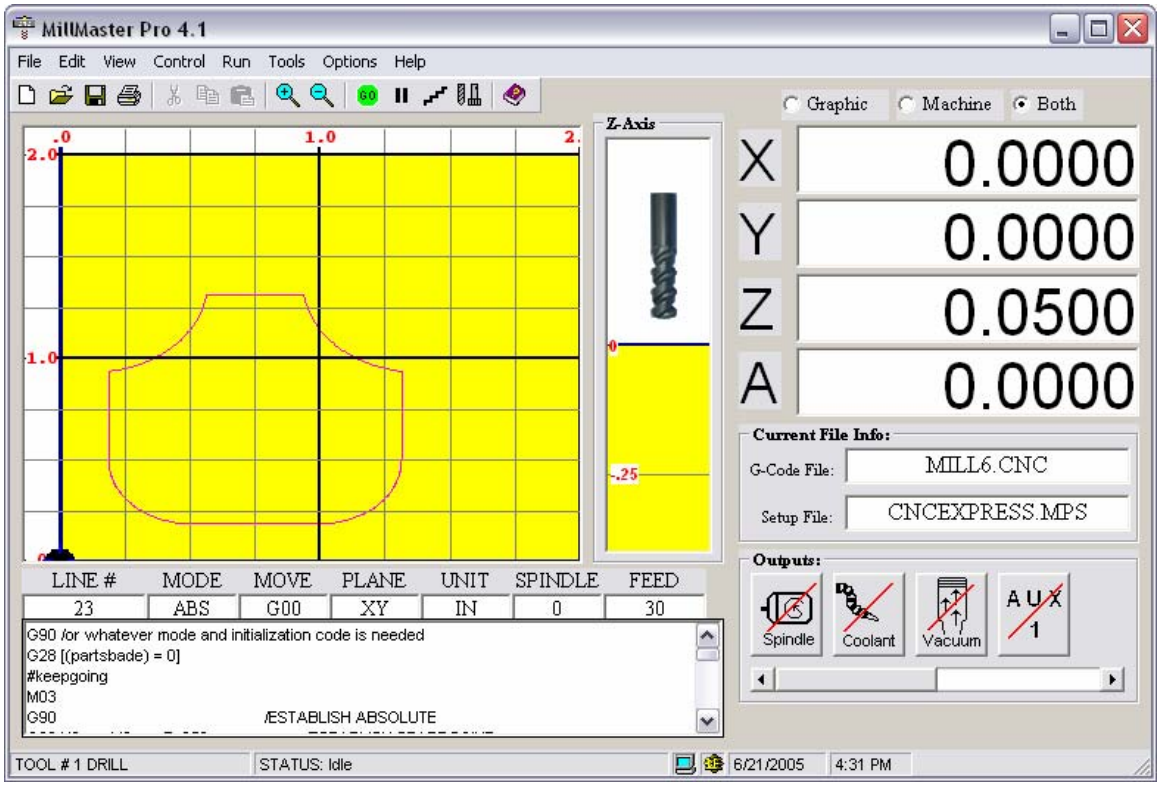
MillMaster Pro for Windows

Milling Machine Control and CNC Program Verification
For Microsoft Windows ™ Based PC's
V4.2



Contents Quick Jump

Minimum System Requirements

- **An IBM PC or 100% compatible**
- **Windows 98SE with Internet Explorer 5.5 or higher installed if running with OptiStep Control Card, 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 a free serial port**
- **15-20 megs of free hard drive space**
- **Minimum 64 megs RAM (128 megs recommended)**
- **Minimum 266mhz Processor (300mhz or higher recommended)**

We do have turnkey systems available which include a CPU, monitor, keyboard, mouse, OEM software, all cables, and customer configuration of MicroKinetics software. If you are having trouble locating a computer for use with your OptiStep or QuickPhase control card, please call and inquire about our systems.

Installing MillMaster Pro for Windows

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

From the Windows Desktop:

1) Place the media containing MillMaster Pro for Windows into a drive

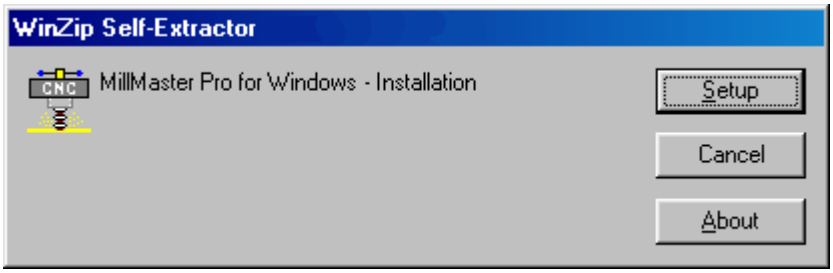
2) Click **'START'; 'RUN'; 'BROWSE'**

3) Select the drive you inserted the media into, then select 'mmpfwv1.exe' and Click **'OK'**

This starts the Auto-Install process. You will see the window below appear on your screen:

When this Dialog box appears on your desktop, click **'OK'.**

Next you will see the window below appear on your screen:



When this Dialog box appears on your desktop, click **'SETUP'.** This will unzip all of the MillMaster Pro files into a temporary directory and start the Setup program.

MillMaster Pro will now be installed to your hard drive with the Standard Windows setup program. The setup screen will reflect the version you are installing. The graphic below shows v1.0, but is for demonstration purposes only.

The next screen you will see has a Bluish/Black background. **(Shown below)**

Follow the prompts until Setup is complete.

**** *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 for Windows

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 an identical **MillMaster Pro** Icon onto your desktop for quick, easy access.

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

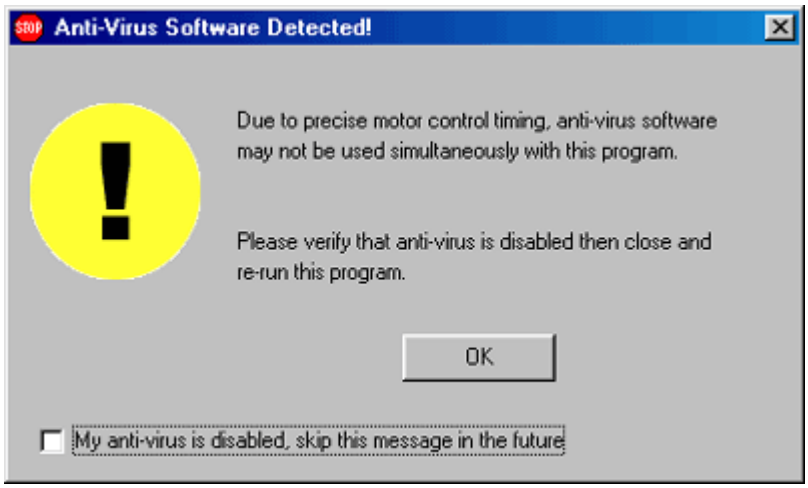
You will notice the evaluation message displayed on this screen until you 'Unlock' the software. This can be done by using the File Menu Bar at the top of the **MillMaster Pro** Main Screen; clicking the word HELP, this will drop down a small sub-menu; click the option that says 'Unlock Software'

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 for Windows** 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 appear:



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.

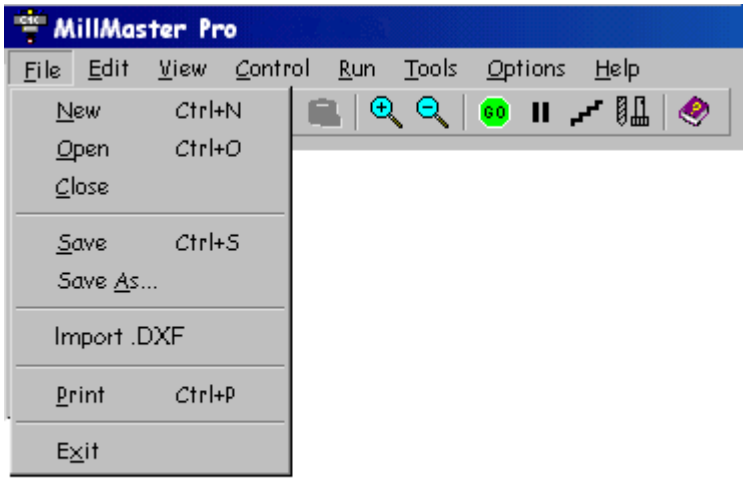
MillMaster Pro only detects Norton Anti-Virus or McAfee Virus Shield. If you are running another brand of Anti-Virus, this message will not be displayed. You will need to disable ANY virus protection, regardless of manufacturer before machining.

Getting Started with MillMaster Pro for Windows

Thank You for purchasing **MillMaster Pro for Windows 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.

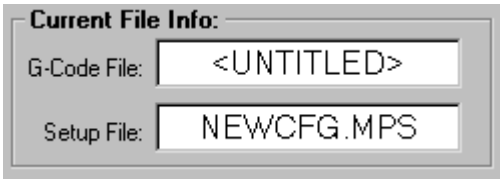
The following graphics show the Machine Parameter Tabs. The graphic on top demonstrates what the tab looks like when the **Mouse-over Hot Key** is **active**.

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** functions are also accessed with a **Mouse QuickSelect** click by left clicking and dragging the mouse over the area to zoom in on in the cutting window. Single clicking the right mouse button in this same window will bring your view back to the previous zoom level. Notice that when you move your mouse over the XY cutting window, the mouse pointer turns to a crosshair for precision zooming.

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 **Fixturing Method (chuck)**, 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.

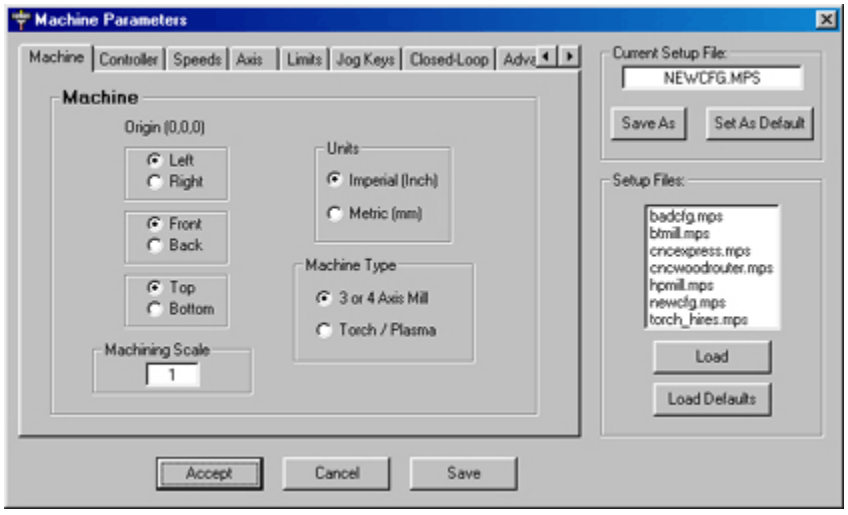
5. Enter the **Stock Width** in the specified **Units**. Enter 3.0.

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**.

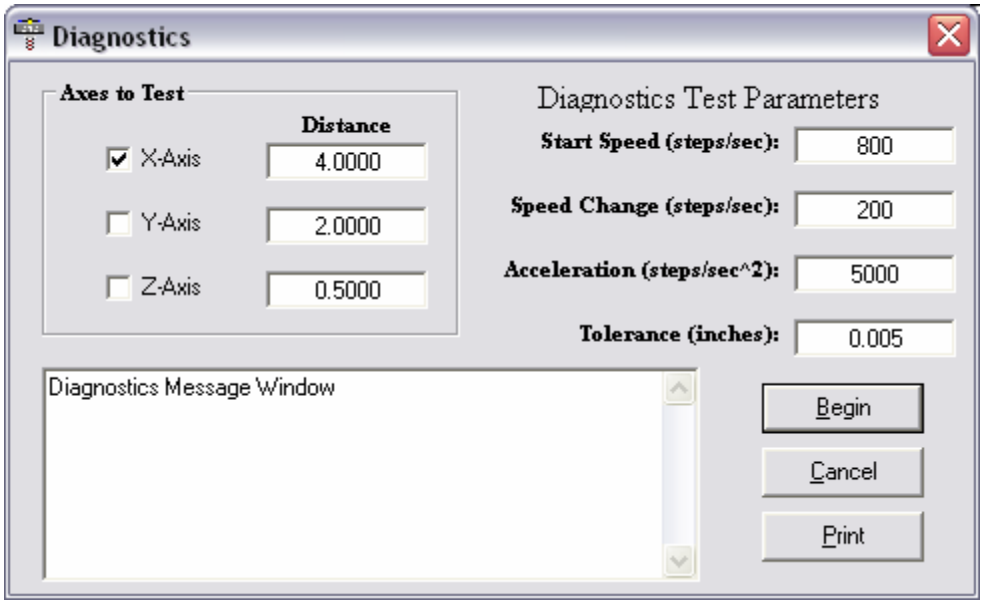
2. Set the **Rapid Traverse Speed** to the maximum speed at which your machine can move **reliably.** The default is set to between 2500 and 3500 Steps/Sec depending on the machine you are running. You may be able to set this at a much higher rate if you are using ballscrews or other low friction mechanical 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 **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 or 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 hitting **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.

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

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.

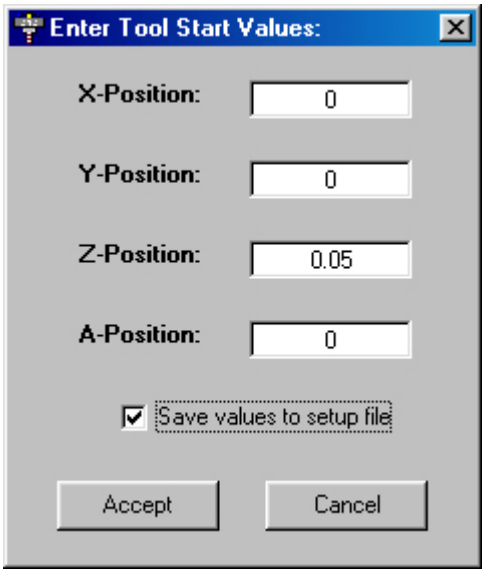
4.5 Setting the Tool Start Position

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".

Set the following values:

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

To work with the tutorial



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.

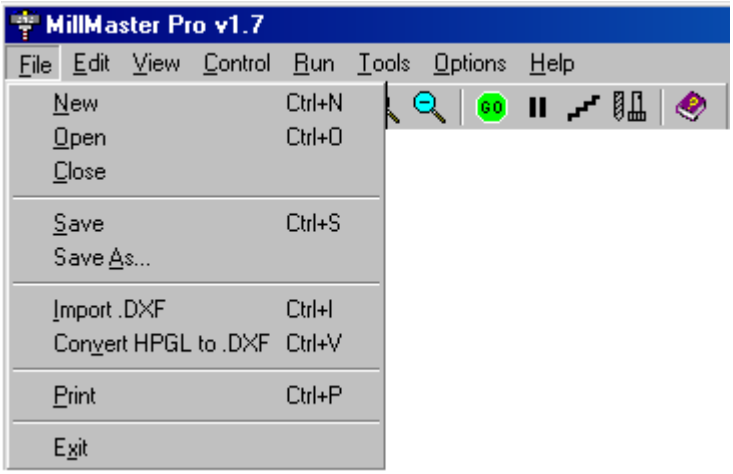
Save Parameters as Default?
When exiting **MillMaster Pro**, you may receive the above message. If you click "**Yes**" MillMaster will store all your specific parameters on disk so that they automatically load at start-up.

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*

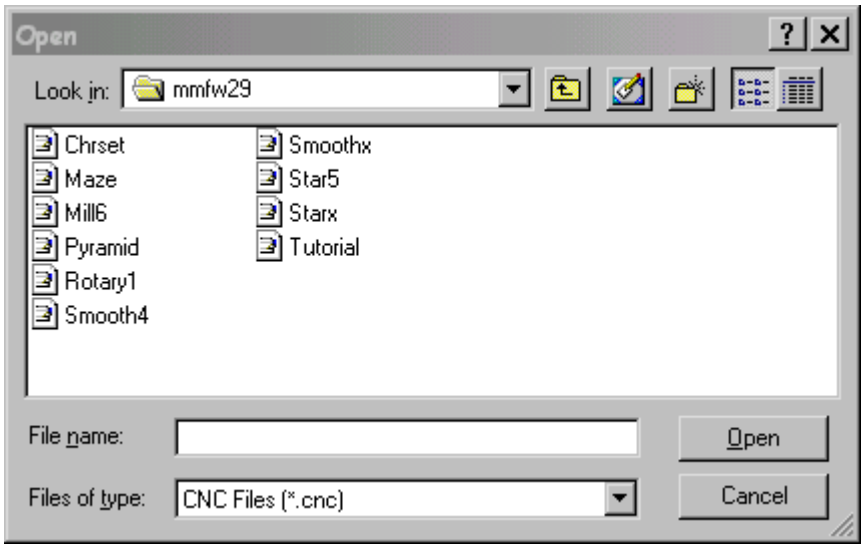
Clears the part program in memory and establishes the standard defaults for a new part program. The first icon in the Toolbar also corresponds to this menu command.

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

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..

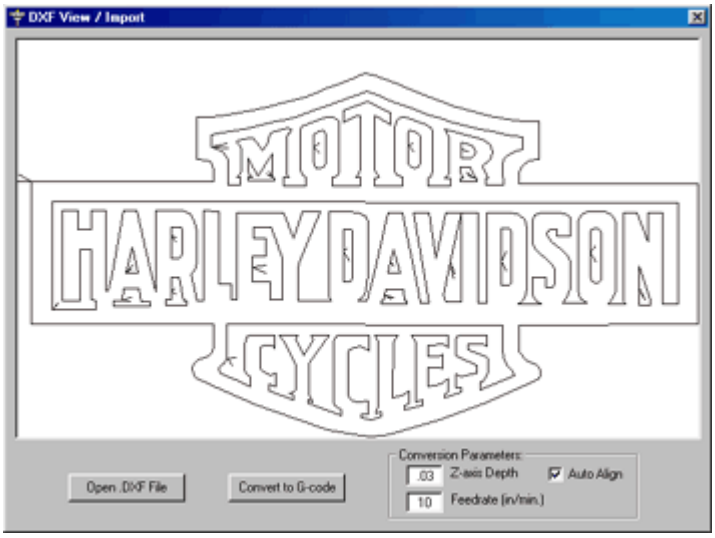
.DXF files that are saved with Tables or that use 3D modeling will not show up in the DXF window, (shown below), and will not convert to G-code. The following entities are supported for conversion to G-code:

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

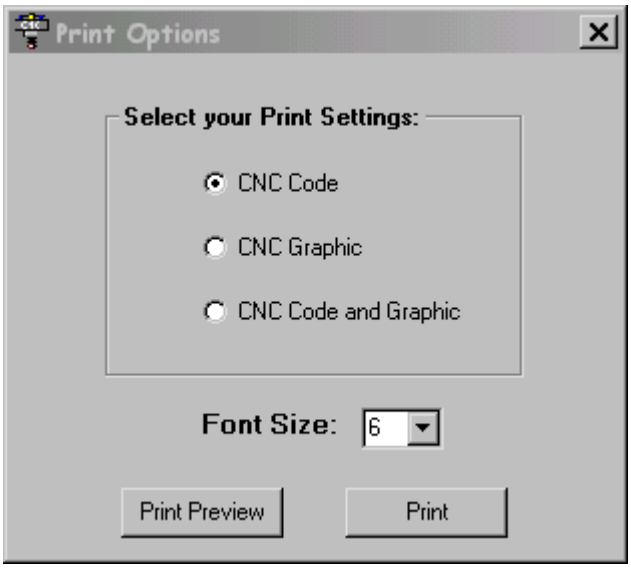
TIP: The conversion from HPGL to DXF is very generic to prevent unwanted alterations of your original .PLT file design in the newly created .DXF file, but the .DXF files created by the converter default to 110" by 110" in length and width. Once the part program is loaded use Options --> Material Size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

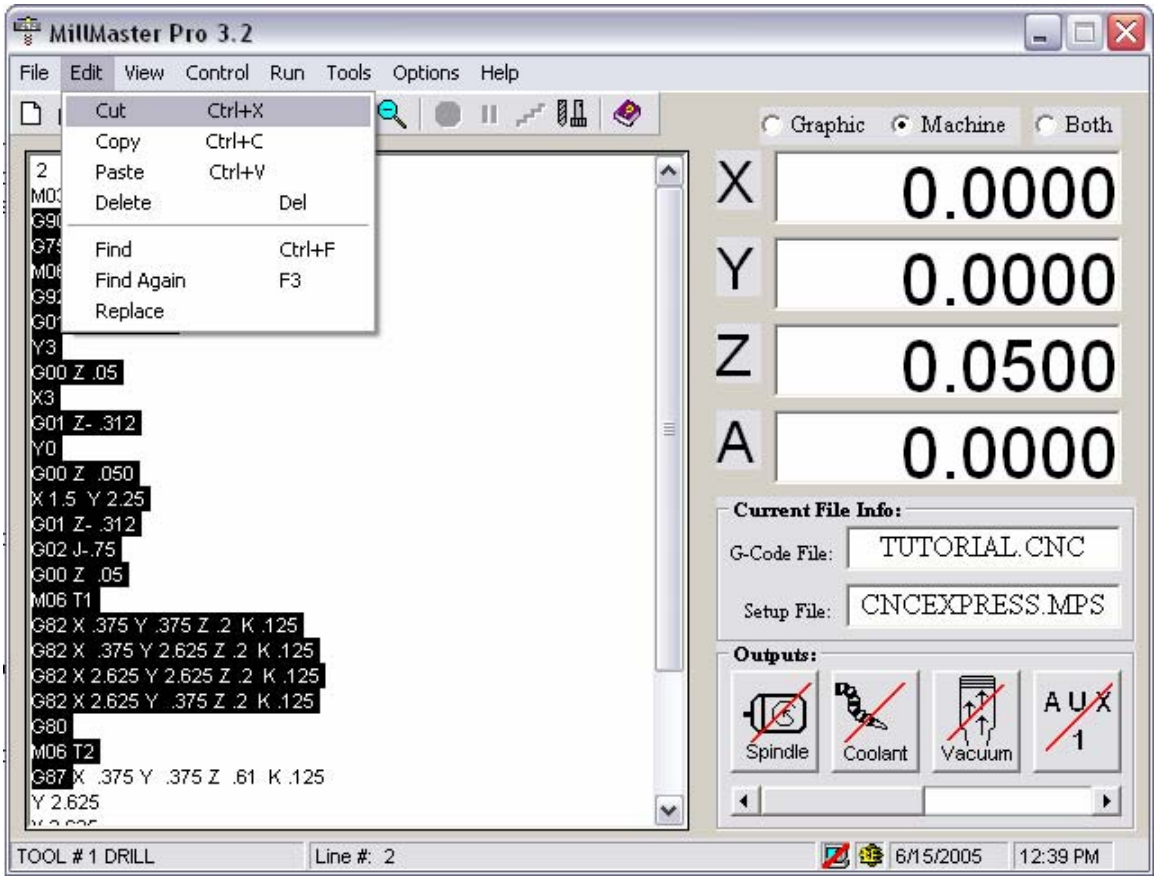
The fourth icon in the Toolbar also corresponds to this menu command.

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.

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.

NOTE: This command is "grayed" out and disabled unless MillMaster Pro is in CNC Edit Mode.**

Edit- *Copy*

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

NOTE: This command is "grayed" out and disabled unless MillMaster Pro is in CNC Edit Mode.**

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.

NOTE: This command is "grayed" out and disabled unless MillMaster Pro is in CNC Edit Mode.**

Edit- *Delete*

Deletes the currently highlighted text permanently.

NOTE: This command is "grayed" out and disabled unless MillMaster Pro is in CNC Edit Mode.**

Edit- *Find*

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

NOTE: This command is "grayed" out and disabled unless MillMaster Pro is in CNC Edit Mode.**

Edit- *Find Again*

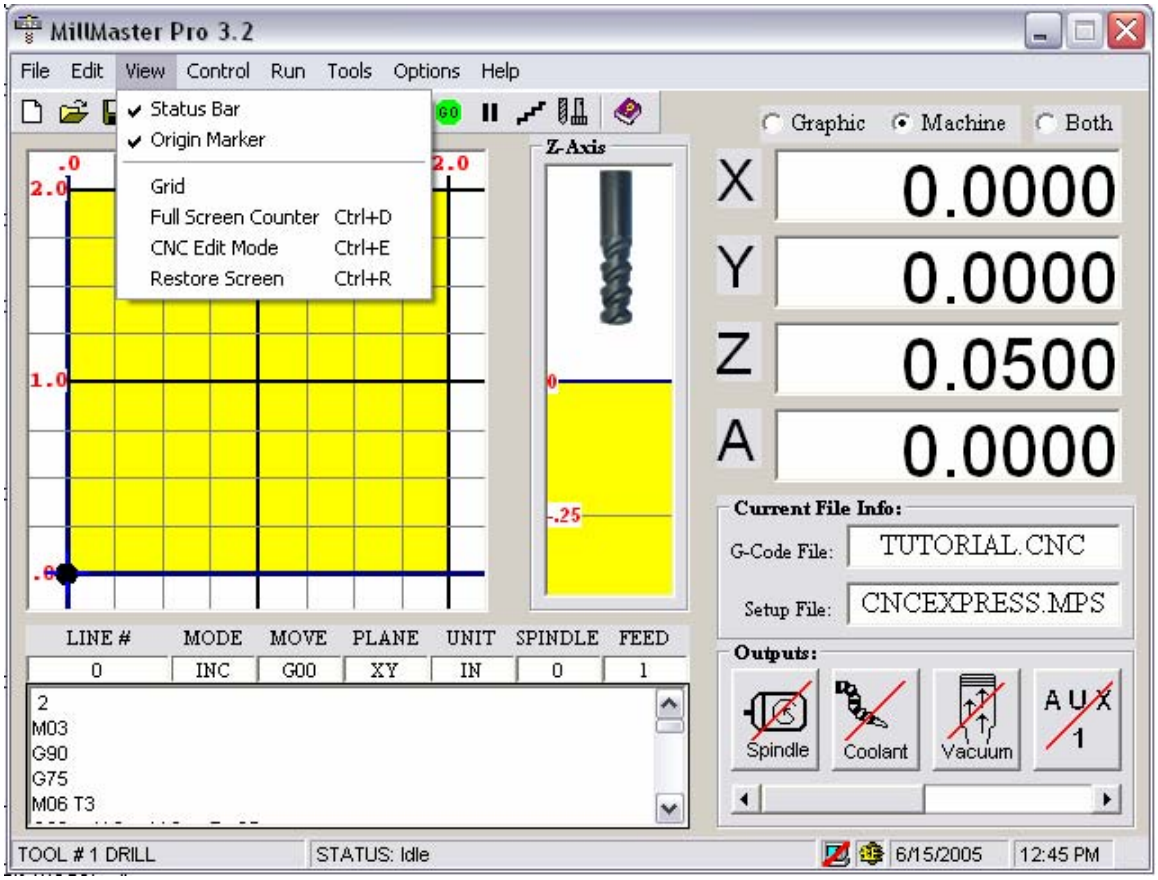
Finds the next iteration of the specified sequence of characters you previously looked for with the FIND command.

NOTE: This command is "grayed" out and disabled unless MillMaster Pro is in CNC Edit Mode.**

Edit- *Replace*

Replaces a specified sequence of characters in your CNC Edit window with a new string input by the user.

NOTE: This command is "grayed" out and disabled unless MillMaster Pro is in CNC Edit Mode.**



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.

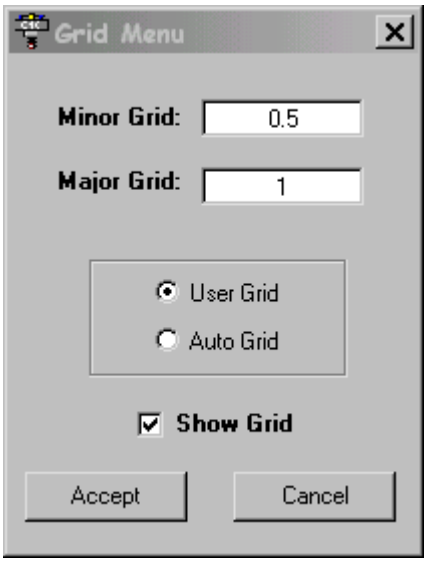
View- *Origin Marker*

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- *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)**

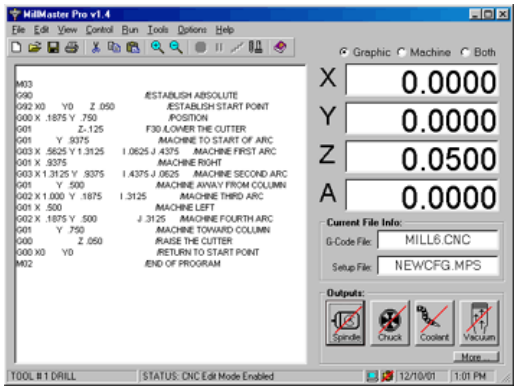
View- *CNC Edit Mode*

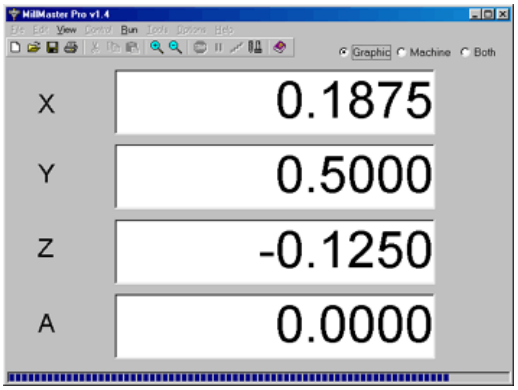
Turns on CNC text edit mode and displays the CNC text editor while temporarily hiding the cutting window. This allows the user to Edit any currently loaded program, or enter a part program manually. This works like any standard Text editor. When a check mark appears next to this function, it is enabled.

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

Main Screen with CNC Edit mode enabled

Main Screen with Full Screen Counter mode enabled



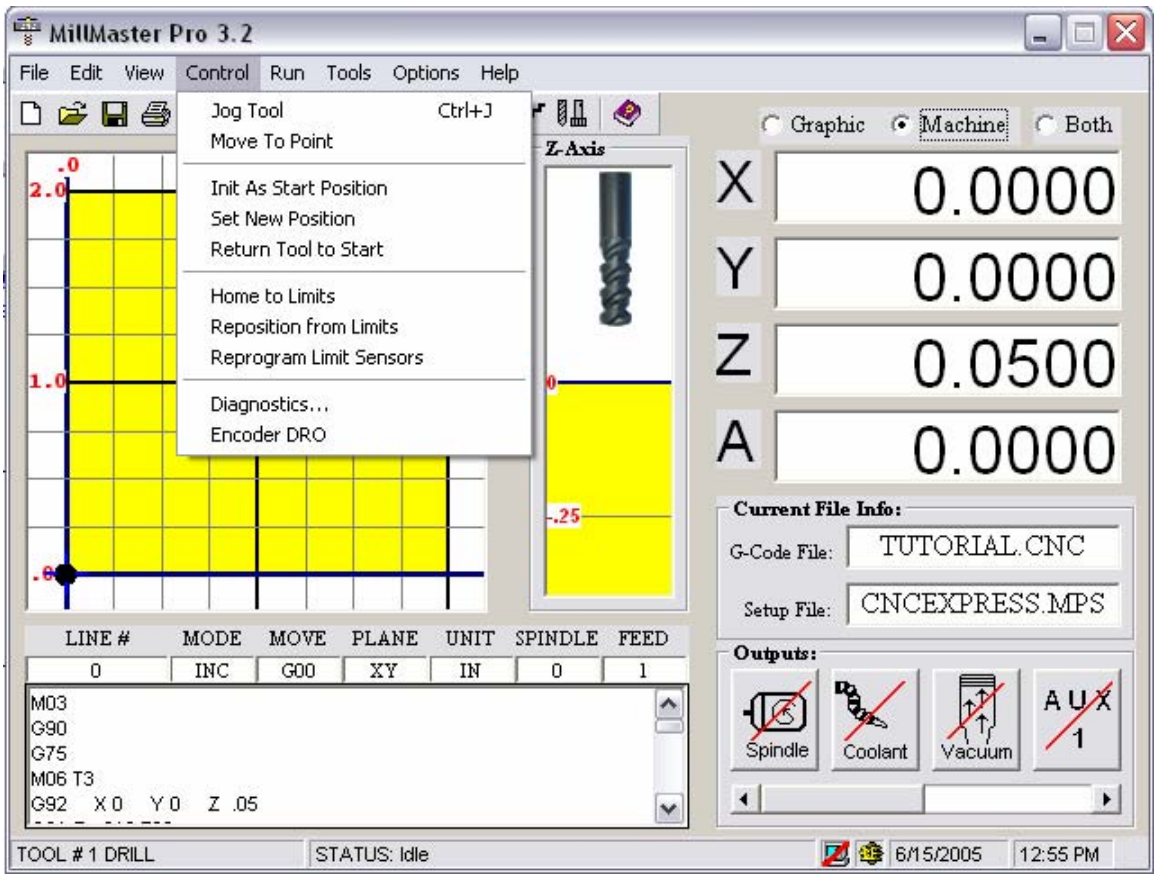


(These graphics have been shrunk for spacing purposes)

View- *Restore Screen*

Redraws 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.

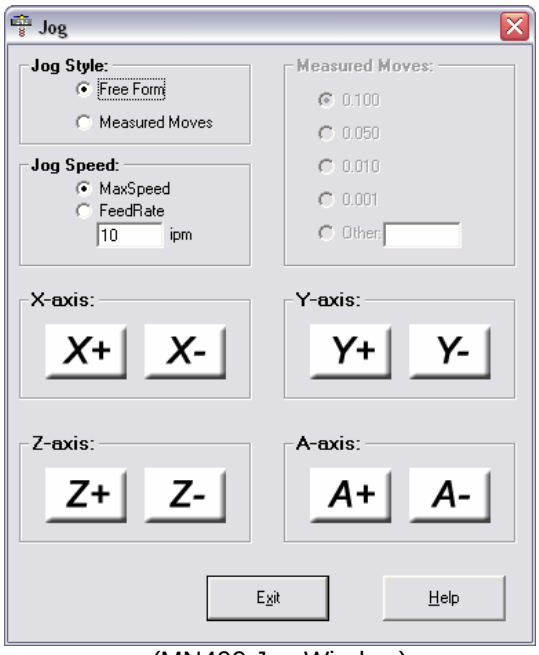
The values in the Material Size menu are applied here for the redrawing of the uncut stock material.



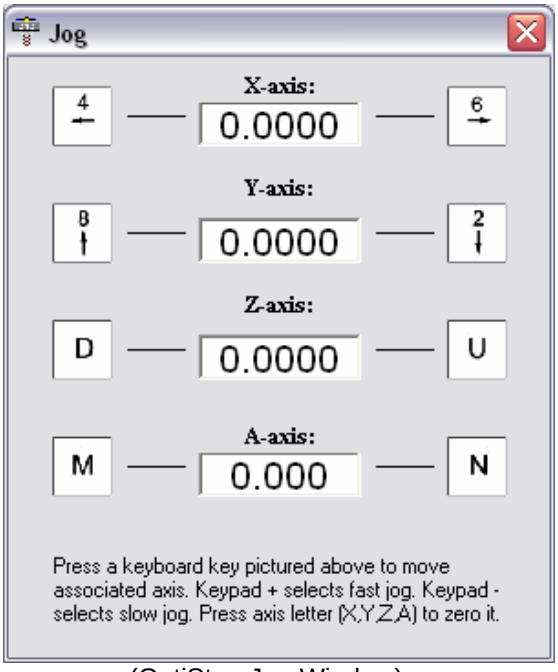
The **Control Menu** is used to jog the tool of the machine, set up the starting position of the machine, and calibrate the machine. The underlined letters are called ***QuickSelect*** keys. These allow access to a command only while the pull-down menu is displayed.

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.



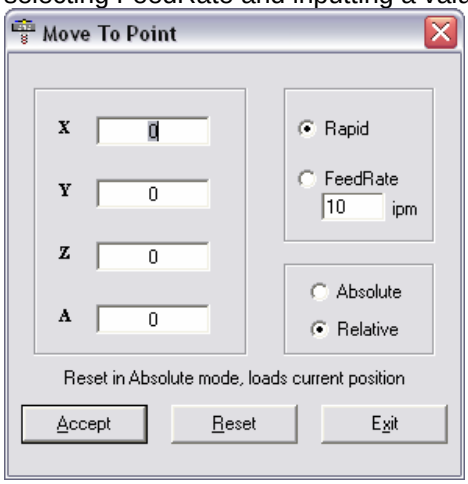
(MN400 Jog Window)



(OptiStep Jog Window)

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*

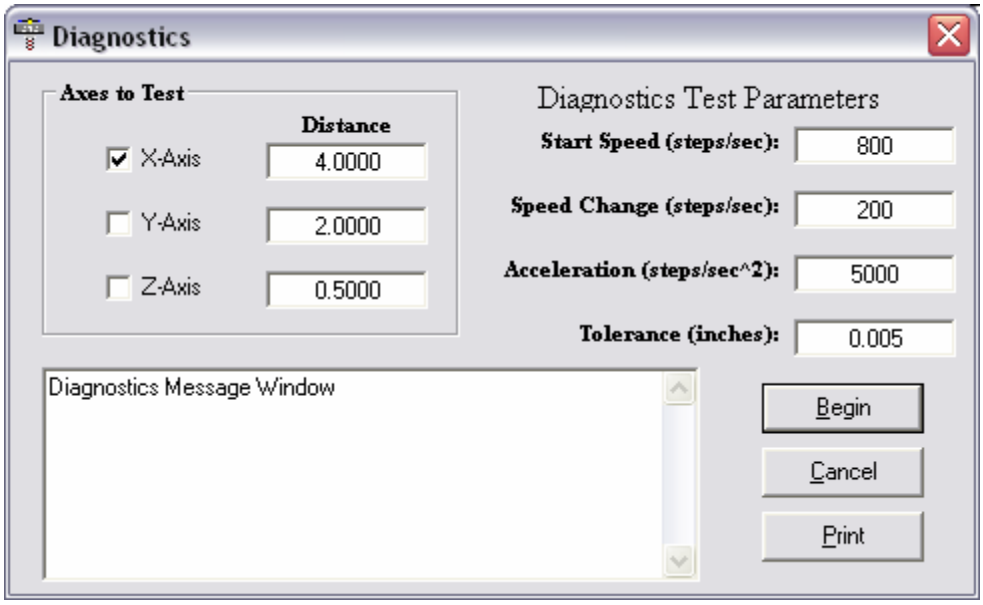
Automatically measures and stores the distance from the Start Point to each of the 3 Limit Switches. Tool must be positioned at its Start Point prior to selecting this function. Limit switches must be connected to use this function.

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)**.

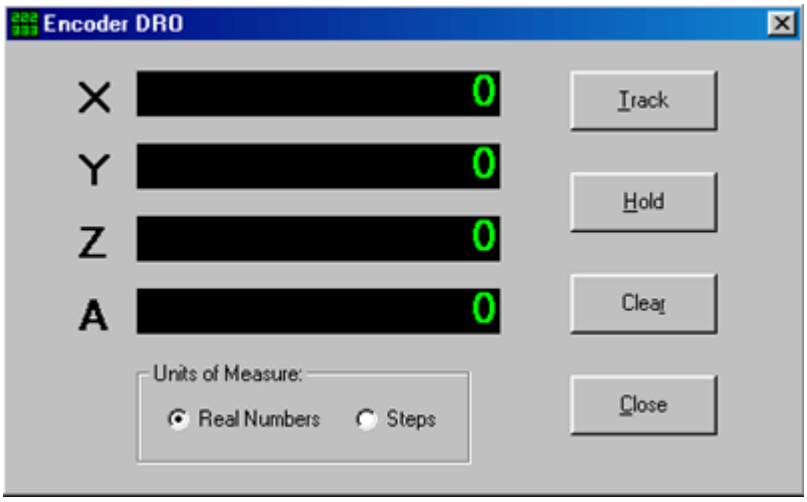
Simply select which axes you want to speed test and enter a value in the Distance field. We recommend a minimum of 4" to allow the machine time to ramp up to the higher speeds. The message window will display the time the move took and how fast the machine was traveling both in steps per second and inches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

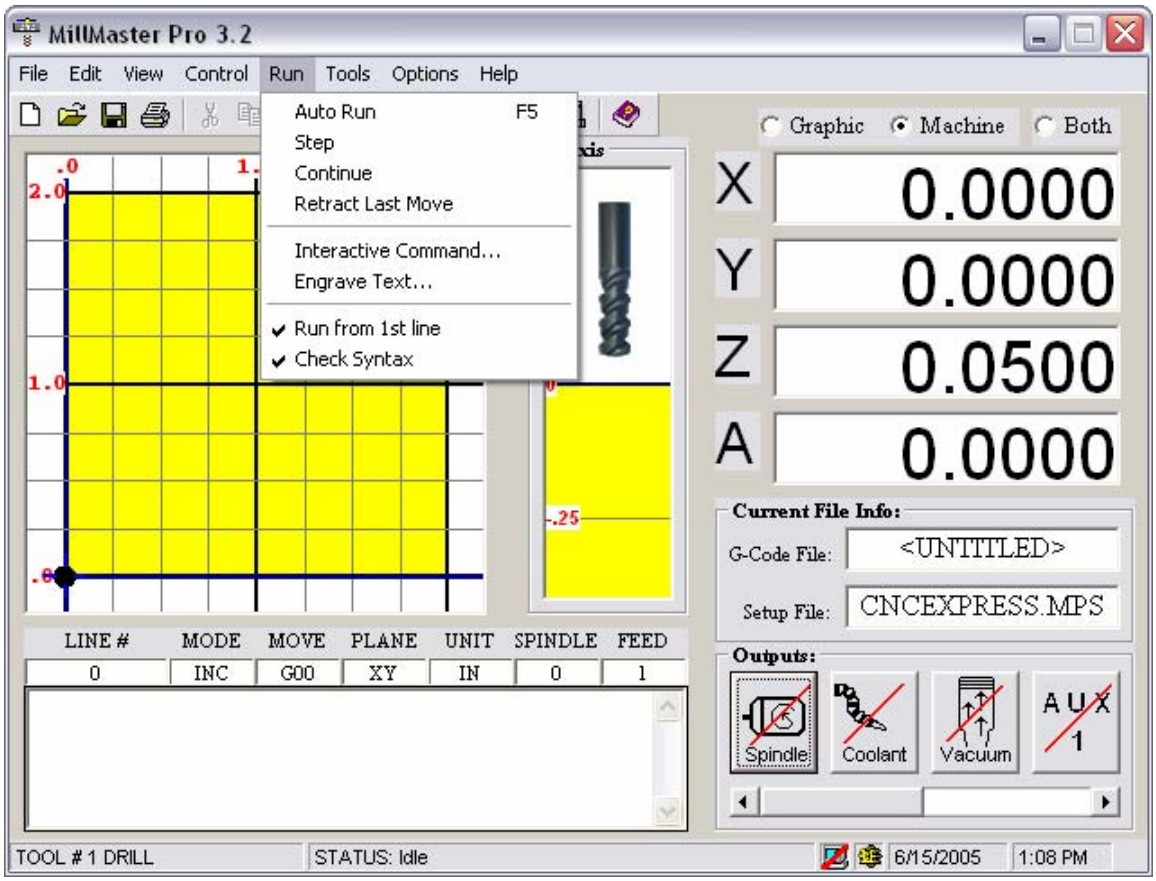


Control- *Encoder DRO*

Displays a **D**igital **R**ead**O**ut that allows the user to test the functionality of the OptiTracker board. The DRO window (shown below), has 4 command buttons, Track, Hold, Clear, Close, and two Unit of Measurement options; Real Numbers and Steps. Track starts the software counter updating, Hold stops tracking and allows you to move the motor without affecting the display, Clear sets the internal OptiTracker counters to 0, and Close exits the DRO window. When exiting, if the physical location of the motor has changed, MillMaster Pro will prompt you to match positions. If you choose YES, the Instep driver and the OptiTracker internal count will be synchronized. If you choose NO, the OptiTracker count will differ from the Instep count and could cause errors when utilizing encoders with part programs. You can switch the Digital Display between Real Numbers and Steps by clicking on the corresponding radio button in the Units of Measure frame.

You must specify the OptiTracker address in the Machine Parameters section of the program. The OptiTracker board and encoders work **ONLY** with the DOS version of the software at this time, but support will be available soon for Windows systems using the MN400 controller.




MillMaster Pro 3.2
File Edit View Control Run Tools Options Help
Auto Run F5
Step
Continue
Retract Last Move
Interactive Command...
Engrave Text...
Run from 1st line
Check Syntax
Graphic Machine Both
X 0.0000
Y 0.0000
Z 0.0500
A 0.0000
Current File Info:
G-Code File: <UNTITLED>
Setup File: CNCEXPRESS.MPS
Outputs:
Spindle
Coolant
Vacuum
AUX 1
LINE # MODE MOVE PLANE UNIT SPINDLE FEED
0 INC G00 XY IN 0 1
TOOL # 1 DRILL
STATUS: Idle
6/15/2005
1:08 PM

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 tenth 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 eleventh 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.

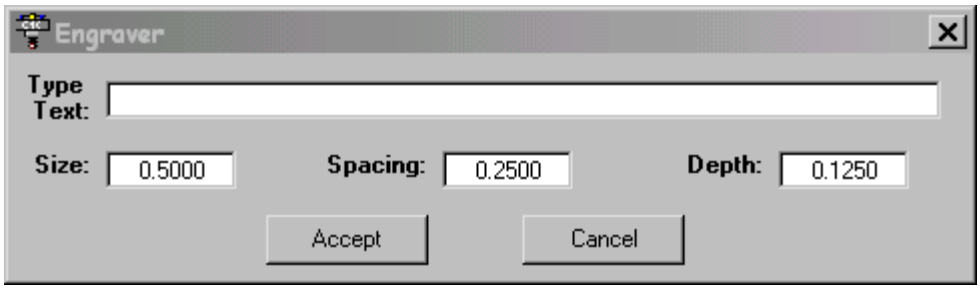
Run- *Interactive Command*

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>**.

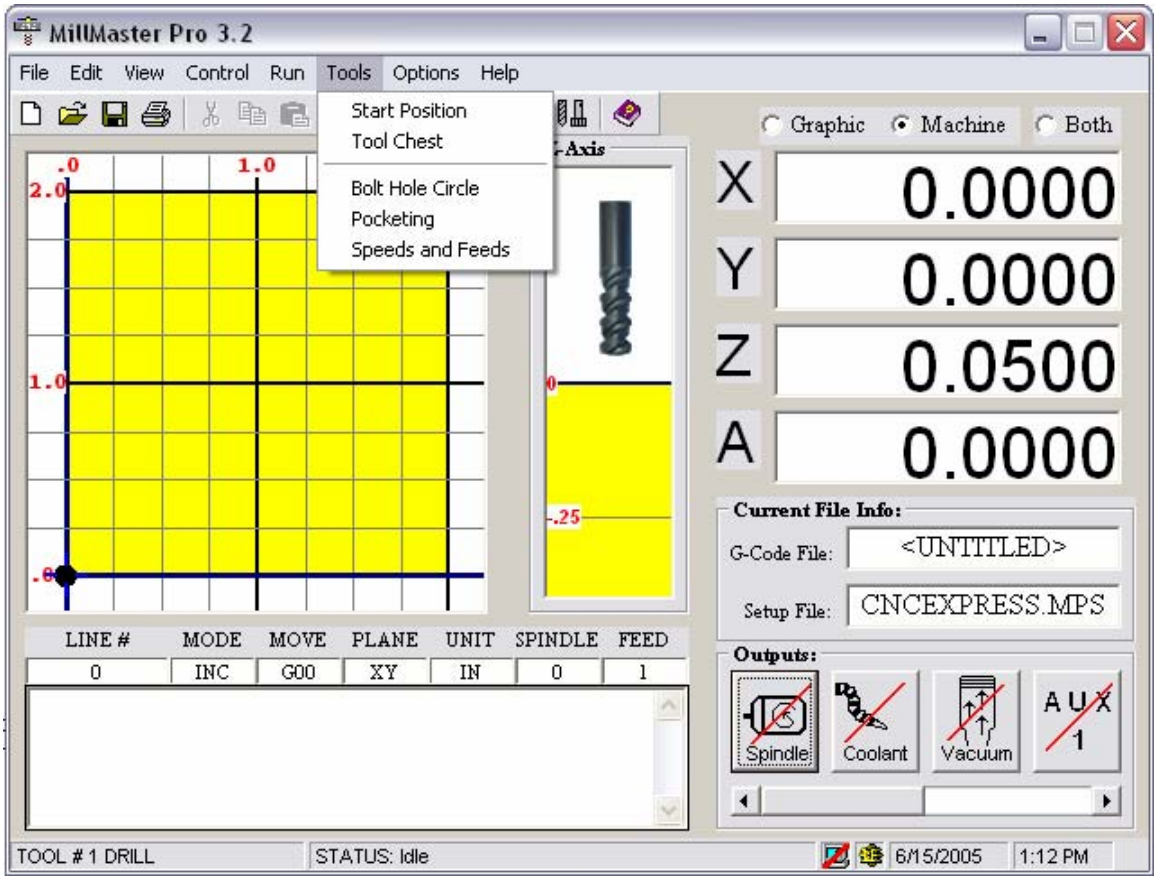
To machine, select **Machine Mode** and then **Auto Run**.

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.



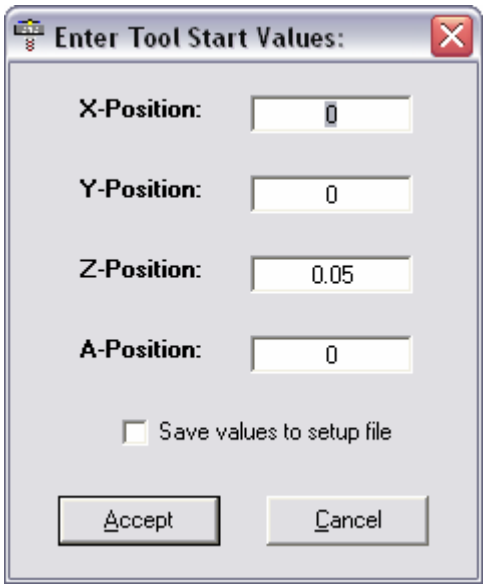
The **Tools Menu** is used to specify the tool to use, its specific attributes, and to specify a starting position for the tool.

The underlined letters are called ***QuickSelect*** keys.

These allow access to a command only while the pull-down menu is displayed.

Tools- *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.

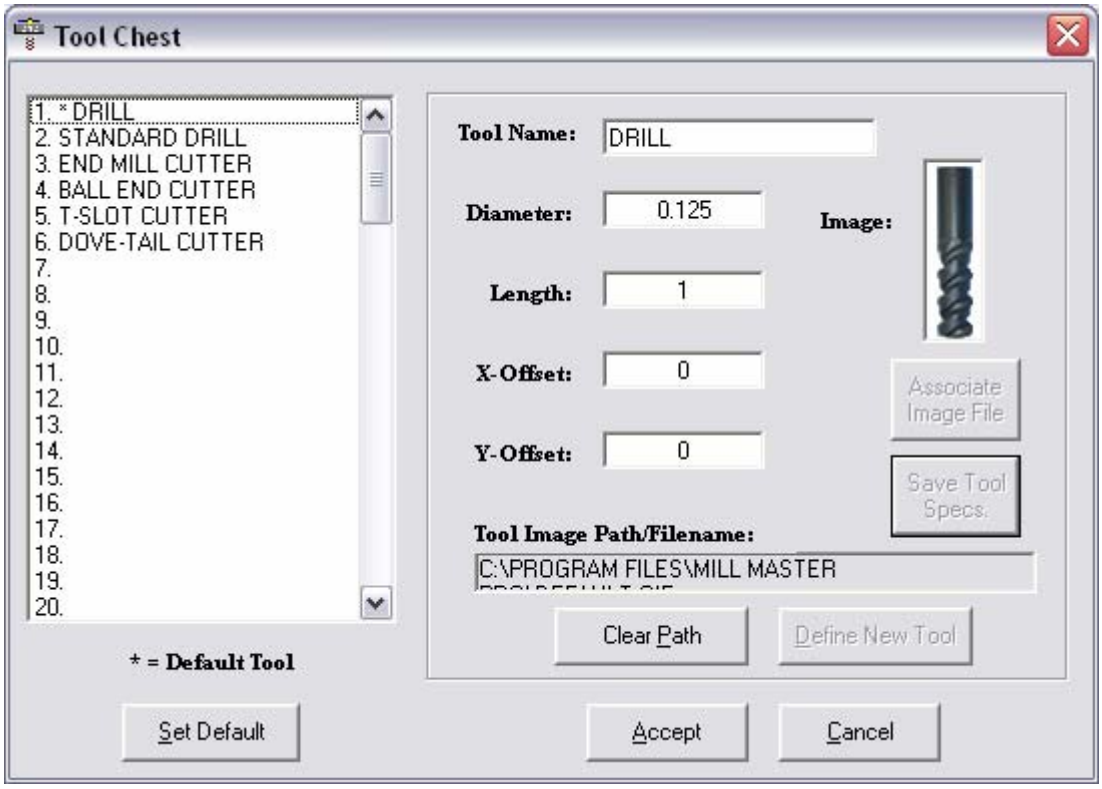


Tools- *Tool Chest*

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 twelfth icon in the Toolbar also corresponds to this menu command.

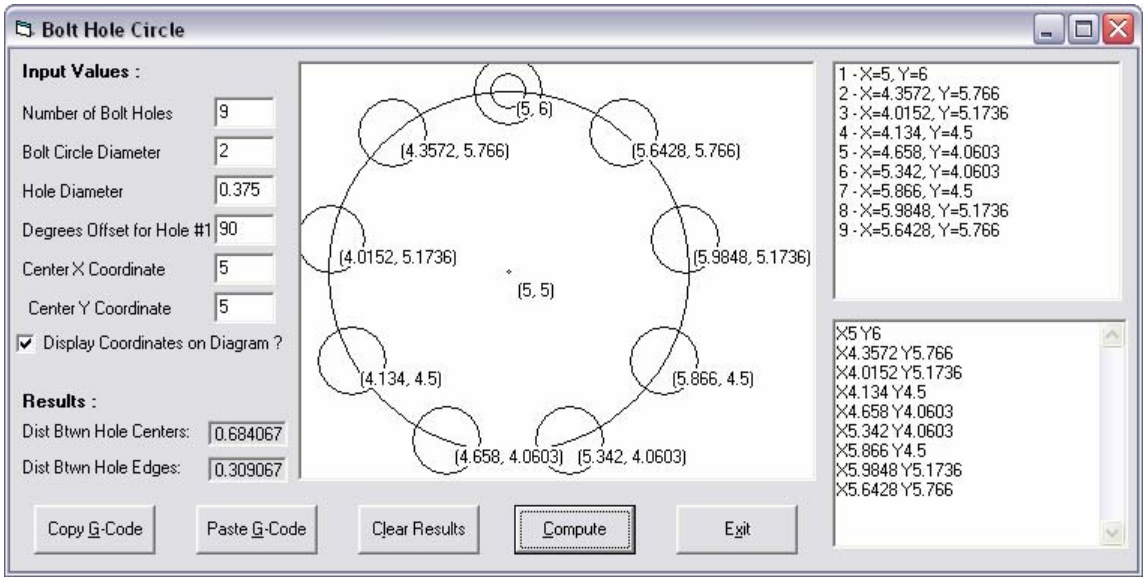
You can create a new tool by selecting an empty slot (or overwriting an existing if you prefer), and then clicking the define new tool button. Now you will need to name your new tool. You can then input the diameter of the tool which is for graphic purposes only. The cut path of your program will vary 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, you can import a tool graphic from a location on your computer. Once you have finished entering all the data, you will need to select Save Tool Specs.

You can also change the default tool through this window as well as change to any other tool before you begin a program by clicking that tool on the left and pressing Accept.



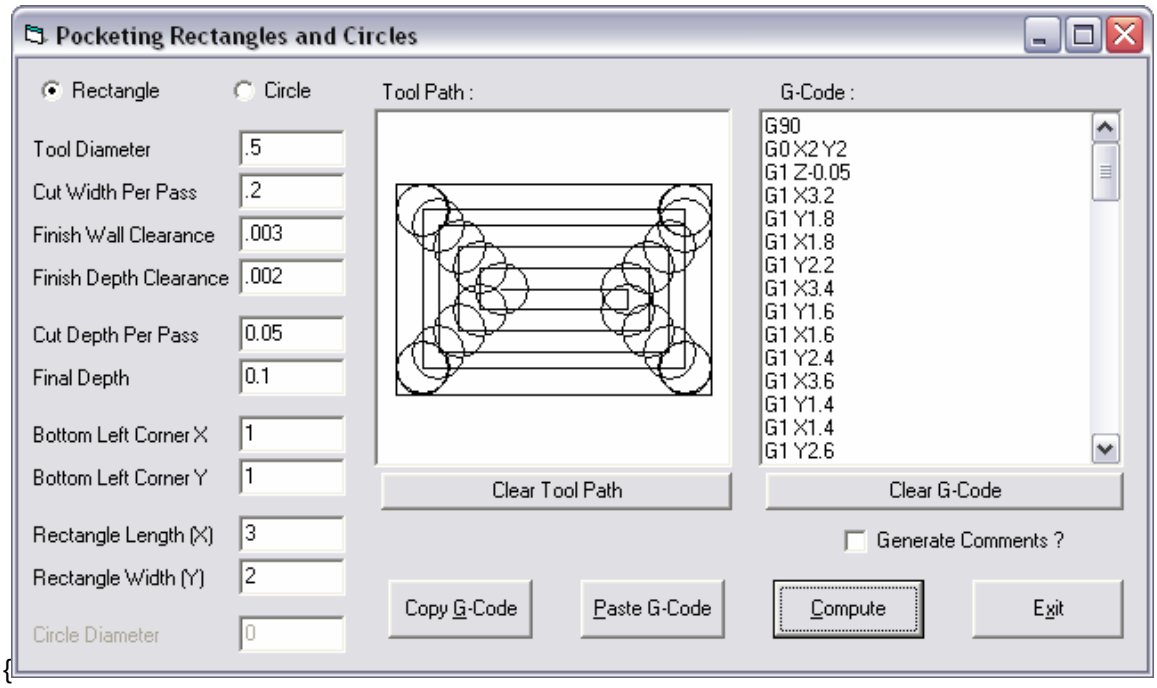
Tools- *Bolt Hole Circle*

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.



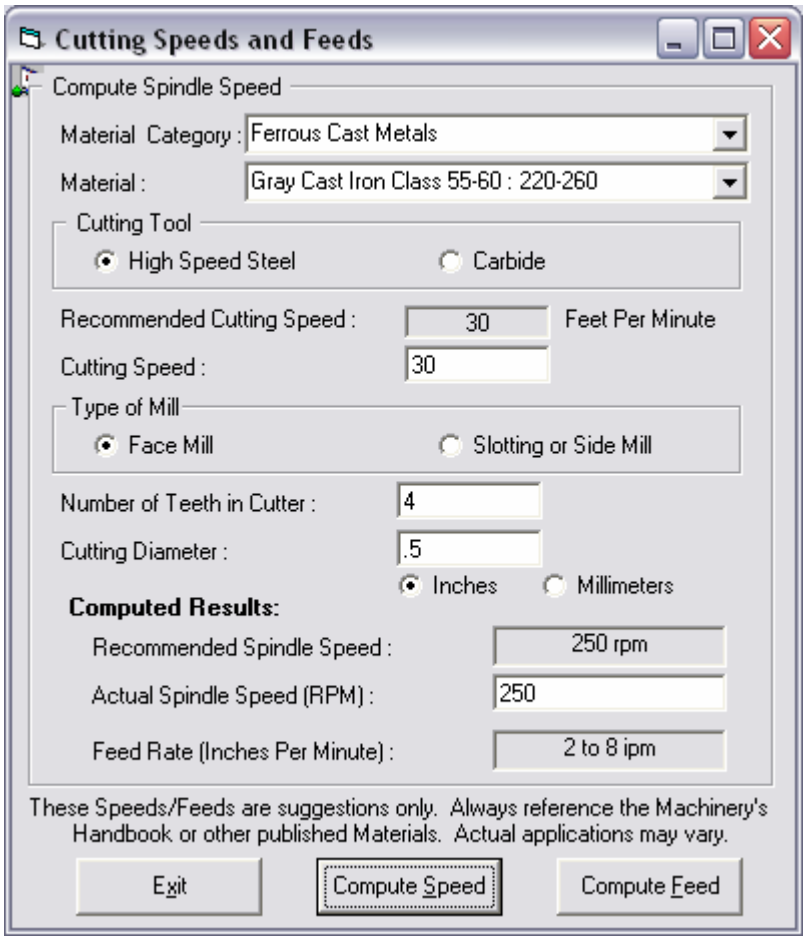
Tools- *Pocketing*

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.



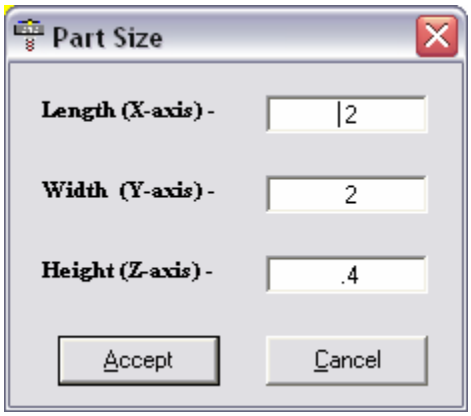
Tools- *Speeds and Feeds*

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.



Options- *Material Size*

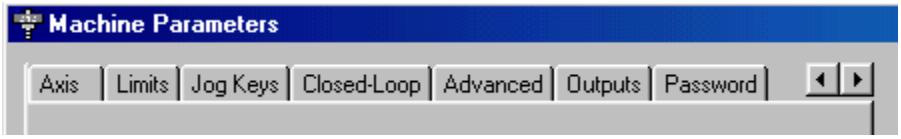
Allows the user to enter the material **Length**, **Width**, and **Height** in the units specified in the **Machine Parameters window**.



Options- *Machine Parameters*

Allows the user to set Machine Parameters such as Unit of Measurement, Origin, select the type of **MicroKinetics** controller, **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 setup 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 you will be using **MillMaster Pro** with. This menu can be accessed with a **Mouse Quickselect** 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.

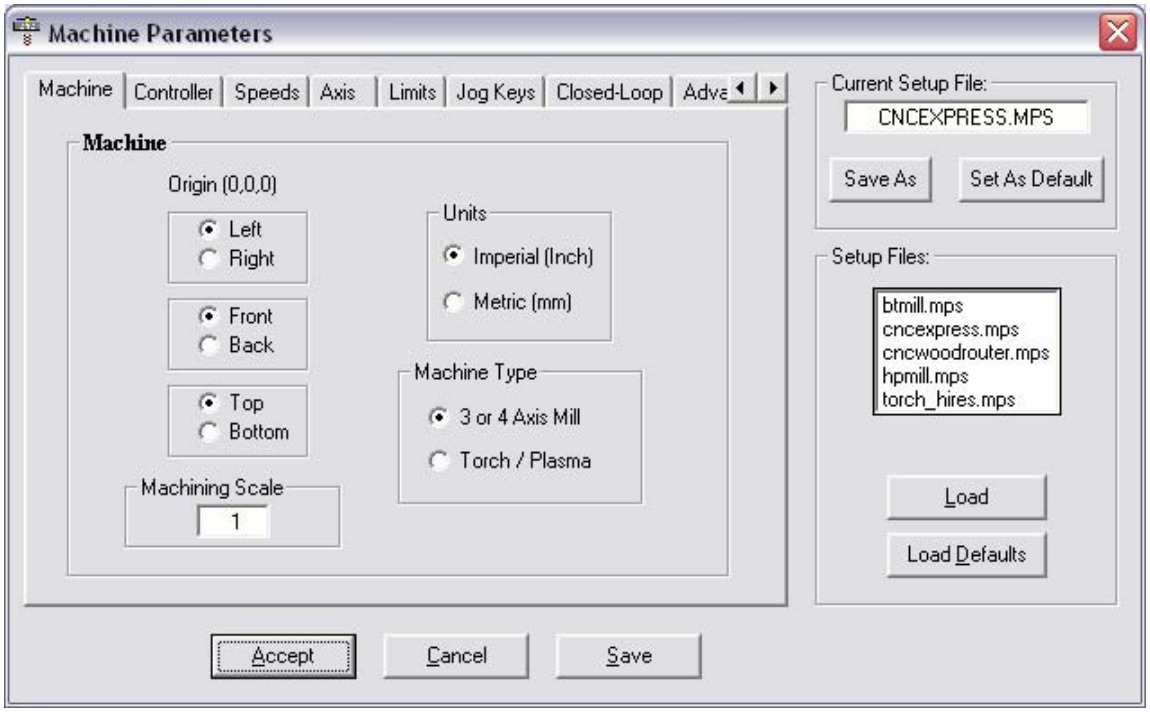
From any tab in **Machine Parameters**, the user can Load a new setup file which stores all of the users settings for their machine. Simply click one of the files under the **Setup Files** frame and press the **Load** button just below this. If you know that this is the file that you are going to want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

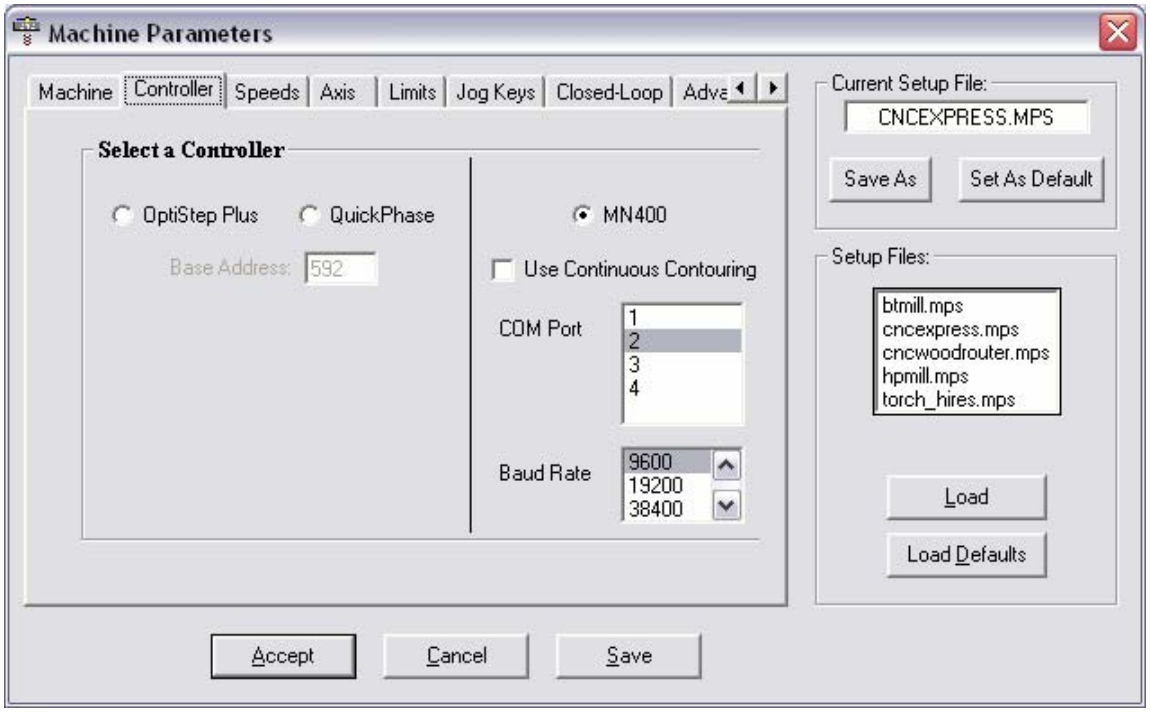
Switching the Machine Type will change certain aspects of the functionality of the program depending on what type of machine you will be running.



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:** The DIP switch settings on the card must match this address for the OptiStep or QuickPhase cards.



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.

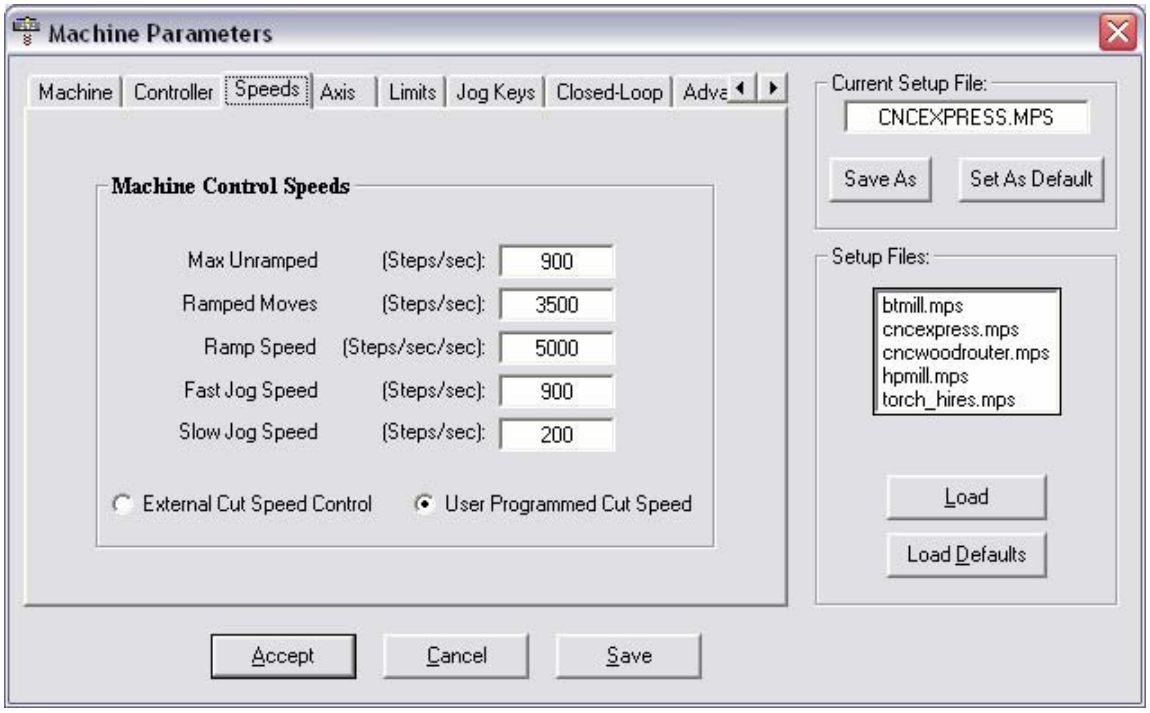
Fast Jog Speed is the rate the machine will travel in the *Jog Mode* after the "Fast Jog" button (+ key on the number keypad) is pressed on the screen.

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.

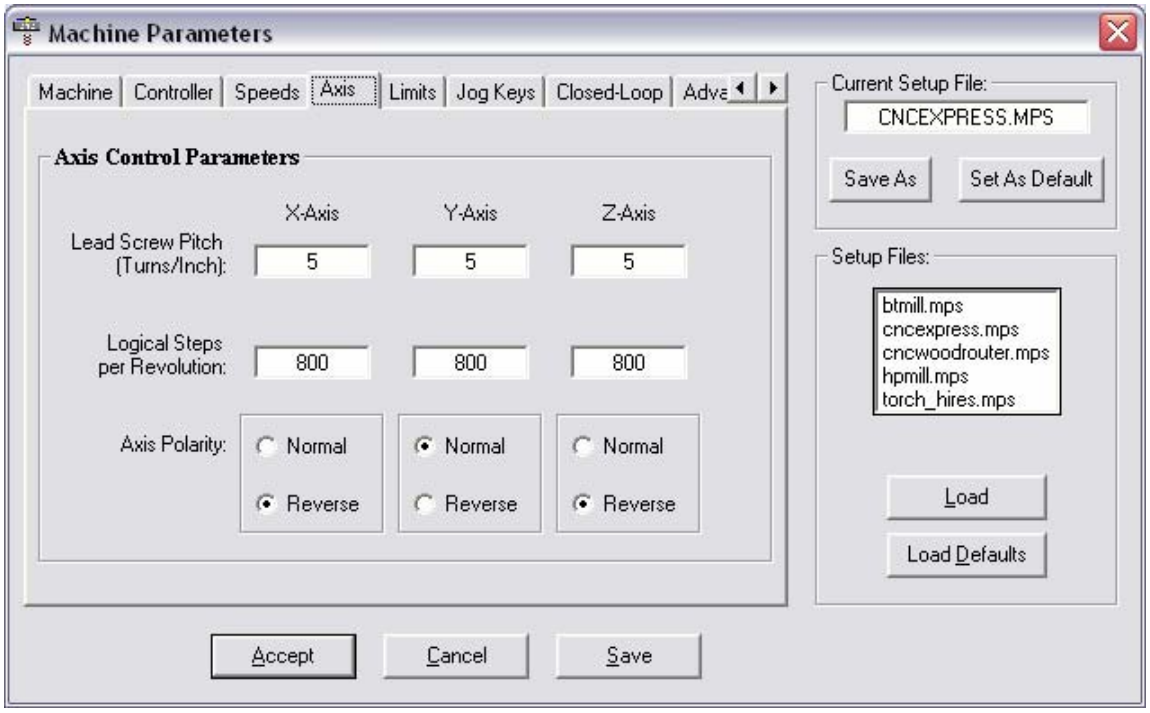
Note: If REVERSE is selected for any axis, then the limit sensor connections must also be reversed **

(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.

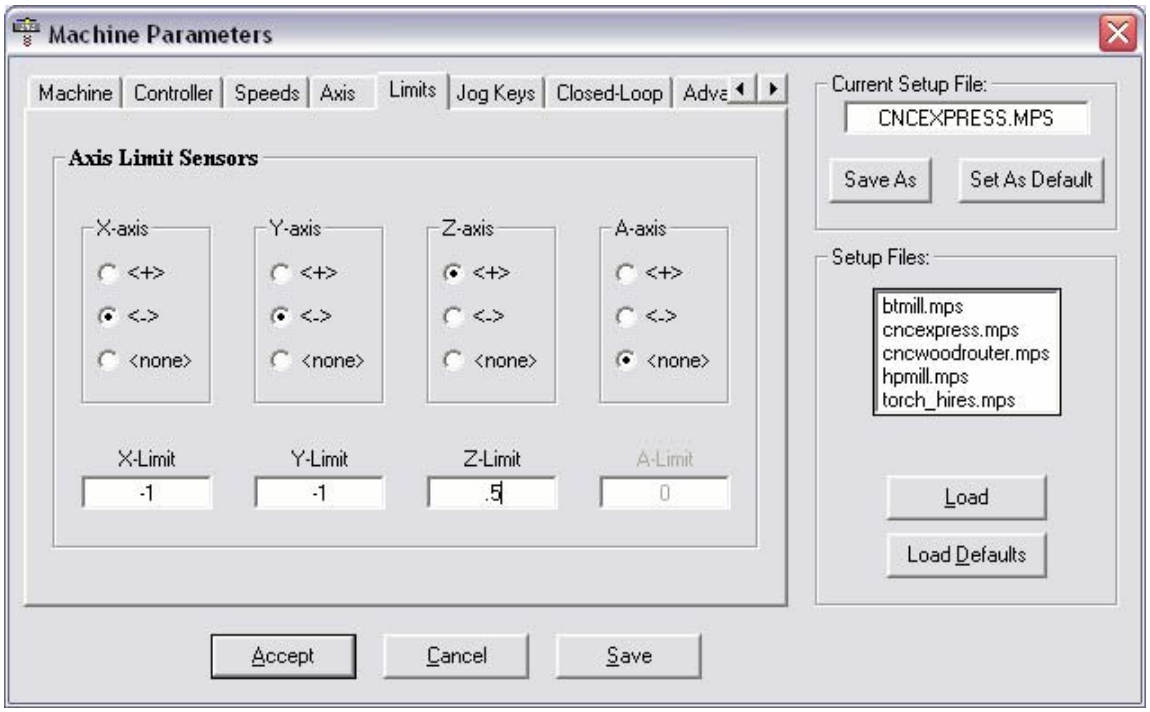
Example:

Motor Res.Driver Res.Logical Steps/Rev
200 FULLSTEP 200
200 HALFSTEP 400



Options- *Machine Parameters-Limit Sensors*

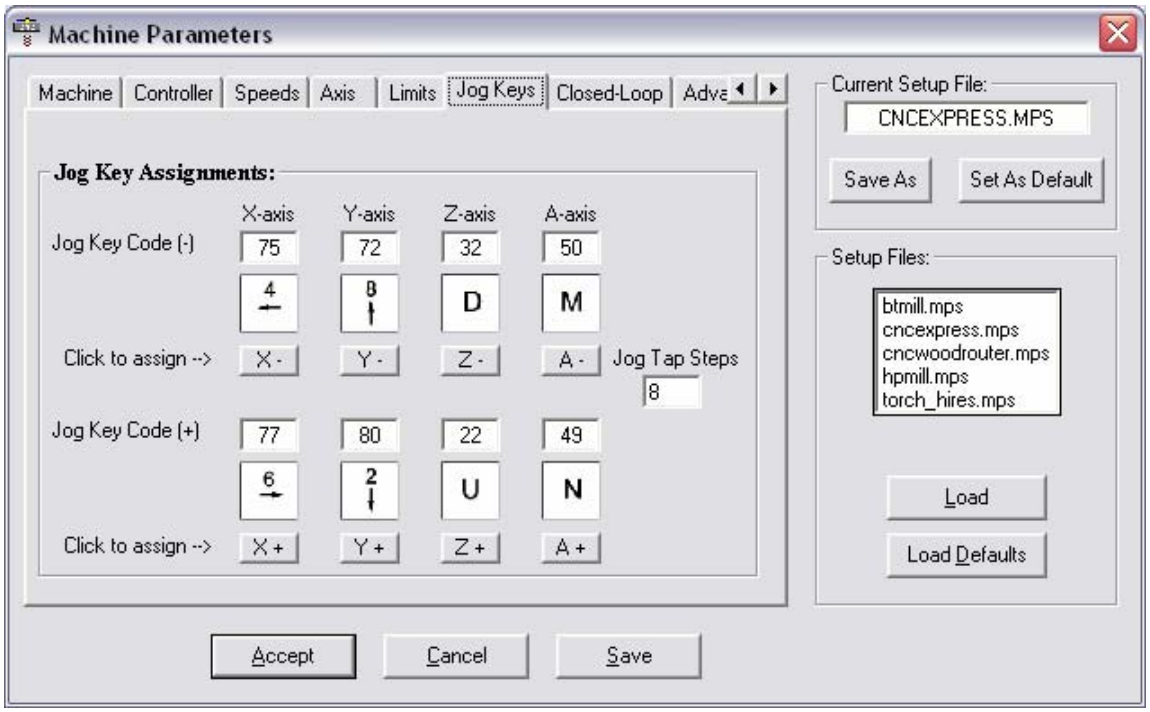
Allows the user to select the limit switches that will be used to home to. Enter the position of the home limit sensor for each axis in reference to the origin of the part. On the Z-axis only, enter the position of the bottom of the tool holder in reference to the origin of the part.



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.

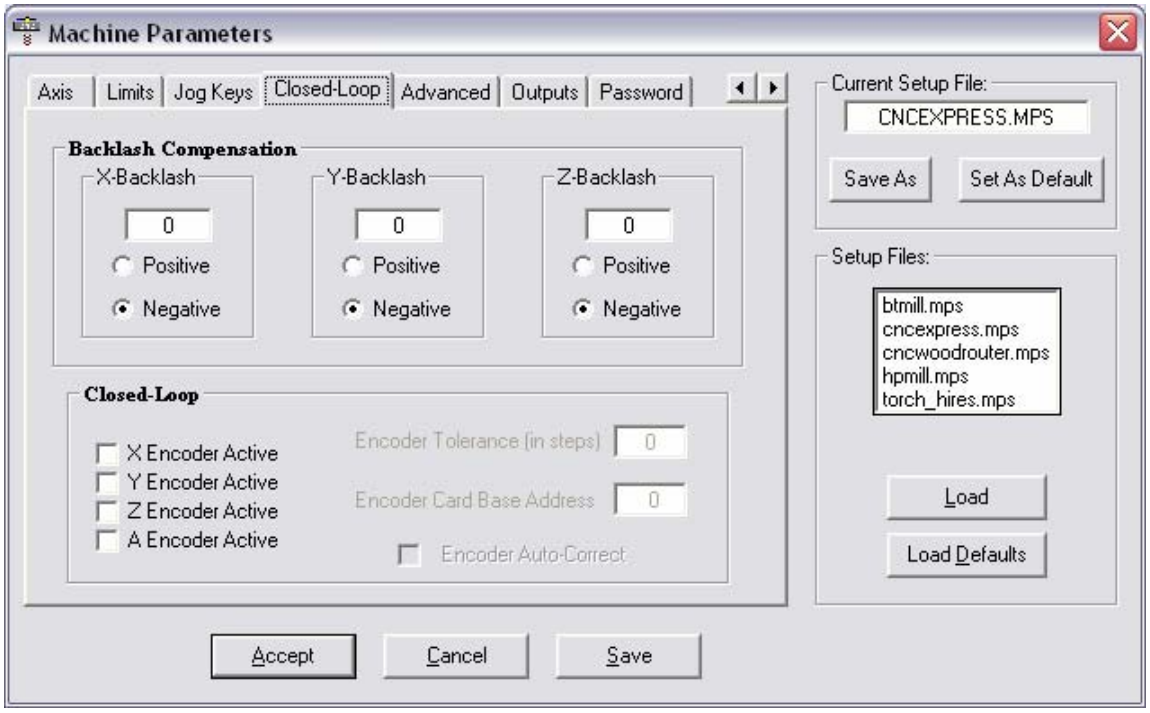
Jog Tap Steps is the number of steps the motor will rotate when a jog key is tapped. A value between 1 and 8 is accepted in this field.



Options– *Machine Parameters-Closed Loop / Backlash compensation*

Allows the user to enter values for XYZ 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.

The Closed-Loop functionality is not available at this time, but will be in the near future.



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.

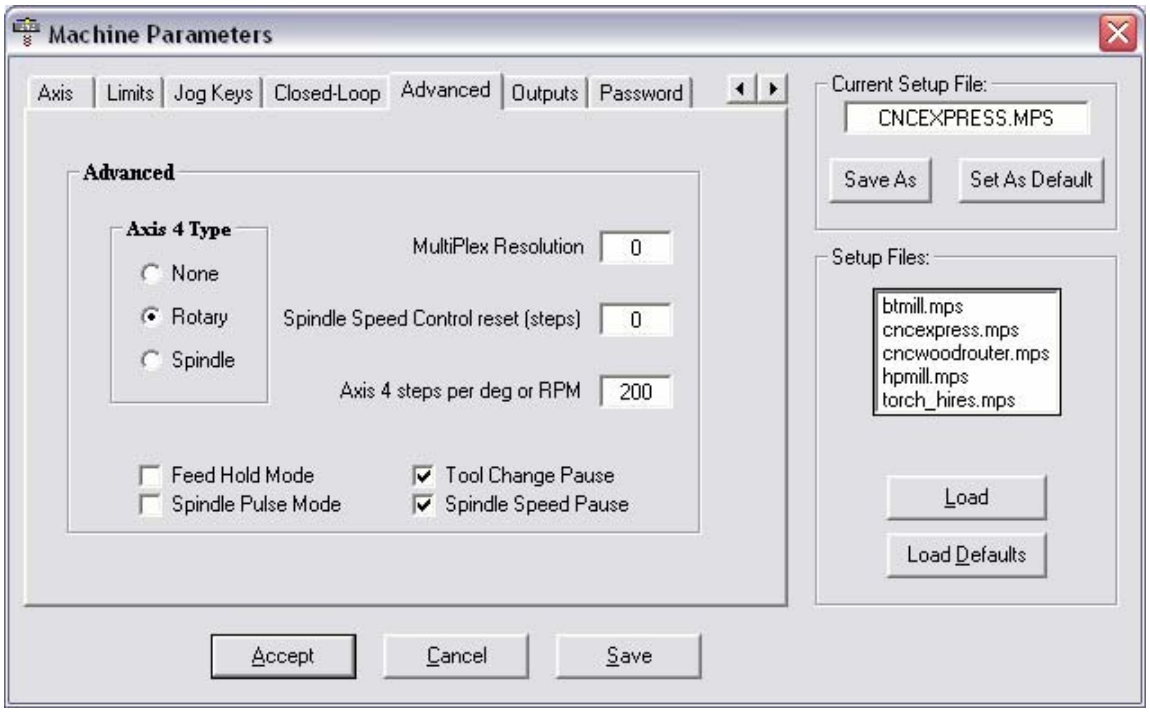
MultiPlex Resolution refers to resolution selection for diagonal linear moves. It allows for smoother movements while sacrificing your top speed. For most applications a multiplex of 2 works as a good compromise. 4 causes an added burden on the CPU but creates very smooth diagonal moves. 1 is for no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

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.

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

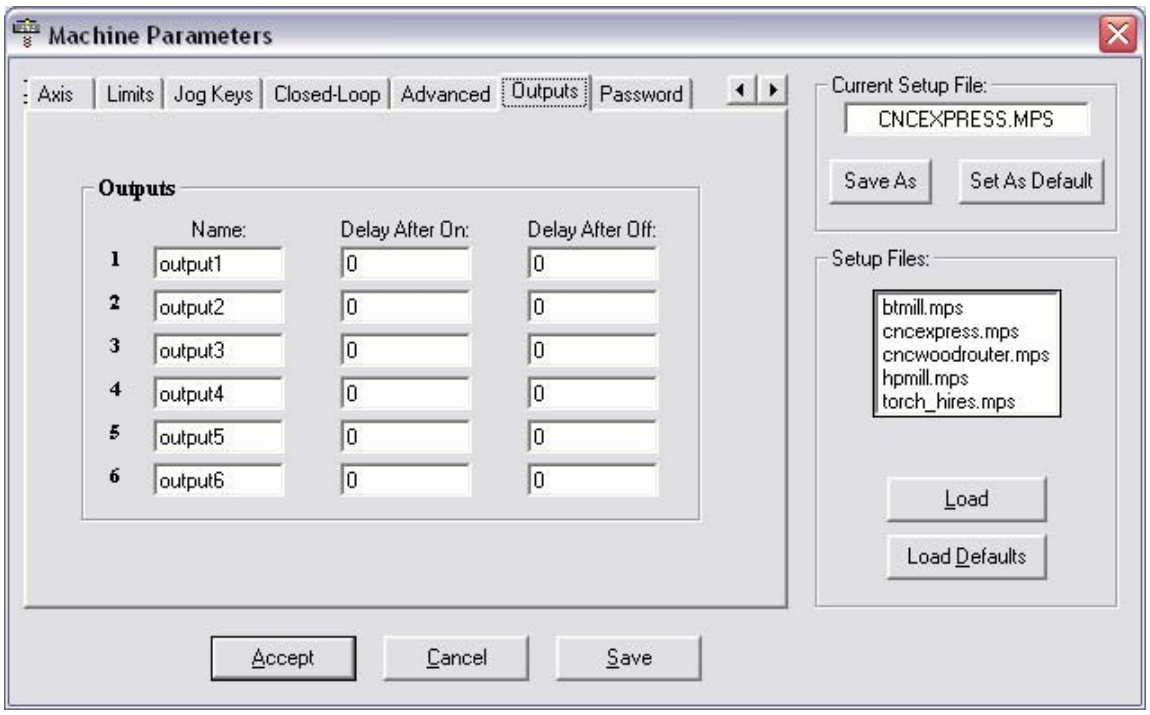
Tool Change Pause - This parameter specifies the action MillMaster Pro will take when it encounters an M06 command in a CNC part program. If checked, MillMaster will pause the CNC program and any machining so the machinist can manually change the tool. If it is not checked, the software will assume you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

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.



Options– *Machine Parameters-Outputs*

Allows the user to enter specific name and delay values for up to six 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-Set As Default*

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

Options- *Machine Parameters-Load Defaults*

Clears the current settings and restores the defaults from disk.

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-Save Setup*

Saves the current setup parameters to the current setup file.

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.

Help- *Contents*

This will display a window containing the contents of this help file. The fourteenth icon in the Toolbar also corresponds to this menu command.

Help- *Search For Help On*

This will search the help file and display a window containing the “searched for” function and a description on how to use it.

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.

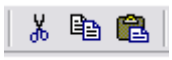
The following are short descriptions of the above graphical commands contained on the toolbar. (from left to right)

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%

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 G03command, 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.

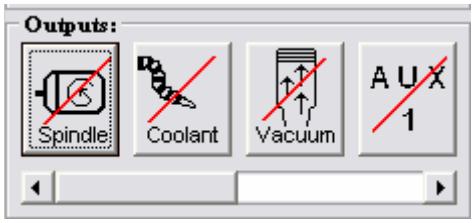
Help - Displays a window showing the contents of this help file.

ICON RELATED COMMANDS

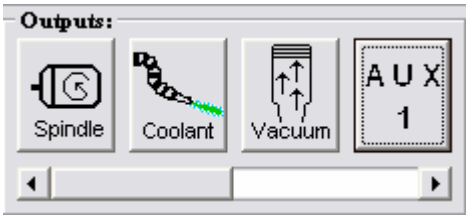
Along with the Toolbar, there are other graphical/icon related command keys.

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

OFF Position



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 - Toggle **On/Off** the **auxiliary 2 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) command 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.

EXAMPLE: G01 X1.2 Y1 F20

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**.

EXAMPLE: G02 X.5 Y2 J1 F10

G03
This code is used for **counterclockwise** cutting moves.
It is used in the same way as the **G02 command** described above.

EXAMPLE: G00 X3 Y2
G03 X4 Y2.2 I.4375 F10

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 her to call the subroutine a specified number of times.

EXAMPLE: 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 a **logical zero** or if the **parameter is true**.

Note: Can be used with **OptiStep-Plus** controller only.

EXAMPLE 1: G26 I1 #HOLESET

Will jump to **#HOLESET** when **input 1** is a **logical zero**.

EXAMPLE 2:

```
G28 (parts) = 5
G90          /ESTABLISH ABSOLUT
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.

EXAMPLE: 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**

G70

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

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 (default)**. **(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)**

EXAMPLE: G75

G79

Deep channel **CANNED CYCLE**. Mill a specified depth in a number of passes cutting in only one direction. The parameters include **X & Y** position of the hole, **Z depth**, and either **Q #** of passes or **K depth** per pass.

Z should be 0.050" above material before command.

EXAMPLE: G00 X2
G79 X3 Y1 Z.2 Q4

Will mill a channel 0.200" deep in 4 passes.

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.**

Z should be 0.050" above material before command.

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

Will drill a hole 0.150" deep.

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 1.2 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
G83 X.5 Y.5 Z1 K.25

Will drill a hole 1.000" deep in 4 passes.

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. **Z** retracts 0.050" from the bottom of each stroke.

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 1/2 second.

G90

This code Sets the **absolute programming mode**. In the **absolute 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. The **default programming mode is incremental**.

EXAMPLE: G90

G91

This code sets the **incremental programming mode**. In the **incremental 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 programming mode** is **incremental**.

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**. Operation **resumes** by pressing **<ENTER>**

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)**

M04

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

M05
This turns **<OFF>** the spindle motor **(control output #1 and #5)**

M06
This command selects the specified tool

EXAMPLE: M06 T2 **/selects tool #2**

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**.

M39
This **<CLOSES>** the chuck **(control output #6)**

M40
This **<OPENS>** the chuck **(control output #6)**

M66
This allows for jogging of the axis 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 and position will be restored to the exact location they were before the M66 command was encountered.

M97
This command pauses **part program execution** until a **logical zero** is detected at the specified input.

Note: Can be used with an **OptiStep Plus** controller only.

EXAMPLE: M97 I2

Will wait for **input 2** to become a **logical zero** 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**
G70	**Set inch programming (default)**
G71	**Set metric programming**
G74	**Sets single quadrant arc programming mode (default)**
G75	**Sets multiple quadrant arc programming mode**
G79	**Canned cycle for milling a deep channel**
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**
G91	**Set incremental programming mode (default)**
G92	**Set current tool position counters to value**
G95	**Chain to next part program**

Miscellaneous Functions (M-Codes)

M00	**Temporary stop**
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**
M39	**Chuck <CLOSE> (output #6)**
M40	**Chuck <OPEN> (output #6)**
M66	**Jogs without affecting counters**
M97	**Wait for true input state then continue**
M99	**Restart part program from beginning**

Tutorial

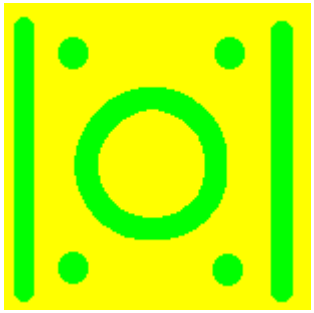
This section will show you how to write a part program using the built in CNC Editor and how to graphically simulate and manufacture the part using MillMaster Pro.

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 magnetic chuck or parallel jaw vise.

- A standard endmill cutting tool.

- A center drill and a twist drill

- 3" x 3" x 0.5" Stock (mild steel, plastic, 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

Fixturing Method = Magnetic Chuck

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 turning 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 problem. 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 re-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 going wrong during machining, **PRESS THE EMERGENCY STOP BUTTON** on your DriveRack.**

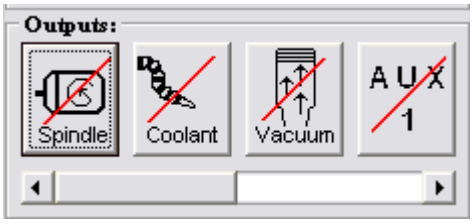
Customizing MillMaster Pro

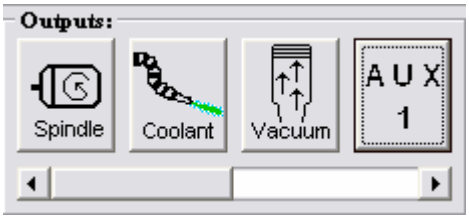
This section will instruct you on how to customize **MillMaster Pro for Windows**.

MillMaster Pro for Windows 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

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 for Windows 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 for Windows 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.

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
F2 – 60
F3 – 61
F4 – 62
F5 – 63
F6 – 64
F7 – 65
F8 – 66
F9 – 67
F10 – 68
1 – 2
2 – 3
3 – 4
4 – 5
5 – 6
6 – 7
7 – 8
8 – 9
9 – 10

0 – 11
A – 30
B – 48
C – 46
D – 32
E – 18
F – 33
G – 34
H – 35
I – 23
J – 36
K – 37
L – 38
M – 50
N – 49
O – 24
P – 25
Q – 16
R – 19

S – 31
T – 20
U – 22
V – 47
W – 17
X – 45
Y – 21
Z – 44
- – 12
= – 13
\ – 43
[– 26
] – 27
; – 39
“ – 40
– 28
, – 51
. – 52
/ – 53

Left ALT – 56
Left SHIFT – 42
Right SHIFT – 54
Caps Lock – 58
SCR LOCK – 70
NUM LOCK – 69
DEL – 83
INS – 82
TAB – 15
CTRL – 29
ESC – 1
SpaceBar – 57
BKSPC –14
Keypad + – 78
Keypad - – 74
Keypad * – 55

Keypad 1 – 79
Keypad 2 – 80
Keypad 3 – 81
Keypad 4 – 75
Keypad 5 – 76
Keypad 6 – 77
Keypad 7 – 71
Keypad 8 – 72
Keypad 9 – 73

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 **metacommands** available along with the **proper syntax: data** is a **decimal number** and **flag** is either **1** or **-1**. (**1** is used for **LEFT** or **DOWN** and -**1** is used for **RIGHT** or **UP**).

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

___ {Fixture}:CHUCK or CLAMPS	___ {Units}: INCH or MM	
___ {SLENGTH}:*data*	___ {SWIDTH}:*data*	___ {SHEIGTH}:*data*
___ {X Start}:*data*	___ {Y Start}:*data*	___ {Z Start}:*data*
___ {X Origin Zero}:*flag*	___ {Y Origin Zero}:*flag*	___ {Z Origin Zero}:*flag*

Use these commands in each part program to **automatically setup MillMaster Pro!**

Example:

```
/ {Fixture}: CLAMPS     {Units}: INCH
/ {SLENGTH}: 2.0000   {SWIDTH}: 2.0000     {SHEIGHT}: 0.2500
/ {X Start}: 0            {Y Start}: 0            {Z Start}: 0.05
/ {X Origin Zero}:1      {Y Origin Zero}:1      {Z Origin Zero}:-1
```

This set of **metacommands** will configure **MillMaster Pro** to **Inch mode** and set the **Fixturing method** to **Hold-down set**.

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**.

Appendix C

Wiring Diagram for External Cutting Speed Control Option

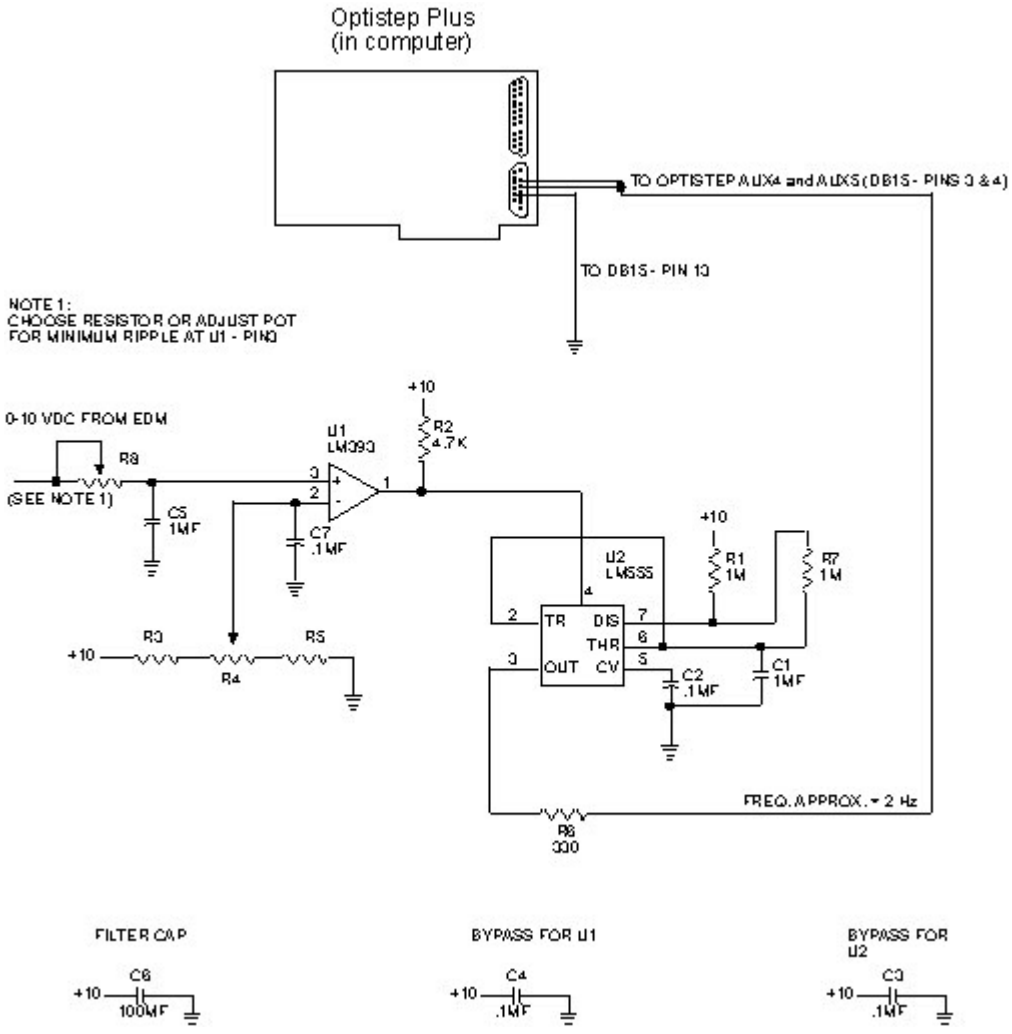
Connect an external pulse source, as in the **example below**, to the **OptiStep Plus controller** to use the **External Cutting Speed Control option**.

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 File HPMILL.MPS

The purpose of this **appendix** is to define each parameter in the **HPMILL.MPS file**. The **HPMILL.MPS file** must be in the same subdirectory as the **MMPRO.DEF file**. This is usually your **MillMaster Pro** home directory **(i.e. C:\Program Files\MillMaster Pro)**.

The **configuration file** for **MillMaster Pro** has changed from the previous **DOS Version**. The new configuration system is designed to allow the user to view and edit the file from within **MillMaster Pro For Windows** without cryptic DOS commands or opening any kind of external program and endlessly searching your hard drive for the correct file. Although, if the need arises to open this file, we have added detailed descriptions for each attribute of your software and of your machine for easy configuration and/or troubleshooting.

These attributes are displayed, in order, below.

Version = 1.4
Card address = 592
Card type = OptiStep plus
Machine type = 592
Max depth per pass = 1
Maximum unramped speed = 62
Maximum speed = 62
Acceleration rate = 62
Fast jog speed = 1000
Slow jog speed = 500
External sync active = 0
X screw pitch = 20
X steps in rev = 400
Y screw pitch = 20
Y steps in rev = 400
Z screw pitch = 20
Z steps in rev = 400
X plus = 75
X minus = 77
Y plus = 80
Y minus = 72
Z minus = 32
Z plus = 22
A minus = 51
A plus = 52
X origin = 1
Y origin = 1
Z origin = -1
X polarity = -1
Y polarity = 1

Z polarity = 1
Part dim x = 2
Part dim y = 2
Part dim z = .4
Machine scale = 1
Using chuck = 0
X start pos = 0
Y start pos = 0
Z start pos = 0
A start pos = 0
X limit pos = -2.522901
Y limit pos = -2.6278
Z limit pos = 3.0623
X limit dir = -1
Y limit dir = -1
Z limit dir = 1
Units = I
X will test = 0
Y will test = 0
Z will test = 0
Diag Start Speed = 0
Diag Speed Increment = 0
X Distance = 0
Y Distance = 0
Z Distance = 0
Tolerance = 0
Default tool = 0
Xbacklash = 0
Ybacklash = 0
Zbacklash = 0
Xbacklash Direction = 1
Ybacklash Direction = 1
Zbacklash Direction = 1
Encoder Tolerance = 6
X Encoder Active = 0
Y Encoder Active = 0
Z Encoder Active = 0
A Encoder Active = 0
Encoder Auto Correct = 0
Jog Tap Steps = 12
Feedhold Mode = 0
Multiplex = 0
OptiTracker Address = 0
Spindle Pulse Mode = 0
Axis #4 Type =
Steps Per Deg RPM = 1

Home Speed Pot Steps = 0
Oem Name = John Doe
Oem Company = Doe, Inc.
License Number = 0

Output number = 1
Output name = preheat
Delay after on = 5
Delay after off = 5

***** Each Output utilizes 4 lines for its particular attributes. There are 5 programmable Outputs. *****

Tool Number = 1
Tool Name = drill
Tool Picture Filename = c:\mmpro\pics\drill.bmp
Tool Diameter = .25
Tool Length = 0
X Tool Offset = 1.1
Y Tool Offset = 1.2

***** Each Tool utilizes 7 lines for its particular attributes.***

**** 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 for Windows 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: QuickPhase, OptiStep, OptiStep plus

This parameter specifies which card type is 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:

$$\text{Steps/Sec} \setminus \frac{\text{Screw Pitch x \#Steps/Rev}}{60}$$

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)

X, Y, Z Origin

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

X, Y, Z Polarity

Valid Range: NORMAL or 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 part dim

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

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 Limit Position

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

X, Y, Z Limit Direction

This parameter specifies the 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..

Origin Visible

X, Y, Z Will Test

Valid Range: 1 or -1

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

Diag Start Speed

This parameter specifies the **Start Speed** of the machine during the **Diagnostics Testing**.

Diag Speed Increment

This parameter specifies the **Increase In Speed Per Test**.

X, Y, Z Distance

This parameter specifies how far to travel for **Diagnostic Testing**.

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.

Xbacklash,Ybacklash, and Zbacklash

Valid Range: 0 to 0.9999

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

Xbacklash, Ybacklash, and Zbacklash 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"**.

Xencoder,Yencoder, Zencoder, Aencoder 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: Yes or 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(TM) card** in the computer. This parameter is ignored if set to 0.

Spindle Pulse Mode

Valid Range: YES or **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.

Output Number

Valid Range: 1-5

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. (###.#)

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.

X Tool Offset

This parameter specifies the tool's X offset.

Y Tool Offset

This parameter specifies the tool's Y offset

MN400 Controller Information

The MN400 controller card is an external motion control card that uses the serial port to communicate with the PC. This allows for the use of the MN400 controller board on any computer that has a serial port. With this controller, you can now 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:** After connecting the MN400 to the serial port on the back of your machine and choosing MN400 in the Machine Parameters, you are unable to jog, switch the cable to the other available serial port and re-try.

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

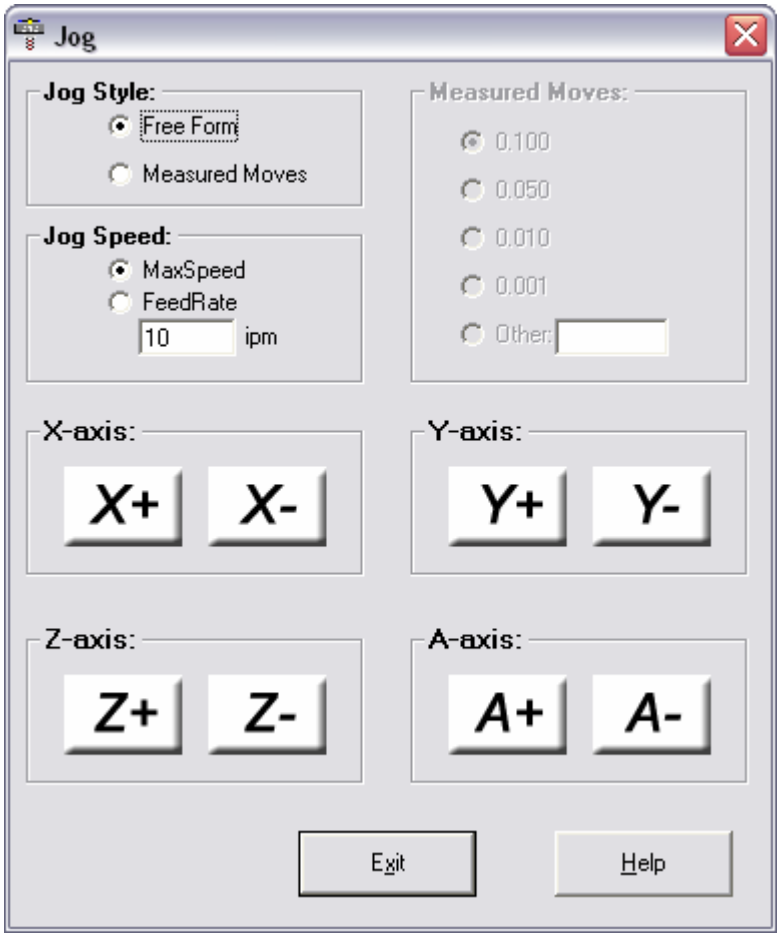
Although the operation of the MN400 is extremely close to the operation of the Instep Library, the MN400 does have some small differences. When running a program with the MN400, you do not experience the "Lockout" that occurs when using Instep and either the OptiStep or QuickPhase cards. The MN400 is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

When the MN400 motion controller board is selected for use with MillMaster Pro, the jogging feature within the MN400 must be turned ON and OFF to operate correctly. When you choose Jog Tool from the Control menu and MN400 is selected, the software will send the command to set Jog mode.

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.

For quick help while in the MN400 jog window, click the button in the bottom right corner with that say 'Help' on it. This will display a message window giving you basic instructions for jogging with the MN400 controller.