starting position of the machine, and calibrate the machine. The underlined letters are called **QuickSelect** keys. These allow access to a command only while the pull-down menu is displayed.

Control- Jog Tool

Allows the user to jog all axes on the machine. The absolute location of the tool is shown and updated when a jog key is released. A machine must be correctly connected to use this function.



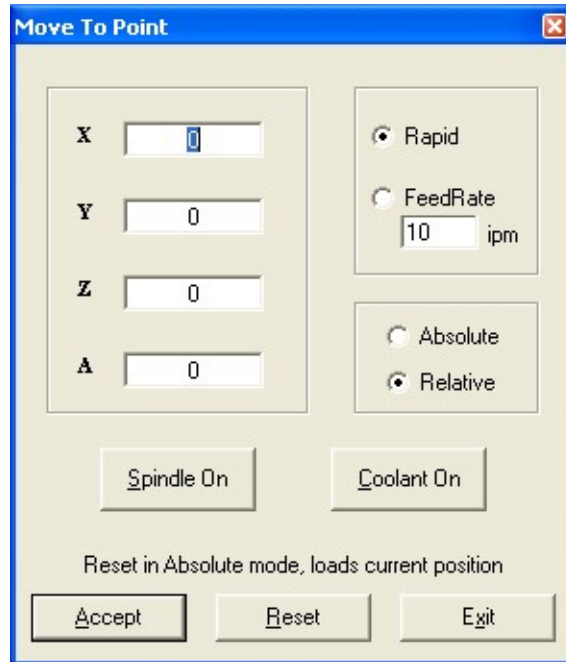
(MN400 Jog Controller)



(OptiStep & QuickPhase Controllers)

Control- Move To Point

Positions the currently selected tool to the **XYZ** coordinates entered in the Move To Point Menu. The program will reflect a cutting path **ONLY** if the **Z**-axis is below the surface of the part (**a negative number.**) You can select between a rapid move or a speed controlled move by selecting FeedRate and inputting a value.



Control- Init. As Start Position

Initiates the current position of the tool as the Start Position and resets the internal counters. The current location of the tool is assumed to be in the tool start position.

Control- Set New Position

Sets the location entered by the user and sets the internal counters to the new values. This does **NOT** affect the options Tool Start Position setting.

Control- Return Tool to Start

Moves the tool to the Tool Start Position. If **Machine Mode** is selected, the tool will move to the tool start position. This does **NOT** reset the internal counters.

Control- Home to Limits

Moves the machine to the limit sensors indicated in the Options-Limit Sensors Dialog Box. Limit switches must be connected to use this function.

Control- **Reposition from Limits**

Returns tool to its Home Limit switches, clears counters and then positions tool at the start point. You must use Reprogram from Limit Switch command to set the distances from each limit switch to the start point before using this command. Limit switches must be connected to use this function.

Control- **Reprogram Limit Sensors**

Automatically measures and stores the distance from the Start Point to each of the 3 Limit Switches. Tool must be positioned at its Start Point prior to selecting this function. Limit switches must be connected to use this function.

Control- **Diagnostics**

This allows the user to find the maximum jog and rapid traverse speeds of any or all axes on the milling machine. It is a self-running test that exercises the selected axes to determine the speed limits of the machine. The maximum speeds that are found by this function can be automatically entered into the **Position and Feed Speeds** window.

A file named DIAG.LOG that contains test information is created in the current directory and is a standard text file that can be read using any text editor. This option uses the home limit switches selected in the **Limit Sensors** option (**if <NONE> is selected for any axis, this test can not be performed for that axis**).

Simply select which axes you want to speed test and enter a value in the Distance field. We recommend a minimum of 4" to allow the machine time to ramp up to the higher speeds. The message window will display the time the move took and how fast the machine was traveling both in steps per second and inches per minute.

Axis	Unramped Distance	Ramped Distance
<input checked="" type="checkbox"/> X-Axis	0.5	4
<input type="checkbox"/> Y-Axis	0.5	2
<input type="checkbox"/> Z-Axis	0.5	2

Diagnostics Test Parameters

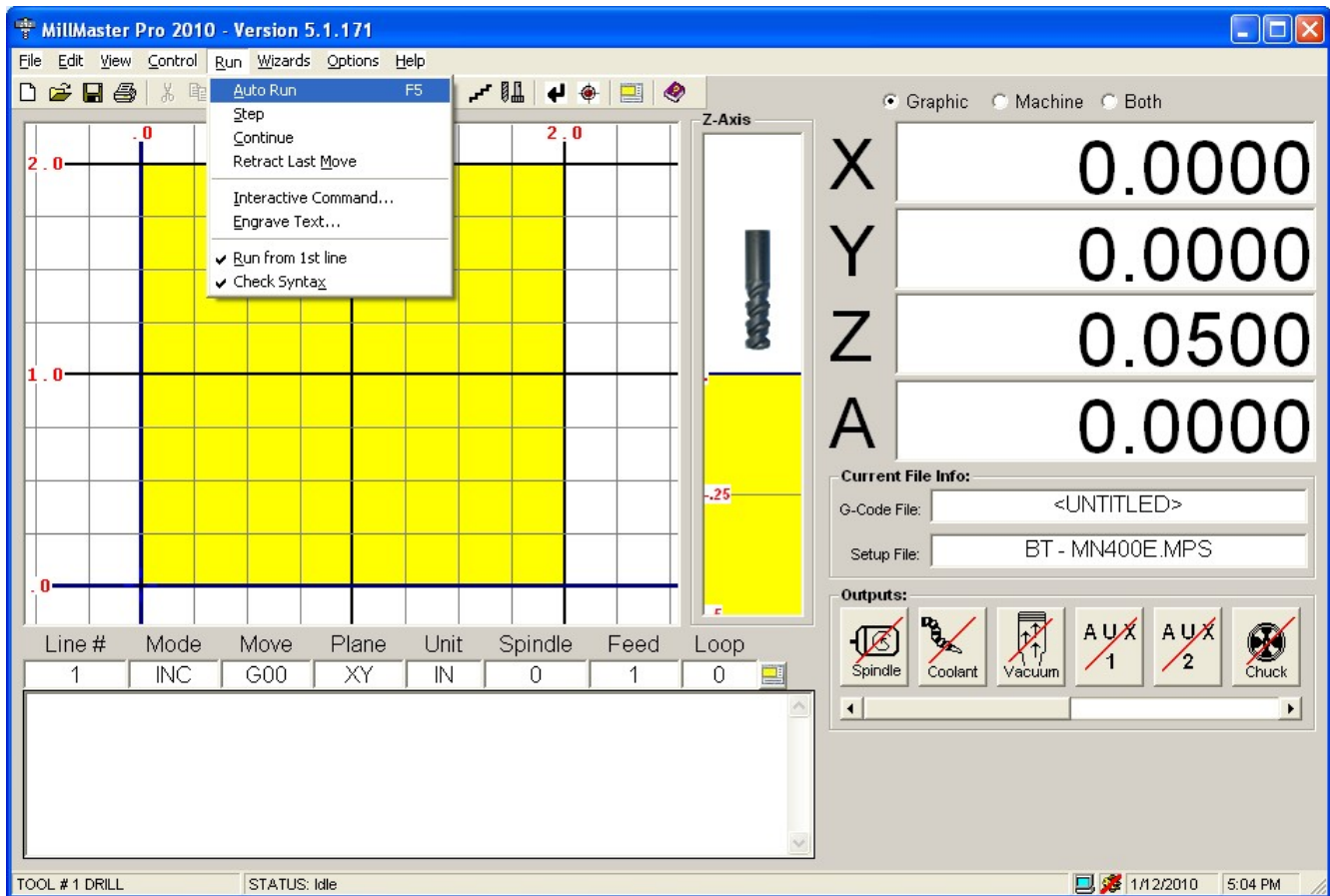
Start Speed (steps/sec): 1000

Speed Change (steps/sec): 500

Acceleration (steps/sec²): 5000

Tolerance (inches): 0.005

Buttons: Begin, Show Last, Print, Exit



The **Run Menu** is used to execute the part program. The part program can be tested graphically or it can be used to actually machine a part.

The underlined letters are called **QuickSelect** keys. These allow access to a command only while the pull-down menu is displayed.

Run- Auto Run

Executes the current part program in memory in continuous mode. Either **Graphics Mode** or **Machine Mode** must be enabled to use this option. The twelfth icon in the Toolbar also corresponds to this menu command

Run- Step Mode

Executes the current part program one line at a time pausing after each line. The user must click **<CONTINUE>** to execute the next program line.

Either **Graphics Mode** or **Machine Mode** must be enabled to use this option. The fourteenth icon in the Toolbar also corresponds to this menu command.

Run- Continue

This is used to continue executing the part program at the current instruction. Either **Graphics Mode** or **Machine Mode** must be enabled to use this option.

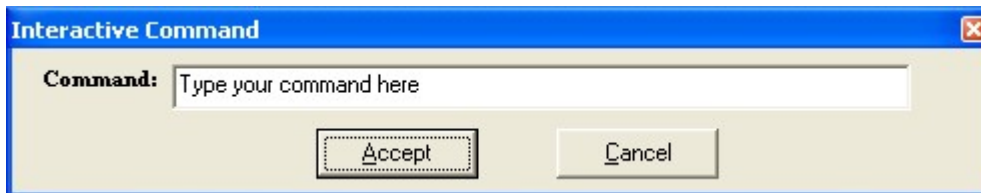
Run- Retract Last Move

This is used to "Undo" the last move executed by the software. This function can only be used in machine mode.

Run- Interactive Command

This allows the user to enter a single program line and see the results immediately following on the screen. This option can **NOT** be used in **Machine Mode**.

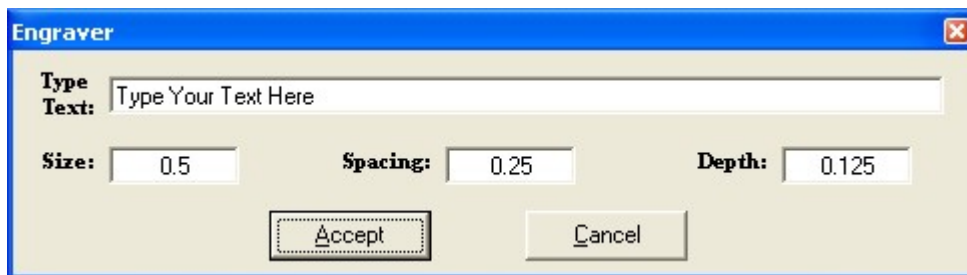
The graphic below represents the **Interactive Command Window**.



Run- Engrave Text

This option allows engraving text. Simply type a text line to be engraved and click **<ACCEPT>**.

MillMaster Pro will automatically enable **Graphics Mode**, disable **Machine Mode**, and will engrave the line graphically starting at the current tool position. The G-codes produced will be appended to the end of the current part program. The graphic below represents the **Engrave Text Window**.



You may change the Size, Spacing, and Depth parameters by entering new values and clicking **<ACCEPT>**.

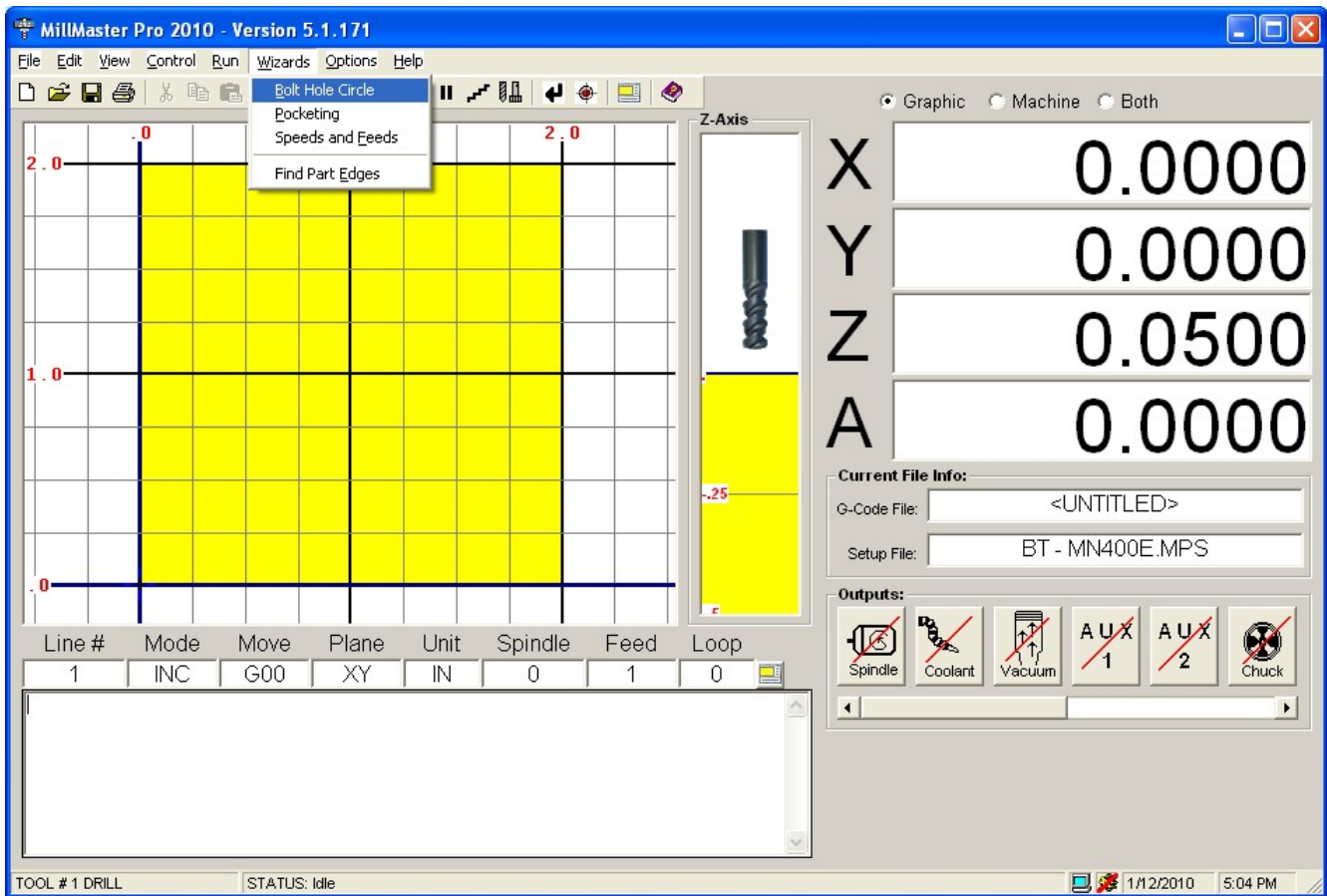
To machine, select **Machine Mode** and then **Auto Run**.

Run- Run from 1st line

This option allows the user to start the part program at any good line number. When checked (default), the part program will start from the first line in the file. If unchecked, the software will prompt the user to enter the line number they wish to start the part program from.

Run- Check Syntax

This option allows the user to check their G-code for errors while executing the program. When checked (default), the syntax is checked right before each line of code is executed. If something is wrong, a message box will pop up indicating the errors which were found and the user will be prompted to continue with the program or stop to correct the problem. If unchecked, the program will run normally.



The **Wizards Menu** gives the user a set of wizards to help in the creation of a part.

The underlined letters are called **QuickSelect** keys. These allow access to a command only while the pull-down menu is displayed.

Wizards- Bolt Hole Circle

Opens the Bolt Hole Circle Window which allows the user to define a bolt hole circle pattern. This subroutine will write g-code for the user based on a series of variables and have the end result be coordinates on the XY plane for where their mill will dive into the material. The user will then be able to edit the g-code based on whether they are running a G81, G82, or G83 command and input the corresponding Z axis cutting move after each move to a new coordinate.

The screenshot shows the 'Bolt Hole Circle' wizard window. It contains input fields for 'Number of Bolt Holes' (9), 'Bolt Circle Diameter' (2), 'Hole Diameter' (0.375), 'Degrees Offset for Hole #1' (90), 'Center X Coordinate' (5), and 'Center Y Coordinate' (5). A checkbox 'Display Coordinates on Diagram?' is checked. The 'Results' section shows 'Dist Btwn Hole Centers' as 0.684067 and 'Dist Btwn Hole Edges' as 0.309067. A central diagram shows a circle with 9 smaller circles inside, each labeled with its coordinates. A list on the right shows the resulting g-code for each hole. At the bottom are buttons for 'Copy G-Code', 'Paste G-Code', 'Clear Results', 'Compute', and 'Exit'.

Hole #	X	Y
1	5	6
2	4.3572	5.766
3	4.0152	5.1736
4	4.134	4.5
5	4.658	4.0603
6	5.342	4.0603
7	5.866	4.5
8	5.9848	5.1736
9	5.6428	5.766

```
1 · X=5, Y=6
2 · X=4.3572, Y=5.766
3 · X=4.0152, Y=5.1736
4 · X=4.134, Y=4.5
5 · X=4.658, Y=4.0603
6 · X=5.342, Y=4.0603
7 · X=5.866, Y=4.5
8 · X=5.9848, Y=5.1736
9 · X=5.6428, Y=5.766
```

```
X5 Y6
X4.3572 Y5.766
X4.0152 Y5.1736
X4.134 Y4.5
X4.658 Y4.0603
X5.342 Y4.0603
X5.866 Y4.5
X5.9848 Y5.1736
X5.6428 Y5.766
```

Wizards- *Pocketing*

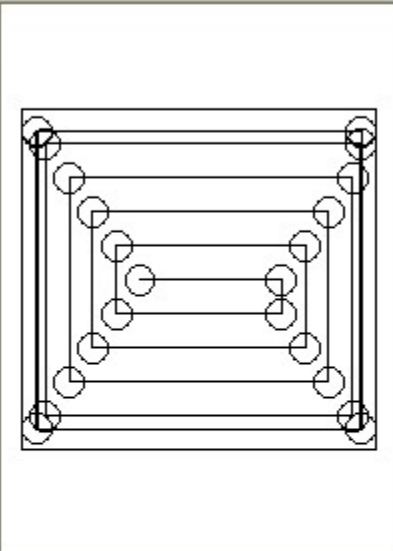
Opens the Pocketing Window which allows the user to define either a rectangular or circular pocket. This subroutine will write g-code for the user based on a series of variables and have the end result be a small program that will cut a pocket in their material.

Pocketing Rectangles and Circles

Shape of Pocket:
 Rectangle Circle

Tool Diameter: .25
Cut Width Per Pass: .2
Finish Wall Clearance: .003
Finish Depth Clearance: .002
Cut Depth Per Pass: 0.05
Start Z-Level: 0
Final Z-Level: -0.1
Bottom Left Corner X: 0
Bottom Left Corner Y: 0
Rectangle Length (X): 3
Rectangle Width (Y): 2
Circle Diameter: 2
Plunge Feed Rate (IPM): 1
Cut Feed Rate (IPM): 8

Tool Path:



Clear Tool Path

Programming Mode:
 Absolute Incremental

Copy G-Code Paste G-Code Compute Exit

G-Code:

```
G90 /Absolute Positioning
/**** Cutting Pass #1, Cut Depth =
-0.05
G00 X1 Y1 /Move to initial drill
G01 Z-0.05 F10 /Drill down to -0.05
G01 X2.2 F80 /Initial cut to the Right
G01 Y0.8 /Cut downward
G01 X0.8 /Cut to the left
G01 Y1.2 /Cut upward
G01 X2.4 /Cut to the right
G01 Y0.6 /Cut downward
G01 X0.6 /Cut to the left
G01 Y1.4 /Cut upward
G01 X2.6 /Cut to the right
G01 Y0.4 /Cut downward
G01 X0.4 /Cut to the left
G01 Y1.6 /Cut upward
G01 X2.8 /Cut to the right
G01 Y0.2 /Cut downward
G01 X0.2 /Cut to the left
G01 Y1.8 /Cut upward
```

Clear G-Code

Generate Comments ?

Wizards- Speeds and Feeds

Opens the Cutting Speeds and Feeds Window which allows the user to the spindle speed that they need to be running their machine at depending on material, number of teeth, and other variables which come into play when determining cutting speed. Simply input the proper data and let the program calculate your spindle rpm for you.

Cutting Speeds and Feeds

Material Category : Plain Carbon and Alloy Steels

Material : Alloy Steels 1330-1345, 4032-4047, 4130, et al

Brinell Hardness : 175-225

Material Condition :

Tool Material : HSS

Delete Change Add New

Recommended Cutting Speed : 30 Cutting Feet Per Minute

Actual Cutting Speed : 30 Cutting Feet Per Minute

Type of Mill

End Mill Face Mill Slit or Slot Mill

Number of Teeth in Cutter : 10

Cutting Diameter : .5

Inches Millimeters

Computed Results:

Recommended Spindle Speed (RPM) : 229 rpm

Actual Spindle Speed (RPM) : 229

Feed Rate (Inches Per Tooth) : 0.008 to 0.015 ipt

Feed Rate (Inches Per Minute) : 18.3 to 34.3 ipm

These Speeds/Feeds are suggestions only. Always reference the Machinery's Handbook or other published Materials. Actual applications may vary.

For End Milling, data is based on a 3-tooth, 20 degree helix angle tool with a diameter of 1.0 inch, an axial depth of cut of 0.2 inch, and a radial depth of cut of 1 inch (full slot).

For Face Milling, data is based on a 10-tooth, 8-inch diameter face mill, operating with a 15 degree lead angle, 3/64 inch nose radius, axial depth of cut = 0.1 inch, and radial depth (width) of cut = 6 inches.

For Slit or Slot Milling, data is based on an 8-tooth, 10 degree helix angle tool with a cutter width of 0.4 inch, diameter D of 4.0 inches, and a depth of cut 0.6 inch.

Exit Compute Speed Compute Feed

Wizards- Find Part Edges

Finds internal and/or external edges of a part or piece of material, based on user-specified options. This screen also allows calibration of the probe diameter by using a known circular or rectangular cutout (pocket).

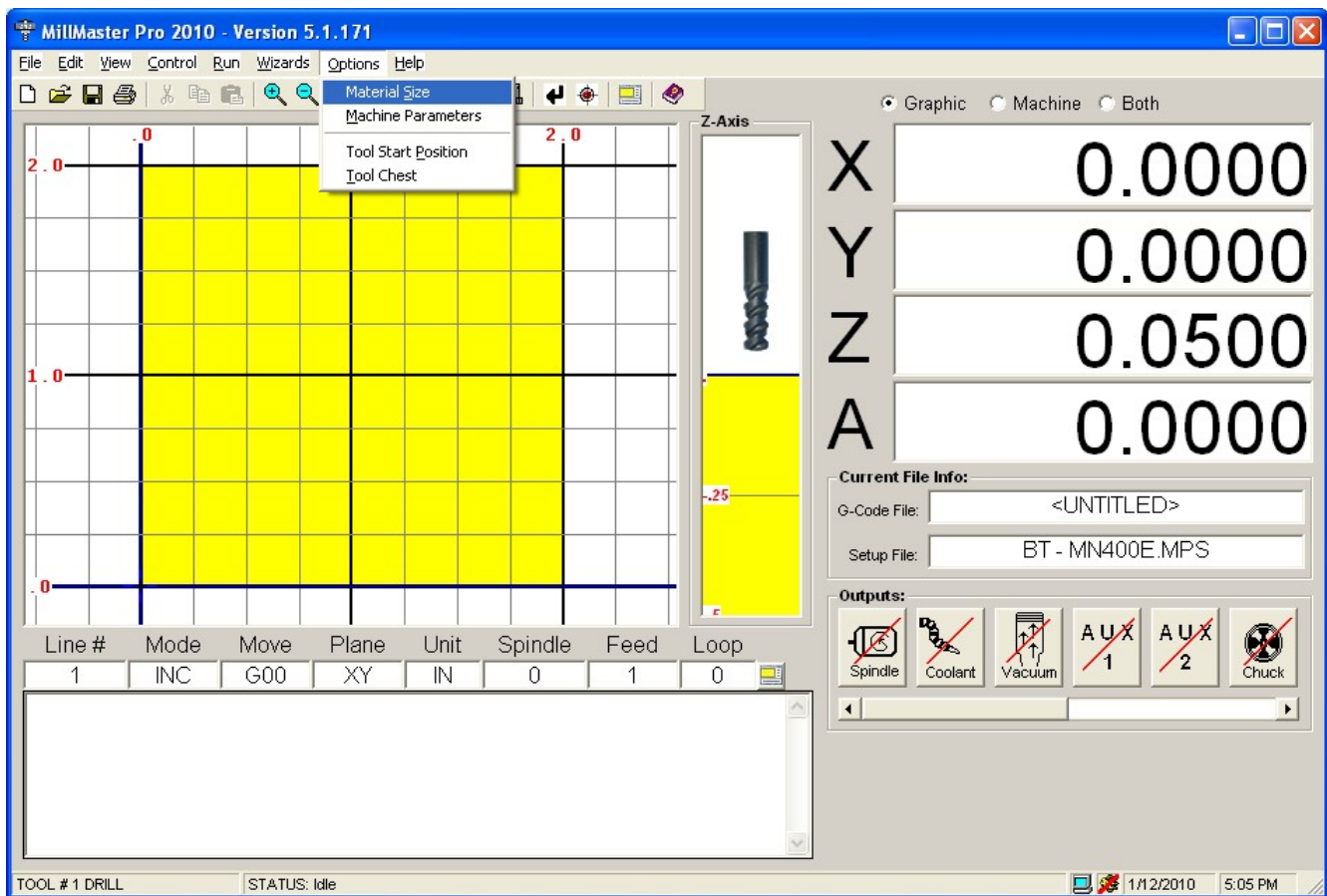
To use this screen, you must have a digital probe installed and wired to one of the Inputs (1-8).

Wizards- Collect Part Geometry

Collects part geometry for circular, rectangular or irregular smooth shaped cutouts (pockets), or an irregular smooth shaped island (boss). A series of points is collected and either stored as G-Code (for circular or rectangular shapes) or stored in a separate **DXF** file as a series of *POINTS* (for irregular smooth shapes).

If a **DXF** file is created, it can be imported by **MillMaster Pro** or **MultiCAM**.

To use this screen, you must have a digital probe installed and wired to one of the Inputs (1-8). Before collecting geometry on a part, you should calibrate the probe using the **Wizards | Find Part Edges** screen.

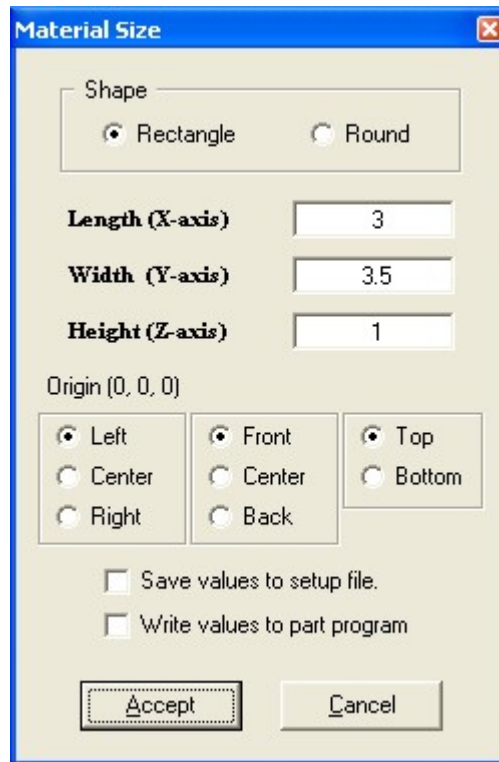


The **Options Menu** is used to set the material size, machine parameters, start position and create a tool chest.

The underlined letters are called **QuickSelect** keys. These allow access to a command only while the pull-down menu is displayed.

Options- *Material Size*

Allows the user to enter the material **Length**, **Width**, and **Height** in the units specified in the **Machine Parameters** window.



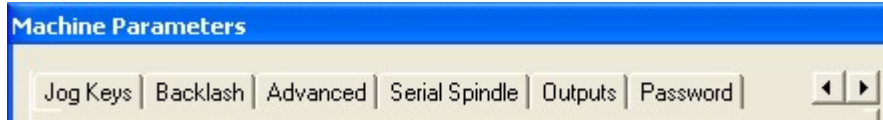
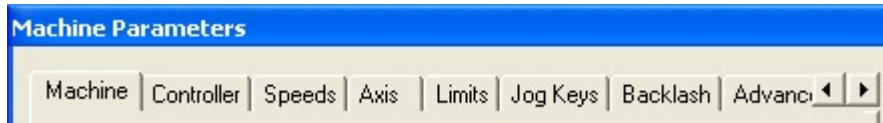
The image shows a dialog box titled "Material Size" with a blue title bar and a close button. The dialog is divided into several sections. At the top, there is a "Shape" section with two radio buttons: "Rectangle" (selected) and "Round". Below this are three input fields: "Length (X-axis)" with the value "3", "Width (Y-axis)" with the value "3.5", and "Height (Z-axis)" with the value "1". Underneath these is the "Origin (0, 0, 0)" section, which contains three groups of radio buttons. The first group has "Left" (selected), "Center", and "Right". The second group has "Front" (selected), "Center", and "Back". The third group has "Top" (selected) and "Bottom". At the bottom of the dialog, there are two checkboxes: "Save values to setup file." and "Write values to part program", both of which are currently unchecked. Finally, there are two buttons at the very bottom: "Accept" and "Cancel".

If the **Save values to setup file** checkbox is checked and the *Accept* button is pressed, then these values are written to the default setup file. If the **Write values to part program** checkbox is checked and the *Accept* button is pressed, then these values are written to the current G-Code program file via *metacommands*.

Options- *Machine Parameters*

Allows the user to set Machine Parameters such as Units of Measure, Origin, select the type of controller (**MN400**, **OptiStep Plus** or **QuickPhase**), set the base address, in decimal, of the controller, Set Machine Control Speeds, Axis Control Parameters, Axis Limit Sensors, Jog Key Assignments, output delays and allows the user to set up a password field to access or change all of these parameters. The user may also select a Machine Type. This specifies what kind of machine **MillMaster Pro** is working with. This menu can be quickly accessed with a **Mouse** click by left clicking in the whitespace of the "Setup File" box in the Current File Info. frame. Below are two images representing the file tabs on the machine parameters page.

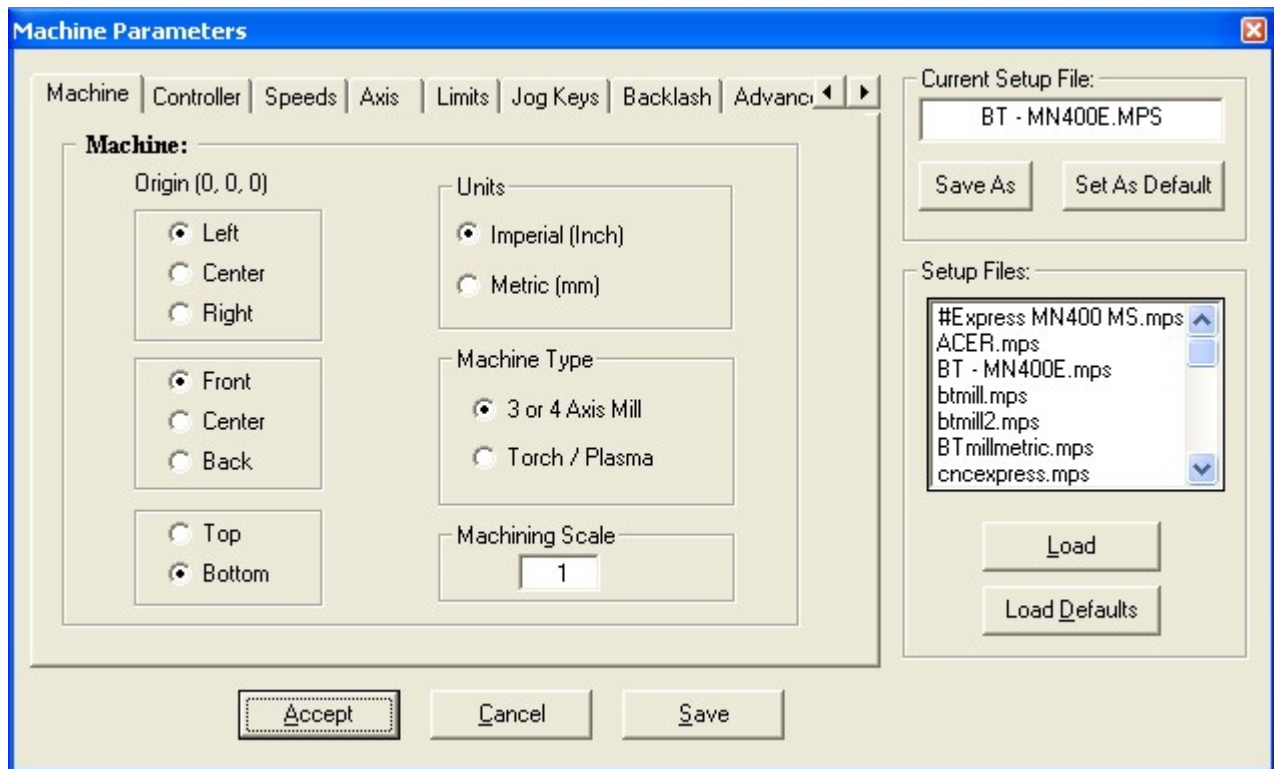
From any tab in **Machine Parameters**, the user can Load a new setup file which stores all of the users settings for their machine. Simply click one of the files under the **Setup Files** frame and press the **Load** button just below this. If you know that this is the file that you are going to want to use every time you start the software, you can then press the **Set As Default** button at the top of this frame.



Options– Machine Parameters - Machine

Allows the user to set the origin of the machine, unit of measurement and machine type.

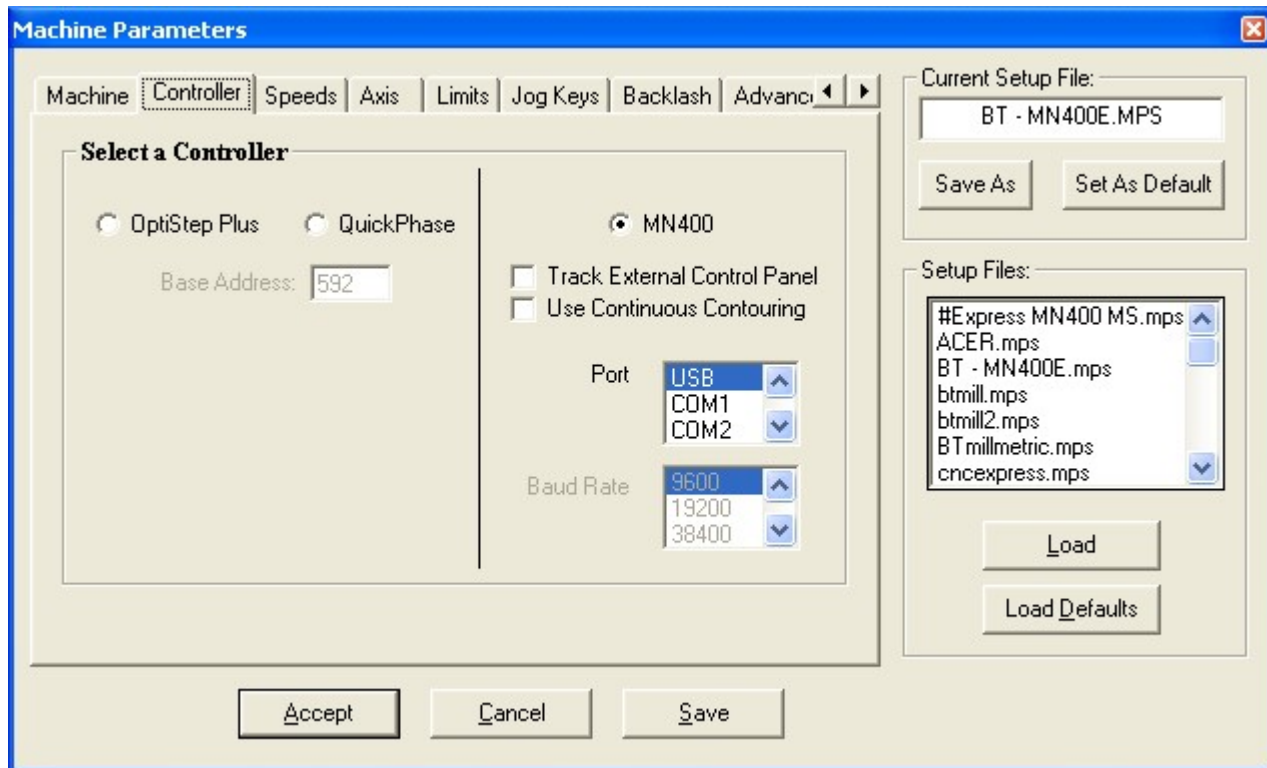
Also allows the user to set the Machining Scale. A value of 1 in this field runs your program with the values unchanged. By entering a 2 here, the movements in your program will all be doubled for every command and your part will essentially be twice the size of the original. Switching the Machine Type will change certain aspects of the functionality of the program depending on what type of machine you will be running.



Options– *Machine Parameters - Controller*

Allows the user to select which controller card they are using and enter its corresponding base address. The MN400 also allows you to select whether or not you want to turn on Continuous Contouring which ramps motors in such a fashion as to get the most performance out of your program. It interprets what type of move is coming next and adjusts the speed of the motor to maximize the efficiency of the transition.

****NOTE:** For the OptiStep Plus and QuickPhase controllers, the DIP switch settings on the card must match the Base Address specified.



Options– *Machine Parameters - Position and Feeds Speeds*

Allows the user to enter the rates at which the tool will travel.

Max Unramped is the maximum instantaneous rate the tool can move reliably. Since this is the speed that the motor is trying to start out at, it is expected to see a lower number here.

Ramped Moves is the top speed the machine can travel at with acceleration. Used for all G00 moves.

Ramp Speed is the acceleration of the stepping motors.

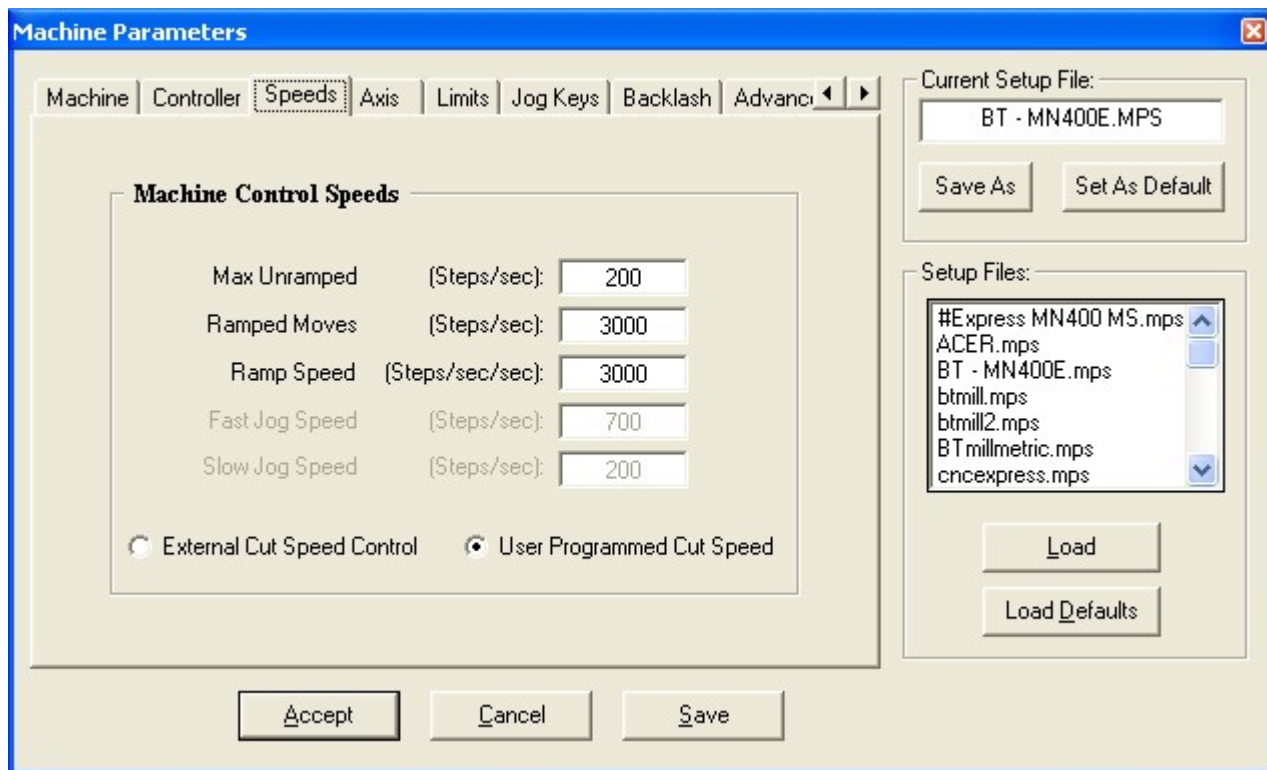
Fast Jog Speed is the rate the machine will travel in the *Jog Mode* after the “Fast Jog” button (+ key on the number keypad) is pressed on the screen.

Slow Jog Speed is the rate the machine will travel in the *Jog Mode* after the “Slow Jog” button (- key on the number keypad) is pressed on the screen.

The radio buttons next to User Programmed Cut Speed or External Cut Speed Control (**for use with an OptiStep Plus controller only**) indicates which cut speed mode is enabled by placing a black dot inside the circle. User Programmed Cut Speed mode uses the feedrate (**F**) codes embedded in the part program to set the cutting speed and External Cut Speed Control mode uses an external pulse to regulate the cutting speed.

Rapid Traverse moves are not affected by this option.

See the wiring diagram in Appendix C for instructions on how to hook-up the **OptiStep Plus** for the External Cut Speed Control option.



Options- *Machine Parameters - Axis Control Parameters*

Allows the user to enter the parameters that describe the lead screw pitch, the axis polarity, and the number of logical steps per stepper motor revolution. NORMAL axis polarity is defined as when the stepper motor rotates CCW, as viewed from the motor end, it generates a positive linear move. REVERSE axis polarity generates a negative linear move under the same circumstances.

****Note:** If REVERSE is selected for any axis, then the limit sensor connections must also be reversed **

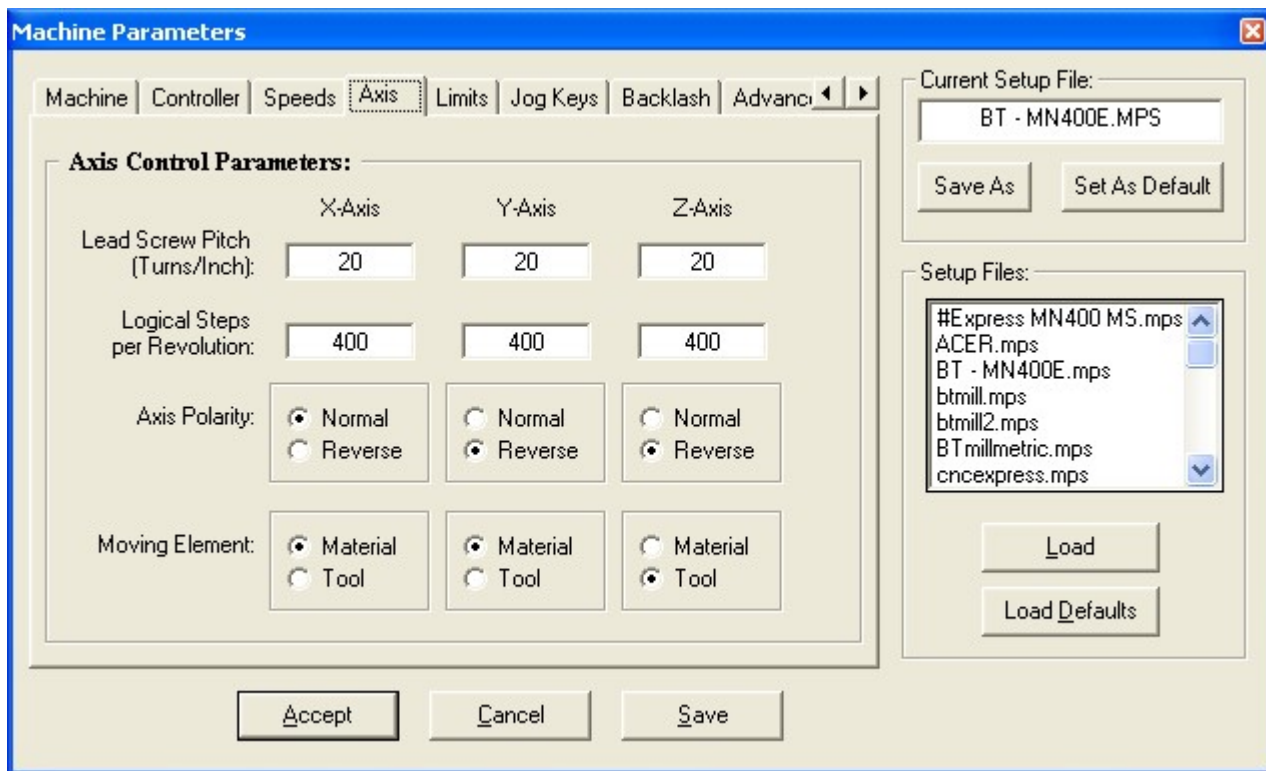
(i.e. connect a positive limit switch to the negative input).

Logical step per revolution is the number of step pulses required to move the stepping motor one full revolution taking into account the motor driver resolution setting.

Example:

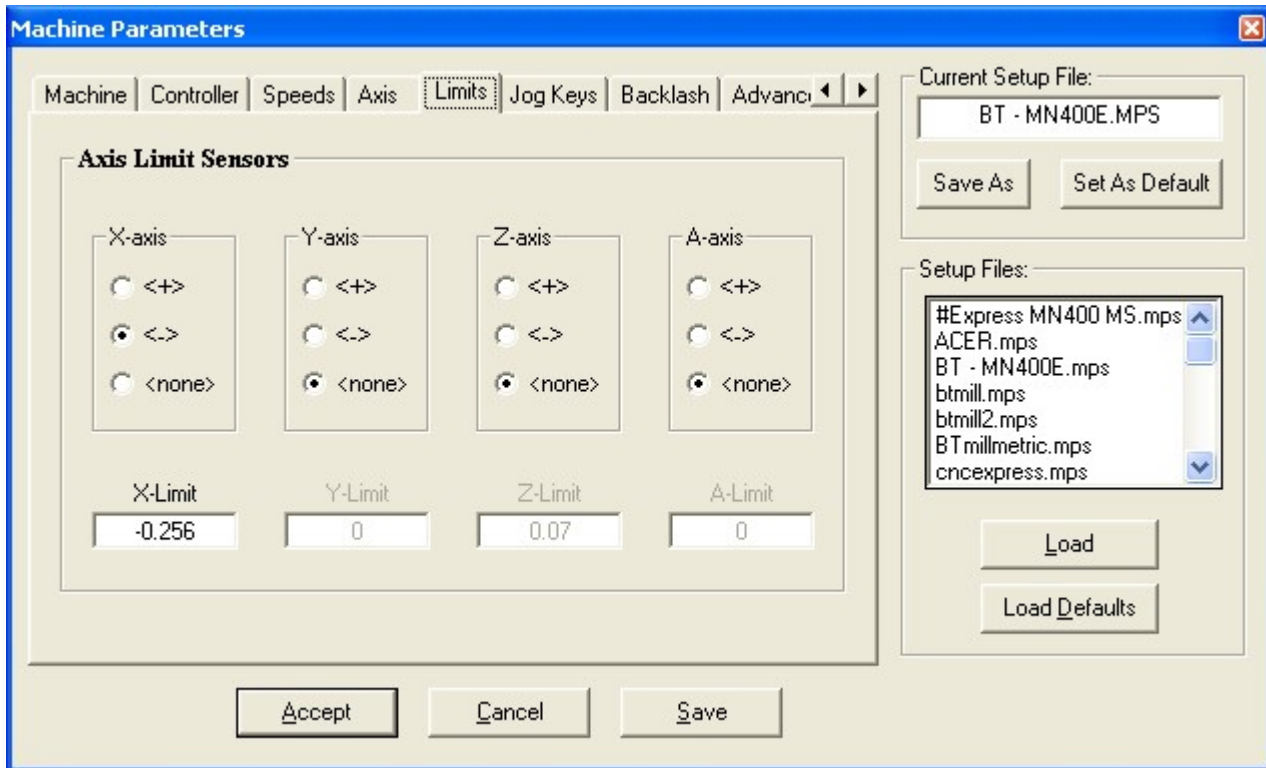
Motor Res.Driver Res.Logical Steps/Rev

200	FULLSTEP	200
200	HALFSTEP	400



Options- *Machine Parameters - Limit Sensors*

Allows the user to select the limit switches that will be used to home to. Enter the position of the home limit sensor for each axis in reference to the origin of the part. On the Z-axis only, enter the position of the bottom of the tool holder in reference to the origin of the part.



Options- Machine Parameters - Jog Keys Assignment

Allows the user to choose which keys on the computer keyboard control the jog movement of the machine. The default sets the Z-axis to jog UP when <U> is depressed and DOWN when <D> is depressed, the X-axis to jog with the left and right arrow keys, and the Y-axis to jog with the up and down arrow keys. If you wish to change the jog keys, you should click on the button that corresponds with the axis you wish to change. This will display a small window with two buttons on it, OK and CANCEL. Press the key you wish to assign to that axis and the display window will tell you if it is a valid key. If it is, an image of the key you pressed will appear in the window. If not, a message stating 'Invalid Key' will be displayed and you will need to choose another key. To accept a valid key, click the OK button and you will notice the corresponding SCAN code is entered automatically into the Jog Keys page along with an image of that key.

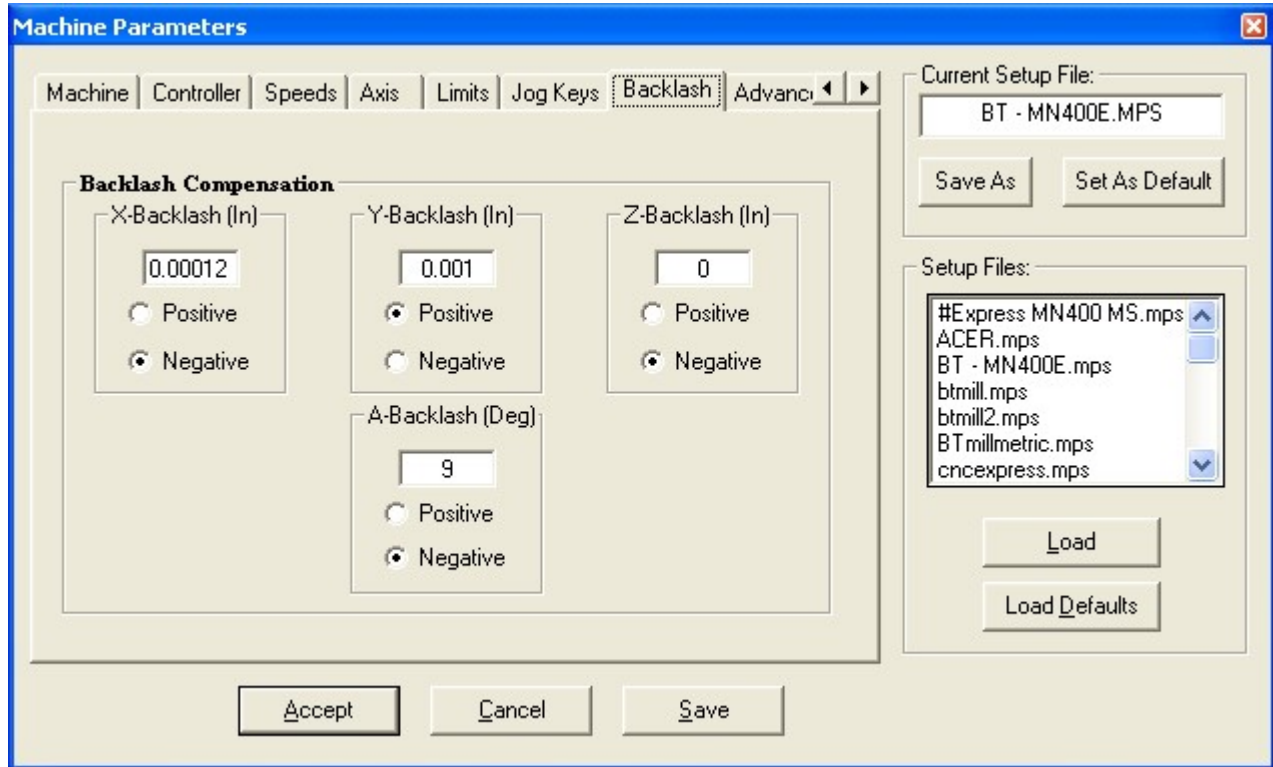
Jog Tap Steps is the number of steps the motor will rotate when a jog key is tapped. A value between 1 and 8 is accepted in this field.

The screenshot shows the 'Machine Parameters' dialog box with the 'Jog Keys' tab selected. The dialog is divided into several sections:

- Jog Key Assignments:** A table with columns for X-axis, Y-axis, Z-axis, and A-axis. Each column has a 'Jog Key Code' field, a key image button, and a 'Jog Res' field. The current assignments are: X-axis (77, right arrow), Y-axis (72, up arrow), Z-axis (17, W), A-axis (50, M). Below this, another set of assignments is shown: X-axis (75, left arrow), Y-axis (80, down arrow), Z-axis (31, up arrow), A-axis (49, A-). A 'Click to assign -->' label is next to the bottom row of key images.
- Resolution Fields:** Four 'Jog Res' fields with values of 0.00025, labeled 'Jog Res High', 'Res Med-High', 'Res Med-Low', and 'Jog Res Low'.
- Current Setup File:** A text box containing 'BT - MN400E.MPS' and buttons for 'Save As' and 'Set As Default'.
- Setup Files:** A list box containing several files: '#Express MN400 MS.mps', 'ACER.mps', 'BT - MN400E.mps', 'btmill.mps', 'btmill2.mps', 'BTmillmetric.mps', and 'cncexpress.mps'. Below the list are 'Load' and 'Load Defaults' buttons.
- Buttons:** 'Accept', 'Cancel', and 'Save' buttons at the bottom of the dialog.

Options– *Machine Parameters - Backlash compensation*

Allows the user to enter values for **X**, **Y**, **Z**, and **A** backlash compensation, activate encoders by axis and enter closed loop encoder values. If you notice some constant backlash on your machine that just can not be eliminated mechanically, then you can input this value into the appropriate axis field. You can also specify what direction the backlash is associated with by selecting either Positive or Negative.



Backlash is play or looseness between machine parts. If there is backlash, some commanded rotation of the motor is needed to take up this backlash before the machine starts to move.

Determine the amount of backlash in inches and select positive or negative initial compensation.

With a positive backlash setting, additional motor rotation is provided for the first negative move.

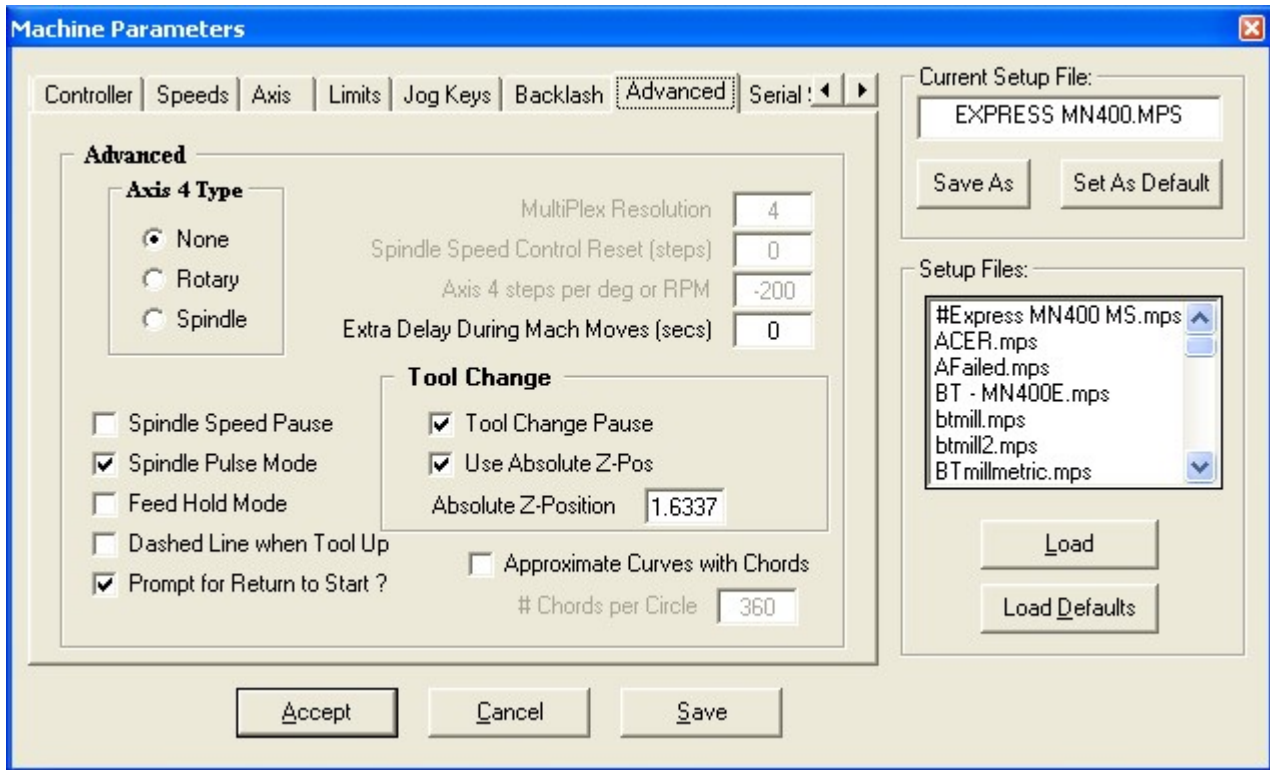
With a negative backlash setting, additional motor rotation is provided for the first positive move.

In both cases additional motor rotation is commanded at each subsequent direction reversal.

Note: After the introduction or change of a backlash setting, it is necessary to exit and restart the control software.

Options– **Machine Parameters - Advanced**

Allows the user to enter the fourth axis type (if applicable), set other attributes associated with a fourth axis, and set the other advanced functions described below.



When the **Rotary** option is selected you will need to enter the steps per degree of your rotary table. The MicroKinetics 8" rotary table requires a value of 200 for its gearing, while the 4" requires a value of 80.

If you select the **Spindle** option, then you will need to enter a value for RPM in this field as well as the reset speed.

MultiPlex Resolution refers to resolution selection for diagonal linear moves. It allows for smoother movements while sacrificing your top speed. For most applications a multiplex of 2 works as a good compromise. 4 causes an added burden on the CPU but creates very smooth diagonal moves. 1 is for no multiplexing at all and can be a bit rough on large router tables and milling machines that do not have microstepping drivers.

Spindle Speed Pause - This parameter specifies the action **MillMaster Pro** will take when it encounters an M03 command in a CNC part program. If checked, MillMaster will pause the CNC program and any machining so the machinist can adjust the spindle speed. If it is not checked, the software will continue without any interruption.

Spindle Pulse Mode - If checked, this parameter allows magnetic starters and 2-coil latching relays to be activated correctly while maintaining full manual control.

Feed Hold Mode - This parameter specifies the action **MillMaster Pro** will take when the shield is opened. If checked, the software will simply pause the machining (spindle motor remains on). If unchecked, the software will stop machining, turn the spindle motor off and prompt the user to Abort or Continue.

Dashed Line when Tool Up - If checked, **MillMaster Pro** will draw a thin dashed line in **Graphics** mode when the tool is above the material. Note that this option only affects **Graphics** mode and does not affect milling.

Tool Change Pause - This parameter specifies the action **MillMaster Pro** will take when it encounters an M06 command in a CNC part program. If checked, MillMaster will pause the CNC program and any machining so the machinist can manually change the tool. If it is not checked, the software will assume you have a multi-tool head and continue without any interruption.

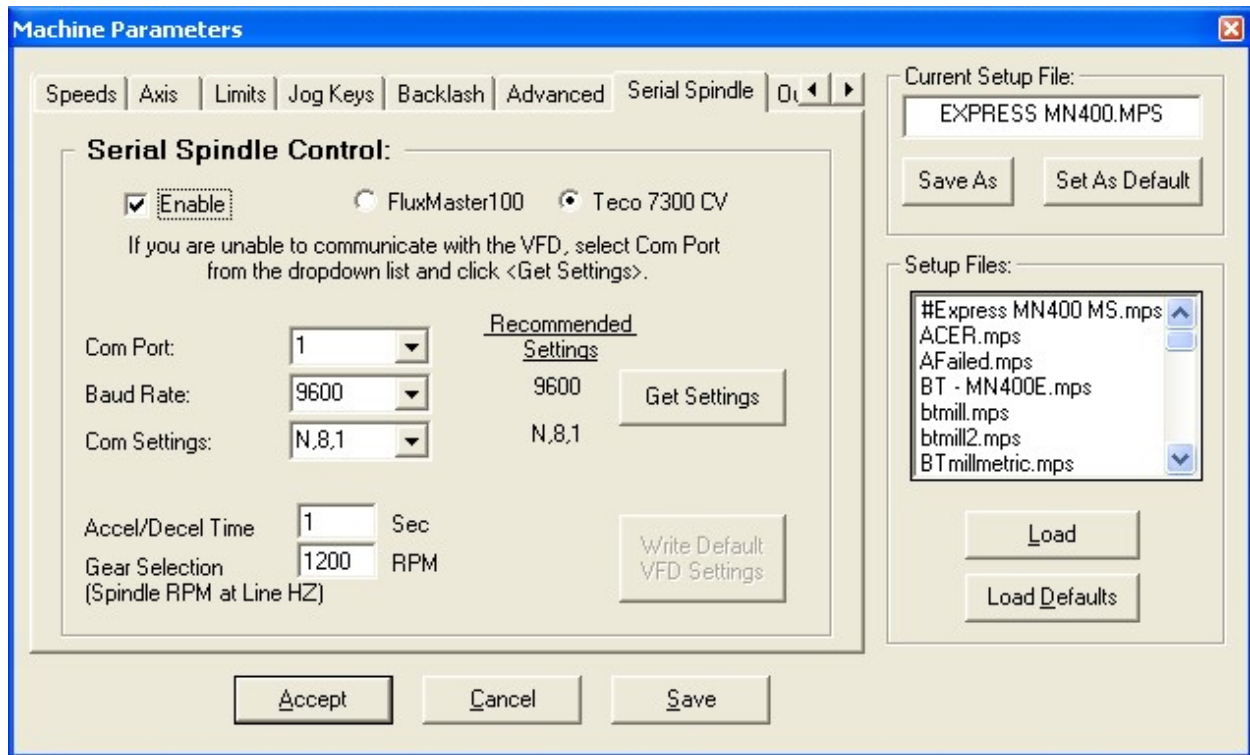
Use Absolute Z-Pos - If checked the Z-axis will move to the specified location before any tool change pause or automatic tool change takes place.

Approximate Curves with Chords - If checked, **MillMaster Pro** will mill arcs as a series of straight lines. The number of straight lines (chords) per circle is determined by user input. The default is 360 chords per circle, or 1 degree per chord. Thus, using the default values, a complete circle would be milled as 360 straight lines. Note that this option only affects milling and does not affect the **Graphics** mode.

Extra Delay During Mach Move - Normally this value is 0. If any value is entered, this delay will occur each time **MillMaster** checks for the final tool position during a machine move. The value entered must be less than 1 second.

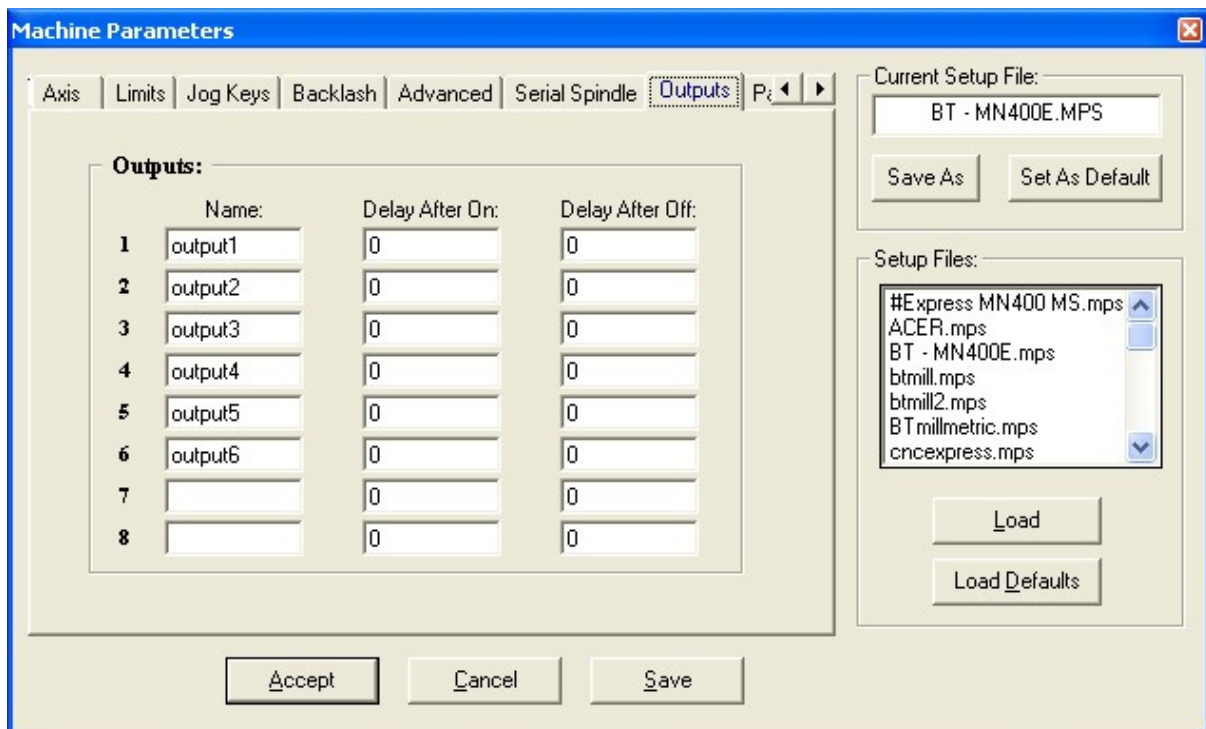
Options– Serial Spindle

Allows the user to enter communications settings if a serial port spindle is used.



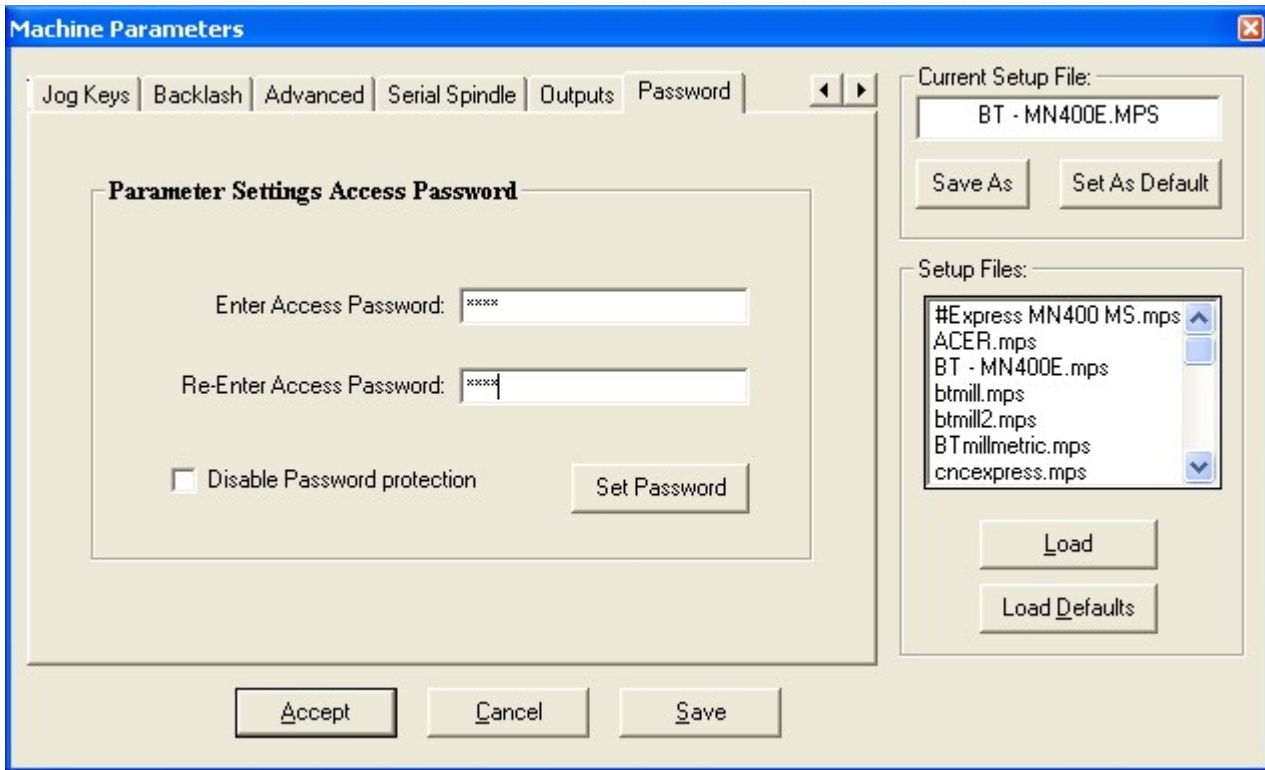
Options– Machine Parameters - Outputs

Allows the user to enter specific name and delay values for up to eight numbered outputs.



Options– Machine Parameters - Password

Allows the user to specify a password to protect the machine parameters page from unauthorized changes that could cause damage to the machine and/or cause injury.



Options- Machine Parameters - Save As

Allows the user to save the current setup parameters to a setup file of choice or create a new one.

Options- Machine Parameters - Set As Default

Saves the current settings to the default configuration file. All the options set will become permanent.

Options- Machine Parameters - Load

Allows the user to load a setup file. A setup file has an .MPS filename extension and configures **MillMaster Pro** with the setup parameters in the file. This allows the user to quickly change between different setups.

Options- Machine Parameters - Load Defaults

Clears the current settings and restores the defaults from disk.

Options- Machine Parameters - Accept

Saves the current setup parameters to the current setup file.

Options- *Machine Parameters - Cancel*

Cancels any changes to the setup parameters you have made and does not save.

Options- *Machine Parameters - Save*

Saves the current setup parameters to the current setup file.

Options- *Start Position*

Allows the user to specify the position of the tool at the beginning of the program. You can also select to have these values saved to your setup file for future use.

The image shows a dialog box titled "Enter Tool Start Values:" with a close button in the top right corner. The dialog has a light beige background and a blue border. It contains four rows of labels and input fields: "X-Position:" with a text box containing "0.5", "Y-Position:" with a text box containing "0", "Z-Position:" with a text box containing "0.05", and "A-Position:" with a text box containing "0". Below these fields is a checkbox labeled "Save values to setup file" which is currently unchecked. At the bottom of the dialog are two buttons: "Accept" and "Cancel".

Options- *Tool Chest*

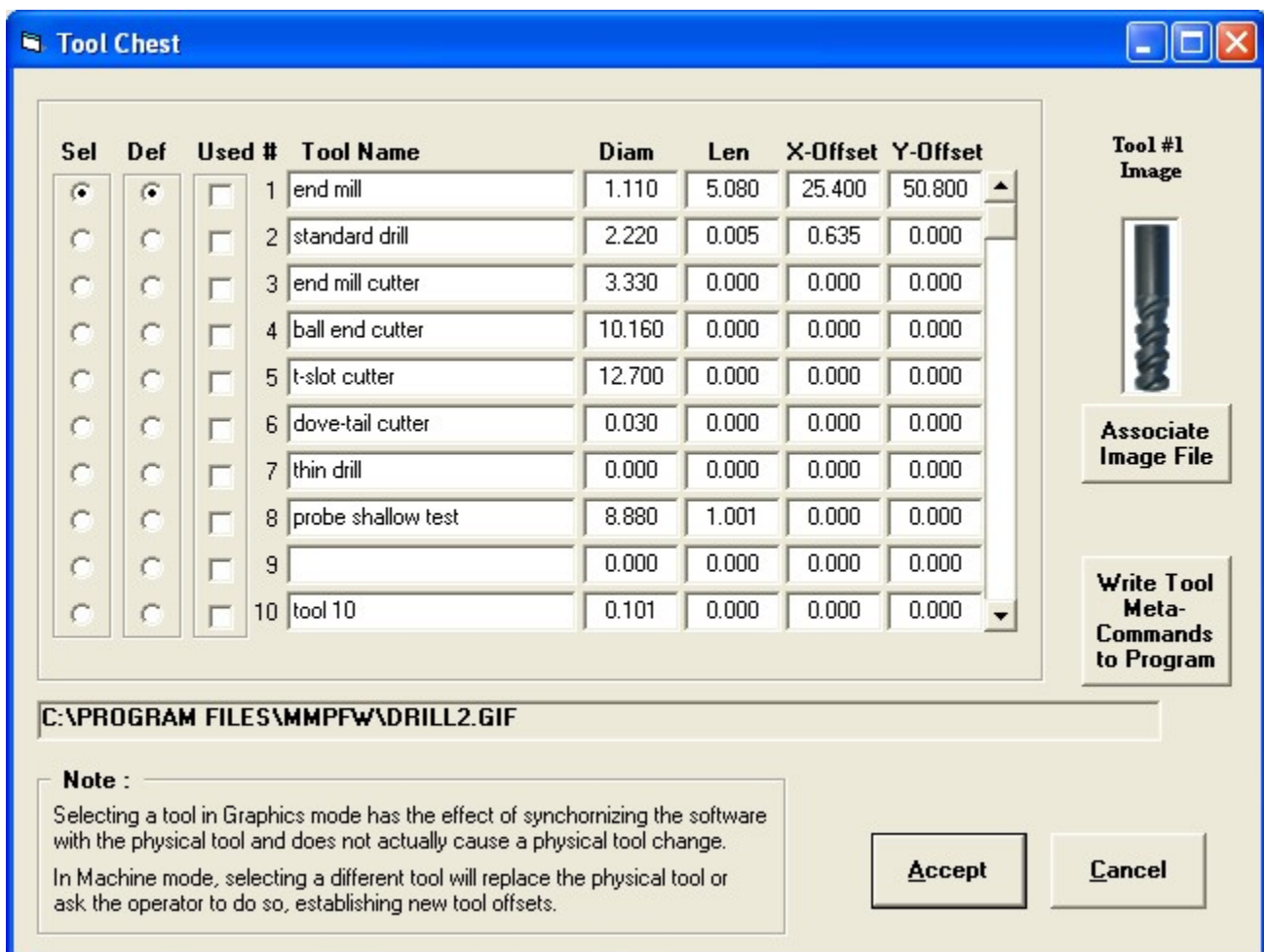
Opens the Tool Chest Window which allows the user to describe the length and radius of each tool. When changing tools, Z-axis compensation will occur automatically based on the difference in tool lengths. The 15th icon in the Toolbar also corresponds to this menu command.

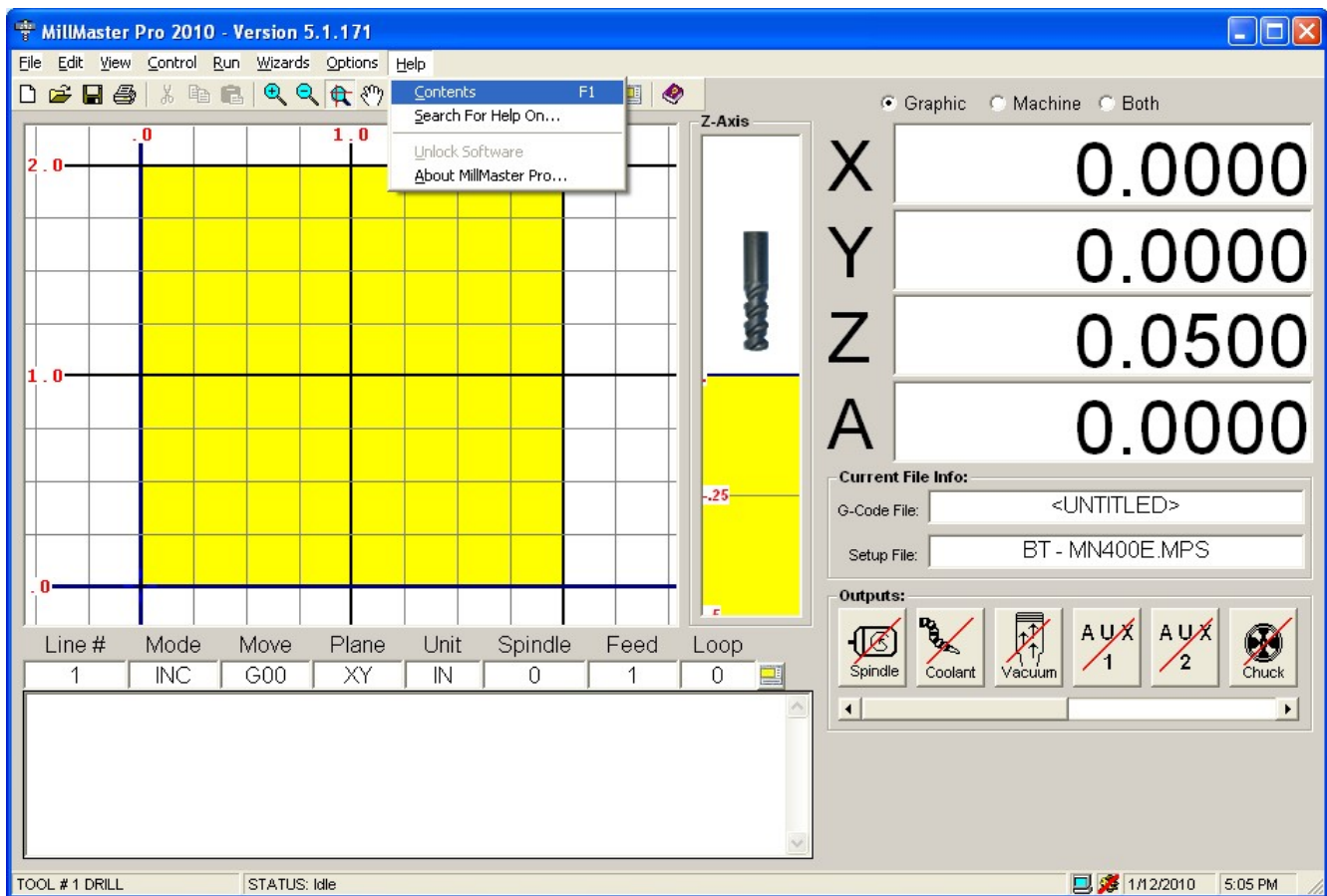
You can create a new tool by selecting an empty column (or overwriting an existing column if you prefer). You can use the scroll bar to find an empty tool. **Millmaster Pro** allows up to 99 tools. Now you will need to name your new tool. You can then input the diameter of the tool which is for graphic purposes only. The cut path of your program will vary depending on what size is entered here. Then you will need to enter the length of the tool. If this varies from the previous tool, your Z-axis will automatically make the height adjustment when the tool change occurs. X and Y-Offsets are more for a multihead spindle as they will adjust the start point of the tool upon a change to this new location. By clicking **Associate Image File** button, you can import a tool graphic from a location on your computer.

You can change the default tool (tool selected when starting **MillMaster Pro**) through this window by clicking the **Def** (Default) option button for desired tool.

You can also change the currently selected tool by clicking the **Sel** (Selected) option button for the desired tool.

Once you have finished entering all the data, you will need to press the **Accept** button to save your changes.





The **Help Menu** is used to display help and unlock the software.

The underlined letters are called **QuickSelect** keys. These allow access to a command only while the pull-down menu is displayed.

Help- Contents

This will display the table of contents from the .PDF Help file. The right-most icon in the Toolbar also corresponds to this menu command.

Help- Search For Help On

This will display the first page of the .PDF Help file where you can begin your search..

Help- Unlock Software

This will display the Unlock Software window, where, when purchased, allows you to enter an Unlock Code to fully maximize the use of the software. Having an unlocked copy allows you to utilize Tech Support for mechanical and software related problems. You will need to contact MicroKinetics to get your unlock code which is randomly generated based on your system ID.

****NOTE:** Once the software is unlocked, the Unlock Software option on the help menu will be disabled permanently.

Help- About

This will display the currently running version of **MillMaster Pro**, Instep Library, Company, and Copyright information.



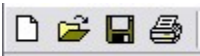
Toolbar and Icon related commands

This section explains the Toolbar and Icon related commands.

The graphic below is the Toolbar at program start-up.



The following are short descriptions of the above graphical commands contained on the toolbar (from left to right).

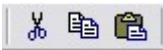


New - Clears the part program in memory and establishes the standard defaults for a new part program.

Open - Displays the Open File Dialog Window which displays all part programs in the current directory.

Save - Quickly saves the current part program to disk. The extension CNC is automatically assigned

Print - Displays the Print Dialog Window, which allows the user to choose Print Options, and utilize the built-in Print Preview Function.



****Notice** that the second set of icons in the top Toolbar graphic appear "grayed" out and in the above Toolbar they are active. These are the Edit Functions that are available only during CNC Edit Mode.

Cut - Cuts the currently highlighted text out of the page and onto the clipboard.

Copy - Copies the currently highlighted text onto the clipboard.

Paste - Inserts the current data from the clipboard to the current cursor location.



Zoom In - Zooms the current cutting window in by 25%

Zoom Out - Zooms the current cutting window out by 25%

Zoom - Zooms to the limits specified by the last mouse drag

Pan - Pans the current cutting window. When in **Pan** mode, the screen cursor will appear as a hand. To exit **Pan** mode, press the **Pan Icon**, or any of the three **Zoom** icons.



Start/Stop Execution - Executes the current part program in memory in continuous mode. After clicking the **GO icon** you will notice that it changes to a **STOP icon**. This allows you to **STOP** the program execution at any time. Either **Graphics Mode** or **Machine Mode** must be enabled to use this option. Alternatively, you can press any key on the keyboard to pause the program.

Pause Execution - Pauses program execution and displays **Step/Auto/Continue window**. Pressing any key during program execution also brings up this window. If you are in the middle of a G02 or G03 command, the program will not pause until after the command is completed.

Step Mode - Starts currently loaded program in **Step mode**, displaying the **Step/Auto/Continue window**.

Tool Chest - Opens the Tool Chest Window which allows the user to describe the length and radius of each tool. When changing tools, Z-axis compensation will occur automatically based on the difference in tool lengths.

Return Tool to Start - Returns the tool to the starting position.

Init as Start Position - Initiates the current position of the tool as the Start Position and resets the internal counters. The current location of the tool is assumed to be in the tool start position.

CNC Full-Screen Edit Mode - Toggles between full-screen CNC edit mode and graphics mode.

Note: This icon is also displayed above the small **CNC Edit** textbox, to the right of the status fields. This allows easy access to the Full-Screen Edit Mode when typing in the small **CNC Edit** textbox.

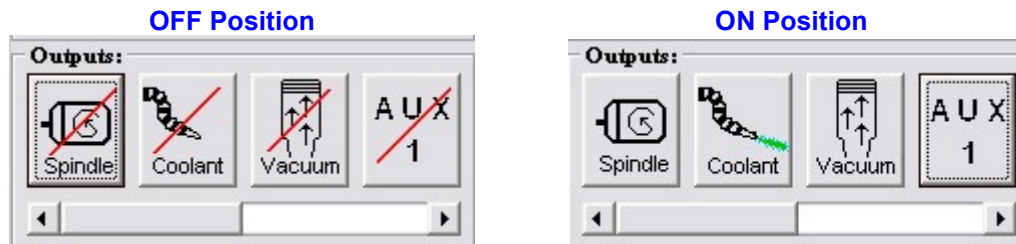


Help - Displays a window showing the contents of this help file.

ICON RELATED COMMANDS

Along with the Toolbar, there are other graphical/icon related command keys.

Below are pictures of the Outputs graphical menu in the **OFF** position and the **ON** position.



These graphical icons/command buttons control all outputs. The following are short descriptions of the above graphical commands contained on the Outputs graphical menu. (from left to right)

Spindle - Toggles **On/Off** the **spindle** of the milling machine.

Coolant - Toggles **On/Off** the flow of **coolant (if applicable)** for the milling machine. If the machine type is set to TORCH, this icon would be replaced with a torch icon and work in the same fashion.

Vacuum - Toggles **On/Off** the power to the **vacuum (if applicable)** of the milling machine.

AUX 1 - Toggles **On/Off** the **auxiliary 1 output (if applicable)**

AUX 2 - Toggles **On/Off** the **auxiliary 2 output (if applicable)**

AUX 3 - Toggles **On/Off** the **auxiliary 3 output (if applicable)**

AUX 4 - Toggles **On/Off** the **auxiliary 4 output (if applicable)**

Chuck - Toggles **Open/Close** the **pneumatic chuck (if applicable)** of the milling machine

****NOTE:** The Red Slash across the button indicates that Output is OFF.

****NOTE:** These outputs will function only while in machine mode.

Function Detail Reference

Preparatory Functions (G-Codes):

G00

This function is used to rapidly locate the tool to a new location.
This should only be used when **no contact** with the material is taking place.

EXAMPLE: G00 X0.1 Y-0.1

G01

This code is used to cut along the path of a straight line.

The first G01 in a program will require an **F** (Feedrate) parameter unless you want to use the feedrate already loaded into memory. You may change the feedrate in the same manner whenever you have a G01 command. F Codes are the **Feedrate** in inches/minute multiplied by 10 when in **Inch** mode. For example **F15** would equate to 1.5 inches/minute. In **Metric** mode F codes are in mm/minute.

For A-axis moves you can specify a **V** parameter. V is expressed in turret revolutions/minute.

EXAMPLES: G01 X1.2 Y1 F20 / Feedrate is 2.0 inches/min.
 G01 A90 V2 / Move turret 90 deg at 2 rev/min.

G02

This is used for a **two-axis** circular cutting move in a **clockwise** direction.

To perform a circular move the cutting tool is moved to the **starting position** before the **G02 code** is used. Following the **G02 code**, the end point of the arc is expressed in **two axes (X and Y)**. Next the distance from the **start of the arc to the circle center point** is given with an **I and/or J specification**. At least one of **I** and **J** must be specified. When an **I** or **J** is not specified, it is assigned to be **zero**.

You may use the **R** (Radius) parameter as an alternative to using the **I** and **J** parameters. **MillMaster** will compute up to 2 possible center points with the specified start and end points and the given radius. If the **R** parameter is positive, the center point that results in the shortest arc will be selected. If the **R** parameter is negative, the center point that results in the longest arc will be selected.

EXAMPLES: G02 X.5 Y2 J1 F10
 G02 X.5 Y2 R1 / Draw shortest arc of Radius 1
 G02 X.5 Y2 R-1 / Draw longest arc of Radius 1

G03

This code is used for **counterclockwise** cutting moves.
It is used in the same way as the **G02 command** described above.

EXAMPLES: G00 X3 Y2
 G03 X4 Y2.2 I.4375 F10
 G03 X4 Y2.2 R1 / Draw shortest arc of Radius 1
 G03 X4 Y2.2 R-1 / Draw longest arc of Radius 1

G04

This code causes the program to **pause** for a set period of time during execution.
A **G04** followed by a **D** and the number of seconds of the delay.

EXAMPLE: G04 D1.5 /this pauses for 1.5 seconds

G17

This code selects the **XY** plane for contouring. (**default**) After this command, all **G02** or **G03** calling moves will occur on the **XY axes**.

G18

This code selects the **XZ** plane for contouring.
After this command, all **G02** or **G03** calling moves will occur on the **XZ axes**.

G19

This code selects the **YZ** plane for contouring.
After this command, all **G02** or **G03** calling moves will occur on the **YZ axes**.

G25

This code causes program execution to continue at the first block of a named subroutine.
Certain modal states are saved automatically and restored upon return from subroutine.
These include **G90/91**, **G74/75**, and current program line. An **L** can be used here to call the subroutine a specified number of times.

EXAMPLES:

G25 #GROOVE L5

Runs the **GROOVE** subroutine in the current file five times.

G25 #SUBS.GROOVE

Runs the **GROOVE** subroutine in the file **SUBS.CNC** in the current directory.

G26

Conditional Branch: this code causes the program to jump to the specified label if the specified input is active (low voltage) or if the **parameter is true**. Valid input #s are I1 - I8.

Note: The I-Parameter is only available with the **OptiStep Plus** and **MN400** controllers.

EXAMPLE 1: G26 I1 #HOLESET

Will jump to #HOLESET when input 1 is active (low voltage).

If an I parameter is not used, the formula between the G26 and the label is evaluated. If the result is arithmetically non-zero or logically true, then the jump will occur. If the formula is evaluated as arithmetic zero or logical false, then the jump will not occur.

Note: To use logical comparisons, use the operands =, >, <, >=, <=, and enclose the entire expression in square brackets [].

EXAMPLE 2:

```
-- If the value of (parts) is greater than or equal to 1, goto #NextPart
G26 [(parts) >= 1] #NextPart
```

EXAMPLE 3:

```
G28 (parts) = 5
G90 /ESTABLISH ABSOLUTE
G75
G92 X0 Y0 Z0.5 /ESTABLISH START POINT
#loop
/program goes here
/
G28 (parts) = [(parts) - 1]
G26 (parts) #loop
M25 /RETURN TO START POINT
M2
```

G27

Unconditional Branch: this code causes the program to jump to the specified label.

EXAMPLE: G27 #HOLESET

Will jump to #HOLESET.

G28

This is used to set a variable to a desired value. The variable should be made up of letters and numbers only and must be enclosed in parentheses. Once the variable is defined, it may be used in any succeeding blocks in place of numerical constants.

You may also use mathematical formulas on the right side of the equal sign. If a formula is used, it must be surrounded by square brackets "[]" in order for the math evaluator to interpret it correctly. See the section "*Using the Math Formula Evaluation Feature*" for more information.

EXAMPLES: G28 (DEPTH)=.050 / Sets the DEPTH to .050
 G91 / Incremental
 G01 X-(DEPTH) F10 / Dig into material
 G00 X(DEPTH) / Rapid traverse out of material

 G28 (DEPTH) = [(DEPTH) * 2] / Set Depth to twice the current depth.

G54

Reset **Fixture Offset Mode**.

Fixture offsets are set to the default values of 0 for the **X**, **Y**, and **Z** axes.

G55 - G58

Set canned **Fixture Offset Mode**.

Fixture offsets are set to the values entered on the **Options | Fixture Offsets** screen.

For **G00**, **G01**, **G02** and **G03**, **X**, **Y**, and **Z** fixture offset values are added to the commanded values during program execution.

For **G79 - G89** Canned Cycle commands, only the **X**, and **Y** fixture offset values are applied.

G59

Set variable **Fixture Offset Mode** (up to 9 offsets are available). Use the **P** parameter (values 1-9) to specify which of the 9 Fixture Offset Modes to invoke.

EXAMPLES: G59 P1 / Resets Fixture Offset Mode. Same as G54
 G59 P2 / Set Fixture Offset # 2. Same as G55
 G59 P3 / Set Fixture Offset # 3. Same as G56
 G59 P4 / Set Fixture Offset # 4. Same as G57
 G59 P5 / Set Fixture Offset # 5. Same as G58
 G59 P6 / Set Fixture Offset # 6.
 G59 P7 / Set Fixture Offset # 7.
 G59 P8 / Set Fixture Offset # 8.
 G59 P9 / Set Fixture Offset # 9.

G60

Turn off **Continuous Contouring Mode**.

The checkbox on the **Machine Parameters** screen will be the initial setting of *Continuous Contouring* each time a program starts up.

G64

Turn on **Continuous Contouring Mode**.

G68

Collect part geometry for an irregular smoothed shape **Pocket** or **Cutout**.

A series of points is collected and saved to a separate **DXF** file.

To use this command, you must have a digital probe installed and wired to one of the Inputs (1-8). Before collecting geometry on a part, you should calibrate the probe using the **Wizards | Find Part Edges** screen.

If a **DXF** file is created, it can be imported by **MillMaster Pro** or **MultiCAM**. The format of the created **DXF** file is:

Geometry_G68_mmddyy_hhmmss_Depth_z-depth.DXF

where *mmddyy* is the current date, *hhmmss* is the current time and *depth* is the current Z-value.

Parameters for this command are imported from the **Wizards | Collect Part Geometry** screen. You may override any screen values with the use of optional parameters. These parameters are as follows:

- I** Input # (1-8) that the probe is connected to.
- D** Initial Seek Distance. The max distance to initially seek for a part edge.
- S** Max Seek Distance. The max distance to seek for a part edge after the initial seek.
- B** Backout Distance (Inches). Distance to backout after each contact.
- W** Distance between samples (Inches).
- C** Contact Speed (Inches/Min). Speed when contacting the part edge.
- A** Move-Away Speed (Inches/Min). Speed when moving away from the part edge.

EXAMPLE: **G00 x0 y0** / Go to "safe" X Y position.
 G00 z-.1 / Z-Depth to Collect Geometry
 G68 D1 / Collect part geometry for Pocket.
 / Max Initial Seek Dist = 1 inch, All other parameters from screen.
 / DXF File will be **Geometry_G68_mmddyy_hhmmss_Depth_-0_01.DXF**

G69

Collect part geometry for an irregular smoothed shape **Island** or **Boss**.

See the notes for **G68** for a detailed explanation of operation.

G70

This code sets the inch programming format. This overrides the unit of measure selected in the **Material Size/Setup** in the **Machine Parameters** screen.

EXAMPLE: G70

G71

This code sets the metric programming format. This overrides the unit of measure selected in the **Material Size/Setup** in the **Machine Parameters window**.

EXAMPLE: G71

G74

This code selects the **Single Quadrant Arc Programming Mode**. (I, J & K parameters must be positive.)

EXAMPLE: G74

G75

This code selects **Multiple Quadrant Arc Programming Mode**. (I, J & K parameters may be positive or negative.) **G75** is the default mode for **MillMaster Pro**.

EXAMPLE: G75

G77

Deep channel **CANNED CYCLE into the X-plane**. Mill a specified depth in a number of passes. The parameters include the ending **Y** and **Z** position of the hole, **X** depth, and either **Q** (# of passes) or **I** (X-depth per pass).

Note: The **X**-parameter is the *depth* of the channel. Thus it is assumed to be an *Incremental* value regardless of the current **G90/G91** setting. The other axes (**Y** and **Z**) are either *Incremental* or *Absolute* based on the current **G90/G91** setting.

Note: The tool edge **X** position should be 0.050" left or right of the material before command.

EXAMPLE 1: G00 X-0.05 Y0 /-- Position X & Y
 G01 Z-0.01 /-- Z-Depth
 G77 X0.1 Y1 Q4 /-- Mill channel to the right in 4 strokes.

Will mill a 0.1" channel to the right, and 1 inch long, in 4 passes at a Z-depth of -0.01".

EXAMPLE 2: G00 X1.05 Y0 /-- Position X & Y
 G01 Z-0.01 /-- Z-Depth
 G77 X-0.5 Y1 I.099 /-- Mill channel to the left at 0.099" per pass.

Will mill a 0.5" channel to the left, and 1 inch long, in 6 passes (5 passes at 0.099" and one finishing pass of 0.05") at a Z-depth of -0.01".

G78

Deep channel **CANNED CYCLE into the Y-plane**. Mill a specified depth in a number of passes. The parameters include the ending **X** and **Z** position of the channel, **Y** depth, and either **Q** (# of passes) or **J** (Y-depth per pass).

Note: The **Y**-parameter is the *depth* of the channel. Thus it is assumed to be an *Incremental* value regardless of the current **G90/G91** setting. The other axes (**X** and **Z**) are either *Incremental* or *Absolute* based on the current **G90/G91** setting.

Note: The tool edge **Y** position should be 0.050" in front of or behind the material before command.

EXAMPLE 1: G00 X0 Y1.05 /-- Position X & Y
 G01 Z-0.01 /-- Z-Depth
 G78 X2 Y-.075 Q3 /-- Mill channel towards the front of the part

Will mill a channel 0.075" towards the front of the part, 2 inches wide, in 3 passes of 0.025" each, at a Z-depth of -0.01".

EXAMPLE 2: G00 X0 Y-0.05 /-- Position X & Y
 G01 Z-0.01 /-- Z-Depth
 G78 X2 Y.075 J.024 /-- Mill channel towards the rear of the part

Will mill a 0.075" channel towards the rear of the part, 2 inches wide, in 4 passes (3 passes at 0.024" each, and one finishing pass of 0.003"), at a Z-depth of -0.01".

G79

Deep channel **CANNED CYCLE into the Z-plane**. Mill a specified depth in a number of passes. The parameters include the ending **X** and **Y** position of the channel, **Z** depth, and either **Q** (# of passes) or **K** (Z-depth per pass).

Note: The **Z**-parameter is the *depth* of the channel. Thus it is assumed to be an *Incremental* value regardless of the current **G90/G91** setting. The other axes (**X** and **Y**) are either *Incremental* or *Absolute* based on the current **G90/G91** setting.

An optional **D**-parameter (Downfeed speed) is available which will override the **F**-Feedrate for the Z-axis only.

Note: The tool **Z** position should be 0.050" above the material before this command.

EXAMPLE 1: G00 X0 Y0 Z0.05 / Position X, Y and Z Axes
 G79 X3 Z.2 Q4 F80 D5 / Mill a 3" channel to the right.

Will mill a channel 0.200" deep in 4 passes. The feedrate for the X and Y axes will be 8 inches/minute. The feedrate for the Z axis will be 0.5 inches/minute.

EXAMPLE 2: G00 X5 Y1 Z0.05 / Position X, Y and Z Axes
 G79 X2 Z.2 K.049 F80 / Mill a 3" channel to the left.

Will mill a channel 0.200" deep in 5 passes (4 passes at 0.049", and one finishing pass of 0.004"). The feedrate for the X and Y axes will be 8 inches/minute.

G80

Cancels a **CANNED CYCLE**. Use this command after each **CANNED CYCLE** function.

G81

Drill canned cycle. The parameters include **X & Y** position of the hole, **Z depth**, and **Feedrate**.

Z should be 0.050" above material before command.

EXAMPLE: G81 X.5 Y.5 Z.15 F5

Will drill a hole 0.150" deep. The Feedrate will be 0.5 inches per minute.

G82

Spot facing **CANNED CYCLE**. The parameters include **X & Y** position of the hole, **Z depth**, and **D** dwell in seconds.

Z should be 0.050" above material before command.

EXAMPLE: G82 X.5 Y.5 Z.2 D.5

Will spot face a hole 0.200" deep and wait ½ second.

G83

Deep hole drilling **CANNED CYCLE**. The parameters include **X & Y** position of the hole, **Z** final depth, and either **Q** (# of passes) or **K** (depth per pass).

Z should be 0.050" above material before command.

EXAMPLE: G83 X.5 Y.5 Z1 Q4
Will drill a hole 1.000" deep in 4 passes.

EXAMPLE: G83 X.5 Y.5 Z1 K.3
Will drill a hole 1.000" deep with (3) 0.300" strokes followed by one 0.1000" stroke.

G85

Boring **CANNED CYCLE**. The parameters include **X & Y** position of the bore, **Z** final depth. Upstroke is at feedrate.

Z should be 0.050" above material before command.

EXAMPLE: G85 X.5 Y.5 Z.2

G87

Drill **canned cycle** with chip breaking. The parameters include **X & Y** position of the hole, **Z** final depth, and either **Q #** of strokes or **K depth** per stroke. After each stroke, **Z** retracts to the depth of the previous stroke only to break the chip.

Z should be 0.050" above material before command.

EXAMPLE: G87 X.5 Y.5 Z1 Q4
 G87 X.5 Y.5 Z1 K.25

Will drill a hole 1.000" deep in 4 passes with chip breaking. Both command lines are equivalent.

G89

Boring with dwell **CANNED CYCLE**. The parameters include **X & Y** position of the bore, **Z** final depth.

Upstroke is at feedrate.

Z should be 0.050" above material before command.

EXAMPLE: G89 X.5 Y.5 Z.2 D.5

Will bore 0.200" deep and wait ½ second.

G90

This code sets **Absolute Programming Mode**. In **Absolute Programming Mode** all positions are expressed as they relate to a *single zero reference point*. This code is modal and will remain in effect in the program until changed by a **G91** code. This is the default programming mode.

EXAMPLE: G90

G91

This code sets **Incremental Programming Mode**. In **Incremental Programming Mode** all positions are given in terms of relative distance and direction from the current tool position. This code is modal and will remain in effect until changed by a **G90** code. The default mode is **Absolute Programming Mode** (G90).

EXAMPLE: G91

G92

This code sets the initial starting point of the cutting tool. This is usually used at the beginning of a program when setting the **starting position**, however, it may be used at any **time to step** and **repeat a set of commands** at a new location.

EXAMPLE: G92 X0 Z-0.05

G95

This code chains to another part program. Include this command at the **end** of a part program file and it will automatically load and continue machining the specified program. Any code placed after this command will **not** be executed.

EXAMPLE: G95 #TEST

Will load and machine **TEST.CNC** out of the current directory at the end of the current file.

Miscellaneous Functions (M-Codes):

M00

This command causes a **temporary stop**, by displaying a *Message Box*.

An optional message can be added to the M00 message box by using the comment character ("/"). Any text after the "/" will be displayed in the message box. Additionally, *user-defined variables*, or *system variables* may be displayed in the message by enclosing the variable name in parentheses "()". *Evaluated formulas* may also be displayed by enclosing them in square brackets "[]".

Operation **resumes** by pressing the **<OK>** button.

Pressing **<Cancel>** will terminate the current program.

EXAMPLES:

```
M00
M00          / This message will be displayed in the message box.
M00          / Debug Only: Variable TestX = (TestX). TextY = (TestY)
M00          / Next Year will be [Year[0] + 1]
```

The first example displays a simple M00 message box. The second example includes the text after the "/". The third example displays the values of the user-defined variables **TestX** and **TestY**. The fourth example displays an evaluated formula (next year), enclosed in square brackets.

M01

M01 is identical to **M00** with the exception that all *Coolant* and *Spindle* outputs (Outputs 1, 2, and 5), and any *VFD Spindle Controllers* (Teco 7300, or Fluxmaster 100) will be turned off while the message box is displayed, and restored to their original state upon completion of the command.

M02

This command causes an **end-of-program stop**.

All operations are terminated and the system returns to the **main menu**.

M03

This turns **<ON>** the spindle motor **clockwise (control output #1)**.

An optional S-parameter, for spindle speed, can be specified when using certain automatic spindles (Teco 7300 or Fluxmaster 100). Normally, **MillMaster Pro** will wait for the spindle to reach the desired speed before continuing to the next command. If a negative S-parameter is encountered, **MillMaster Pro** will not wait for the spindle to spool up before continuing.

EXAMPLES:

```
M03          / Turn spindle ON, clockwise
M03 S100     / Turn spindle ON, clockwise (100 RPM). Wait for spindle to spool up.
M03 S-100    / Turn spindle ON, clockwise (100 RPM). Do not wait for spindle to spool up.
```

M04

This turns <ON> the spindle motor **counterclockwise (control output #5)**.

An optional S-parameter can be added, as described for the M03 command.

M05

This turns <OFF> the spindle motor **(control output #1 and #5)**.

M06

This command selects the specified tool.

Note: You should first raise the tool to allow clearance for the tool change.

After the tool change, the new tool length will be incorporated into the next machine move. Therefore it is recommended that you then command the Z-axis back to its previous position before continuing.

EXAMPLE: Assume the Z-axis is at its start point of 0.05" (above the part surface).

```
G01 Z1.05 / Raise the tool 1 inch for clearance.  
M06 T2 / Select Tool #2. (You will be prompted to change the tool.)  
G01 Z0.05 / Lower the tool back to the original position.  
/ The new tool length will be incorporated into this move.
```

M08

This turns <ON> **coolant (control output #2)**.

M09

This turns <OFF> the **coolant (control output #2)**.

M10

This turns <ON> the **vacuum (control output #3)**

M11

This turns <OFF> the **vacuum (control output #3)**.

M12

This turns <ON> the **Auxiliary Output (control output #4)**.

This may be used to activate additional devices.

M13

This turns <OFF> the **Auxiliary Output (control output #4)**.

M17

This command causes a return from subroutine. Must be used at the end of every subroutine. When executed this command will **restore** the **G90/91**, **G74/75** status and continue from the line following the **G25** subroutine call that invoked this subroutine.

M25

This command homes the **Z axis**, the **Y axis**, then the **X axis**.

M29

This command homes to the limit of one axis (X, Y or Z axis) and returns the result.

EXAMPLE: M29 x (varname)

If in absolute programming mode then the X, Y or Z position is returned in *varname*. Otherwise the incremental movement (the distance from the last position to the limit) is returned in *varname*.

M39

This <CLOSES> the chuck (**control output #6**).

M40

This <OPENS> the chuck (**control output #6**).

M41

This turns <ON> the Auxiliary Output #3 (**control output #7**). This may be used to activate additional devices.

M42

This turns <OFF> the Auxiliary Output #3 (**control output #7**).

M43

This turns <ON> the Auxiliary Output #4 (**control output #8**). This may be used to activate additional devices.

M44

This turns <OFF> the Auxiliary Output #4 (**control output #8**).

M66

This allows for jogging during g-code execution without affecting the counters.

EXAMPLE: M66

Note: When **MillMaster** encounters an **M66** command in G-code, the Jog Tool window will appear on screen. When you jog, the counters will change. As soon as you close the Jog Tool window, the counters will be restored to the exact values they were before the **M66** command was encountered, but no motion will occur.

M94

This command sends a *literal* command to the **MN400** Controller. The command should be a native **MN400** command. MillMaster does not validate or parse the string before passing it to the **MN400**. The command may be embedded in double quotes but this is not required.

Note: The **M94** command is ignored if any other controller is selected, or if not in *Machine* mode.

EXAMPLES: M94 m400,0,-200 / Move X 400 steps, Y -200 steps
 M94 "m400,0,-200"
 M94 c2,0 / Turn Output #2 ON

M97

This command pauses **part program execution** until the specified input is **active** (low voltage). Valid input #s are I1 - I8.

Note: This feature is only available with the **OptiStep Plus** and **MN400** controllers.

EXAMPLE: M97 I2

Will wait for **input 2** to become active (low voltage) before continuing program execution.

M99

This command restarts execution of the **part program** from the **beginning**.

Function Quick Reference

Preparatory Functions (G-Codes)

G00	Rapid positioning move
G01	Linear cutting move
G02	Clockwise circular cutting move
G03	Counterclockwise circular cutting move
G04	Set dwell in seconds
G17	Selects the XY plane for contouring (default)
G18	Selects the XZ plane for contouring
G19	Selects the YZ plane for contouring
G25	Execute subroutine
G26	Conditional branch
G27	Unconditional branch
G28	Set system or user defined variable to value
G54	Reset Fixture Offset Mode
G55-58	Set canned Fixture Offset Mode
G59	Set variable Fixture Offset Mode
G60	Turn off Continuous Contouring mode
G64	Turn on Continuous Contouring mode
G68	Collect Part Geometry for Pocket (Cutout) **
G69	Collect Part Geometry for Island (Boss) **
G70	Set inch programming (default)
G71	Set metric programming
G74	Sets single quadrant arc programming mode
G75	Sets multiple quadrant arc programming mode (default)
G77	Canned cycle for milling a deep channel into the X-plane
G78	Canned cycle for milling a deep channel into the Y-plane
G79	Canned cycle for milling a deep channel into the Z-plane
G80	Cancel canned cycle
G81	Canned cycle for drilling a hole
G82	Canned cycle for spot facing
G83	Canned cycle for deep hole drilling
G85	Canned boring cycle
G87	Canned drilling cycle with chip break
G89	Canned boring cycle with dwell
G90	Set absolute programming mode (default)
G91	Set incremental programming mode
G92	Set current tool position counters to value
G95	Chain to next part program

** G68 and G69 commands require a digital probe.

Miscellaneous Functions (M-Codes)

M00	Temporary stop
M01	Temporary stop - Spindle & Coolant OFF
M02	End of program stop
M03	Spindle <ON> CW (output #1)
M04	Spindle <ON> CCW (output #5)
M05	Spindle <OFF> (output #1, output #5)
M06	Tool change
M08	Coolant <ON> (output #2)
M09	Coolant <OFF> (output #2)
M10	Vacuum <ON> (output #3)
M11	Vacuum <OFF> (output #3)
M12	Auxiliary Output <ON> (output #4)
M13	Auxiliary Output <OFF> (output #4)
M17	Return from subroutine
M25	Rapid traverse to home position
M29	Home to Axis Limit
M39	Chuck <CLOSE> (output #6)
M40	Chuck <OPEN> (output #6)
M41	Auxiliary Output #3 <ON> (output #7)
M42	Auxiliary Output #3 <OFF> (output #7)
M43	Auxiliary Output #4 <ON> (output #8)
M44	Auxiliary Output #4 <OFF> (output #8)
M66	Jogs without affecting counters
M94	Send literal command to MN400 Controller
M97	Wait for true input state then continue
M99	Restart part program from beginning

Tutorial

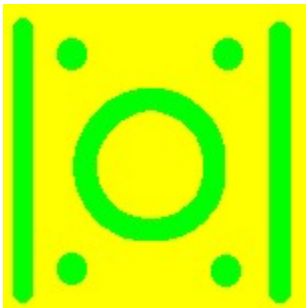
This section will show you how to write a part program using the built in CNC Editor and how to graphically simulate and manufacture the part using MillMaster Pro.

This tutorial will touch on basic machining functions such as linear and circular cutting moves, rapid traverse moves, and homing moves.

Before attempting to create and manufacture this part, your machine should be setup properly. Please refer to the *Configuration and Setup* section to assure proper Machine and software setup.

A drawing of the part that will be produced is shown in the graphic below.

The yellow represents the stock material, and the green represents the cuts in the stock material.



5.1 Writing a Part Program

From the Main screen press <CTRL> + E Simultaneously to start the text editor. You can easily create, edit, and save your part programs from this screen.

1. Enter the first three lines. They set the programming mode, starting position and arc mode of the machine. These two commands are usually included in most part programs. The comments are optional.

```
G90           /SET PROGRAMMING MODE
G92 X0 Y0Z 050 /SET STARTING POSITION
G75           /SET MULTIPLE QUAD ARC MODE
```

2. Enter the remaining lines.

```
M06 T3          /SELECTS TOOL #3 (END MILL)
G01 Z-.312 F30  /LOWERS THE CUTTER
Y3             /CUT THE FIRST FLANGE
G00 Z.05       /RAISE THE CUTTER
X3
G01 Z-.312
Y0            /CUT SECOND FLANGE
G00 Z.05
X1.5 Y2.25    /GO TO TOP OF CIRCLE
G01 Z-.312    /LOWER THE CUTTER
G02 J-.75     /CUT A CW CIRCLE
G00 Z.05
M06 T1        /CHANGE TO TOOL #1
              /SPOT FACE 4 HOLES
G82 X.375 Y.375 Z.2 K.1
Y2.625
X2.625
Y.375
G80          /CANCEL CANNED CYCLE
M06 T2
              /DRILL HOLES
G87 X.375 Y.375 Z.61 K.1
Y 2.625
X 2.625
Y.375
M25         /RETURN TO HOME
G74        /SET SINGLE QUAD ARC MODE
M02        /END OF PROGRAM
```

3. Choose **File-Save**. Type in **TUTORIAL** and press **<ENTER>**. This will save the part program in the current directory as **TUTORIAL.CNC**.

Some other helpful hints:

1. **Optional N-Sequence Numbers**. Each program line may be numbered with an xxxx sequence code. xxxx can be any number from 0001 to 9999.

2. **Align your code in columns**. This will make it easier for you to examine other portions of your program and to refer to it in the future.

3. **Add frequent comments**. Especially where the command itself does not explain the operation you are performing.

5.2 Machining a Part Program

Now that you've entered the part program, you will be instructed in this section on how to actually produce it. Before attempting to manufacture this part, your machine should be setup properly.

Please refer to the [Configuration and Setup](#) section to assure proper Machine and Software setup.

The following equipment is needed for this exercise.

- A vise or holdown set.
- A standard endmill cutting tool.
- A center drill and a twist drill
- 3" x 3" x 0.5" Stock (Aluminum, brass, plastic, or wax)

Return to the Main screen by pressing **<CTRL> + E** simultaneously again.

1. Choose [Options-Machine Parameters](#).

Check that the options and values are set as follows:

Units = Inch

Origin 0,0 = Left,Front,Top

Length = 3.000"

Width = 3.000"

Height = 0.500"

Machining Scale = 1

Save these parameters as default.

2. Choose [File-Open](#) with the mouse or **<ALT>-F O** from the keyboard. Select [TUTORIAL.CNC](#) and choose **[OK]**.

The program file is now loaded into **MillMaster Pro**.

3. Choose **Run-Auto Run** with the mouse or **<ALT>-R A** from the keyboard. This runs the simulation program. You should see the tool cutting the stock as the program lines scroll at the bottom of the screen. Press the **<SPACE- BAR>** after a few lines have executed to pause the simulation. In the dialog box choose the **[STEP]** option and then click **<CONTINUE>**. You are now in **STEP MODE**, click **<CONTINUE>** and the next program line will be executed.

This is a useful tool in troubleshooting a programming errors. Hit the **<SPACE- BAR>** again and choose **[AUTO]** and then click **<CONTINUE>**.

The part program should be free running again.

4. Choose **Run-Machine Mode** with the mouse or **<ALT>-R M** from the keyboard. Note that the status bar at the bottom of the screen indicates machine mode with an icon of a gear. If a slash appears across the gear, **Machine Mode** is disabled.

5. If you have **Limit Sensors** on your machine, choose **Control-Return Tool To Start**. This will automatically move your tool to the start position. If you do **NOT** have **Limit Sensors**, jog your tool to the start position.

6. Choose **Run-Auto Run** again. **MillMaster Pro** will now produce the part.

**** NOTE:** The **<Space-Bar>** or the **<ESC>** key will pause the machining process at any time.

If something starts to go wrong with the machining, use the space bar to pause or **PRESS THE EMERGENCY STOP BUTTON** on your DriveRack or other controller.

Using the Math Formula Evaluation Feature

Beginning with **Millmaster Pro** and **TurnMaster Pro** Versions 4.1.xxx, a math formula evaluation feature was added to the G-Code parser. This math evaluation feature combined with user-defined variable names is sometimes referred to as Parametric Programming.

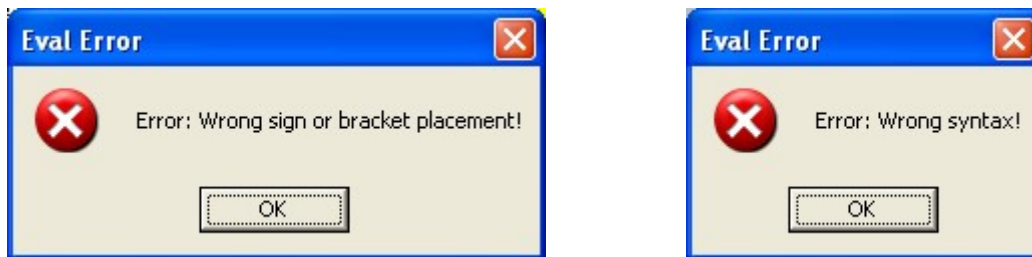
Parser:

Internally, the G-Code parser first converts any system variables or user-defined variables, in the G-Code line being parsed, to their equivalent numerical values.

The parser then looks for any square bracket pairs [...]. If found, the parser will attempt to evaluate the formula inside the brackets.

If the formula is a legal math expression, it will be converted internally to the equivalent numerical value and executed. For example, the G-Code line `G00 X[1+2]` would be executed as `G00 X3`.

If the formula is not valid, an error message is displayed, and the parser returns a value of **0** for the expression. For example:



Note: If the formula is not valid, the G-Code program continues to execute with a zero value returned from the expression. Be sure to test all G-Code programs in graphics only mode, and syntax checking turned on before running any programs in machine mode.

Syntax:

Multiple bracket pairs can exist on a single line. They may exist separated by parameters `G01 X[2*3] Y[1-.2]` or they may be nested to any level within the same formula `G00 X[1-[0.1*2]]`.

You must, however, make sure the entire expression is within the brackets. For example, `G00 X 1+[1-0.1]`, or `G00 X[1-0.5]+1` would both be invalid. This type of error would not be caught by the evaluation parser since the data inside the bracket pair is valid. In the examples above the program would continue to run, but with the wrong values for X.

Also, do not use the parenthesis characters () inside the expression. Parenthesis are reserved for system variables or user-defined variables. For example:

```
G28 (X1) = 0.1 /Set User Defined Variable X1 = 0.1
G92 X(X1) Y[1-(X1)] Z[0+[0+0.050]] /Establish Start Point
```

would establish the start point at X0.1, Y0.9 and Z0.5.

However the line:

```
G92 X(X1) Y[1-(0+0.1)] Z[0+[0+ .050]] /Establish Start Point
```

would be invalid since the value inside the parenthesis (**0+0.1**) is not a valid variable name.

Functions:

The following arithmetic functions are valid:

Arithmetic Operators:

+	Addition
-	Subtraction
*	Multiplication
\	Division (do not use the / char as it is reserved for comments)

abs[x]	Absolute Value of x
fix[x]	Integer portion of x (negative numbers round up)
int[x]	Integer portion of x (negative numbers round down)
rnd[x]	Random number between 0 and 1. X is the random # seed.
sgn[x]	Sign of x. Returns 1 if positive, -1 if negative, 0 if zero.

arcsin[]	Arcsine (in radians)
arccos[]	Arccosine
arcsec[]	Arcsecant
arccosec[]	Arccosecant
arccotan[]	Arccotangent
atn[]	Arctangent
sin[]	Sine
cos[]	Cosine
cosec[]	Cosecant
cotan[]	Cotangent
sec[]	Secant
tan[]	Tangent
exp[x]	Inverse Natural Logarithm or the base of a natural logarithm.
log[x]	Natural Logarithm of x
sqr[x]	Square Root of x

timer[0]	Seconds since midnight (dummy parameter [0] is required)
Year[0]	Current year
Month[0]	Current month
Day[0]	Current day

Logical Operators:

=	Equal To
>	Greater Than
<	Less Than
>=	Greater Than or Equal To
<=	Less Than or Equal To

Note: The logical operators are best used with the G26 (Conditional Jump) command. If True, the jump occurs, otherwise execution falls to the next line in the G-Code program.

However, these logical operators will return an arithmetic -1 if the condition is True, and 0 if the condition is false.

Also, be sure the entire logical expression is surrounded by square brackets.

For example:

```
[1 <= 2] would return -1 (True)
[99 = [33*3]] would return -1 (True)
[5 > [3*2]] would return 0 (False)
```

Examples:

<u>G-Code Line</u>	<u>Evaluates To</u>
G00 X[1+0.1]	G00 X1.1
G01 X[2*.1] Y[1\4]	G01 X0.2 Y0.25
G00 X[1+[1.1*2]]	G00 3.2
G92 X[cos[3.1416]] Y[sqr[4]]	G92 X-1 Y2
G28 (CurrentYear)=[YEAR[0]]	G28 (CurrentYear) = 2008
G28 (NextYear)=[(CurrentYear)+1]	G28 (NextYear)=2009
G01 [(NextYear)-(CurrentYear)]	G01 1
G28 (blnTest) = [(CurrX) >= 1]	Sets var (blnTest) to -1 (True) if (CurrX) >= 1, else (blnTest) is set to 0 (False).
G26 [(NumPasses) <= 0] #Done	Jump to #Done if (NumPasses) <= 0.

Sample Program:

```
-- Draw a variable number of "spokes" around a center point.
/ To change the program performance, simply change the variables
/ (NumberOfSpokes), (LengthOfSpoke), (XCenter), and (YCenter) at the
/ beginning of the program.

M03
G90 / Establish Absolute Positioning

G28 (LengthOfSpoke) = 1.5 / Length of Spoke
G28 (NumberOfSpokes) = 12 / Number of Spokes

G28 (XCenter) = 1.5 / Center point of spokes
G28 (YCenter) = 1.5

G28 (PI) = 3.1416 / Mathematical PI
G28 (SpokesDrawnSoFar) = 0 / Running count of spokes drawn

G00 x(XCenter) y(YCenter) / Establish center point of spokes
G01 z-.1 / Tool down

/---- Main loop. Each Iteration will draw one spoke -----
#DrawSpoke
```

```

/-- Add to the number of Spokes Drawn after this iteration.
G28 (SpokesDrawnSoFar) = [(SpokesDrawnSoFar)+1]

/ Compute angle in Radians for this spoke.
G28 (Angle) = [[2 * (PI) \ (NumberOfSpokes)] * (SpokesDrawnSoFar)]

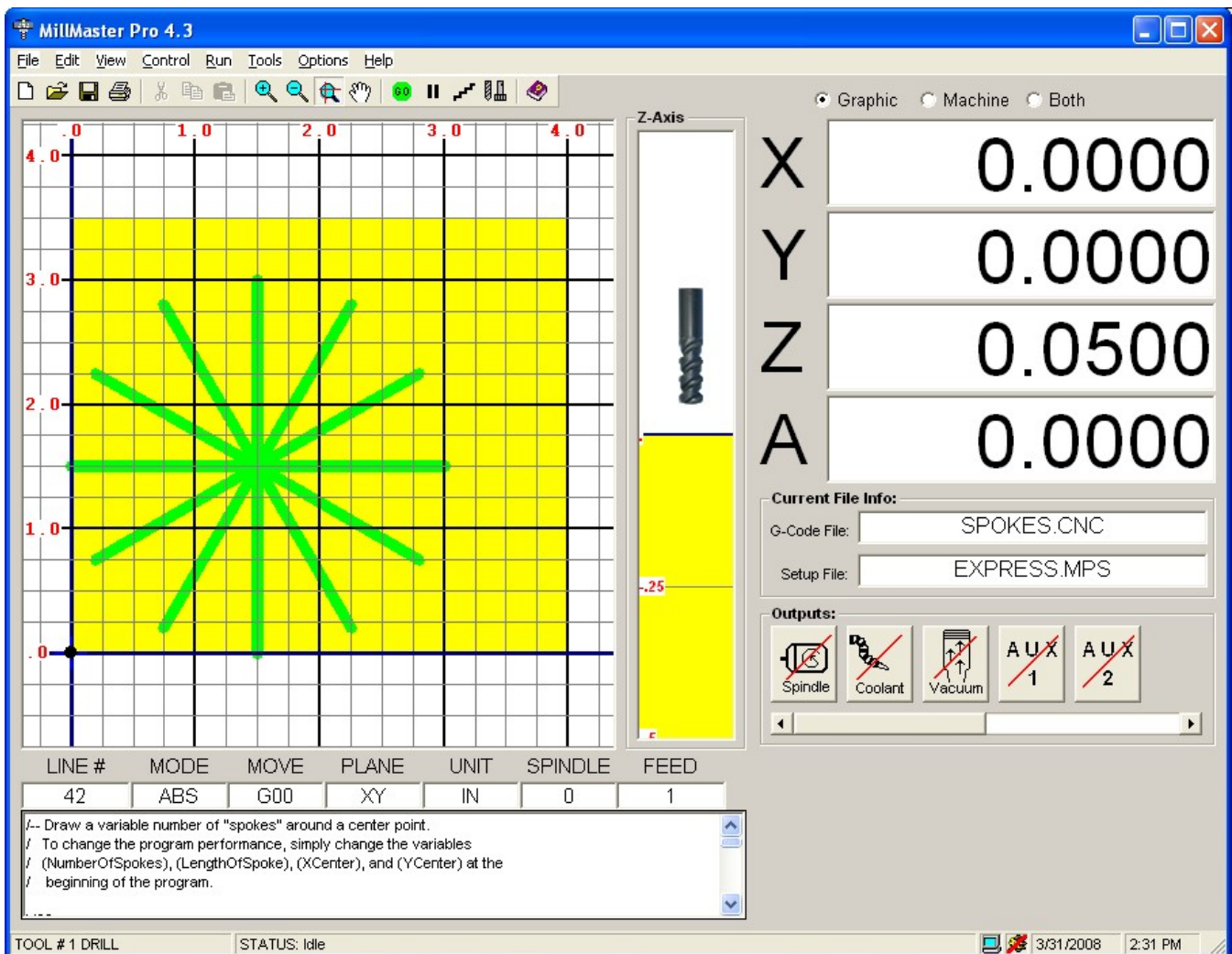
/ Draw the spoke.  x-coordinate is Cos(Angle), y-coordinate is Sin(Angle)
G1 x[[cos[(Angle)]*(LengthOfSpoke)]+(XCenter)]  y[[sin[(Angle)]*(LengthOfSpoke)]+(YCenter)]

g0 x(XCenter)  y(YCenter)          / Go back to center point

/-- If Spokes Drawn = Number of Spokes, then fall through,
/-- Else repeat loop by going to #DrawSpoke
G26 [(NumberOfSpokes)-(SpokesDrawnSoFar)] #DrawSpoke

/-- End of Main loop -----
G00 Z .050                          / Raise the cutter
G00 X0 Y0                            / Return to start point
M02                                  / End of program

```



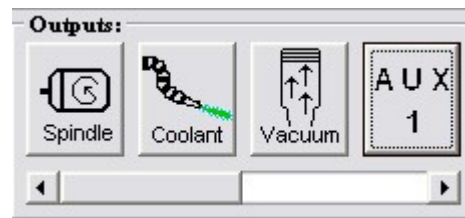
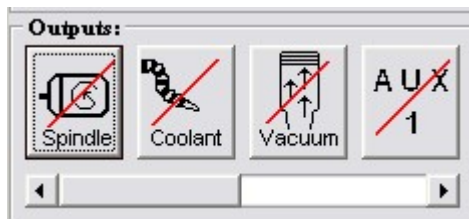
Customizing MillMaster Pro

This section will instruct you on how to customize **MillMaster Pro 2014**.

MillMaster Pro 2014 is easily customizable. You can change the **Output** icons and also add or change tools of your choice.

Customizing Output Icons:

The following graphic is the **Outputs** menu with the default icons used. By using the scroll bar located just below this window, AUX 1 and AUX 2 will appear in the Outputs menu. The scroll bar adjusts the view between the main and auxiliary outputs.



In the **MillMaster Pro** home directory there are 10 output icons in standard Windows Icon format (.ico)

out1-on.ico
out1-off.ico
out2-on.ico
out2-off.ico
out4-on.ico
out4-off.ico
aux1-on.ico
aux1-off.ico
aux2-on.ico
aux2-off.ico
aux3-on.ico
aux3-off.ico
aux4-on.ico
aux4-off.ico

Outputs 1, 2, 4 and Auxiliary 1, 2 are programmable. Output 3 is **NOT** programmable. This output is constant and is based on the machine type parameter specified in the configuration file. It will show coolant if the machine type = Mill and will show a torch if the machine type = torch.

**** NOTE: Changing the icon for output 3 will produce unwanted results and may cause unexpected machine operation.**

These standard Window Icon files must meet the following requirements:

Size: 48X48

Background: Transparent

Once you have created an Icon that meets the above requirements, you can save it in the **MillMaster Pro 2014** home directory with the name that corresponds to the output position you wish to change.

There are many Freeware / Shareware icon editor / creators on the internet today. Anyone that

runs under Windows 9.x should be capable of creating these icons easily.

Customizing Tool images

MillMaster Pro 2014 comes with a default set of tool images for you to choose from. However, you may require a special tool for your machining needs that we do not provide. It is very simple to create your own tool images for display in **MillMaster Pro**.

Below are a few samples of default tool images shipped with **MillMaster Pro**. These images are contained within the **MillMaster Pro** home directory, but can be placed in a separate images directory. Using 'Associate Tool Image' will allow you to specify its exact location.



These standard .GIF files must meet the following requirements:

Size:

Height: 88 pixels Width: 27 pixels

Background: Transparent (preserve transparency)

File Type: .GIF

You will notice that the tips of each tool are exactly aligned, whereas the tops are not. It is not required to utilize the entire 88 pixels of height for the tool, but it is required that the tip of the tool be placed exactly at the bottom of the image. This will assure EXACT Z depth in the Z viewport.

Making these images is similar to the icons, but requires a little more work. You can utilize a 3rd party image editor to create these files. There are also many shareware / freeware image editors on the internet that would be capable of these simple graphical operations.

MN400 Controller Information

The MN400 is an external (to the computer) motion control card that uses a serial or USB port to communicate with the PC. This allows for the use of the MN400 controller board on any computer that has a serial or USB port. With this controller, you can use **MillMaster Pro** with Windows ME/2000/XP and/or a laptop computer.

Within this version of **MillMaster Pro**, the MN400 can be activated and used to machine part programs, jog the machine, and do almost everything the OptiStep/QuickPhase and the Instep Library can do with the exception of utilizing Encoder feedback.

The MN400 is programmed to use COM ports at 9600 BPS. These parameters are adjustable.

**** NOTE:** If after connecting the MN400 to a serial or USB port on the back of your machine and choosing MN400 in the Machine Parameters, you are unable to jog, switch the cable to another available port and try again.

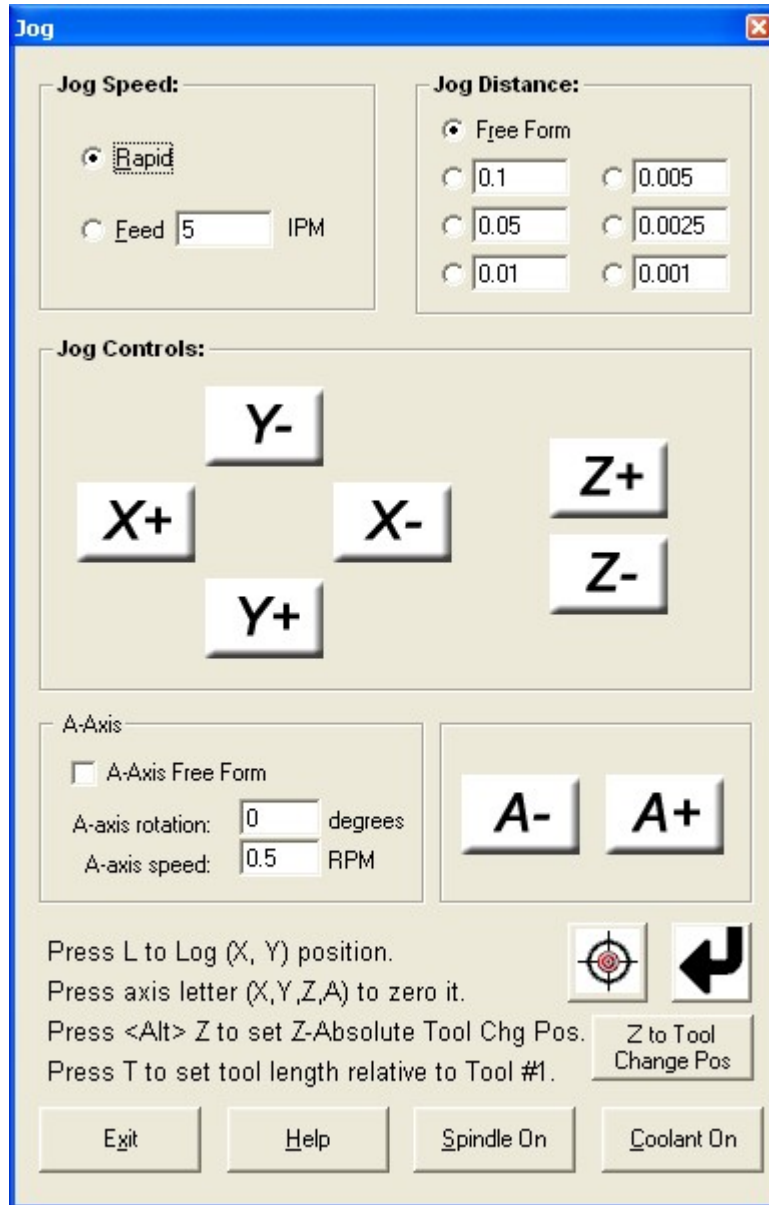
All commands work the same in **MillMaster Pro** when using the MN400 except the jogging feature. The differences are described below.

Although the operation of the MN400 is similar to the operation of the Instep Library, the MN400 does have some small differences. When running a program with the MN400, you do not experience the "Lockout" that occurs when using Instep and either the OptiStep or QuickPhase cards. The MN400 is a more advanced controller that accepts commands directly from the software through a serial or USB port, giving you more control over moves and program operation.

You may experience some delay when reading counters, when limits are hit or when running special operations such as Diagnostics, Home to Limits and Reprogram Limit Sensors. These delays are minor and are usually less than 1 second in length.

Jogging with the MN400 controller

With the MN400 motion controller board, you can now jog your machine with ease using the mouse. The graphic below shows the MN400 Jog window.



Each axis has its own + and - buttons organized in an easy to use fashion. The large white buttons contain the axis letter and the direction. To manually jog the machine, make sure Free Form is selected in the Jog Style frame and place your mouse cursor over one of the white buttons. Click and hold the mouse button down. The MN400 will jog the corresponding axis continuously until you release the mouse button.

You can also dispense Measured Moves by selecting Measured Moves in the Jog Style frame. Now, each click of one of the axis buttons will dispense the exact number of steps needed to move the Measured amount specified in the frame in the top left corner. To specify a custom distance, select 'Other' and enter the distance you wish to move in inches/mm. Now when you press any of the axis buttons, the machine will jog the distance specified.

The Jog Speeds frame allows the change of the jog speed based on either the MaxSpeed, defined under Machine Parameters, or by selecting feedrate and entering a value in inches per minute (ipm). For instance, to jog at a rate of 10 inches per minute, the user would select FeedRate and enter the number 10 in the field. Now when jogging any axis, the motors will move at 10 inches per minute.

Note that the A-axis has its own **Free Form** checkbox and **Rotation Amount** textbox.

For quick help while in the MN400 jog window, click the **Help** button at the bottom of the screen. This will display a message window giving you basic instructions for jogging with the MN400 controller.

Appendix A

Keyboard Scan Code Reference

The numbers to the right of the keys are the scan (or "make") codes. These codes are used for programming the active key for each tool jog direction.

Due to different Windows keyboard configurations and drivers based on region, keyboard type, and also typematic rate settings in BIOS, some keys may not work correctly. If you encounter a key that is not responding, try changing the key and try again.

F1 – 59	0 – 11	S – 31	Left ALT – 56	Keypad 1 – 79
F2 – 60	A – 30	T – 20	Left SHIFT – 42	Keypad 2 – 80
F3 – 61	B – 48	U – 22	Right SHIFT – 54	Keypad 3 – 81
F4 – 62	C – 46	V – 47	Caps Lock – 58	Keypad 4 – 75
F5 – 63	D – 32	W – 17	SCR LOCK – 70	Keypad 5 – 76
F6 – 64	E – 18	X – 45	NUM LOCK – 69	Keypad 6 – 77
F7 – 65	F – 33	Y – 21	DEL – 83	Keypad 7 – 71
F8 – 66	G – 34	Z – 44	INS – 82	Keypad 8 – 72
F9 – 67	H – 35	-- 12	TAB – 15	Keypad 9 – 73
F10 – 68	I – 23	= – 13	CTRL – 29	
1 – 2	J – 36	\ – 43	ESC – 1	
2 – 3	K – 37	[– 26	SpaceBar – 57	
3 – 4	L – 38] – 27	BKSPC – 14	
4 – 5	M – 50	; – 39	Keypad + – 78	
5 – 6	N – 49	“ – 40	Keypad - – 74	
6 – 7	O – 24	– 28	Keypad * – 55	
7 – 8	P – 25	, – 51		
8 – 9	Q – 16	. – 52		
9 – 10	R – 19	/ – 53		

Appendix B

Metacommand Reference

Metacommands are special commands found in comment lines that affect the **setup** when the **CNC program file** is loaded.

If the same **Metacommand** appears more than once in the same file, the last one found is used.

Below is a list of the available **Metacommands** along with the proper syntax:

data is a decimal number

string is a text string

flag is either **1** or **-1**. (**1** is used for **LEFT** or **BELOW** and **-1** is used for **RIGHT** or **ABOVE**).

Some of the commands can only be used with the constants listed next to them.

___ {Fixture}: **CHUCK** or **CLAMPS**

___ {Units}: **INCH** or **MM** - Inches or Millimeters.

___ {Material}: **string** - Textual description of the material to be milled.

___ {Material Shape}: **RECTANGULAR**, **RECTANGLE** or **ROUND**

___ {SLENGTH}: **data** - Length of part along the X-axis.

___ {SWIDTH}: **data** - Width of part along the Y-axis.

___ {SHEIGHT}: **data** - Height of part along the Z-axis.

___ {DIAMETER}: **data** - Diameter of **ROUND** part along the X and Y-axes.

___ {X Start}: **data** - Tool starting position on the X-axis.

___ {Y Start}: **data** - Tool starting position on the Y-axis.

___ {Z Start}: **data** - Tool starting position on the Z-axis.

___ {X Origin Zero}: **flag** - (1 = Origin is LEFT of the part, -1 = Origin is RIGHT of the part)

___ {Y Origin Zero}: **flag** - (1 = Origin is BELOW the part, -1 = Origin is ABOVE the part)

___ {Z Origin Zero}: **flag** - (1 = Material is BELOW the tool, -1 = Material is ABOVE of the tool)

___ {Tested} **YES** or **NO** - Has the program been tested successfully?

___ {Tool}: **#x**, **string**, **DIAMETER = data** - Set a Tool's name (**string**) and diameter to a numeric value (**data**).

___ {Defvar}: **string**, **data** - Set variable of name (**string**) to a numeric value (**data**).

You can use these commands in each part program to **automatically setup MillMaster Pro!**

Examples:

```
/* {Fixture}: CLAMPS
/* {Units}: INCH
/* {SLENGTH}: 2.0
/* {SWIDTH}: 2.0
/* {SHEIGHT}: 0.25
/* {X Start}: 0
/* {Y Start}: 0
/* {Z Start}: 0.05
/* {X Origin Zero}: 1
/* {Y Origin Zero}: 1
/* {Z Origin Zero}: -1
/* {TOOL}: #1, Drill, DIAMETER=0.1
/* {defvar}: depth,0.5
```

This set of **metacommands** will configure **MillMaster Pro** to **Inch mode** .

The material size will be set to a 2" x 2" square, 0.25" thick.

The origin will be placed on the top of the front left corner of the material and the tool **start position** will be set at the origin, **0.05" above the material**.

Tool #1 "Drill" is used and has a diameter of 0.1 inch.

The user-defined variable (*depth*) will be initially set to the value 0.5.

Appendix C

Wiring Diagram for External Cutting Speed Control Option

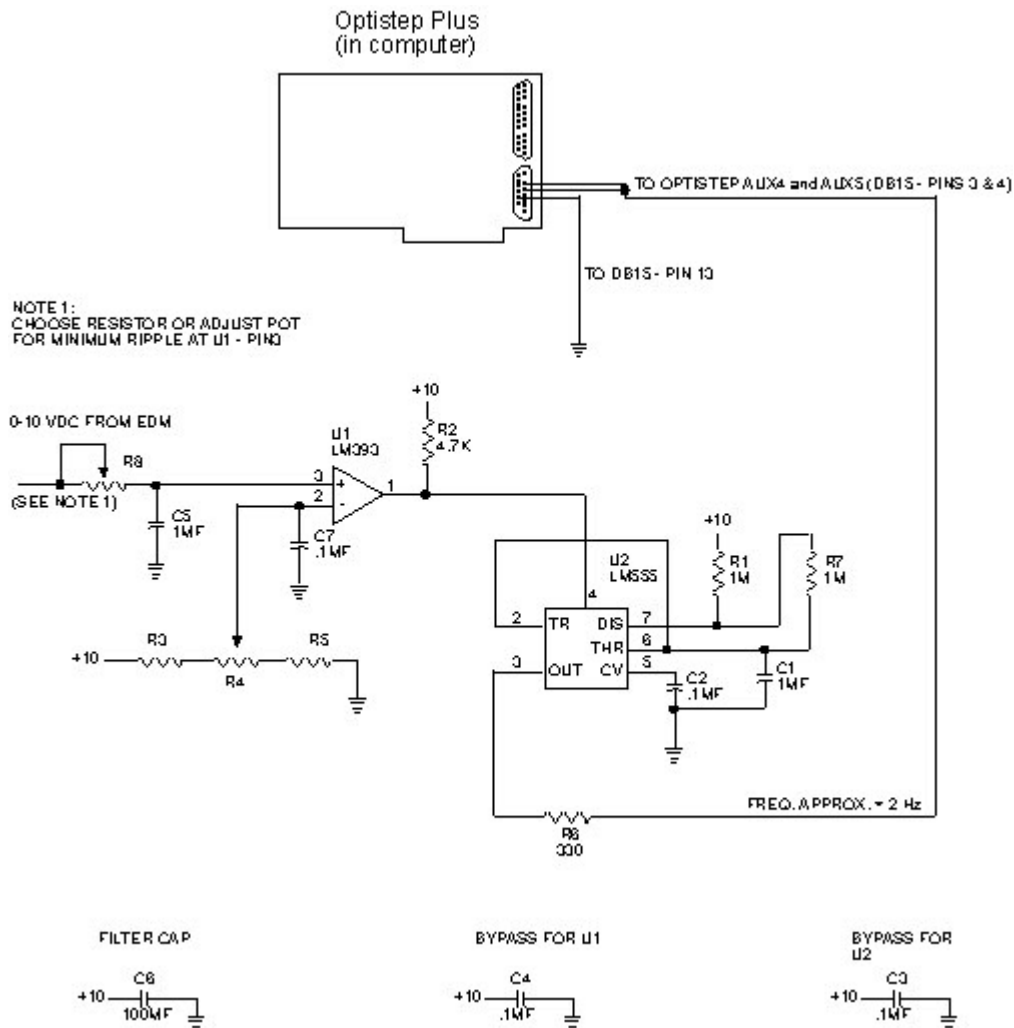
Connect an external pulse source, as in the example below, to the OptiStep Plus controller to use the External Cutting Speed Control option.

To enable this option, select External Cut Speed Control under Options-Position & Feed Speeds.

In this example, a wire EDM machine is supplying a DC voltage (0-10 Vdc) that changes proportionally with the amount of material it must remove before it moves the electrode.

If this voltage goes above the comparator threshold, it enables free-running oscillator (approx. 2 Hz) to move the stepping motors.

Example:



Appendix D

Using the configuration Files: "MMPRO.DEF" and the default ".MPS" file

The purpose of this **appendix** is to define each parameter in the default **.MPS** (*MillMaster Pro Setup*) file. The default **.MPS** file is specified in the **MMPRO.DEF** file. **MMPRO.DEF** and the default **.MPS** file are always located in the **MillMaster Pro** home directory. (i.e. **C:\Program Files\MillMaster Pro**)

To find out what your current default **.MPS** file is, use *Notepad™* or some other text-only editor to open **MMPRO.DEF**. The filename listed is your default **.MPS** file. Several sample default **.MPS** files are included in the installation package.

All configuration parameters can be viewed and edited from within **MillMaster Pro 2014** via the **Options | Machine Parameters** screen. However, should the need arise to manually edit these parameters, we have added a listing and detailed descriptions for selected parameters below.

Attributes from "**Express MN400.MPS**" are displayed below:

```
version = 0

card address = 592
card type = mn400

machine type = mill
max depth per pass = 1
maximum unramped speed = 800
maximum speed = 7000
acceleration rate = 10000
fast jog speed = 5000
slow jog speed = 500

external sync active = 0

x screw pitch = 5
x steps in rev = 800
y screw pitch = 5
y steps in rev = 800
z screw pitch = 5
z steps in rev = 800

x plus = 75
x minus = 77
y plus = 80
y minus = 72
z minus = 32
z plus = 22
a minus = 50
a plus = 49
tapsteps = 8
```

```
x origin = 1
y origin = 1
z origin = 1
x polarity = -1
y polarity = 1
z polarity = -1
x moving element = 0
y moving element = 0
z moving element = 1

part dim x = 2
part dim y = 2
part dim z = .75
circular part = False
machine scale = 1

x start pos = 0
y start pos = 0
z start pos = .05
a start pos = 0
x limit pos = -1
y limit pos = -1
z limit pos = .5
a limit pos = 0
x limit dir = -1
y limit dir = -1
z limit dir = 1
a limit dir = 0

units = i

x will test = 0
y will test = 0
z will test = 0
diag start speed = 700
diag speed increment = 300
diag acceleration = 10000
tolerance = .005
x dist unramped = .3
y dist unramped = .3
z dist unramped = .3
x dist ramped = 4
y dist ramped = 4
z dist ramped = 4

default tool = 1
```

```
x backlash = 0
y backlash = 0
z backlash = 0
a backlash = 0
x backlash direction = -1
y backlash direction = -1
z backlash direction = -1
a backlash direction = 0

encoder tolerance = 0
x encoder active = 0
y encoder active = 0
z encoder active = 0
a encoder active = 0
encoder auto correct = 0

jog tap steps = 8
feedhold mode = 1
multiplex = 4
optitrackker address = 0
spindle pulse mode = 0
axis #4 type = rotary
steps per deg rpm = -200
home speed pot steps = 0
tool change pause = 1
spindle speed pause = 1

oem name =
oem company =

use chords = False
chords per circle = 0

mn400 com port = 0
mn400 baud rate = 9600
mn400 continuous contouring = False

track external controls = False
spindle control com port = 1
serial spindle control = 0
teco 7300cv spindle = False
serial spindle baud rate = 9600
serial spindle comm settings = N
gear selection rpm = 1200

measured a-axis move = 0
draw dashed line when tool up = False
tool change clearance = 0
```



```
minor grid = 0.25
major grid = 1
auto grid = 1
show grid = 1
measured jog move1 = 0
measured jog move2 = 0
measured jog move3 = 0
measured jog move4 = 0
measured jog move5 = 0
measured jog move6 = 0
```

```
output number = 1
output name = output1
delay after on = 0
delay after off = 0
```

**** Each Output utilizes 4 lines for its particular attributes. There are 8 programmable Outputs. ****

```
tool number = 1
tool name = Drill
tool picture filename = c:\program files\mmpfw\drill2.gif
tool diameter = .125
tool length = 0
x tool offset = 0
y tool offset = 0
```

**** Each Tool utilizes 7 lines for its particular attributes. There are up to 99 tools available.**

**** NOTE:** The **configuration file** will continue past this point depending on the number of tools defined in the "Tool Chest".

Below is an explanation of each attribute and Valid Ranges these attributes will accept.

Version

This parameter specifies the current **Mill Master Pro 2014** version number.

Card Address

Valid Range: Specific Card Address (See card instructions for valid addressing)

This parameter specifies the card base address. (**Factory Address = 592**)

Card Type

Valid entries: MN400, QuickPhase, OptiStep, OptiStep plus

This parameter specifies which card type is to be used to operate the machine.

Machine Type

Valid entries: Mill or Torch

This parameter specifies which type of machine you will be using **MillMaster Pro** with.

Max Depth Per Pass

This parameter specifies the maximum amount of material removed in each pass.

Max. Unramped Speed

This parameter specifies the maximum instantaneous speed at which the machine can be moved without acceleration. This speed is in steps per second and can easily be converted from inches per minute by using the following formula:

$$\frac{\text{inches}}{\text{min}} = \left(\frac{\# \text{ steps}}{\text{sec}} \right) * \left(\frac{1}{\text{screw pitch}} \right) * \left(\frac{1}{\# \text{ steps/rev}} \right) * \left(\frac{60 \text{ sec}}{\text{min}} \right)$$

Max Speed

This parameter specifies the **maximum speed** in which the machine can be moved.

Acceleration Rate

This is the rate at which the speed will increase from the **Max UnRamped** speed to the **Max Ramped** speed.

Fast Jog Speed

This parameter specifies the speed at which the machine will **Jog in Fast Mode**. This parameter should be set to the same speed as the **Max Unramped** speed.

Slow Jog Speed

This parameter specifies the speed at which the machine will **Jog in Slow Mode**. Set the speed that can be used for fine positioning. A good value to start with is **200. (Adjust the value as needed)**.

External Sync. Active

This parameter specifies the signal for wire **EDM** machining. This parameter is normally set to **0**.

X, Y, Z Screw Pitch

This parameter specifies the **Teeth/Turns Per Inch (TPI)** of the lead screw.

X, Y, Z Steps In Rev

This parameter specifies the number of steps the motor needs to complete 1 full revolution.
(MicroKinetics Motors are typically: Fullstepping = 200 and Halfstepping = 400)

X Plus / X Minus

These parameters reflect the **Keyboard Scan Codes** that will control the **X Plus** and **X Minus** jog control keys.
(See Appendix A for Keyboard Scan Code information)

Y Plus / Y Minus

These parameters reflect the **Keyboard Scan Codes** that will control the **Y Plus** and **Y Minus** jog control keys.
(See Appendix A for Keyboard Scan Code information)

Z Plus / Z Minus

These parameters reflect the **Keyboard Scan Codes** that will control the **Z Plus** and **Z Minus** jog control keys.
(See Appendix A for Keyboard Scan Code information)

TapSteps

Valid Range: # of steps (Non-MN400 controllers only)

This parameter specifies the number of steps an axis will move when a **Jog Key** is **Tapped**. This value is not used for MN400 controllers.

X, Y, Z Origin

These parameters specify the **X,Y & Z** origin of the tool.

X Origin: 1 (Left), 2 (Center), -1 (Right)

Y Origin: 1 (Front), 2 (Center), -1 (Back)

Z Origin: 1 (Top), -1 (Bottom)

X, Y, Z Polarity

Valid Range: 1 (Normal) or -1 (Reverse)

These parameters specify the **Polarity** for each **Axis**. **NORMAL Axis Polarity** is defined as when the stepper motor rotates **CCW**, as viewed from the motor end. **REVERSE Axis Polarity** is defined as when the stepper motor rotates **CW**.

X, Y, Z Moving Element

Valid Range: 0 (Material) or 1 (Tool)

These parameters specify whether the material or the tool moves for each of the X, Y, and Z-axes.

X, Y, Z part dim

These parameters specify the **X,Y & Z** dimensions of the part.

Circular Part

Valid Range: True or False

Machine Scale

This parameter specifies the **Scaling Factor**. The scaling factor is used to scale the size of the actual part of an existing part program without changing any code.

Example: If a part program milled a part that is 2.0" square with the scale factor equal to 1, then with a scale factor equal to 2, the part would be milled to a 4.0" square.

X, Y, Z, A Start Pos

These parameters specify the **X, Y, Z & A** start position of the tool.

X, Y, Z, A Limit Position

These parameters specify the **X, Y, Z & A** limit positions for the machine. Your machine **must** be equipped with a **limit sensor package** to utilize this feature.

X, Y, Z, A Limit Direction

This parameter specifies the **X, Y, Z & A** direction in which the motor should **home/reposition** to.

Units

Valid Range: "i" for Inches OR "m" for Metric

This parameter specifies the unit of measure to be used for all software and machining operations..

X, Y, Z Will Test

Valid Range: 1 (test) or 0 (do not test)

These parameters specify which axes will be tested during the **Diagnostics Testing** of the machine.

Diag Start Speed

This parameter specifies the **Start Speed** (steps/sec) of the machine during the **Diagnostics Testing**.

Diag Speed Increment

This parameter specifies the **Increase In Speed** (steps/sec) per test.

Diag Speed Increment

This parameter specifies the **Tool Acceleration** (steps/sec²) used during diagnostic testing.

X, Y, Z Distance Unramped

This parameter specifies how far to travel for diagnostic testing of unramped moves.

X, Y, Z Distance Ramped

This parameter specifies how far to travel for diagnostic testing of ramped moves.

Tolerance

This parameter specifies the **Tolerance** for machining.

Example: A 1 inch square block with a tolerance of **+/- 0.005"** could be in the range of **0.995" to 1.005"** and still be within tolerance. **Total tolerance in this case is 0.010" - (0.005" for the plus and 0.005" for the minus)**

Default Tool

Valid Range: 0 - 99

This parameter specifies the default tool that **MillMaster Pro** will load at program startup or when restoring defaults.

X Backlash, Y Backlash, Z Backlash, and A Backlash

Valid Range: 0 to 0.9999

These parameters specify the amount of **backlash** in each of the **3 axes**. Enter the amount of **backlash** in the units specified in the **Options - Material Setup Parameters dialog window**.

X Backlash, Y Backlash, Z Backlash, and A Backlash Direction

Valid Range: 1 or -1

These parameters specify the **uncompensated** tool direction in each of the 3 axes. Enter **1** for **positive** or **-1** for **negative backlash setup direction**.

Encoder Tolerance

Valid Range: # of steps

This parameter specifies acceptable error in number of encoder pulses.

Example: If **Encoder Tolerance = 6** then **MillMaster Pro** would not flag a positioning error until the absolute position of the tool was 7 or more encoder pulses off.

If the encoder has **400 pulses per revolution** and the lead screw is **20 T.P.I.**, then this example would have a tolerance of **+/- 0.00075"**.

X Encoder, Y Encoder, Z Encoder, and A Encoder Active

Valid Range: YES or NO

These parameters turn encoder feedback **ON** or **OFF**. Specifying **YES** turns **ON** encoder feedback for that axis (**closed loop mode**) and a **NO** turns it **OFF** (**open loop mode**).

Encoder Auto Correct

Valid Range: YES or NO

This parameter specifies the action **MillMaster Pro** will take if a positioning error exists. Specifying **YES** will direct **MillMaster Pro** to auto correct the error and **NO** will prompt the user to [**Abort**] or [**Correct**] and **Continue**.

Jog Tap Steps

Valid Range: # of steps

This parameter specifies the number of steps an axis will move when a **Jog Key** is **Tapped**.
Example: If **Jog Tap Steps = 8** then **MillMaster Pro** will move any axis 8 steps when one of its respective **Jog Keys** is **Tapped**.

Feed Hold Mode

Valid Range: 1 (Yes) or 0 (No)

This parameter specifies the action **MillMaster Pro** will take when the shield is opened. Specifying **NO** will stop machining, turn **OFF** the **Spindle Motor** and prompt the user to [**Abort**] or [**Continue**]. Specifying **YES** will simply pause machining (**spindle motor remains on**).

Multiplex Resolution

Valid Range: 1, 2 or 4

This parameter adjusts the **Interpolation Resolution**. With **higher resolution**, there is less vibration on diagonal moves due to **Velocity Ripple**. A value of **2** is suitable for most applications.

OptiTracker address

Valid Range: Decimal base address

This parameter specifies the base address of the **OptiTracker™ card** in the computer. This parameter is ignored if set to 0.

Spindle Pulse Mode

Valid Range: 1 (Yes) or 0 (No)

Setting Spindle Pulse mode to **YES** causes M03 spindle on (CW) command to generate a pulse on output #1, M04 (CCW) causes a pulse on Output #5, and M05 (Spindle OFF) causes a pulse on output #4. These pulses are active low for a period of 1 second for M03 and M04. The period for the OFF command is 0.25 seconds.

Axis #4 type

Valid Range: ROTARY or SPINDLE

This parameter specifies what type the 4th axis is. (usually a rotary table)

Steps Per Deg RPM

Valid Range: 80 - 200

This parameter specifies the number of steps that the motor must make for the rotary table to rotate one full degree when using the 4th axis as a rotary axis.

Home Speed Pot Steps

Valid Range: Trial and Error

This parameter is used if the 4th axis is specified as **SPINDLE**. Home speed pot steps is the maximum number of steps it takes to move the speed control / knob motor from anywhere in the range back to zero.

Tool Change Pause

Valid Range: 1 (Yes) or 0 (No)

If "yes" then the program will pause for user input before changing tools.

Spindle Speed Pause

Valid Range: 1 (Yes) or 0 (No)

If "yes" then the program will pause for user input before changing spindle speed.

OEM Name, OEM Company

The machine owner's name and company.

Use Chords

Valid Range: True or False

This parameter specifies whether to use a series of straight lines (chords) to simulate an arc or circle.

Chords Per Circle

Valid Range: 4 - 7200

The number of chords used to simulate a 360° circle.

MN400 Com Port

Valid Range: 0 - 8

This parameter specifies COM Port for the MN400 (if used). A value of 0 is used for the MN400 with USB communications.

MN400 Baud Rate

This parameter specifies the Baud Rate for the MN400 using serial communications. This parameter is ignored if the **MN400 Com Port = 0**.

MN400 Continuous Contouring

This parameter specifies whether to use *Continuous Contouring* with the MN400. If used, commands will buffer in the MN400 and execute continuously after buffering.

Track External Controls

Valid Range: True or False

Should the display continuously query the MN400 and update the X, Y, Z, and A positions? These values may change if the user is jogging the MN400 via the external controls.

Spindle Control Com Port

Valid Range: 0 - 8

This parameter specifies COM Port for the external VFD spindle controller (if used).

Serial Spindle Control

Valid Range: 1 (Yes) or 0 (No)

Is a FluxMaster 100 external VFD spindle controller attached?

Teco 7300CV Spindle

Valid Range: True or False

Is a Teco 7300 external VFD spindle controller attached?

Serial Spindle Baud Rate

Valid Range: 4800 - 38400

The Baud Rate for an attached external VFD spindle controller.

Serial Spindle Comm Settings

The parity, # of data bits, and # stop bits for serial communications with the serial spindle. Normally this value will be "N,8,1"

Gear Selection RPM

The Gear RPM for the serial spindle, as specified by the manufacturer.

Measured A-axis Move

This parameter specifies how far to move the A-axis in the *MN400 Jog Screen* when a measured move is selected.

Draw Dashed Line When Tool Up

Valid Range: True or False

This parameter specifies whether MillMaster should draw a dashed line on the Main Display during moves when the tool is above the surface of the part. Normally, nothing is drawn if the tool is up.

Tool Change Clearance

This parameter specifies any extra clearance desired during tool changes

Minor Grid

Valid Range: 0.01 - 200

The distance between minor grid lines on the Main Display. This values are only used if **Auto Grid** is turned OFF.

Major Grid

Valid Range: 0.05 - 1000

The distance between major grid lines on the Main Display. This values are only used if **Auto Grid** is turned OFF.

Auto Grid

Valid Range: 1 (Use Auto Grid) or 0 (Do not use)

If Auto Grid is on, each time the main screen is redrawn, MillMaster automatically determines the major and minor grid lines. If **Auto Grid** is ON, the values for **Major Grid** and **Minor Grid** are ignored.

Show Grid

Valid Range: 1 (show grid) or 0 (do not show grid)

Measured Jog Move1 - 6

These parameters specify the values loaded into the MN400 Jog Screen for the 6 selectable measured moves under the *Jog Distance* panel.

Output Number

Valid Range: 1 - 8

This parameter specifies the output number.

Output Name

This parameter is a text description of the output. For example, on a torch machine a sample output name might be 'Preheat'

Delay after on

Valid Range: an integer specifying the delay in seconds

This parameter specifies the amount of time **MillMaster** should delay after turning on an output, before proceeding. You may specify, in seconds, 3 places to the left and one to the right. (###.#)

Delay after off

Valid Range: an integer specifying the delay in seconds

This parameter specifies the amount of time **MillMaster** should delay after turning off an output, before proceeding. You may specify, in seconds, 3 places to the left and one to the right. (###.#)

Note: The preceding 4 fields can be repeated for each of up to 8 outputs.

Tool Number

Valid Range: 1-99

This parameter specifies the tool number in the index.

Tool Name

This parameter specifies the unique name for the tool. This parameter is displayed in the Tool Geometry window when the tool is selected.

Tool Picture Filename

This parameter specifies the full path and filename for the image associated with the tool.

Example: C:\images\drill.gif

Please refer to Customizing **MillMaster Pro** for further information on image association.

Tool Diameter

This parameter specifies the diameter of the tool. This parameter will effect the size of the tool top in the XY viewport.

Tool Length

This parameter specifies the length of the tool. This parameter will effect movement of the Z-Axis during tool changes.

X Tool Offset

This parameter specifies the tool's X offset.

Y Tool Offset

This parameter specifies the tool's Y offset

Note: The preceding 7 fields can be repeated for each of up to 99 tools.

Appendix E

Using Machine Parameters as Read-Only Variables

Selected values entered in the **Machine Parameters** screen are available for use in your G-Code programs.

These values are similar to user-defined variables, in that they must be used with surrounding parenthesis. As with user-defined variables, they may be used in arithmetic expressions.

These values are **read-only** and cannot be set using a **G28** command.

For example:

```
G01 x(MM_PARTLENGTH) y(MM_PARTWIDTH) /-- back right of part
G01 z[1000 \ [(MM_ZSCREWPITCH) * (MM_ZSTEPSINREV) ] ] /-- move 1000 steps
```

The following values are available:

<u>Variable Name</u>	<u>Value</u>
MM_PARTLENGTH	X-dimension of the work area
MM_PARTWIDTH	Y-dimension of the work area
MM_PARTHEIGHT	Z-dimension of the work area
MM_CONTROLLER	1 - MN400, 2 - OPTISTEP PLUS, 3 - QUICKPHASE
MM_UNITS	1 - Inches (English) , 2 - Millimeters (Metric)
MM_MACHINETYPE	1 - Mill, 2 - Torch
MM_MACHININGSCALE	Machining Scale (Default = 1)
MM_TRACKJOG	Track External Control Panel ? (Yes = 1, No = 0)
MM_CONTCONTOURING	Use Continuous Contouring ? (Yes = 1, No = 0)
MM_MAXUNRAMPEDSPEED	Maximum initial axis speed (steps/sec)
MM_MAXRAMPEDSPEED	Maximum final axis speed (steps/sec)
MM_ACCELERATION	Axis acceleration (steps/sec ²)
MM_XGPOS	Current graphics X-position.
MM_YGPOS	Current graphics Y-position.
MM_ZGPOS	Current graphics Z-position.
MM_XMPOS	Current machine X-position.
MM_YMPOS	Current machine Y-position.
MM_ZMPOS	Current machine Z-position.
MM_XCPOS	Current counter X-position.
MM_YCPOS	Current counter Y-position.
MM_ZCPOS	Current counter Z-position.

MM_XSCREWPITCH	X-axis screw pitch (revolutions per inch)
MM_YSCREWPITCH	Y-axis screw pitch (revolutions per inch)
MM_ZSCREWPITCH	Z-axis screw pitch (revolutions per inch)
MM_XSTEPXINREV	X-axis steps per revolution
MM_YSTEPXINREV	Y-axis steps per revolution
MM_ZSTEPXINREV	Z-axis steps per revolution
MM_XPOLARITY	X-axis polarity (Normal = 1, Reverse = -1)
MM_YPOLARITY	Y-axis polarity (Normal = 1, Reverse = -1)
MM_ZPOLARITY	Z-axis polarity (Normal = 1, Reverse = -1)
MM_XORIGINZERO	X-origin (Left = 1, Right = -1, Center = 2)
MM_YORIGINZERO	Y-origin (Front = 1, Back = -1, Center = 2)
MM_ZORIGINZERO	Z-origin (Top = 1, Bottom = -1)
MM_XBACKLASH	X-backlash value (Inches or MM)
MM_YBACKLASH	Y-backlash value (Inches or MM)
MM_ZBACKLASH	Z-backlash value (Inches or MM)
MM_ABACKLASH	A-backlash value (degrees)
MM_XBACKLASHDIR	X-backlash direction (Positive = 1, Negative = -1)
MM_YBACKLASHDIR	Y-backlash direction (Positive = 1, Negative = -1)
MM_ZBACKLASHDIR	Z-backlash direction (Positive = 1, Negative = -1)
MM_ABACKLASHDIR	A-backlash direction (Positive = 1, Negative = -1)
MM_SERIALSPINDLE	Spindle (None = 0, Teco7300 = 1, Fluxmaster100 = 2)
MM_SPINDLECOMPORT	Spindle Serial COM Port
MM_SPINDLEBAUD	Spindle Serial Baud Rate
MM_SPINDLEACCDECTIME	Spindle Acceleration/Deceleration Time (seconds)
MM_SPINDLEGEARRPM	Spindle Gear RPM
MM_SPINDLERESETSTEPS	Spindle Speed Control Reset (steps)
MM_AXIS4TYPE	Axis 4 (None = 0, Rotary = 1, Spindle = 2)
MM_AXIS4RPM	Axis 4 Steps Per Degree (Rotary) or RPM (Spindle)
MM_FEEDHOLD	Feedhold Mode ? (Yes = 1, No = 0)
MM_PULSEMODE	Spindle Pulse Mode ? (Yes = 1, No = 0)
MM_TOOLCHANGEPAUSE	Tool Change Pause ? (Yes = 1, No = 0)
MM_SPINDLESPEEDPAUSE	Tool Change Pause ? (Yes = 1, No = 0)
MM_DASHEDLINEWHENTOOLUP	Draw Dashed Line When Tool Up ? (Yes = 1, No = 0)
MM_USECHORDS	Use Chords for Arcs ? (Yes = 1, No = 0)
MM_CHORDSPERCIRCLE	Chords per Circle (4-720)
MM_MULTIPLEXRESOLUTION	MultiPlex Resolution
MM_TOOLCHGUSEABSOLUTEZ	Absolute Z when Changing Tools ? (Yes = 1, No = 0)
MM_TOOLCHGCLEARANCE	Z-Clearance or Absolute Z when Changing Tools
MM_EXTRACOMMDELAY	Extra Delay During Machine Move

MM_TOOLNUM	Active Tool # (1-99)
MM_TOOLDIAMETER	Tool Diameter for the Currently Selected Tool
MM_TOOLLENGTH	Tool Length for the Currently Selected Tool
MM_TOOLXOFFSET	Tool X-Offset for the Currently Selected Tool
MM_TOOLYOFFSET	Tool Y-Offset for the Currently Selected Tool
MM_TOOLDIAMETER_{xx}	Tool Diameter for Tool # <i>xx</i> (where $1 \leq xx \leq 99$)
MM_TOOLLENGTH_{xx}	Tool Length for Tool # <i>xx</i>
MM_TOOLXOFFSET_{xx}	Tool X-Offset for Tool # <i>xx</i>
MM_TOOLYOFFSET_{xx}	Tool Y-Offset for Tool # <i>xx</i>