

FLASHCUT CNC

CONTROL MADE SIMPLE

Quick Start Guide

The screenshot displays the Flashcut CNC software interface. The 'Setup' tab is active, and the 'Tooling' sub-tab is selected. The 'Setup File:' field is empty, and buttons for 'New', 'Load', 'Save', and 'Save As...' are visible. The left sidebar shows a tree view with categories: System, Preferences, Machine, Sensing, and Programming. The 'Machine' category is expanded, and 'Axes' is highlighted. The main area shows two configuration panels:

Motor Line/Axis Mapping

Line #	Axis	Sub Axis	Offset	Direction
1	Y	1	0 in	Positive Negative
2	X	N/A	0 in	Positive Negative
3	Z	N/A	0 in	Positive Negative
4	Y	2	0 in	Positive Negative
5	None	N/A	0 in	Positive Negative

Axis Settings

Axis	Use Axis	Limited travel	Length	Backlash	Direction Change Delay (sec)
X	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	48 in	0 in	0
Y	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	48 in	0 in	0
Z	<input checked="" type="checkbox"/>	<input type="checkbox"/>	6.5 in	0 in	0
W	<input type="checkbox"/>	<input type="checkbox"/>	10 in	0 in	0

Midwest Office

444 Lake Cook Road, Suite 22

Deerfield, IL 60015

Phone (847) 940-9305 ♦ Fax (847) 940-9315

www.flashcutcnc.com

©1998-2025 WPI, Inc.

Getting Started.....	1
About This Manual.....	1
Turning Off the Controller.....	1
Legal.....	1
Safety and Usage Guidelines.....	2
Using the Quick Start Guide.....	3
Software Requirements.....	3
<i>Minimum System Specifications.....</i>	<i>3</i>
<i>Recommended System Specifications.....</i>	<i>3</i>
Using the Configuration Menu.....	4
System.....	5
General.....	5
Controller.....	5
Preferences.....	6
Language.....	6
Display.....	7
Jogging.....	8
Machine.....	9
Axes.....	9
Drive Parameters.....	10
<i>Control Parameters.....</i>	<i>10</i>
<i>Drive Mechanism.....</i>	<i>11</i>
Motion Calibration.....	13
Optimizing Feedrates and Ramping.....	14
Start/Stop Feedrate (in/min).....	14
Maximum Feedrate (in/min).....	14
General Ramp Rate (in/sec ²).....	15
Feedrate Move Ramp Rate (in/sec ²).....	16
Emergency Ramp Rate (in/sec ²).....	16
Continuous Contouring Feedrate Tol. (in/min).....	16
Inputs.....	17
Drive Faults.....	17
Plasma.....	18
General.....	18
Communication (Add-On).....	19
Torch Control.....	19
Cutting.....	20
<i>Start Of Cut.....</i>	<i>20</i>
<i>End Of Cut.....</i>	<i>20</i>
<i>Cut leadin when returning to toolpath.....</i>	<i>21</i>
<i>Standalone Piercing.....</i>	<i>21</i>
Marking.....	21
<i>Start of Mark.....</i>	<i>21</i>
<i>End of Mark.....</i>	<i>22</i>

Arc-Transfer Monitoring.....	22
Torch Height Control.....	23
General Sensing.....	23
Program Zero Sensing.....	25
Additional Controls.....	26
Touch Off.....	26
OxyFuel.....	27
General.....	28
Lifter.....	29
<i>Lift Control</i>	29
<i>Lift Control: None</i>	29
<i>Lift Control: Two Position</i>	29
<i>Lift Control: Axis</i>	31
<i>Lift Control: Momentary</i>	31
On/Off Control.....	33
Automatic Ignition.....	34
Preheat.....	36
<i>Basic Preheat</i>	36
<i>Bi-Level Preheat</i>	37
<i>Configure Low Preheat</i>	37
<i>Configure High Preheat</i>	38
Water.....	40
Cutting.....	41
<i>Start Of Cut</i>	41
<i>End Of Cut</i>	42
<i>Cut Leadin When Returning to Toolpath</i>	42
Standalone Piercing.....	43
Additional Controls.....	44
<i>Support Purge Delay</i>	44
Waterjet.....	45
General.....	46
Waterjet Control.....	48
Low Pressure Control.....	49
Abrasive Control.....	50
Cutting.....	51
<i>Start Of Cut</i>	51
<i>End Of Cut</i>	52
<i>Cut leadin when returning to toolpath</i>	52
Standalone Piercing.....	53
General Sensing.....	54
Program Zero Sensing.....	56
Touch Off.....	57
Additional Controls.....	57
Inputs Continued.....	58
Homing and Limits.....	58
Input Configuration.....	59
<i>Set Up Homing</i>	59

Safety Features.....	62
<i>As an Input</i>	62
<i>As an Accessory</i>	62
Servo Communications.....	64
<i>ServoWare</i>	64
<i>Hardware</i>	64
<i>COM Port</i>	65
<i>Connecting to Drives</i>	65
Revision History.....	67

Getting Started

About This Manual

CNC is a unique application involving hardware and software. We recommend that you read all of these instructions before using the product.



Since automated machining is potentially dangerous, please take the time to completely read through this manual and the software User's Guide to understand the operation of the electronics, software and machine before cutting a part.

Turning Off the Controller



Always turn off the CNC Controller when it is not in use.

Legal

The products and company names listed herein are trademarks or registered trademarks of their respective owners, and the use of the marks does not and should not imply any affiliation with, endorsement by, or authorization from, those companies.

Safety and Usage Guidelines



When running an automated machine tool, safety is of the utmost importance. For proper and safe use of the CNC program and your CNC machine, the following safety guidelines must be followed:

- 1. Never let the machine tool run unattended.**
- 2. Require any person in the same room as a running machine tool to wear safety goggles and to stay a safe distance from the machine.**
- 3. Allow only trained operators to run the machine tool. Any operator must have:**
 - Knowledge of machine tool operation.**
 - Knowledge of personal computer operation.**
 - Knowledge of Microsoft Windows.**
 - Good common sense.**
- 4. Place safety guards around the machine to prevent injury from flying objects. It is highly recommended that you build a safety shield around the entire tool envelope.**
- 5. Never place any part of your body within the tool envelope while the machine is online, since unexpected machine movement can occur at any time.**
- 6. Always keep the tool envelope tidy and free of any loose objects.**
- 7. Be on alert for computer crashes at all times.**

WPI, Inc. is not responsible for the safe installation and use of this product. You and only you are responsible for the safety of yourself and others during the operation of your CNC machine tool. We supply this product but have no control over how it is installed or used. Always be careful!

WPI, Inc. or its affiliates are not responsible for damage to any equipment or workpiece resulting from use of this product.

If you do not understand and agree with all of the above safety guidelines, do not use this product.

Using the Quick Start Guide

The **Quick Start Guide** is intended for the initial setup of a CNC machine. It is important that the software is configured properly as incorrect parameters can cause the machine to behave in a potentially dangerous manner. Due to the large variety in CNC machines, not all sections may apply.

Please refer to the **FlashCut CAD/CAM and CNC 10 User's Guide** for additional information regarding the software's functions and features not covered in this guide.

Software Requirements

Before installing the software, please ensure the computer either meets or exceeds the following:

Minimum System Specifications

- **OS:** 64-bit Windows, Windows 10 or newer
- **CPU:** 2-core Intel i3 or equivalent, clock speed at or greater than 2.1 GHz
- **Memory:** 8GB RAM
- **Storage:** Solid-State Drive
- **Communications:**
 - 1x USB
 - +1x USB (Servo only)
 - +1x USB (Plasma communications only)
 - +1x USB (Control pendant only)

Recommended System Specifications

- **OS:** 64-bit Windows, Windows 10 or newer
- **CPU:** 4-core Intel i3 or equivalent, clock speed at or greater than 3.2 GHz
- **Memory:** 16GB RAM
- **Storage:** Solid-State Drive
- **Communications:**
 - 1x USB
 - +1x USB (Servo only)
 - +1x USB (Plasma communications only)
 - +1x USB (Control pendant only)

Using the Configuration Menu

The **Configuration Menu** can be accessed by clicking on the “double-gear” icon in the upper-right corner of the software.



The setup file tools are located at the upper right of the configuration window and allow for the following functions:



New: This loads a pre-packaged setup file. Select the file then click **OK**. A copy of that file will be loaded.

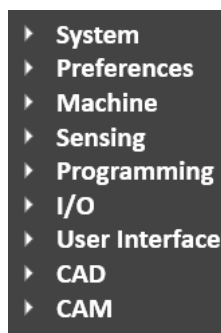
Load: This loads a different setup file. Setup files have the extension ***.SETUP**. Legacy files from Version 4.5, which use the extension ***.STP**, can also be used. The software will create a backup of the original file before updating it to the current version.

Save: This button will save the current configuration state and close the Configuration Menu. This will overwrite changes to an existing setup file.

Save As...: This button will save a copy of the setup file allowing for a different name and/or destination for the file.

Note: It is recommended to save the setup with a unique name. An example is to use the organization name and date the setup it was created.

The **configuration page browser** is found on the left-hand side of the menu, and will be used to navigate between sections and subsections.



Note: Because the setup file contains all of the configured parameters for motion and control, it is recommended to create and store backups of this file. In the event of a computer crash or failure, a finalized setup file can be loaded as opposed to configuring the setup from scratch.

System

General

Using the configuration page browser, navigate to the **System>>General** section. Use the drop-down to choose from **English** or **Metric** for **System Units**. For the purposes of this guide, we will be using English units.

Controller

Using the configuration page browser, navigate to the **System>>Controller** section.

The first sub-section, **Controller**, will have parameters that default to the following:

- **Controller Type:** 501A (USB)
- **USB Speed:** Full-Speed
- **Jog Buffer:** 0.100 sec
- **General Buffer:** 0.300 sec

For most configurations, the default should be used.

Note: When loading a configuration from an older version of FlashCut, ensure the **USB Speed** is set to **Full-Speed**. The controller will then require a power cycle to update the USB Speed settings.

The **Connect at Startup** checkbox will prompt a connection to the controller upon launching the software when selected. A safety dialog will appear to establish a connection at startup, effectively the same as pressing the **Connect** button in the CNC screen.

In the subsection, **Motor Signals**, use the **Controller Model** drop down to select the appropriate model matching the one found on the controller's specification label.

Based on the **Controller Model** that is selected, default values for the following will be generated:

- **Drive Type**
- **Step Pulse Polarity**
- **Step Pulse Width**
- **Min Time Between Steps**
- **Direction-Step Setup**
- **Min Step-Direction Lag**
- **Enable Signal Polarity**

These fields are grayed out and cannot be changed.

When utilizing drives other than those provided by Flashcut, select **Other** in the **Controller Model** drop-down to make changes to the grayed out values in accordance with the documentation of the driver model.

Preferences

Language

Navigate to the **Preferences>>Language** section of the configuration. Use the drop-down to choose the language for FlashCut's user interface.

The following languages are available:

- English (United States)
- Spanish (Spain)
- Italian (Italy)
- German (Germany)
- French (France)
- Dutch (Netherlands)
- Portuguese (Portugal)
- Japanese (Japan)

Note: Although several items may become translated immediately, an application restart is necessary for the language change to take effect throughout the application.

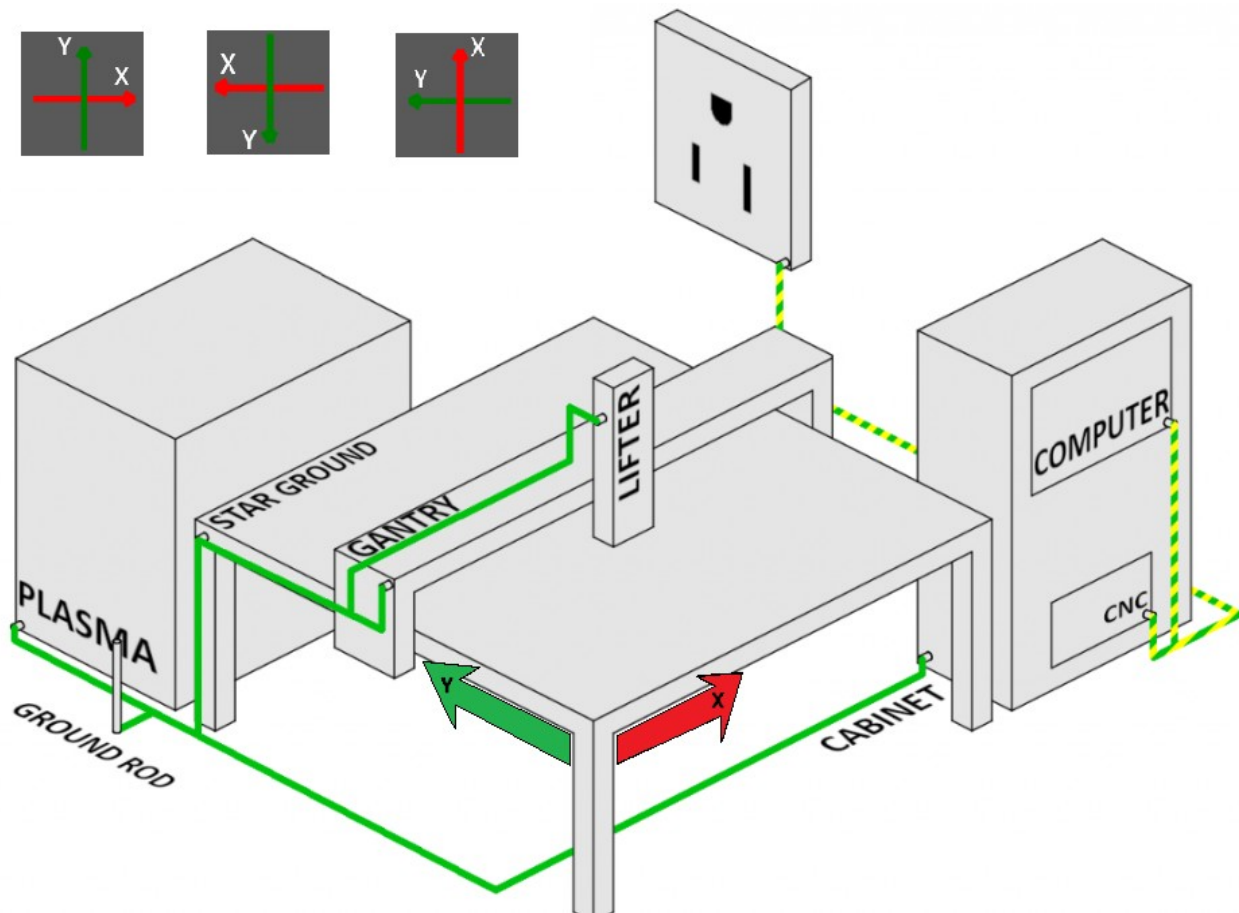
Display

Navigate to the **Preference>>Display** section.

Under the **Axis Display** sub-section, use the following check-boxes to modify the axis layout:

- **Display X axis horizontally in the viewport (and Y vertically)**
- **If X is horizontal, positive is to the right. If X is vertical, positive is up.**
- **Display all axes parallel to Z with positive up.**

The orientation of the axes is constrained by the right-hand rule for Cartesian coordinates. Ensure the axes layout matches the orientation of the computer in relation to the machine.



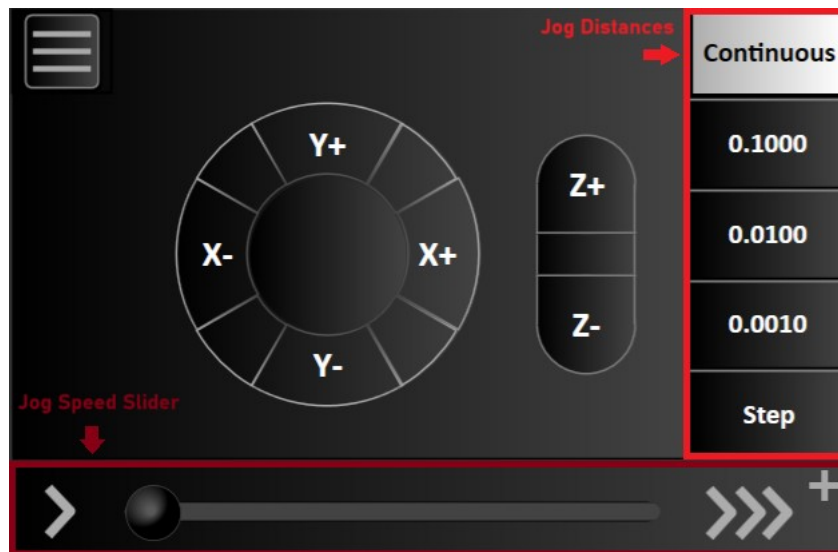
Jogging

Navigate to the **Preferences>>Jogging** section of the configuration. The minimum jog and point-move speeds can be set and adjusted here.

Under the **Jog and Point Move Rates** subsection, enter a conservative speed for each axis that is slow enough to position the machine precisely.

These speeds correspond to the left-most position on the speed slider of the jog panel in the **CNC screen**.

Under the **Jog Distances** subsection on the same page, set discrete jog distances that can be toggled between in the jog panel in the **CNC screen**.



Machine

Axes

Navigate to the **Machine>>Axes** section to assign the axis lines in use.

Motor Line/Axis Mapping						
Line #	Axis	Sub Axis	Offset	Direction		
1	Y	1	0 in	<input type="radio"/> Positive	<input checked="" type="radio"/> Negative	
2	X	N/A	0 in	<input type="radio"/> Positive	<input checked="" type="radio"/> Negative	
3	Z	N/A	0 in	<input checked="" type="radio"/> Positive	<input type="radio"/> Negative	
4	Y	2	0 in	<input type="radio"/> Positive	<input checked="" type="radio"/> Negative	
5	None	N/A	0 in	<input checked="" type="radio"/> Positive	<input type="radio"/> Negative	

Note: The example above outlines a dual-driven gantry. For this reason, Lines 1 and 4 are defined as the same **Axis** letter but a different value is assigned in the **Sub Axis** column to distinguish the two.

The **Line #** corresponds to the axis number on the back of the CNC controller. Each signal generator can command up to five independent signals simultaneously.

Use the **Axis** column's drop downs and assign a logical axis letter (X, Y, Z, etc.) to a physical motor line on the signal generator (as described under Line #).

If more than one motor is required for an axis, assign the same axis letter to all the appropriate lines. Then, assign a different number in the **Sub Axis** column. This drop down allows the software to distinguish between multiple motors used on the same axis for operations that require unique identifiers, such as homing.

The next column, **Direction**, can modify the direction polarity of each motor. Verify proper direction by saving the setup and using the jog buttons in the CNC screen to drive each axis. If it moves as expected, the polarity is correct. If any direction is incorrect, use the buttons under **Direction** to change the polarity from Positive to Negative (or vice-versa).

With a dual-driven gantry, it is important that the sub-axis column is configured correctly. If the shafts for the motors both point in the same direction, ensure that the polarity is the same. If their shafts point toward each other (or away from each other), the two lines should be set to opposite polarity.

Note: A servo system with a slaved drive should set that line's axis and sub axis as N/A.

Drive Parameters

Navigate to the **Machine>>Drive Parameters** section to make adjustments to the **Control Parameters** and **Drive Mechanism** of each axis.

Control Parameters

Set the control parameters to the default value based on the drive. Click on the drop down for an axis and select **Edit Control Parameters...** to open up the dialog outlined below. Selecting the appropriate controller model under **Motor Signals** of the **System>>Controller** section will default the values to the following:

- **2.5A/5A Compact Micro Stepper**
 - Step Mode: 4 micro steps/full step
 - Steps per Rev: 200 full steps/motor rev
- **8A Pro/Titanium Series Micro Stepper**
 - Step Mode: 10 micro steps/full step
 - Steps per Rev: 200 full steps/motor rev
- **25A Pro/Titanium Series Servo**
 - Encoder Divisor: 5 encoder ticks/SG pulse
 - Encoder Resolution: 1000 lines/rev

Note: These values may vary significantly if using third-party drives or motors. Consult the **FlashCut CAD/CAM and CNC 10 User's Guide** for additional information on the configurable parameters.

Drive Mechanism

Next, define the mechanics that drive each of the axes. Click on the drop down for an axis and select the **Edit Drive Mechanism...** option.

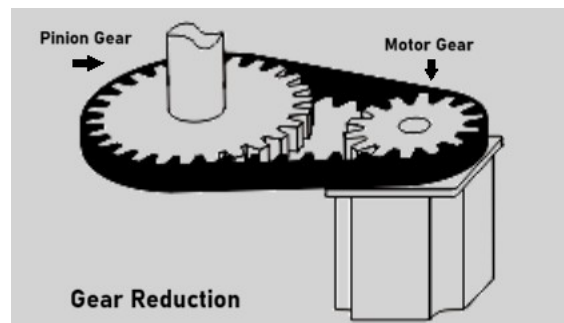
Note: This section's units can be defined as either English or Metric units independent of those defined under **System Units** in the **System>>General** section of the configuration. For the purposes of this guide, we will be defining them as English units.

General Linear

By default, the mechanism for X and Y axes will be defined as **General Linear**. This works best if the specifications of the mechanics are not available, as only one parameter is required and can be further refined using the calibrate button.

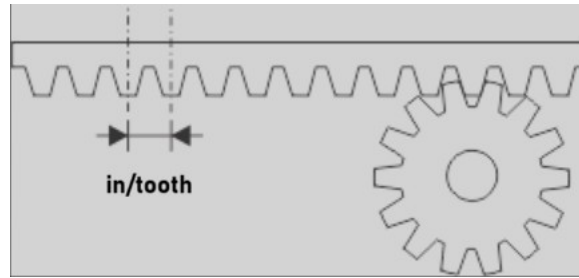
The **Drive Ratio** drop down is used to define the distance traveled per motor revolution or the number of motor revolutions per unit distance. Since this process will require commanded motion to observe motor rotations, ensure the axes have already been mapped out correctly under the **Machine>>Axes** section.

Note: For all drive mechanisms other than general linear, a **Gear Reduction** value is required. The first option in the drop down, **Enter Decimal Gear Ratio**, is used to account for all gearing between the motor and the drive mechanism. If a drive mechanism is composed of multiple gearing systems, use the decimal gear ratio of the whole system, found by multiplying each ratio together. In a system without gearing, like a direct-drive mechanism, use a value of 1.0 for the ratio. The second option in the drop down, **Enter Number of Teeth**, requires tooth counts for **on Motor Gear** as well as **on Pinion Gear/on Lead Screw Gear**.



Rack and Pinion

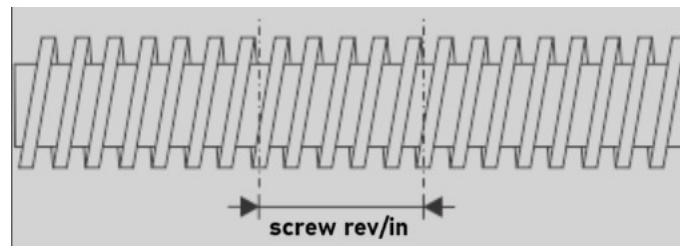
A common drive mechanism that consists of a gear running back and forth on a track.



This type of drive mechanism will require **Rack Spec** (Specifications). Select between **Tooth Pitch** and **Tooth Count** in the drop down. For tooth pitch—the distance between each tooth—be sure to measure from the exact same point across each tooth. Alternatively, enter in the number of teeth per inch if using tooth count, but don't count the first tooth. Lastly, ensure to include the **Pinion Tooth Count**.

Lead Screw

By default, the mechanism for the Z axis will be defined as a Lead Screw but other axes may also be defined as Lead Screw. This mechanism consists of a threaded screw that transforms rotary motion into linear motion.



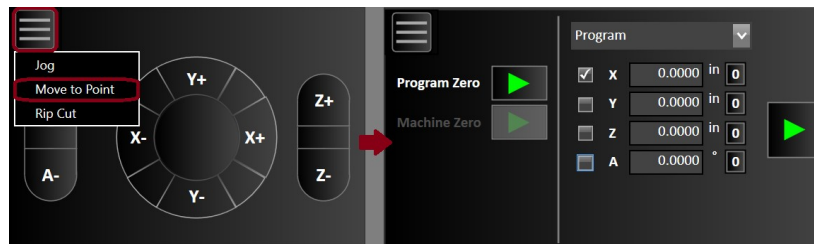
This type of drive mechanism will require **Lead Screw Spec** (Specifications). Similar to a rack and pinion, select between **Thread Pitch** and **Thread Count** in the drop down. For thread pitch—the distance between each thread—be sure to measure from the exact same point between threads. Alternatively, enter in the number of threads per inch if using thread pitch and don't count the first thread.

Motion Calibration

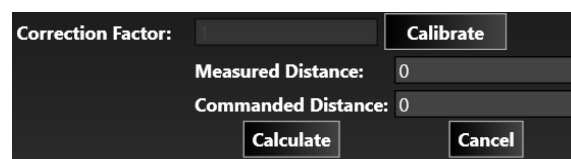
To fine-tune motion, use the primary fab head to determine motion error and correct it on each axis.

Note: All settings in this section of the guide assume inch units; multiply these settings by 25 if using metric units.

1. To begin, load the most precise tool or consumable for the primary fab head and put a piece of scrap material that runs along the X axis.
2. Navigate to the **CNC** screen and use the **Jog** controls to move the fab head to the negative extent of the X axis along the scrap material.
3. Mark the fab head's current position above the scrap material.
4. Use the **Set** drop down in the **Program** coordinates section to select **Zero X**.
5. Then, use the **Move to Point** mode to move at least 10" (or just short of max travel if the total travel length is shorter than 10".) Only the X axis checkbox should be selected so no other axes execute motion.



6. Measure the distance physically traveled.
7. Open the configuration and navigate to the **Machine>>Drive Parameters** section. Open the **Edit Drive Mechanism...** window of the X axis.
8. Click the **Calibrate** button and enter in both the measure and commanded distances. Then, click **Calculate** followed by the **OK** button and save the configuration.



9. Repeat steps 1-8 for the Y axis and Z axis. For most applications, the Z axis will have a smaller scale of travel, and the surface of the material should be considered the negative end of travel.

Note: These steps can be used multiple times to dial-in on the Tool Positioning Resolution which will increase the accuracy of the machine's cuts.

Optimizing Feedrates and Ramping

All parameters must be set conservatively to prevent the system from losing position. Due to variations in the drive mechanism for each axis, ensure that these instructions are performed at several positions along each axis, in both directions. Navigate to the **Machine>>Axes** section and scroll down to the **Feedrates and Ramping** sub-section.

Note: All settings in this section of the guide assume inch units; multiply these settings by 25 if using metric units.

Start/Stop Feedrate (in/min)

This sets the initial and ending feedrate for ramping moves.

1. Start by entering 349 in/min for the X axis **Start/Stop Feedrate**, and 350 in/min for the **Maximum Feedrate**, and then save the configuration. If the software doesn't allow the **Maximum Feedrate** to be set to 350 in/min, use the largest value that is allowed, and set the **Start/Stop Feedrate** to 1 less than that value.
2. Navigate to the **CNC screen**. Use the **Jog** panel to move the fabrication head to a location on the machine with sufficient room to continuously move in the X positive direction, an inch at a time.
3. Switch from the **Jog** panel to the **Move to Point** panel. Use the drop down to change from **Program** to **Incremental**. Only the X axis should be selected to prevent movement in the other axes. Enter 1 inch in the appropriate field.
4. Use the '+' button in the upper-right corner of the jog-speed slider to enter the feedrate manually. The feedrate should be set at a relatively slow value (such as 5 in/min). Then, press play to execute the move.
5. If the motor slips, repeat this process with a slower feedrate. If the motor doesn't slip, try a faster feedrate.

Note: Slight slippage can be detected using a dial indicator, homing to limit switches, or reading the values on a hand-wheel if available on the machine.

6. Repeat this process until the highest feedrate that doesn't cause motor slippage is found. Run along the length of the X axis in both directions to verify there is no slippage at any point.
7. Enter in 70% of the value for the highest feedrate that doesn't cause motor slippage in the **Start/Stop Feedrate** column.
8. Repeat this process for all axes.

Maximum Feedrate (in/min)

This sets the maximum feedrate for the system. No matter how fast the system is commanded to move, the axis will never move faster than this value. Use the **Start/Stop Feedrate** from the previous section and follow the steps below.

1. Enter 1 in/sec² in the **General Ramp Rate** column for the X axis. Then, set the **Maximum Feedrate** to 350 in/min and save the configuration. If the software settings don't allow the **Maximum Feedrate** to be set to 350 in/min, use the largest value the software will allow.
2. Navigate to the **CNC screen**. Use the **Jog** panel to move the fabrication head to a location on the machine with sufficient room to continuously move in the X positive direction, an inch at a time.
3. Switch from the **Jog** panel to the **Move to Point** panel. Use the drop down to change from **Program** to **Incremental**. Only the X axis should be selected to prevent movement in the other axes. Enter 1 inch in the appropriate field.
4. Use the '+' button in the upper right of the jog-speed slider to enter the feedrate manually. The feedrate should be set to double the **Start/Stop Feedrate** of the X axis. Then, press the play button to execute the movement.
5. If the motor slips, repeat this process with a slower feedrate. If the motor doesn't slip, try a faster feedrate.

Note: Slight slippage can be detected using a dial indicator, homing to limit switches, or reading the values on a hand-wheel if available on the machine.

6. Repeat this process until the highest feedrate that doesn't cause motor slippage is found. Run along the length of the X axis in both directions to verify there is no slippage at any point.
7. Enter in 70% of the value for the highest feedrate that doesn't cause motor slippage in the **Maximum Feedrate** column.
8. Repeat this process for all axes.

General Ramp Rate (in/sec²)

Ramping Rates typically range from 1 to 10 in/sec². Slower ramping rates require more time to ramp up to the maximum feedrate and to ramp down to a stop. The goal is to set a fast ramping rate that will start and stop the toll responsively without losing steps. Fast ramping rates can also allow acceleration past resonant speeds of a stepper motor. Use the **Start/Stop Feedrate** and **Maximum Feedrate** from the previous section and follow the steps below.

1. Enter 1 in/sec² in the **General Ramp Rate** for the X axis and save the configuration.
2. Navigate to the **CNC screen**. Use the **Jog** panel to move the fabrication head to a location on the machine with sufficient room to continuously move in the X positive direction, an inch at a time.
3. Switch from the **Jog** panel to the **Move to Point** panel. Use the new drop down to change **Program** to **Incremental**. Make sure that X is the only axis that is selected and enter 1 inch in the appropriate field.

Note: If the table doesn't reach a continuous speed before ramping down to a stop, there isn't enough room to fully ramp up and down. Increase the length of the move and repeat.

4. Use the '+' button in the upper right of the jog-speed slider to select the **Rapid Move** checkbox. Then, press play to execute the move.
5. If the motor slips, repeat this process with a lower ramping rate. If the motor doesn't slip, try a higher ramping rate.
Note: Slight slippage can be detected using a dial indicator, homing to limit switches, or reading the values on a hand-wheel if available on the machine.
6. Repeat this process until an optimal **General Ramping Rate** for the X axis is found. Run along the length of the X axis in both directions to verify there is no slippage at any point.
7. Enter in 70% of the value found in the **General Ramping Rate** field.
8. Repeat this process for all axes.

Note: Once the optimal ramping rate is determined for each axis, re-test the **Maximum Feedrate** to see if they can be set any higher.

Feedrate Move Ramp Rate (in/sec²)

This sets the rate of acceleration and deceleration for feedrate moves. Most setups will simply copy the General Ramping Rate to the Feedrate Move Ramp Rate.

Emergency Ramp Rate (in/sec²)

This sets the rate of acceleration and deceleration used in an emergency stop, typically via a "Feed Hold (Emergency)" input trip. Once again, most setups copy the General Ramping Rate to the Emergency Ramp Rate.

Continuous Contouring Feedrate Tol. (in/min)

This allows smooth transitions from one G-Code line to the next if the two moves are in line with one another. A typical default setting for the continuous contouring feedrate tolerance is to copy the Start/Stop Feedrate (in/min) value.

The higher the tolerance is set, the smoother a G-Code file will be executed. If the values are set too high, loss of steps or servo errors can occur. Since this parameter is enabled by default, setting the Continuous Contouring Feedrate Tolerance to zero for all axis will essentially disable the feature.

Inputs

Drive Faults

Setting up input lines as drive faults has two advantages: fault reporting and safety. With fault lines set up, the operator will be notified if a fault occurs and unable to move the machine until the fault is resolved.

Take note of the **Motor Line/Axis Mapping** order under the **Machine>>Axes** page. Each numbered line corresponds with a different input line for fault reporting.

Then, navigate to **I/O>>Input Lines** section and assign each motor line to the corresponding input lines using the following chart.

Axis Line #	Input Line #
1	9
2	11
3	13
4	15
5	17

Assign a **Safety** function and use the description field to fill in which axis it corresponds to.

The lines in a stepper system should be configured as Normally Open (N.O.). In a servo system, these fault lines should be set as Normally Closed (N.C.).

Plasma

To get started, open the configuration window by clicking the 'Double Gear' icon found in the upper-right of the software screen:

Navigate to the **Machine>>Fabrication Heads** page in the configuration using the navigation tree.

When adding a new plasma fabrication head (fab head), click the **New Fab Head** button.

If editing an existing plasma fab head, select one to edit from the list by clicking on the name of the plasma in the chart. This will expand additional configuration settings.

When adding a new fab head where the **Type** is **Plasma**, ensure to select a **SubType** from the list of the supported plasma models.

General

The **ID#** number is automatically generated, incrementing the value for each instance of the same fab head type. For example, if there are two plasma fab heads already configured, adding another will set its ID# as 3, though this can be overwritten. It is used in combination with Type to create a unique identifier. In the example given, this would be plasma3.

Use the **Model** drop down to select the appropriate model of plasma fab head.

The **Name** will default to the fab head type (or Model). The name is used on the CAM and CNC screens to identify the fab head type. It can be edited, if desired.

When using more than one fab head, defining the **Offset** for **X and Y** will allow the software to transition from one fab head to the next. As an example, a system that has both a plasma fab head and a laser pointer fab head would benefit from an offset. With the plasma fab head considered as the primary its offset (X, Y) should be set as 0,0. The laser pointer fab head would require an offset relative to that of the primary tool.

Next, use the **Lift Axis** drop down to select the relevant letter for the lift axis of the plasma fab head. This letter is also used to map the Drive Mechanism under the Drive Parameters section to ensure proper motion. This axis letter is moved using the jog controls in the CNC screen, much like the control for any X or Y positioning. Most commonly the lift axis will be assigned to **Z, W, V,** and **U**.

A **Lift Axis** of **None** indicates all positioning of the lifter is done outside the control software. This could include a torch which is positioned by hand, pneumatic lift, or even a motor-driven torch that does not use the control software to position.

The **Lift Axis Offset** is used to indicate the offset from one fab head to another, similar to the (X, Y) offset. This offset is necessary only if multiple fab heads share the same axis letter. This means that for most fab heads, the lift axis offset is 0.

Defining **Lead Length** will be important for a proper cut-chart lookup of the plasma's cut parameters. Use the drop down to select the correct lead length.

Communication (Add-On)

Some plasma models offer additional controls through a communications port. Depending on the model, this may simply allow control of the amperage but complex plasma systems may offer the control of cutting gas mixes, report status codes, as well as diagnostics. This feature will require the appropriate license and hardware.

In the **Communication** line, select the **RS232/RS422/RS485** button. This will expand to show additional parameters. Ensure that the correct **COM Port** is selected using the drop down. If this is not known, it can be found in the **Windows Device Manager** under **Ports (COM & LPT)**.

After setting your COM Port, save the configuration. The plasma fab head will now have a status window with options that vary from plasma to plasma, but they should all include a Connect button to establish communication.



If **None** is selected, there is no communication between the plasma power supply and the software.

Torch Control

In this sub-section, the basic on/off controls for **Torch Control** are defined. These parameters will correspond to the behavior of the Torch On/Off button in the **CNC screen**.



Disabled/Enabled

For **Torch On**, use the drop down next to **Output Line(s) State** to select the correct output line and state. The majority of plasma systems, especially those using the Stingray THC Kit, should set the output line to **1** and state to **On**.

Similarly, for **Torch Off**, use the drop down next to **Output Line(s) State** to select the correct output line and state. The output line should complement that of the Torch On parameter, **1**, but its state should be set to **Off**.

Some plasma systems may require multiple configured output lines. In this case press the **[+]** button to add more output lines as necessary.

For most plasma systems, **Turn On Using** and **Turn Off Using** should be set to **Output Lines**. Advanced or custom plasma systems may benefit from using a macro. This macro—a sequence of commands—must include at least one command that sets the **Output Line(s) State** appropriately. To configure the macro, click the **Edit** button which will launch a macro editor box. Lastly, configure an optional **Delay** after setting the output line state. For the majority of configurations, no delay is required.

Cutting

This subsection is where the commands for the start and end of each cut in a G-Code program are specified. Most plasma models will have **Cutting** by default but plasma models with marking capabilities have an **Enable** checkbox that can be deselected.

Start Of Cut

This is used to describe the routine that is executed automatically at the beginning of any feedrate move for a cut process. To configure the macro, ensure that the **Start Of Cut** checkbox is selected.

For most plasma systems, the default macro—a sequence of commands—will work best. To make changes to the macro, click the **Edit** button which will launch a macro editor box. The following is the default **Start of Cut** macro for a plasma fab head:

```
G605 (Move to Initial Height)

#Fabhead.TurnOn (Torch On)

#FabHead.WaitForReadyToFab

#FabHead.RunPierceStyle

G601(Begin Plasma Cutting)
```

If the checkbox for **Start of Cut** is left unchecked, no macro will run at the beginning of any feedrate move.

End Of Cut

This used to describe the routine that is executed automatically at the beginning of a rapid move, typically at the end of a cut. To configure the macro, ensure the **End Of Cut** checkbox is selected.

For most plasma systems, the default macro will work best. To make changes to the macro, click the **Edit** button which will launch a macro editor box. The following is the default end of cut macro for a plasma fab head:

```
#Fabhead.TurnOff (Torch Off)

G600 (End Plasma Cutting)

#FabHead.WaitForFabStopped
```

If the checkbox for **End Of Cut** is left unchecked, no macro will run at the end of the cut process.

Cut leadin when returning to toolpath

This checkbox details the behavior of the fab head when the software resumes a program after a pause, such as a feed-hold, and the tool has been moved away from the toolpath. If the checkbox is selected, the torch will begin cutting from its current position towards the toolpath after a pause. This feature is helpful to prevent gouging the part in the event that cutting has to be interrupted. If the box is unchecked, the torch will instead rapid move to the point at which the program was paused and resume cutting.

Standalone Piercing

This checkbox is used to describe the routine that is executed automatically for a standalone (separate) pierce in the G-Code program. To configure the macro, ensure the **Enable** checkbox under **Standalone Piercing** is selected.

For most plasma systems, the default macro will work best. To make changes to the macro, click the **Edit** button which will launch a macro editor box. The following is the default end of cut macro for a plasma fab head:

```
G605 (Move to Initial Height)
```

```
#Fabhead.TurnOn (Torch On)
```

```
#FabHead.WaitForReadyToFab
```

```
#FabHead.RunPierceStyle
```

```
#Fabhead.TurnOff (Torch Off)
```

```
#FabHead.WaitForFabStopped
```

If the checkbox for **Standalone Piercing** is left unchecked, no macro will run for a standalone piercing process.

Marking

If a plasma system is capable of both cutting and marking, this subsection is where the commands for the start and end of each mark in a G-Code program are specified. To begin, ensure that the **Enable** checkbox under **Marking** is selected.

Start of Mark

This is used to describe the routine that is executed automatically at the beginning of a feedrate move for a mark.

To configure the macro, ensure that the **Start Of Mark** checkbox is selected. For most plasma systems, the default macro will work best. To make changes to the macro, click the **Edit** button which will launch a macro editor box. The following is the default Start of Mark macro for a plasma fab head:

```
G605 (Move to Initial Height)
```

```
#Fabhead.TurnOn (Torch On)
```

```
#FabHead.WaitForReadyToFab
```

G601 (Begin Plasma Marking)

If the checkbox for **Start of Mark** is left unchecked, no macro will run at the start of the mark process.

End of Mark

This checkbox is used to describe the routine that is executed automatically at the beginning of a rapid move during a mark process, typically at the end of a mark.

To configure the macro, ensure that the **End of Mark** checkbox is selected. For most plasma systems, the default macro will work best. To make changes to the macro, click the Edit button which will launch a macro editor box. The following is the default End of Mark macro for a plasma fab head:

```
#Fabhead.TurnOff (Torch Off)
```

G600 (End Plasma Marking)

```
#FabHead.WaitForFabStopped
```

If the checkbox for **End of Mark** is left unchecked, no macro will run at the end of the mark process.

Arc-Transfer Monitoring

This subsection will cover the configuration of the **Arc-Transfer** signal, also commonly referred to as the “OK to Move” signal. The commands to monitor the arc-transfer state, **WaitForReadyToFab** and **WaitForFabStopped**, are baked into the **Start of Cut** and **End of Cut** macros for plasma operation.

To enable these commands, check the **Configure** checkbox under **Arc Transfer Monitoring**.

Next, use the **Monitor Input Line** drop down to select the appropriate input line. Then, use the **Arc-Transferred State** drop down to select the signal state for the input line. The default values assume the use of the Stingray Torch Height Control Kit so Monitor Input Line should be set to **20**, and its Arc-Transferred State should be set to **Tripped**.

In the event the arc transfer signal does not reach the appropriate state during a **WaitForReadyToFab** or **WaitForFabStopped** command, the **Timeout** value entered is the duration the system will wait before an error message pops up.

The entered value for **Runtime Estimate** will be calculated into the progress meter when running a G-Code program. It is also relevant job costing report, an optional nesting feature available in CAM.

Lastly, selecting the **Execute Feed Hold On Loss of Arc** checkbox will pause the program if the system detects a loss of the arc transfer signal while running a G-Code program.

Torch Height Control

This subsection will allow

The value entered for **Arc Voltage Divisor** corresponds to the ratio of the cutting voltage from the plasma power supply divided by the control voltage. The ideal arc voltage divisor for the control system is 50:1. Check with your plasma manufacturer as some plasma systems have a configurable voltage divisor. If that is the case, select a divisor at or close to 50:1.

The **Lock Out Factor** is a velocity lock out factor. This will pause changes in torch height when the velocity of the machine slows down to take corners to prevent the plasma torch from diving into the material as the kerf widens. The factor dictates at what percent of the specified feedrate this occurs. By default, this value is 94%.

The **Lock Out Hysteresis** is a buffer value used to prevent the height control from rapidly pausing and resuming during velocity changes. For example—if the lockout factor is 94% and the lockout hysteresis is 5%—when the velocity slows down to 94% of the program feedrate, the height control will pause lifter adjustments. The height control will only then resume lifter adjustments once velocity has reached 99% (lockout factor plus hysteresis) of the program feedrate. By default, this value is 5%.

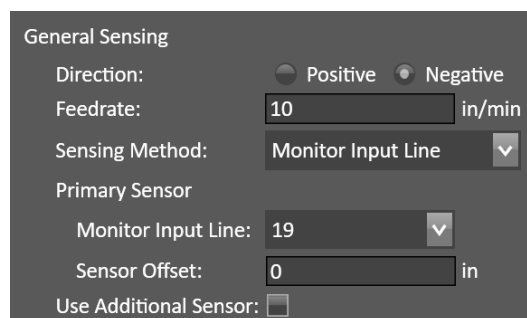
When the plasma torch is idle, the actual displayed voltage is very low and may bounce around by several volts. To avoid this, any voltage below the entered value for **Idle Voltage Threshold** will not be displayed. This is just for aesthetics, and the value is typically set to 20 volts.

The slider for the **Sensitivity** will adjust how reactive the height control will behave with voltage fluctuation. The slider corresponds to the Dead Band Voltage displayed below. Most plasma systems work well when the dead band voltage is about 1 volt. This can be adjusted to be more or less sensitive based on your plasma system and the type of material that is being cut.

For most plasma systems, the **Use Advanced Adjustments** should not be used unless directed by a Flashcut technician.

General Sensing

Here, you will specify the sensing parameters to be used for the **Touch Off** and **Program Zero Sensing** process of the plasma fab head.



The screenshot shows a configuration panel titled "General Sensing" with the following settings:

- Direction:** Radio buttons for "Positive" and "Negative", with "Negative" selected.
- Feedrate:** A text input field containing "10" followed by "in/min".
- Sensing Method:** A dropdown menu showing "Monitor Input Line".
- Primary Sensor:** A sub-section containing:
 - Monitor Input Line:** A dropdown menu showing "19".
 - Sensor Offset:** A text input field containing "0" followed by "in".
- Use Additional Sensor:** An unchecked checkbox.

First, select the **Direction** in which the fab head will sense the workpiece. This is typically negative.

Then, define the **Feedrate**—or speed—at which the fab head will travel during the sensing process. Setting the value too high can result in inaccurate results or other errors. A typical value is 10 in/min. The sensing feedrate cannot exceed the **Start/Stop Feedrate** of the lift axis found in the “**Machine... Axes**” settings.

Next, use the **Sensing Method** drop down to select between ‘*Monitor Input Line*’ or ‘*Monitor Motor Current*’. Traditional switches (mechanical, proximity sensors, etc.) fall under the ‘*Monitor Input Line*’ section. ‘*Monitor Motor Current*’ will only be available to servo systems.

Select the input line number to be used as the primary sensor. This input line must also be

configured as a control-type input in the “**I/O...Input Lines**” section of the configuration.

The sensor offset is used to define a distance from when the sensor returns to a normal state after contacting the workpiece and the point when the plasma torch lifts off the material. The value is typically negative.

If a second switch is being used as a backup, check the **Use Additional Sensor** box. As with the primary sensor, you will need to define an additional input line as well as the corresponding offset value.

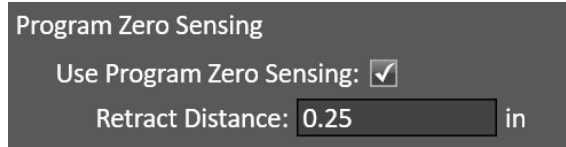
The alternate sensing method, ‘*Monitor Motor Current*’, is only available for a servo system.

Start by choosing a **Current Limit** that will indicate when the lift axis has encountered the material. By default, this is 1A. If the system is causing significant deflection in the material, this value needs to decrease. If the system instead is setting the zero before it reaches the material, this value should be increased.

The **Current Debounce** dictates the time that the motor current must exceed the limit. If the sensing is not consistent, this value should be increased. If the sensing is slow to react, decrease this value.

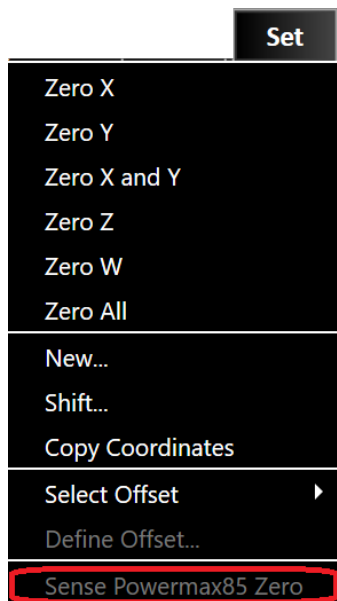
Program Zero Sensing

Once the General Sensing settings are configured, the sensing input line can now be used for Program Zero sensing. Select **Program Zero Sensing** checkbox to enable its use with the fab head's lift axis.



The entered value for **Retract Distance** is the position the lift axis will move to after sensing the workpiece. By default, this value is a quarter inch.

When checked, an option to 'Sense [Plasma] Zero' will appear in the **Set** drop down available in the CNC screen. When this option is selected, you'll be able to zero your Z program coordinates using the input line(s) configured under the general sensing section.



Additional Controls

Safe Height is a parameter on the CNC screen that defines the height the lift axis will travel to at the start of the program and between cuts. It is important to note that the system will only move to the safe height during this process and no motion will occur if the lift axis is already above the safe height.

Safe Height: 0.2500 in

Use the **Specify Safe Height As** buttons to define the safe height parameter as a *'Program Coordinate'* or an *'Incremental Distance'*.

Additional Controls
Specify Safe Height As: Program Coordinate Incremental Distance

For example, if the safe height is 0.25 inches and it is listed as a *'Program Coordinate'*, the lift axis will rise to a coordinate of 0.25 inches when a move to safe height is required. Instead, if it is set as an *'Incremental Distance'*, the lift axis will move 0.25 inches from its current height.

Touch Off

In order to sense the material prior to a cut, select the **Touch Off At Start (G605)** checkbox. This will add a parameter on the CNC screen to toggle **Touch Off** during a Move to Initial Height (G605) command. This command is commonly found in the Start of Cut, Start of Mark, and Standalone Piercing macros.

Touch Off
Touch Off At Start (G605):
Zero Lift Axis Program Coord:
Rapid Move to Start Height:
Enable Suppression Radius:

When **Zero Life Axis Program Coord** is checked, the lift axis's zero will be reset every time a touch is performed during a program. If this is unchecked, the lift axis's zero will not change during a touch off.

When **Rapid Move to Start Height** is checked, it adds the **Start Height** parameter below the **Touch Off** checkbox in the **CNC screen**. The system will make a rapid move from the Safe Height to the Start Height before performing a touch off. This can decrease sensing time while allowing larger clearance to reduce the risk of contacting a tip up.

Touch Off
Start Height: 0.5000 in
Suppression Radius: 0.0000 in

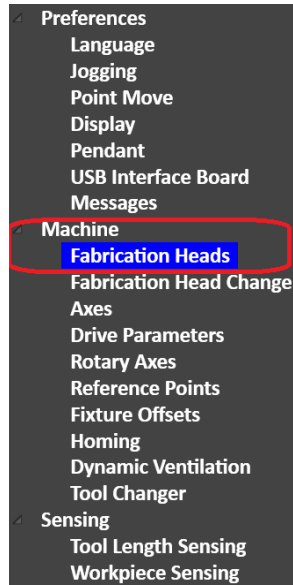
When **Enable Suppression Radius** is checked, the system will skip touch offs within a specified distance of the previous touch off. This feature helps with efficiency by reducing the number of touch off sequences.

OxyFuel

To get started, open the configuration window by clicking the ‘*Double Gear*’ icon found in the upper-right of the software screen:



Navigate to the “**Machine...Fabrication Heads**” page using the configuration page browser on the left side of the configuration window.



When adding a new OxyFuel fab head (fabrication head), click the ‘*New Fab Head*’ button.

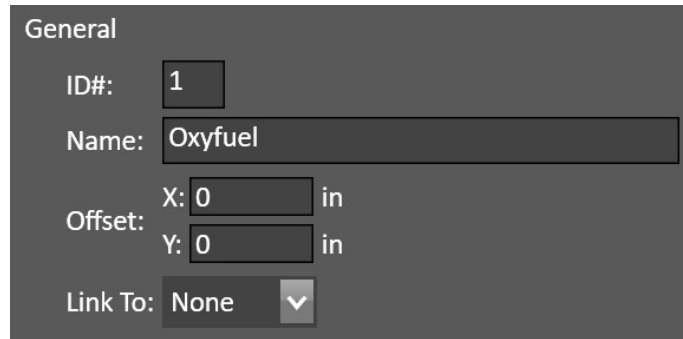


If editing an existing OxyFuel fab head, select which one to edit from the list and click on it to expand additional configuration settings.

ID	Name	Model	Offset
oxyfuel1	Oxyfuel	Oxyfuel	X:0.0000 Y:0.0000
laserpointer1	Laser Pointer	Laser Pointer	X:0.0000 Y:0.0000

General

Next, you'll need to establish the general settings for the OxyFuel fab head.



The screenshot shows a 'General' settings panel with the following fields:

- ID#:** A text input field containing the number '1'.
- Name:** A text input field containing 'Oxyfuel'.
- Offset:** Two input fields, one for 'X' and one for 'Y', both containing the number '0'. Each field is followed by the unit 'in'.
- Link To:** A dropdown menu currently set to 'None'.

The **ID#** number is automatically generated, incrementing the value for each instance of the same fab head type. For example, if there are currently three OxyFuel fab heads configured, adding another will set its ID# as 4. This number is used with the fab head type to create a unique identifier. However, it can be overwritten.

The **Name** will default to the fab head type, in this case OxyFuel. The name is used on the CAM and CNC screens to identify the fab head type. It can be edited, if desired.

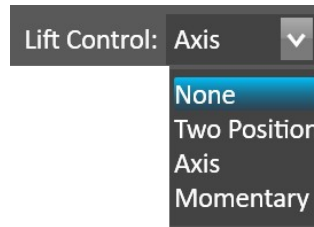
When using more than one fab head, defining the **Offset** for **X** and **Y** will allow the software to transition from one fab head to the next. For example, an end user with an OxyFuel cutting system might also add a laser pointer fab head. When configuring the laser pointer, they would need to enter in an offset (X, Y) in relation to the primary fab head. In this case, the OxyFuel fab head is considered the primary, and its offset (X, Y) should be set as 0,0.

Lastly, the **Link To** drop down allows for another configured OxyFuel to be linked for multiple simultaneous use. When selected, some configuration parameters which are linked to the primary OxyFuel fab head will be disabled and only unique parameters will be available. For most OxyFuel systems, the **Link To** can left as 'None'.

Lifter

Lift Control

After establishing the general parameters, you'll need to use the **Lift Control** drop down and select the appropriate type for your OxyFuel fab head. Choose between *'None'*, *'Two Position'*, *'Axis'*, and *'Momentary'*.

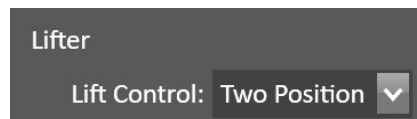


Lift Control: None

When the Lift Control is set to *'None'*, this will indicate that all positioning of the lifter is done outside the control software. This could include a torch which is positioned by hand, a pneumatic lift, or even a motor-driven torch that does not use the control software to position.

Lift Control: Two Position

A Lift Control of *'Two Position'* describes a lifter that is either all the way up or all the way down in a default state and relies on an output to move the torch into position.



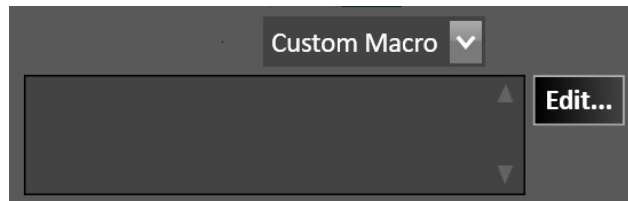
A common example of this type of lift control is a pneumatic lifter. Lifters of this type will normally rest all the way up and use compressed air to push the torch all the way down when the appropriate Output is enabled. These types of lifters also commonly use a roller system to maintain proper cut height.

Using a pneumatic lifter as an example, the **Up** state will need its **Output Line(s) State** set to *'4'* and *'Off'*. If multiple Output lines are required, the **+** button will add Output lines to configure.



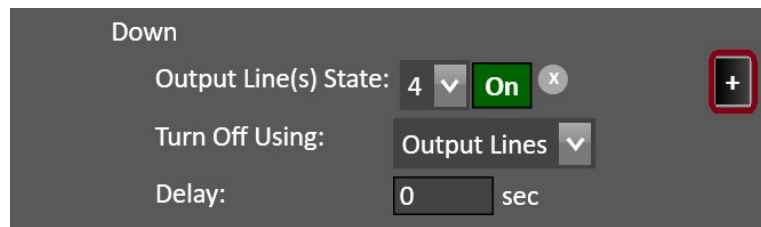
The **Turn On Using** drop down should be set to *'Output Lines'*.

For advanced/custom systems, you can use a *'Custom Macro'*. This macro—a sequence of commands—must include at least one command that sets the **Output Line(s) State** appropriately. To configure the macro, you can enter the routine into the macro editor box of click the *'Edit...'* button to launch a macro editor dialog box.



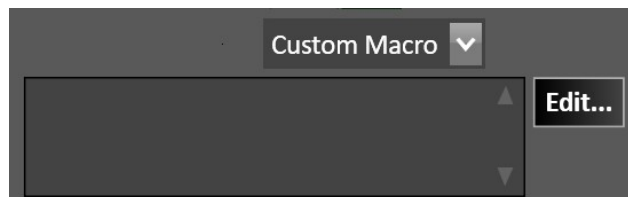
An optional **Delay** can be added to allow time for the lifter to reach the **Up** position.

The **Down** state will need its **Output Line(s) State** set to '4' and 'On'. If multiple Output lines are required, the '+' button will add Output lines to configure. The **Turn Off Using** drop down should be set to 'Output Lines'.



For advanced/custom systems, you can use a 'Custom Macro'. This macro—a sequence of commands—must include at least one command that sets the **Output Line(s) State** appropriately. To configure the macro, you can enter the routine into the macro editor box of click the 'Edit...' button to launch a macro editor dialog box.

Once again, an optional **Delay** can be added to allow time for the lifter to reach the **Down** position.

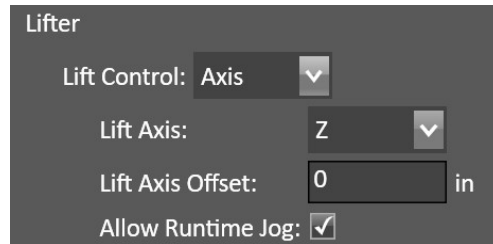


After saving your changes for the two position lifter, buttons will be added to the fab head's control panel on the CNC screen to manually control the position of the lifter. The lit-up arrow will reflect the current state.



Lift Control: Axis

A Lift Control of 'Axis' describes a lifter that is precisely controlled using a lettered axis configured in the control software. This type of lift control typically utilizes a stepper or servo motor.



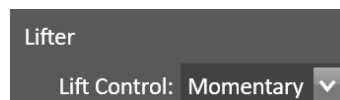
The **Lift Axis** is the relevant letter for the OxyFuel fab head. This letter is also used to map the Drive Mechanism under the Drive Parameters section to ensure proper motion. This axis letter is moved using the jog controls on the CNC screen, much like the control for any X or Y positioning. Most commonly, the lift axis will be assigned to 'Z', 'U', 'V', or 'W'.

The **Lift Axis Offset** is used to indicate the offset from one fab head to another, similar to the (X, Y) offset. This offset is necessary only if multiple fab heads share the same axis letter. This means that for most fab heads, the lift axis offset is 0.

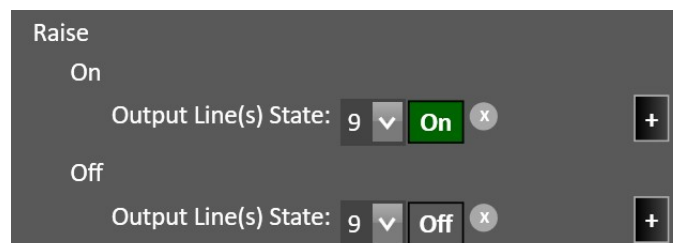
When the **Allow Runtime Jog** checkbox is selected, it will enable the lift axis to be moved using the jog controls while running a program. In order to allow runtime jog for a lift axis, there cannot be any motion commands for that axis in the program.

Lift Control: Momentary

A Lift Control of 'Momentary' describes a lifter that is controlled (typically) by a motor. The motor is spun in one direction to raise the lifter and spun in the other direction to lower it. This behavior is controlled with the momentary press of the corresponding button in the CNC screen.



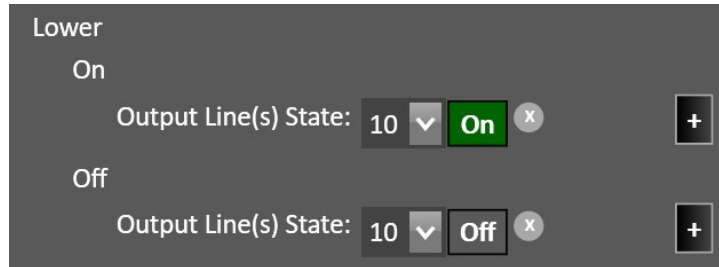
First, describe the behavior of the lifter when the Raise button is pressed and released. You'll notice that the **Raise** section is split into **On** and **Off** states that represent a press and release, respectively. In the example below, the Raise button is toggled by Output 9. To accomplish this, ensure that the **Output Line(s) State** for **On** is set to '9' and 'On'. Then, set the **Output Line(s) State** for **Off** to '9' and 'Off'.



If multiple output lines are required, the '+' button will add Output lines to configure. For most Oxyfuel systems, **Turn On Using** and **Turn Off Using** should be set to 'Output Lines'.

Next, you'll need to describe the behavior of the lifter when the lower button is press and released. In the pictured example below, you'll notice that the **Lower** section is also spit into **On** and **Off** states that represent a press and release, respectively.

Use the '+' button to add the output line in the **Output Line(s) State** for **On** and **Off**. The example below will use output 10 to lower the lifter when pressed and stop motion when released.



After saving your changes, the buttons for the momentary lifter will be added to the fab head's control panel on the CNC screen.



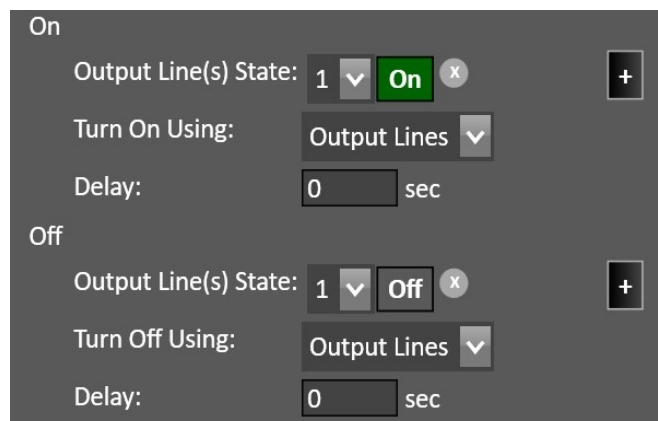
On/Off Control

This sub-section allows the software control over the oxygen valve and is used to configure the behavior of the Oxyfuel Cut button.

To begin, select the **Configure** checkbox under **On/Off Control**. Once configured, this will provide a button in the CNC screen that enables and disables the Oxyfuel Cut valve. If the Configure box is left unchecked, control of the Oxyfuel Cut valve is handled manually outside of the control software.

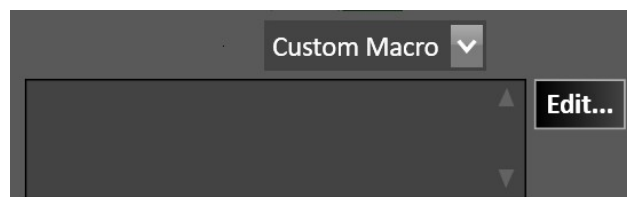


Next, describe what happens when the OxyFuel Cut button is enabled and disabled in the CNC screen. In the example below, the Oxyfuel Cut button is toggled by Output 1, but other Outputs can be used. To accomplish this, ensure that the **Output Line(s) State** for **On** is set to '1' and 'On'. Then, set the **Output Line(s) State** for **Off** to '1' and 'Off'.



If multiple output lines are required, the '+' button will add Output lines to configure. For most Oxyfuel systems, **Turn On Using** and **Turn Off Using** should be set to 'Output Lines'.

For advanced/custom systems, you can use a 'Custom Macro'. This macro—a sequence of commands—must include at least one command that sets the **Output Line(s) State** appropriately. To configure the macro, you can enter the routine into the macro editor box of



click the 'Edit...' button to launch a macro editor dialog box.

Lastly, you can configure an optional **Delay** after setting the Output line state. For the majority of configurations, no delay is required.

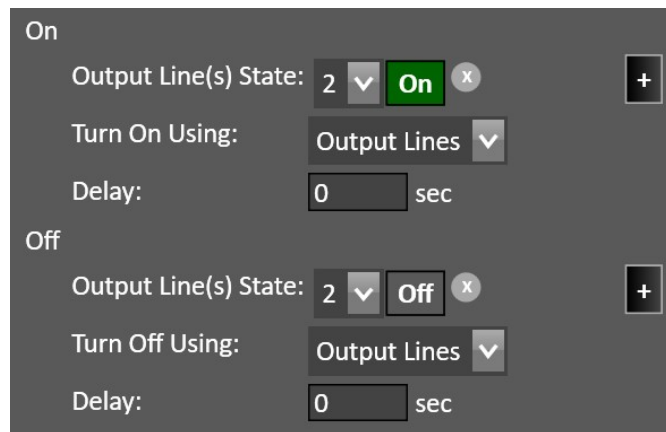
Automatic Ignition

This sub-section allows the software control over an automatic striker used to ignite the flame of an Oxyfuel fab head and is used to configure the behavior of the automatic striker button.

To begin, select the **Configure** checkbox under **Automatic Ignition**. Once configured, this will provide a button in the CNC screen that enables and disables the automatic striker. If the configure box is left unchecked, control of the automatic striker is handled manually outside of the control software.

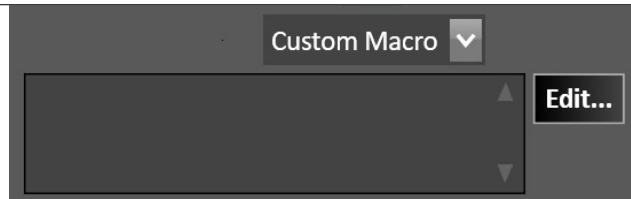


Next, describe what happens when the automatic striker button is enabled and disabled in the CNC screen. In the example below, the automatic striker button is toggled by Output 2, but other Outputs can be used. To accomplish this, ensure that the **Output Line(s) State** for **On** is set to '2' and 'On'. Then, set the **Output Line(s) State** for **Off** to '2' and 'Off'.



If multiple output lines are required, the '+' button will add Output lines to configure. For most Oxyfuel systems, **Turn On Using** and **Turn Off Using** should be set to 'Output Lines'.

For advanced/custom systems, you can use a 'Custom Macro'. This macro—a sequence of commands—must include at least one command that sets the **Output Line(s) State** appropriately. To configure the macro, you can enter the routine into the macro editor box of click the 'Edit...' button to launch a macro editor dialog box.

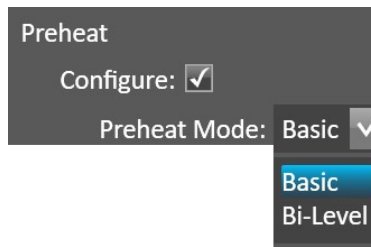


Lastly, you can configure an optional **Delay** after setting the Output line state. For the majority of configurations, no delay is required.

Preheat

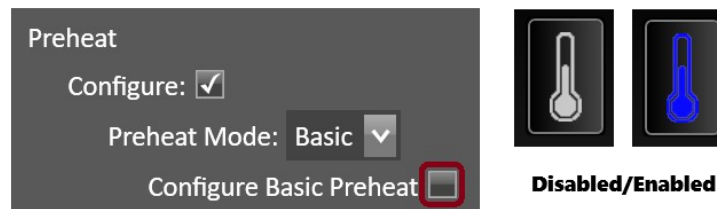
This sub-section allows the software control over the preheat valve and is used to configure the behavior of the various Preheat buttons.

To begin, select the **Configure** checkbox under **Preheat**. Next, use the **Preheat Mode** drop down to select between *'Basic'* or *'Bi-Level'*. This is used to describe the type of preheat system or how many preheat valves will be managed through the control system. If the Configure box is left unchecked, control of the Preheat valve is handled manually outside of the control software.

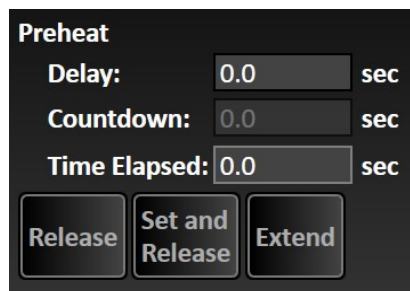


Basic Preheat

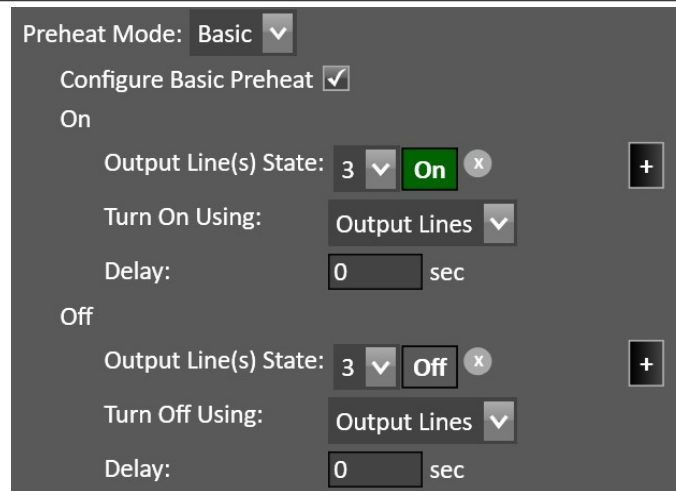
This option describes a preheat system that utilizes only one preheat valve. Use the **Preheat Mode** drop down to select *'Basic'*. Then, select the **Configure Basic Preheat** checkbox. Once configured, this will provide a button in the CNC screen that enables and disables the Basic Preheat valve.



If it is left unchecked, the preheat valve is managed manually outside of the control software. However, you will have access to the preheat parameters under the fab head's control panel in the CNC screen.



If **Configure Basic Preheat** is checked, you'll need to describe what happens when the Basic Preheat button is enabled and disabled in the CNC screen. In the example below, the Basic Preheat button is toggled by Output 3, but other Outputs can be used. To accomplish this, ensure that the **Output Line(s) State for On** is set to *'3'* and *'On'*. Then, set the **Output Line(s) State for Off** to *'3'* and *'Off'*.



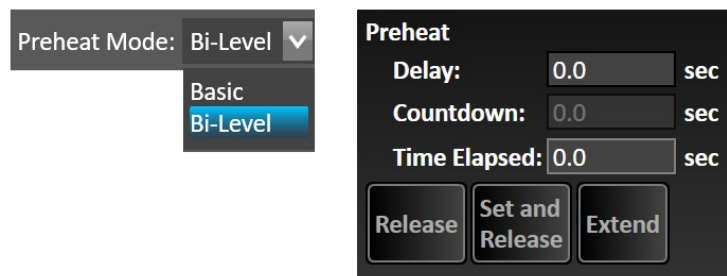
If multiple output lines are required, the '+' button will add Output lines to configure. For most Oxyfuel systems, **Turn On Using** and **Turn Off Using** should be set to 'Output Lines'.

For advanced/custom systems, you can use a 'Custom Macro'. This macro—a sequence of commands—must include at least one command that sets the **Output Line(s) State** appropriately. To configure the macro, you can enter the routine into the macro editor box of click the 'Edit...' button to launch a macro editor dialog box.

You can configure an optional **Delay** after setting the Output line state. For the majority of configurations, no delay is required.

Bi-Level Preheat

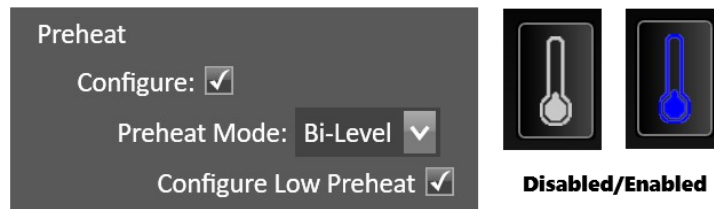
This option describes a preheat system that utilizes two preheat valves: a low preheat and a high preheat. Use the **Preheat Mode** drop down to select 'Bi-Level'. You will now have a **Configure Low Preheat** and **Configure High Preheat** checkbox. If neither option is selected, the low and high preheat valves are managed manually outside of the control software. However, you will



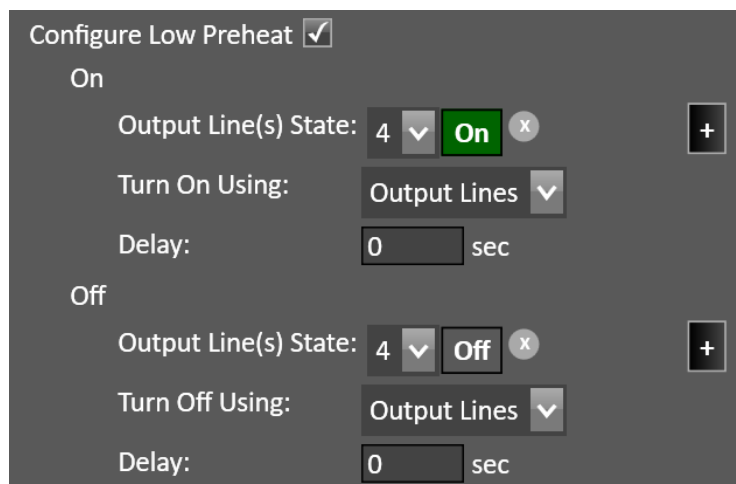
have access to the preheat parameters under the fab head's control panel in the CNC screen.

Configure Low Preheat

To begin, ensure that the **Configure Low Preheat** checkbox is selected. Once configured, this will provide a button in the CNC screen that enables and disables the Low Preheat valve.

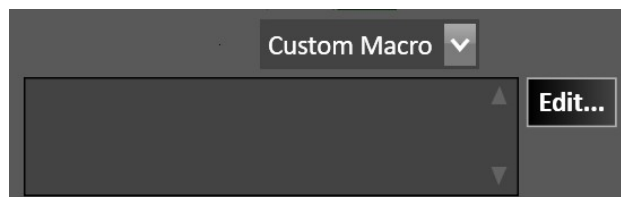


If **Configure Low Preheat** is checked, you'll need to describe what happens when the Low Preheat button is enabled and disabled in the CNC screen. In the example below, the Low Preheat button is toggled by Output 4, but other Outputs can be used. To accomplish this, ensure that the **Output Line(s) State** for **On** is set to '4' and 'On'. Then, set the **Output Line(s) State** for **Off** to '4' and 'Off'.



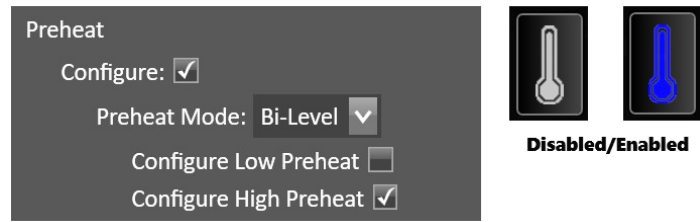
If multiple output lines are required, the '+' button will add Output lines to configure. For most Oxyfuel systems, **Turn On Using** and **Turn Off Using** should be set to 'Output Lines'.

For advanced/custom systems, you can use a 'Custom Macro'. This macro—a sequence of commands—must include at least one command that sets the **Output Line(s) State** appropriately. To configure the macro, you can enter the routine into the macro editor box of click the 'Edit...' button to launch a macro editor dialog box.

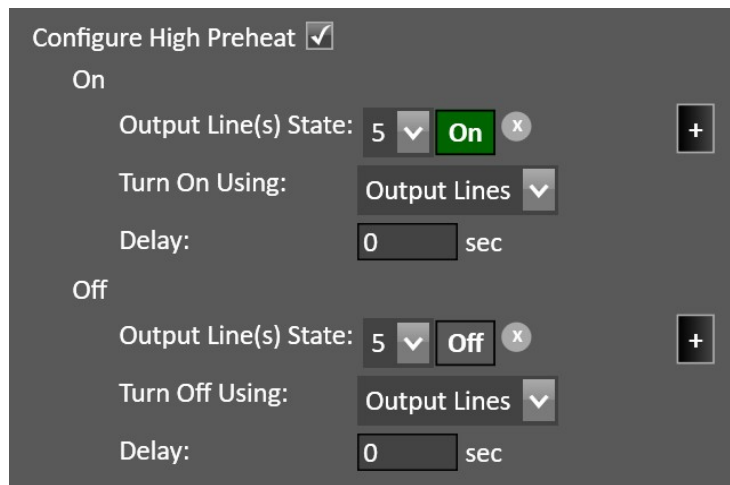


Configure High Preheat

To begin, ensure that the **Configure High Preheat** checkbox is selected. Once configured, this will provide a button in the CNC screen that enables and disables the High Preheat valve.

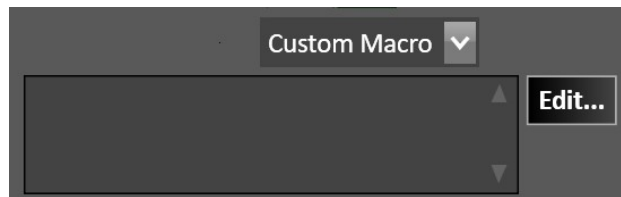


If **Configure High Preheat** is checked, you'll need to describe what happens when the High Preheat button is enabled and disabled in the CNC screen. In the example below, the High Preheat button is toggled by Output 5, but other Outputs can be used. To accomplish this, ensure that the **Output Line(s) State** for **On** is set to '5' and 'On'. Then, set the **Output Line(s) State** for **Off** to '5' and 'Off'.



If multiple output lines are required, the '+' button will add Output lines to configure. For most Oxyfuel systems, **Turn On Using** and **Turn Off Using** should be set to 'Output Lines'.

For advanced/custom systems, you can use a 'Custom Macro'. This macro—a sequence of commands—must include at least one command that sets the **Output Line(s) State** appropriately. To configure the macro, you can enter the routine into the macro editor box of click the 'Edit...' button to launch a macro editor dialog box.



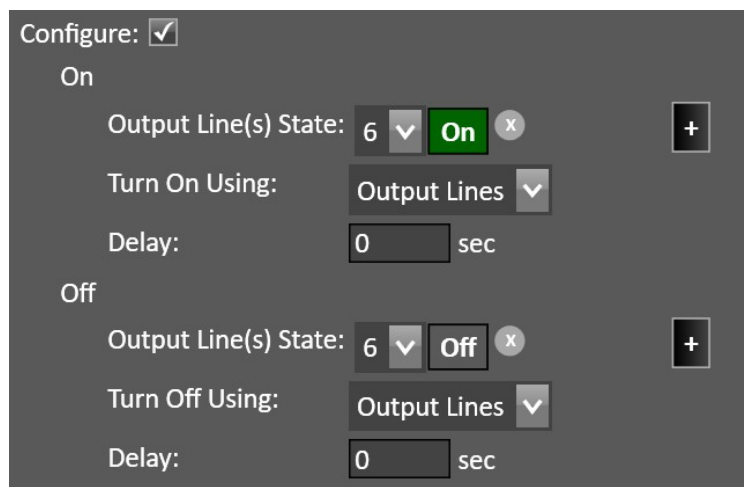
Water

This sub-section allows the software control over a water valve and is used to configure the behavior of the Water button. This is typically used to minimize plate warping.

To begin, select the **Configure** checkbox under **Water**. Once configured, this will provide a button in the CNC screen that enables and disables the water valve. If the configure box is left unchecked, control of the water valve is handled manually outside of the control software.

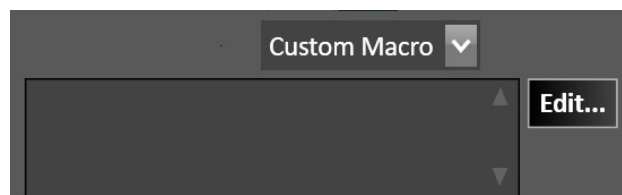


Next, you will need to describe what happens when the Water button is enabled and disabled in the CNC screen. In the example below, the Water button is toggled by Output 6, but other Outputs can be used. To accomplish this, ensure that the **Output Line(s) State** for **On** is set to '6' and 'On'. Then, set the **Output Line(s) State** for **Off** to '6' and 'Off'.



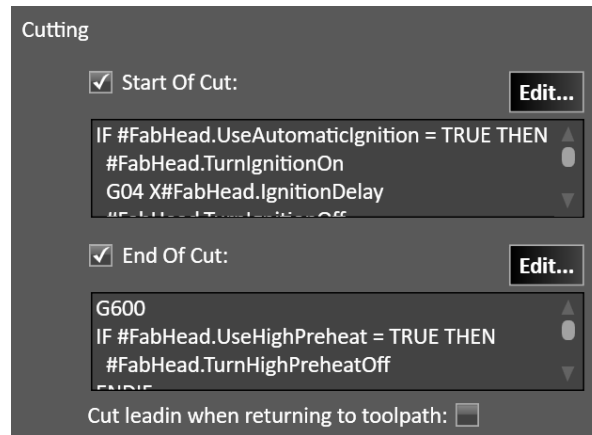
If multiple output lines are required, the '+' button will add Output lines to configure. For most Oxyfuel systems, **Turn On Using** and **Turn Off Using** should be set to 'Output Lines'.

For advanced/custom systems, you can use a 'Custom Macro'. This macro—a sequence of commands—must include at least one command that sets the **Output Line(s) State** appropriately. To configure the macro, you can enter the routine into the macro editor box of click the 'Edit...' button to launch a macro editor dialog box.



Cutting

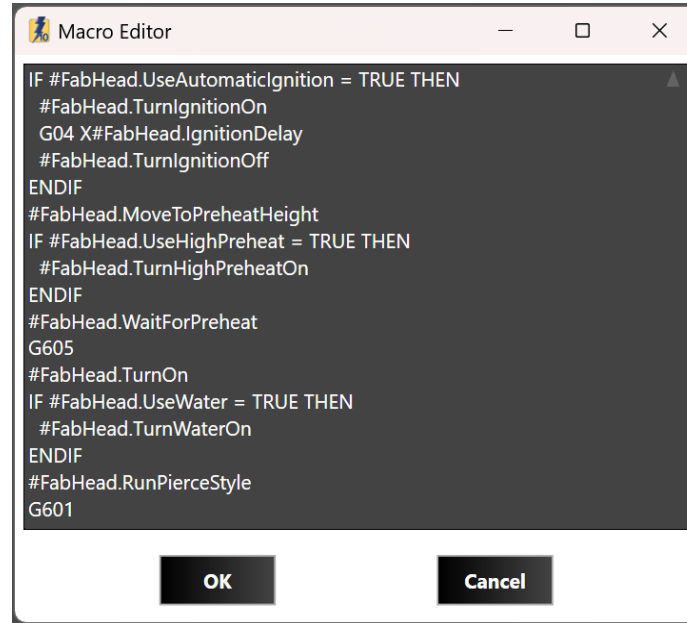
This sub-section will allow you to specify the commands that are executed at the start and end of each cut in a G-Code program.



Start Of Cut

This is used to describe the routine that is executed automatically at the beginning of any feedrate move. To configure the macro, ensure that the **Start Of Cut** checkbox is selected.

For most Oxyfuel systems, the default macro—set of commands—will work well. To configure the macro, you can enter the routine into the macro editor box or click the 'Edit...' button to launch a macro editor dialog box. The following is the default start of cut macro for an Oxyfuel fab head:



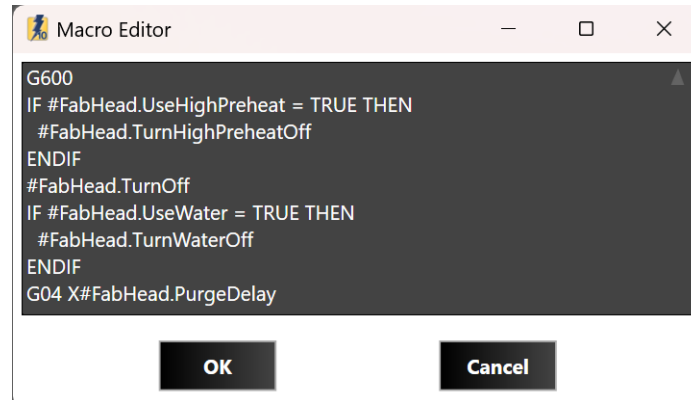
This includes 'IF' statements to account for configurable options such as Automatic Ignition, Preheat, and Water before executing a pierce.

If the checkbox for Start Of Cut is left unchecked, no macro will run at the start of cut process.

End Of Cut

This checkbox is used to describe the routine that is executed automatically at the beginning of any rapid move, typically at the end of cut. To configure the macro, ensure **End Of Cut** checkbox is selected.

For most Oxyfuel systems, the default macro—set of commands—will work well. To configure the macro, you can enter the routine into the macro editor box or click the *'Edit...'* button to launch a macro editor dialog box. The following is the default start of cut macro for an Oxyfuel fab head:



This includes turning the OxyFuel fab head off and 'IF' statements to account for configurable options such as Preheat, Water, and Purge Delay.

If the checkbox for End Of Cut is left unchecked, no macro will run at the end of cut process.

Cut Leadin When Returning to Toolpath

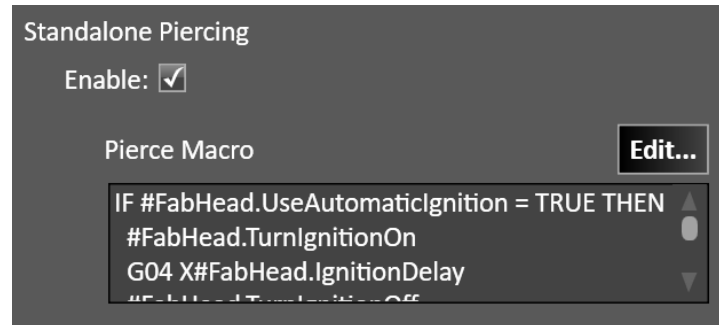
This feature is used to automatically cut a lead-in when a program is paused.

Cut leadin when returning to toolpath:

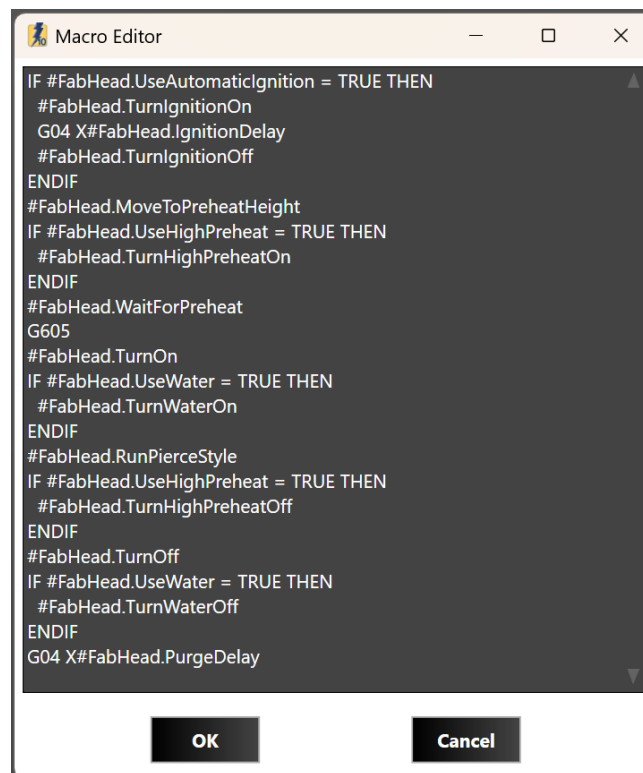
If this box is checked, the program can be paused (through a feed hold or optional pause) and the torch can be moved off the toolpath. When the program is resumed, the torch will begin cutting from the current position. This feature is helpful to prevent gouging the part in the event that cutting has to be interrupted. If the box is unchecked, the torch will instead rapid move to the point at which the program was paused and resume cutting.

Standalone Piercing

This checkbox is used to describe the routine that is executed automatically for a standalone (separate) pierce in the G-Code program. To configure the macro, ensure the **Enable** checkbox under **Standalone Piercing** is selected.



For most Oxyfuel systems, the default macro—set of commands—will work well. To configure the macro, you can enter the routine into the macro editor box or click the 'Edit...' button to launch a macro editor dialog box. The following is the default pierce macro for an Oxyfuel fab head:



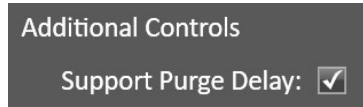
This includes 'IF' statements to account for configurable options like Preheat, Automatic Ignition, Water, and Purge Delay.

If the checkbox for **Standalone Piercing** is left unchecked, no macro will run for a standalone piercing process.

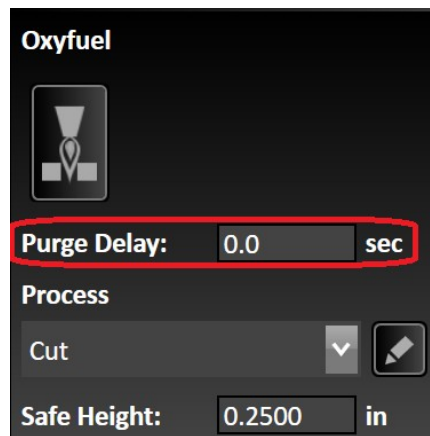
Additional Controls

Support Purge Delay

This section is used to add a delay that is applied after the cut oxygen valve is disabled to allow the gas to purge and prevent gouging the part before moving to the next cut. To add this delay, select the **Support Purge Delay** checkbox. If the Support Purge Delay checkbox is not selected, no delay will be available to configure.



If checked, this will provide a configurable parameter to the Oxyfuel fab head's control panel in the CNC screen.

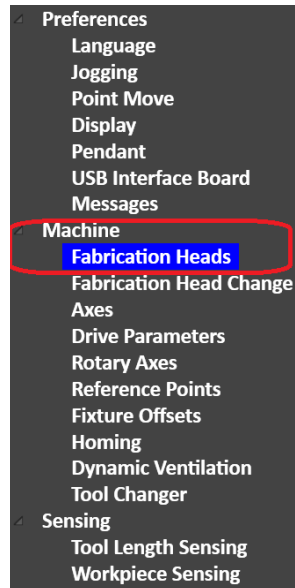


Waterjet

To get started, open the configuration window by clicking the 'Double Gear' icon found in the upper-right of the software screen:



Navigate to the "Machine...Fabrication Heads" page using the configuration page browser on the left side of the configuration window.



When adding a new Waterjet fab head (fabrication head), click the 'New Fab Head' button.

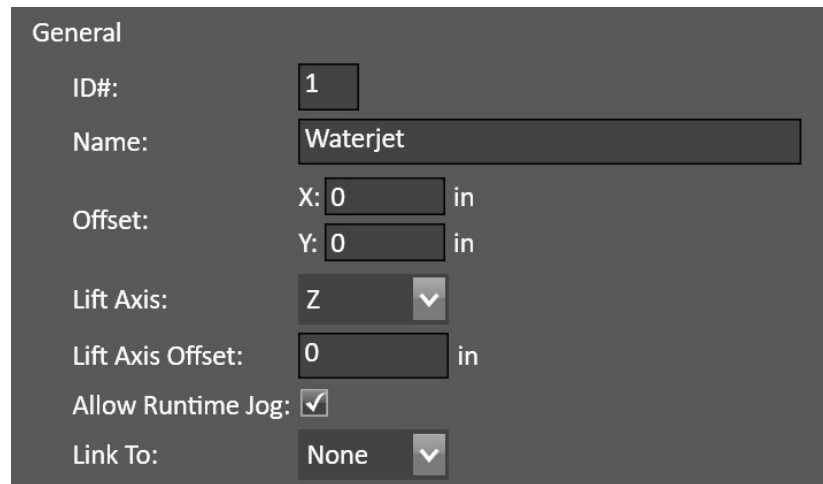


If editing an existing Waterjet fab head, select which one to edit from the list and click on it to expand additional configuration settings.

ID	Name	Model	Offset
waterjet1	Waterjet	Waterjet	X:0.0000 Y:0.0000
laserpointer1	Laser Pointer	Laser Pointer	X:0.0000 Y:0.0000

General

Next, establish the general settings for the Waterjet fab head.



General

ID#: 1

Name: Waterjet

Offset: X: 0 in
Y: 0 in

Lift Axis: Z

Lift Axis Offset: 0 in

Allow Runtime Jog:

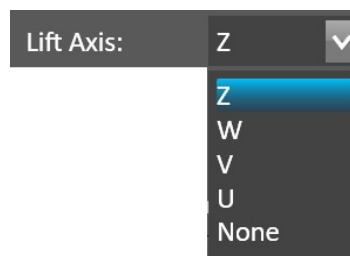
Link To: None

The **ID#** number is automatically generated, incrementing the value for each instance of the same fab head type. For example, if there are currently three Waterjet fab heads configured, adding another will set its ID# as 4. This number is used with the fab head type to create a unique identifier. However, it can be overwritten.

The **Name** will default to the fab head type (or Model). The name is used on the CAM and CNC screens to identify the fab head type. It can be edited, if desired.

When using more than one fab head, defining the **Offset** for **X and Y** will allow the software to transition from one fab head to the next. For example, an end user with an Waterjet cutting system might also add a laser pointer fab head. When configuring the laser pointer, they would need to enter in an offset (X, Y) in relation to the primary fab head. In this case, the Waterjet fab head is considered the primary, and its offset (X, Y) should be set as 0,0.

Next, use the **Lift Axis** drop down to select the relevant letter for the lift axis of the Waterjet fab head. This letter is also used to map the Drive Mechanism under the Drive Parameters section to ensure proper motion. This axis letter is moved using the hog controls on the CNC screen, much like the control for any X or Y positioning. Most commonly the lift axis will be assigned to 'Z', 'U', 'V', or 'W'.



Lift Axis: Z

- Z
- W
- V
- U
- None

A **Lift Axis** of 'None' indicates all positioning of the lifter is done outside the control software. This could include a torch which is positioned by hand, pneumatic lift, or even a motor-driven torch that does not use the control software to position.

The **Lift Axis Offset** is used to indicate the offset from one fab head to another, similar to the (X, Y) offset. This offset is necessary only if multiple fab heads share the same axis letter. This means that for most fab heads, the lift axis offset is 0.

Lift Axis Offset: in

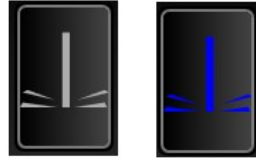
When the **Allow Runtime Jog** checkbox is selected, it will enable the lift axis to be moved using the jog controls while running a program. In order to allow runtime jog for a lift axis, there cannot be any motion commands for that axis in the program.

Lastly, the **Link To** drop down allows for another configured Waterjet to be linked for multiple simultaneous use. When selected, some configuration parameters which are linked to the primary Waterjet fab head will be disabled and only unique parameters will be available. Most Waterjet systems will only have the single fab head and **Link To** can left as 'None'.

Waterjet Control

This subsection is used to specify the basic On/Off controls for a Waterjet system.

To configure the **Waterjet Control**, describe what happens when the Waterjet button is enabled and disabled in the CNC screen.



Disabled/Enabled

In the example below, the Waterjet button is toggled by Output 1, but other Outputs can be used. To accomplish this, ensure that the **Output Line(s) State** for **Waterjet On** is set to '1' and 'On'. Then, set the **Output Line(s) State** for **Waterjet Off** to '1' and 'Off'.

Waterjet Control

Waterjet On: Output Line(s) State: 1

Turn On Using:

Delay: sec

Waterjet Off: Output Line(s) State: 1

Turn Off Using:

Delay: sec

Pump Pressure: psi

If multiple output lines are required, the '+' button will add Output lines to configure. For most Waterjet systems, **Turn On Using** and **Turn Off Using** should be set to 'Output Lines'.

For advanced/custom systems, use a 'Custom Macro'. This macro—a sequence of commands—must include at least one command that sets the **Output Line(s) State** appropriately. To configure the macro, enter the routine into the macro editor box or click the 'Edit...' button to launch a macro editor dialog box.

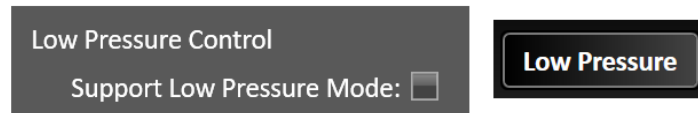
Configure an optional **Delay** after setting the Output line state. For the majority of configurations, no delay is required.

Lastly, the **Pump Pressure** line sets the water pressure at which water is exerted from the nozzle. This is set at 65000 psi by default.

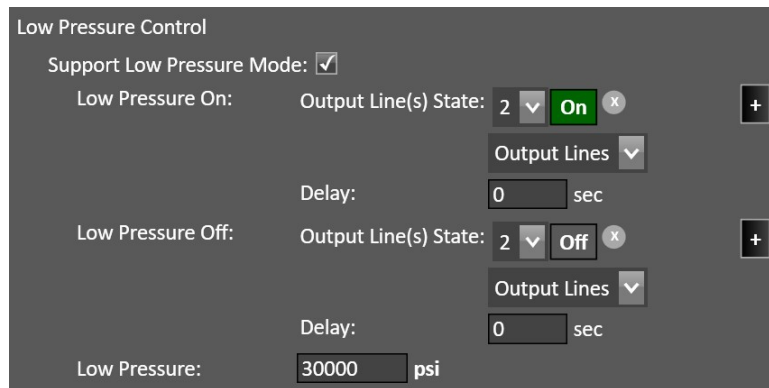
Low Pressure Control

This sub-section is used to configure a **Low Pressure Control** mode, which allows the Waterjet fab head to cut with a lower pressure for fragile materials.

To begin, select the **Support Low Pressure Mode** checkbox. Once configured, this will provide a button on the CNC screen to be used to enable and disable the Low Pressure Control. If this box is left unchecked, control of a low pressure mode is handled manually outside of the control software.

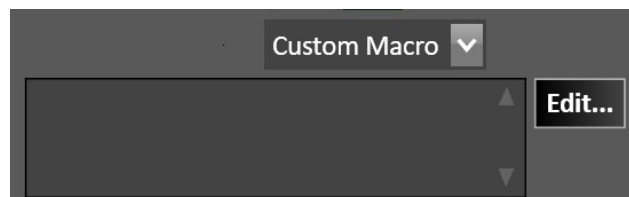


Next, describe what happens when the Low Pressure Control button is enabled and disabled in the CNC screen. In the example below, the Low Pressure Control button is toggled by Output 2, but other Outputs can be used. To accomplish this, ensure that the **Output Line(s) State for Low Pressure On** is set to '2' and 'On'. Then, set the **Output Line(s) State for Low Pressure Off** to '2' and 'Off'.



If multiple output lines are required, the '+' button will add Output lines to configure. For most Waterjet systems, **Turn On Using** and **Turn Off Using** should be set to 'Output Lines'.

For advanced/custom systems, use a 'Custom Macro'. This macro—a sequence of commands—must include at least one command that sets the **Output Line(s) State** appropriately. To configure the macro, enter the routine into the macro editor box or click the 'Edit...' button to launch a macro editor dialog box.



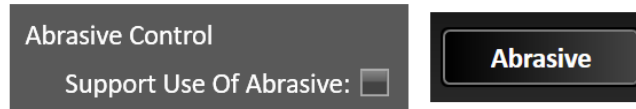
Configure an optional **Delay** after setting the Output line state. For the majority of configurations, no delay is required.

Lastly, specify the **Low Pressure** pump pressure for the Waterjet system, which will allow for the automatic lookup of cut-chart values.

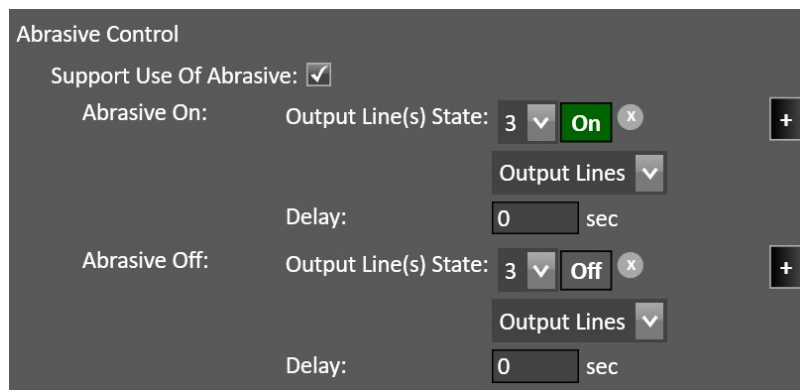
Abrasive Control

This sub-section is used to configure **Abrasive Control**. The use of an abrasive will allow the Waterjet to cut through a wider variety of thick materials.

To begin, select the **Support Use Of Abrasive** checkbox. Once configured, this will provide a button on the CNC screen to be used to enable and disable the Abrasive Control. If this box is left unchecked, Abrasive Control is handled manually outside of the control software.

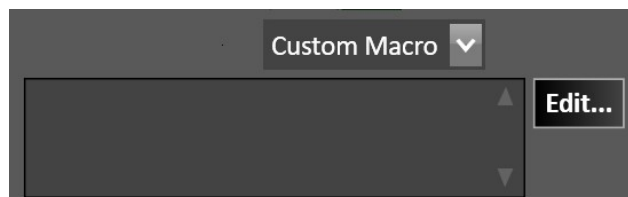


Next, describe what happens when the **Abrasive Control** button is enabled and disabled in the CNC screen. In the example below, the Abrasive Control is toggled by Output 3, but other Outputs can be used. To accomplish this, ensure that the **Output Line(s) State** for **Abrasive On** is set to '3' and 'On'. Then, set the **Output Line(s) State** for **Abrasive Off** to '3' and 'Off'.



If multiple output lines are required, the '+' button will add Output lines to configure. For most Waterjet systems, **Turn On Using** and **Turn Off Using** should be set to 'Output Lines'.

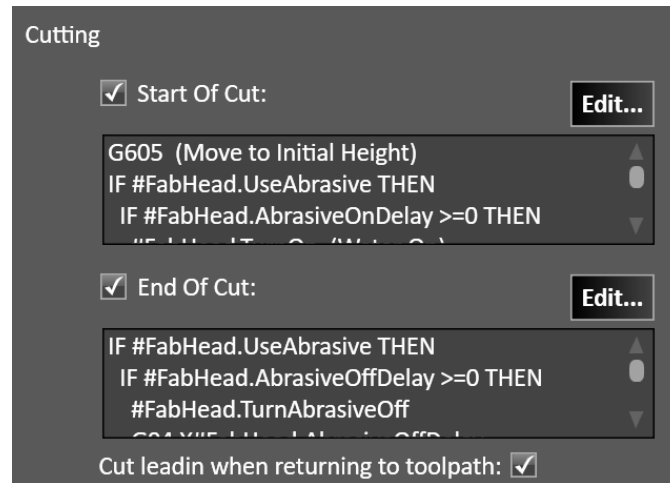
For advanced/custom systems, use a 'Custom Macro'. This macro—a sequence of commands—must include at least one command that sets the **Output Line(s) State** appropriately. To configure the macro, enter the routine into the macro editor box or click the 'Edit...' button to launch a macro editor dialog box.



Configure an optional **Delay** after setting the Output line state. For the majority of configurations, no delay is required.

Cutting

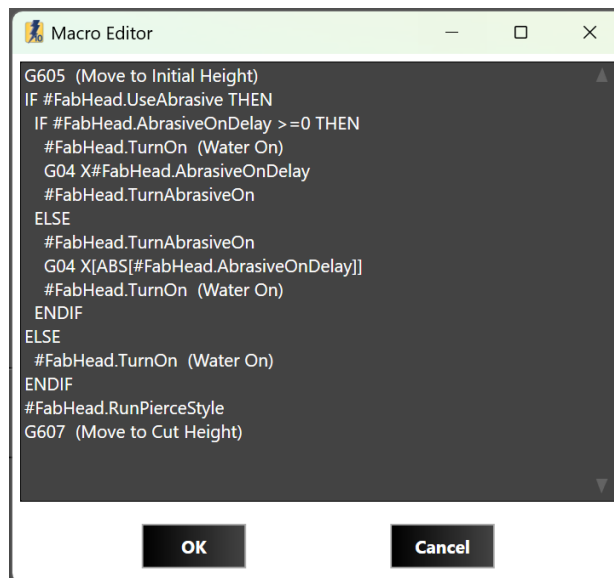
This subsection is used to specify the commands that are executed at the start and end of each cut in a G-Code program.



Start Of Cut

This is used to describe the routine that is executed automatically at the beginning of any feedrate move. To configure this macro, ensure that the **Start Of Cut** checkbox is selected.

For most Waterjet systems, the default macro—set of commands—will work well. To configure the macro, enter the routine into the macro editor box or click the 'Edit...' button to launch a macro editor dialog box. The following is the default start of cut macro for a Waterjet fabrication head:



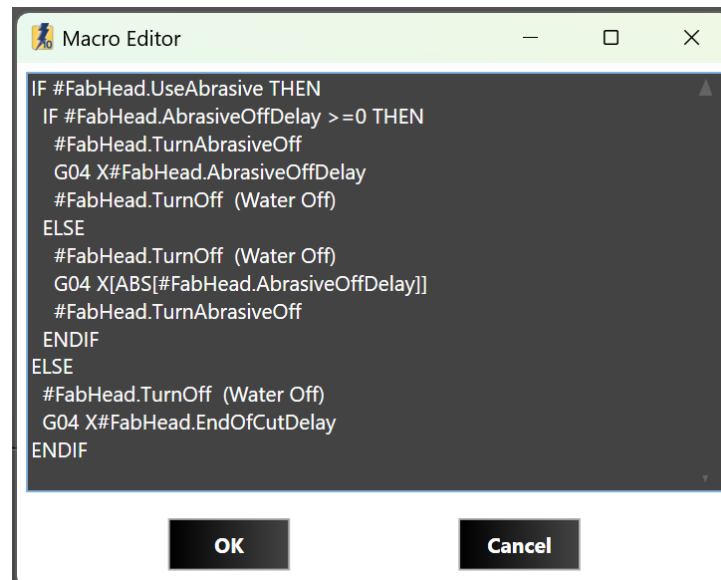
The includes 'IF' statements to account for Abrasive Control before executing a pierce.

If the checkbox for **Start Of Cut** is left unchecked, no macro will run at the start of cut process.

End Of Cut

This is used to describe the routine that is executed automatically at the beginning of any rapid move, typically at the end of a cut. To configure the macro, ensure the **End Of Cut** checkbox is selected.

For most Waterjet systems, the default macro—set of commands—will work well. To configure the macro, enter the routine into the macro editor box or click the 'Edit...' button to launch a macro editor dialog box. The following is the default end of cut macro for a Waterjet fabrication head.

The image shows a screenshot of a 'Macro Editor' dialog box. The dialog has a title bar with a small icon on the left and standard window controls (minimize, maximize, close) on the right. The main area is a dark text field containing the following G-code macro:

```
IF #FabHead.UseAbrasive THEN
IF #FabHead.AbrasiveOffDelay >=0 THEN
#FabHead.TurnAbrasiveOff
G04 X#FabHead.AbrasiveOffDelay
#FabHead.TurnOff (Water Off)
ELSE
#FabHead.TurnOff (Water Off)
G04 X[ABS[#FabHead.AbrasiveOffDelay]]
#FabHead.TurnAbrasiveOff
ENDIF
ELSE
#FabHead.TurnOff (Water Off)
G04 X#FabHead.EndOfCutDelay
ENDIF
```

At the bottom of the dialog, there are two buttons: 'OK' and 'Cancel'.

This includes 'IF' statements to account for Abrasive control and turning the Waterjet fab head off.

If the checkbox for End of Cut is left unchecked, no macro will run at the end of cut process.

Cut leadin when returning to toolpath

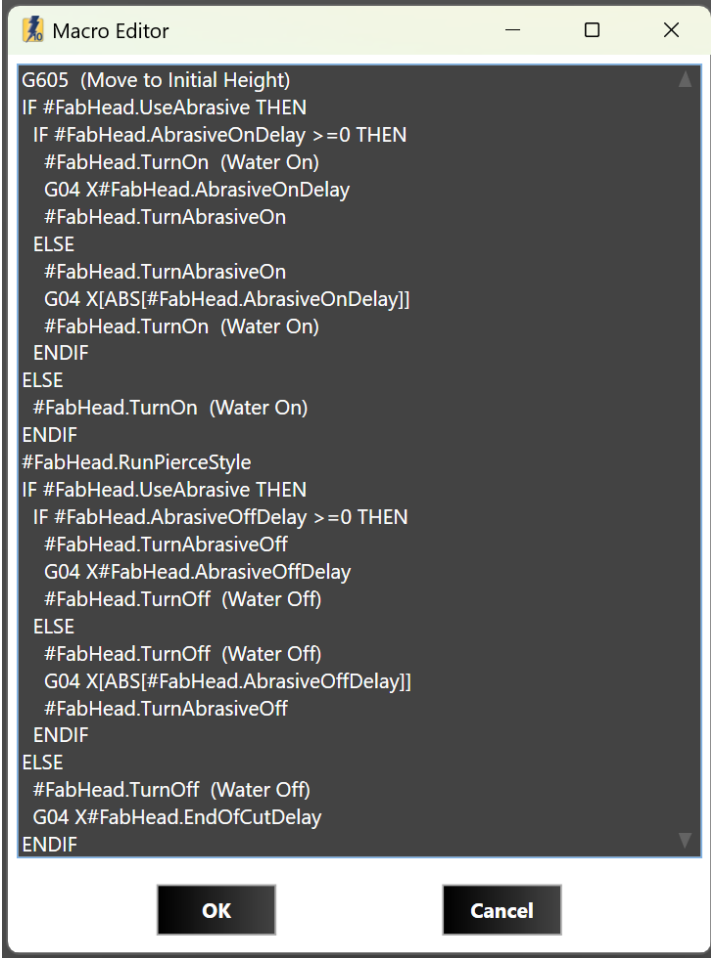
This feature is used to automatically cut a lead-in when a program is paused.

If this box is checked, the program can be paused (through a feed hold or optional pause) and the Waterjet can be moved off the toolpath. When the program is resumed, the Waterjet will begin cutting from the current position. This feature is helpful to prevent gouging the part in the event that cutting has to be interrupted. If the box is unchecked, the Waterjet will instead rapid move to the point point at which the program was paused and resume cutting.

Standalone Piercing

This checkbox is used to describe the routine that is executed automatically for a standalone (separate) pierce in a G-Code program. To configure the macro, ensure the **Enable** checkbox under the **Standalone Piercing** is selected.

For most Waterjet systems, the default macro—set of commands—will work well. To configure the macro, enter the routine into the macro editor box or click the 'Edit...' button to launch a macro editor dialog box. The following is the default pierce macro for a Waterjet fabrication head:



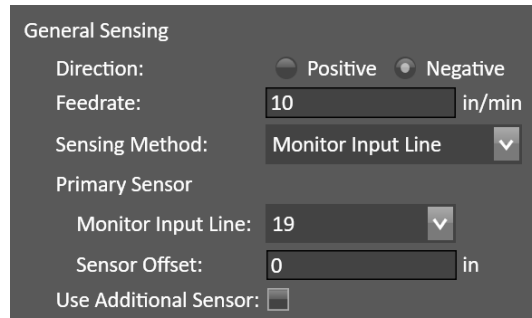
```
Macro Editor
G605 (Move to Initial Height)
IF #FabHead.UseAbrasive THEN
  IF #FabHead.AbrasiveOnDelay >=0 THEN
    #FabHead.TurnOn (Water On)
    G04 X#FabHead.AbrasiveOnDelay
    #FabHead.TurnAbrasiveOn
  ELSE
    #FabHead.TurnAbrasiveOn
    G04 X[ABS[#FabHead.AbrasiveOnDelay]]
    #FabHead.TurnOn (Water On)
  ENDIF
ELSE
  #FabHead.TurnOn (Water On)
ENDIF
#FabHead.RunPierceStyle
IF #FabHead.UseAbrasive THEN
  IF #FabHead.AbrasiveOffDelay >=0 THEN
    #FabHead.TurnAbrasiveOff
    G04 X#FabHead.AbrasiveOffDelay
    #FabHead.TurnOff (Water Off)
  ELSE
    #FabHead.TurnOff (Water Off)
    G04 X[ABS[#FabHead.AbrasiveOffDelay]]
    #FabHead.TurnAbrasiveOff
  ENDIF
ELSE
  #FabHead.TurnOff (Water Off)
  G04 X#FabHead.EndOfCutDelay
ENDIF
OK Cancel
```

Typically, this includes 'IF' statements to account for configurable options such as Abrasive Control as well as Waterjet fab head on/off controls.

If the checkbox for **Standalone Piercing** is left unchecked, no macro will run for a standalone piercing process.

General Sensing

This subsection is where the sensing parameters for the **Touch Off** and **Program Zero Sensing** process of the Waterjet fab head are specified.



General Sensing

Direction: Positive Negative

Feedrate: in/min

Sensing Method: ▼

Primary Sensor

Monitor Input Line: ▼

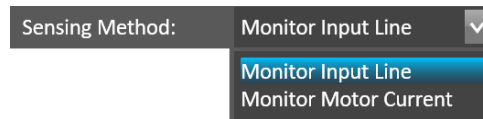
Sensor Offset: in

Use Additional Sensor:

First, select the **Direction** in which the fab head will sense the workpiece. This is typically negative.

Then define the **Feedrate**—or speed—at which the fab head will travel during the sensing process. Setting the value too high can result in inaccurate results or other errors. A typical value is 10 in/min. The sensing feedrate cannot exceed the **Start/Stop Feedrate** of the lift axis found in the **“Machine...Axes”** settings.

Next, use the **Sensing Method** drop down to select between *‘Monitor Input Line’* or *‘Monitor Motor Current’*. Traditional switches (mechanical, proximity sensors, etc) fall under the *‘Monitor Input Line’* section. *‘Monitor Motor Current’* will only be available to servo systems.

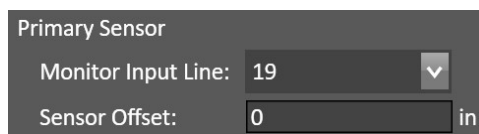


Sensing Method: ▼

Monitor Input Line

Monitor Motor Current

Select the input line number to be used as the primary sensor. This input line must also be configured as a control-type input in the **“I/O...Input Lines”** section of the configuration.



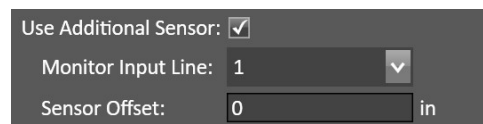
Primary Sensor

Monitor Input Line: ▼

Sensor Offset: in

The sensor offset is used to define a distance from when the sensor returns to a normal state after contacting the workpiece and point when the plasma torch lifts off the material. The value is typically negative.

If a second switch is being used as a backup, check the **Use Additional Sensor** box. As with the primary sensor, define an additional input line as well as the corresponding offset value.



Use Additional Sensor:

Monitor Input Line: ▼

Sensor Offset: in

The alternate sensing method, '*Monitor Motor Current*', is only available for a servo system.

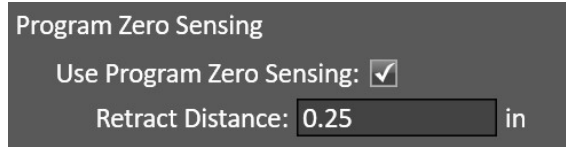
Sensing Method:	Monitor Motor Current	▼
Current Limit:	1.00	A
Current Debounce:	200	msec

Start by choosing a **Current Limit** that will indicate when the lift axis has encountered the material. By default, this is 1A. If the system is causing significant deflection in the material, this value needs to decrease. If the system instead is setting the zero before it reaches the material, this value should be increased.

The **Current Debounce** dictates the time that the motor current must exceed the limit. If the sensing is not consistent, this value should be increased. If the sensing is slow to react, decrease this value.

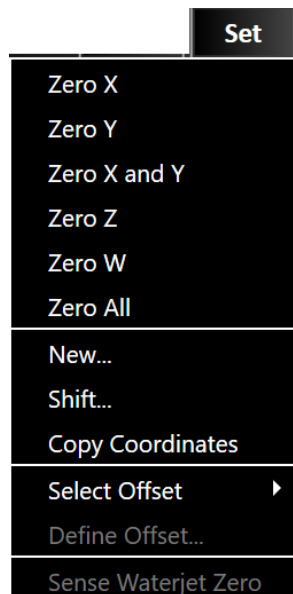
Program Zero Sensing

Once the General Sensing settings are configured, the sensing input line can now be used for Program Zero sensing. Select **Program Zero Sensing** checkbox to enable its use with the fab head's lift axis.



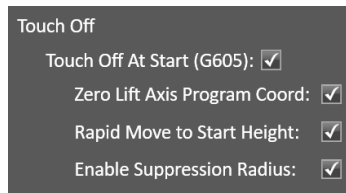
The entered value for **Retract Distance** is the position the lift axis will move to after sensing the work piece. By default, this value is a quarter inch.

When checked, an option to 'Sense Waterjet Zero' will appear in the **Set** drop down available in the CNC screen. This function will zero the Z program coordinate using the input line(s) configured under the general sensing section.



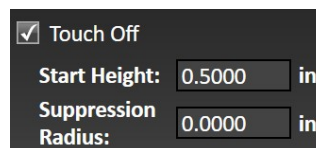
Touch Off

In order to sense the material prior to a cut, select the **Touch Off At Start (G605)** checkbox. This will add a parameter on the CNC screen to toggle **Touch Off** during a Move to Initial Height (G605) command. This command is commonly found in the Start of Cut, Start of Mark, and Standalone Piercing macros.



When **Zero Life Axis Program Coord** checkbox is selected, the lift axis's zero will be reset every time a touch is performed during a program. If this is unchecked, the lift axis's zero will not change during a Touch Off.

When **Rapid Move to Start Height** is checked, it adds the **Start Height** parameter below the **Touch Off** checkbox in the **CNC screen**. The system will make a rapid move from the Safe Height to the Start Height before performing a touch off. This can decrease sensing time while allowing larger clearance to reduce the risk of contacting a tip up.



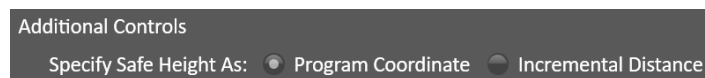
When **Enable Suppression Radius** is checked, the system will skip touch offs within a specified distance of the previous touch off. This feature helps with efficiency by reducing the number of touch off sequences.

Additional Controls

Safe Height is a parameter on the CNC screen that defines the height the lift axis will travel to at the start of the program and between cuts. It is important to note that the system will only move to the safe height during this process and no motion will occur if the lift axis is already above the safe height.



Use the **Specify Safe Height As** buttons to define the safe height parameter as a '*Program Coordinate*' or an '*Incremental Distance*'.



For example, if the safe height is 0.25 inches and it is listed as a '*Program Coordinate*', the lift axis will rise to a coordinate of 0.25 inches when a move to safe height is required. Instead, if it is set as an '*Incremental Distance*', the lift axis will move 0.25 inches from its current height.

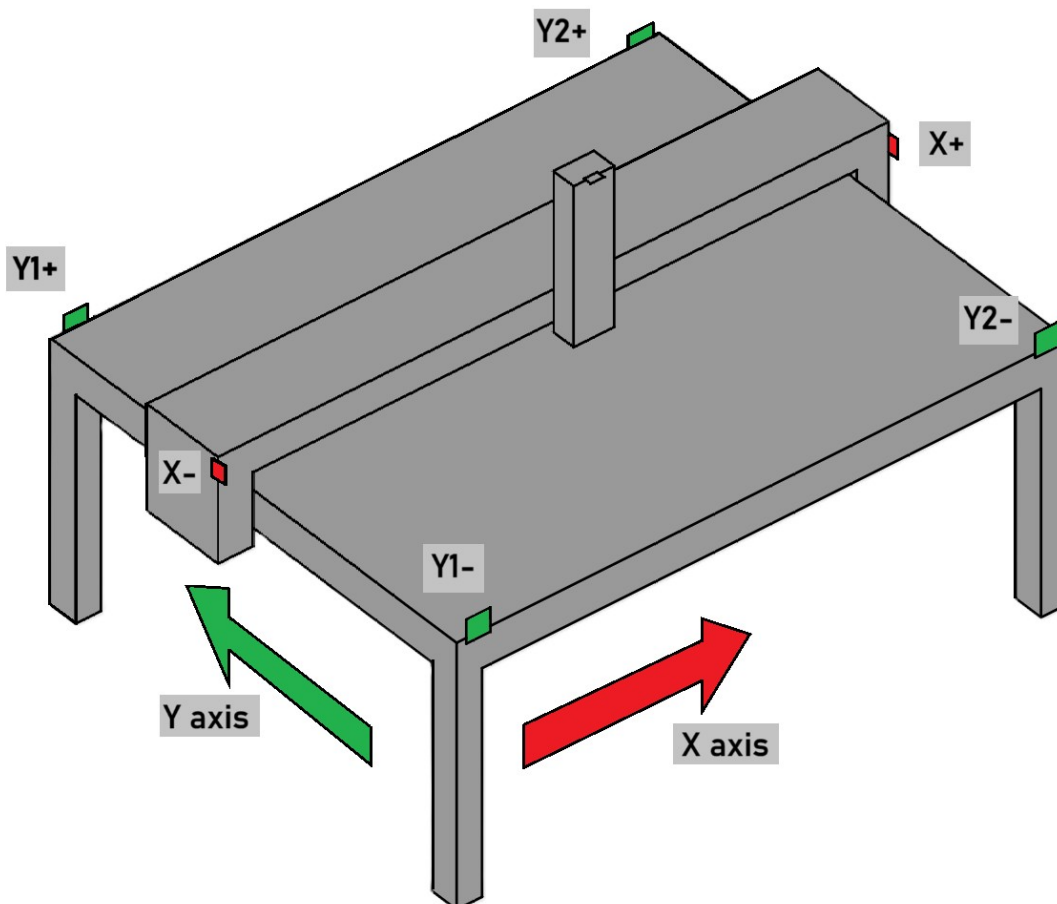
Inputs Continued...

Homing and Limits

The homing function is used to establish the physical limit of travel for the machine, also known as the machine envelope. It also allows the system to check for positioning errors and to re-zero any axis enabled for homing. This function requires the use of mechanical hardware, such as mini-switches or proximity sensors, and a configured input line to monitor the corresponding input trip.

Similarly, limits are used to alert the user when a movement would exceed the physical limit of an axis. Much like the homing inputs, they will require mechanical hardware and a configured input line.

To begin, determine the axis, position, and input line that the switch is wired to. Use the example below to help you visualize the location of each of your switches.



Input Configuration

The next step will involve configuring the inputs to match the location of the switches. Navigate to the **I/O>>Inputs** section of the configuration.

Assign the axis, position, and (if applicable) sub-axis to each input in relation to each switch’s physical location and wiring.

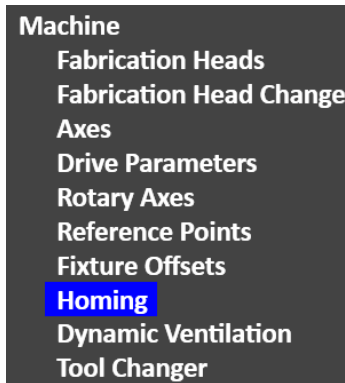
On every axis you intend to home, decide which position you will home to. Set the function of these inputs as ‘Home/Limit’. The rest should be set as ‘Limit’.

Using the figure in the previous page, a table that homes to the negative end of travel for X and Y should be configured as follows:

Line #	Description	Function	Axis	Sub Axis	Position	Sensing Debounce	Wiring
1	X- Home/Limit (All Subaxes)	Home/Limit	X	N/A	Negative	0.001	<input type="radio"/> N.O. <input checked="" type="radio"/> N.C.
2	Y+ Limit (All Subaxes)	Limit	Y	N/A	Positive	0.001	<input type="radio"/> N.O. <input checked="" type="radio"/> N.C.
3	X+ Limit (All Subaxes)	Limit	X	N/A	Positive	0.001	<input type="radio"/> N.O. <input checked="" type="radio"/> N.C.
4	Y- Home/Limit (All Subaxes)	Home/Limit	Y	N/A	Negative	0.001	<input type="radio"/> N.O. <input checked="" type="radio"/> N.C.

Set Up Homing

Navigate to the “**Machine... Homing**” section.



Once there, configure the ‘**Axis Settings**’.

Axis	Home Axis	Home End	Home Order	Home Switch Offset	Homing Rate	Homing Tolerance
X	<input checked="" type="checkbox"/>	<input type="radio"/> Positive <input checked="" type="radio"/> Negative	3	0.5 in	40 in/min	0 in
Y	<input checked="" type="checkbox"/>	<input type="radio"/> Positive <input checked="" type="radio"/> Negative	2	0.5 in	40 in/min	0 in
Z	<input checked="" type="checkbox"/>	<input checked="" type="radio"/> Positive <input type="radio"/> Negative	1	0.1 in	10 in/min	0 in

Start by checking the **Home Axis** box for each axis that has a *'Home/Limit'* input line configured in the **"I/O... Input Lines"** section.

The **Home End** buttons toggle the end that the program will home to and should match the position of the homing switch.

Use the drop downs in the **Home Order** column to select the order in which you wish the axes to seek home. We recommend that the order is set to prevent the tool through the material. This means homing the lift axis—Z in the example above—first, followed by X and Y.

In the **Home Switch Offset** column, define the distance the axis will retract after triggering the switch. This is used to create a buffer space between the machine's maximum travel and the switch.

The **Homing Rate** is the speed at which the axis will seek the home switch. Although this is dependent on the characteristics of the machine, 10 in/min is usually a safe starting point. We do not recommend exceeding the **Start/Stop Feedrate** for a particular axis, found in the **"Machine... Axes"** section, in order to ensure an accurate and consistent home location. If you are finding homing position discrepancies, try slowing down the **Homing Rate**.

The value entered under the **Homing Tolerance** column is the parameter that **'Homing Initiated by User'** and **'Homing Initiated While Running G-Code'** subsections reference for displaying homing results.

As mentioned above, **'Homing Initiated by User'** and **'Homing Initiated While Running G-Code'** allow you to toggle the preferences for displaying the homing results after a re-home.

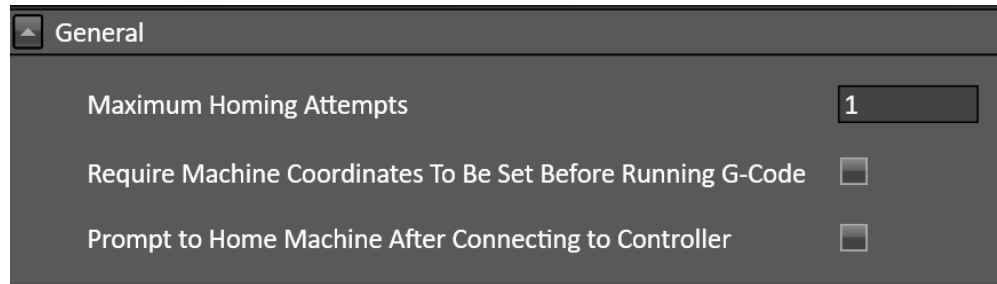
'Always' will show the results after every re-home, even if the discrepancy from the previous Machine Zero location is within the **Homing Tolerance**.

If *'When Out of Tolerance'* is selected, the system will display the results of re-homing when the discrepancy from the previous Machine Zero location is outside the **Homing Tolerance**. Additionally, the checkbox *'Reset Machine Zero if in Tolerance'* will re-zero the machine coordinates when enabled.

The *'Never (Always Resets Machine Zero)'* will not display the results of re-homing, even if the discrepancy from the previous Machine Zero location is outside the Homing Tolerance for any axis. Machine Zero will always be reset to the new location.

Homing Initiated by User	Homing Initiated While Running G-Code
Display Homing Results <input type="radio"/> Always <input checked="" type="radio"/> When Out of Tolerance <input checked="" type="checkbox"/> Reset Machine Zero if in Tolerance <input type="radio"/> Never (Always Resets Machine Zero)	Display Homing Results <input type="radio"/> Always <input checked="" type="radio"/> When Out of Tolerance <input checked="" type="checkbox"/> Reset Machine Zero if in Tolerance <input type="radio"/> Never (Always Resets Machine Zero)

Under the **'General'** subsection, you will want to dictate the **Maximum Homing Attempts**. By default this is one (1).



General	
Maximum Homing Attempts	<input type="text" value="1"/>
Require Machine Coordinates To Be Set Before Running G-Code	<input type="checkbox"/>
Prompt to Home Machine After Connecting to Controller	<input type="checkbox"/>

Following the importance of machine coordinates, the system can *'Require Machine Coordinates To Be Set Before Running G-Code'* with the appropriately labeled checkbox.

Additionally, the system can *'Prompt to Home Machine After Connection to Controller'* in the appropriately labeled checkbox. This will give you an additional dialog box after connecting to the controller that will home your machine.

Safety Features

As an Input

For all systems, an input line can be utilized to wire in an emergency stop (E-Stop). This includes collision-detection systems.

After identifying which input line the E-Stop will be wired to, navigate to the **“I/O... Input Lines”** section and set the **Function** to ‘Safety’ in the corresponding **Line #**. When tripped, the system will stop all motion until the safety fault is cleared.

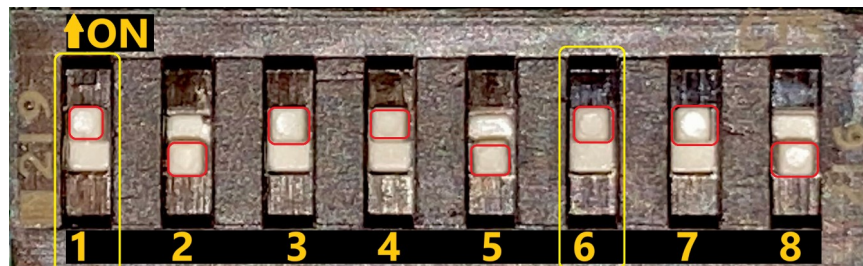
Line #	Description	Function	Axis	Sub Axis	Position	Sensing Debounce	Wiring
1	E-Stop	Safety ▾	N/A ▾	N/A ▾	N/A ▾	0.001	<input type="radio"/> N.O. <input checked="" type="radio"/> N.C.

As an Accessory

A controller that has an advanced power board also comes equipped with a 10-pin accessory port on the back panel. An E-Stop can be installed as a normally closed circuit across pins 1 and 2 of the accessory port. Additionally, this system can be set to wait for a reset signal after executing an E-Stop. This reset switch should be a normally open, momentary switch wired to pins 9 and 10 of the 10-pin accessory port.



To enable the E-Stop function, ensure that switch 1 on the advanced power board is set to the ‘ON’ position. Similarly, ensure that switch 6 is set to the ‘ON’ position to wait for a reset signal after executing an E-Stop.



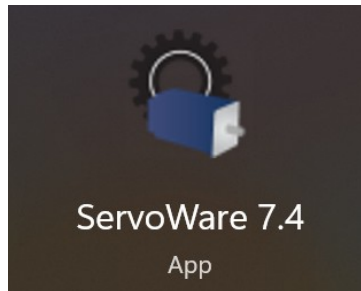
By default, this E-Stop signal is hard-wired for Input 24. Navigate to **“I/O... Input Lines”** section and set the **Function** to *‘Safety’* in *Line# 24*. When tripped, the system will stop all motion until the safety fault is cleared.

Line #	Description	Function	Axis	Sub Axis	Position	Sensing Debounce	Wiring
24	E-Stop	Safety ▼	N/A ▼	N/A ▼	N/A ▼	0.001	<input type="radio"/> N.O. <input checked="" type="radio"/> N.C.

Servo Communications

ServoWare

A Pro Series controller that includes servo drives will require the use of ServoWare software for communication with the drives. ServoWare will allow you to adjust the tuning on a drive in addition to help you identify a fault in the drive.



Note: Adjustments to the preinstalled tuning file should not be done without the help of a FlashCut technician.

Hardware

The hardware, shipped alongside a servo system, includes a USB-to-Serial adapter and a DB-9 Male to Female Serial Extension Cable. The adapter will interface between your computer and the extension cable. The extension cable will connect to the COM port on the back panel of the controller.



DB-9 Serial Extension Cable



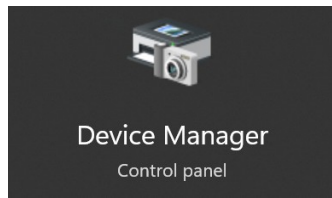
USB-to-Serial Adapter



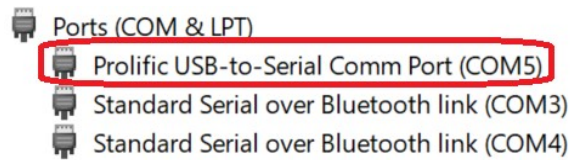
Note: In a Titanium series, this software and the adapter are preinstalled.

COM Port

To identify the correct port to use, open the **Device Manager** program for **Windows**.

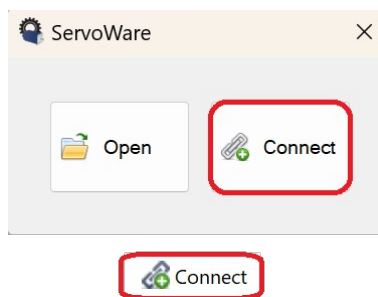


Find and expand the 'Ports (COM & LPT)' category. Identify the device and make a note of its COM port.

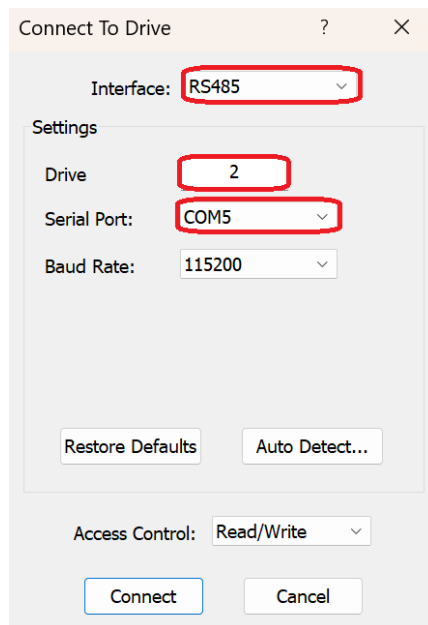


Connecting to Drives

1. To interact with a drive, open the ServoWare software and click the 'Connect' button.

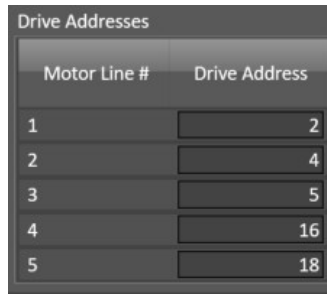


2. This will display the following **Connect To Drive** screen:



3. Set the **Interface** to 'RS485'.

- The **Drive** address will correspond to the following order.



Motor Line #	Drive Address
1	2
2	4
3	5
4	16
5	18

- Serial Port** should match the 'COM#' that you identified in Device Manager.

