

MACHMOTION

Plasma/Oxy Torch Operations Manual

Mark Worth

4/25/2012



Everything you need to know to configure and operate your plasma and/or oxy torch system.

Copyright © 2012, MachMotion.com
All rights reserved.

MachMotion

www.machmotion.com

14518 County Road 7240, Newburg, MO 65550

(573) 368-7399 • Fax (573) 341-2672

Contents

1	Control Overview	5
1.1	Screen Overview	5
1.2	Operator Panel Overview.....	6
2	Homing and Jogging	7
2.1	Homing.....	7
2.2	Jogging	8
2.2.1	Jog On/Off	8
2.2.2	Jog Buttons.....	8
2.2.3	Jog Speed	8
2.2.4	Continuous and Step Jogging	8
3	Torch Control	9
3.1	Tabs and Plasma and Oxy Fuel Mode Selection.....	9
3.2	Plasma Settings	9
3.2.1	THC	10
3.2.2	Torch	10
3.2.3	Rip Cut.....	10
3.2.4	Pierce Counter	11
3.2.5	Pierce Height.....	11
3.2.6	THC Height	11
3.2.7	Pierce Delay.....	11
3.2.8	THC Delay	11
3.2.9	Ignition Height.....	11
3.2.10	Jog Speed	11
3.2.11	Ref Feed	11
3.3	Oxy Fuel Settings.....	12
3.3.1	Oxy Fuel PreHeat.....	13
3.3.2	Oxy Fuel Cut	13
3.3.1	Rip Cut.....	13
3.3.2	Pierce Counter	13
3.3.3	PreHeat Delay	14
3.3.4	Pierce Delay.....	14
3.3.5	Cut Height	14
3.3.6	Jog Speed	14
4	Running Files	15
4.1	Zeroing (Setting Part Zero).....	15
4.2	GOTO.....	15
4.2.1	Load.....	15
4.2.2	Part Zero.....	15
4.3	Loading G-Code Files.....	15
4.3.1	G-Code Window	15

4.3.2	Edit G-Code	15
4.3.3	Recent File.....	15
4.3.4	Close G-Code.....	16
4.3.5	Load G-Code.....	16
4.3.6	Set Next Line	16
4.3.7	Line.....	16
4.3.8	Rewind	16
4.3.9	Single BLK.....	16
4.3.10	Block Delete	16
4.3.11	M1 Optional Stop.....	16
4.4	Plate Alignment.....	16
4.5	Simulate	17
4.6	Control Buttons.....	17
4.6.1	Cycle Start	17
4.6.2	Rewind	17
4.6.3	Reset	17
4.6.4	Stop	17
5	Lost Cut Recovery	19
5.1	Lost Cut Recovery Overview	19
5.2	Lost Cut Recovery Procedure.....	20
5.2.1	Plasma – Auto	20
5.2.2	Plasma – Manual.....	20
5.2.3	Oxy Fuel.....	21
5.2.4	Drill.....	22
5.2.5	Plate Marker.....	22
6	Appendix A – Button Definitions	25
6.1	Feed Rate	25
6.2	Load Location.....	25
6.3	Max and Min THC.....	25
6.4	Pierce Reference Distance	25
6.5	Plate Height Offset.....	25
6.6	Z Rapid Height	26
7	Appendix B – Diagnostics.....	27
7.1	Tool Change Location.....	27
7.2	Lost Arc Location.....	27
8	Appendix C – MDI and Program Limits	28
8.1	Program Limits	29
8.2	MDI.....	29

This page was intentionally left blank.

1 Control Overview

This manual has been designed to lead operators through the process of operating a Victory CNC plasma system. It discusses everything from jogging and homing to lost cut recovery and plate alignment. The plasma control interface is built on the Mach3 software.

1.1 Screen Overview

Use the following figure as a reference throughout this manual for finding buttons and controls.

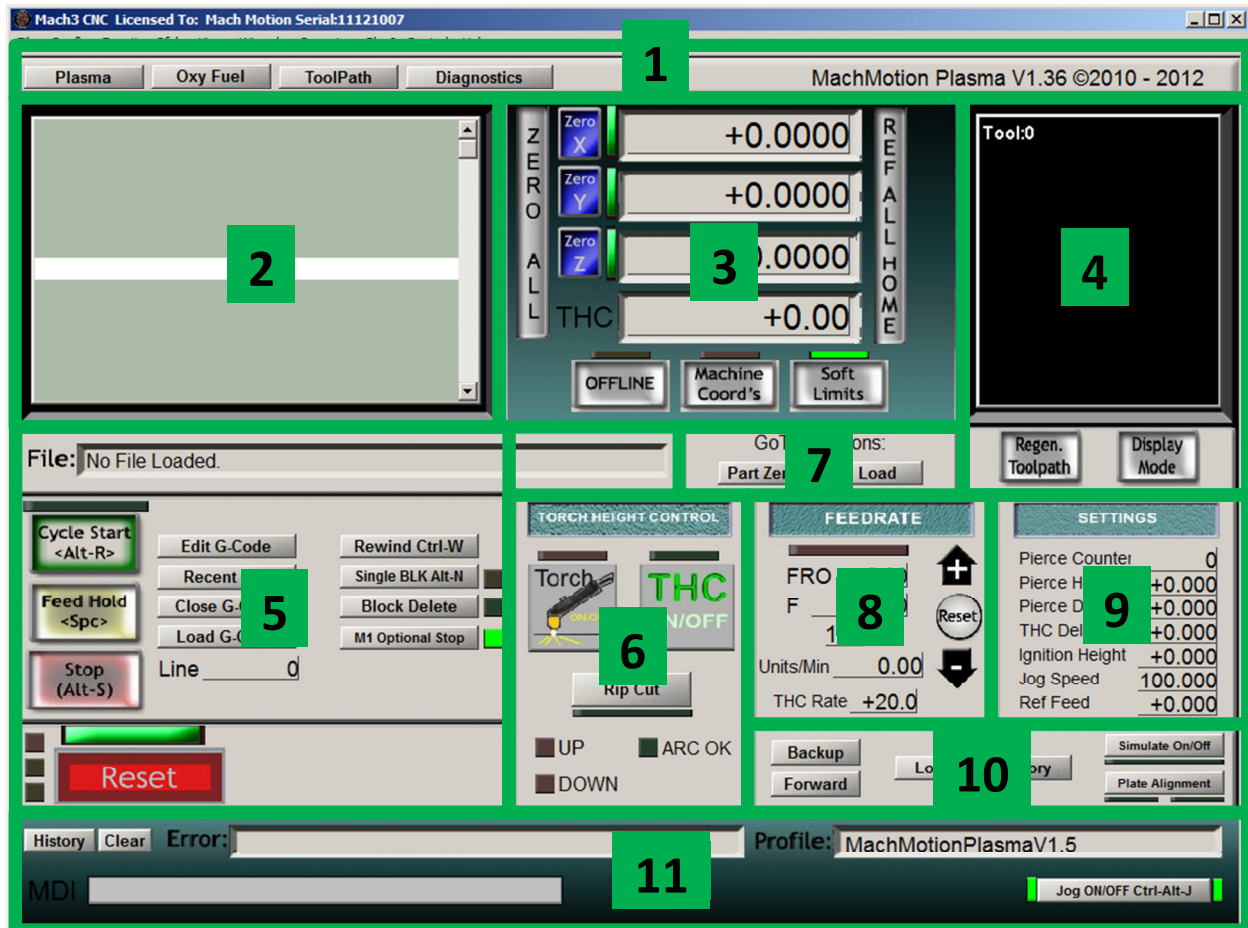


Figure 1 - Screen Break Down

For a quick reference, the description of each zone is located below.

- Zone 1. Tab Bar and Mode Selection (See Section 3.1 – Tabs and Plasma and Oxy Fuel Mode Selection)
- Zone 2. G-Code Window (See Section 4.3.1 - G-Code Window)
- Zone 3. Axis Position and Homing (See Section 2.1 - Homing)
- Zone 4. G-Code File Tool Path Rendering
- Zone 5. G-Code Operations (See Section 4.3 - Loading G-Code Files)
- Zone 6. Torch Control (See Section 3 -Torch Control)

- Zone 7. GOTO (See Section 4.2 - GOTO)
- Zone 8. Feedrate Control (See Section 6.1 - Feed Rate)
- Zone 9. Plasma and Oxy Fuel Settings (See Section 3.2 – Plasma Settings and 3.3 – Oxy Fuel Settings)
- Zone 10. Lost Cut Recover, Simulate and Plate Alignment (See Sections 5, 4.5, and 4.4 respectively)
- Zone 11. Status Field, Profile Field, Jog ON/OFF, and MDI Field (See This page was intentionally left blank)

Zone 13. Appendix C – MDI and Program Limits)

1.2 Operator Panel Overview

The operator panel is located to the right of the screen. Its purpose is to provide quick and easy access to important and frequently used information and controls.

- Meter – The digital readout shows set tip volts or actual tip volts
- Status LEDs
 - THC Up – Torch Height Control (THC) commanding the torch to be raised
 - THC Down – THC commanding the torch to be lowered
 - Torch On – Software command signal for torch on
 - Arc Ok – Signal showing the arc has established feedback from plasma cutter
 - Ready – THC is ready for operation
- Cut Height Setting Knob – Control for defining desired tip volts (cut height)
- Emergency Stop
- Cycle Start
- Feed Hold



Figure 2 - Operator Panel

2 Homing and Jogging

2.1 Homing

To reference all the axes on the homing switches use the **REF ALL HOME** button. The homing sequence does each axis individually. The Z axis is homed first, followed by the X axis, and then the Y axis. Another way to home the machine is to home each axis individually using the **Ref** buttons for each axis on the *Diagnostics* tab. Homing defines the systems machine coordinates.

*Note: When Mach3 starts up **Reset** must be pressed before any motion can be performed.
(See Section 4.6.3)*

The machine needs to be homed before any cutting or significant jogging is done so that the controller knows where the axes are in order to avoid crashes.

The system will need to be homed every time one of the following happens: Mach3 is restarted, there is an external E-Stop, or a limit switch is triggered.

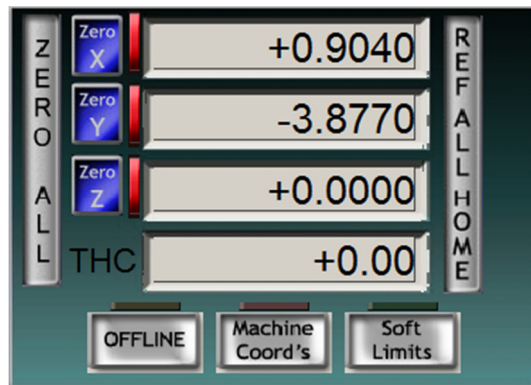


Figure 3 - Machine Position Before Being Homed

It is possible to know if the system needs to be homed by simply looking at the Mach3 screen. If the LED associated with an axis is red, then the axis is not homed. If it is green, then it is homed.



Figure 4 - Machine Position After Being Homed

2.2 Jogging

To access the jog panel in Mach3 press the **Tab** key on the keyboard to toggle the panel on and off (See Figure 5).

2.2.1 Jog On/Off

Checking the Jog On/Off box enables jogging. If it is unchecked then the jogging buttons on the screen and keyboard will not function.

2.2.2 Jog Buttons

The jog buttons allow the operator to move the axes manually. This can also be done using the **Up** and **Down** keys on the keyboard to control the Y-axis and the **Left** and **Right** keys to control the X-axis.

The torch can be moved manually using the **Z+** and **Z-** arrows on the screen or the **+** and **-** keys on the keyboard.

2.2.3 Jog Speed

The value in the DRO that is controlled by the up and down buttons under *Slow Jog Rate* is the percentage of the maximum jog speed. This is used to slow the jog speed. Acceptable values range from 1-100.

2.2.4 Continuous and Step Jogging

Continuous jogging moves the axis as long as the jog key is held down.

Step jogging moves the axis a fixed distance as defined by the DRO. The speed at which it moves is defined by the feed rate not the jog speed. See Figure 1 Zone 8.



Figure 5 - Jog Controls on Plasma

3 Torch Control

3.1 Tabs and Plasma and Oxy Fuel Mode Selection

The tab bar is used to switch between Plasma and Oxy Fuel and to go to the ToolPath screen and Diagnostics.



Figure 6 - Screen Tabs

3.2 Plasma Settings

The figure below shows the plasma cut sequence.

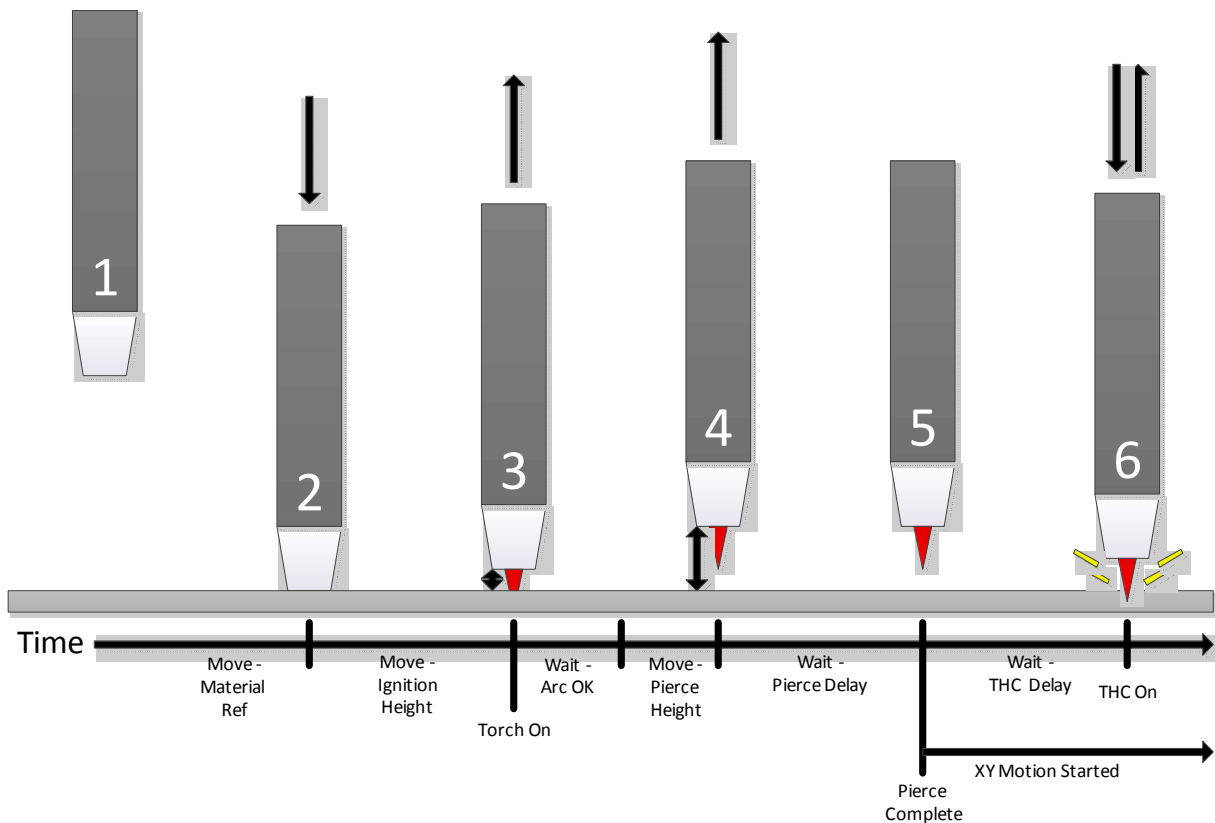


Figure 7 – Plasma Cut Sequence

Below is a description of each step.

- Step 1. The height that the Z-Axis was at when the cut sequence began
- Step 2. The torch head comes down to contact the material to determine the material height
- Step 3. The torch head lifts to ignition height and turns on (See Section 3.2.9)
- Step 4. Once the Arc Ok signal turns on the torch lifts to the pierce height (See Section 3.2.5)
- Step 5. Pierce delay is complete and XY motion begins (See Section 3.2.7)
- Step 6. THC delay is complete and the THC takes over Z-Axis control (See Section 3.2.8)

The following settings affect the cut sequence and will change the times and distances referred to in Figure 7.

SETTINGS	
Pierce Counter	17
Pierce Height	+0.100
Pierce Delay	+0.000
THC Delay	+0.000
Ignition Height	+0.000
Jog Speed	+100.000
Ref Feed	+100.000

Figure 8 - Plasma Operating Settings

3.2.1 THC

The Torch Height Control (THC) reads the tip voltage while cutting and automatically adjusts the cutting head up and down to maintain a constant height above the material. See Figure 9. If THC is off the cutting head will stay at the pierce height while running.

3.2.2 Torch

This button controls whether the torch is on or off.



Figure 9 - Torch and THC Controls

3.2.3 Rip Cut

The rip cut feature is used to do quick trimming operations on material. The cutting head will cut a defined distance from the start position in whichever direction is chosen. The federate can be defined by the operator. Rip cut goes through the same cut sequence as a normal cut and uses THC if enabled.

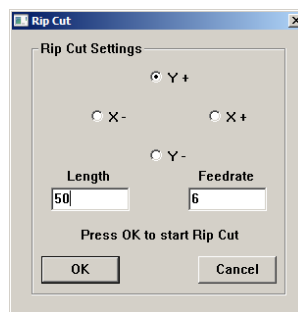


Figure 10 - Rip Cut Dialog

3.2.4 Pierce Counter

The pierce counter records the number of pierces the controller performs. It can be zeroed out by the operator at any time. It does not control anything it is simply a reference. See Figure 8.

3.2.5 Pierce Height

The *Pierce Height* defines, in inches or millimeters, the height above the material that the cutter head will sit while conducting a pierce action. See Figure 1 Zone 1.

A good value for pierce height can be found by looking in the consumables chart for the system's plasma cutter.

3.2.6 THC Height

The THC correction height displays the distance the Z-Axis has moved since the torch turned on to the current location. It is labeled *THC*. See Figure 4.

3.2.7 Pierce Delay

The **Pierce Delay** is the time, in seconds, that the cutter head will pause while piercing. This gives time for the hole to go all the way through the material. See Figure 8.

The **Pierce Delay** can be interrupted at any time by pressing the Cycle Start button to continue running the G-Code file.

A good value for **Pierce Delay** can be found by looking in the consumables chart for the system's plasma cutter.

3.2.8 THC Delay

The THC Delay allows the cutting head to get away from the molten metal surrounding the initial pierce hole. This is used primarily for thick material or small holes. See Figure 8.

3.2.9 Ignition Height

The Ignition Height is used on thick material where the pierce height is higher than the plasma head can easily establish an arc. On thinner material the Ignition Height can be set to zero and the system will skip the feature. See Figure 8.

3.2.10 Jog Speed

The **Jog Speed** field is used to adjust the *Jog Speed* without opening the jog panel. See Figure 5 and Figure 8.

3.2.11 Ref Feed

The Ref Feed defines the cutting head's feed rate in inches per minute while the torch is referencing. See Figure 8.

3.3 Oxy Fuel Settings

The figure below shows the Oxy Fuel cut sequence.

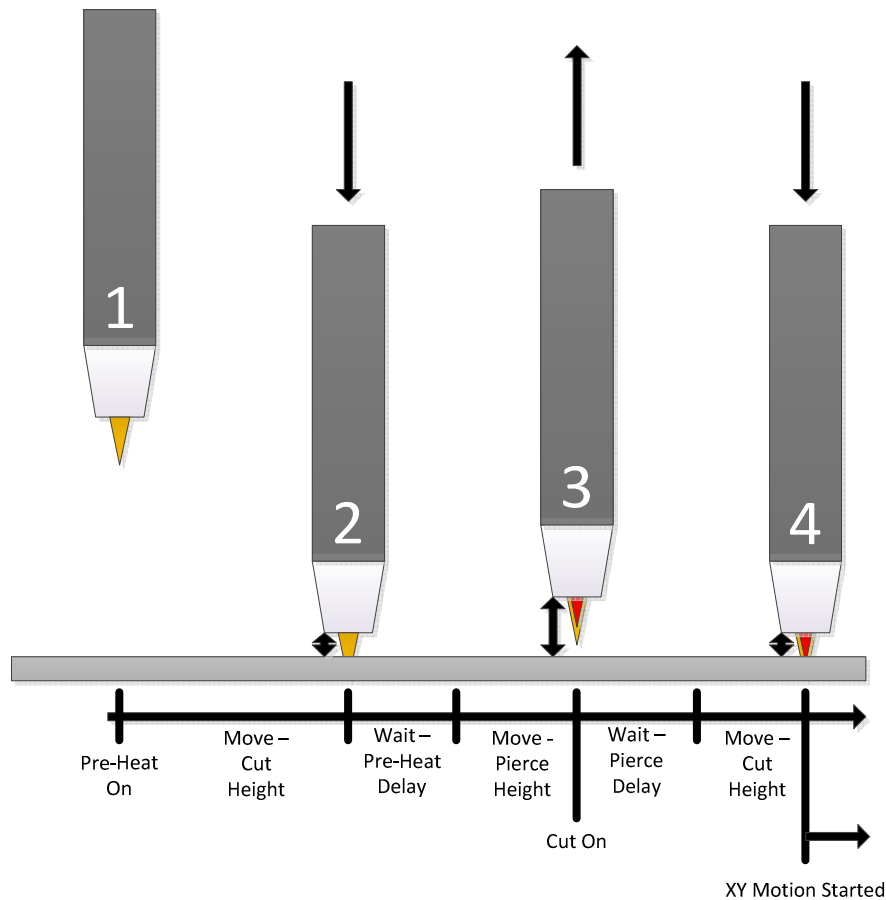


Figure 11 – Oxy Fuel Cut Sequence

Below is a description of each step.

- Step 1. The height that the Z-Axis was at when the cut sequence began
- Step 2. The Z-Axis lowers to the cut height and the pre-heat delay begins (See Section 3.3.3)
- Step 3. Once the cutting head has arrived at the pierce height, **Cut** (Oxygen) turns on and begins piercing the material
- Step 4. When the pierce delay is completed the cutting head lowers to the cut height and begins XY motion

The following settings affect the cut sequence and will change the times and distances referred to in Figure 11.

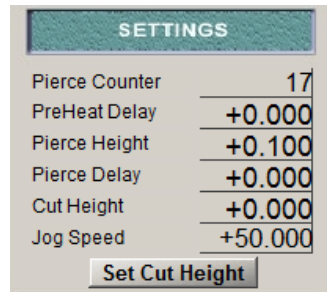


Figure 12 - Oxy Fuel Operating Settings

Note: When in Oxy Fuel mode the **Torch** button text will change from **Torch** to **Cut**.

3.3.1 Oxy Fuel PreHeat

Preheat must be manually turned on and the torch must be lit before any program is run in order to prep the material for piercing. The control will automatically turn it off at the end of the file.

Turning on preheat turns on the mixed gas to the cutting head.

3.3.2 Oxy Fuel Cut

Cut turns on the oxygen flow to the cutting head when the program is ready to pierce or cut.

3.3.1 Rip Cut

The rip cut feature is used to do quick trimming operations on material. The cutting head will cut a defined distance from the start position in whichever direction is chosen. The federate can be defined by the operator. Rip cut goes through the same cut sequence as a normal cut and uses THC if enabled. See Figure 10.



Figure 13 - Oxy Fuel Control

3.3.2 Pierce Counter

The pierce counter records the number of pierces the controller performs. It can be zeroed out by the operator at any time. It does not control anything it is simply a reference. See Figure 12.

3.3.3 PreHeat Delay

The **Preheat Delay** is the time, in seconds, that the cutter head will pause while heating up the material before piercing. The **Preheat Delay** can be interrupted at any time by pressing the Cycle Start button to continue running the G-Code file.

3.3.4 Pierce Delay

The **Pierce Delay** is the time, in seconds, that the cutter head will pause while piercing. This gives time for the hole to go all the way through the material. See Figure 8. The **Pierce Delay** can be interrupted at any time by pressing the Cycle Start button to continue running the G-Code file.

3.3.5 Cut Height

The **Cut Height** is the distance above the material that the cutting head should be kept at.

1. Determine the desired cut height by jogging the Z-Axis.
2. Enter that value into the Cut Height field. See Figure 12.
3. Press the set cut height button on the axis position bar. See Figure 1 Zone 1.
4. To maintain the desired cut height the operator must jog the Z-Axis up and down using the Torch arrows on the screen.

3.3.6 Jog Speed

The **Jog Speed** field is used to adjust the *Jog Speed* without opening the jog panel (See Figure 5).

4 Running Files

4.1 Zeroing (Setting Part Zero)

Pressing **ZERO ALL** zeros the system which defines the part zero and therefore the part coordinates. It will not move any of the axes. It will simply set the X and Y axis positions to zero. See Figure 4.

4.2 GOTO

Before running a file, you may want to use one of your GoTo buttons to move the machine to the correct location.

4.2.1 Load

The load location is an operator defined location for the cutting head to go to be out of the way for material loading and unloading. See Figure 25.

4.2.2 Part Zero

The Part Zero button allows the machine to go to any of the last five defined part zeros.

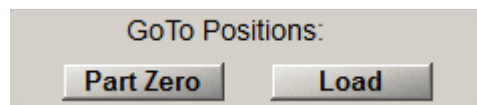


Figure 14 - GoTo Buttons

4.3 Loading G-Code Files

4.3.1 G-Code Window

G-Code files can be viewed in the window shown in Figure 15.



Figure 15 - G-Code Window

4.3.2 Edit G-Code

This buttons opens the current G-Code file in Notepad window to allow manual editing. See Figure 16.

4.3.3 Recent File

This buttons opens a selection box with the last 10 G-Code files used for quick access.

4.3.4 Close G-Code

This buttons unloads the currently loaded G-Code file from Mach3. See Figure 16.

4.3.5 Load G-Code

This opens a window to select a desired file from a directory.

4.3.6 Set Next Line

This buttons defines the current line as the line to begin running off of other than the first line in a G-Code file.

4.3.7 Line

Shows the current line and allows the operator to jump to a desired line.

4.3.8 Rewind

This buttons rewinds the loaded G-Code file to the beginning.

4.3.9 Single BLK

When Single BLK is activated the current G-Code file will run one line at a time requiring the cycle start button to be pressed at each line.

4.3.10 Block Delete

The delete block function temporarily removes all lines in the G-Code file that begin with “/” when it is activated. This can be used for trouble shooting, diagnostics, features etc.

4.3.11 M1 Optional Stop

If the M1 Optional Stop is activated the G-Code file will pause at every instance of M1 in the file.

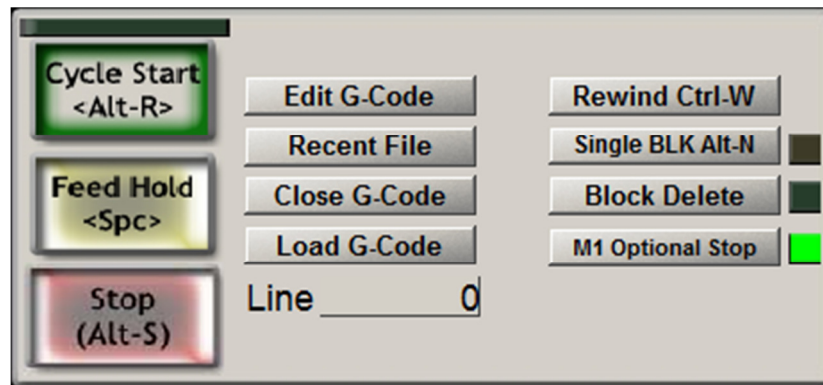


Figure 16 - G-Code File Controls

4.4 Plate Alignment

The purpose of plate alignment is to compensate for the material not being exactly straight in relation to the cutter axis. See Figure 16.

To use Plate Alignment:

1. Locate the cutter head along one edge of the material

2. Press the **Plate Alignment** button

Note: When it is pressed the first time a window will pop up asking if the operator wants to restore the last plate alignment. This allows the operator to restore the previous settings if a shutdown is required or if it got turned off accidentally.

3. After the dialog box is closed the left LED under the **Plate Alignment** button will turn on
4. Move the cutter head to a point further down the same edge of the material
5. Press the **Plate Alignment** button a second time

When **Plate Alignment** is active the both LEDs under the **Plate Alignment** button will be green and the X and Y axis position DROs will be red signifying that the axes have been rotated.

Note: The system must be zeroed before plate alignment is used

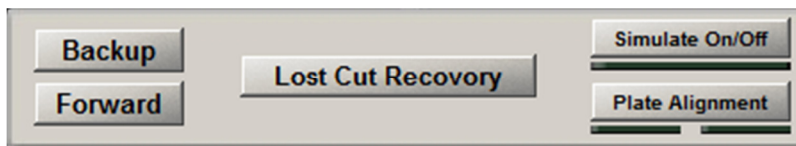


Figure 17 - Lost Cut Recovery, Simulate, and Plate Alignment Controls

4.5 Simulate

The simulate function allows the operator to run the machine through a program without activating any of the outputs or turning on the cutter. This allows the operator to trouble shoot or test the program and get an idea of what it will look like without cutting anything. To switch states simply press the button. The green LED below the button will be on if the simulate function is on. See Figure 17.

Note: This is also used in conjunction with Lost Cut Recovery. See Section 5.2.2.

4.6 Control Buttons

4.6.1 Cycle Start

This button will run the loaded G-Code file, from the beginning of the file or if the **Stop** button was pressed from the last line that was run. The start point can be adjusted using the **Up** and **Down** arrows on the G-Code window. See Figure 16.

4.6.2 Rewind

This buttons rewinds the loaded G-Code file to the beginning. See Figure 16.

4.6.3 Reset

Reset stops all motion, turns off all outputs, and stops any G-Code programs.

When *Mach3* starts up, **Reset** must be pressed before any motion can be performed.

4.6.4 Stop

The **Stop** button will stop any programs running but will keep the place in the G-Code file. It is used along with **Simulate** to start a cut in the middle of a G-Code line or arc segment. See Section 6.6.

This page was intentionally left blank.

5 Lost Cut Recovery

5.1 Lost Cut Recovery Overview

When running a G-Code file and something happens to interrupt the process **Lost Cut Recovery** can be used to pick up where the operator left off or it can be used to start a G-Code file part way through.

Here is a brief overview of the five lost cut recovery modes:

1. Plasma Auto Lost Cut Recovery: The machine will move to where the arc signal was lost and begin cutting.

When to Use:

- The plasma torch was on, but the arc signal was lost.
- To re-start when the arc signal was lost.

2. Plasma Manual Lost Cut Recovery: The machine will begin cutting at the current location and then continue executing from the selected line of G-Code.

When to Use:

- The torch never started (there never was an arc ok signal).
- The system cannot start at the position that lost the arc signal.
- When starting between two lines of G-Code.

3. Oxy Fuel Lost Cut Recovery: For lost cut recovery using Oxy Fuel

When to Use:

- When starting to cut from the middle of a G-Code file

4. Drill: For starting a drill cycle in the middle of a G-Code file

When to Use:

- When starting to drill from the middle of a G-Code file.

5. Plate Marker

When to Use:

- When starting to mark a plate from the middle of a G-Code file.

5.2 Lost Cut Recovery Procedure

5.2.1 Plasma – Auto

The procedure for conducting a Plasma auto lost cut recovery is as follows:

1. To start Lost Cut Recovery press the **Lost Cut Recovery** button (See Figure 17).
2. Press the **Plasma** button under *Auto Lost Cut Recovery* causing the following dialog to appear

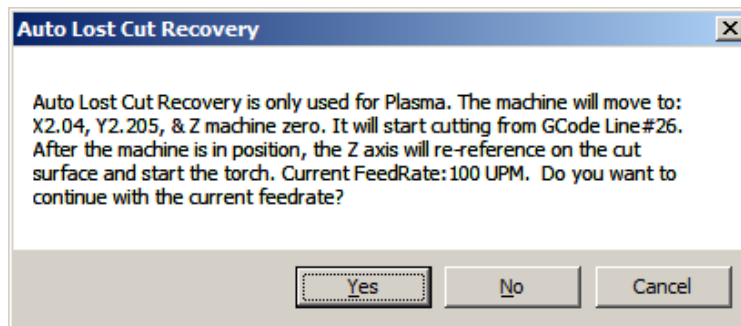


Figure 18 - Auto Plasma Lost Cut Recovery Dialog

3. Press **Yes** to continue with the current feed rate or **No** to set a new feed rate.

The machine will move to where the arc was lost, re-reference the Z axis, and then begin cutting. See Figure 7.

WARNING: The machine will move to the last location where an arc signal was lost. Make sure that the torch had been on. If not done properly, this mode could cause severe damage to the machine and/or material.

5.2.2 Plasma – Manual

The procedure for conducting a Plasma manual lost cut recovery is as follows:

1. Scroll through the G-Code to select the line to begin cutting on.
2. Move the machine to the EXACT X and Y position where you want the machine to begin cutting using the **Forward** and **Backup** buttons. The **Forward** and **Backup** buttons will move the machine to the final commanded position of a G-Code line. See Figure 17.

Note: If you want to start in the middle of an arc or line segment you must turn Simulate on and press Cycle Start. The machine will begin moving along the profile. Press Stop when the cutter reaches the point that you want to begin cutting from.

3. Press the **Lost Cut Recovery** button. See Figure 17.

4. Press the **Plasma** button under *Manual Lost Cut Recovery* (See Figure 23) causing the following dialog to appear

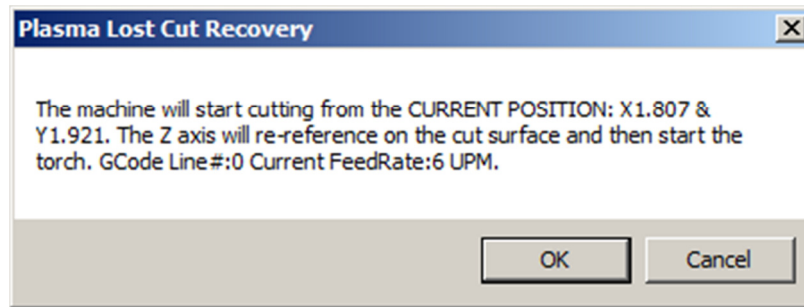


Figure 19 – Manual Plasma Lost Cut Recovery Dialog

Your machine will re-reference the Z axis, and then begin cutting from the current position. See Figure 7.

WARNING: Your machine will pierce and then start cutting at the current position.

5.2.3 Oxy Fuel

The procedure for conducting an Oxy Fuel lost cut recovery is as follows:

1. Scroll through the G-Code until you locate the line where you want your program to begin cutting.
2. Move the machine to the EXACT X and Y position where you want the machine to begin cutting using the **Forward** and **Backup** buttons. The **Forward** and **Backup** buttons will move the machine to the final commanded position of a G-Code line. See Figure 17.

Note: If you want to start in the middle of an arc or line segment you must turn Simulate on and press Cycle Start. The machine will begin moving along the profile. Press Stop when the cutter reaches the point that you want to begin cutting from.

3. Press the **Lost Cut Recovery** button. See Figure 17.
4. Press the **Oxy Fuel** button under *Manual Lost Cut Recovery* (See Figure 23) causing the following dialog to appear

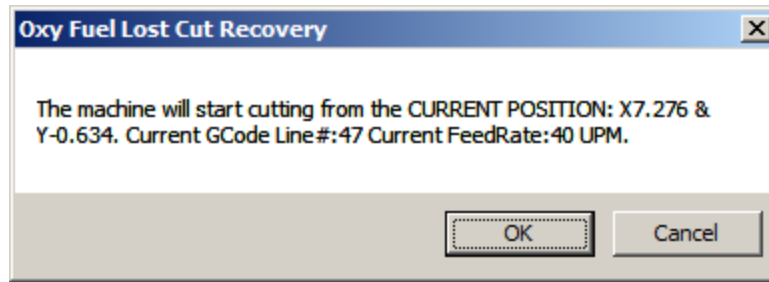


Figure 20 - Oxy Fuel Lost Cut Recovery Dialog

Your machine will begin cutting from the current position. See Figure 11.

5.2.4 Drill

The procedure for conducting a Drill cycle is as follows starting from the middle of G-Code file:

1. Scroll through the G-Code until you locate the line where you want your program to begin drilling.
2. Move the machine to the EXACT X and Y position where you want the machine to begin cutting using the **Forward** and **Backup** buttons. The **Forward** and **Backup** buttons will move the machine to the final commanded position of a G-Code line. See Figure 17.
3. Press the **Lost Cut Recovery** button. See Figure 17.
4. Press the **Drill** button under *Manual Lost Cut Recovery* (See Figure 23) causing the following dialog to appear

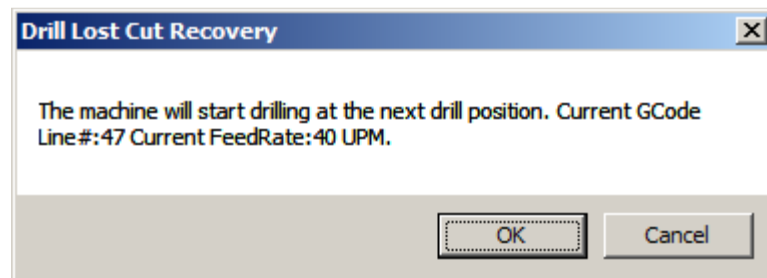


Figure 21 - Drill Cycle Dialog

Note: Your machine will begin drilling from the current position.

5.2.5 Plate Marker

The procedure for conducting a Plate Marker cycle is as follows:

1. Scroll through the G-Code until you locate the line where you want your program to begin marking the plate.
2. Move the machine to the EXACT X and Y position where you want the machine to begin marking using the **Forward** and **Backup** buttons. The **Forward** and **Backup** buttons will move the machine to the final commanded position of a G-Code line. See Figure 17.

Note: If you want to start in the middle of an arc or line segment you must turn Simulate on and press Cycle Start. The machine will begin moving along the profile. Press Stop when the plate marker reaches the point that you want to begin marking from.

3. Press the **Lost Cut Recovery** button. See Figure 17.
4. Press the **Plate Marker** button under *Manual Lost Cut Recovery* (See Figure 23) causing the following dialog to appear

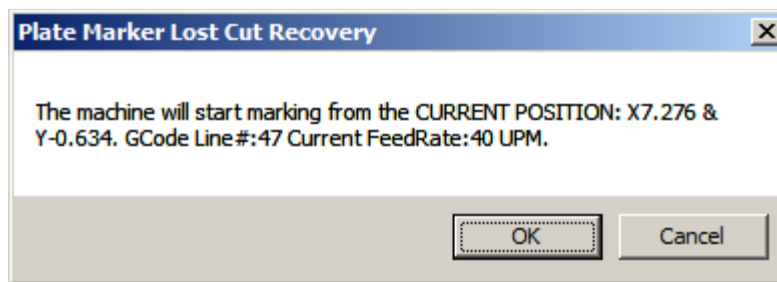


Figure 22 - Plate Marker Dialog

Note: Your machine will move the plate marker down to the material and begin marking from the current position.

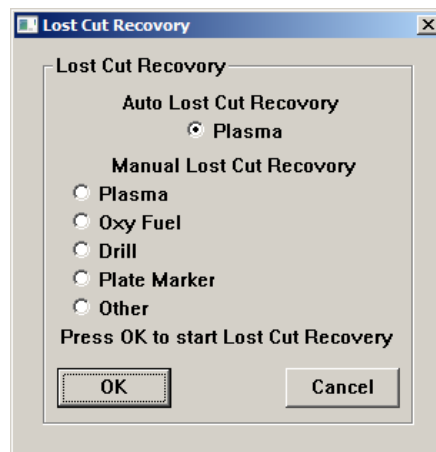


Figure 23 - Lost Cut Recovery Dialog

This page was intentionally left blank.

6 Appendix A – Button Definitions

6.1 Feed Rate

The window shows the current commanded feed rate. The feed rate can be overridden using the **UP** and **Down** buttons to scale the feed rate from 0-300% of the commanded value. The **Reset** button returns the override percentage back to 100%.

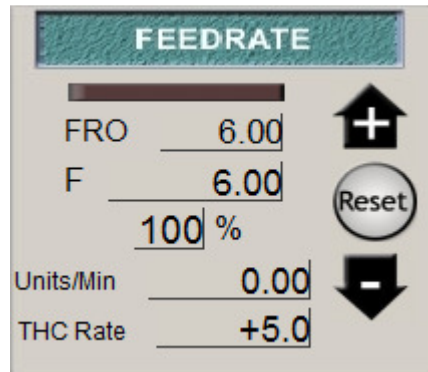


Figure 24 – Feed Rate Window

6.2 Load Location

The load location is an operator defined position for the cutting head to go to be out of the way for material loading and unloading.

It will not interfere with homing or the part zero.

6.3 Max and Min THC

The min and max THC values are the allowable Z-Axis deltas from the position the torch is in when torch turns on.

6.4 Pierce Reference Distance

Due to temperature fluctuations the material on a system often warps and bends. To compensate for this and avoid crashes the controller needs to reference the material height. However, it is not necessary to reference the material every time a new cut is made. Doing so can significantly slow down production times. See Figure 25.

The pierce reference distance allows the operator to define a distance within which the controller will not re-reference the material. So after it references for a cut any following cut within the pierce reference distance from the first cut will not require a second reference.

6.5 Plate Height Offset

The Plate Height Offset is the travel distance between the floating head's position at rest to the reference switch. The distance as entered into Mach3 is always negative. See Figure 25.

6.6 Z Rapid Height

Between cuts the cutting head will go up to the defined Z rapid height during while moving from one cut position to the next. Using a low rapid height in combination with a long pierce reference distance can significantly speed up part cut times. This is especially the case for parts with lots of holes and small cuts. See Figure 25.

7 Appendix B - Diagnostics

7.1 Tool Change Location

The tool change location is an operator defined location chosen to conveniently change tools.

7.2 Lost Arc Location

When the Arc OK signal is lost while cutting the location and G-Code line the cutter was on when it was lost is recorded here.

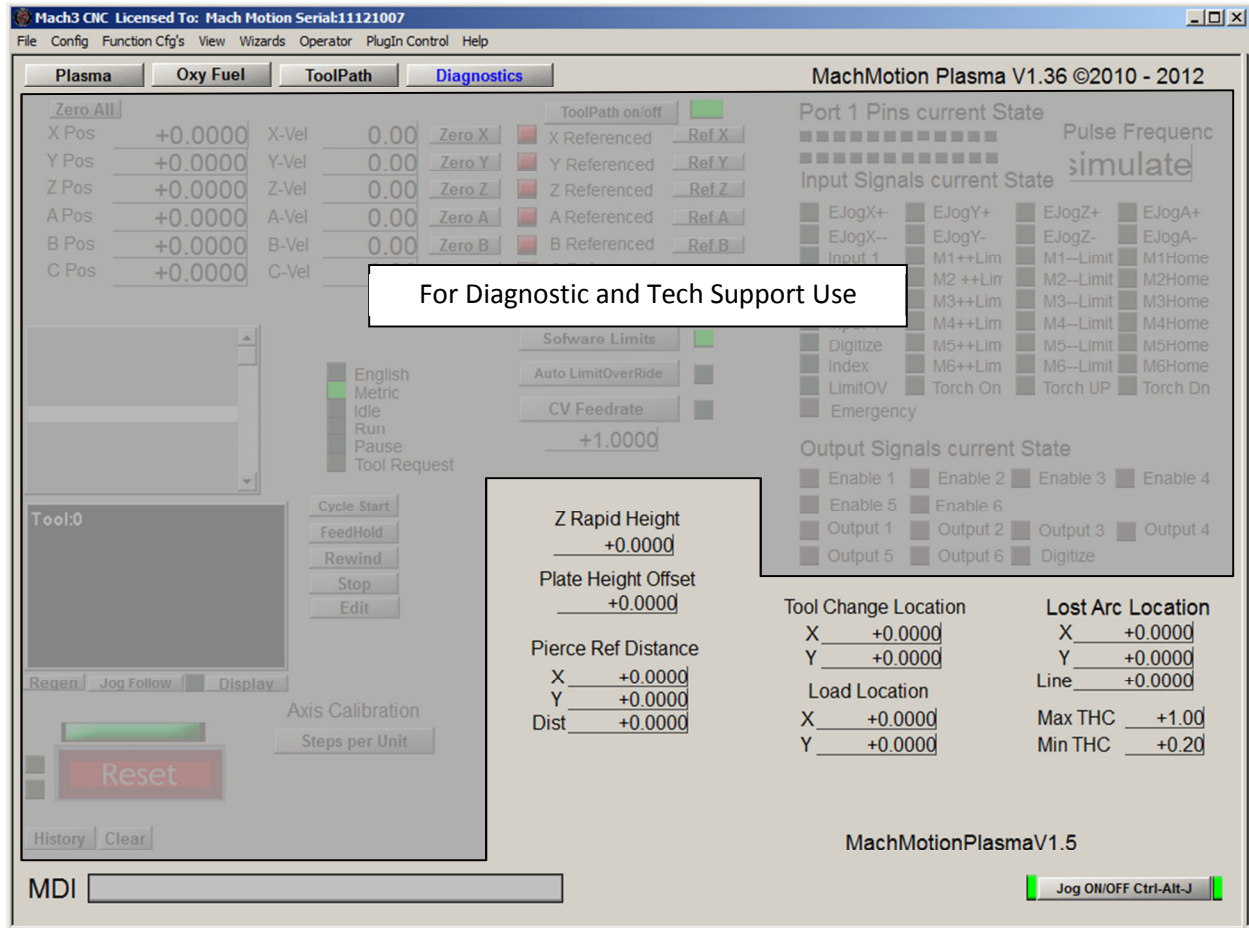


Figure 25 - Diagnostics Screen

This page was intentionally left blank

8 Appendix C – MDI and Program Limits

8.1 Program Limits

The program limits show the axis limits of a G-Code file when one has been loaded.

8.2 MDI

The MDI line allows the operator to input G and M-Code commands manually.

Figure 26 – MDI Tab