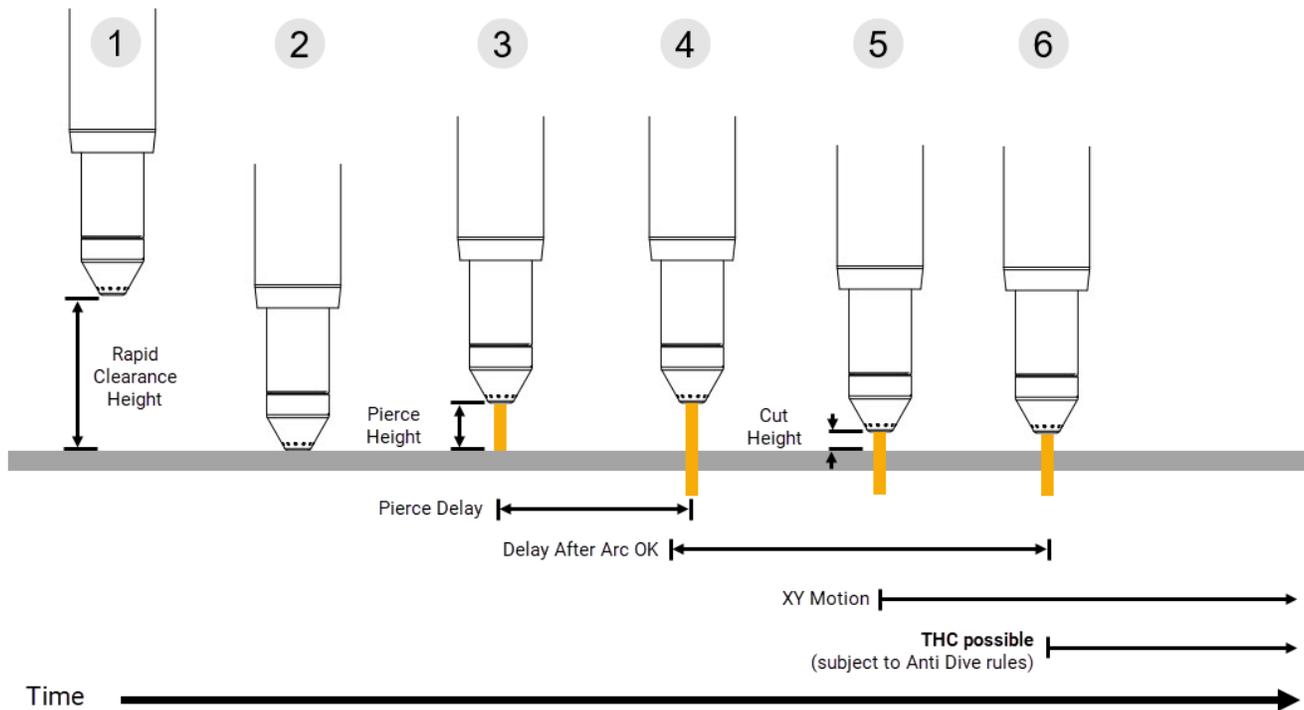


Avid CNC Plasma Software User's Guide - 2.4.1 Beta

| | |
|--|----------|
| General Avid CNC Plasma Information | 2 |
| Plasma Process Overview | 2 |
| Ohmic Probe Function | 3 |
| Software User's Guide | 4 |
| SheetCam with Avid CNC Post and Tools | 4 |
| Choosing the Correct Installer | 4 |
| Post Processor Settings | 4 |
| Toolsets | 5 |
| Drawing Import | 7 |
| Operation Setup | 8 |
| Mach4 - Feed Rate Override | 10 |
| Mach4 Plasma Screen Overview | 11 |
| Help Pop-Ups | 11 |
| A: Plasma Settings group | 11 |
| Torch Settings | 12 |
| THC and Logging | 13 |
| THC Settings | 14 |
| B: Plasma Monitors group | 15 |
| Typical Program Workflow | 16 |
| Load Gcode | 16 |
| Set Z Rapid Height | 16 |
| Check Consumable State | 17 |
| Check THC state | 17 |
| Verify THC Settings | 17 |
| Torch Height Control (THC) | 18 |
| Operating Theory | 18 |
| Avid CNC Mach4 THC Tuning | 19 |
| Material Profile | 19 |
| Fine Tuning - Up and Down Sensitivity | 20 |
| Additional THC Settings | 21 |
| Velocity Anti-Dive Thresholds | 22 |
| Voltage Anti-Dive Thresholds | 23 |
| Advanced Manual THC Anti-Dive Settings | 25 |
| Getting Help | 27 |
| Recording a THC Log | 27 |
| Recording a Mach4 Log | 27 |

General Avid CNC Plasma Information

Plasma Process Overview



1. Safe Height / Retract Height

- At **rapid clearance height** the machine moves to the XY pierce position and begins the material touch-off.

2. Material Touch-Off

- The torch moves down until the torch tip makes contact with the material and the 'Ohmic Probe' signal reads 'Active' in Mach4.
- The Z height is set to Z = 0.

3. Pierce Start

- The torch is moved up to the programmed **Pierce Height**.
- The torch fires. This starts the programmed **Pierce Delay** time to allow the arc to fully pierce the material.

4. Pierce Complete

- When the arc transfers to the material, the torch will send an 'Arc Okay' signal triggering the **Delay After Arc OK** which inhibits Torch Height Control (THC) motion.
- After the **Pierce Delay** the arc has fully pierced the material and cutting motion can begin.

5. Cut Start

- The torch is moved to the programmed **Cut Height**.

- XY Motion starts.

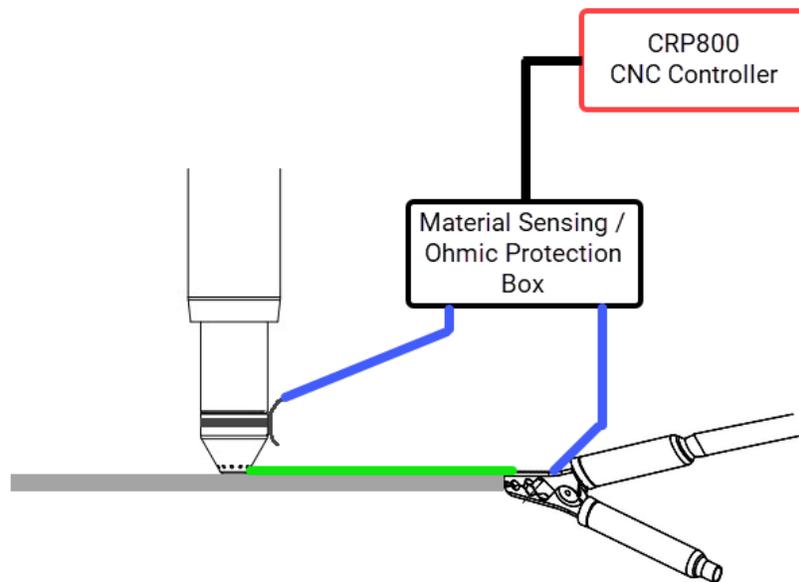
6. Main Cut

- After the **Delay After Arc OK** time, THC may be active **IF** enabled by the g-code **AND** the Anti Dive rules allow it.

Ohmic Probe Function

The Ohmic Probe senses the top surface of the material in order to set a local Z axis reference for pierce and cut heights. The circuit sends the Ohmic Probe Active signal to Mach4 when an electrically conductive path between the Ohmic Ring (on the torch) and the ground wire (on the material clamp) is completed.

Note: the Ohmic Probe system is NOT involved in Torch Height control. Its only purpose is to set the initial pierce and cut heights.



Software User's Guide

SheetCam with Avid CNC Post and Tools

Gcode for Avid CNC Mach4 version 2.4.0+ should be created using version 1.9.0+ of the Avid CNC Mach4 post processor and tools for SheetCam.

Choosing the Correct Installer

The correct installer depends on your **torch** model.

For Hypertherm Powermax torches with **SYNC** torches, choose the “Avid CNC Post and Powermax **SYNC** Tools for SheetCam” installer.

For Hypertherm Powermax torches with **Duramax** torches, choose the “Avid CNC Post and Powermax Tools for SheetCam” installer.

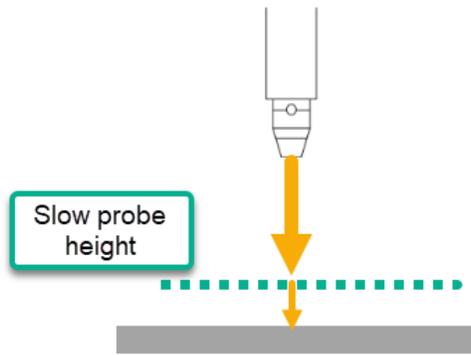
Note: this includes tools for Duramax consumables and SYNC consumables with adapter.

Post Processor Settings

These settings are all found in **Options -> Machine -> Post Processor -> Set custom post options.**

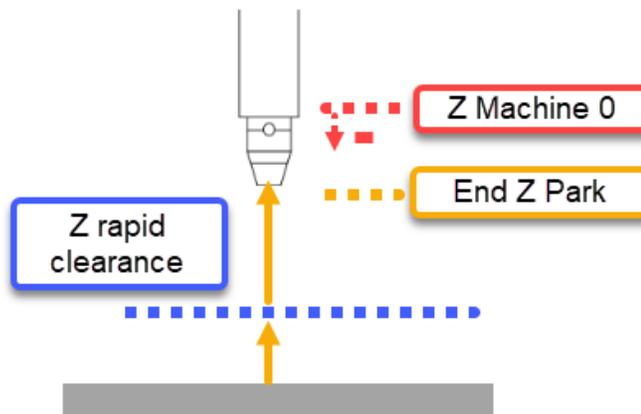
| Setting | Value | Number |
|--|------------|--------|
| Distance between references | 0 in | 1 |
| Reference feed rate | 19.685 ipm | 2 |
| Slow Probe Height | 0.25 in | 3 |
| Touchoff, 1 = Mechanical 0 = Ohmic | 0 | 4 |
| End program Z park (machine coordinates) | -0.5 in | 5 |

1. **Distance between references:** this sets a radius around the most recent probe where subsequent pierces will start without a probe. Use this to save run time on flat, stable (usually thick) material that will not have a significant height difference between pierce locations.
2. **Reference feed rate:** the feedrate during the probe portion of a move. Increasing this from the default will reduce run time but also reduce the accuracy of the Z positioning.
3. **New: Slow Probe Height:** this sets a height above the *last* material Z position that the torch will rapid down to before starting the slow probe move.



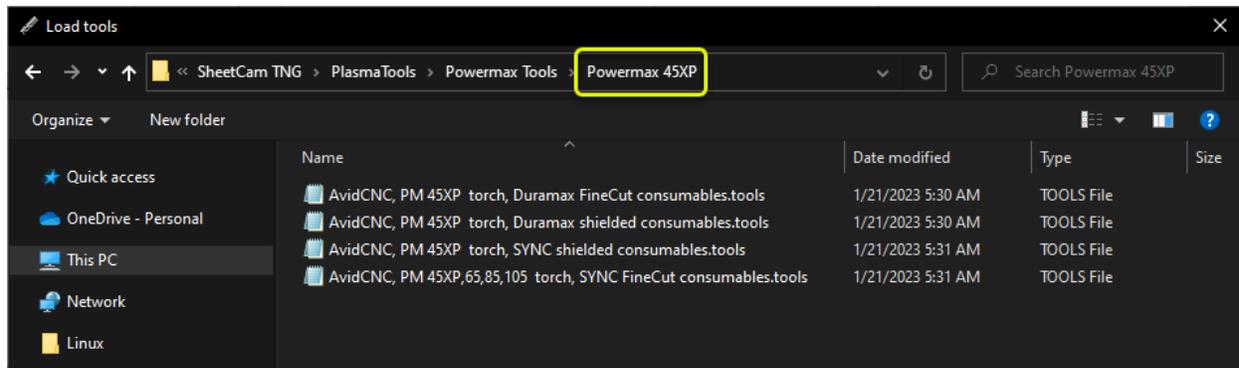
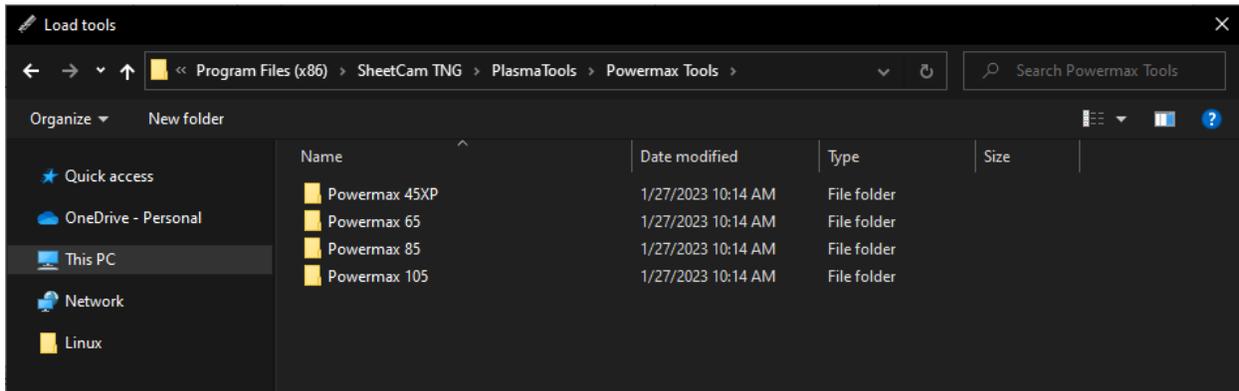
The first probe in a program will always be full height at slow speed to initially find the material. Using the Slow Probe Height greatly reduces the run time of programs with many pierces.

4. **Touchoff type:** sets Ohmic or Mechanical touchoff. Ohmic is recommended.
Note: using Mechanical touchoff requires opening the Ohmic Protection Box and changing a jumper position.
5. **New: End Program Z Height:** sets a machine coordinate location to park the Z after a program. This is used to drive the Z up and reduce water splashing at the end of a cut.

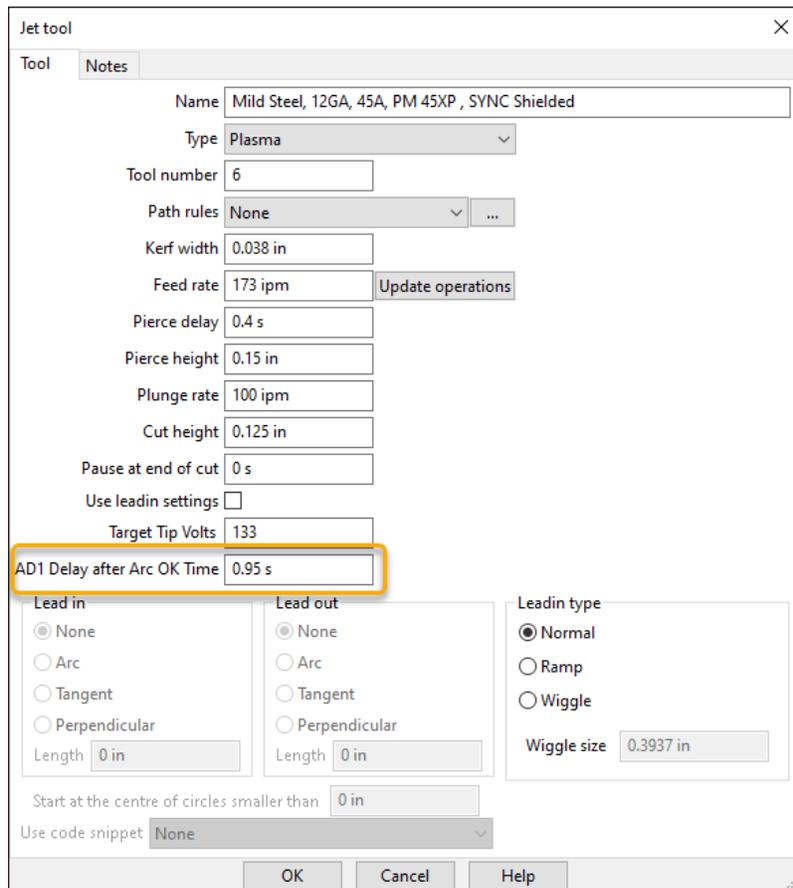


Toolsets

The toolsets are organized by **Torch** and **Consumable**. Each toolset includes a range of materials and English/Metric thicknesses. It is very important to choose the correct toolset for your torch and consumable.



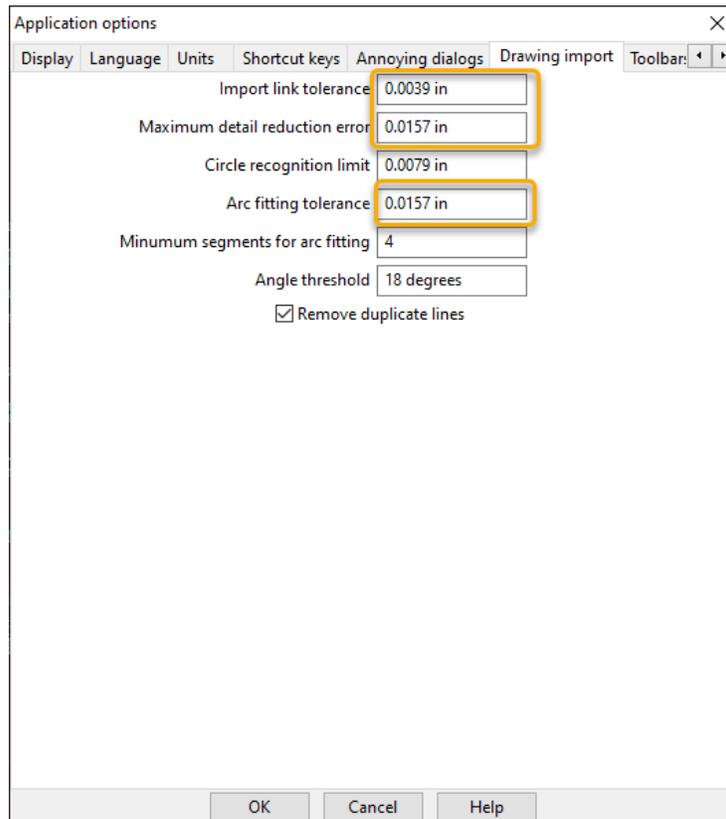
Within each tool are pre-populated values for all parameters that are tool-specific. A new addition is a pre-populated **AD1 Delay after Arc OK Time**.



This is an Anti Dive setting that inhibits THC motion until X seconds after the arc transfers to the material. This is a period of very unstable voltage and the duration is related to other material/tool properties.

Drawing Import

There are settings in **Options -> Application Options -> Drawing Import** that SheetCam uses to smooth and simplify vectors when drawings are imported. More detail on the settings is available in **Help -> Help Window -> Menu Items -> Options Menu**.



The Avid CNC Post and Tools installer changes some of the settings from default to better match the capabilities and needs of plasma cutting. Plasma is a relatively rough cutting method that cannot achieve very fine tolerances. Smoothing the imported vectors will result in smaller file size ($\frac{1}{3}$ - $\frac{1}{2}$ reduction in gcode lines), smoother motion. Smoothing will result in small geometry differences but the differences are smaller than the normal variation in kerf width throughout a cut and the final products of these two files will be indistinguishable.

| Setting | SheetCam default | Avid default |
|----------------------------|------------------|-----------------|
| Import link tolerance | 0.0004" (0.01mm) | 0.0039" (0.1mm) |
| Max detail reduction error | 0.0008" (0.02mm) | 0.0157" (0.4mm) |
| Arc fitting tolerance | 0.0039" (0.1mm) | 0.0157" (0.4mm) |

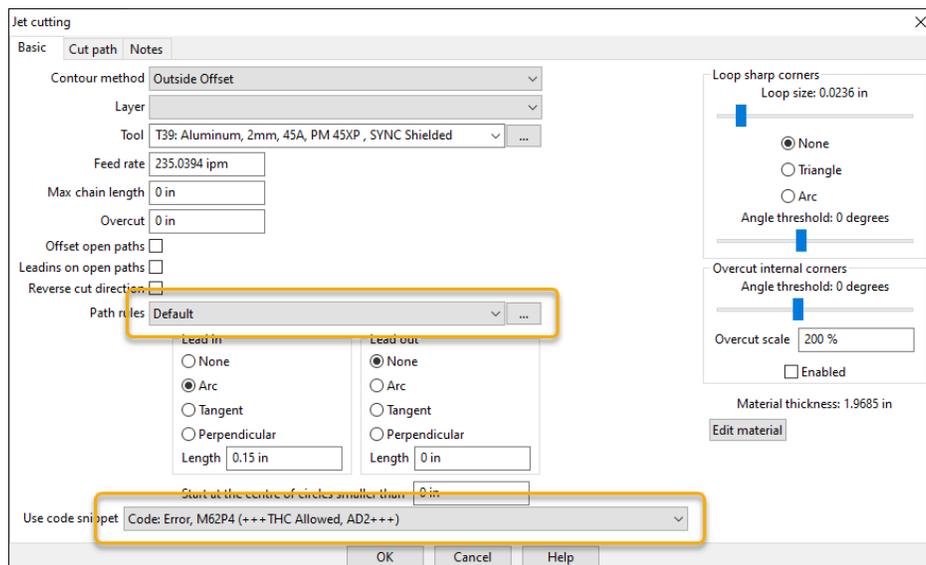
Below is an example of SheetCam defaults (left) and Avid CNC defaults (right).



Operation Setup

When a new Jet Cutting operation is created, the Path Rules should be set to Default and the Code Snippet should be set to “M62P4 (+++THC Allowed, AD2+++)”.

Note: the code snippet may say “Error, M62P4...” initially. This is OK.

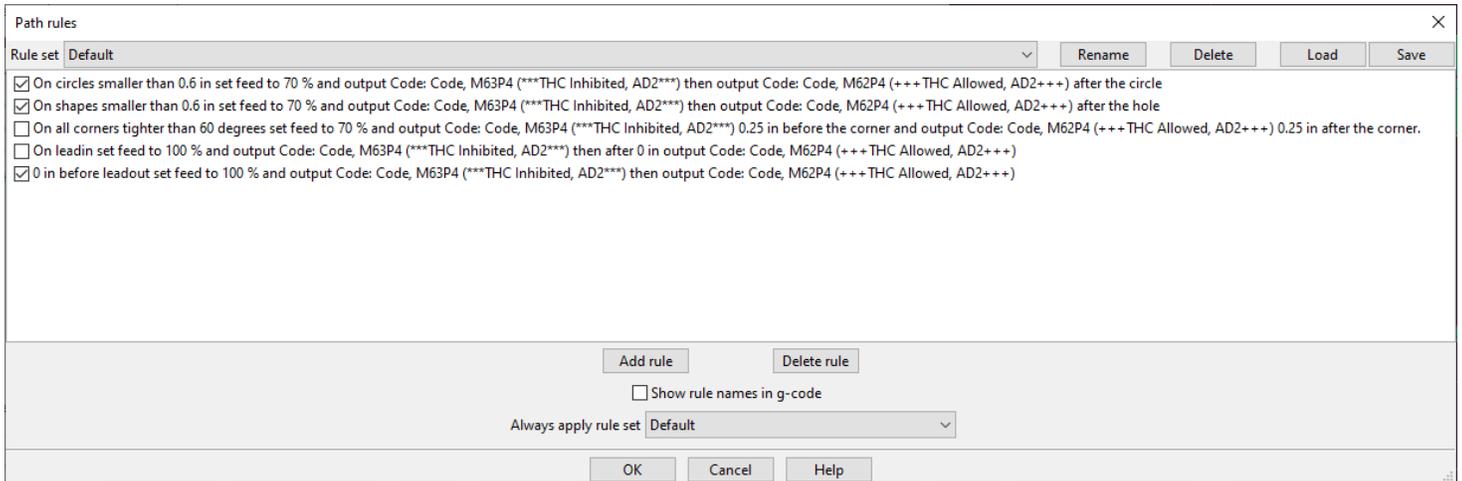


The Code Snippet is necessary to ensure that THC is allowed in the operation. The Path Rules will then allow/inhibit THC for certain features.

Note that if you don't want THC active at all during a specific operation within a gcode program, it is important to also disable the Cut/Path rules for that operation to avoid M62P4 being added automatically on a feature.

| | Use code snippet | Use path ruleset | Enable path rules |
|-----------------------|-----------------------------------|------------------|-------------------|
| THC wanted | M62P4 (+++THC Allowed, AD2+++) | Default | All |
| THC not wanted | M63P4 (***)THC Inhibited, AD2***) | None | None |

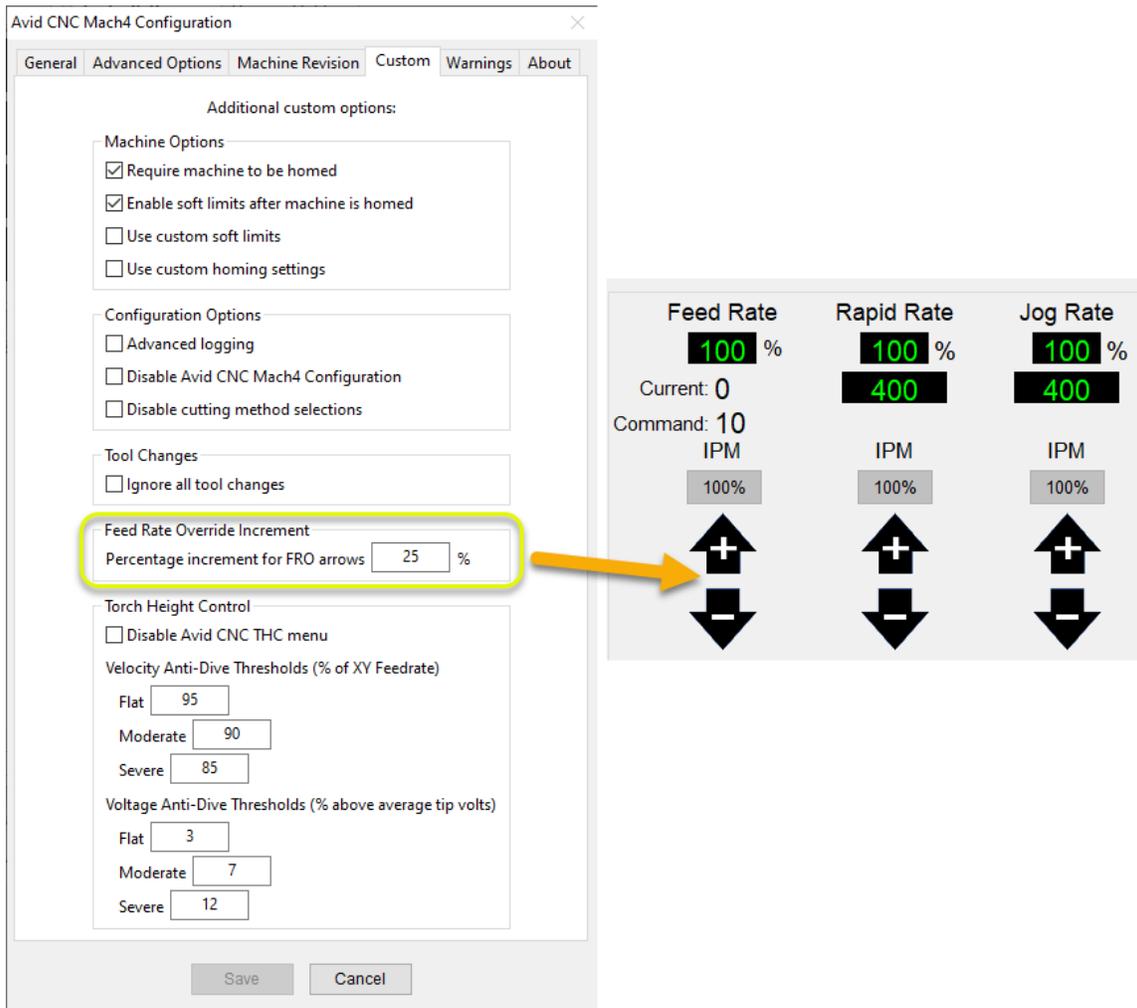
The default path rules are shown below.



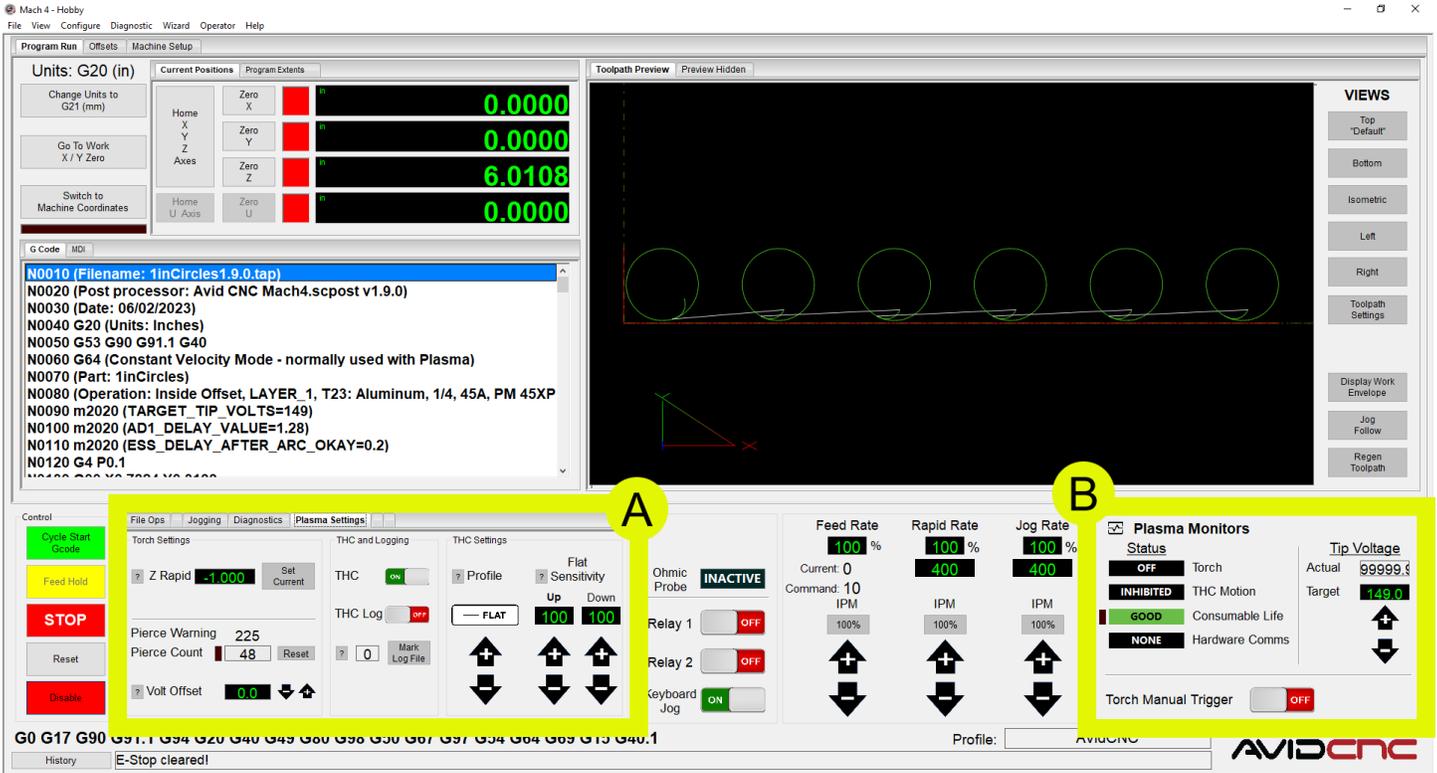
- Enabled: On circles/shapes smaller than 0.6in set feed to 70% and disable THC
 - This turns off THC and slows down the cut on small holes/features to prevent dives and improve the cut accuracy.
- Disabled: On all corners tighter than 60 degrees set feed to 70% and disable THC
 - This is generally redundant and not necessary with the new THC settings.
- Disabled: On leadin disable THC
 - This is generally redundant and not necessary with the new THC settings.
- Enabled: On leadout disable THC
 - This is needed to prevent dives on leadouts that have no material left to cut.

Mach4 - Feed Rate Override

It is now possible to change the adjustment increment for the Feed Rate Override buttons. This setting is in **Configure -> Avid CNC Mach4 Configuration -> Custom**.



Mach4 Plasma Screen Overview

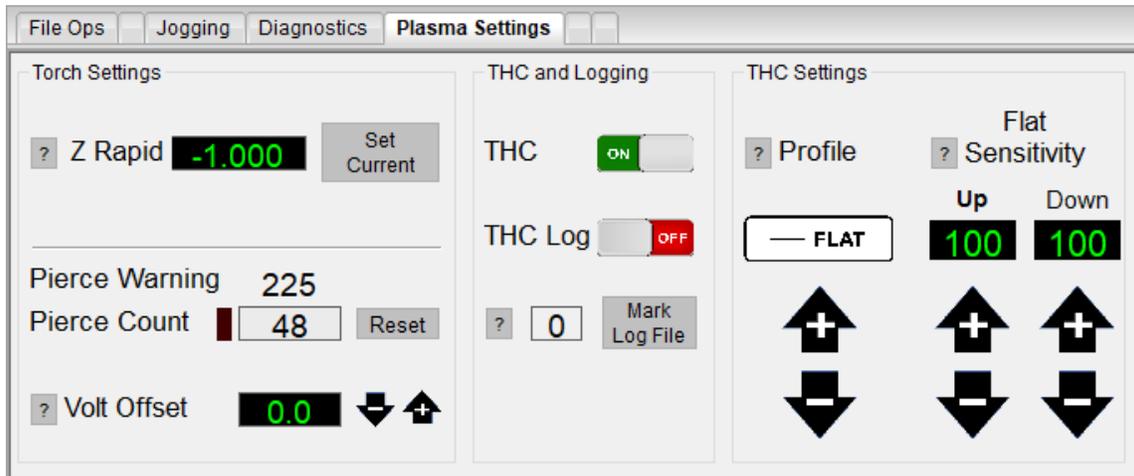


Help Pop-Ups

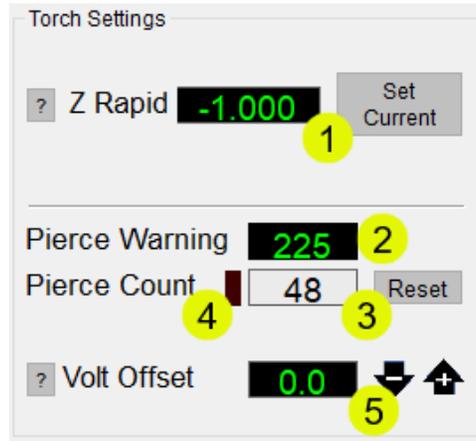


Throughout the screen clicking on the “?” icons will bring up brief help windows to describe screen functions.

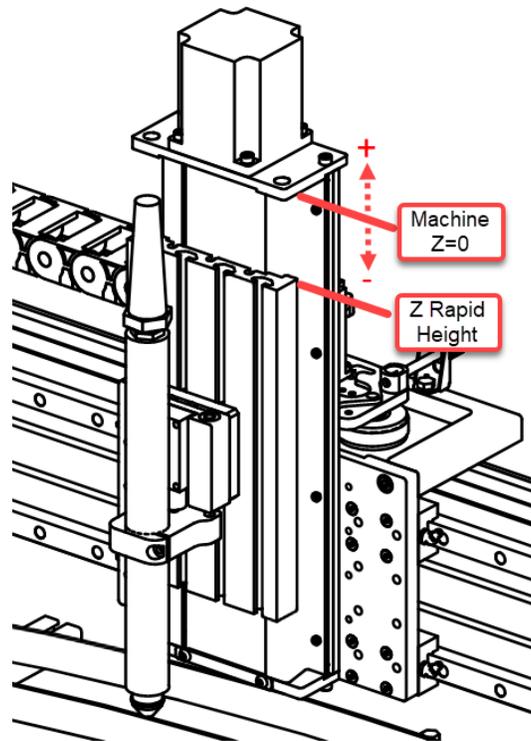
A: Plasma Settings group



Torch Settings



1. Z Rapid height: In between plasma cuts the torch will retract to this position **IN MACHINE COORDINATES**. The default setting of -1.000in (-25.4mm) means the torch will retract to 1" from the Z home sensor. The Z Rapid setting should always be (-).



The **Set Current** button will set the Z Rapid to the current position. Set this for each new program or material. The rapid position should be above the highest point of the material and above any likely tip-ups.

2. Pierce Warning: This DRO is editable when Mach4 is Disabled. This value is the number of pierces that will trigger a consumable/pierce life warning to indicate that the consumables are old.
3. Pierce Count: The number of torch fires since the last pierce count reset. Not editable. Reset to **0** with the **Reset** button.
4. Pierce Count Warning LED: Turns red when Pierce Warning limit is exceeded.

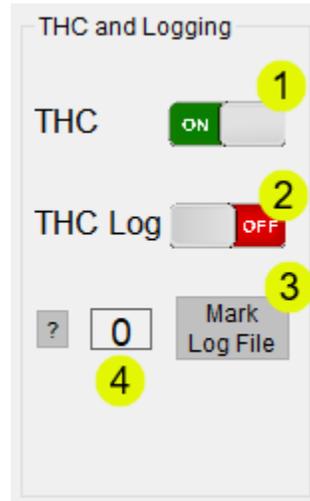
5. Volt Offset: Editable DRO and buttons that increment the value by +/- 1V. This sets the Tip Volt zero point and can be used to trim the voltage for consumable wear or a known offset.

A **negative** Volt Offset will cause the torch to cut **higher**. Use this for worn consumables that are cutting too low.

- Rule of thumb: 1V = 0.003" of electrode wear

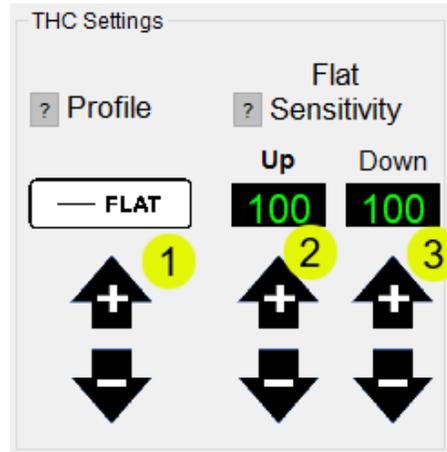
A **positive** Volt Offset will cause the torch to cut **lower**.

THC and Logging

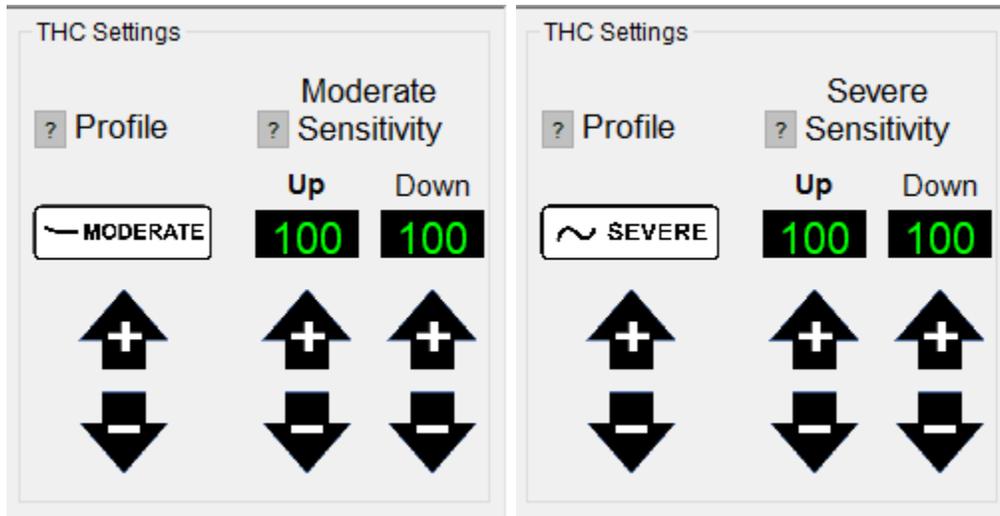


1. THC toggle: Toggles the Torch Height Control (THC) on and off. Default and recommended state is ON.
2. THC Log toggle: Toggles THC log creation on and off. When ON, THC logs are saved in the C:\Mach4Hobby\W9_HC folder
3. Mark Log File: adds a numbered mark in the THC log file at the current timestamp
4. Log file mark number

THC Settings



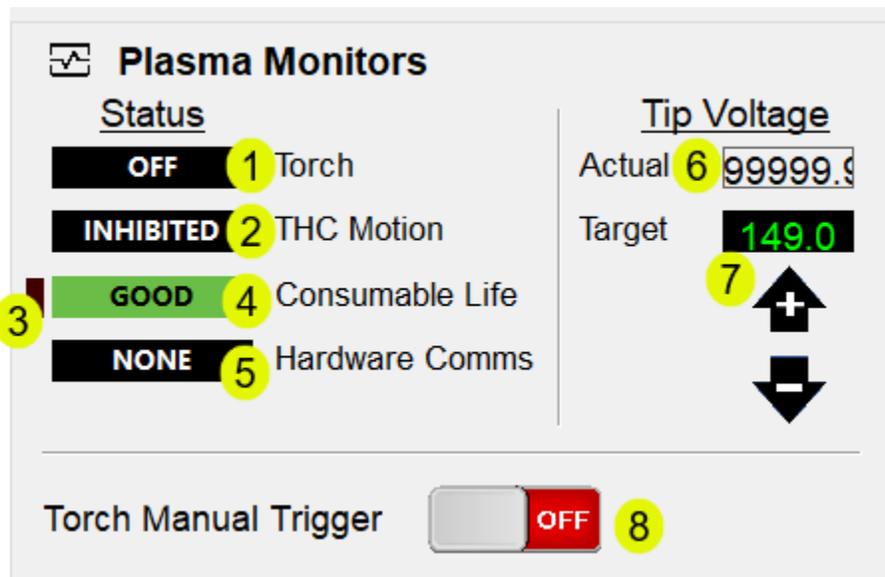
1. Material Profile: Use the arrows to choose the profile that best matches the Z height fluctuations of the material. Changes preset values for THC speed and Anti-Dive settings.



FLAT will be used in the vast majority of situations.

2. Up Sensitivity: Default of 100 uses the recommended precalculated THC settings. Increasing will increase the Z axis response speed in the UP direction. Decreasing will slow Z speed in the UP direction. This override setting is specific to a Material Profile and will be saved for FLAT, MODERATE, SEVERE.
3. Down Sensitivity: Default of 100 uses the recommended precalculated THC settings. Increasing will increase the Z axis response speed in the DOWN direction. Decreasing will slow Z speed in the DOWN direction. This override setting is specific to a Material Profile and will be saved for FLAT, MODERATE, SEVERE.

B: Plasma Monitors group



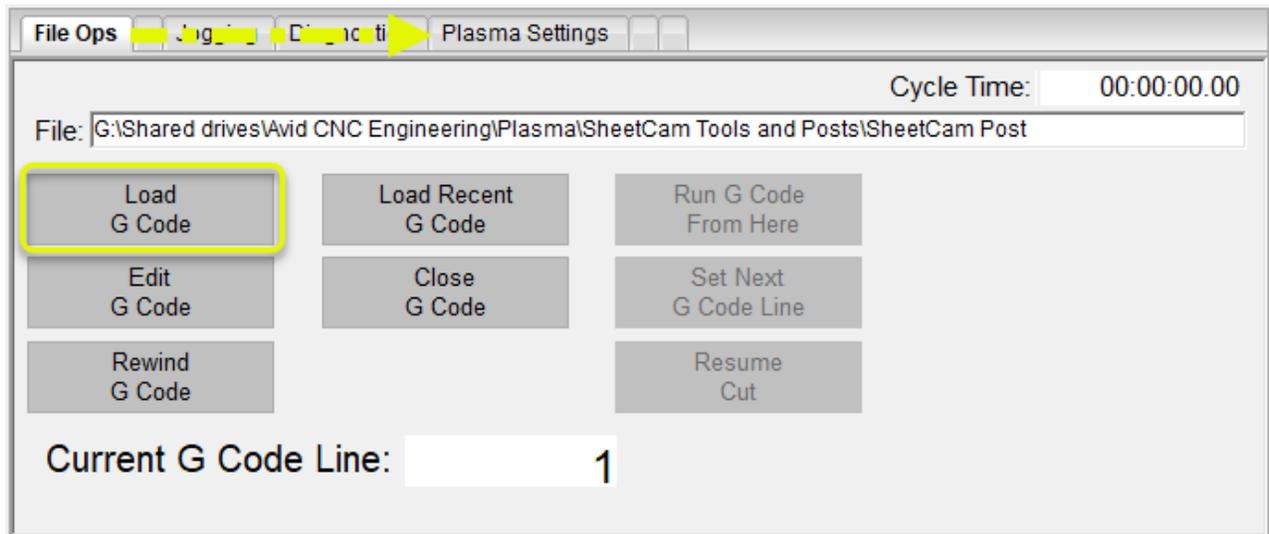
1. Torch: shows if the torch is commanded ON or OFF
2. THC Motion: shows if THC motion is INHIBITED, UP, or DOWN.
Note: INHIBITED does not mean that THC is turned off or is not working. It means that for some reason THC motion is currently inhibited which could be because of an inhibit in the gcode or an Anti Dive rule.
3. Consumable LED: turns red if the Pierce Warning limit is exceeded.
4. Consumable Life: shows if the number of pierces since last reset is GOOD or the consumables should be changed (REPLACE).
5. Hardware Comms: shows if Mach4 has established communication with the WarpRunner/TMC3in1 THC controller. States are NONE, ACTIVE, and UPDATING.
 - ACTIVE is required to start gcode
 - UPDATING means the WarpRunner/TMC3in1 is updating its firmware. This is common after updating Mach4 with new plugin versions.
6. Actual: displays the actual Tip Volts measured by the plasma power supply and WarpRunner/TMC3in1. The Volt Offset is also applied.
7. Target: the Target Tip Volts are read from properly prepared gcode once the program is started. The target can be adjusted up and down during a program using the arrows.
8. Torch Manual Trigger: turns the torch ON or OFF. **Caution: this will fire the torch with no additional warning so make sure to use proper PPE**

Typical Program Workflow

Typical operation will look something like this:

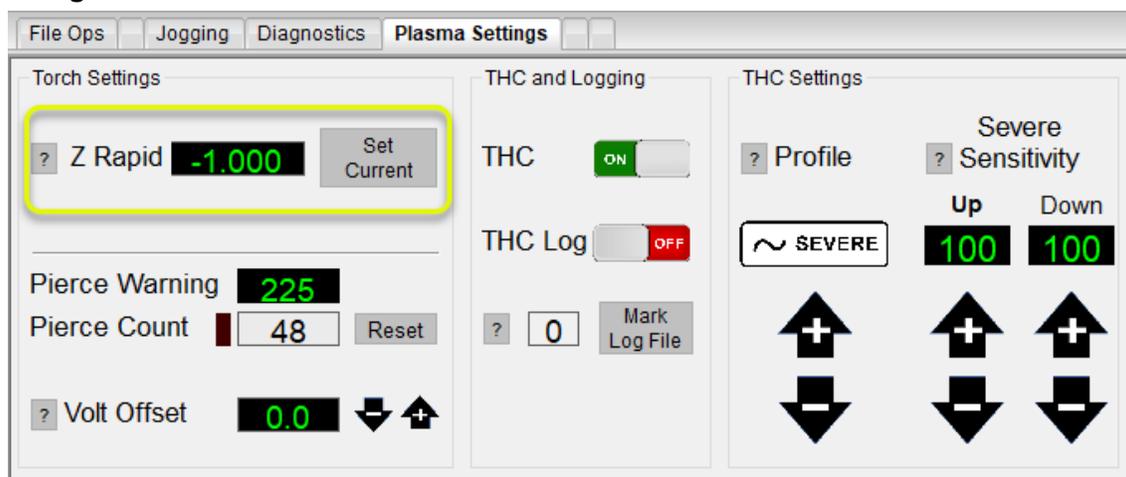
1. Home machine and set work XY zero
2. Set Z Rapid Height
3. Check consumable state
4. Check THC state
5. Verify THC settings
6. Cycle Start gcode

Load Gcode



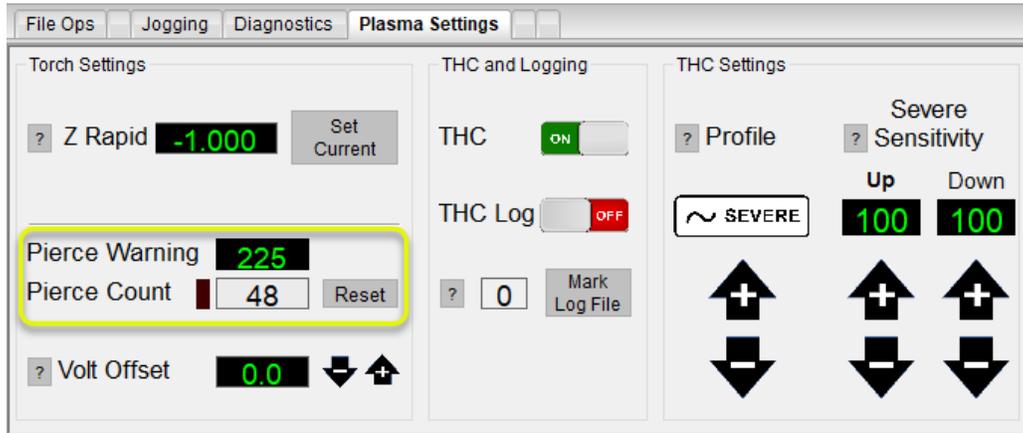
After loading new gcode the tab focus will switch to Plasma Settings to continue setting up for the program.

Set Z Rapid Height



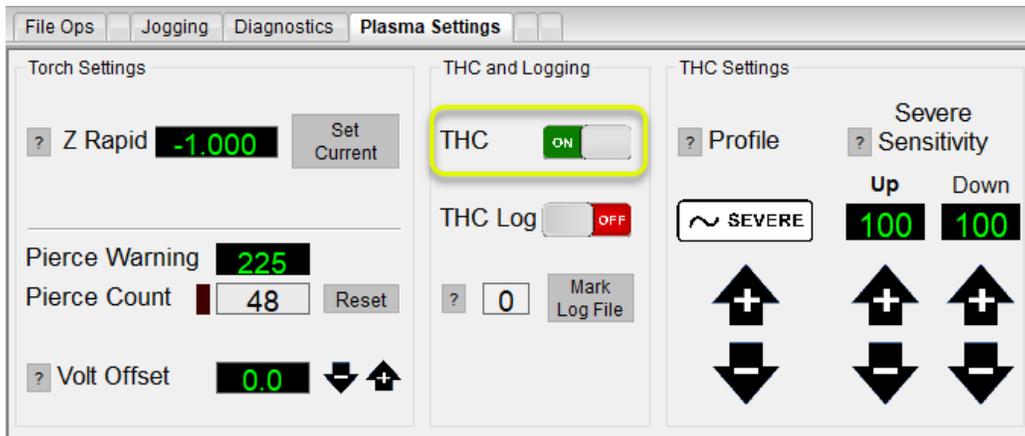
Jog the torch to a safe Z height for the material and tip ups. **Set Current** to set the Z Rapid height.

Check Consumable State



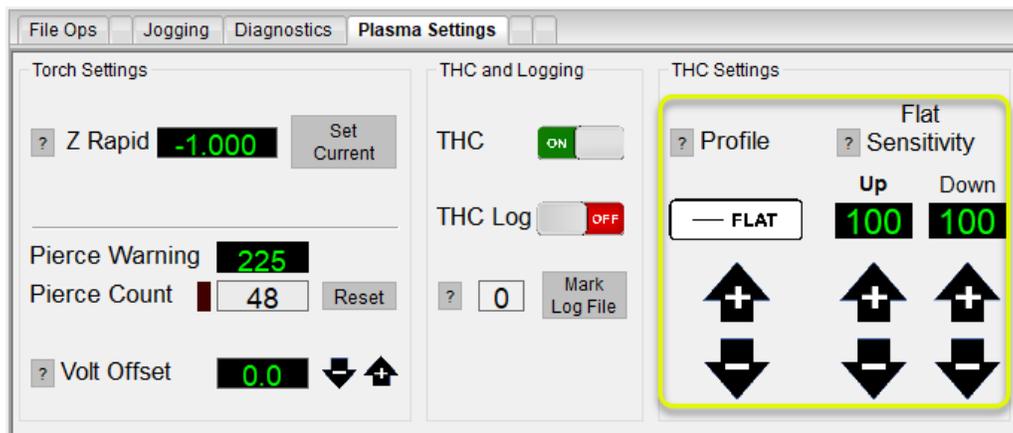
Make sure the current consumables are within their useful life.

Check THC state



In most cases this should be ON.

Verify THC Settings



Make sure the material profile and sensitivity settings match the program's needs.

Torch Height Control (THC)

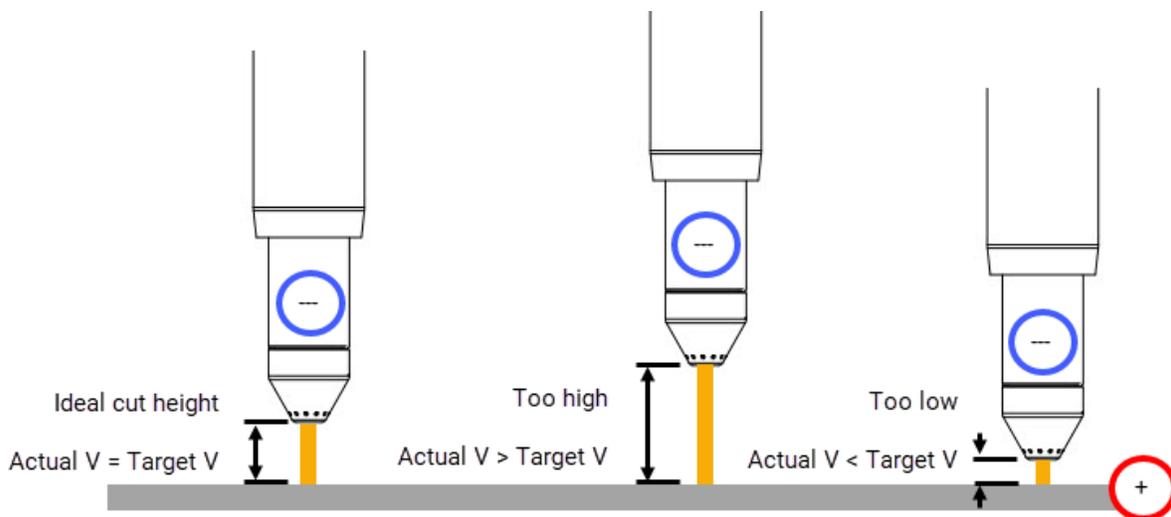
Operating Theory

The purpose of a THC system is to maintain a consistent torch-to-material offset across a workpiece. This allows for the best cut quality on materials that vary in height due to warping or inherent waviness.

Most THC systems use Arc Voltage (Tip Volts) as an indirect measurement of height. The arc voltage is measured between the Electrode (in the consumable stack) and the workpiece. The plasma arc voltage between the electrode (-) and workpiece (+) is governed by the equation

$$V = I \times R \quad \text{or} \quad \text{Voltage} = \text{Current} \times \text{Resistance}$$

The arc current is constant but the resistance increases with increasing arc length. This means the arc voltage increases as the arc lengthens and can be used as an indirect measurement of torch-to-material offset.



A THC system will move the Z axis / torch up and down as it attempts to bring the Actual Tip Volts closer to the Target Tip Volts.

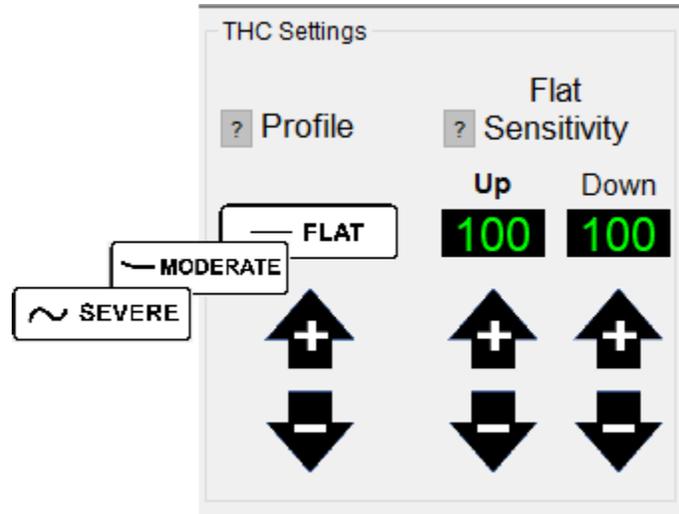
Because it is an indirect measurement, Tip Volts and actual torch height will not always directly correlate. Some of these situations and the likely results in tip voltage:

- **Leadin/leadout** - the voltage is unstable when the XY velocity changes significantly.
- **Tight corners** - the voltage typically increases as the XY motion slows around a corner.
- **Small holes/shapes** - the voltage will increase as XY motion slows
- **Pre-cut material (voids)** - the voltage will increase rapidly as the torch-material distance gets very large because there is no material directly under the torch.

All of these situations can lead to unintended THC motion. In most cases this will be an unwanted dive towards the material. This unwanted motion can be prevented in some cases with **Anti Dive** settings and in all cases can be reduced with correct THC tuning for the material.

Avid CNC Mach4 THC Tuning

Material Profile



The Z axis speed required for the torch to successfully follow material height variations is dependent on the severity of the slope and the XY feedrate (set in SheetCam). In the two exaggerated illustrations below, the torch needs to follow the orange path. The XY feedrate remains constant but the slope increases so the material on the right will need a greater Z axis speed to follow the material.

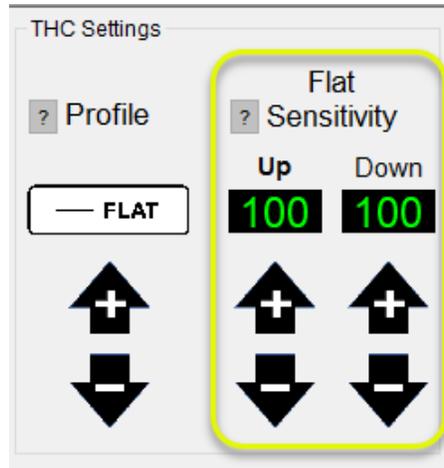


Because most disruptive events (corners, holes, etc) during a cut lead to an increase in tip voltage and a likely dive, it is useful to only allow enough Z axis speed to follow the likely material curve and correct for delayed response. The **Profile** settings of FLAT, MODERATE, SEVERE allow increasingly faster Z motion to correct for voltage errors. Each setting also adjusts some Anti Dive settings for best performance.

The FLAT material profile should be used in most cases. This is intended for material that is nominally flat but may have some warping or may warp during cutting. A 5% slope (0.6in/12in) can easily be handled by this profile.

MODERATE and SEVERE should only be used for significant material slopes or severely warping thin material. In order to perform on sloped material these settings allow faster Z motion and have looser Anti Dive settings that will not catch as many voltage spikes. This can lead to torch crashes on flatter material or very complex cuts on sloped material.

Fine Tuning - Up and Down Sensitivity



Within each Profile setting the Z axis response speed for Up and Down movements can be adjusted independently. This acts as a percentage override to the pre-calculated values. 100 is the default and the range is 10-200.

If the torch fails to closely track the material Up or Down, use the **Plus** arrow to increase the Up or Down setting(s).

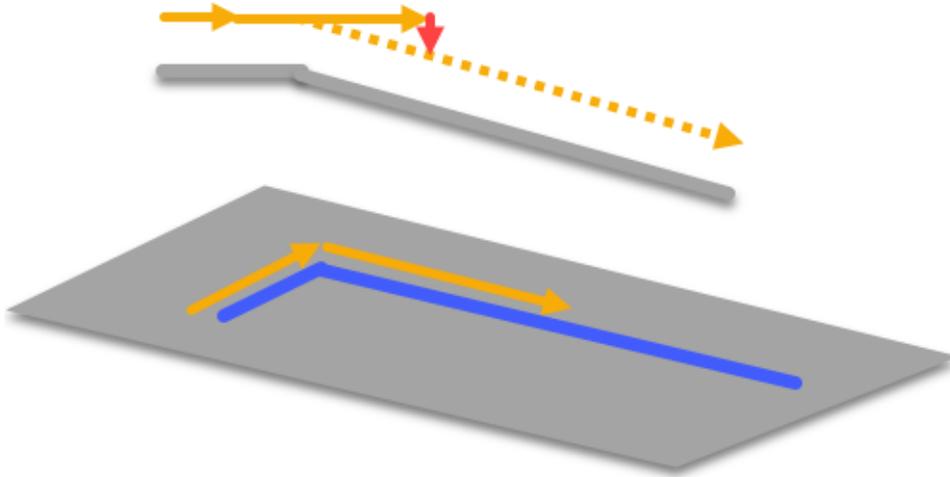
If the torch moves up or down too much (diving/crashing or excessive height), use the **Minus** arrow to decrease the Up or Down setting(s) appropriately.

These settings are specific to each Profile. They can be adjusted on-the-fly during a cut.

Additional THC Settings

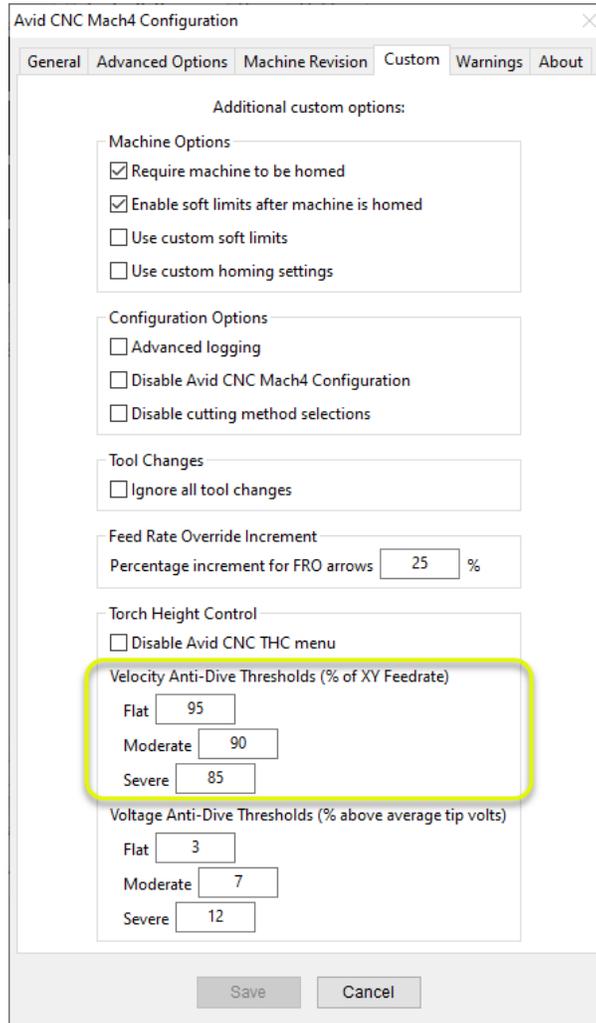
Several other settings are available but rarely needed.

Each material Profile has default Anti Dive settings that are intended to only allow the Z motion necessary to track the material. In general steeper material slopes require Anti Dive settings that allow motion more often.



The example above is a corner on sloped material. The torch path is shown in orange, following the blue profile. The upper image shows that **Velocity Anti-Dive** may inhibit THC motion around the corner because the tip volts are likely to spike. But as the torch starts to move past the corner the material will drop away before the Z axis is allowed to respond. Depending on the Anti Dive settings this could result in poor quality or a failed cut.

Velocity Anti-Dive Thresholds



Avid CNC Mach4 Configuration

General | Advanced Options | Machine Revision | Custom | Warnings | About

Additional custom options:

Machine Options

- Require machine to be homed
- Enable soft limits after machine is homed
- Use custom soft limits
- Use custom homing settings

Configuration Options

- Advanced logging
- Disable Avid CNC Mach4 Configuration
- Disable cutting method selections

Tool Changes

- Ignore all tool changes

Feed Rate Override Increment

Percentage increment for FRO arrows %

Torch Height Control

- Disable Avid CNC THC menu

Velocity Anti-Dive Thresholds (% of XY Feedrate)

| | |
|----------|---------------------------------|
| Flat | <input type="text" value="95"/> |
| Moderate | <input type="text" value="90"/> |
| Severe | <input type="text" value="85"/> |

Voltage Anti-Dive Thresholds (% above average tip volts)

| | |
|----------|---------------------------------|
| Flat | <input type="text" value="3"/> |
| Moderate | <input type="text" value="7"/> |
| Severe | <input type="text" value="12"/> |

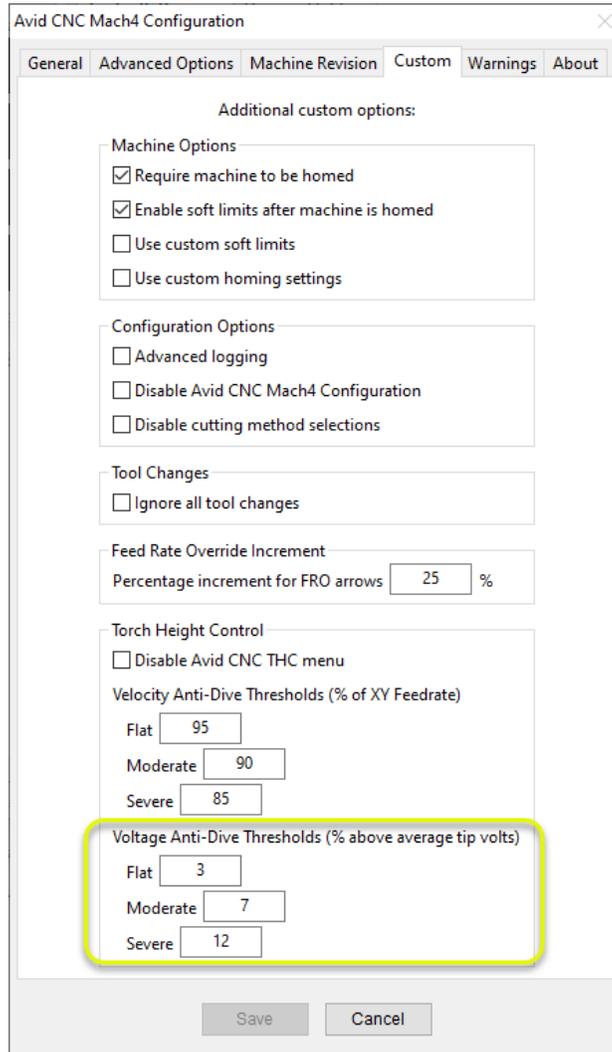
Save Cancel

This setting is in the **Configure -> Avid CNC Mach4 Configuration** menu. It sets a threshold at the given percentage of programmed XY feedrate. If the actual feedrate drops below the threshold (due to acceleration limits in corners and small arcs), THC motion is inhibited until the feedrate recovers.

If the torch is frequently diving in low speed situations but otherwise tracking the material properly, increasing the Velocity Anti-Dive Threshold could help prevent the dives.

If the torch is failing to track the material during or after low speed situations, decreasing the Velocity Anti-Dive Threshold could help to allow more motion.

Voltage Anti-Dive Thresholds



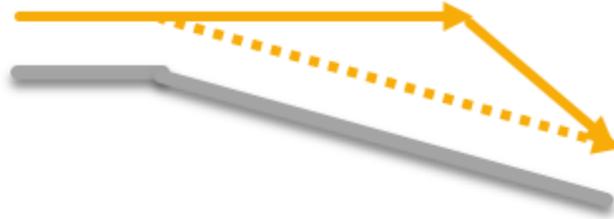
This setting is in the **Configure -> Avid CNC Mach4 Configuration** menu. It sets a threshold at the given percentage above the Average Tip Volts during a cut. If the Actual (or “instantaneous”) tip voltage exceeds the threshold, THC motion is inhibited until the actual voltage drops.



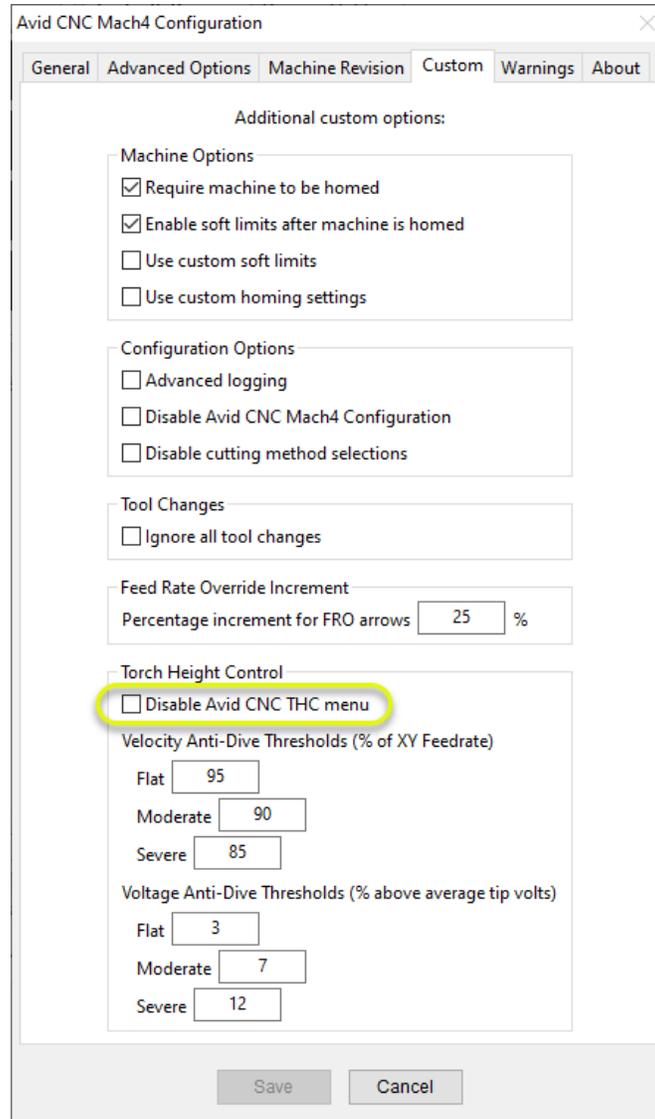
The image above shows the Actual voltage (red) spike above the Average (blue) and threshold (orange). In this case THC motion would be inhibited during the spike.

If the torch is frequently diving when crossing voids or other irregularities, decreasing the Voltage Anti-Dive Threshold could help prevent the motion.

If the torch fails to begin following changes in material slope, increasing the Voltage Anti-Dive Threshold could help allow motion sooner. This would look like the image below where the torch does not follow the intended path (dashed orange) and has to quickly recover.

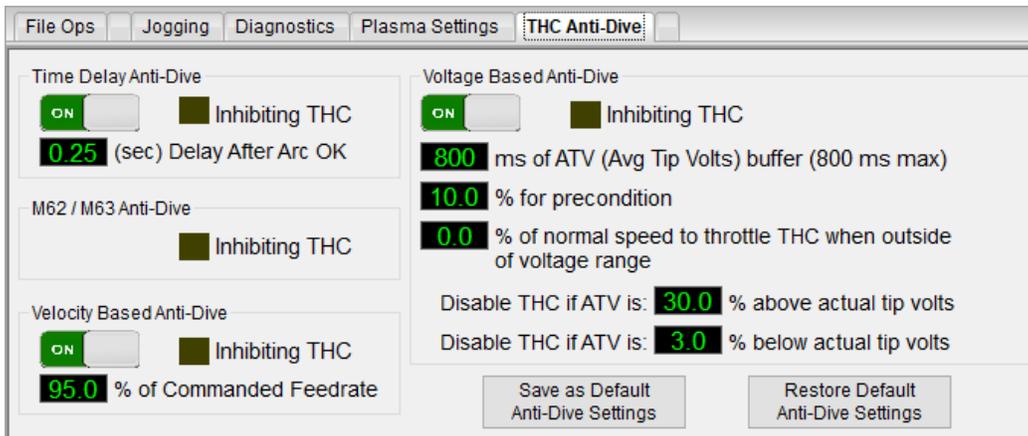
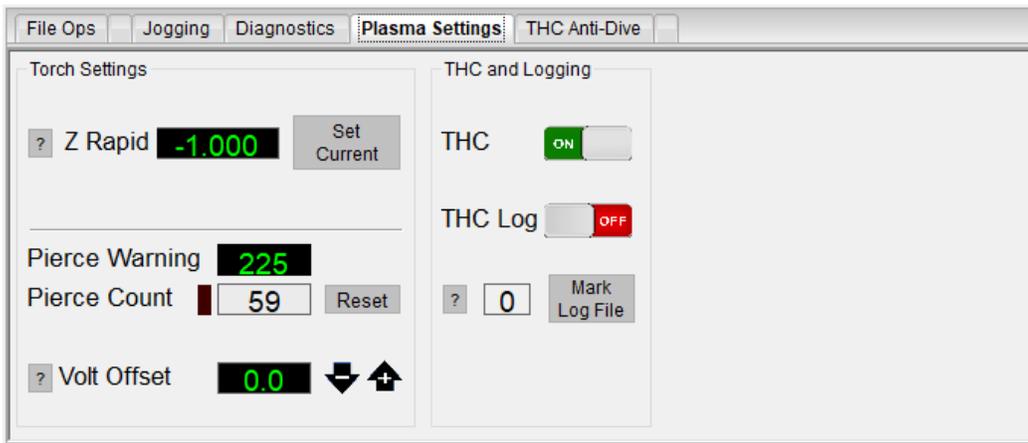


Advanced Manual THC Anti-Dive Settings



If necessary or desired, the Avid CNC THC parameter calculations can be disabled. Doing so will disable all feedrate and profile based Z axis response calculations. This should only be done by users that have an understanding of each Anti-Dive setting, including the Linear Response Band.

This will hide the Profile and Sensitivity settings and expose the THC Anti-Dive tab



Getting Help

If you have performance issues that cannot be resolved with the available settings, please contact us with as many of the following files as possible:

1. SheetCam project file (.job extension)
2. G-code file (.tap or other text file equivalent)
3. THC log file (.thc extension). These will be saved by default in **C:\Mach4Hobby\W9_HC**
4. Mach4 log file (.log extension), saved wherever you choose.

Recording a THC Log



1. Make sure the THC Log toggle is ON.
2. Run the program that is experiencing issues.
3. Copy the log file. Log files are saved in **C:\Mach4Hobby\W9_HC** with a timestamp filename and a “.thc” extension.

Recording a Mach4 Log

<https://www.avidcnc.com/support/instructions/software/mach4UsersGuide/#record-mach4-diagnostic-log>

Find the Diagnostic menu at the top of Mach4 and select Logging. This will pop up the window below. Use the Play button to start logging before hitting Cycle Start, and save the log file after the relevant program is finished to minimize the log file size.

