MACHMOTION

Plasma/Oxy Torch

Operations Manual

4/11/2012

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

MachMotion
www.machmotion.com
14518 County Road 7240, Newburg, MO 65550
(573) 368-7399 •Fax (573) 341-2672

Contents

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.

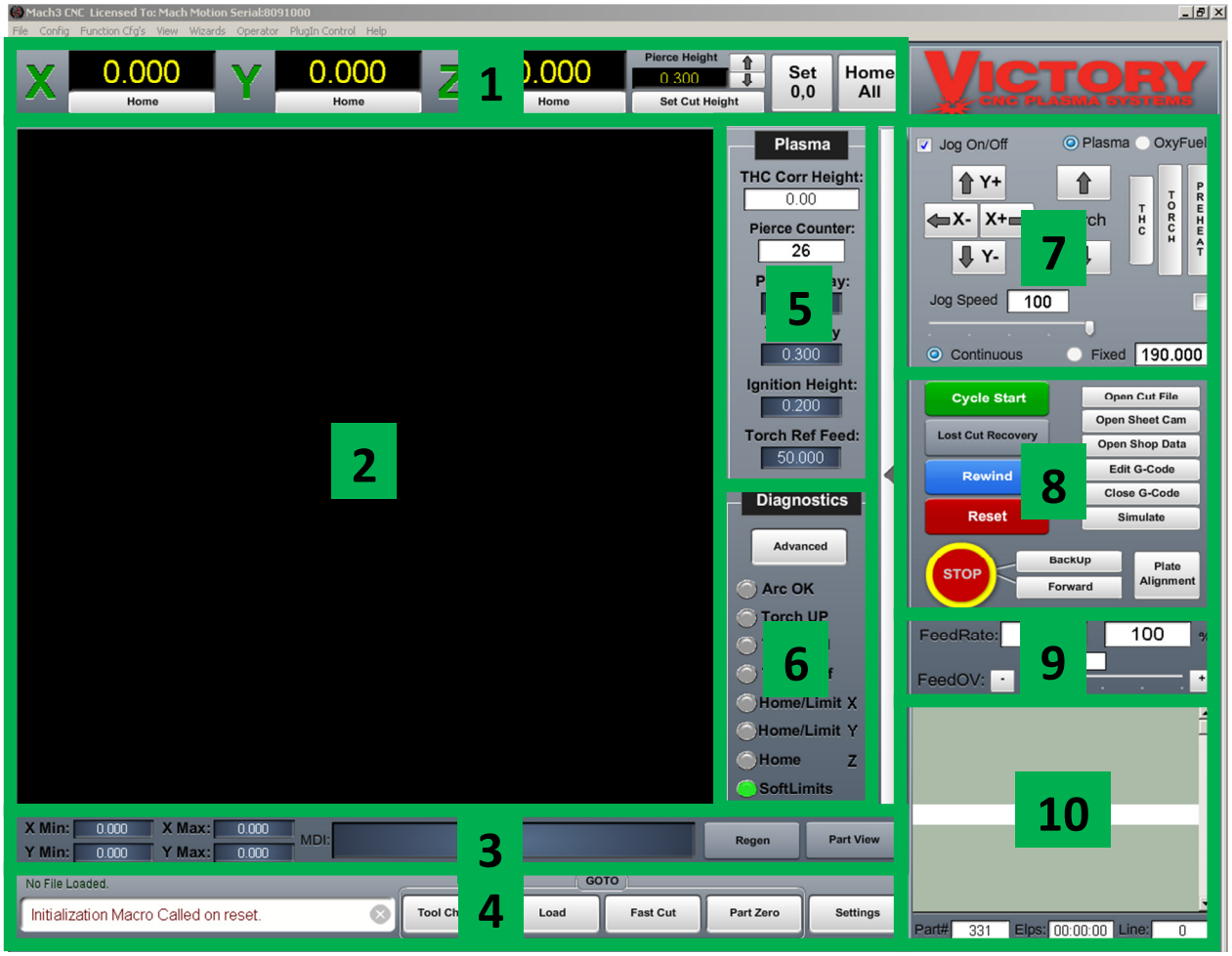

Figure 1 - Screen Break Down

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

Zone 1. Axis Position and Homing (See Section 2.1 - Homing)
Zone 2. G-Code File Tool Path Rendering
Zone 3. MDI and File Limits (See Section 6 - Appendix C – MDI and File Limits)
Zone 4. Status and GOTO (See Section 4.2 - GOTO)
Zone 5. Plasma (See Section 3.2 - Plasma Settings) or OxyFuel (See Section 3.3 - OxyFuel Settings) Mode Operating Settings

Zone 6. Diagnostics Screen (See Section 7 - Appendix B – Diagnostics)
Zone 7. Jogging (See Section 2.2 - Jogging) and Torch Control (See Section 3 -Torch Control)
Zone 8. G-Code Operations (See Section 4.3 - Loading G-Code Files)
Zone 9. Feedrate Control (See Section 6.1 - Feed Rate)
Zone 10. G-Code Window (See Section 4.3.1 - G-Code Window)

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 **Home All** 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 **Home** buttons under each axis position DRO. 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 an axis label (X, Y or Z) is red, then the axis in not homed. If it is green, then it is homed.

Figure 4 - Machine Position After Being Homed

2.2 Jogging

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 axis 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 up and down 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 slider 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 Fixed Jogging

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

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

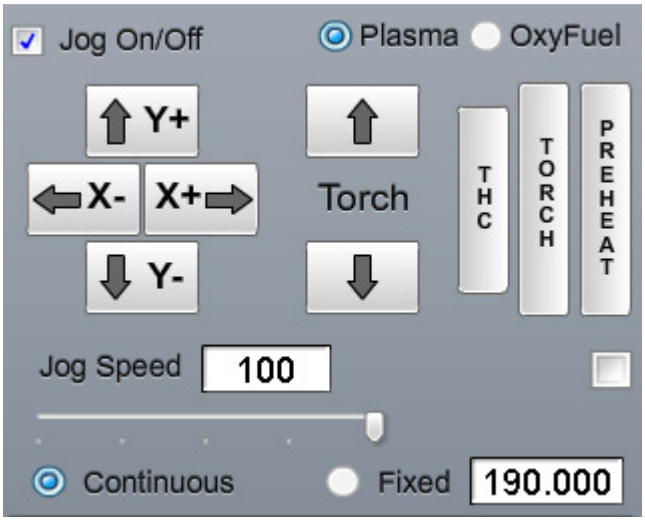

Figure 5 - Jog Controls on Plasma

3 Torch Control

3.1 Plasma and OxyFuel Mode Selection

This option is used to switch the torch controls from Plasma to Oxy Fuel mode. It also changes the operating settings window between the two modes. See Figure 5.

3.2 Plasma Settings

The figure below shows the plasma cut sequence.

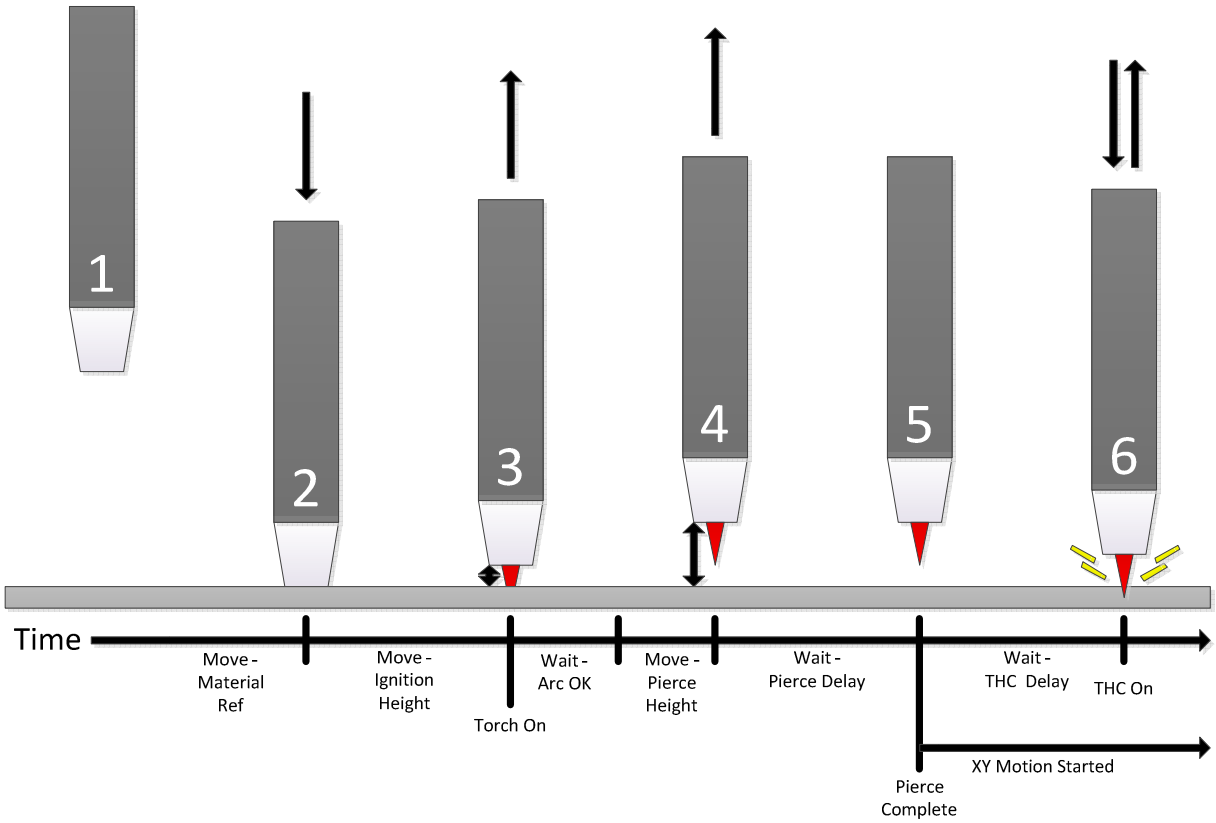

Figure 6 – Plasma Cut Sequence

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 3.2.8)
Step 4. Once the Arc Ok signal turns on the torch lifts to the pierce height (See Section 3.2.3)
Step 5. Pierce delay is complete and XY motion begins (See Section 3.2.6)
Step 6. THC delay is complete and the THC takes over Z-Axis control (See Section 3.2.7)

The following settings affect the cut sequence and will change the times and distances referred to in Figure 6.

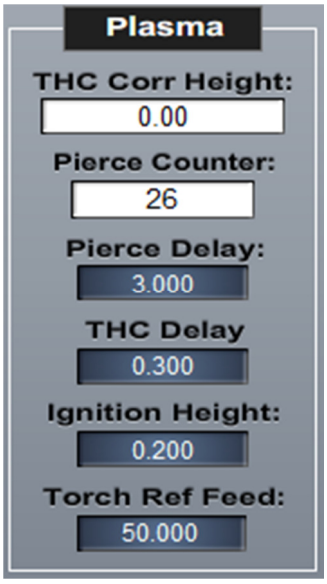

Figure 7 - Plasma Operating Settings

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 5. 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. See Figure 5.

3.2.3 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.

3.2.4 THC Corr Height

The *THC Correction Height* displays the distance the Z-Axis has moved since the torch turned on to the current location. See Figure 7.

3.2.5 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 7.

3.2.6 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 7.

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.

3.2.7 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 7.

3.2.8 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 7.

3.2.9 Torch Ref Feed

The Ref Feed defines the cutting head's feed rate in inches per minute while the torch is referencing. See Figure 7.

3.3 OxyFuel Settings

The figure below shows the oxyfuel cut sequence.

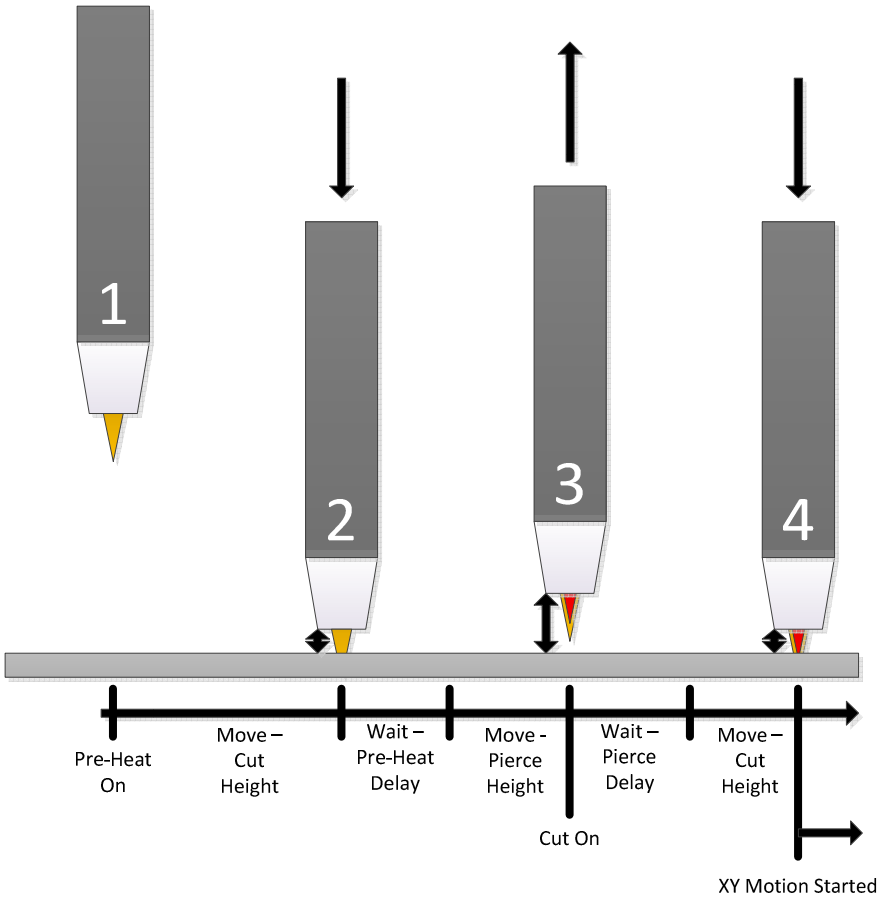

Figure 8 - OxyFuel Cut Sequence

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 3.3.4)

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 8.

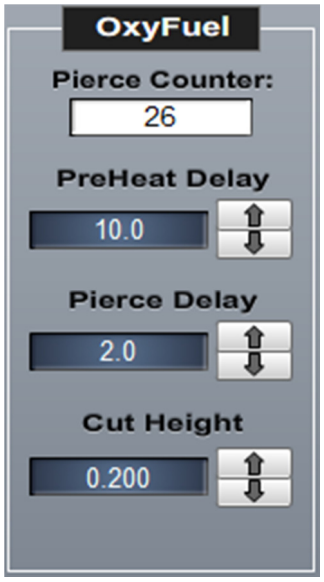

Figure 9 - Oxy Fuel Operating Settings

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

3.3.1 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 Cut

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

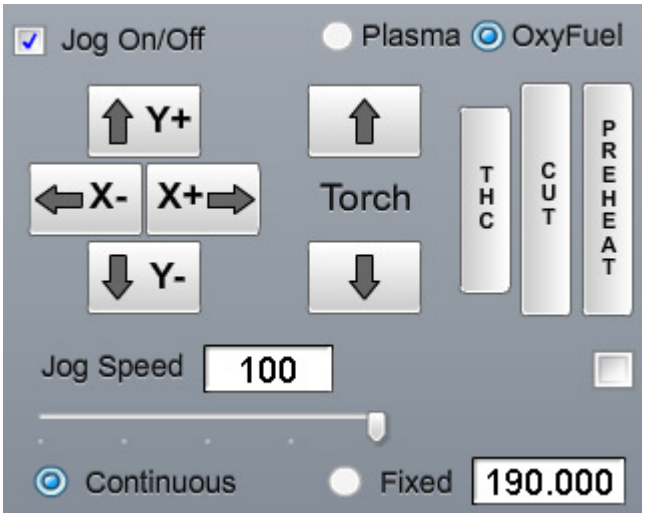

Figure 10 - Jog Controls on OxyFuel

3.3.3 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 10

3.3.4 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.

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

3.3.6 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 9.
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.

4 Running Files

4.1 Zeroing (Setting Part Zero)

Pressing the **Set 0,0** 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 Tool Change

The tool change position is an operator defined location for changing the consumables.

4.2.2 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.

4.2.3 Fast Cut

Fast Cut is an operator defined location used to quickly move the cutting head to the corner of the material.

4.2.4 Part Zero

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

Figure 11 - GOTO Buttons

4.3 Loading G-Code Files

The **Open Cut File** button opens a file menu window for finding a desired G-Code file. See Figure 14.

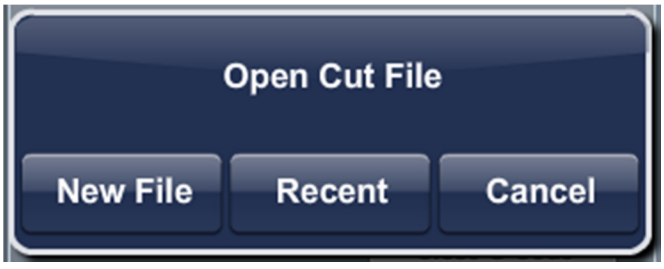

Figure 12 - Open Cut File Dialog

4.3.1 G-Code Window

G-Code files can be viewed in the window shown in Figure 13.

1. Part# – The **Part#** counter records the number of files the controller runs. It can be zeroed out by the operator at any time. It does not control anything it is simply a reference.
2. Elps – The Elps or elapsed time records how long the current file has been running.
3. Line – Shows the number of the current G-Code line highlighted.

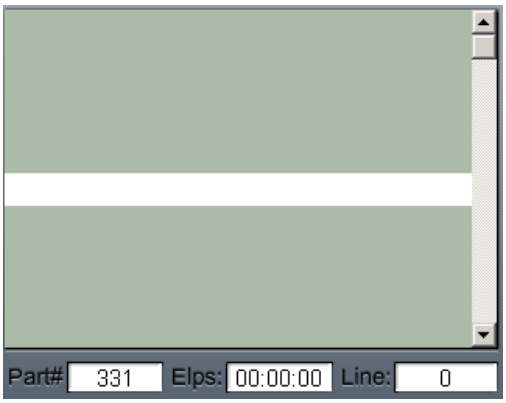

Figure 13 - G-Code Window

4.3.2 Edit G-Code

This buttons opens the current G-Code file in Notepad window to allow manual editing. See Figure 14.

4.3.3 Close G-Code

This buttons unloads the currently loaded G-Code file from Mach3. See Figure 14.

4.3.4 Open Sheet Cam

This is a shortcut to open Sheet Cam.

4.3.5 Open Shop Data

This is a shortcut to open Shop Data.

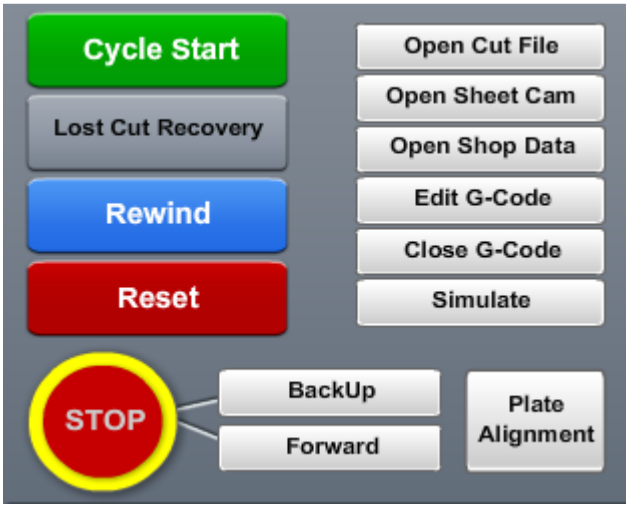

Figure 14 - G-Code File Controls

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 14.

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 accidently.
3. After the dialog box is closed the **Plate Alignment** button will turn yellow
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 PA is active the **Plate Alignment** button will be green and there will be a small R in the top left corner of the X and Y axis position DRO windows signifying that the axis has been rotated.

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

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 14.

Note: This is also used in conjunction with Lost Cut Recover. 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 **Backup** and **Forward** buttons. See Figure 14.

4.6.2 Rewind

This buttons rewinds the loaded G-Code file to the beginning. See Figure 14.

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 0)

This page was intentionally left blank.

5 Lost Cut Recovery

5.1 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 OK* signal was lost.
   - To re-start when the *Arc OK* 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 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 14).

2. Press the **Plasma** button under *Auto Lost Cut Recovery* causing the following dialog to appear

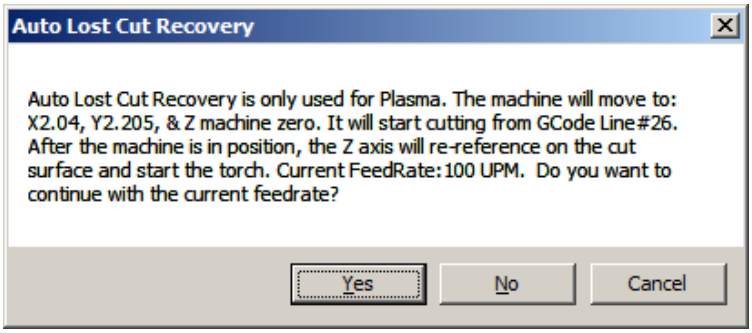

Figure 15 - Auto Plasma Lost Cut Recovery Dialog

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 6.

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 the machine should 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 14.

   *Note: To start in the middle of an arc or line segment **Simulate** must be used. Turn **Simulate** on and press Cycle Start. The machine will begin moving along the profile. Press Stop when the cutter reaches the desired starting point.*

3. Press the **Lost Cut Recovery** button. See Figure 14.

4. Press the **Plasma** button under *Manual Lost Cut Recovery* (See Figure 20) causing the following dialog to appear

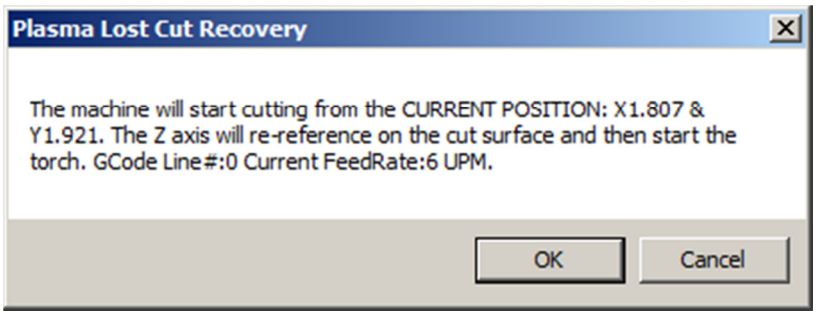

Figure 16 – Manual Plasma Lost Cut Recovery Dialog

The machine will re-reference the Z axis, and then begin cutting from the current position. See Figure 6.

WARNING: The 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 to select the line to begin cutting on.

2. Move the machine to the EXACT X and Y position where the machine should 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 14.

   *Note: To start in the middle of an arc or line segment **Simulate** must be used. Turn **Simulate** on and press Cycle Start. The machine will begin moving along the profile. Press Stop when the cutter reaches the desired starting point.*

3. Press the **Lost Cut Recovery** button. See Figure 14.

4. Press the **Oxy Fuel** button under *Manual Lost Cut Recovery* (See Figure 20) causing the following dialog to appear

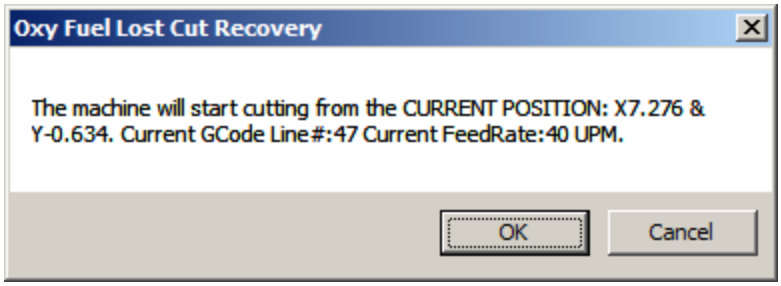

Figure 17 - Oxy Fuel Lost Cut Recovery Dialog

The machine will begin cutting from the current position. See Figure 8.

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 to select the line to begin cutting on.

2. Move the machine to the EXACT X and Y position where the machine should 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 14.

3. Press the **Lost Cut Recovery** button. See Figure 14.

4. Press the **Drill** button under *Manual Lost Cut Recovery* (See Figure 20) causing the following dialog to appear

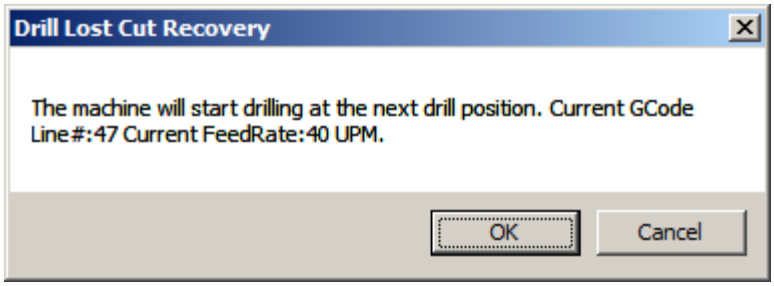

Figure 18 - Drill Cycle Dialog

Note: The 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 to select the line to begin cutting on.

2. Move the machine to the EXACT X and Y position where the machine should 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 14.

   *Note: To start in the middle of an arc or line segment **Simulate** must be used. Turn **Simulate** on and press Cycle Start. The machine will begin moving along the profile. Press Stop when the cutter reaches the desired starting point.*

1. Press the **Lost Cut Recovery** button. See Figure 14.

2. Press the **Plate Marker** button under *Manual Lost Cut Recovery* (See Figure 20) causing the following dialog to appear

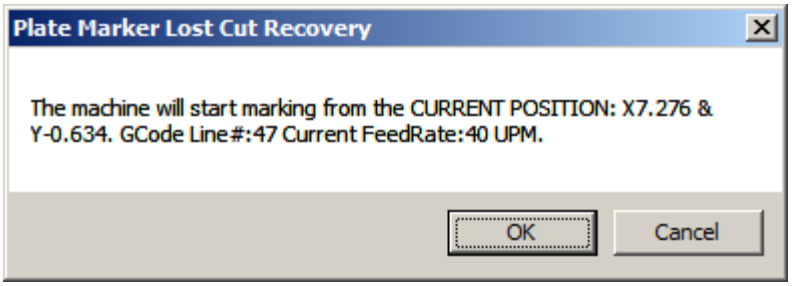

Figure 19 - Plate Marker Dialog

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

Figure 20 - 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 **FeedOV** slider to scale the feed rate from 0-300% of the commanded value.

Figure 21 – 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.

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.

6.6 Set G-Code Position

The **Set G-Code Position** button takes the XY coordinates from the currently loaded G-Code line and resets the part location to that position.

This is allows the operator to find the part zero from any point on the part.

The **Set G-Code Position** Procedure:

1. Jog the machine to the desired starting position on the part

2. Use the up and down buttons on the scroll bar in the G-Code window to select the desired starting line in the G-Code file
3. Open the Diagnostics Tab and press **Advanced** -> **Set G-Code Position** Button.

The part will now be zeroed and the file can be run from the set location.

6.7 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.

7 Appendix B – Diagnostics

7.1 Advanced Button

1. Drill Cycle – Used to test the drill cycle
2. Coolant – Used to turn the coolant On/Off on select machines
3. Set G-Code Position - Sets the current part location to be equal to the current XY G-Code file position (See Section 6.6)

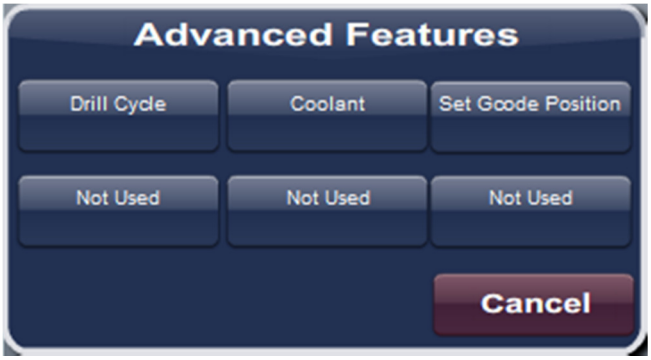

Figure 22 - Diagnostics Advanced Features

7.2 Status LEDs

The following are the status signals shown in the diagnostics screen. The LEDs show the real time state of each of the signals.

1. Arc OK
2. Torch UP
3. Torch DN
4. Torch Ref
5. Home/Limit X
6. Home/Limit Y
7. Home Z
8. SoftLimits

Figure 23 - Diagnostics Screen

8 Appendix C – MDI and File Limits

8.1 File Limits

The file 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 24 – MDI and File Limits