Mach4 CNC Controller
Mill Operations Guide

Table of Contents

Introduction

The purpose of this manual is to teach the basic operation and functionality of the Mach4 CNC control software. This manual is based on the core Mach4 product using the standard screen set. The appearance of the interface may vary depending on the machine builder. Mach4 includes a screen designer that allows manufactures and operators to create custom screen layouts for their specific need.

More information on the screen designer can be found in the customization manual. Programming and configuration also have their own respective manuals which must be read before setting up and running a Mach4 controlled machine.

Note: Operators should be familiar with general CNC and machining practices before operating any CNC machine. Some great resources for additional learning at the machsupport.com forums, other user groups/forums, and books such as CNC Programming Handbook and CNC Control Setup for Milling and Turning, both by Peter Smid and Programming of CNC Machines by Ken Evans

Before You Begin

Any machine tool is potentially dangerous. Computer controlled machines are potentially more dangerous than manual ones because, for example, a computer is quite prepared to rotate an 8" unbalanced cast iron four-jaw chuck at 3000 rpm, to plunge a panel-fielding router cutter deep into a piece of oak, or to mill away the clamps holding your work to the table. Because we do not know the details of your machine or local conditions we can accept no responsibility for the performance of any machine or any damage or injury caused by its use. It is your responsibility to ensure that you understand the implications of what you design and build and to comply with any legislation and codes of practice applicable to your country or state. If you are in any doubt, be sure to seek guidance from a professionally qualified expert rather than risk injury to yourself or to others.

What is Mach4?

Mach4 is software that operates on a personal computer to create a powerful and cost efficient CNC controller. It makes up one small piece of a computer numerical control (CNC) machine. Machines can range from basic mills and lathes to wood routers, plasma cutters, multi axis machining centers, quilting machines, anything requiring motion control. The system is capable of interpreting multiple programming languages, the default and most common being G code, to provide instructions for machine movement and other functions. These instructions are passed to an external motion device, which in turn controls all the inputs and output signals and motion.

Mach4 is designed to be flexible and adaptable to a wide variety of machines. Part of this flexibility is the ability for hardware and software developers to create addons or plugins for Mach4 to expand its capabilities. Addons are small programs installed into the Mach4 directory that give Mach the ability to talk to hardware devices such as motion controllers and pendants, communicate with other software, add additional wizards or conversational machining functions, or anything a developer can dream up. Addons to Mach4 are so diverse it would be impossible to cover them in this manual. The developer should provide detailed information on the installation, configuration and use of their addon or plugin.

What is an external motion device? Originally, Mach-series software only worked with the parallel port (via the parallel port driver), which was a standard port on every PC. Technologies have advanced over time, and not only is the parallel port becoming obsolete, but the Windows codebase has changed to the point where it is technical impossible for the parallel port driver to work. An external motion device is a piece of hardware that is an improvement over the parallel port. It enables a PC running Mach3/Mach4 to control outputs and read inputs. They typically communicate with the PC via an Ethernet or USB connection (but are not limited to those two means of communication). In order to control a machine using an external motion device, the developer of the hardware must write a plugin (driver) for that specific device, so no standard USB-to-parallel

How Do I Get My Drawing Into Mach4?

Mach4 cannot directly interpret drawings. Mach4 is motion (motor) control software.

General process

1) Part is drawn in a computer aided design (CAD) program.

2) Resulting file transferred into a computer aided manufacturing (CAM) program to create the tool paths for machining.

3) The CAM program uses a post processor to generate a formatted G code program from the tool paths created in the software for Mach

4) The G code program is then loaded and executed by Mach4.

*This is only one method of creating a G code program, they can also be manually written or created with a wizard or conversational program. The wizards in Mach are essentially simple graphical CAM programs designed to easily create programs for basic shapes and tasks.

Typical CNC Cutting Process

Create CAD model of part → Use CAM software to generate G code program from CAD model → Machine safety check → Turn on machine and control power

Reference all active axes ← Jog machine away from home switches ← Enable system ← Start Mach4 and select appropriate profile

Load work piece into fixture/vise → Load edge finder into spindle → Find work piece zero location and set fixture offset → Load tools and set tool height offsets

Cut part ← Dry run program above part ← Check tool path display for any obvious errors ← Load program

Measure critical feature BEFORE removing part if possible → Make required adjustments and re-run if necessary

Screen Layout and Controls

The standard Mach4 screen is designed to be easy to navigate for first time users. Figure 2-1 shows the layout of the main page. This is the screen that will be loaded when Mach4Gui.exe is launched and the Mach4Mill profile is selected. Each part will be discussed in more detail in separate sections.

Figure 2-1: Mach4 screen layout.

- **Menu Bar:** Contains drop down menus similar to many programs. Options here range from loading programs, configuration of Mach4 and its plug-ins and exiting the program.
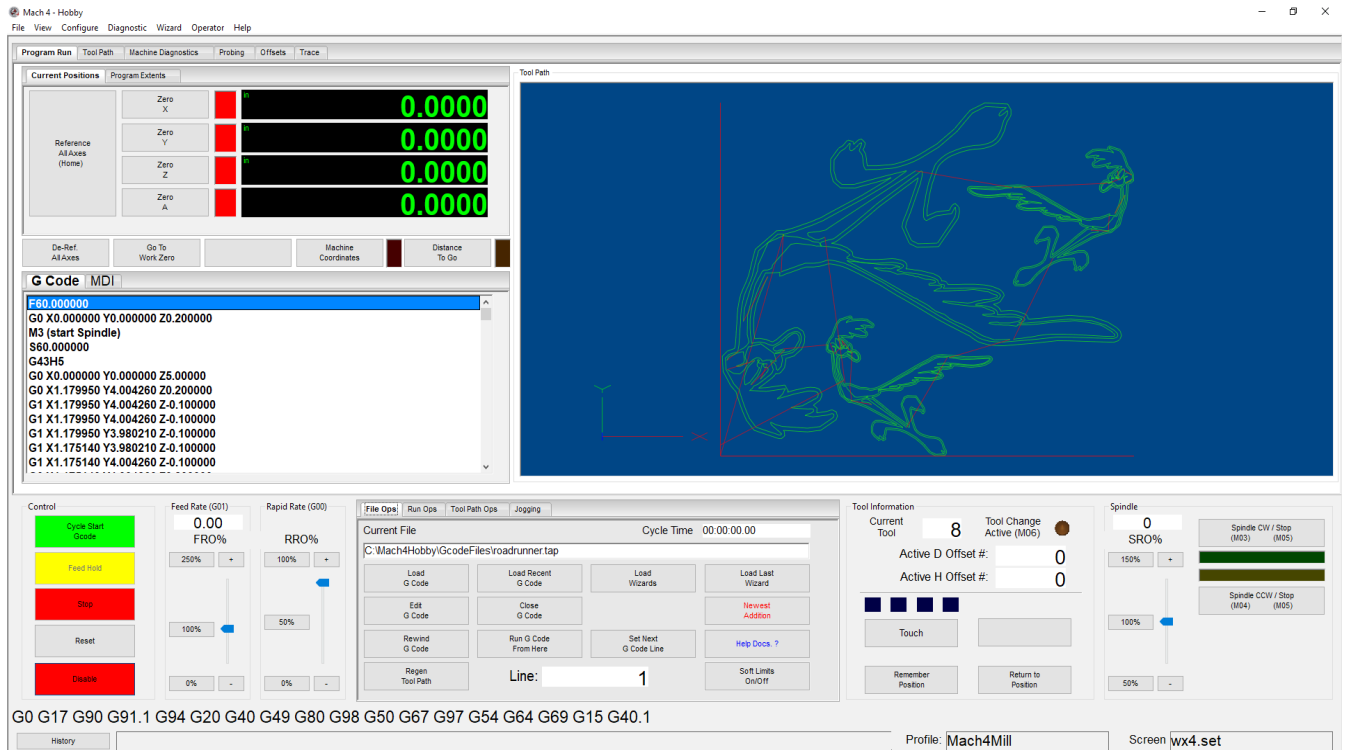
- **Operation Mode Tabs:** These tabs select the operation mode of Mach4 and change the layout of the upper half of the screen based on the selection made.

- **Program Execution Controls:** Cycle Start, Feed Hold and Stop buttons to control the execution of loaded G Code programs or MDI programming.

- **Reset:** The Reset button returns the control to its default state and rewinds the program.

- **Enable/Disable:** The enable button does just that, it enables Mach4. The control must be disabled before any configuration changes can be made.

- **Status Bar:** Shows the current state of the modal G Code groups.

- **Message History:** Displays a window that shows the history of messages displayed in the message bar and recent button pushes.

- **Message Bar:** Displays Mach4 errors and comments from a program or script.

- **Operator Control Tabs:** Display a variety of buttons for control of loading and editing programs, running of programs, tool path display and jogging.

- **Current Tool Data:** Displays current tool number and offset data.

- **Active Profile:** Displays the name of the current profile.

- **Override Controls:** Provide control over the feed rate, rapid and spindle speed overrides.

- **Spindle Control:** Spindle on/off button.

Menu Bar

The menu bar in Mach 4 consists of drop downs that provide access to, among other things, configuration settings, diagnostics, and offset information.

File Menu

The file menu contains commands for opening and closing G code files and exiting the program (see figure 2-2).
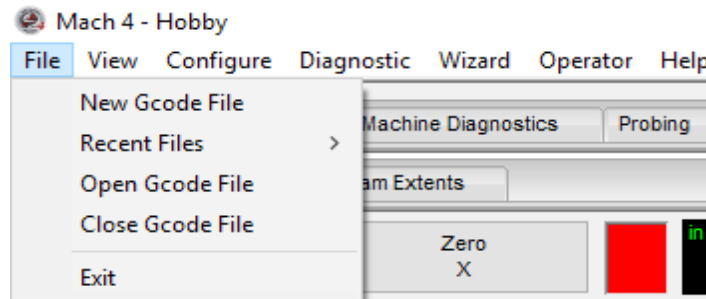
Figure 2-2: File menu

- **New Gcode File:** Selecting this option opens the G code editor with a new blank file. A program can then be manually typed in by the user and saved for later execution.

- **Recent Files:** Display a list of the most recently opened G code files. Select one from the list to load it into Mach4 for execution.

- **Open Gcode File:** Opens the "Select File" dialog, figure 2-3. The user can then navigate to the desired G code file to select and open it in Mach4.
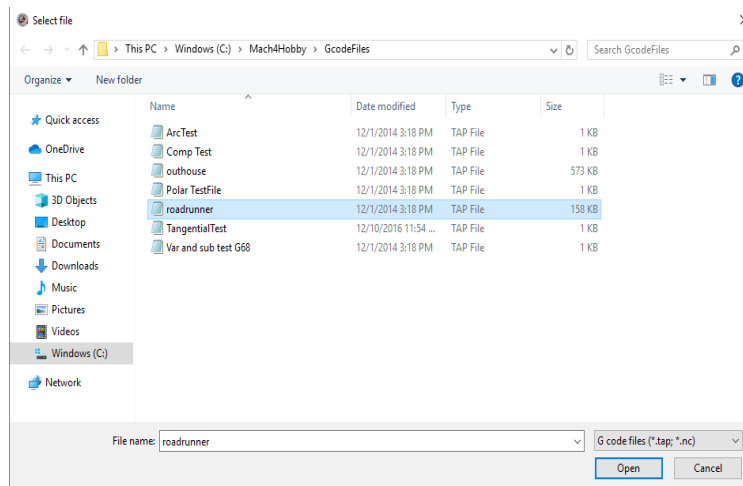
Figure 2-3: Select file

- **Close G code File:** Closes the currently loaded G code file.

- **Exit:** Exits Mach4.

View Menu

The view menu, figure 2-4, contains controls for changing the current display.

Figure 2-4: View menu

- **Load Screen:** Opens a dialog that allows the user to choose a new screen, or display.

- **Full Screen:** Puts Mach4 into full screen mode. Mach4's display will be enlarged to cover the entire display.

- **Fixture Offsets:** Displays the fixture offset table, more information is available in the fixture offset section of this manual.

- **Tool Table:** Displays the tool offset table, more information is available in the tool offset section of this manual.
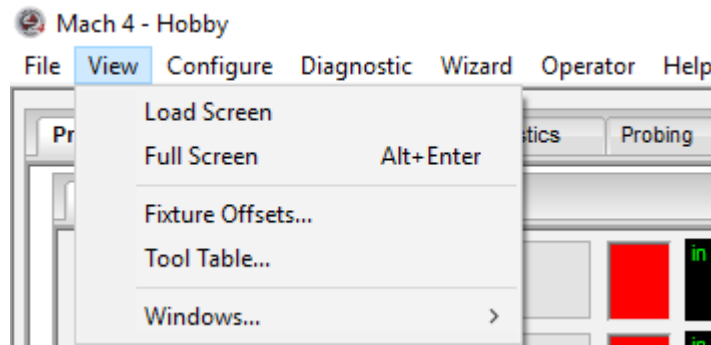
- **Windows:** Provides options for what toolbars are displayed when in the screen editor mode. The screen editor and its associated functions are explained in the customization manual.

Configure Menu

The configure menu, figure 2-5, provides access to Mach and Plugins configurations. Configuration settings are disabled when Mach4 is enabled. To allow access to these settings, first disable the controller. More detailed information on these settings can be found in the Mach4 CNC Controller Configuration Manual.

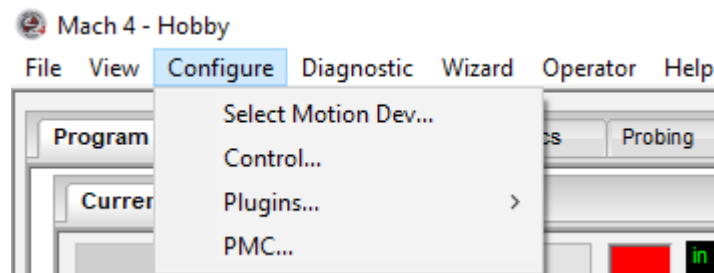*Figure 2-5: Configure menu*

- **Select Motion Dev:** Use this menu option to select the external motion device that is to be used.

- **Control:** Opens a window that displays all of Mach4's configuration settings. This is where all motor and input and output settings are entered.

- **Plugins:** Opens a window that lists all installed plugins and provides access to each plugin's configuration settings.

- **PMC:** Programmable Machine Control Is a later logic style control for Mach4 interface

Diagnostic Menu

The diagnostic menu, figure 2-6, provides functions for logging and viewing the status of inputs and outputs. Any installed plugins may have their own diagnostic, varying the contents of this menu based on the machine configuration. Some of the options shown may not be discussed in this manual as they are 3rd party plugin diagnostics. The menu items should be covered in detail in the manual provided by the developer of the plugin.
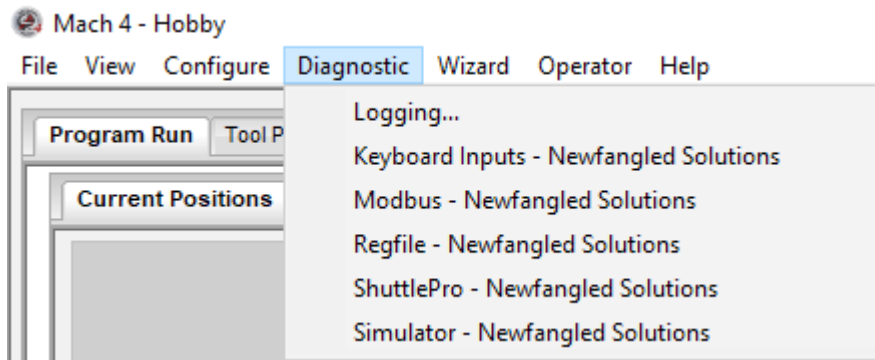
Figure 2-6: Diagnostic menu

- **Logging:** Opens the logging facility. This is handy error checking tool that logs events in Mach4 in real time. The machine builder can choose what gets logged, so each machine may log events differently.

- **Modbus:** Opens a window that shows the current state of Modbus communications.

- **Regfile:** Opens a window that displays the current value of a variety of registers and variables in Mach4. The contents and range of what is displayed here will vary with each machine build.

Wizard Menu

The wizard menu, figure 2-7, provides access to the pick wizard window (figure 2-8). Any installed wizards can be found here. A wizard is a small program that allows a user to easily generate G code for common or moderately complex machining processes. Wizards are their capabilities can be as varied as the programmers that write them.
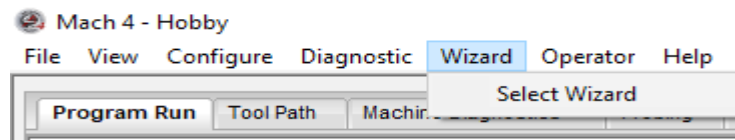
Figure 2-7: Wizard menu
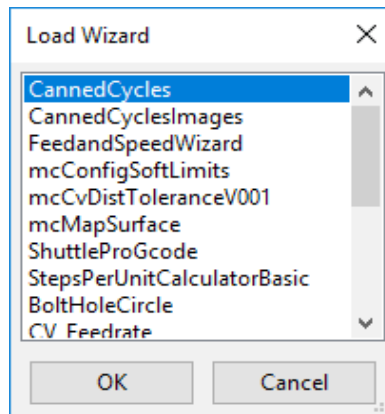
Figure 2-8: Load wizard window

Operator Menu

The operator menu, figure 2-9, provides the operator with options for editing the Mach4 interface and Lua scripting. The screen editor and an overview of Lua can be found in the customization manual.
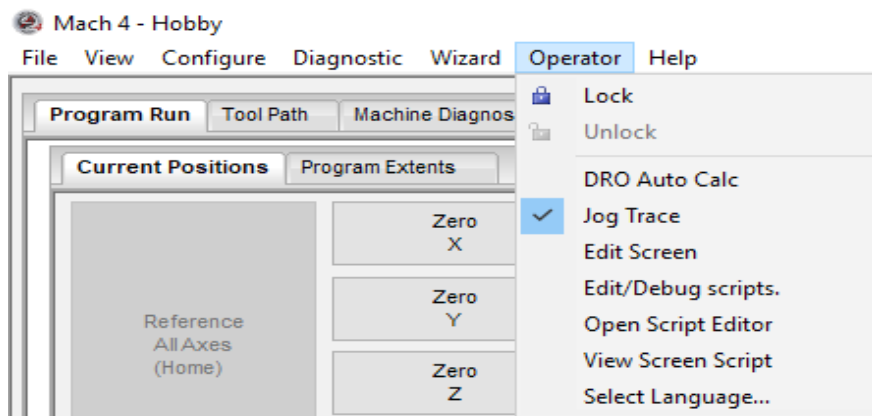
Figure 2-9: Operator menu

- **Lock:** Locks the screen from being edited, a password is required to unlock.

- **Unlock:** Unlocks the screen for editing, a password is required.

- **DRO Auto Calc:** A calculator for a DRO by double clicking in the box to perform any mathematically calculation for values of the DRO

- **Jog Trace:** An onscreen path, showing jog movements

- **Edit Screen:** switches Mach4 to the screen editing mode, see figure 2-10.

- **Edit/Debug Scripts:** Opens the Lua editor for editing scripts saved as macros, such as custom M codes.

11
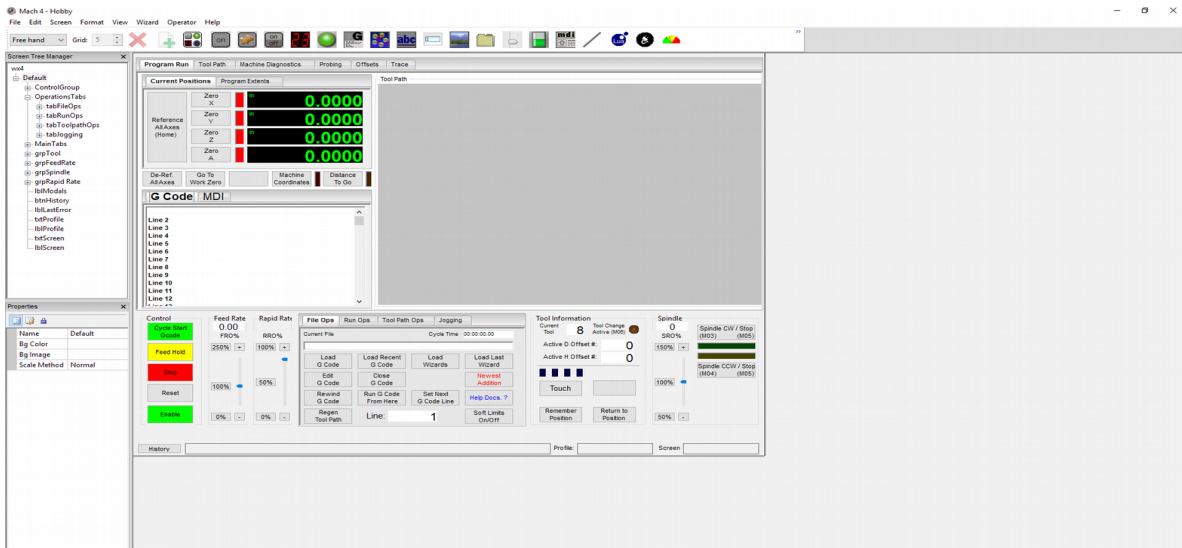
Figure 2-10: Edit Screen

Help Menu

The help menu, figure 2-11, is where the operator will find the "About" page, figure 2-12.
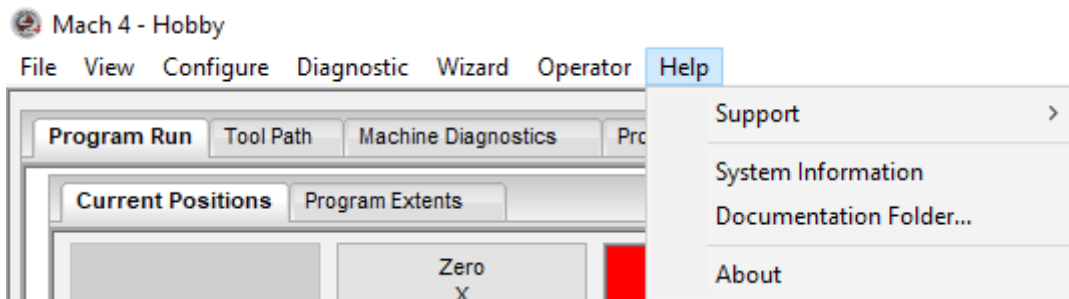
Figure 2-11: Help menu

- **About:** Opens a window that displays information about Mach4 and the computer it is installed on. The about screen indicates what version of Mach4 is installed, Lite, Hobby, Industrial, etc. Next is the version number, followed by the build number. The PCID is required for licensing. The PCID is based on the configuration of the computer Mach4 is installed and the license will be locked to that PCID / computer system. Copy the PCID number by clicking on the "Copy ID to Clipboard" button. Use the "Load License File" button to load a license file received from Artsoft.
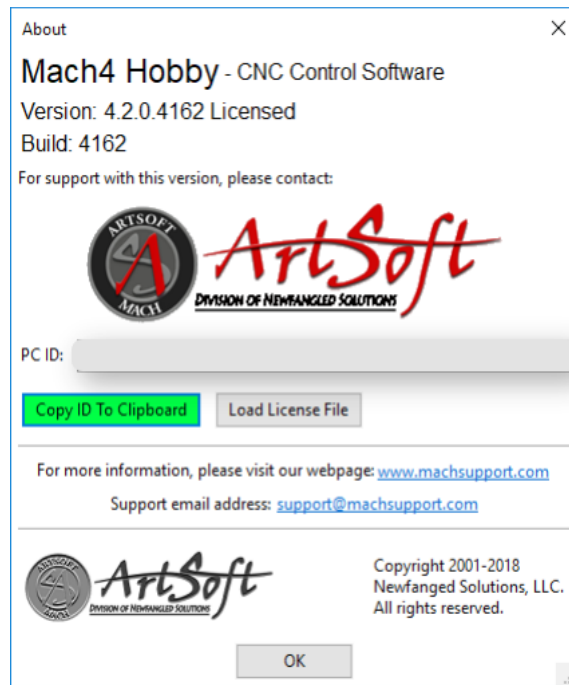
Figure 2-12: About Window

Operation Mode Tabs

The operation mode tabs allow the user to navigate between program run, manual data input (MDI) and diagnostics. Click on a tab to change the view in Mach4 and change modes.

Program Run Tab

The program run tab, figure 2-13, is the default on startup. This tab displays the current G code file, a plot of the tool path and current position.

Figure 2-13: Program run tab

- **Operation Mode Selection:** These are the tab markers, select the desired tab by clicking on its name.

- **G Code Window:** Displays the current G code file.

- **Go to Zero:** Moves machine to X0, Y0 in the active fixture offset.

- **Position Display Mode Selection:** Changes the contents of the position display. The "To Go" button changes the display to show the distance to go to the programmed point. The "Machine Coordinates" button changes the display to show the machine coordinate system. The LED above each button will light up when the button is activated. If neither LED is on the display shows the current position in the active fixture offset.

- **Reference All:** References/Homes all axes in the order set on the Homing/Soft limits tab in the Mach configuration. The axes will move to the home switches and machine zero set.

- **Zero Axis:** The zero axis buttons will set the current position to zero in the active fixture offset. This sets the active fixture offset and will over write any data that was entered previously.

- **Reference Status LEDs:** When an axis is referenced or homed its status LED will change from red to green.

- **Position Display:** The digital readouts (DROs) display position data as selected by the display mode buttons.

- **Tool Path Window:** The tool path window displays the programmed path for the currently loaded G code file. Below it are some basic controls for the display.

- **Display Mode:** This button toggles the display of the soft limits in the tool path window. Showing the extents of the soft limits can provide visual reference of where the program is in the machine travel and if it fits.

MDI Tab

Manual Data Input, or MDI, is a mode that allows the user to manually type in blocks of G code to be executed. A block is a single line of G code containing multiple words, each started with a letter or symbol. The code is executed by pressing the cycle start button.
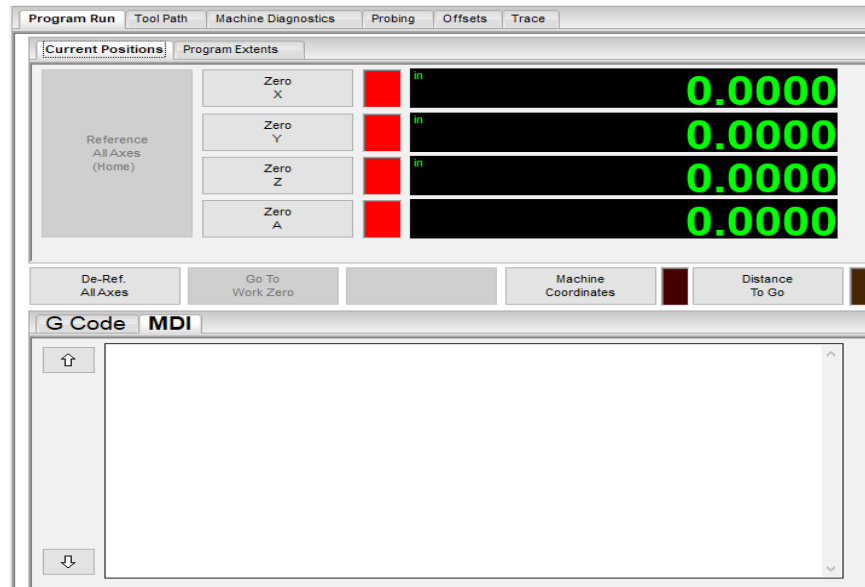
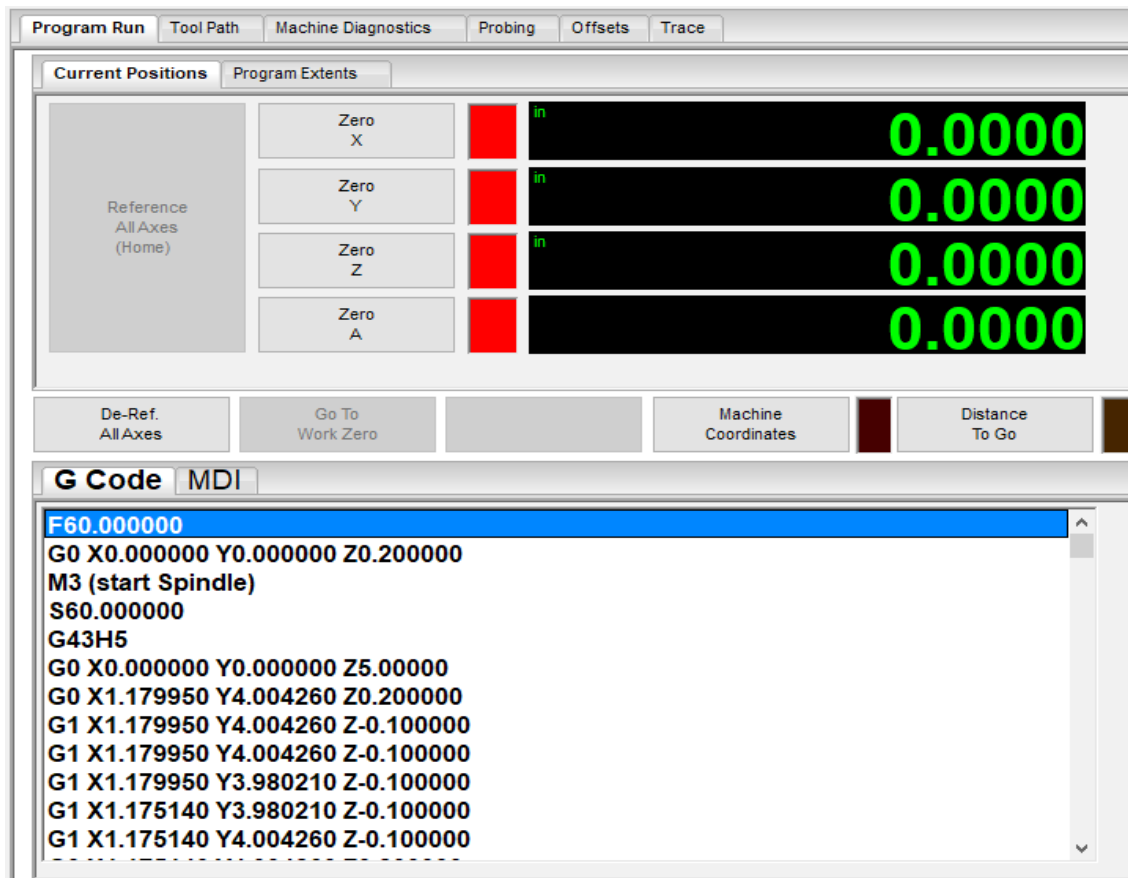*Figure 2-14: MDI tab*

- **Operation Mode Selection:** These are the tab markers. Select the desired tab by clicking on its name.

- **Reference All:** References/Homes all axes in the order set on the Homing/Soft Limits tab in the Mach configuration. The axes will move to the home switches and machine zero set.

- **Zero Axis:** The zero axis buttons will set the current position to zero in the active fixture offset. This sets the active fixture offset and will over write any data that was entered previously.

- **Reference Status LEDs:** When an axis is referenced or homed its status LED will change from red to green.

- **De-Ref All Axes:** This button sets all axes to an un-referenced state.

- **Manual Data Input Window:** Manually input G code to be executed in this window. Multiple lines can be entered. The up and down arrows to the left scroll through the history of executed code.

- **Position Display Mode Selection:** Changes the contents of the position display. The "To Go" button changes the display to show the distance to go to the programmed point. The "Machine Coordinates" button changes the display to show the machine coordinate system. The LED

above each button will light up when the button is activated. If neither LED is on the display shows the current position in the active fixture offset.

- **Position Display:** The digital readouts (DROs) display position data as selected by the display mode buttons.

Tool Path Tab

The tool path tab, figure 2-15, is a simple display with a tool path window dominating the screen.

Figure 2-15: Tool path tab

- **Operation Mode Selection:** These are the tab markers. Select the desired tab by clicking on its name.

- **G Code Window:** Displays the current G code file.

- **Position Display:** The digital readouts (DROs) display position data as selected by the display mode buttons. There are not position display mode buttons on this tab, the display will be changed based on the state of the buttons on the Program Run or MDI pages.

- **Tool Path Window:** The tool path window displays the programmed path for the currently loaded G code file.

Diagnostic Tab

In the diagnostic tab, figure 2-16, the operator can view the status of inputs and outputs as well as individual axis homing.

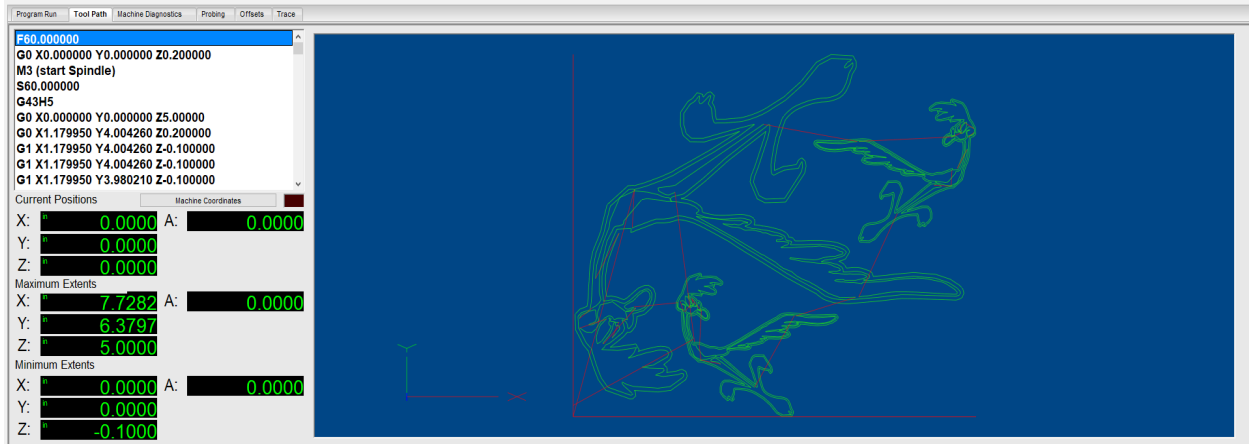*Figure 2-16: Diagnostic tab*

16

- **Operation Mode Selection:** These are the tab markers. Select the desired tab by clicking on its name.

- **Input Status LEDs:** LEDs show the status of a variety of inputs to Mach4.

- **Output Status LEDs:** LEDs show the status of a variety of outputs from Mach4.

- **Axis Homing:** Buttons to home individual axes and status LEDs to show the reference status.

- **DRO's:**

- **Spindle Movements:**

Operator Control Tabs

These tabs give the operator control of G code file loading and editing, program execution option, tool path and jogging.

File Ops Tab

The file operation tab, figure 2-17, displays controls to affect the current G code file. Here the operator can select which file to load, where to start program execution and edit the G code file.
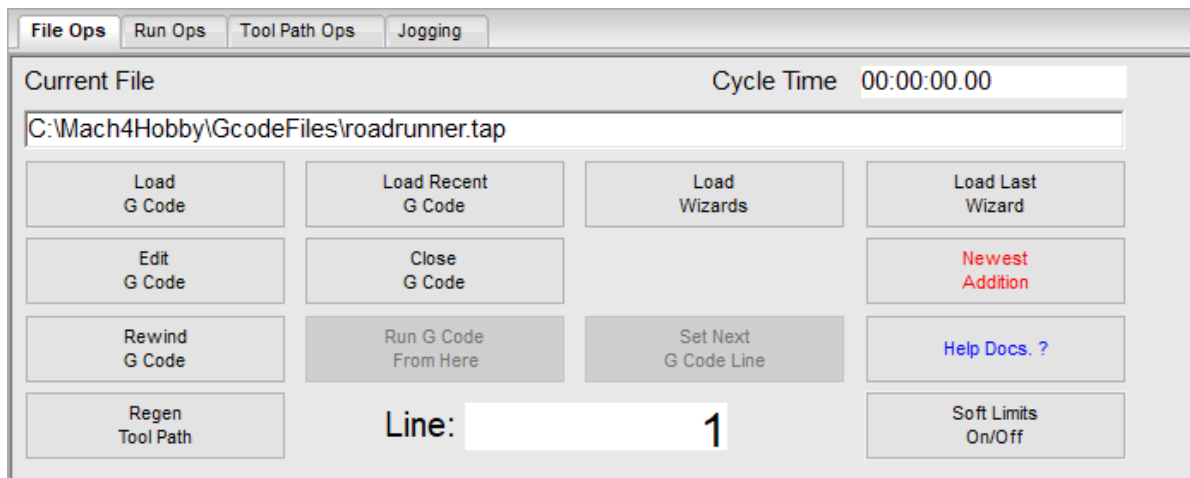
Figure 2-17: File ops tab

- **Current G Code File:** Displays the file path and name of the current G code file.

- **Wizard Controls:** The "Load Wizards" button opens a window that displays all the installed wizards available to the user, select one and press "OK" to run the wizard. "Load Last Wizard" loads the last wizard that was run.

- **Regenerate Tool Path:** This button regenerates or refreshes the display. When pushed the tool path will be recalculated with any changes.

- **Operator Control Selection:** These are the tab markers. Select the desired tab by clicking on its name.

- **Edit G Code:** The edit button opens the current G code file in the G code editor. The operator can then make changes to the program. When the editor is closed the G code file is reloaded into Mach4.

17

- **Load Recent G Code:** Opens a window that displays a list of the most recently loaded G code files. A file can be selected to be loaded into Mach4.

- **Close G Code:** Closes the current G code file.

- **Load G Code:** Opens a window to search for a G code file to be loaded into Mach4.

- **Run From Here:** Opens a wizard to aid in starting program execution from anywhere in the program. This function is described in greater detail later in this manual.

- **Line:** Displays the current line in the G code file. This is the line being currently executed by the controller.

- **Set Next Line:** Sets the line to start program execution from.

Run Ops Tab

The run operations tab, figure 2-18, gives the operator control of how a G code file is executed. Options such as single block, block delete, and optional stop can be turned on or off.

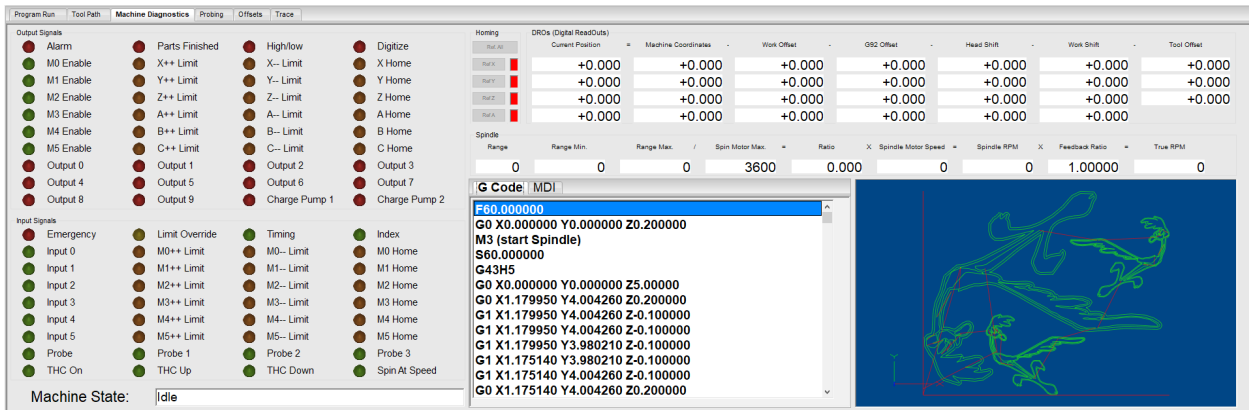*Figure 2-18: Run ops tab*

- **Operator Control Selection:** These are the tab markers, select the desired tab by clicking on its name.

- **Single Block:** Activates single block mode, the LED will light when active. In single block mode Mach4 will execute only one line, or block, of code for every press of the cycle start button.

- **Flood Coolant:** Turns on the flood coolant output.

- **Optional Stop:** When activated program execution will pause at M01 in the program.

- **Block Skip:** It is possible to skip lines, or blocks, or a program using the block buttons. When activated any block starting with a slash, "/", will be ignored by Mach4.

- **Mist Coolant:** Turn on the mist coolant output.

Tool Path Ops Tab

The tool path operation tab, figure 2-19, contains tools for controlling the tool path display.

18
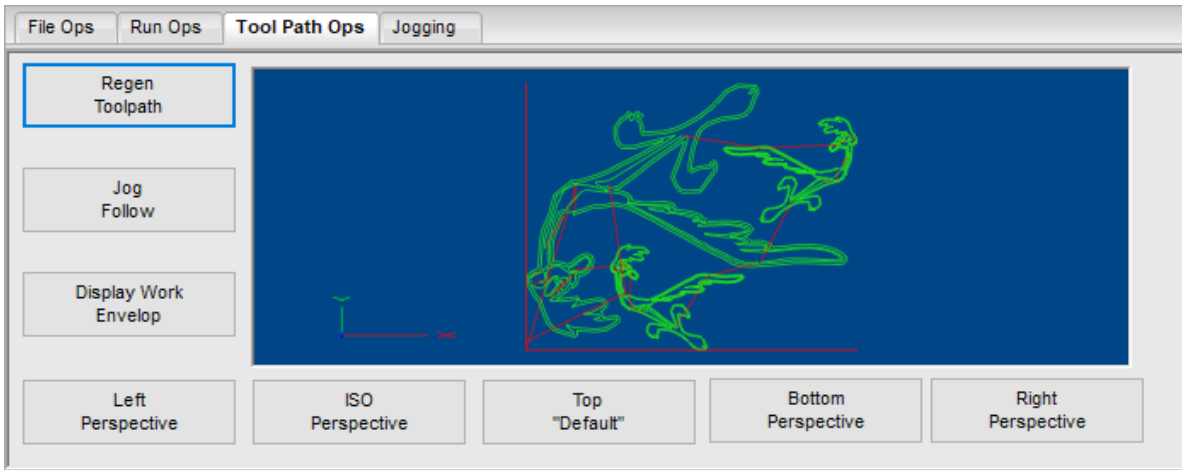
Figure 2-19: Tool path ops tab

- **Jog Follow:** This button toggles how motion is displayed in the tool path window. With jog follow off the tool path stays stationary and the red lines showing the tool position moves around. When on, the tool position stays centered on the window and the tool path moves around. For large parts this mode can be used to view the tool path in more detail.

- **Operator Control Selection:** These are the tab markers. Select the desired tab by clicking on its name.

- **Regenerate Toolpath:** Reloads the program and recalculates the tool path.

- **Display Mode:** Toggles the display of the soft limits.

- **Jog Follow:** When active the center of the tool is locked to the center of the tool path display. The display will follow tool movement.

- **Perspectives:** Several buttons allow for different perspectives of the toolpath including the default Top view.

Jogging Tab

The jogging tab, figure 2-20, displays controls for jogging the machine's axes and changing the jog mode.

Figure 2-20: Jogging Tab

19

- **Axes Limit Override:** If a limit switched is tripped motion is disabled. This can cause a problem trying to move the machine off the limit. The limit override essentially ignores the limit switches until they are deactivated. Extreme caution should be used when activating the limit override, it is possible to overrun the switch and cause damage to the machine.
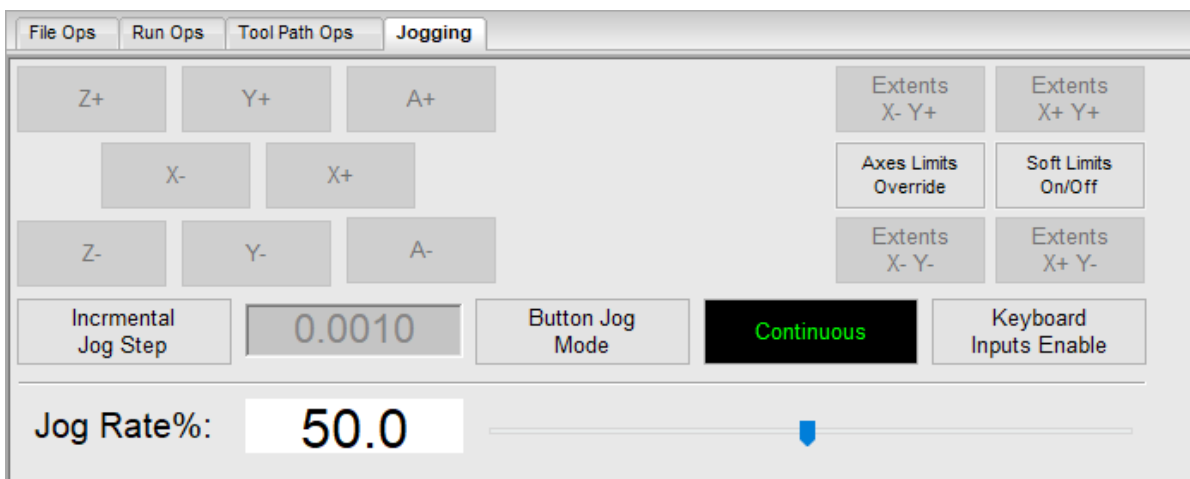
- **Soft Limits:** Activates or deactivates the soft limits. Soft limits are exactly as the same implies, software limits to contain machine motions. These are usually set just inside of the actual limit switches. Soft limits are set up in the Homing/Soft limits tab in the Mach configuration.

- **Operator Control Selection:** These are the tab markers. Select the desired tab by clicking on its name.

- **Jog Buttons:** Buttons to jog X, Y, Z and A axes.

- **Cycle Incremental Jog Distance:** In incremental jog mode each press of an axis jog button will move the machine on a specified distance, this button cycles through the distances defined in the general configuration. The current step distance in shown in the adjacent DRO.

- **Toggle Jog Mode:** Toggles between continuous and incremental jog modes.

- **Jog Rate:** Sets the velocity for continuous jog as a percentage of the machine's max velocity.

- **Toggle Keyboard Jog:** Turns on/off the ability to jog using the keyboard.
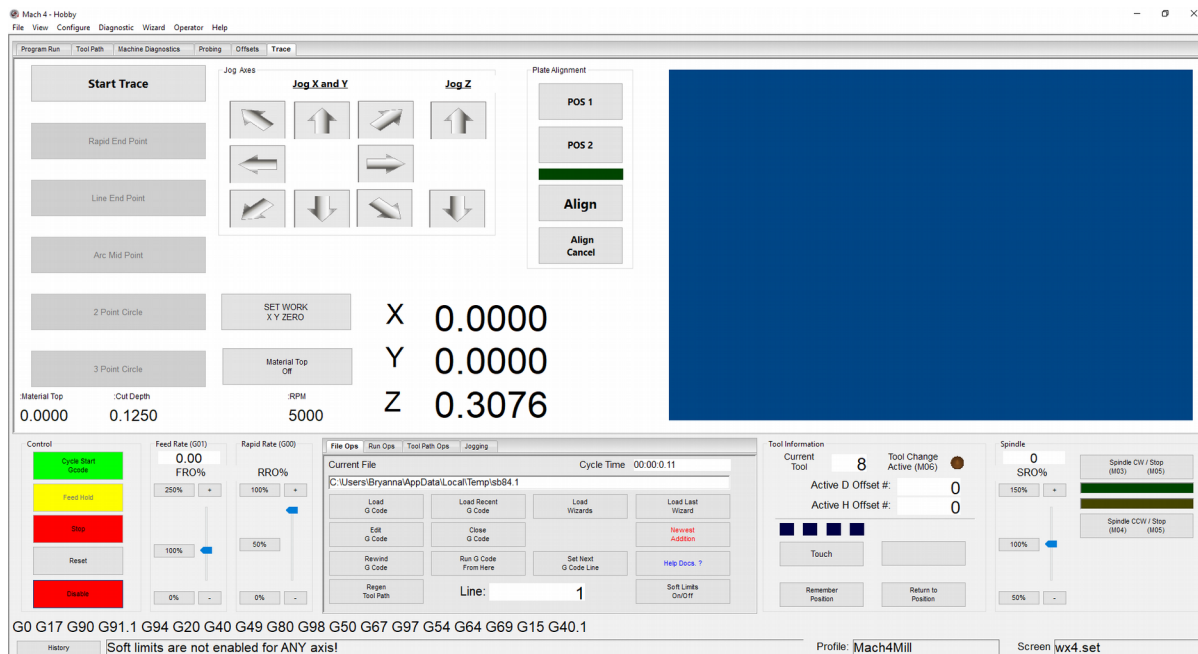
2.2.5   Trace tab

Figure 2-21: Trace Tab

Trace tab gives users the opportunity to create a G Code file using an existing template, with or without preexisting dimensions set. By jogging axes to the desired shape using the on screen buttons.

- **Start Trace:** To begin jogging enable start trace. When End Trace is selected a prompt will appear to close profile with a line, to enclose the shape just made

- **Rapid End Point:** Rapid end point allows for movement to a different area of the screen without cutting the material

- **Line End Point:** Straight movements from the current point to the selected point, in a direct cut

- **Arc Mid Point:** After selecting the Arc Mid Point the button will change to Arc End Point to allow for a complete arc to be traced and finished

- **2 Point Circle:** Selecting the center of the circle and one additional point along the outside of the circle

- **3 Point Circle:** Creating a circle by selecting three points along the radius of the existing circle

- **Plate Alignment:** Adjusting for an uneven material in the workspace to allow for the proper alignment of the trace tool, regardless of the materials position on the machine

- For more information and description of how to use the trace tab, go to our official YouTube page Mach Support. Direct link: https://www.youtube.com/watch?v=t-4hqKA68jI&t=333s

Getting Started

How to start Mach4 will vary depending on the installation. In all cases there will be a Mach4GUI.exe located in the Mach4 folder usually located on the C drive. Some machines may have a shortcut to this executable on the desktop. When Mach4 is started the user will be presented with an option to choose a profile, figure 3-1.
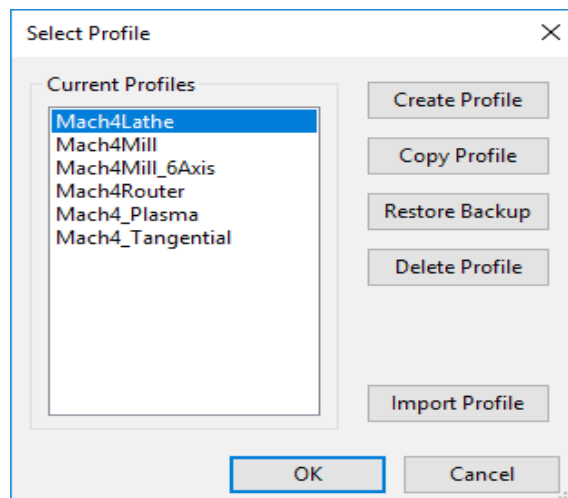
Figure 3-1: Select profile window

In the select profile window, the user can choose a profile to open, create a new profile from scratch, copy an existing profile, restore a profile or delete a profile. Once a machine is set up, using a shortcut to run a specific profile is often convenient. Create a shortcut to Mach4GUI.exe on the desktop. Right click on the shortcut and select properties. In the properties dialog, figure 3-2, the Target is the file that will be run. This should read the location of Mach4GUI.exe. Add "/p profilename" to the end of this line, replacing "profilename" with the name of the desired profile. The shortcut will open Mach4 into the named profile.

Figure 3-2: Shortcut properties dialog

Enabling the System

Once open the user will be presented with Mach4 in the disabled state. The system must be enabled before any operation of the machine can take place. Enable the controller by pressing the flashing green enable button in the lower left corner of the screen. Figures 3-3 and 3-4 show Mach4 in its disabled and enabled states, respectively.

22
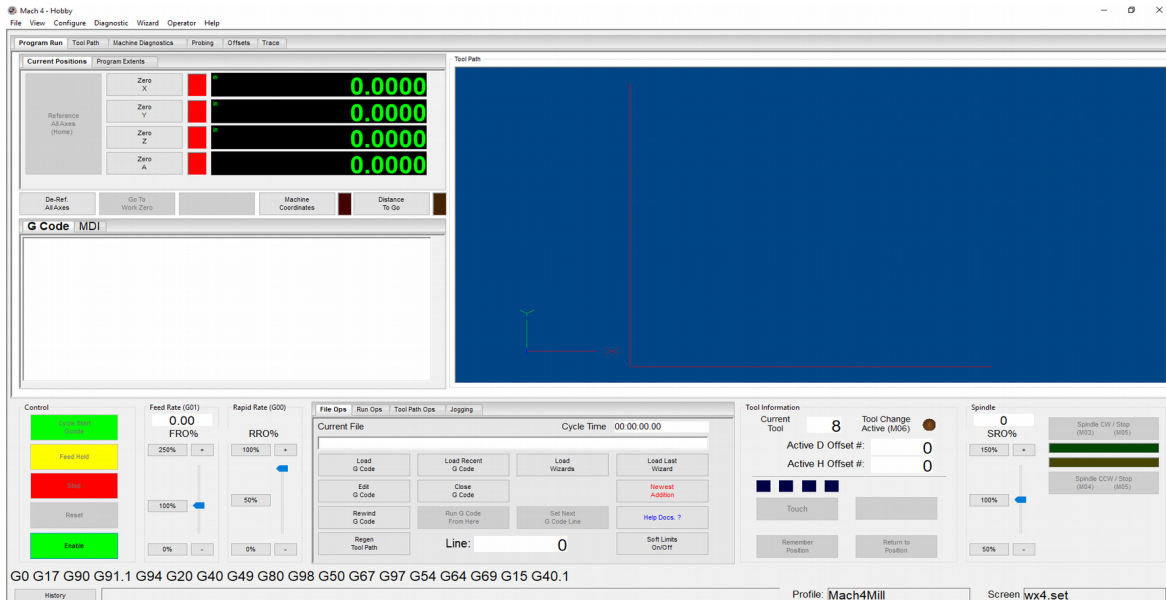
Figure 3-3: Mach4 main screen in disabled state

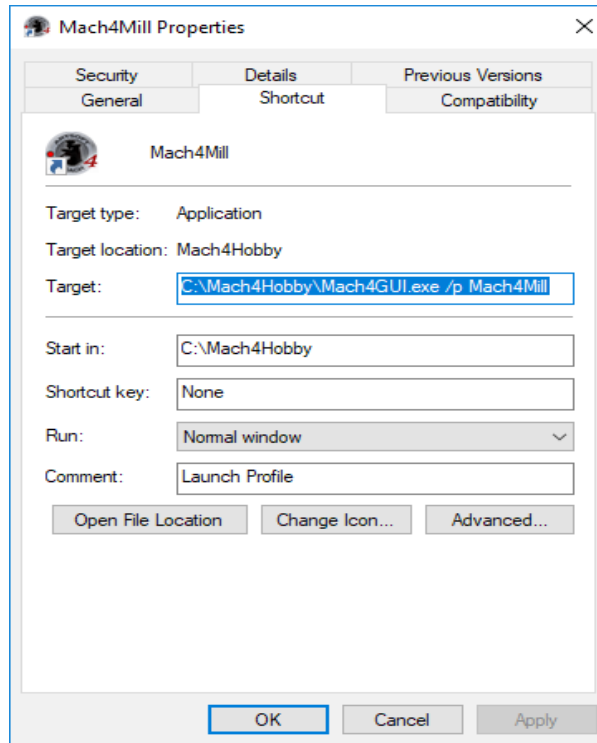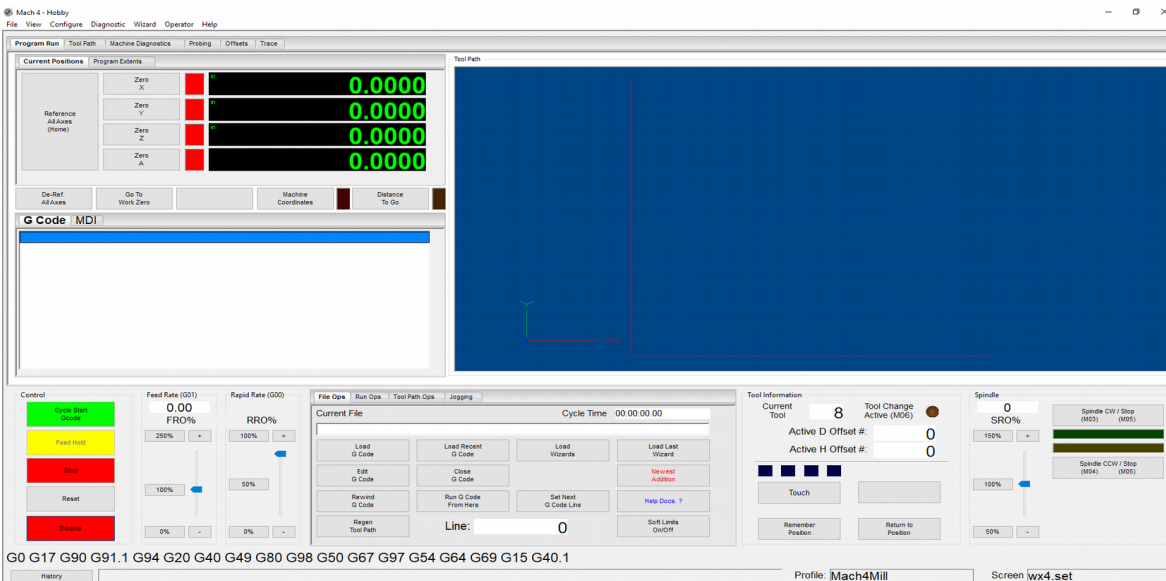*Figure 3-4: Mach4 main screen in enabled state*

Notice that when Mach4 is disabled buttons that control motion or program execution are unavailable to the user, they are greyed out. On this main screen these controls are "Ref All Home", "Goto Zero", "Cycle Start", and "Reset". Once enabled these controls are made available to the user.

Home the Machine

The vast majority of machines running Mach software use incremental encoders for position feedback, some have no position feedback to Mach. As a result, when Mach4 is started and enabled, the control has no idea where the machine is positioned. To determine the location of the machine Mach4 uses a referencing sequence started by pressing the "Ref All Home" button. Before pressing the reference all button ensure that no axis is on its home switch, jog the machine away from the switch if necessary. Each axis is sent to its home position, signaled by the activation of a switch, in the order set in the homing and limits configuration. When the home position is found the DROs in Mach will be updated to the desired machine position, usually 0 for all axes. The machine is now ready to run.

Jogging

Manual jogging allows the operator to move the machine around using on screen controls, figure 3-5, the keyboard, or an attached control panel / pendant (see figure 3-6). To view the onscreen controls, click on the jogging tab on the screen.

Figure 3-5: Jogging tab
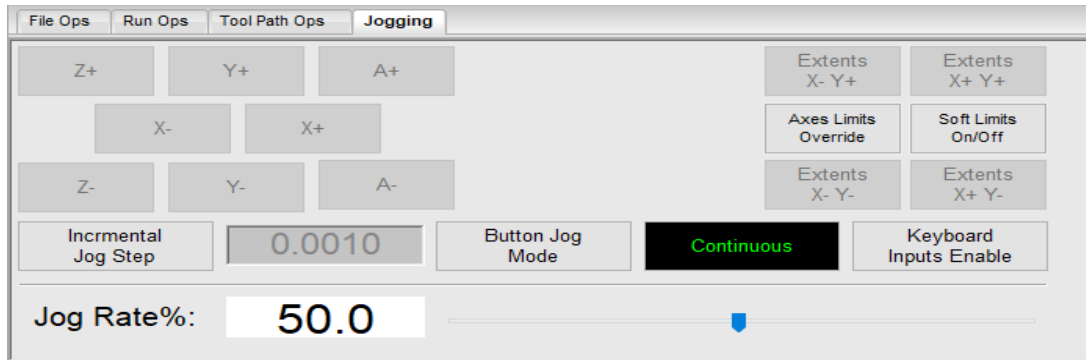
Jogging is enabled whenever the controller is enabled and a program is not being executed. However, keyboard jog defaults to off for safety. The default screen provides controls for jogging 4 axes; X, Y, Z, and A. These controls work just like a physical button, in continuous jog mode the machine will move as long as the button is held down. Keyboard jogging is deactivated by default, this feature can be activated by pressing the "Keyboard Jog" button. The default jog buttons are left arrow for X-, right arrow for X+, up arrow for Y+, down arrow for Y-, page up for Z+ and page down for Z-.

There are 2 basic jog modes which are selectable by pressing the "Jog Mode" button. The current selection will be indicated by an LED. Figure 3-5 shows the continuous mode selected. In continuous mode the machine will move at the specified jog rate as long as a jog button is held down. The jog rate can be changed by moving the slider at the bottom of the jogging tab or by directly typing the desired rate into the DRO in the bottom right corner. The number shown in the DRO is a percentage of the max velocity, or rapid rate, of the machine. Continuous mode is useful for moving the machine relatively large distances and rough positioning. When finding the edge of a work piece, locating a hole, or other precision position is required, step jog is much more accurate. In step jog, the machine moves only by the specified increment amount, regardless of how long the jog button is held down. The jog increment can be changed by pressing the "Cycle Jog Inc" button. This button will change the DRO to the right of it to show the current increment amount. The increment values can be changed on the General tab in the Mach configuration.

Figure 3-6: Control panel and pendant examples

Changing Tools

Tool changes will vary depending on machine configuration. Some machines have automatic tool changers, while others require the operator to physically change the tool. Each type of tool changer must be configured properly in Mach4, this is covered in the configuration manual. For automatic tool changers, a Lua script will likely be required to command it or communicate between an external controller and Mach4. Scripting is covered in a separate scripting manual.

Fixture Offsets Tab
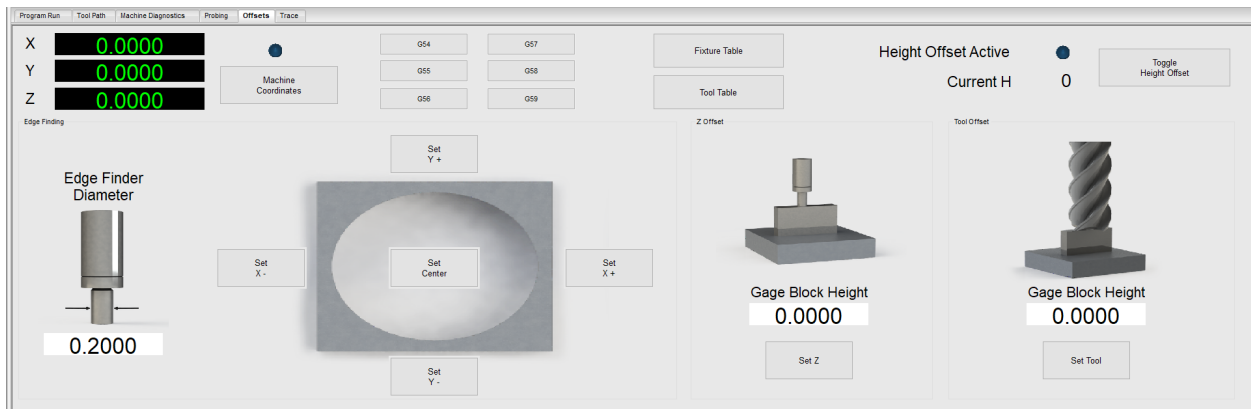
Figure 3-7: Offsets Tab

Fixture offsets are used to tell the machine where a work piece is located. The values are stored in the fixture offset table, figure 3-8. These values offset the machine coordinate system to create a user coordinate system at the desired point. Offsets are activated by calling the corresponding G code in the program. See the programming manual for details on calling fixture offsets. G54-G59 are the standard fixture offsets with G54.1 P1 through G54.1 P248. There are also two additional offsets in the table Work Shift and Head Shift. Values in work shift will shift the entire table, this can be useful when test

running programs (the Z axis can be lifted slightly more than the max depth of the program to run above the part) or making adjustments without changing offset values. The work shift is always active, so when changed, it is important to remember a change has been made. Changing fixture offsets in the program will not deactivate work shifts. Head shift is used for multi head/multi spindle machines and not covered in this manual.

Figure 3-8: Fixture offset table

There are a variety of ways to find a work piece in the machine; edge finding is probably the most commonly used method and will be covered in this manual. Probing is a great method when available, sometimes all that is required is to visually jog the machine to position and zero it. The accuracy required is determined by the required part accuracy.

Probing Tab

26
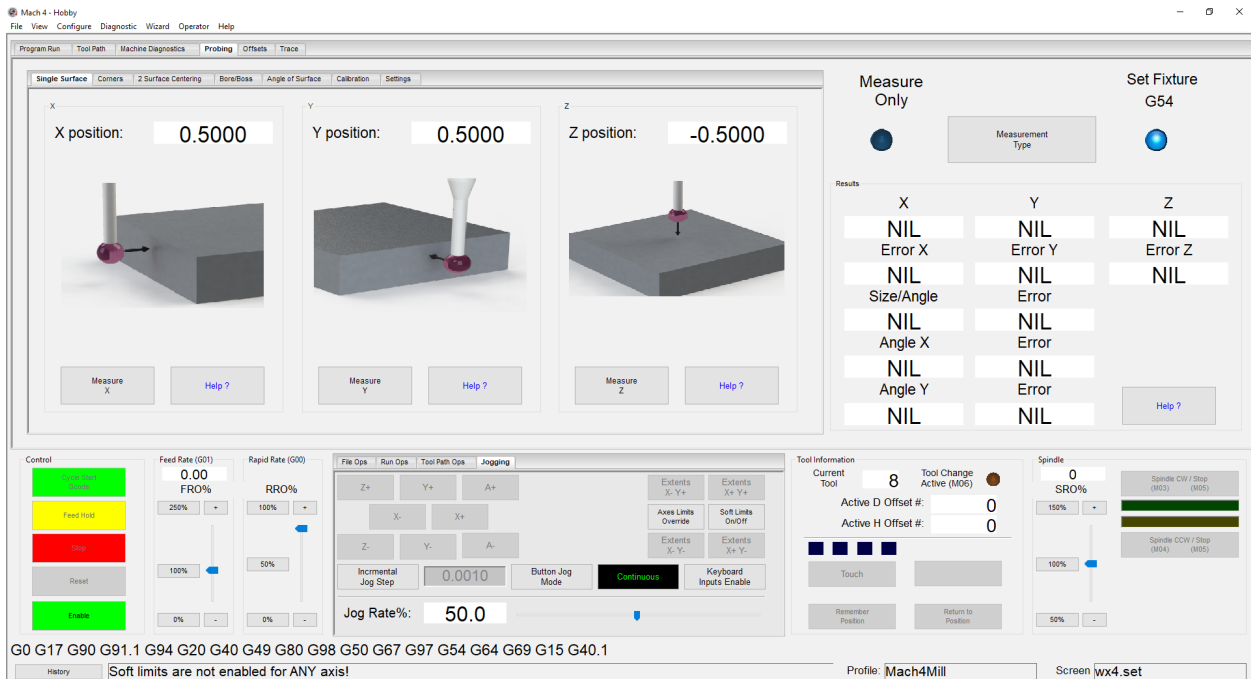
Figure 3-9: Probing Tab

Edge Finding

Edge finders, figure 3-10, come in many shapes and styles, each specialized for specific applications. The type used for examples in this manual are highlighted in figure 3-10. The shaft diameters can vary, but the tip diameter is 0.200 inch, making the math for calculating the offsets very easy.

Figure 3-10: Edge finders

An edge finder is used by spinning it at a low RPM, usually 1000 is perfect. The tip of an edge finder is held in place under spring tension and is moveable. In figure 3-11 all the edge finders are shown with the tip offset. When spinning in free space, away from a part edge, the tip will move away from the center line and wobble. Choose an axis to touch off first. For this example, we will use the X axis and make the

left edge of our part the fixture zero. Jog the machine to a position that will allow the edge finder to come down below the surface of the part just to the left of the work piece, see figure 3-11 position A.
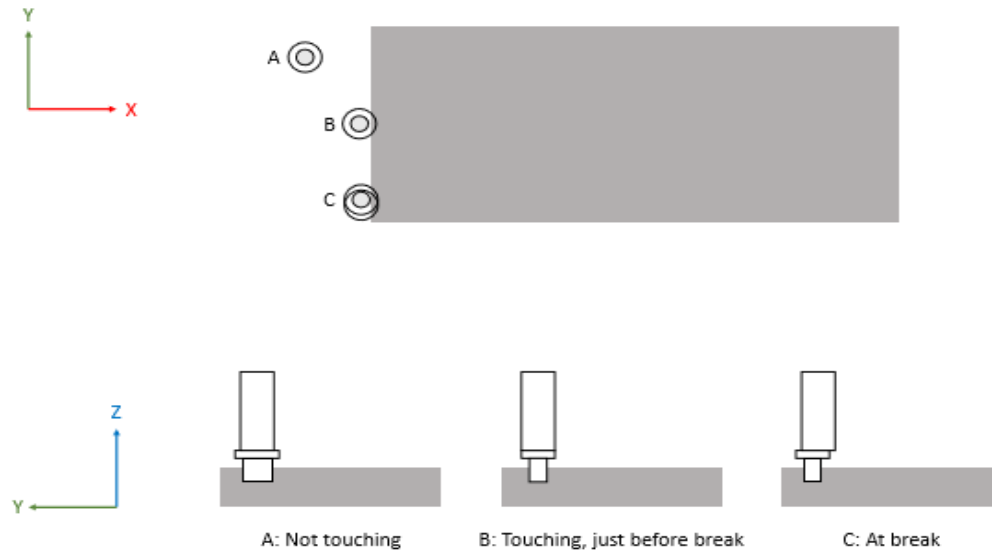
Figure 3-11: Using an edge finder

When clear of the work piece, the edge finder can be moved down in the Z axis until the tip is just below the surface of the part, about .100 inch. The tip will be wobbling and appear bigger than it actually is. Jog the machine, slowly and carefully, towards the part. When the edge finder just starts to touch the part stop and switch to incremental jog. A larger step amount of about 0.010 inch is appropriate here, move one increment at a time until the edge finder starts to look like figure 3-11, position B. As the machine moves closer to the edge, the edge finder will start spinning more truly. As the machine gets closer, the edge finder will get more sensitive to movement. Turn the step increment down to 0.001 inch and move in one increment at a time. When the correct position is reached, the edge finder will break, as shown in figure 3-11 position C.

Offsets Tab
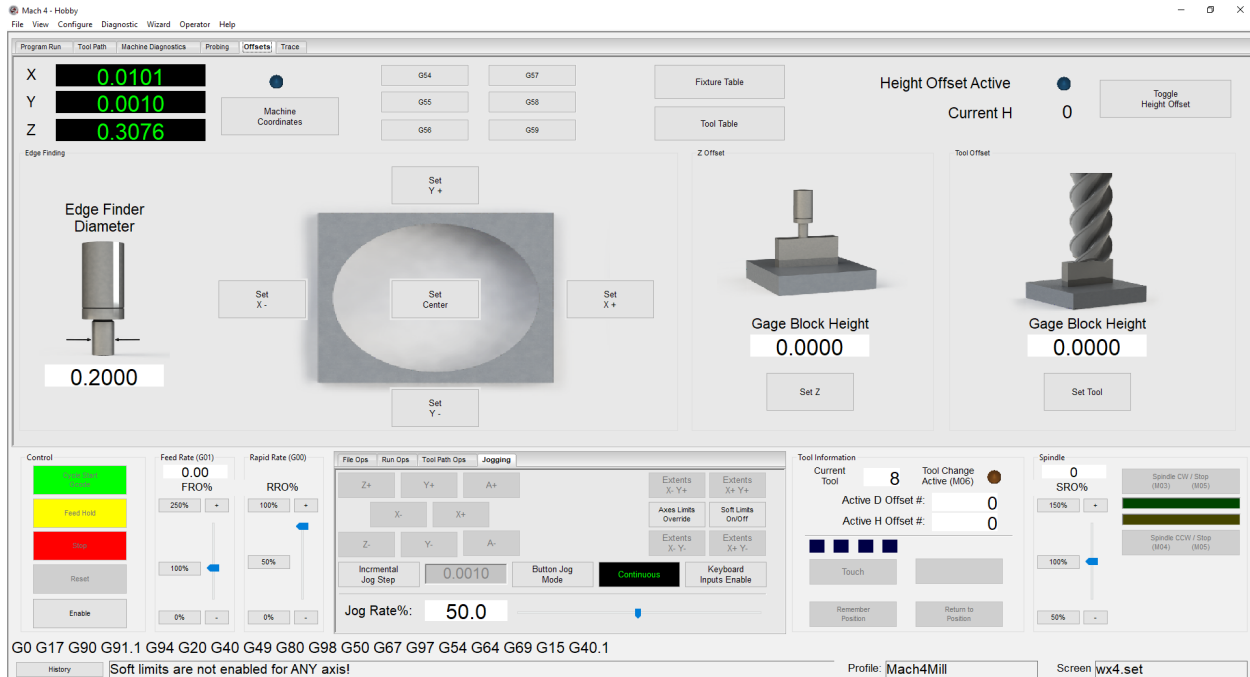
Figure 3-12: Offsets Tab

Inputting Fixture Offset Values

Now to input a value into the fixture offset table. There are a couple ways to do this. The value can be calculated manually and directly typed into the field or the input box can be used to enter values. When using the input box there are three ways to enter data, see figure 3-13. The first is the "Input" button. Pressing input enters the value from the input box into the selected offset, over writing any existing value. Next, is the "Input +" button. Pressing the plus input adds the value in the input box to the value in the selected offset. For instance, if the current offset is 1.5 and .1 is typed into the input box then plus input is pressed the new value of the offset will be 1.6. If input was pressed, the new offset would be .1. Play around with these input functions to get familiar with them and how they function. The third button is the measure button. This button will do the math for us.

Figure 3-13: Fixture offset table

Let's first input the offset using the "Input" and "Input +" buttons. The value we're looking to enter into the offset is the machine position of the X axis at the desired zero location on the work piece. The position we are looking for is the location of the edge of the part. The edge finder was used to find the edge, but the radius of the edge finder has to be accounted for, see figure 3-14.

Figure 3-14: Touch off location for 0.200 inch diameter edge finder

Select the desired offset. In figure 3-13 the X offset of G54 is selected. In the input box type in the position displayed in the X axis machine position DRO and press the input button. The data will be written to the offset. Now turn off the machine position to see the fixture offset position in the DRO. If everything was entered correctly, the X DRO should now read 0.0000. If not, make sure the desired fixture offset is active by commanding it in the MDI mode. Now, although the data was input correctly to

obtain zero, the edge finder radius has not been accounted for. The position we are actually looking for is -0.1000 as shown in figure 3-14. To make this adjustment type 0.1000 into the input box and press the "Input +" button. The X DRO will update to show -0.1000, this is the correct position. Double check your offset data by commanding the fixture offset in MDI to activate it, jog the machine so the edge finder is just above the surface of the part and move to X0.0000; the edge of the part should be at the center of the edge finder.

The next method, and more reliable as human error is reduced, is to use the measure button. With the edge found with the edge finder, type in the desired position, in this case it is -0.1000 and press the "Measure" button; the offset value is calculated and input into the offset. The X axis DRO will now read -0.1000. The math is completed by Mach4, just input the value that the DRO should read at the current position.

Repeat the same process to find the Y axis position. The Z axis will be a little different. A good way to find the Z axis position is to use a fixed tool. The edge finder is a good one, and a gage block. Move the machine above the part and jog down slowly. When near the gage block height, use incremental mode to avoid crushing anything. Slide the gage block under the edge finder, if it slides freely pull it out and step the Z axis down. Keep stepping down until the gage block just hits and won't slide under. Go back up one increment, usually around 0.001 but that will vary with desired accuracy. This is the touch off position. Entering the offset data is the same as described above, only this time instead of taking the edge finder radius into account the gage block thickness must be used. Select the Z axis offset in the fixture offset window, enter the gage block thickness in the input box and press the "Measure" button. Activate the offset and check the position DRO. The Z axis should show the gage block thickness.

Tool Length Offsets

Tool length offsets are used to tell the machine how long a tool is or where it is in relation to the rest of the machine or work piece and are stored in the tool offset table, figure 3-15. There are as many ways to touch off tools are there are CNC machinists, this manual can't attempt to cover them all. The examples used here are to demonstrate how the offset system works.
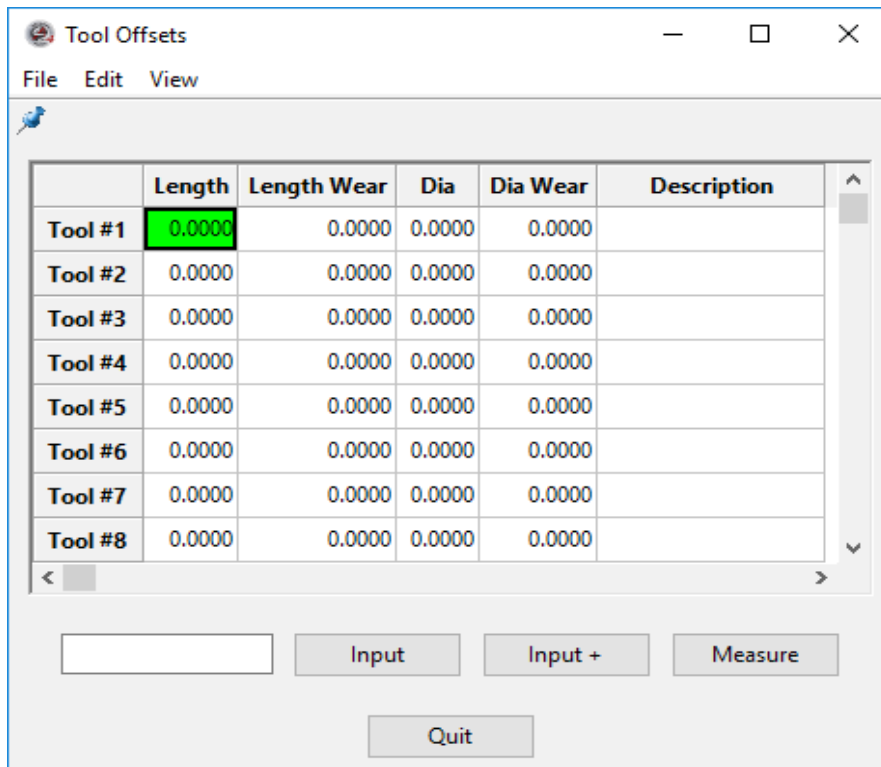
Figure 3-15: Tool offset table

There are a couple of methods for touching off the tool. A gage block can be used as described for setting the Z fixture offset in the previous chapter. A piece of paper can be used in place of a gage block, using the same procedure. Electronic blocks and touch off devices are available, as are blocks with indicators, and even automatic tool setters (see figure 3-16). Another great method for high production environments is an offline tool pre-setter outside the machine.

Figure 3-16: Types of tool setting devices

Master Tool

Setting up a master tool is a great way to do tool offsets, especially on a knee mill type machine. This method involves having a tool at a fixed length and a tool offset value of zero. An edge finder is great for this. Select a holder and lock the edge finder at a fixed length. Use the edge finder to set a work offset at

the top surface of a work piece or any flat surface to touch all the tools off. This allows the operator to set up a library of tools with set offsets. When a new part is set up, just touch off the Z fixture offset and start machining. All the tools will follow the master tool. Make sure to disable tool length offsets (G49) or activate the tool length offset for the master tool (G43 Hhh, where hh is the offset number of the master tool) before setting the fixture offset for touch off surface.

Once a surface is found using the master tool and the fixture offset entered, bring each tool down to the surface using the same method, some type of gage block or tool setter of known height. The position shown in the Z axis DRO will show the difference in length between the master tool and the tool being touched off. There are a few options for inputting the value into the tool offset table, just like when setting fixture offsets. The value can be entered directly into the offset. A value can be typed into the input box at the bottom and then the "Input" button will over write the existing data with the value in the box, the "Input +" button will add the value from the input box to the existing value, and the "Measure" button will calculate the offset value using the data in the input box as the desired position. For this example, type in the gage block thickness then press the "Measure" button. The value will be set. Double check the data by activating the current tool offset, MDI G43 Hhh where hh is the offset number of the tool being touched off. The Z axis DRO should read the gage block thickness if done correctly.

Tool Length from Gage Line

Measuring the actual tool length from the spindle gage line (figure 3-17) is the preferred method when using automatic tool setters and manual electronic/light-up or indicator style setters. It is also particularly suited to machines with fixed tables like bed mills, vertical and horizontal machining centers, gantry mills, and routers, etc. The distance from the spindle gage line to the measurement point must be known. When tools are touched off, the offset value will be the distance from the tip of the tool to the gage line. Because the offset is simply a length measurement and not a machine position, the offset value can also be measured outside the machine on a tool pre-setter.
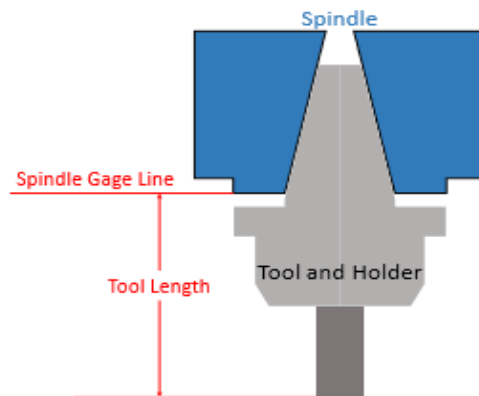
Figure 3-17: Gage line and tool length

To use this type of tool offset system, a fixed point in the machine should be chosen to measure tools. A great position is the table of a bed mill. The distance from the spindle gage line to this point must be found. One way to do this, is remove any tool from the spindle then jog the Z axis down. Generally, the spindle cannot reach the table, but a 4" block usually provides enough height to obtain a measurement. Use the same procedure as was used when touching off tools, step the spindle down in small increments until the block just slides under. The machine position Z axis DRO will read the distance the spindle traveled down to reach this point. Record this number. For our example machine, let's say it is -10.0000. If the gage block used is the same one that will be used to touch off tools, then this is the number that

will be used for tool touch offs. If not, subtract the gage block thickness (-10.0000-4.0000=-14.0000) to get down to the table and then add back the height of whatever gage will be used.

Now, we will explain how to touch off tools. A fixture offset can be set at this position or a macro can be created. If a fixture offset is used, it must be active and tool height offsets inactive when touching tools to get the correct data. With this set up, the user position Z axis DRO will read the length of the tool. This number should be directly input into the tool height offset. With a macro (see customization manual for more detailed information) a button like the "Auto Tool Zero" button can be used to do the math. The macro would take the absolute value of the difference between -10.0000 and the machine position when the button is pressed. The resulting number will be the length of the tool. For instance, if a tool is touched off and the Z axis machine position is -8.5000 the tool length, tool height offset, is -8.5000+10.0000=1.5000. So, the tool in the spindle is 1.5000 long, which is the tool offset. This method is a little more complicated to set up, but once set can be a time saver. Fixture Z offsets can be set with any tool as long as the correct height offset is activated.

Tool Diameter Offsets

Tool diameter offsets allow the operator to make adjustments to the finish size of parts provided that the program contains the proper codes to activate and use the diameter offset. Data is entered in the "Dia" column in the tool offset table, figure 3-15. Detailed information on the programming and use of tool diameter offsets can be found in the cutter compensation section of the Mach4 programming guide.

Tool Wear Offsets

Cutting tools wear during use. This is true for everything from steak knives in the kitchen to high tech coated carbide metal cutting tools. Wear and dull cutting edges induce deflection in the cutting tool and machine that can throw off the precision of a part being cut. Mach4 provides wear offsets for both tool length and diameter to compensate for this deflection, and sometimes actual reduction in length or diameter of the cutting tool. These wear offsets are found in the tool offset table, figure 3-15, in columns labeled "Length Wear" and "Dia Wear". When a tool offset is called, Mach4 adds the wear offset to the set offset value to determine the actual offset to be used. Using wear offsets is recommended for making small adjustments over making changes to the other offsets. For example, a CNC mill is facing a steel part and the finished thickness is 0.005" too big. The operator adjusts the tool length offset, -0.005", to cut deeper. The next part is an aluminum one, the operator doesn't remember the old offset so just faces the part and it is 0.007" too small. The difference in material hardness can affect the finished size of the part. In this case, the operator would have had to record the previous length offset or remember the adjustment amount. Using wear offsets, the operator could just zero out the wear offset, or even use it to cut a little oversize and work down without doing a lot of math. The original offset is never changed, unless the tool is changed. Wear offsets are a valuable resource, learn them, understand them, and use them.

Tool Information

During operation the current tool information is displayed on the main screen, figure 3-18. This section of the screen shows the current tool number and its diameter and height offset information. It is a quick way to check that the correct data has been input into the tool offset. During a tool change the "Tool Chng" LED will blink, indicating that the machine is either changing the tool automatically or waiting for the user to manual change tools and press the cycle start button.
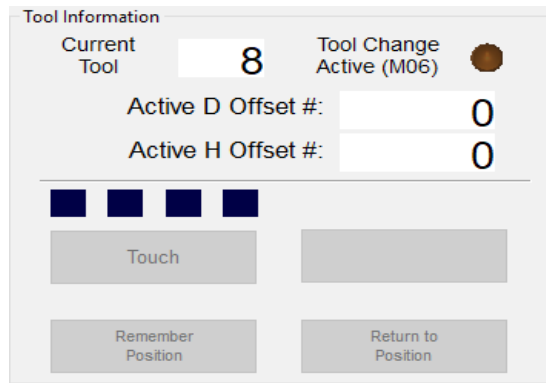
Figure 3-18: Tool information

The "Tool" DRO shows the currently selected tool, this DRO will be updated at every tool change. It can also be changed by the user by clicking on the DRO, typing in the desired tool number, and pressing enter or return on the keyboard. The "Diameter" and "Height" offset DROs will update to show information for the selected tool number. These DROs are read only and not user editable.

Creating Programs

Programs are required for automatic operation of the machine. The most common format is G code. This type of programming uses a system of G codes (G01, G10, G81, etc.) to specify the desired function. Position codes (X, Y, Z, etc.) are used to tell the machine where to go, or where an operation is to take place. Other codes starting with M, T, S, F and a few others enable control of just about every aspect of the controller and machine. The Mach4 Programming Guide is a complete description of the required G code format.

Programs can be created by using the wizards, a CAM (computer aided manufacturing) system, or by manually typing it. There are a variety of CAM systems that can be used for generating programs from 2D or 3D drawings. When CAD models are available using a CAM package is usually the best and quickest way to generate a G code program for machining. However, some of these software packages can be very expensive. For this reason, Mach4 includes the ability to run wizards, a form of conversational programming. Wizards can be created by anyone with some computer programming skill for just about any task needed.

Mill Wizard

Mill Wizard provides the user with a set of tools to generate G code files for common milling jobs. The close link with Mach4 allows created G code to be loaded into Mach4 with the push of a single button. Mill Wizard can create programs with multiple tools and features. Features and shapes include rectangles, circles, links, slots, hole patterns, engraving, gears, splines and more. Figure 3-19 shows the main screen and all the available operations.

Jobs can be created using the stock materials or the user can add special materials to the table. Tools can be added to a table for later use. Operations can be modified and re-ordered as desired. Complete documentation is provided in the help menu of the software. The follow figures give a basic idea of what the software looks like.
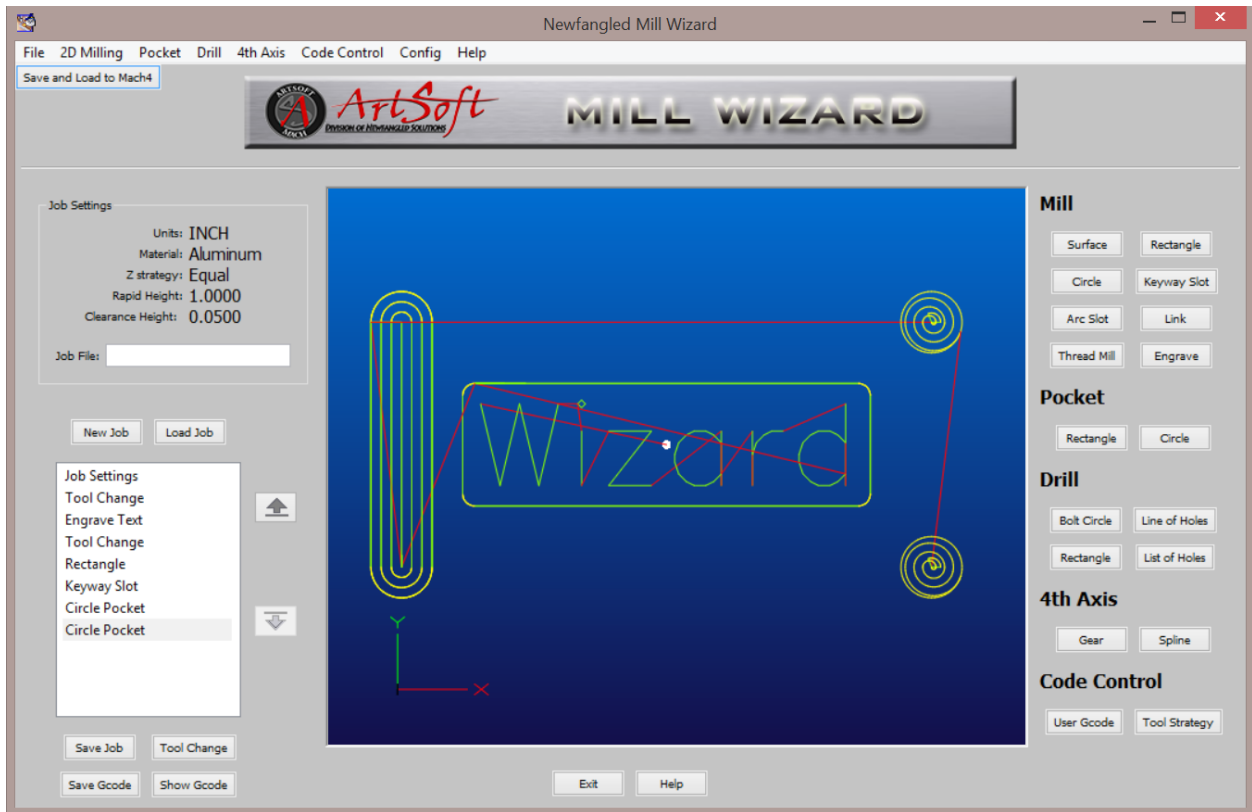
Figure 3-19: Mill Wizard main screen
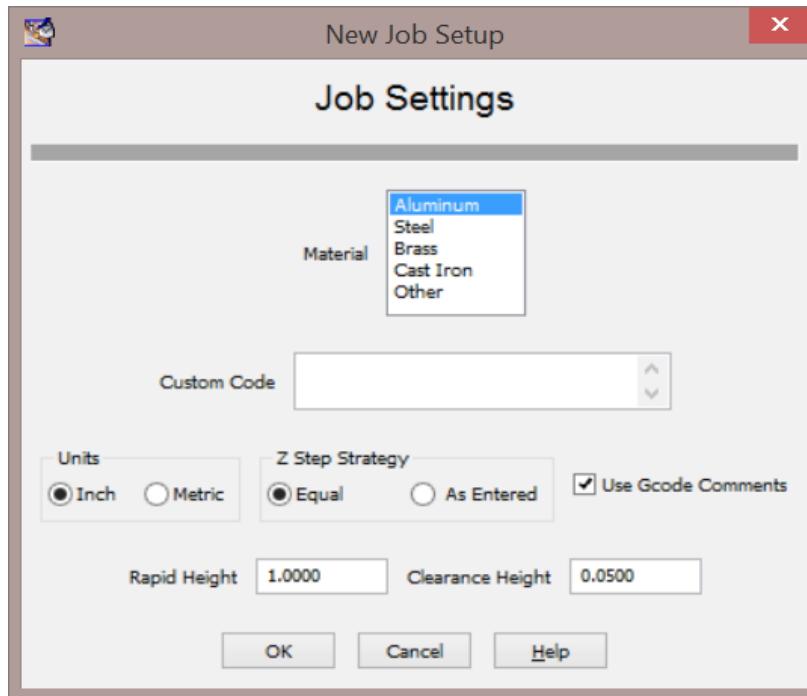

Figure 3-20: New job window, select material, units, etc.
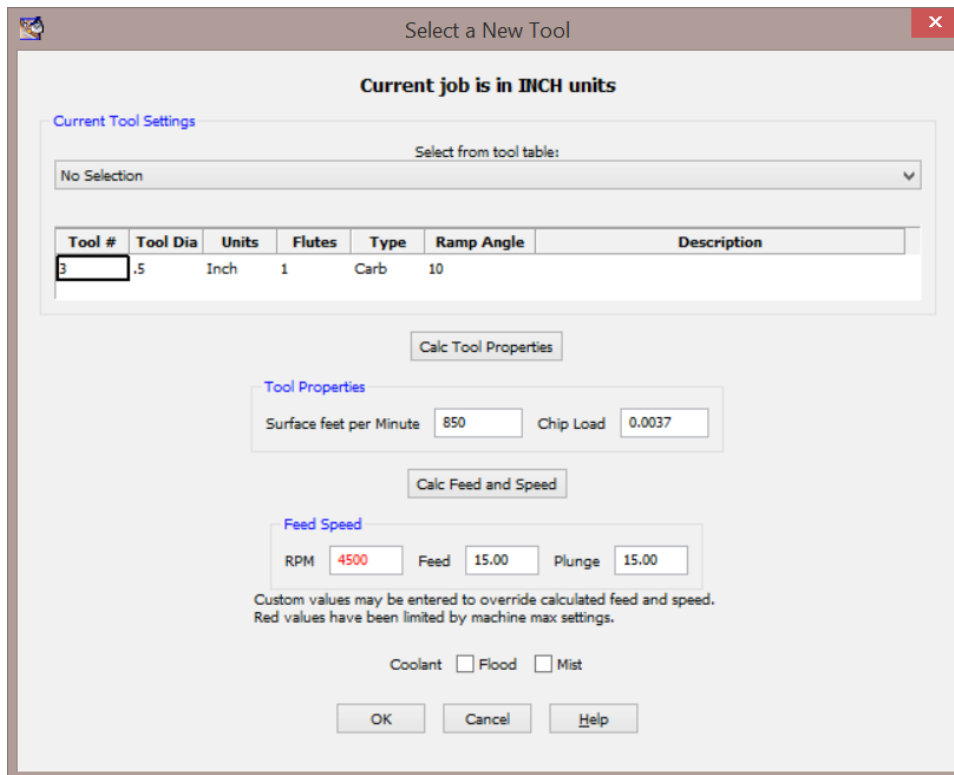
Figure 3-21: New tool window, select tool and set feed and speed
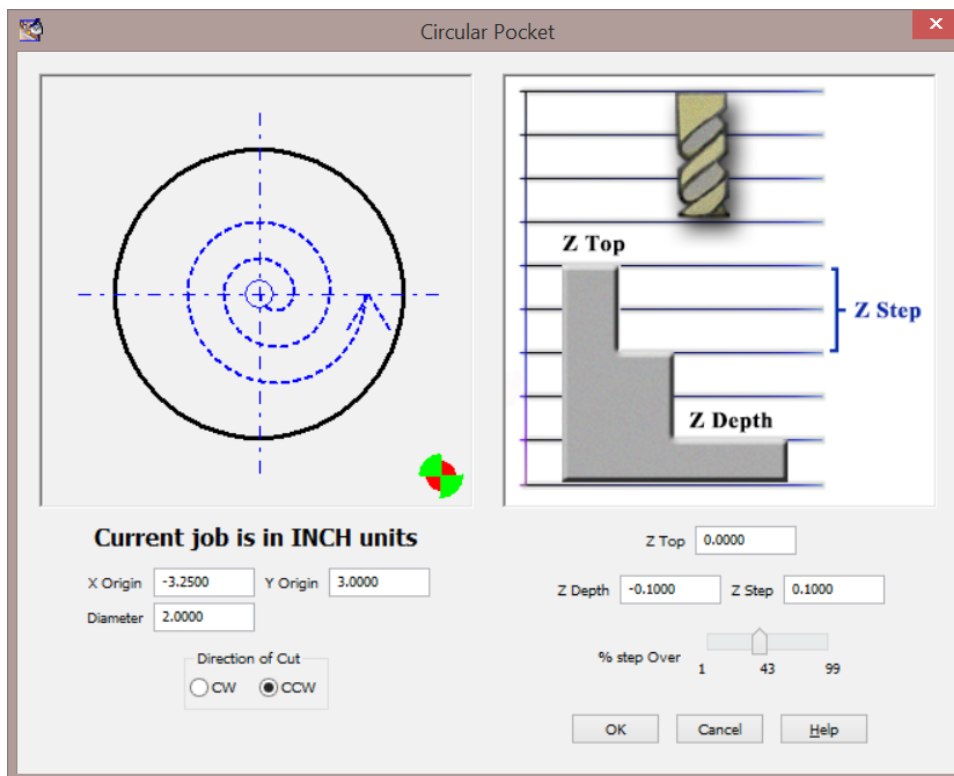
Figure 3-22: Circular pocket wizard
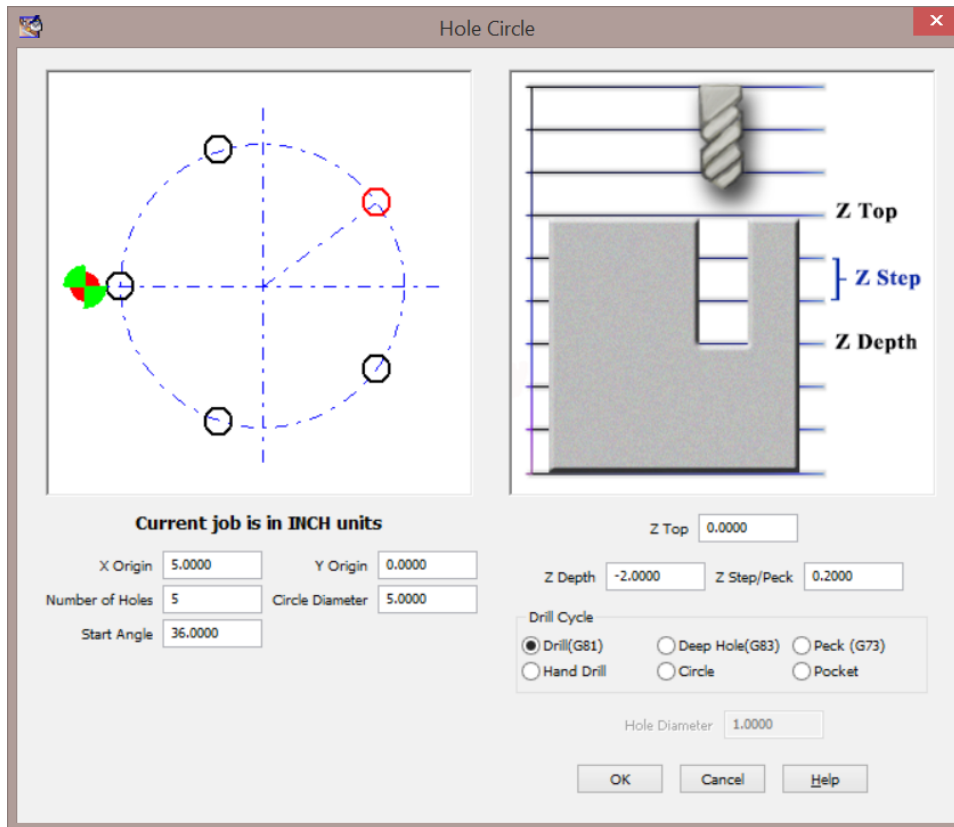
Figure 3-23: Hole circle wizard
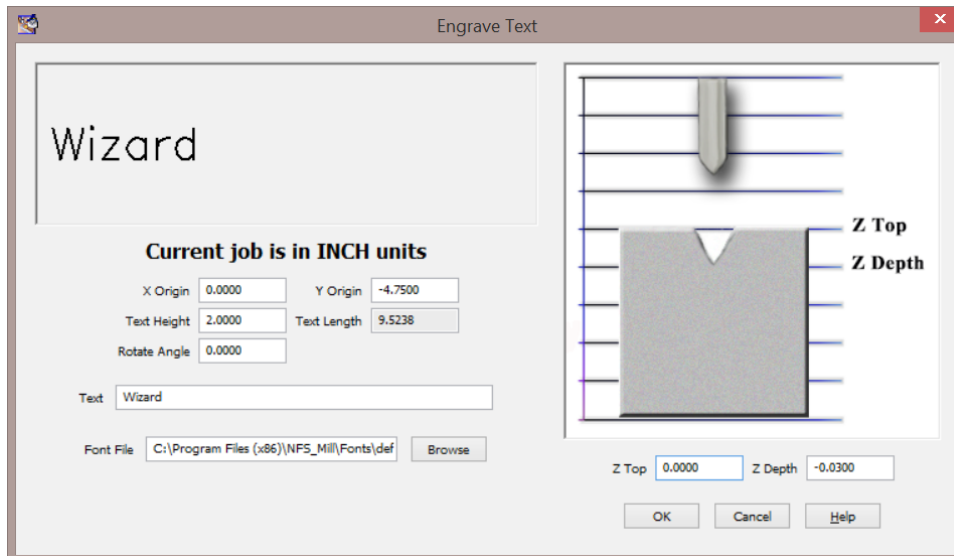
Figure 3-24: Text engraving wizard
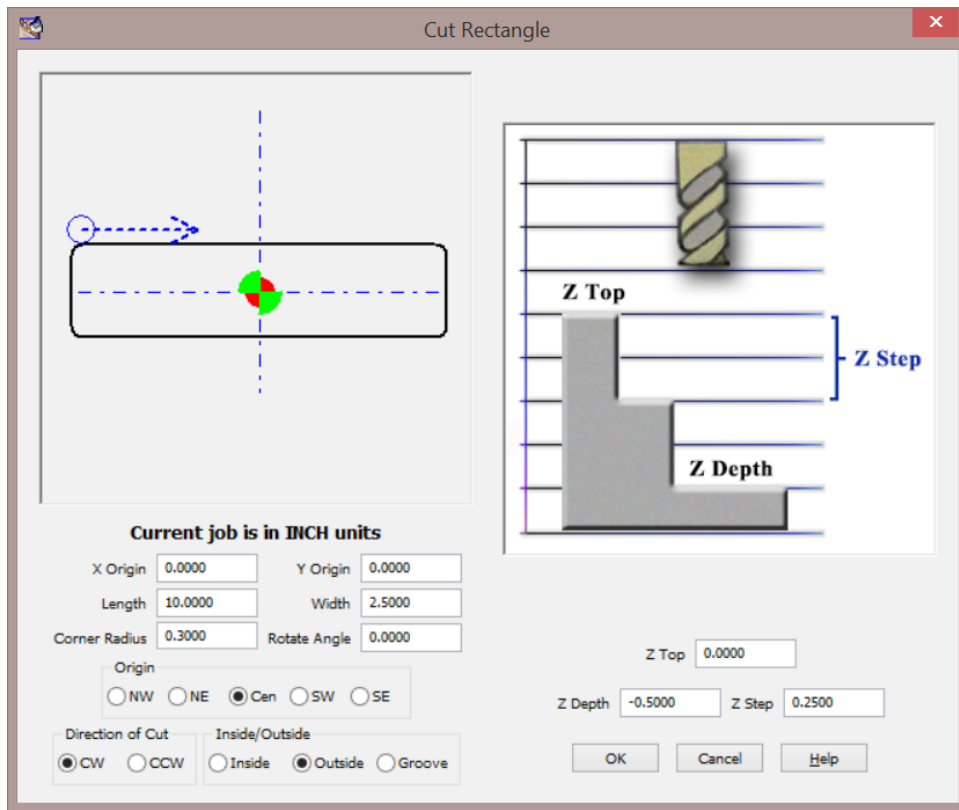
Figure 3-25: Rectangle cutting wizard, cut inside, outside or groove

Running the Machine

Manual Data Input (MDI)

Manual Data Input (MDI) mode allows the operator to manually enter G code to be executed. The operator should have a solid understanding of G code and machine programming before attempting to command the machine using the MDI mode. This is useful for activating or deactivating fixture offsets, tool offsets, positioning modes, tool changes, moving to known positions, etc. In Mach4, MDI programs can be multiple lines and a history of commands is saved that can be navigated using the up and down arrows adjacent to the MDI window.
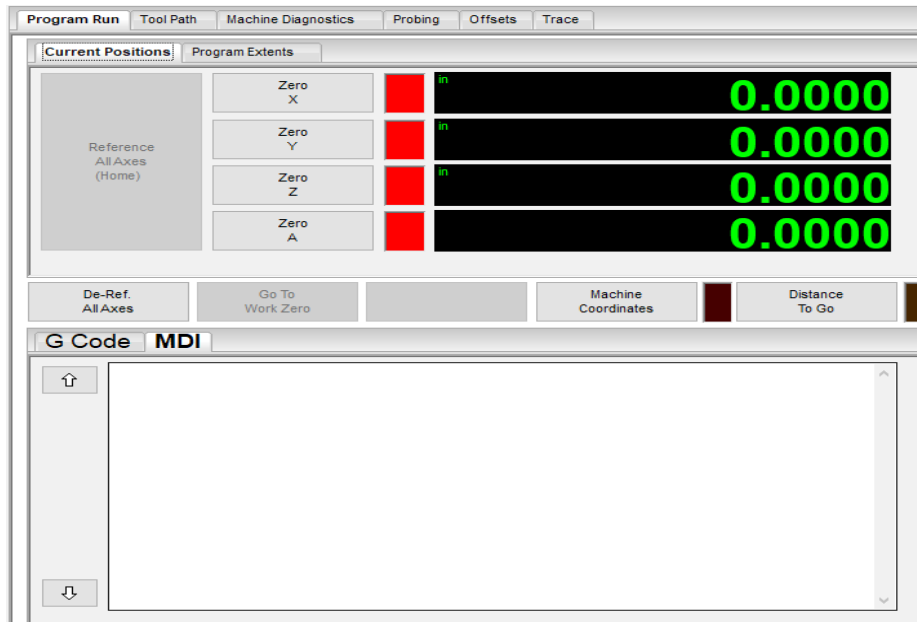
Figure 4-1: MDI screen

To enter code into the MDI window, select the window and type in the desired code. Press the "Cycle Start" button to execute the code in the window. Once executed, the code will be stored in the history where it will remain until Mach4 is shut down. Press the up arrow on the left of the MDI window to scroll through the history. The most recent code is shown first. The tool path window on this page shows the tool path of the currently loaded G code file and not the MDI program.

Some examples:

- Change to tool number 10: T10 M6

- Activate tool length offset number 6: G43 H6

- Move to machine zero position: G53 G90 G0 X0 Y0 Z0

- Move to part zero position: G54 G90 X0 Y0 Z0

Loading a G Code Program

G code programs contain code for cutting a part. The first step to running a program is to load it into Mach4. In the File Ops tab in the operator control section of the screen press the "Load G Code" button. A window will open that will allow the operator to navigate to the desired file and open it. By default Mach4 will show .tap and .nc files. Any basic text format can be opened and the default file types can be changed in the Mach configuration on the General tab. When the file loads, Mach4 will quickly run through it to generate a tool path for the tool path window. If there are any major code errors or unknown codes they will be found on this initial run through. The tool path window will show the actual tool path including cutter comp and any other adjustments made in the program. Check it to make sure it looks as intended before running.

Viewing the Tool Path

The tool path window shows the programmed path exactly as it will be run by the machine. It is possible to pan rotate and zoom the tool path window using the mouse. Move the mouse pointer into the tool path window and perform the following:

- Rotate – Left click and drag

- Pan – Left + right click and drag

- Zoom – Right click and drag or scroll wheel

- Reset View – Double left click

Editing a G Code Program

To edit the currently loaded G code file press the "Edit G Code" button in the File Ops tab. Mach4 will open the file in the included G code editor: GC Edit. When the desired changes are made, save the file and close the window. Mach4 will re-load the file and re-generate the tool path.

Closing a G Code Program

To close a G code program, press the "Close G Code" button in the File Ops tab.

Executing a G Code Program

Cycle Start

Pressing the cycle start button will start program execution from the current line shown in the File Ops tab. The program will run until a program pause or end G code is read or execution is stopped by the operator.

Feed Hold

The feed hold button will immediately pause the program, press cycle start to continue. Feed hold will only pause axis motion. If the spindle is running it will stay on and any active outputs will remain active (coolant, air blow, etc).

Stop

The stop button will immediately end program execution. All motion will stop as well as the spindle and other outputs. This is similar to commanding a program stop in the G code program. Although program execution is halted by pressing the stop button, the program is not rewound.

Reset

To rewind a program back to the beginning press the reset button. This will also return the controller to its default state.

Starting from the Middle of a Program

There are a couple ways to start execution from a point other than the beginning of a program. If the desired start point is a safe start line or other safe position, not including any motion, the desired start line can be selected and cycle start pressed. Select the desired start line in the G code window and then press the "Set Next Line" button. When the cycle start button is pressed the program will begin execution from this selected line.

Programs can also be started from any position in the program, including in the middle of a cut. To do this, select the point to start by scrolling through the G code program in the G code window and select

the desired line. Manually jog the machine close to the start point. Next, press the "Run From Here" button in the File Ops tab. A window will open, figure 4-2, to start the process of restarting the program.
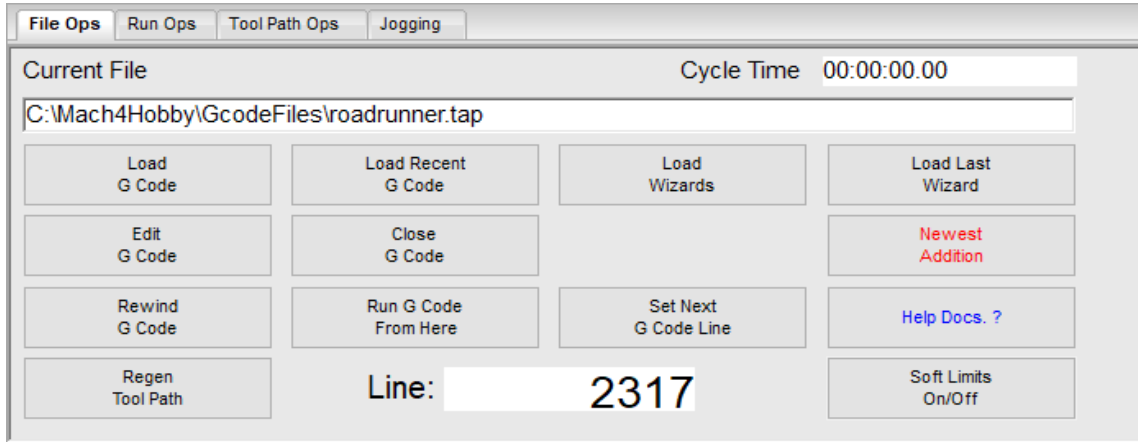

Figure 4-2: Run G Code From Here button

In the run from here window, there is a position read out showing distance to go. These distances are how far the machine needs to travel to reach the start point. Before starting execution, the machine must be moved to the start point. The axes can be moved using the controls in the "Move Axis to Start Position" section of the window. Enabled axes will be available for selection. Select one or multiple axes to move and press the "Move Selected" button. The selected axes will move to their start positions. Another button, "Move Unselected" will move the unselected axes. Use this button to move any remaining axes to their start positions. Once the machine is in the start position, press the "OK" button. Mach4 will scan the preceding part of the program to get the machine into the proper state: coolant/spindle on or off and setting up cutter comp if active. Press cycle start to start machine motion and run the rest of the program.

Help, I've Crashed…

…and now my machine won't move. Inevitably, as careful and conscious as we may be, there will be a crash or run in with hard limit switches. Recovery from these situations can range from careful jogging to removing tools and debris before moving (try to avoid the latter). As always, prevention is better than recovery, both will be discussed.

Limit Switches

Limit switches are physical switches that mark the end of a machine's range of motion. Sometimes one or more of these will double as a home switch. The hard limit switches are basically the last line of defense before major damage occurs. Without these switches the machine could slam into a hard stop at the end of travel or even send the table right off the end of the rails, depending on the configuration. Running a machine without limit switches is risking damage to ball screws and nuts, bearings, castings, people, floors and walls.
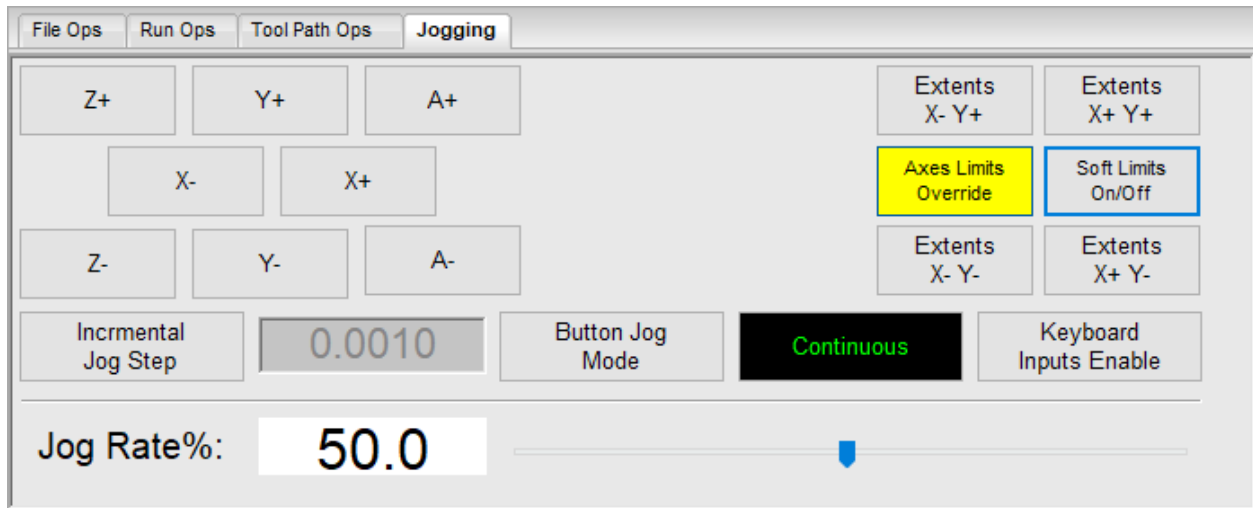
Figure 5-1: Limit override on program run screen

When Mach senses that a limit switch is activated it immediately disables all motion. In this state jogging is disabled. At first it can seem impossible to recover from this situation. On small machines one solution could be to turn the ball screw or slide the axis by hand, on bigger machines this is not possible. In this case jogging the machine off the limit is the only option.

When a machine is on a limit switch it can be jogged by temporarily overriding the limit switches. Press the "Limit Override" button on the program run screen, figure 5-1. This will re-enable axis motion and jogging so the machine can be moved away from the limit switch. As soon as the switch is deactivated the limit override is automatically turned off.

When the limit override button is pushed the machine can be moved in any direction and regardless of the state of the limit switches. Exercise EXTREME caution to avoid serious damage to human operator or machine. Check and double check which axis should be moved and in which direction. Start movement with a very small move, just bump the axis, to verify your choice of direction is correct. Then move the machine the rest of the way off the switch.

Soft Limits

Running into machine hard limits can be prevented by using soft limits. Soft limits are software end of travel switches. They are generally set up by the machine builder to be about ¼ to 1 inch from the physical switch depending on machine size, velocity and acceleration. When an axis is moved into the soft limit it is decelerated to a stop. Generally there is a deceleration window of about a ½ inch. Inside this window axis velocity is reduced. When a machine runs into a soft limit it can simply be jogged the other way, no risky limit override button.
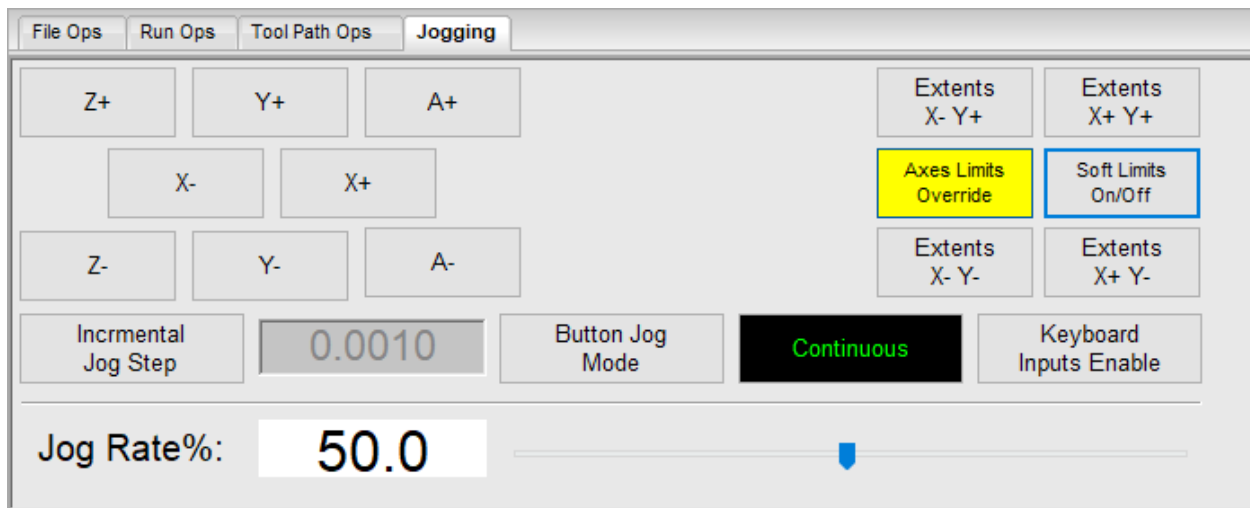
Figure 5-2: Soft Limits On/Off in Jogging tab

Soft limits can be displayed in the tool path display by pressing the "Display Mode" button. The soft limits show up as dashed yellow lines, figure 5-2. When the cycle start button is pressed to run a program, Mach automatically checks if the tool path extends beyond the soft limits, if it does a warning is issued.

Crash!!!

It happens to everyone who has ever run a CNC machine. The machine is running great, the part is being cut exactly as we expected. Then, CRASH. A tool breaks, the work piece comes out of the vise or chuck, the machine hits a clamp, or any number of seemingly impossible feats for a machine to perform. Sometimes it's very minor, sometimes it's very serious. These moments are extremely frustrating for operators and programmers, it is usually best to take a minute to survey the situation before taking action. Before attempting to move a machine that has been crashed take a good look at everything involved. Make sure there is no major damage to the machine, if there is don't try to jog it which could cause even more damage. If something happened while cutting a part, check the tool and where in cut it stopped. Sometimes it is best to loosen a tool in its holder so the Z axis can be jogged up to remove the tool. End mills can be easily damaged when jogged along a part edge or ripped out of a pocket when they're not spinning. Also, if the tool is currently welded to the part (which does happen) trying to pull it out with the machine can potentially cause damage to the machine or it could break, sending small very sharp bits of carbide or tool steel flying in every direction, could cause harm to humans and other things nearby. If a tool is stuck in a part try to loosen or remove it from the spindle or unclamp the part before attempting recovery.

Preventing the Inevitable

As always, the best method of crash recovery is to avoid the crash in the first place. Most crashes are due to human error and operator complacency. As humans we have a tendency to get overly confident and stop paying complete attention to what we're doing. This is especially common when running production type parts, large volumes and very repetitive. Always remain focused on the work at hand, CNC machines can be very dangerous and require 100% of your respect and attention. They will not think twice about removing parts of your body.

New programs are always a huge source of crashes involving the tool and work piece or work holding devices. Usually these crashes are minor, at worst resulting in a broken tool. However, the worst crashes can result in broken spindles, damaged bearings, cracked or bent castings and machine frames, bent tool changer arms, or even fire. However, these types of collisions can almost always be avoided by dry running the program, either without a part or above the part. Procedures for checking programs can be found in text books or talking to experienced operators.

Jogging is another source of crashes. We've all done it, jogging tool towards a part to check or set the height and bang, we jogged too far and crushed the tool into about 1,000 pieces. These crashes are usually very minor as the velocities tend to be low and the button pushes are short, but they do happen. Biggest thing to avoiding these is pay attention and take your time. This usually happens when you're rushing that last job and you don't want to switch to incremental jog just yet, one more bump of the jog button will get it that much closer. Now a new tool needs to be put in and touched off. Take the extra time and use the incremental jog when nearing a part. This prevents crashes caused by holding the jog button down a millisecond too long.

Computers. They make this technology possible, they can also bring it crumbling down and make us want to leap out of tall buildings. Computers can crash and fail, when set up properly this will simply cause the machine to stop, hopefully avoiding any damage. When a computer locks up the charge pump signal from Mach to the hardware motion control stops as well, this tells the motion controller that Mach is no longer in control of the system. In this situation the motion controller will essentially hit the emergency stop button ending all motion and shutting down all outputs. Using a good, reliable computer can almost completely avoid these situations. Keep the computer running the machine cleaned up (inside and out) and don't use it for any unnecessary tasks. It is best if this computer is 100% dedicated to running the machine and nothing else. Keep the control software and any plugins up to date with the most current stable releases. Don't move too fast for the computer. Many times a computer crash is caused when the computer is "thinking" and the user is frustrated and clicks on something 1,000,000,000 times. Don't do it, let the computer react to your input before moving on. Be deliberate and precise in your actions.

Sometimes, it's just out of our control. CNC machines are a complex web of mechanical and electrical systems. These systems can fail. No machine is ever failure proof. Motor drivers can fail, causing the motor to stop or go max RPM until something stops it. Wires can corrode and break, welds can crack, or lightening can strike. Whatever the failure, use the same processes described above. Step back and take it all in before taking action.