

# Ultimate Screen Reference Guide

8/11/2011



Everything you need to know to use and setup the MachMotion Ultimate Screen.

Copyright © 2011, MachMotion.com  
All rights reserved.

MachMotion  
[www.MachMotion.com](http://www.MachMotion.com)  
14518 County Road 7240, Newburg, MO 65550  
(573) 368-7399 • Fax (573) 341-2672

<b>Installation</b>	<b>5</b>
Screen Installation	5
User Macros Installation	6
Mach3 Setup	7
<b>Overview</b>	<b>9</b>
General Layout	9
Bottom Menu Bar	10
Reset	11
MDI	11
User Buttons	11
Ref Home	14
Offsets	14
GoTo	14
Limits	15
Control	15
Tool Path	15
Gcode	16
Error Codes	16
Scripting	17
Tool Panels	17
Jog	17
Spindle	18
Feedrate	18
Tools	19
User Input	20
<b>Main Tabs</b>	<b>21</b>
Program Run	21
Control Buttons	23
Load	24
Advanced	26
Tool Offset	27
User	28
Jog	28
Tool Path	31
Offsets	32
Homing	32
Offset Setup	33
Work Offset	34
X/Y Probing & Z Probing	35

<b>MDI</b>	<b>36</b>
<b>Diagnostics</b>	<b>37</b>
I/O	37
Mach Settings	38
HMI Settings	39
Special Functions	41
Reference	41
<b>Screen Customization</b>	<b>43</b>

# Installation

Follow the instructions below if the Ultimate Screen is not already installed and running on your control. Begin by downloading the Ultimate Screen at [www.machmotion.com](http://www.machmotion.com). Select the **Support** tab and then click **Downloads**. Click on the link “Ultimate Screen.” Save the file to your computer.

## Screen Installation

When the download is complete, open up the file. You should see the following window. Press **Run**.



Figure 1 Open File

You will see the following screen. Press **Next** until you reach the *Confirmation* window.

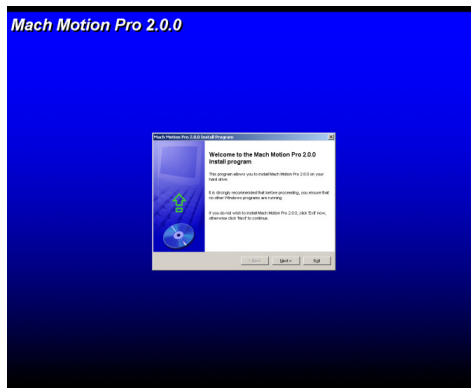


Figure 2 Ultimate Screen Installation

When you see the window below, press **Start** to begin the installation.

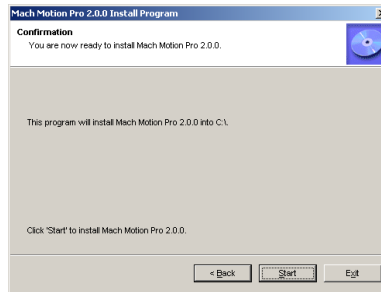


Figure 3 Ultimate Screen Confirmation Window

When it is finished, make sure to select **Install User Macros**.

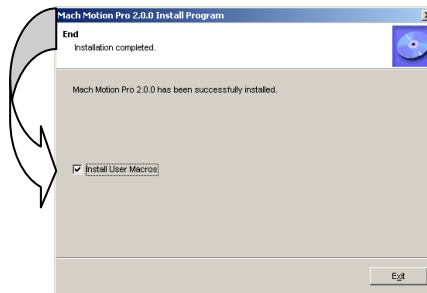


Figure 4 User Macros Selection

Your screen is now installed. However, the buttons will not work unless you install the user macros.

## User Macros Installation

When you press **Exit** on the previous installation window, it will start installing the user macros if you selected the check box.

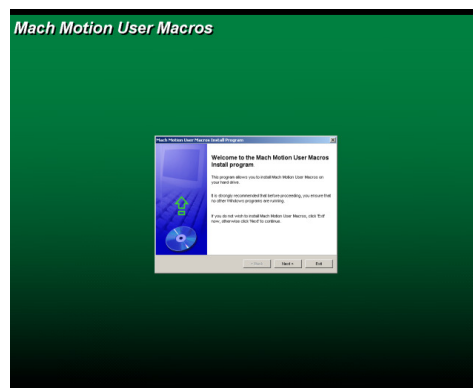


Figure 5 User Macros Installation

Press **Next** until you get to the following window:

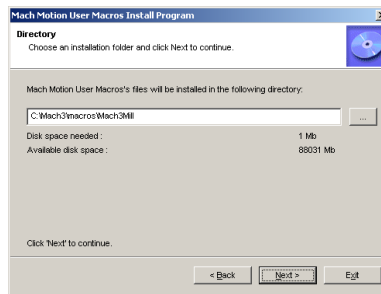


Figure 6 Macro Directory

If you are not using Mach3Mill, press the ... button to browse for the profile you are using or just type it into the file directory. Press **Next** and then **Start**.

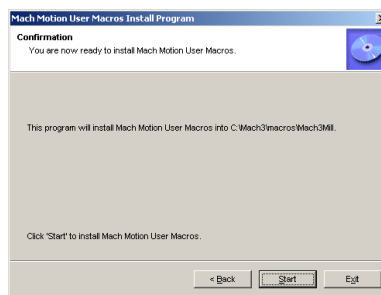


Figure 7 User Macro Confirmation

Press **Exit** to end.

The MachMotion's Ultimate Screen user macros are now installed on your computer. Now you must configure Mach3 to use the screen.

## Mach3 Setup

Begin by starting Mach3. Click **View** on the top menu bar and select **Load Screens**.

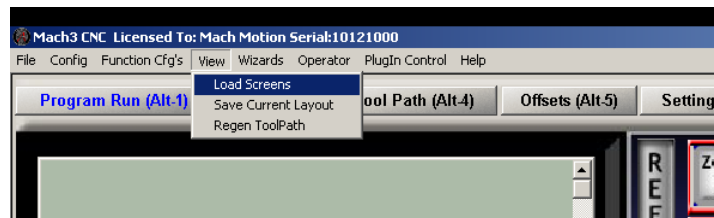


Figure 8 Load Screens

Navigate to the **C:\Mach3\Flash** folder. In the file type, select **Flash Screens** as shown below.

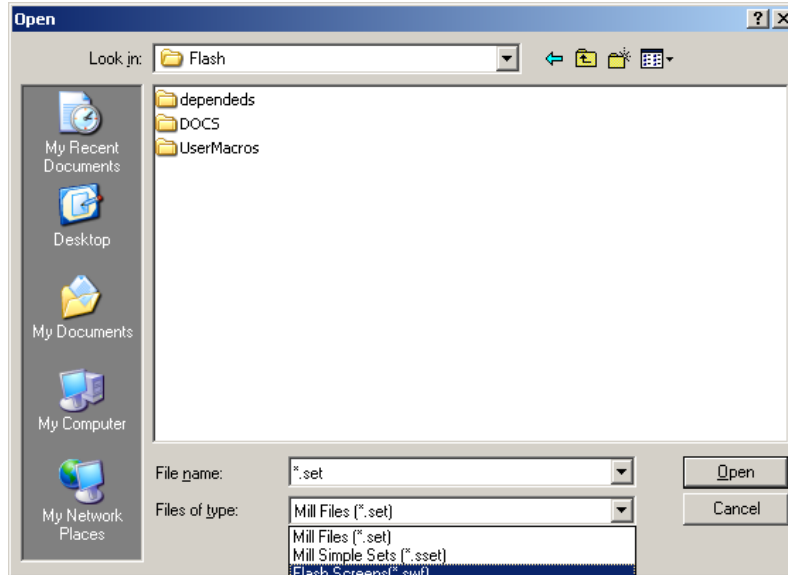


Figure 9 Select File Type

You should then see the **MachMotion2** screen. Double click on it.

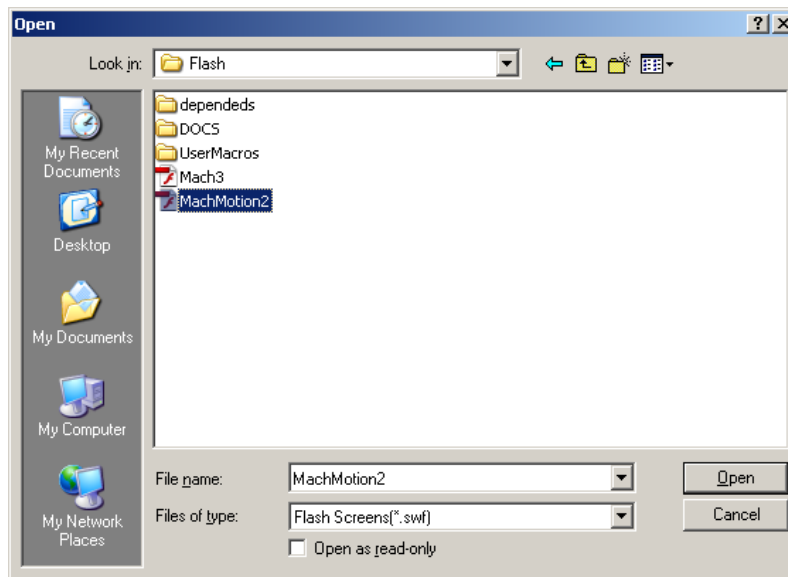


Figure 10 Open Screen

The Ultimate Screen should begin to load. Restart the Mach3 software.

If you had a previous version of the Ultimate Screen, you will NOT need a license key. Otherwise call MachMotion for a license key for your new screen. Now with your Ultimate Screen installed, it is time to learn how to use it.



# Overview

The Ultimate Screen is designed to be a powerful CNC control interface that is quick and easy to learn. This reference guide will lead you through each part of the screen and give you the resources you need to fully utilize this control interface.

## General Layout

The Ultimate Screen has 5 main tabs as shown below.



Figure 11 Main Tabs

The 5 main tabs are **Prog Run** (Program Run), **ToolPath**, **Offsets**, **MDI**, and **Diags** (Diagnostics). On every tab except MDI, the top of the screen shows the current position DROs. If the axis labels are red, then the machine has not been homed.



Figure 12 Machine Not Homed

The axis label will turn green when an axis is homed.



Figure 13 Machine Homed

If the numbers are red, then you are viewing the machine coordinates.



Figure 14 Machine Coordinates

The position DROs can also display the GoTo position. To change the top DRO display from position, machine, or GoTo coordinates, see Limits on page 15.

Above the position DROs the Ultimate Screen displays the current profile that you have loaded, the current offset that you are using (e.g. G54), and the G-code mode.



Figure 15 Program Information

Near the bottom of the screen there is a status line and a file location display. The status line will always display the latest error code or message that took place in Mach3. Make sure to watch it carefully if something does not seem to be working correctly. The file display will always show the current file that is loaded into Mach3.



Figure 16 Status Bar

On the very bottom of the screen is the bottom menu bar. Read the next section for an in-depth description of it.

## Bottom Menu Bar

The bottom menu bar enables you to easily access many different functions from any tab. Begin by selecting the function you want to use by clicking on the button. Then when you are finished, press the **Sub Menu** button to return to the main menu bar. You can also scroll the menu bar back and forth with the right and left arrow keys.



Figure 17 Bottom Menu Bar



Figure 18 Sub Menu Button

## Reset

On the right side of the menu bar is the **Reset** button. **Reset** is used to stop everything immediately. It turns off all outputs. However, to disable your motors you must press the big emergency stop button on the Operator Panel (or another External Emergency Stop). After **Reset** has been pressed there is no way restart your G-code program. You must rewind the program and start from the beginning or use the **Run From Here** button (See Advanced on page 26).



Figure 19 Reset Button

## MDI

The **MDI** menu displays a large MDI (Manual Data Entry) line so you can command your machine from any page. Click on the actual MDI line to enter your command. You can type any valid line that could appear in a part program and it will execute upon pressing the *Enter* key or clicking **SEND** on the on-screen keyboard. Discard the line by pressing *Esc* or by pressing the **STOP** button.



Figure 20 MDI

## User Buttons

The **USER Buttons** menu has 6 user buttons that can be configured for any function. The 6 user buttons are also displayed in the **Prog Run** tab under the **User** section (See User on page 28).



Figure 21 User Buttons

You can edit them by pressing the **Edit Buttons Function** and then selecting the button you want to edit. Press the **DONE** button when you have finished editing. Two Notepad documents will open up. The file **usercodeX** is run when the button is pressed. The file named **usercodeXr** is run after the button is released. The X stands for the user button (1-6). This gives you complete control of the buttons.

You can change the button text by using **SetUserLabel(User Label #, "PLACE TEXT HERE")**. You can also turn on the LED behind the button by using **SetUserLED(LED #, 1)** and you can turn it off by using **SetUserLED(LED #, 0)**. Use the table below as a reference.

User Button	Run On Press	Run On Release	User Label #	LED #
User 1	Usercode1	Usercode1r	23	1801
User 2	Usercode2	Usercode2r	24	1802
User 3	Usercode3	Usercode3r	25	1803
User 4	Usercode4	Usercode4r	26	1804
User 5	Usercode5	Usercode5r	27	1805
User 6	Usercode6	Usercode6r	28	1806

Follow the instructions below to program the buttons for two common configurations.

### Push Button

In the usercodeX file place this code:

**SetUserLabel(User Label #,"BUTTON ON TEXT")**

**SetUserLED(LED #,1)**

**'ADD YOUR CODE HERE:**

In the usercodeXr file place this code:

**SetUserLabel(UserLabel #,"BUTTON OFF TEXT")**

**SetUserLED(LED #,0)**

**'ADD YOUR CODE HERE:**

For example, the button scripts below will turn on output 1 as long the button 1 is held down.

#### **Usercode1:**

SetUserLabel(23,"Output 1 On")

SetUserLED(1801,1)

ActivateSignal(Output1)

#### **Usercode1r:**

SetUserLabel(23,"Output 1 Off")

```
SetUserLED(1801,0)
```

```
DeActivateSignal(Output1)
```

### **Toggle Button**

In the usercodeX file place this code:

```
If Not(GetOEMLED(LED #)) Then
```

```
SetUserLabel(User Label #,"BUTTON ON TEXT")
```

```
SetUserLED(LED #,1)
```

```
'ADD YOUR CODE HERE:
```

```
Else
```

```
SetUserLabel(User Label #,"BUTTON OFF TEXT")
```

```
SetUserLED(LED #,0)
```

```
'ADD YOUR CODE HERE:
```

```
End If
```

In the usercodeXr file place this code:

```
' Nothing
```

For example, the button scripts below will toggle output 1 when button 6 is pressed.

#### **Usercode1:**

```
If Not(GetOEMLED(1806)) Then
```

```
SetUserLabel(28,"Output 1 On")
```

```
SetUserLED(1806,1)'USER LED 6 ON
```

```
ActivateSignal(Output1)
```

```
Else
```

```
SetUserLabel(28,"Output 1 Off")
```

```
SetUserLED(1806,0)'USER LED 6 OFF
```

```
DeActivateSignal(Output1)
```

```
End If
```

#### Usercode1r:

```
'Nothing
```

When you finish editing a file, press the X to close it out. If it asks if it should save the changes, press **Yes**. For more information on coding in VB see the manuals **Mach3 Script Calls** and **Script Language** on the **Diagns->Reference** tab.

## Ref Home

The **Ref Home** menu allows you to home your machine. You can home all your axes simultaneously by pressing the **Ref Home** button or you can home each axis individually by pressing the **Ref** axis buttons.



Figure 22 Ref Home

## Offsets

The **Offsets** menu allows you to configure your current offset (G54, G57, etc). To zero your offsets you can press **Zero All** or you can zero each axis's offset individually by pressing the separate **Zero** buttons.



Figure 23 Offsets

## GoTo

The **GoTo** menu gives a quick and easy way to go to the part zero, the machine zero, and the tool changer position (which is configured in Diagns->Mach Settings). Press any of the **Goto** buttons and your machine will begin to move to the defined location.



Figure 24 GoTo

## Limits

The **Limits** menu allows you to switch between different coordinate displays and enable or disable various limits.



Figure 25 Limits

You can view the Goto position coordinates by pressing the **View To Go** button. The next two buttons (**View Machine Cords**, **View Part Cords**) allow you to switch between part and machine coordinates. All three buttons change the main position DROs on the top of the screen.

The button **Z Inhibit** allows you to setup a limit for the Z axis. The limit is only used for files or MDI commands. It does not stop you from jogging past the Z inhibit value. After enabling Z inhibit, you can enter in a limit into the button.



Figure 26 Z Inhibit

Soft limits can be enabled or disabled by pressing the **Soft Limits** button. You can also turn your system off line by pressing the **ON LINE** button. If your control is off line, all external motion will be disabled.

## Control

Inside the **Control** menu you can press **Cycle Start**, **Feed Hold**, **Stop**, and **Rewind**. See Control Buttons on page 23 for a description of each button.

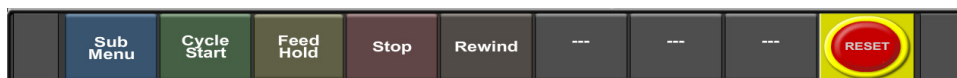


Figure 27 Control

## Tool Path

The **Tool Path** menu allows you to configure the tool path display however you want it.



Figure 28 Tool Path

### *Regen*

**Regen** regenerates the tool path display from the G-Code file.

### *Jog Follow*

Selecting **Jog Follow** causes the tool path display to always keep the machine position in the center of the screen.

### *Job View*

Pressing **Job View** gives an up-close view of the part.

### *Machine View*

Pressing **Machine View** gives an overview of the whole machine table.

### *Rotate*

After selecting **Rotate** you can click inside the tool path display and rotate your part in any direction.

### *Pan*

Selecting **Pan** disables **Rotate** and allows you to move the tool display around.

### *Zoom*

After clicking the **Zoom** button you can zoom in and out of the tool path display by clicking inside the tool path and moving your mouse up and down.

## **Gcode**

To reach the **Gcode** menu, make sure to press the small arrow on the right side of the **RESET** button. You can open, close, or edit G-code from this menu. For more information see Load on page 24.

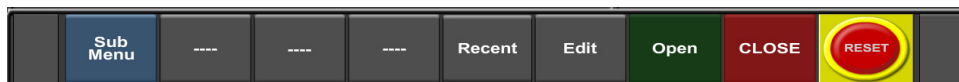


Figure 29 G-Code

## **Error Codes**

The **Error Codes** menu allows you view the status line history. To view the previous error codes, press the **Status History** button. Otherwise you can clear all the existing status errors by clicking **Clear Statuses**.



Figure 30 Error Codes



## Scripting

In the **Scripting** menu you can open the brain control window, the brain editor, or the VB script editor. See Reference on page 41 and <http://www.machsupport.com/videos/> for more information.



Figure 31 Scripting

## Tool Panels

In the **Offsets** tab, **MDI** tab, and **Diagns** tab there is a group of tool panels which allow you to easily access different functions. Each tool panel can be enlarged for more functionality by pressing the **More** button. The Ultimate screen has one for jogging, spindle, feedrate, and tools.

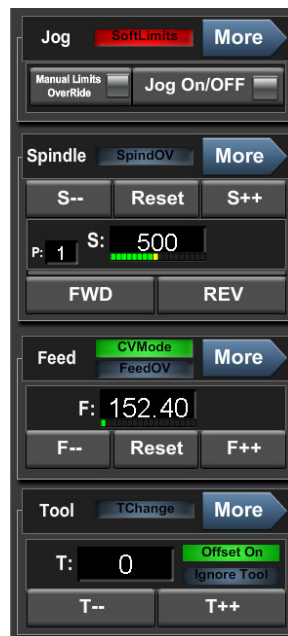


Figure 32 Tool Panels

## Jog

The **Jog** tool panel allows you to turn jog on and off (**Jog On/Off**). It also allows you to manually override a limit switch (**Manual Limits OverRide**). The status LED **SoftLimits** shows if the soft limits are enabled or disabled. If you enlarge the panel you can switch between Continuous, Step, and MPG mode. For more information on the different jogging modes, see Jog on page 28.



Figure 33 Jog Tool Panel

## Spindle

The **Spindle** tool panel allows you to increase (**S++**), decrease (**S--**), or reset the spindle speed to 100% (**Reset**). You can also change the pulley (**P**), enter in a new speed (**S**), or turn the spindle on forward (**FWD**) or reverse (**REV**). The status LED **SpindleOV** is only on (red) if the spindle speed is affected by a spindle override speed (i.e. if the spindle override percentage is anything but 100%). If you enlarge the tool panel, it also gives you the actual spindle speed (**TSpindle**), the spindle override speed (**SO**), and the spindle override percentage (%).

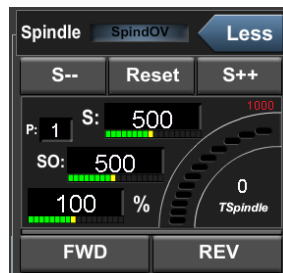


Figure 34 Spindle Tool Panel

## Feedrate

The **Feed** tool panel allows you to increase (**F++**), decrease (**F--**), or reset the feedrate to 100% (**Reset**). You can also enter in a new feedrate (**F**). The status LED **CVMode** will be green if the control is in constant velocity mode. If the feedrate override is used, then the **FeedOV** will be red (i.e. if the feedrate override percentage is anything but 100%). If you enlarge the tool panel, it also gives you the actual feedrate in units per minute (**Units/Min**), the feedrate override (**OV**), and the feedrate override percentage (%).

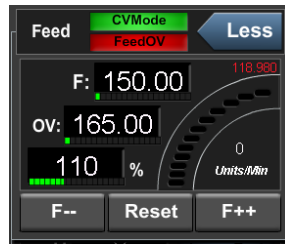


Figure 35 Feedrate Tool Panel

## Tools

The Tools tool panel allows you to easily change tools using the **T—** or **T++** buttons. It always shows the current tool (**T**), whether or not a tool change is taking place (**TChange**), if the tool has an offset (**Offset On**), and if the tool change is suppose to be ignored (**Ignore Tool**). If you enlarge the tool panel, it also gives you the tool description (**Empty** in the example below), the tool height (**H**), and the tool diameter (**D**).

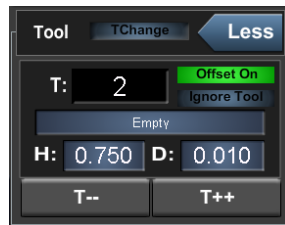


Figure 36 Tools Tool Panel

## User Input

To adjust any value in the Ultimate Screen, start by clicking on the value you want to change. If your on-screen keyboard is turned on, you will see a keyboard appear. Depending on the user input, you may get a number pad or you may get the entire keyboard.

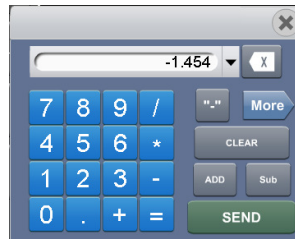


Figure 37 On-Screen Number Pad



Figure 38 On-Screen Keyboard

Enter your value into the keyboard and then press **SEND**.

You can also just input data directly from the keyboard. If your on-screen keyboard is turned off, then this is the only way to enter data. Click on the user input, type in your value, and then press *Enter* to save it.

# Main Tabs

## Program Run

You will probably spend most of your time in the Mach3 software using the **Prog Run** tab. It provides all the functionality you need to run your machine.



Figure 39 Prog Run

On the left of the **Prog Run** tab the tool path window displays the loaded part. You can regenerate the tool path display by pressing **Regen** or you can reload the file by pressing **ReLoad**.

Below the tool path display you can view the loaded G-code file. Under the G-code display it shows the program part number, the current G-code line that the program is on, and the time that has elapsed since the program started.

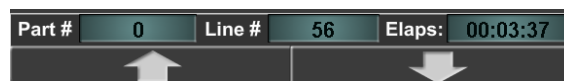


Figure 40 G-Code Information

On the top right of the screen it has the **Spindle** and **Feedrate** tool panels in addition to a bunch of status LEDs. For more information on the tool panels, see Spindle on page 18 and Feedrate on page 18. The small DROs at the top of the screen show the **Goto** position.

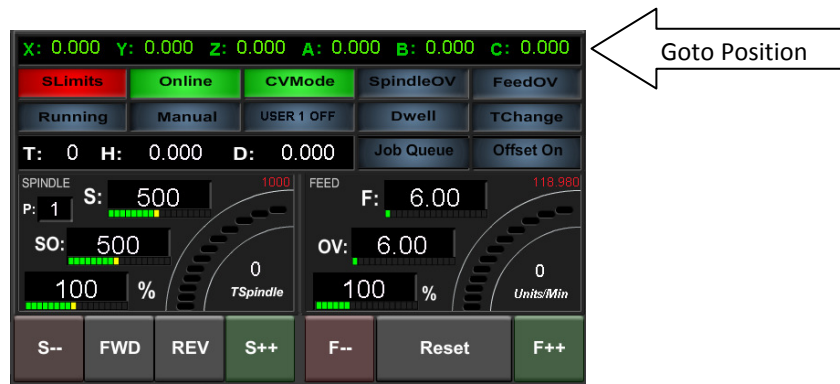


Figure 41 Program Run Tool Panels

Below is a short description of all the different status LEDs.

### *SLimits*

If the soft limits are enabled, this will be green. See Limits on page 15.

### *Online*

Mach3 has an online and offline feature. If this is red, nothing you do in Mach3 will cause the machine to move. See Limits on page 15.

### *CVMode*

If your control is running in constant velocity mode, then this will be green.

### *SpindleOV*

If your spindle speed is overridden, this will be red.

### *FeedOV*

If your feedrate is overridden, this will be red.

### *Running*

This turns green whenever a G-code file is running.

### *Manual*

When manual mode is enabled, this will be green.

### *User 1 Off*

This LED can be controlled from any macro or VB script. The LED is user # 1808 and the label is user # 22. See the manual **Mach3 Script Calls** on the Diagnostics->Reference tab.

### *Dwell*

If G04 is used or if Mach3 is told to dwell for any period of time (like after turning the spindle on), this LED will be green until the dwell has completed.

### *TChange*

During a tool change this LED turns green.

### Job Queue

If the job queue is enabled this will be green. See Load on page 24.

### Offset On

If a tool offset is active, this will be green.

The Prog Run page also always displays the current tool, the tool's height, and the tool's diameter.

T: 0 H: 0.000 D: 0.000

Figure 42 Tool Information

## Control Buttons

The **Prog Run** tab has a small window with 5 different tabs. Each tab displays cycle start, feed hold, stop, and rewind at all times.

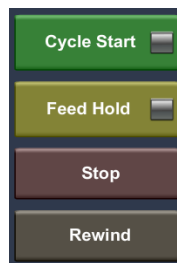


Figure 43 Control Buttons

Below is a description of each control button.

### Cycle Start

**Cycle Start** tells the control to start the loaded program.

### Feed Hold

If you are running a file and press **Feed Hold** the motion will pause before completing the current line of G-code depending on how long the current move is. **This is an immediate stop!** **Cycle Start** can then be pressed to complete the current line and run the file to the end. After **Feed Hold** has been pressed there is no way to rewind the program without pressing **Reset** first.

### Stop

**Stop** is used to stop everything inside Mach3. It cancels all scripts and macros, stops your program, and turns off the spindle, flood, and mist. You cannot start the program again after pressing **Stop** unless you use **Run From Here** or start again at the beginning.

### Rewind

**Rewind** can only be pressed when a file is not running and not paused from a **Feed Hold**. Rewind resets the program back to the starting line.

To run a program, begin by loading your program into the controller. When the program is loaded, press the **Cycle Start** button on the Ultimate Screen or on the Operator Panel. Your program file will begin to execute.

Read the section below for more information about each tab in **Prog Run**.

## Load

The first tab is **Load**. It consists of all the buttons you need to load and edit your files.

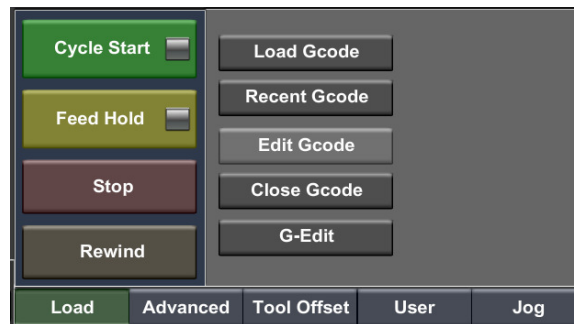


Figure 44 Load Tab

### Load Gcode

Opens up a browse window for you to locate your g-code file. After locating your file, press **Open** and your file will load.

### Recent Gcode

Displays a list of all the recently opened files. Select the file you want to load and press **OK**.

### Edit Gcode

Opens up a Notepad window which allows you to edit your program.

### Close Gcode

Closes the loaded G-code file.

### G-Edit

Opens up the G-code file in an on-screen editing software.

### Open Job Queue (Only displayed if enabled in **Diagns->HMI Settings**)

Allows you to load in multiple G-code files and organize them in the order you want to run them. After loading in all your programs, press **Run Job Queue**. When you press **Cycle Start** your job queue will begin executing.





Figure 45 Job Queue

### *Scroll Down*

Selects the next file below the currently selected file.

### *Scroll Up*

Selects the next file above the currently selected file.

### *Cycle Start Between Jobs*

After each file is run, you will have to press **Cycle Start** again to continue. Otherwise it will run all the way through.

### *Delete Job*

Deletes the selected job.

### *Add Job*

Brings up a window to be able to browse for your G-code files. After selecting your file, press **Open** and it will appear in the *Job Queue* window.

### *Move Up*

Moves the selected program up one level.

### *Move Down*

Moves the selected program down one level.

### *Clear Queue*

Clears all the programs from the job queue.

### *Save*

Saves the current job queue.

### *Close Job Queue*

Closes the job queue setup window.

## Advanced

The **Advanced** tab offers more advanced options for running your G-code programs.

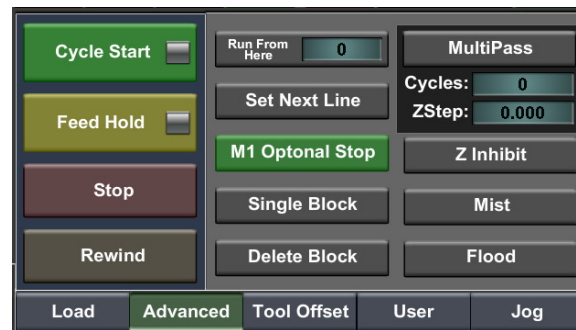


Figure 46 Advanced Tab

### Run From Here

Use the **Run From Here** button to start from the middle of a program. You can either enter the desired line number in the button or use the up and down arrow keys below the G-code display to find the correct place in your program. Press **Cycle Start** when you are ready to begin. The controller basically simulates the program up to the desired line and loads in the current offsets and feedrates. The controller will ask you if you want the motors to start at the actual G-code position. When everything is in place, you can press **Cycle Start** again and your program will begin.

### Set Next Line

Another way to start from the middle of a program is to use the **Set Next Line**. You can use the up and down arrow keys to find the correct place in your program. Press the **Set Next Line** button. Then press **Cycle Start** when you are ready to begin. Any code above the run from here line will be skipped. Regardless of where the motors are, the program will begin running from the current position.

### M1 Optional Stop

**M1 Optional Stop**, when activated, makes the controller stop at every M1 in the program. You must press **Cycle Start** to begin again after a M1.

### Single Block

**Single Block** allows you to execute the program one line at a time.

### Delete Block

**Delete Block** allows you to skip desired lines of code when activated. It is rarely used.

### MultiPass

**MultiPass** allows you to run a G-code file multiple times while moving the Z between each run. **Cycles** is how many times the file will be run and the **ZStep** is the distance that the Z axis will move between each run.

### Z Inhibit

**Z Inhibit** limits the distance that the Z axis can go. Press the button and then you can enter in the inhibit value. You can set the inhibit value to the top of your part and it will never move below it during a program or a MDI line command. Note that it does not stop you from jogging below the inhibit value.

### *Mist*

**Mist** turns on and off your mist control.

### *Flood*

**Flood** turns on and off your flood control.

## Tool Offset

The **Tool Offset** tab is used for setting up and changing tools.



Figure 47 Tool Offset Tab

You can change the current tool by pressing the **T—** or the **T++** buttons. In between the two tool change buttons it displays the current tool. Below the buttons it shows the tool's description. It also lists the tool's diameter and height. Also notice that it allows you to enter a gage block if you are using one (**Current Gage Block Height**).

### *Set Tool Length*

**Set Tool Length** loads in a new value for tool height.

### *Offset On/Off*

**Offset On/Off** turns the tool offset on or off. Note that pressing **Set Tool Length** also turns on the tool offset.

### *GoTo Tool Change Position*

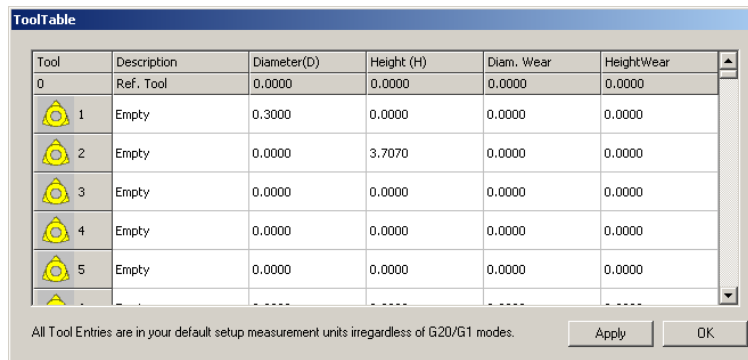
**GoTo Tool Change Position** tells your machine to move to the defined tool change position (See Mach Settings on page 38).

### *Ignore M6 ToolChange*

**Ignore M6 ToolChange** disables M6 so changing tools only affects the tool number.

### *Tool Table*

**Tool Table** opens up a table with all the tool information including the description, diameter, and height.



Tool	Description	Diameter(D)	Height (H)	Diam. Wear	HeightWear
0	Ref. Tool	0.0000	0.0000	0.0000	0.0000
1	Empty	0.3000	0.0000	0.0000	0.0000
2	Empty	0.0000	3.7070	0.0000	0.0000
3	Empty	0.0000	0.0000	0.0000	0.0000
4	Empty	0.0000	0.0000	0.0000	0.0000
5	Empty	0.0000	0.0000	0.0000	0.0000

All Tool Entries are in your default setup measurement units regardless of G20/G1 modes.

Apply OK

Figure 48 Tool Table

## User

The **User** tab displays all the current user buttons. These can be configured under the **USER Buttons** menu on the bottom menu bar (See User Buttons on page 11).



Figure 49 User Tab

The other labels and values above the user buttons can be written to from any macro or VB script. See the manual **Mach3 Script Calls** on the Diagnostics->Reference tab. Use the table below as a reference.

Label	User Label #	User DRO #
AMPS	29	2000
L	30	2001
K	31	2002
P	32	2003

## Jog

The **Jog** tab allows you to select the way you want to jog. You can jog continuously, incrementally, or by using an external MPG. You can also turn the jog completely off by pressing the **Jog On/Off** button.



Figure 50 Jog Tab

In **Continuous** mode you can enter in a jog rate or adjust it by 5 by pressing the **Jog—** or **Jog++** buttons. Pressing **Reset** sets it back to 100. You can also manually override the limit switches by pressing **Manual Limits OverRide**.



Figure 51 Continuous Jog

In **Step** mode each time a jog button is pressed, the selected axis will move the step increment. Set your step size by entering it in or by pressing one of the buttons (0.0001 – 1.0).

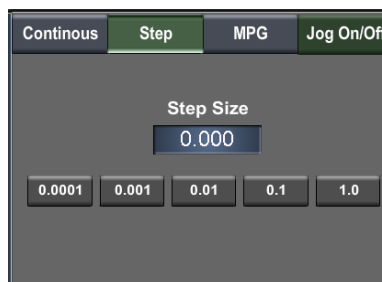


Figure 52 Step Jog

MPG mode is only used if you have an external pendant with a MPG. Use the button **Cycle Axis** to switch between different axes. You can also select your jogging mode (velocity, step/velocity, single step, & multi step).



Figure 53 MPG Jog

## Tool Path

The **ToolPath** tab is very similar to the **Prog Run** tab, except that it has a much larger tool path display and only a few basic functions. You may also choose to run your programs from this mode.



Figure 54 Tool Path Display

You can load or edit G-code with the buttons right above the G-code display. The **ToolPath** tab also gives you the **Estimated Program Run** time. On the top of the tool path display it gives the file limits. These are the maximum and minimum positions that the machine will travel in the program.

Consult Control Buttons on page 23 and Advanced on page 26 for more information about the functions/buttons in the **ToolPath** page.

## Offsets

The **Offsets** tab has 5 different tabs with different groups of offsets and settings. On the right side of the page it has the four main tool panels. Examine the section below for more information about the different tabs inside the **Offset** page.



Figure 55 Offsets

## Homing

The first column of DROs in the **Homing** tab is the machine coordinates. You can reference all the axes by pressing **Reference All Home** or you can individually reference each axis by pressing the **Ref** button for the correct axis. When the LED next to the axis label turns green, the axis is homed. You can also de-reference all the axes by pressing the **Def-Ref All Axis**.

Your machine's soft limits are listed on the right side of the homing tab. **Note that you cannot update your soft limits here.** Go to **Config->Homing/Limits** to edit your soft limits. If the **SoftLimits** button is on, then you can never move your machine outside of the limits. The control will automatically stop all motion.

The **G28 Home Coordinates** are listed on the bottom right of the **Homing** tab. The G28 coordinates define the position in absolute machine coordinates to which the axes will move when a G28 is executed.



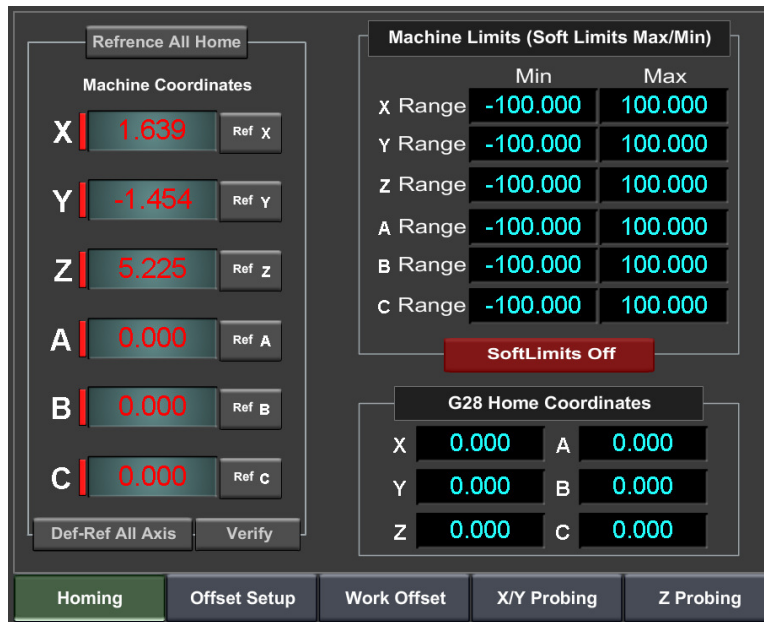


Figure 56 Homing

## Offset Setup

The **Offset Setup** tab gives you all the tools you need to setup your work and tool offsets. For more information on setting up your tools see the **Tool Setup** manual from [www.machmotion.com/support-overview/documentation.html](http://www.machmotion.com/support-overview/documentation.html).

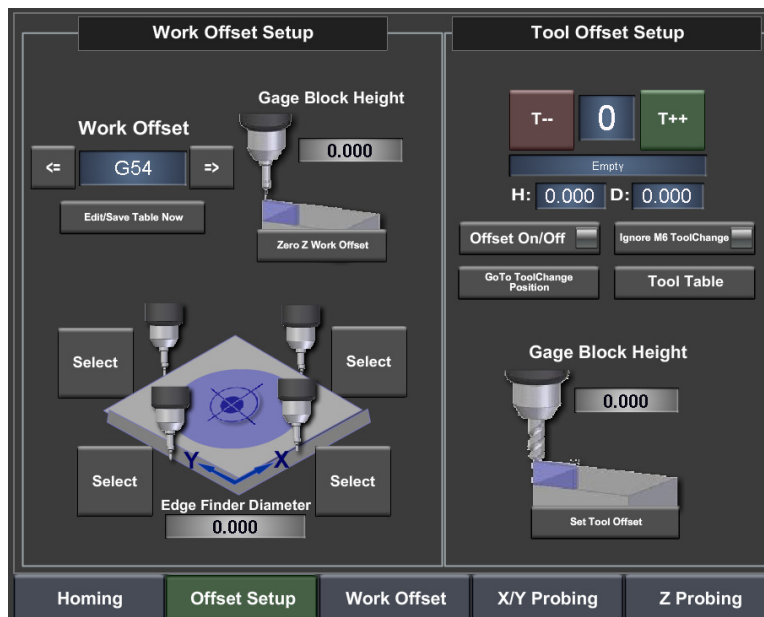


Figure 57 Offset Setup

## Work Offset

The **Work Offset** tab shows how the machine coordinates are actually calculated. You can switch between your different work offsets by pressing the <= and => buttons. The program coordinates can be zeroed by pressing the small **Zero** button underneath the coordinate display. You can also update the work offset by pressing the small **Touch** button underneath the offset display. To home an axis press the **Ref** button underneath the machine coordinate display.

The screenshot shows the 'Work Offset' tab with the following layout:

- Top Section:**
  - Work Offset:** Includes 'Edit/Save Table Now' and buttons for switching between G54 and G52/G92 offsets.
  - TableOffset:** Includes 'Edit/Save Table Now', a 'Tool' selection (currently 0), and an 'Offset On/Off' toggle.
- Coordinate Displays:**
  - X coord:** 0.000 (Zero) + 1.639 (Touch) + 0.000 = X 1.639 (Ref)
  - Y coord:** -1.454 (Zero) + 0.000 (Touch) + 0.000 = Y -1.454 (Ref)
  - Z coord:** 5.225 (Zero) + 0.000 (Touch) + 0.000 + 0.000 (Length Touch) = Z 5.225 (Ref)
  - A coord:** 0.000 (Zero) + 0.000 (Touch) + 0.000 = A 0.000 (Ref)
  - B coord:** -23.240 (Zero) + 0.000 (Touch) + 0.000 = B 0.000 (Ref)
  - C coord:** -25.000 (Zero) + 0.000 (Touch) + 0.000 = C 0.000 (Ref)
- Additional Features:**
  - Tool Diameter:** 0.000 (Touch)
  - Applied with G41/42 in Program Coord**
  - Touch Correction:** 0.000
  - Home key Ref ZXY**
- Bottom Navigation:** Homing, Offset Setup, **Work Offset** (active), X/Y Probing, Z Probing

Figure 58 Work Offset

You can also view and edit all your work offsets by clicking on the **Edit/Save Table Now** button.

The screenshot shows the 'Work Offsets' table with the following data:

G-Code Pos	X	Y	Z	A	B	C	Name
G54	0.0000	0.0000	0.0000	0.0000	23.2400	25.0000	G54
G55	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	
G56	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	
G57	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	
G58	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	
G59	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	
G59P7	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	
G59P8	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	

Buttons at the bottom: Select, Save, Close

Figure 59 Work Offset Table

Use this formula to calculate the machine coordinates:

Program Coordinate + Work Offset + G52/G92 Offset + Tool Offset (For the Z and possibly X) = Machine Coordinate

### **X/Y Probing & Z Probing**

X/Y Probing and Z probing are currently not used. Just ignore these tabs.

## MDI

The **MDI** tab is used for manually moving your machine. On the top left it lists all the different axes' positions, with a zero button and a homing button for each. Below the axis positions there are a few common buttons. For descriptions of each read Control Buttons on page 23 and Advanced on page 26.

In the middle of the screen there is a tool path display. Below the small tool path display, it lists all the machine limits again. The MDI line is located below the machine limits table. On the right side of the page it has the four main tool panels. You can also open up M-code or G-code help (**Mcode Help**, **Gcode Help**) for more detailed descriptions of each code.



Figure 60 MDI

The MDI line is used to command your machine. You can enter any G-code or M-code into the MDI line and it will run. See MDI on page 11 for more information.

## Diagnostics

The **Diagns** tab is used for setting up your I/O, monitoring data, and configuring Mach3. Like the **Offsets** tab, the **Diagns** tab has 5 different tabs. Read the section below for more information about each tab.



Figure 61 Diagnostics

## I/O

While setting up your inputs and outputs, you may spend some time in the I/O tab.

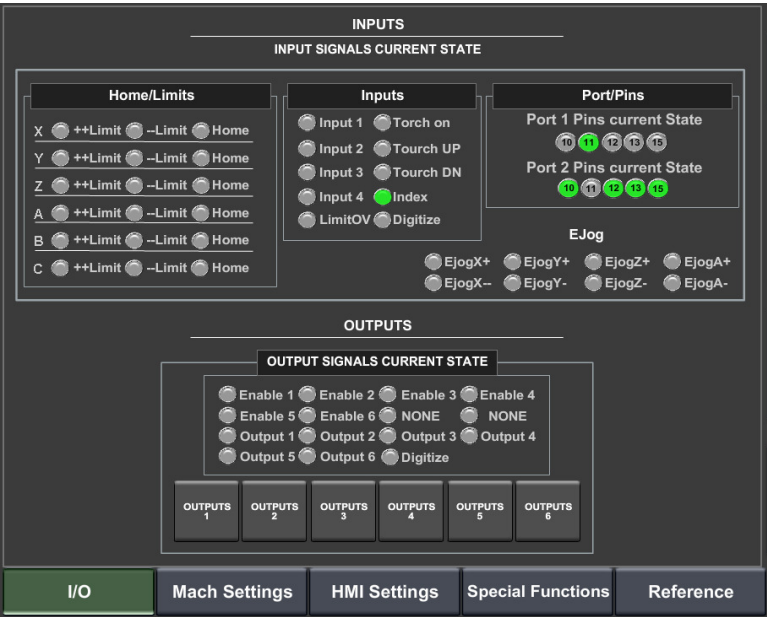


Figure 62 I/O

### Home/Limits

The Home/Limits section gives the current state of all the limit switches and home switches for all the different axes. The *++Limit* is the forward limit, the *--Limit* is the reverse limit, and the *Home* is the homing switch.

### Inputs

The current states of inputs 1 – 4, limit override, torch inputs, index, and digitize are all listed in this box.

### Port/Pins

This box displays the current state of the inputs from the breakout board. These pins are assigned inside Config->Ports & Pins to different signal names (e.g. Input 1, Index, etc).

### EJog

External jog inputs for the X-A axes are also listed here.

### Outputs

Enable signals 1-6, outputs 1-6, and the digitize output are all displayed here. If an output is enabled in ports and pins, you can press the **Output X** button to turn on and off that output. This is an excellent way to test an output.

## Mach Settings

The **Mach Settings** tab has a lot of random Mach3 settings. Review the information below.

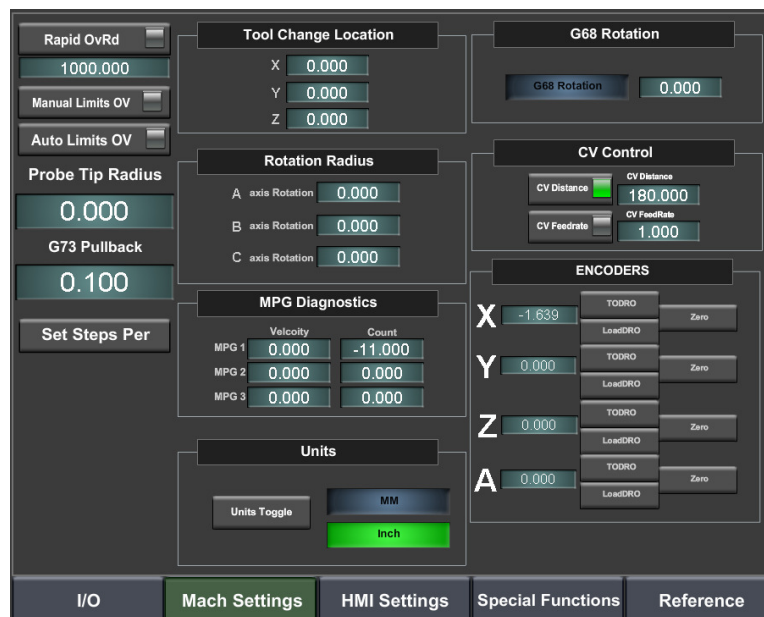


Figure 63 Mach Settings

### Rapid OvRd

Turns on the rapid override.

### *Manual Limits OV*

If a limit switch is activated, the control will not allow you to jog unless manual limits override is selected. If auto limits override is enabled, then manual limits override will happen automatically when you reset the control.

### *Auto Limits OV*

Turning on **Auto Limits OV** causes the control to automatically turn the **Manual Limits Override** on after resetting the control from a triggered limit switch.

### *Tool Change Location*

Set your tool change position for the X, Y, and Z axes here.

### *Rotation Radius*

Setup your rotational radius for the A, B, and C axes in this section.

### *MPG Diagnostics*

Mach3 can have up to three MPGs. This section gives the current velocity and count of each MPG.

### *Units*

You can toggle the units from mm to inch or inch to mm by pressing the **Units Toggle** button.

### *G68 Rotation*

To rotate your coordinate system, enter a value for G68.

### *CV Control*

CV Distance is the number of inches or mm that the machine will move outside of its motion path to keep a constant velocity. If it exceeds the distance value, then the velocity will not be constant any more.

### *Encoders*

If Mach3 has separate encoder inputs, you can copy the counts to the axis DROs, zero them, or write to them.

## **HMI Settings**

The **HMI Settings** tab allows you to change the actual Ultimate Screen interface.

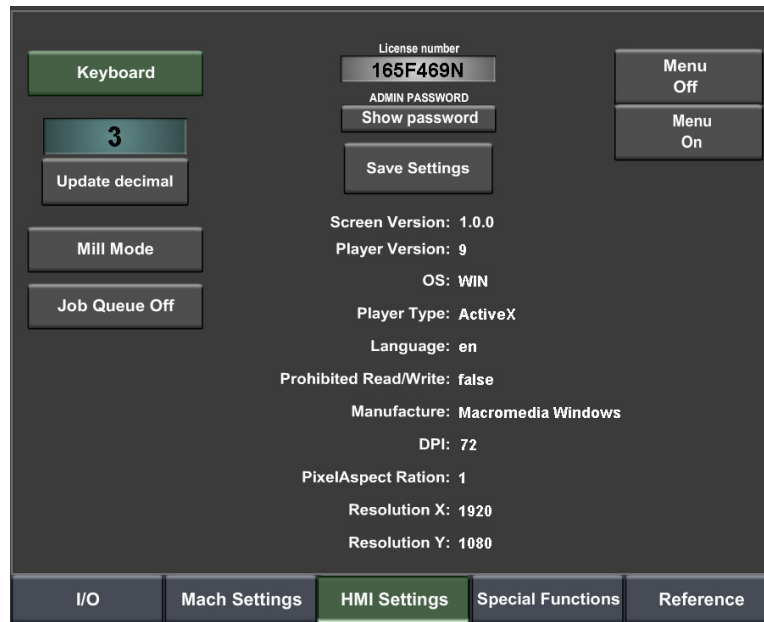


Figure 64 HMI Settings

The **HMI Settings** page displays your screen version, the operating system, resolution, and all other information about the screen. Below are the descriptions of the buttons.

### *Keyboard*

The **Keyboard** button turns on or off the on screen keyboard. If it is green, then the on-screen keyboard will appear whenever you click on a user input. If it is gray, then you have to enter your values directly from an actual keyboard (remember to press *Enter* or your value will not be saved!).

### *Update decimal*

**Pressing Update** decimal updates how many decimal places are displayed in all the user inputs. To have more accuracy, set the number above the button to 4 or 5 and then press the button.

### *Mode*

The **Mode** button allows you to switch between Mill Mode, Torch Mode, and Water Jet Mode. The modes change different tool panels and the names of different buttons.

### *Job Queue*

If the Job Queue is enabled, then a Job Queue button will be displayed on the run mode under the load tab. Job queue allows you to load in multiple programs and run them all at the same time. See Load on page 24.

### *Show Password*

If you click **Show Password**, the current **Diagns** password will be displayed. Otherwise it will give a blank input which will allow you to enter in your own password. The password is used to keep people out of the **Diagns** tab. If any text is placed in this user input, then you will not be able to access the **Diagns** tab without entering the same code again.



### Save Settings

Any changes made to the Ultimate Screen (except the user button's code) will not be saved unless this button is pressed.

### Menu Off and Menu On

To turn the top menu on or off, use the two menu buttons. You may want the menu hidden so the user does not mess anything up.

## Special Functions

The Special Functions tab provides access to a few other special functions.

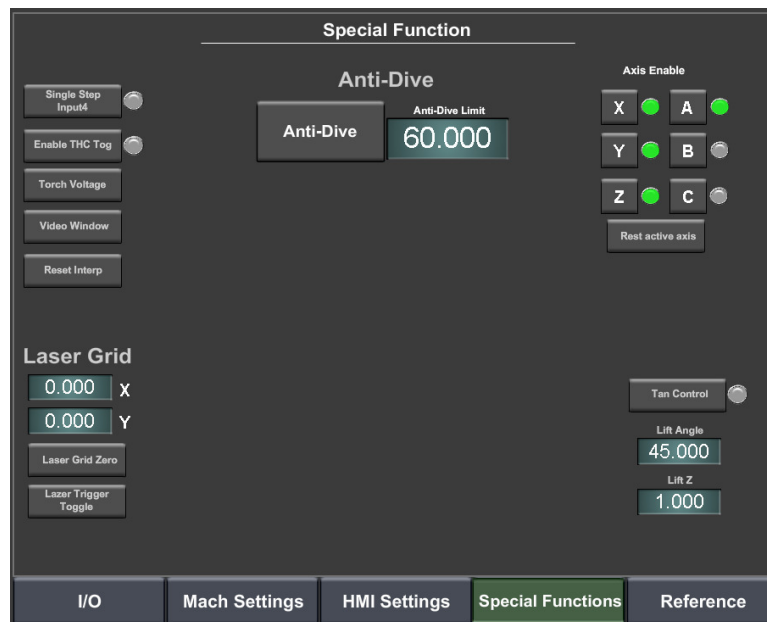


Figure 65 Special Functions

You can setup your torch settings, enable your axes, or use the laser grid and tan control.

## Reference

The **Reference** tab provides different resources for using Mach3. Click on the manual you want to read and the manual will display as a PDF in a separate window.

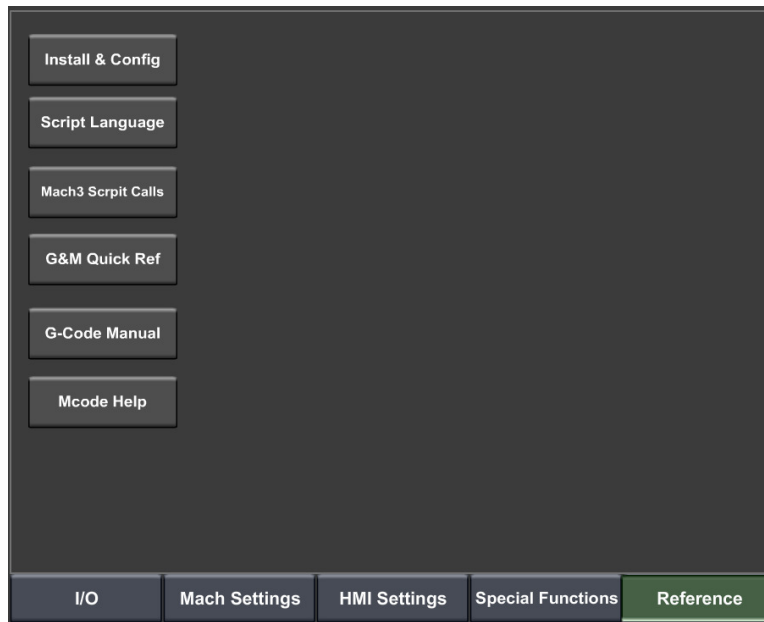


Figure 66 Reference

Below is a short description of each manual:

#### ***Install & Config***

To learn how to install, configure, and use the Mach3 software.

#### ***Script Language***

To learn how to use the visual basic programming language.

#### ***Mach3 Script Calls***

To learn other visual basic programming commands specific for Mach3.

#### ***G&M Quick Ref***

To view a quick overview of the different G-codes and M-codes.

#### ***G-Code Manual***

To take a detailed look at what each g-code does.

#### ***M-Code Help***

Currently Unused.

# Screen Customization

On the very top of the screen it has the words Mach Motion. You can edit this to be your company name or anything you want. Just use the standard procedure for any user input (See User Input on page 20).

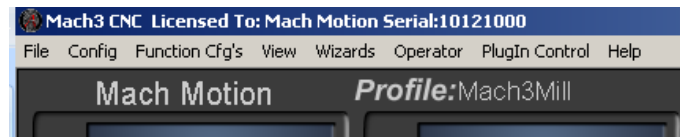


Figure 67 Screen Title

You can also change the axis names. Changing the names in the top DROs change the names everywhere in the screen.



Figure 68 Editing Axis Names

We hope that you have found this reference manual very helpful.

Please let us know if you have any questions.

Sincerely,

The Mach Motion Team

<http://www.machmotion.com>

14518 County Road 7240, Newburg, MO 65550

(573) 368-7399 • Fax (573) 341-2672