DTHC IV -SETUP/INSTALL MANUAL FOR COMMANDCNC PART 1

PART1

REV 0.8
DECEMBER 2016

For use with the following CandCNC products:

MP3600E-DTHCIV LINUX version
BladeRunner EtherCut(all versions) with CommandCNC®
Plazpak Series (all versions) with CommandCNC® option
CommandCNC® Upgrades all series (DTHC IV)

Options Covered in PART2

DCP-01 Digital Current Probe (11/02/2010)
DCC Dynamic Cut Control
HyT-Connect Hypertherm Connection Kits
TAP Total Automation Plasma
Using router with CommandCNC

Use this manual to install and setup a DTHC IV Expansion Module in the field for any of the above listed CandCNC products OR to setup and test the DTHC IV functions in a CandCNC Product that already has the DTHC IV Module installed.

Disclaimer: This is a preliminary manual and may contain typos and references the older CandCNC products or software.

TOC

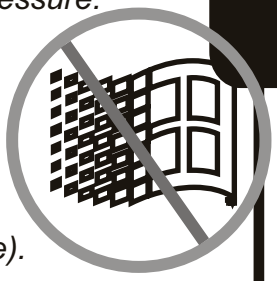
DTHC IV Setup and Installation

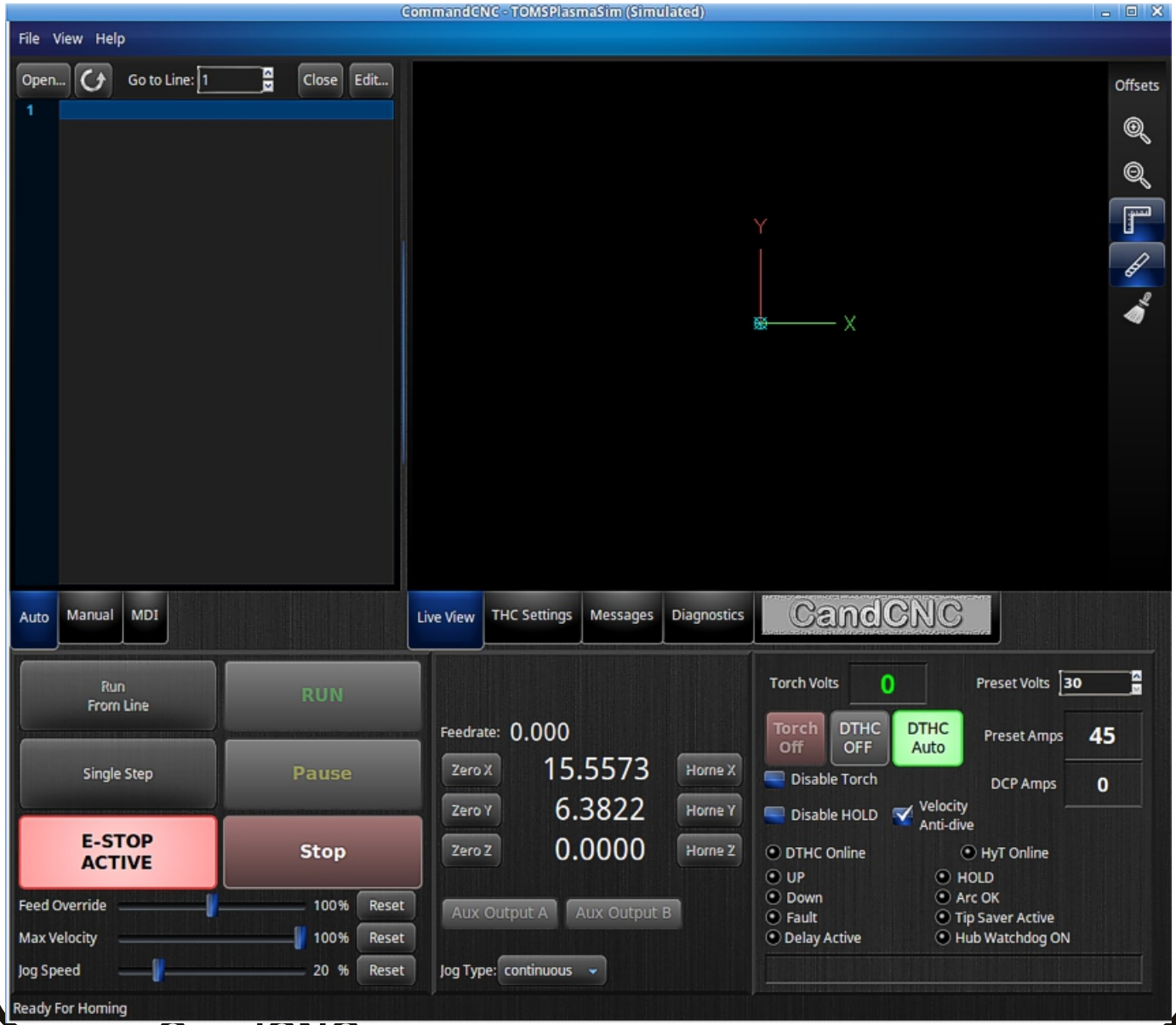
TOC

NEW! CommandCNC Control Software from CandCNC

A

- *Designed for small shop or commercial/industrial applications.*
- *Clean, uncluttered easy-to-use interface with layers of menus.*
- *Optimized for Plasma cutting. Alternate screens and profile for routing.*
- *Inneractive lighted buttons show status at a glance.*
- *Fully DCC ready for automatic cutting. Supports all DTHCIV setting options.*
- *Supports TAP for Hypertherm RS485 to control Cut Current and Air Pressure.*
- *Built on stable LINUX OS.*
- *Full REAL TIME operation .*
- *Integrated High Speed Ethernet Motion Card for smooth motor control.*
- *Most system hardware settings hidden from operator. Easy setup.*
- *Supports 10 opto-isolated inputs. 5 full independent axis of control.*
- *Instant DTHC ON/OFF (hardware control).*
- *Custom POST for SheetCAM for Electronic Cut Chart (plasma Tool table).*
- *Requires one Ethernet port and one USB port (for RS485 adapter).*
- *Engineered to work with control electronics from CandCNC.*



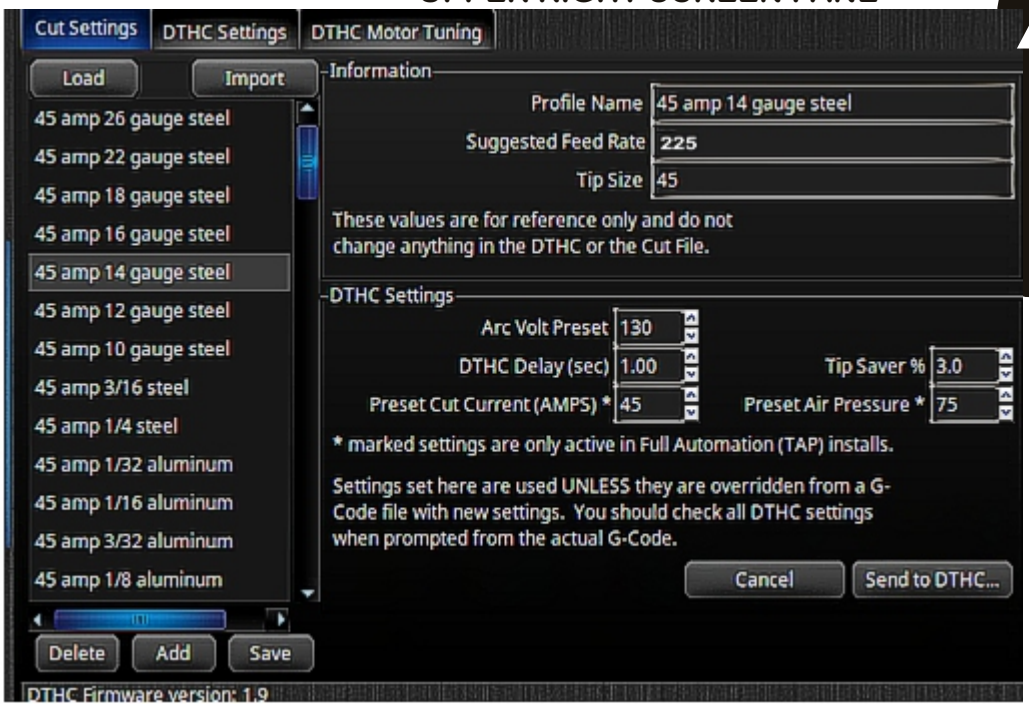


CommandCNC Control Software Screen Details

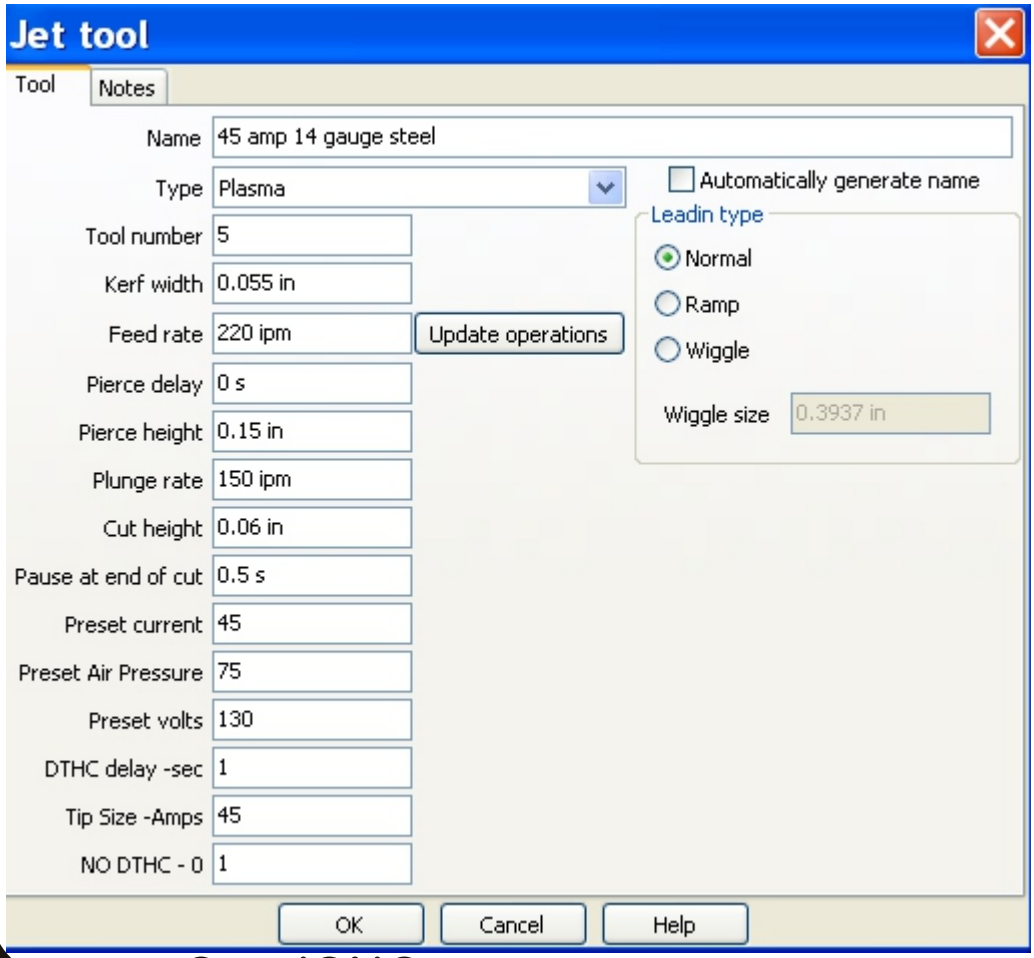
Set any parameter manually on screen or run from local Stored Cut Settings. Use the DCC feature with SheetCAM to use the Electronic Cut Chart (ECC) in the SheetCAM tool tables and run in Automatic Mode where the G-Code job file sets all of your DTHC Settings.

TAB interface lets you ripple thorugh Cut Settings. DTHC Settings and the DTHC Motor Tuning. to quickly confirm or change any setting.

UPPER RIGHT SCREEN PANE



TOOL select screen from SHEETCAM TNG using CandCNC POST



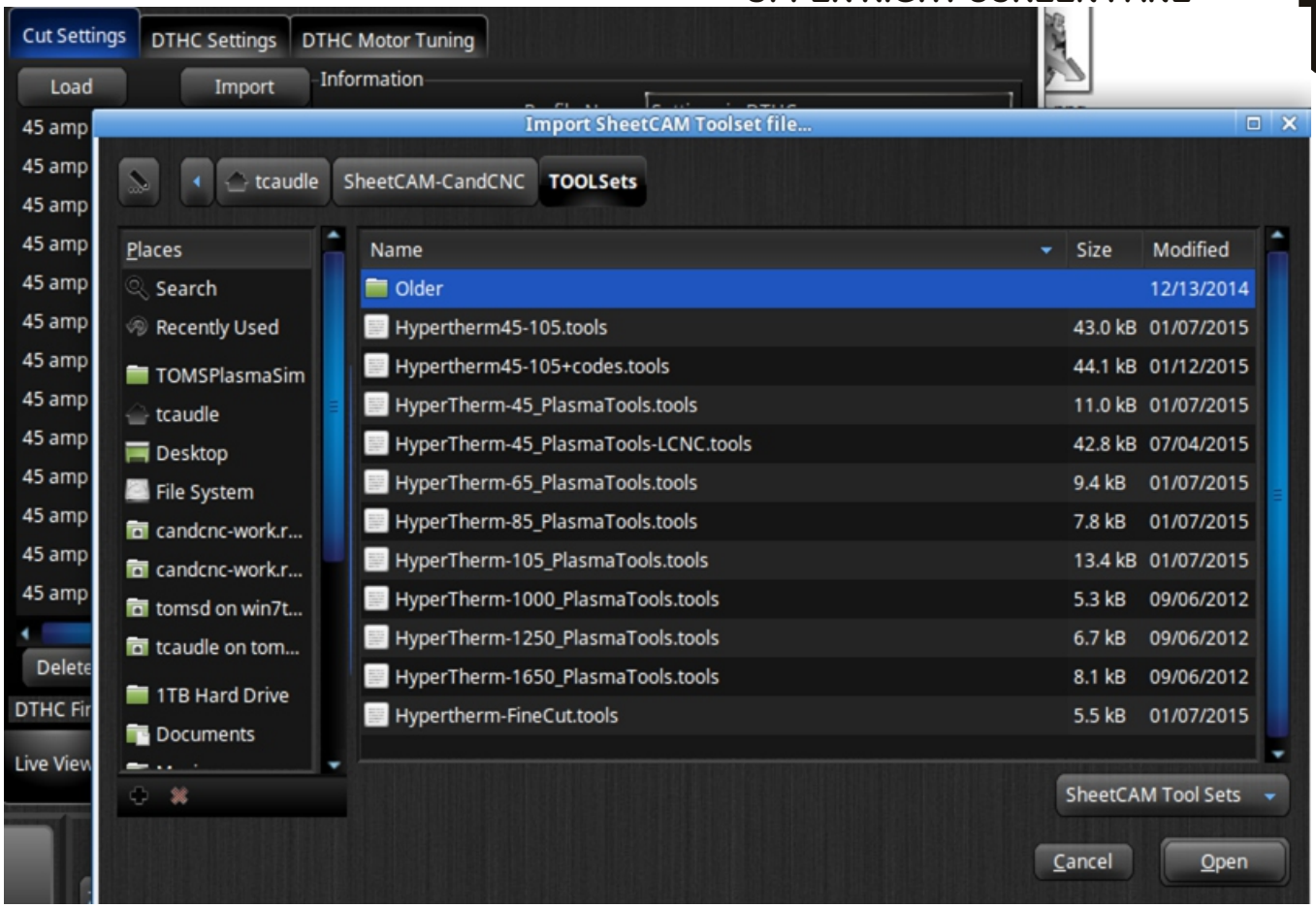
Using a pre-loaded toolset for the Plasma (Hypertherm 45 in this example) you can select by cut current and material. If you are running TAP on the Hypertherm you will have Preset Current and Preset Air Pressure. The settings are embedded in the G-Code and when you run the job it automatically changes the settings in CommandCNC®, and updates the hardware to those values. Full automatic cutting.

A

CommandCNC Control Software Screen Details

Use (import) SheetCAM toolsets for Local tool definitions. If you decide to use local Stored Settings and manually control the cutting process with a non-automated POST or another CAM program you can still use the stored toolsets we provide for SheetCAM.

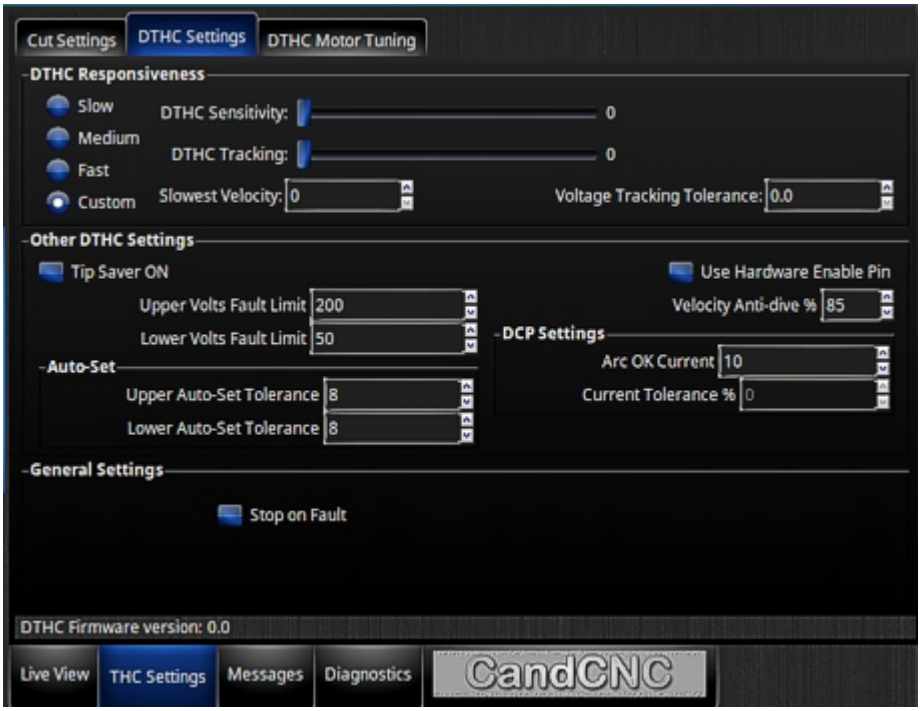
UPPER RIGHT SCREEN PANE



Pre-configured cut profiles have always been part of the CandCNC DTHC Cut Profiles but have used a different file format, Now you can take advantage of a rich set of pre-built tool sets complete with the TAP options of Cut Current and Air Pressure for use with the Hypertherm RS485 options. CommandCNC fully supports all of the Advanced Automation features of the Windows based version.

A

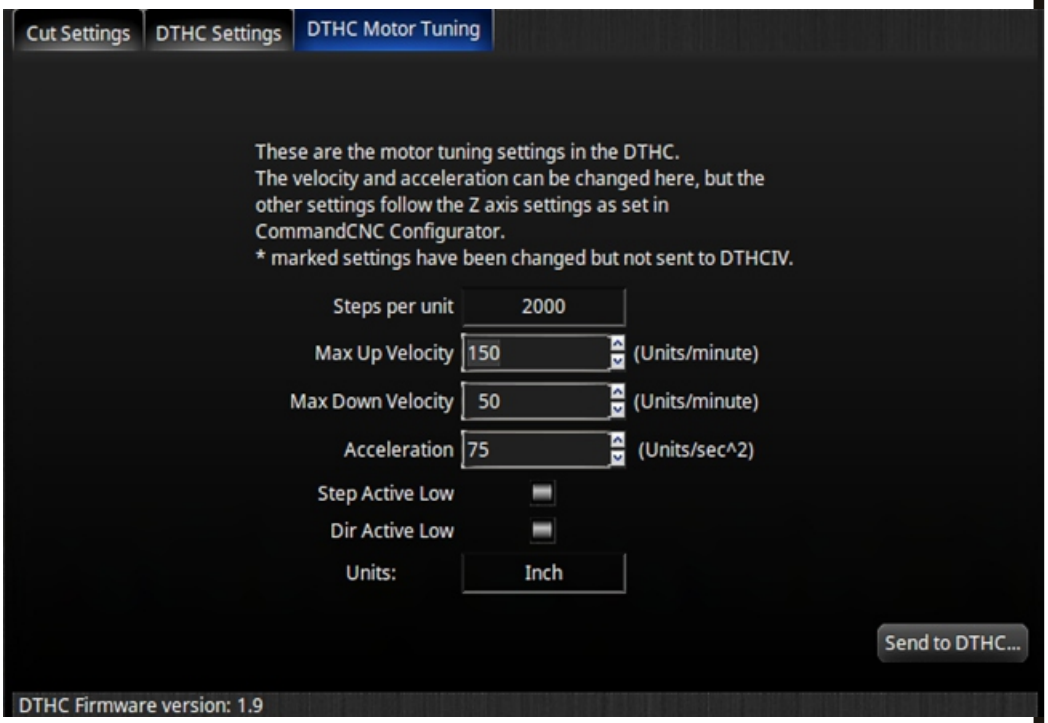
UPPER RIGHT SCREEN PANE



With full access to the PID settings for the high performance DTHCIV with the exclusive CandCNC presets (selectable in SheetCAM as a cut parameter) you get the accuracy and flexibilty to cut everything from flat plate to highly warped HVAC material and get the response needed for high feedrates.

Set your DTHC Motor Tuning for Z the same as your normal Z (default) or tune it up or down to alter the reaction speeds for DTHC. The DTHCIV uses a unique Z axis sharing algorithm that lets each process control the same axis using different parameters.

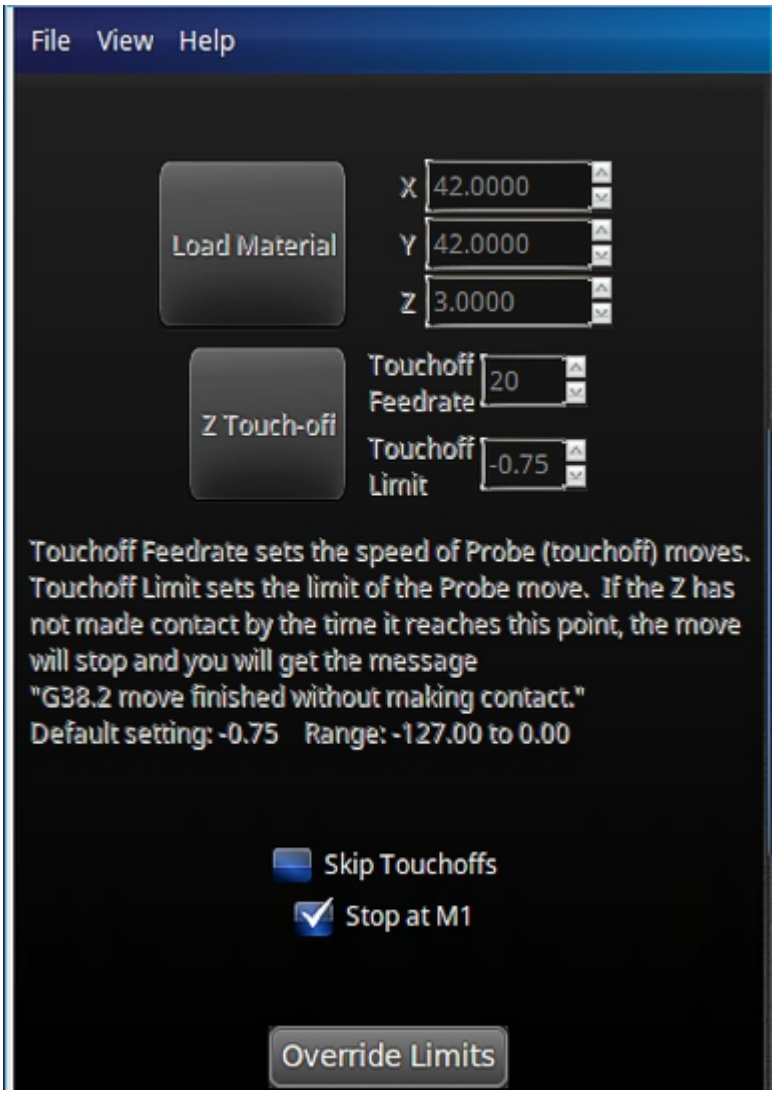
NEW! For a more precise control of DTHC response you can tune the UP Velocity separate



You can setup and test the DTHC Motor Tuning in CommandCNC or in the Hub Admin. Some settings are automatic (not settable). Initial setup should be done in the Hub Admin Utility .

A

MANUAL window in Upper Left



The MANUAL button in the upper left frame opens a set of manual sub routines that perform specific manual actions.

The LOAD MATERIAL button moves th XYZ to a preset position defined by the values entered into the XY and Z Input boxes. The values are stored and only need to be entered once and can be changed if needed. After the values have been set you can push the **Load Material** button and it will manually move the three axis to that position. **NOTE: You have to be out of RESET before you can enter values or activate the button**.

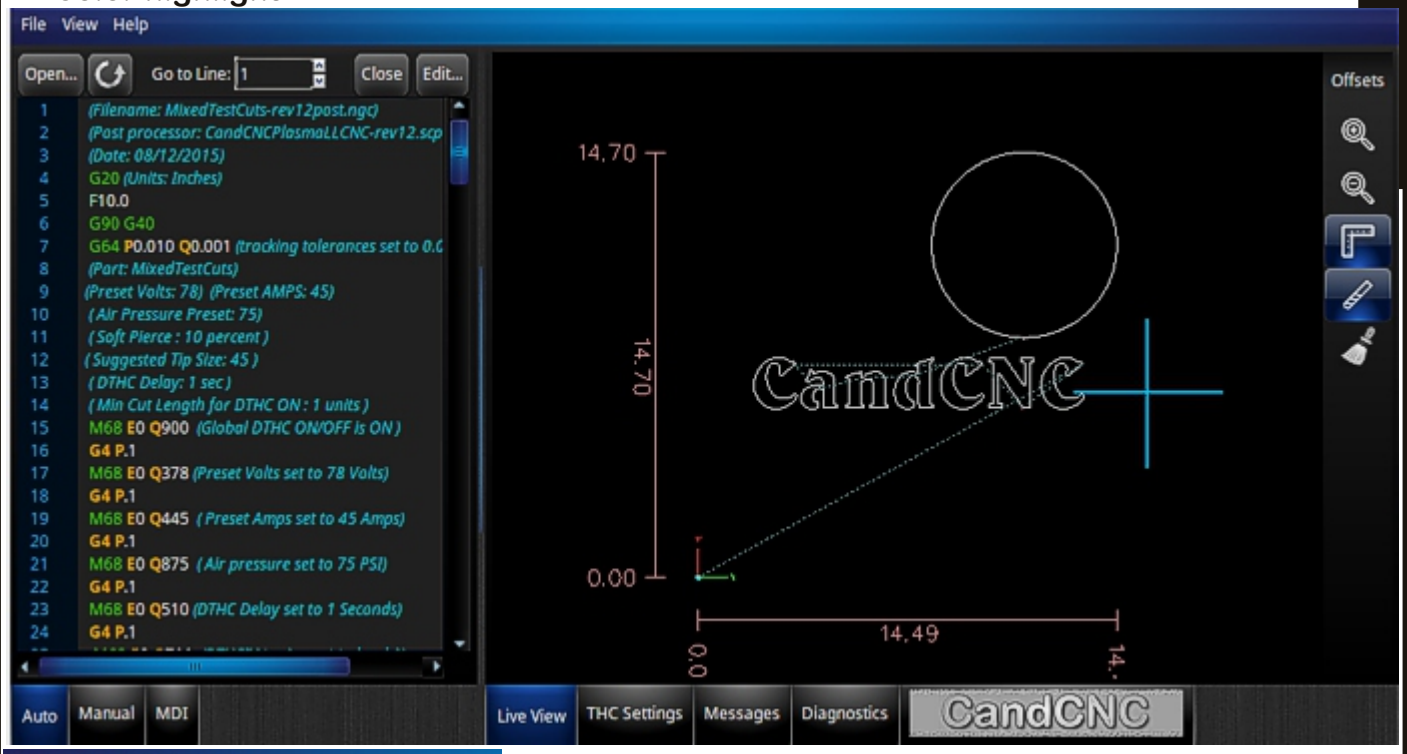
The Z touch-off button is a special function sub routine that moves the Z down until the probe input is activated then lifts the Z the amount to cancel the switch offset. It then zeros the Z. THIS IS DIFFERENT than a homing move on Z since homing just moves to the switch and stops. Homing does not apply the offset or lift the Z to the new zero. The Z Switch Offset is entered into the Z motor tuning as the **Home Switch Offset**

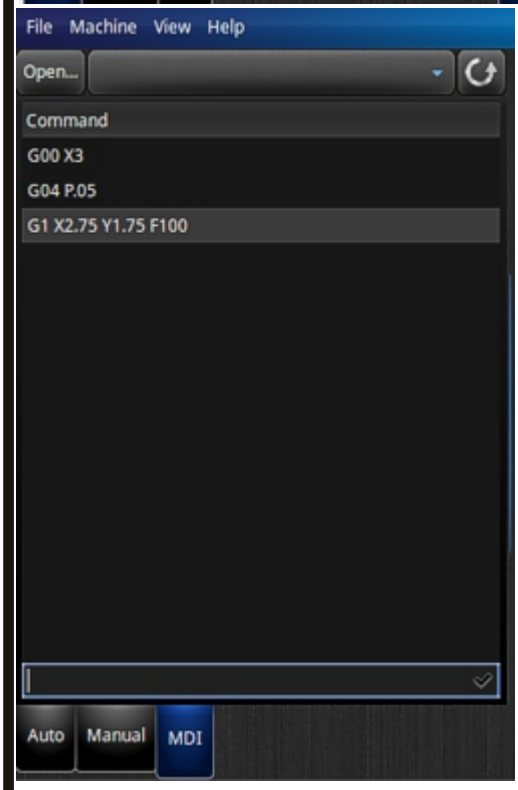
Touchoff Feedrate set the speed of the Touchoff moves. That number needs to be slow enough to prevent significant travel after the switch is tripped. The slower speeds increase the accuracy.
Touchoff Limit sets a lower limit for the probe move. This number is NEGATIVE in value (below zero) It prevents the Z from going below this point before it stops and displays an error. Because some material may not be level or have a consistant height the actual zero may change and you may need to increase the negative number
Skip Touchoffs (checkbox) This box is normally unchecked except on “SIMULATED” profiles. You can check it on a normal profile to do a test run without the torch and no touch-offs, DO NOT USE THIS DURING NORMAL CUTTING. It goes off by defaut if you reload the Config. ***DEFAULT = UNCHECKED***
Stop on M1 (checkbox) This sets CommandCNC so that if the GCode uses an M0 as a Pause, it converts it automatically to an M1 for better continuty. It does not cahnge the base fiel I simply changes the imported version. ***DEFAULT - CHECKED***

A

G-code display uses key word color highlight

Toolpath zoom with mouse or from menu





MDI screen lets you type one or multiples lines of G-Code and execute

Load your G-Code generated by the *CommandCNCLinux* Post and see the toolpath . Take the system out of RESET and RUN the code and watch it simulate the cut. You can enlarge or shrink the toolpath display independent of the screen and add or remove the rulers using the right-side menu bar.

Context highlighting on the G-Code scroll lets you better read and understand what it is doing.

You can use the MDI screen to write and run code interactively to do a quick cut or to troubleshoot.

Tab menus for easy navigation

Large, bright digital readouts (DRO's)

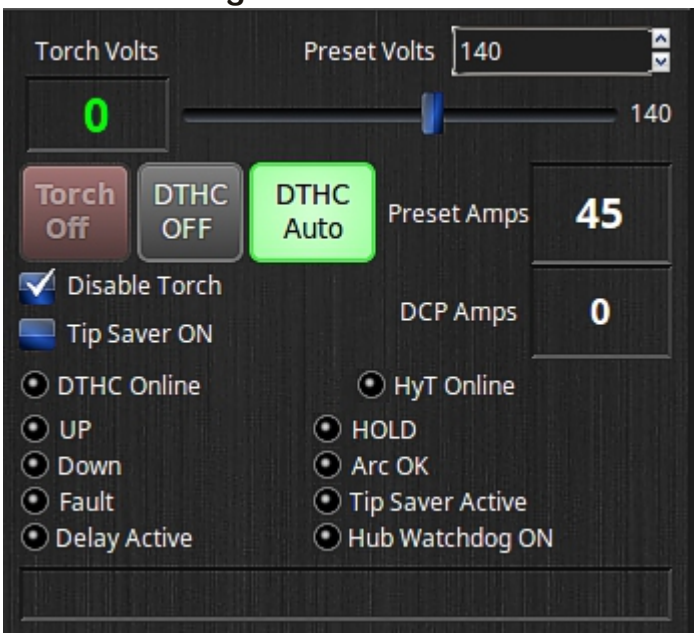
Handy slider controls for variable settings

Full DTHC control at your fingertips
See the status of your DTHC at a glance

Button Text color changes when lighted

Button text shows state when active

Buttons light in colors when activated



Lighted indicators show system and DTHC status

Disclaimer: This document contains information about software that is still in a pre-release form. Features and screens may change in the initial release version and subsequent releases. .

This document describes a product from CandCNC that provides a new approach to Automated Torch Height Control. The DTHC IV is a 4th generation design using a faster more complex microprocessor and is the culmination of over 10 years of THC designs from the engineering team at CandCNC, and is the pentacle of high performance / low cost digital torch height control.

The DTHC IV for LINUX has the following features: *(new features in RED)

Σ

Σ Compact Surface Mount Design. High Speed Processor, Boot Loader on chip for easy firmware updates

Σ Uses ECC (Electronic Cut Chart) for stored settings (2)

Σ RS485 C3Bus® communication using standard UTP cabling (CAT 5) (2)

Σ Isolated external HOLD circuit. No other inputs on BoB used or needed

Σ Works with LinuxCNC (3)

Σ Single Operator Console for all CNC settings and operations....no knobs, no manual settings.

Σ Supports Dynamic Cut Control on all plasma cutters (2)

Σ Usable with virtually any plasma cutter with or without CNC connectors (4)

Σ Z motion speeds of up to 250 IPM with STEPPERS!

Σ Setup and testing via RS485 and Windows/Linux app "HUB ADMIN utility"(1)

Σ Z Response tuning has true PID settings to prevent overshoot and instability

Σ 4 Presets for response tuning cover most types of cutting and Z mechanical types

Σ Plug compatible with existing CandCNC MP3600's and CommadnCNC based BladeRunner/Plazpaks after Oct 2015 (1)(5)

Σ Ethernet pulse interface (4)

Σ More than 5X faster than systems using MACH3 internal THC logic

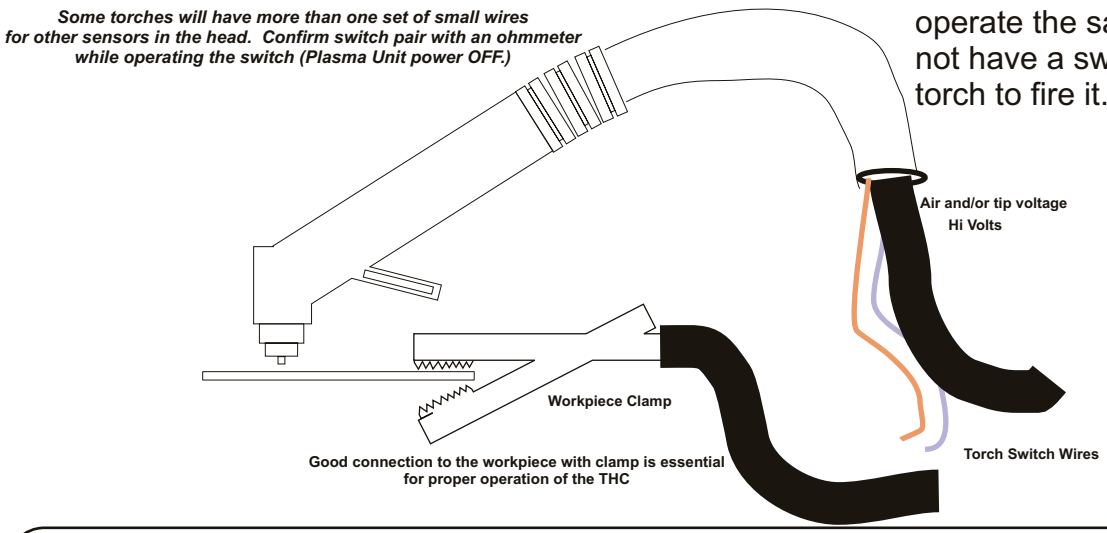
Σ Engineered to do extreme cutting including corrugated and HVAC type materials

Σ Unique Patent Pending Hybrid Design shares single Z axis allowing G-Code position control mixed with high speed local loop under DTHC control

(1) Requires optional USB to RS485 4 Port C3BUS Hub from CandCNC
(2) Requires SheetCAM TNG and custom CandCNC POST Processor
(3) Requires CommandCNC running on LINUX.
(4) May require other options/ connection kits
(5) Upgrade kits available

INTRODUCTION: How THC works

NOTE: Machine Torches operate the same but do not have a switch in the torch to fire it.



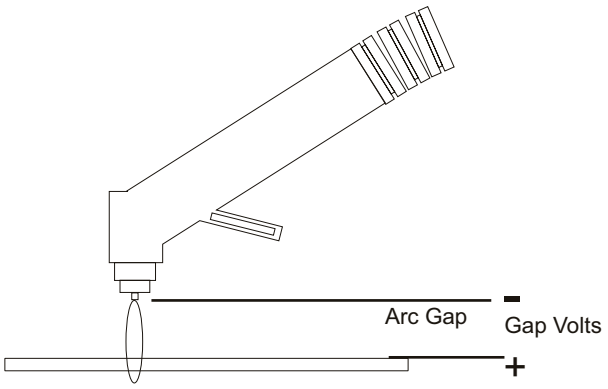
HOW DTHC (THC/AVC/DTHC) WORKS

Automatic Torch Height Control (often called just THC) works by reading the Arc Gap Voltage while cutting. Plasma uses constant current cutting. If the current stays constant and you vary the gap (either by moving the torch or moving the material UP or DOWN) then the voltage will change in proportion to the change in arc gap. Much like the altimeter on a plane (that measures barometric pressure to determine altitude) the Arc voltage indicates the RELATIVE distance from the end (tip) of the nozzle to the top of the material.

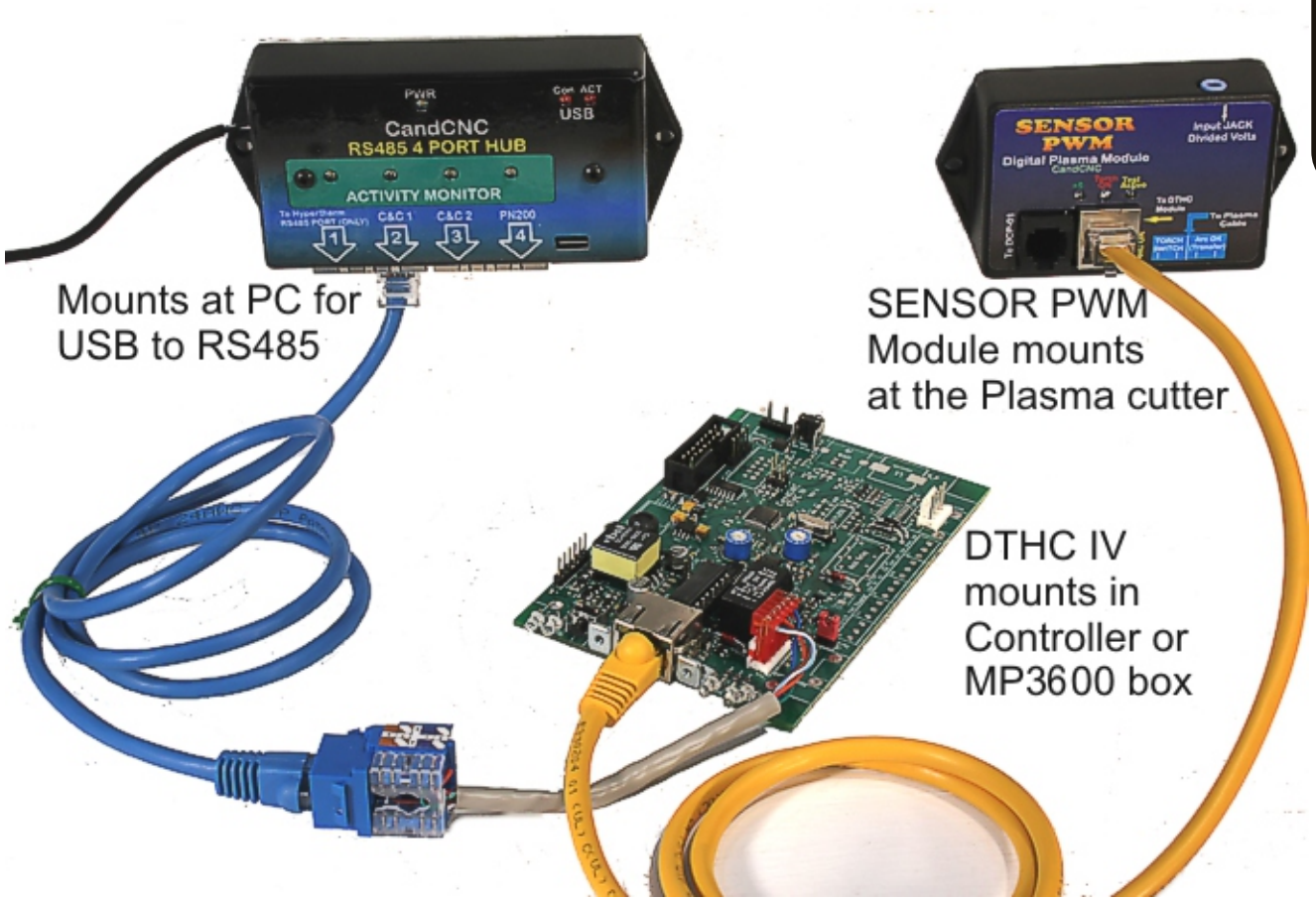
The change in voltage for a change in height is a small percentage of the overall cutting voltage. A 1% change in voltage (100 to 101 volts) is equal to several thousands (typically .010 or more) of arc gap change so the THC must be able to see and act on a small change in a large number.

The THC control must take the actual cut voltage and compare it to a preset “target” and move the Torch Up or Down to try and correct the height based on it’s arc voltage. The process forms a “servo loop” where an “error” voltage from a preset is used to physically move the torch Up or Down to “correct” the error. Under normal cutting conditions the voltage stays constant with height but certain conditions that effect the arc gap voltage can skew the gap volts and cause the THC circuit to overreact. The feed rate (how fast the tip is moving across the material) determines the current density and the Gap Volts. A slower feed rate will cause an increase in Torch Volts (if no THC servo is there to correct). With THC engaged the circuit will sense the higher voltage and based on the error created lower the torch to try and compensate. This occurs if the arc crosses a void or existing kerf or at the end of a cut when the scrap drops out. The DTHCII includes an adjustable Anti-Dive (Tip Saver) that senses a rapid voltage spike and locks down motion.

- Arc Gap = Arc volts=Torch Volts
- 1volt (change) = approx .015”(change)
- >Arc volts = > Arc Gap. (greater the Arc Gap the greater the Arc Volts)
- Z moves opposite Arc Volts based on Preset Volts. Torch Volts Above Preset: LOWERS torch; Torch Volts below Preset: RAISES torch.
- Control has “window” (Span Volts) where no UP or DOWN occurs. (prefect cut height) Anything inside the Span (+ or -) from the Preset generates NO change. SPAN VOLTS is set in 1/4V increments in the Cut



C3BUS™ USB to RS485 4 Port Hub
See text for details



The following pages will cover the identification, hookup, setup and testing of the DTHCIV Digital Torch Height Control System. Each card/module has a specific function and set of tests. A minimum system will consists of a DTHC IV Expansion Module (interfaced to a CandCNC UBOB III Universal Breakout Board) the THC Sensor PWM Module and the RS485 C3 Bus(required).

The DTHC IV is a later version of our popular DTHC II that was introduced in 2011, and is the 3rd generation DTHC and the 4th generation of Digital THC's from CandCNC. The DTHC line has always been a feature-rich product with advanced technology based on powerful embedded processor chips and modern surface mount components. From the innovative **Total Isolation** (including the analog Arc Volts readings) to the **Self-test** and built in **"Tip Saver" anti-dive,** the DTHC has earned a reputation for being solid, reliable and user-friendly, while offering the operator a wide range of options for precise cutting with plasma and all at a very effective price.

The DTHC IV builds on that success by making the product faster, with more features and the first ever "Shared Z " design for MACH3 and CommadnCNC for Linux to access Automation control signals in the plasma. Using the same external PWM Module as the DTHC II, it takes a divided Arc Volts signal (available as an option on several plasma models or from our optional RAV-01 card) and changes the small analog signal to a constant level PWM signal that is many times more immune to external noise and EMF problems. Designed to work with your existing equipment including HF and CD start units or the most modern plasma units in built-in Automation Interfaces, it sets a new standard for Torch Height Control. You can pay MORE for a THC but cannot get more features or accuracy than the DTHC IV offers.

This shows a hookup with a single cable to a Plasma cutter with an external CNC connector (aka CPC Connector) such as the Hypertherm and the Thermal Dynamics "A" Series

DCP-01 (option)

Used for ARC Current Readout; NOT Required

DTHC IV Module

Cat5 UTP cable

UP to 25 ft

Located in the control box with the UBOB III card

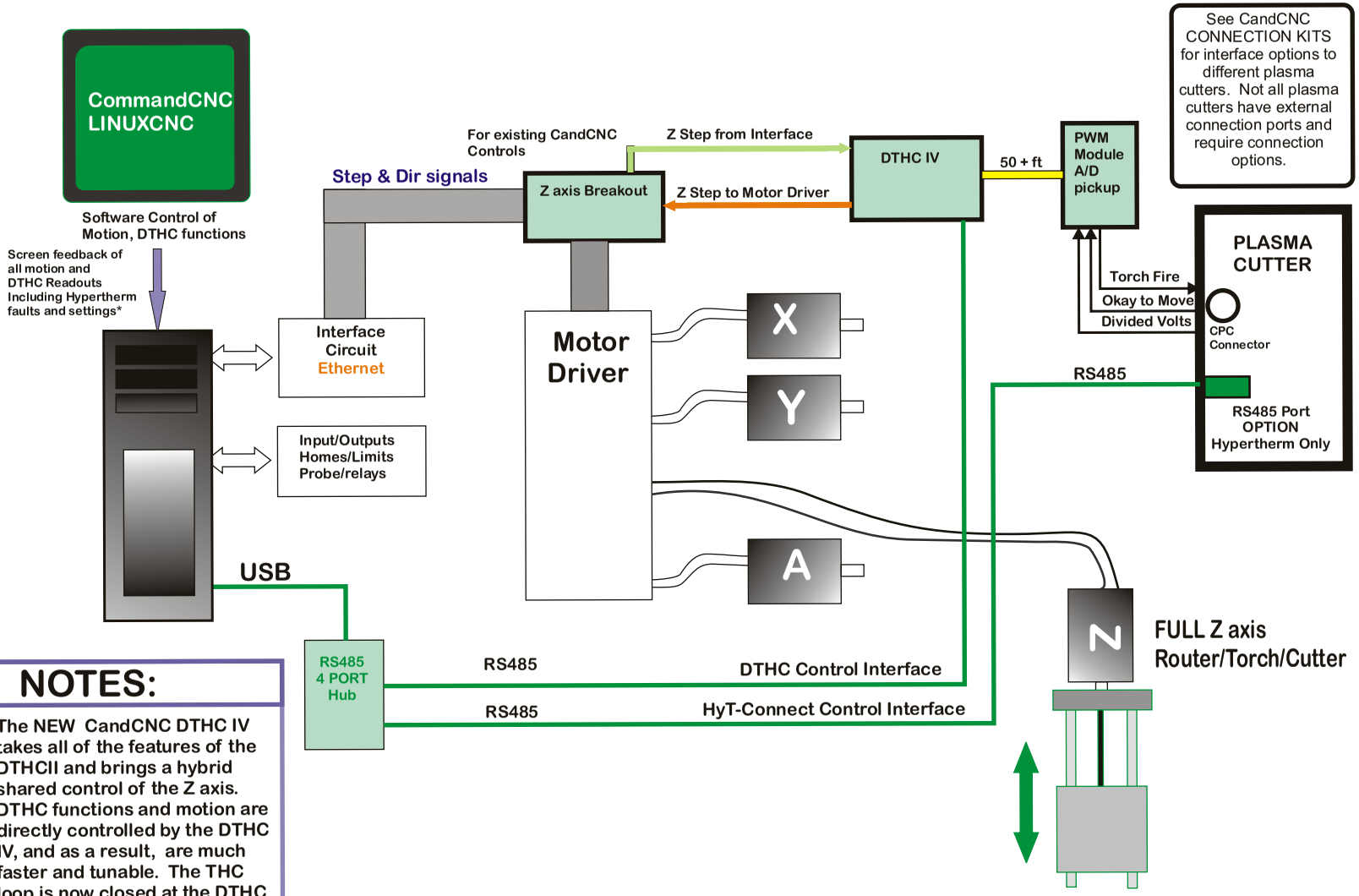
PWM MODULE
Plasma Pickup Module located external and close to the Plasma cutter

MIC-01 Custom Cable

CPC Plug on Plasma Unit

Single cable hookup to Plasma Unit

NEW DTHC IV BASED CONFIGURATION



NOTES:

The NEW CandCNC DTHC IV takes all of the features of the DTHCII and brings a hybrid shared control of the Z axis. DTHC functions and motion are directly controlled by the DTHC IV, and as a result, are much faster and tunable. The THC loop is now closed at the DTHC IV so there is no longer a requirement for the THC logic in the software controller nor for a real-time high speed set of inputs. It still offers the same high level operator interface and the Advanced DCC and TAP options and still allows full plasma control from G-Code

SETUP OUTLINE

NOTE some of these steps may already have been performed in the BladeRunner User Manual or MP3100/MP3500 User manuals. If so just check the settings and move to the next step

1.Do the install for the BladeRunner AIO LINUX system (separate Manual)

2. Run CommandCNC Installer (or later version) to install the products you have from MENU.

6. If not yet done, Use Motor Tuning section in CommandCNC User Manual to setup motor tuning in MACH.

7. Test JOG on all XY and Z axis and Simple G-Code (MDI) motion commands in CommandCNC

8. Update the hub firmware and DTHCIV Device firmware if needed. (use **Hub Admin** to see firmware versions and update firmware)

9. Use HUB ADMIN and DTHC IV Setup section to test and calibrate the Z tuning for DTHC operation. Test JOG using screen buttons

10. Install the Plasma Modules (connection kit) and Make connections to plasma cutter.

11. Run tests in this manual to confirm operations

12. Run simple straight line tests with DTHC OFF (in MANUAL MODE) cutting metal to get baseline numbers and confirm configurations.

14. Install SheetCam CandCNC Support file.to add in custom POSTS and Toolsets

15. Generate simple shapes files using correct post in SheetCam TNG and perform test cuts.

Items in RED are the most often skipped steps and cause problems getting the DTHC to work properly

- Open CommandCNC with either the Plasma (Icon) depending on which product(s) you have.

- Do some preliminary motion checks. If this is a new install and you have no motor tuning numbers to go from, refer to the Motor Tuning Section of the ***BladeRunner AIO LINUX User Manual*** (even if you have the MP3600-DTHCIV version or a Plazpak for LINUX).

- Continue through the *DTHC IV Setup and Config Manual* to do the testing and fine tuning of the DTHC IV. There are settings and tuning requirements for the DTHC IV that are critical to get acceptable results

IMPORTANT! IF you are moving from a MACH based DTHC . DTHCII or DTHCIV to the LINUX based DTHCIV it is a radically different system and requires the tuning steps outlined in this manual. If you skip this than your results will be less than optimal. There are separate documents that will cover the physical part of the upgrade (swapping out the hardware) but the setup, calibration and testing of the new DTHCIV needs to be done using this manual.

3. **DTHC IV High Speed Digital Torch Height Control**. The DTHC IV uses the RS485 (C3BUS) for the following:

- Transfer of screen information to CommandCNC like Torch Volts, Torch Amps and the status LED's for Torch UP, Torch DOWN and ARC OKAY. Unlike the DTHC II that uses defined parallel port inputs for the UP, DOWN and ARC OK, the DTHC IV sends the data across the RS485. The only "hard" input from the DTHC IV to CommandCNC is the HOLD signal .
- Communication to CommandCNC to update Z position when DTHC is active(tells CommandCNC where the DTHC has moved the Z while under THC mode)
- Pass parameters and settings in CommandCNC to the memory of the DTHC IV. This includes all of the DTHC Cut Profile Settings either from the screen or from the G-code when using DCC

C3BUS® MANAGING THE HUB

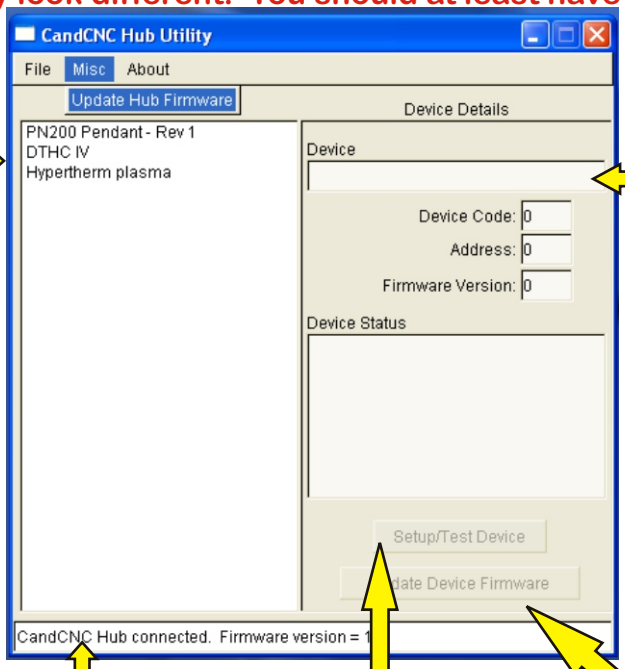
During the install the CandCNC Hub Utility was added and an ICON was placed on your desktop. Click on the icon to open the Hub Utility.

IMPORTANT: The purpose of the HUB UTILITY is primarily to confirm proper communication with each device and to be able to do some base level testing . **You cannot run the Hub Utility at the same time as MACH is running.**

The following screens show the CandCNC Hub Utility displaying information about different devices connected to it. The Hub is "smart" and automatically detects any compatible device connected to it. Review the screens that follow and understand what each section of the screen is used for.

C3 HUB MANAGEMENT
HUB UTILITY

Depending on the devices you have attached your Hub Admin may look different. You should at least have the DTHCiv Device



DEVICES:

Shows a list of connected devices. As a device is connected and sensed by the hub it will appear in the list. If you connect a device and it does not show up immediately, stop (close) the application and restart.

DEVICE DETAILS

Provides the:
Device Name
Device Code: Each CLASS (type) of device has a different Device CODE. Codes can be from 1 to 99.
Device Address: The unique address of a device within a class (values from 1 to 8). Devices of the same type must have a different Device Address. Future cards will have address jumpers to allow multiple cards of the same type to work on the same hub.
Firmware Version: This displays the current

Hub Connected
(to USB on PC)
Displays current firmware of the HUB. This MUST show connected status before any other of the functions will display.

TEST DEVICE
Action button that opens a new dialog window. You must have a device highlighted in the Devices List to use the Setup/Test Device

Update Device Firmware.
This allows you to update each module with new firmware. New firmware will be designated by a REV number. Firmware updates will be available for down load in a special download section of the CandCNC website and on the Yahoo CandCNCSupport Forum.

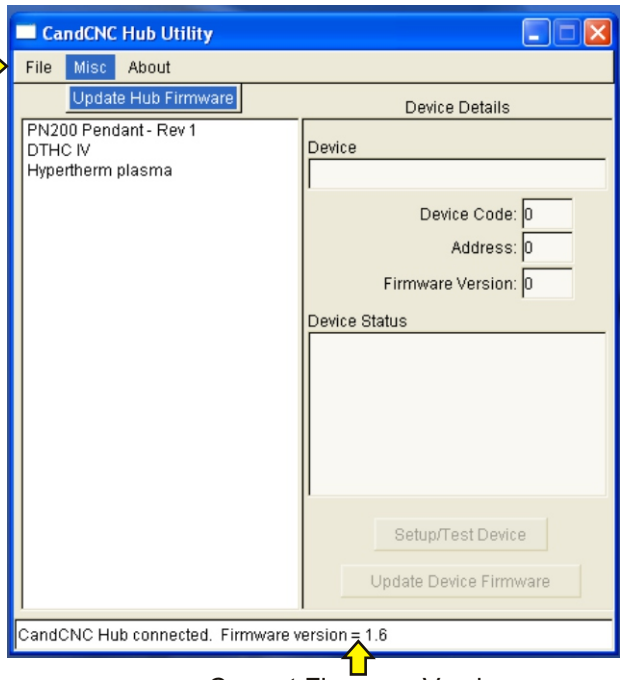
The Hub Utility is an application provided to manage the C3BUS USB to RS485 4 Port Hub and the devices attached to it. With the utility you can:

Update the Hub Firmware (Firmware is the program that runs the hub)
Update Firmware in the Devices attached (like the DTHC IV)
Test basic communications
Test device functions
Device level diagnostics
Tune the DTHCIV Z settings
Set some device settings

HUB MANAGMENT
HUB UTILITY

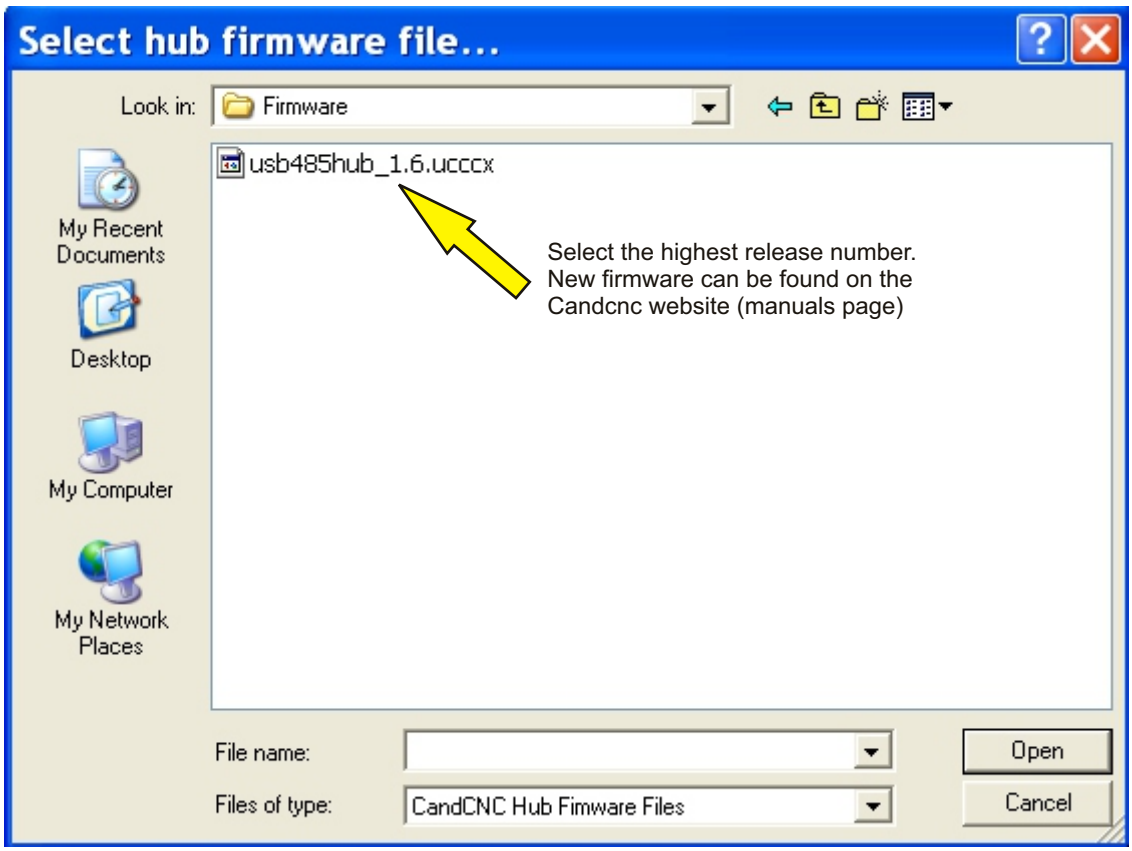
MISC:

Opens choice to update firmware. This is the first step in updating the HUB FIRMWARE.

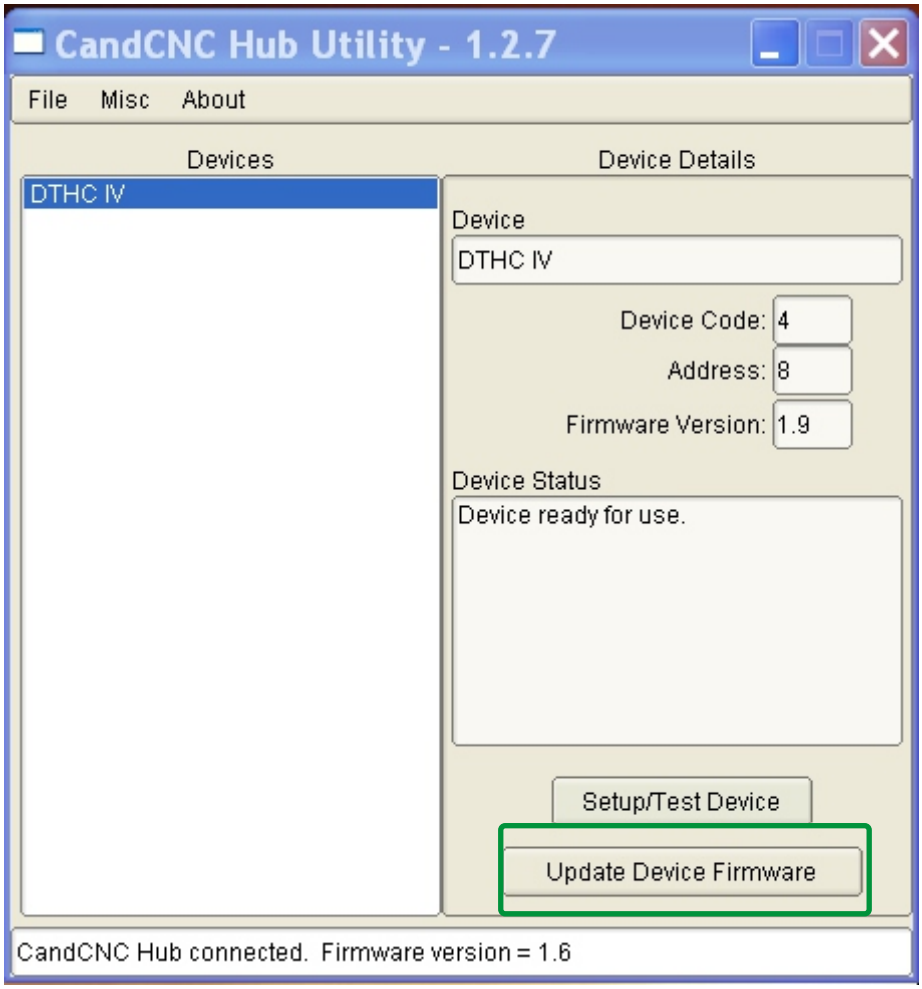


Current Firmware Version

Updating the HUB FIRMWARE is done from the MISC top menu and the *Update Hub Firmware* selection. It will open a File Explorer window and you should navigate to the USB-485HUB4/Firmware and select the highest REV number if the Current Firmware Version is the same as the highest it does not need updating

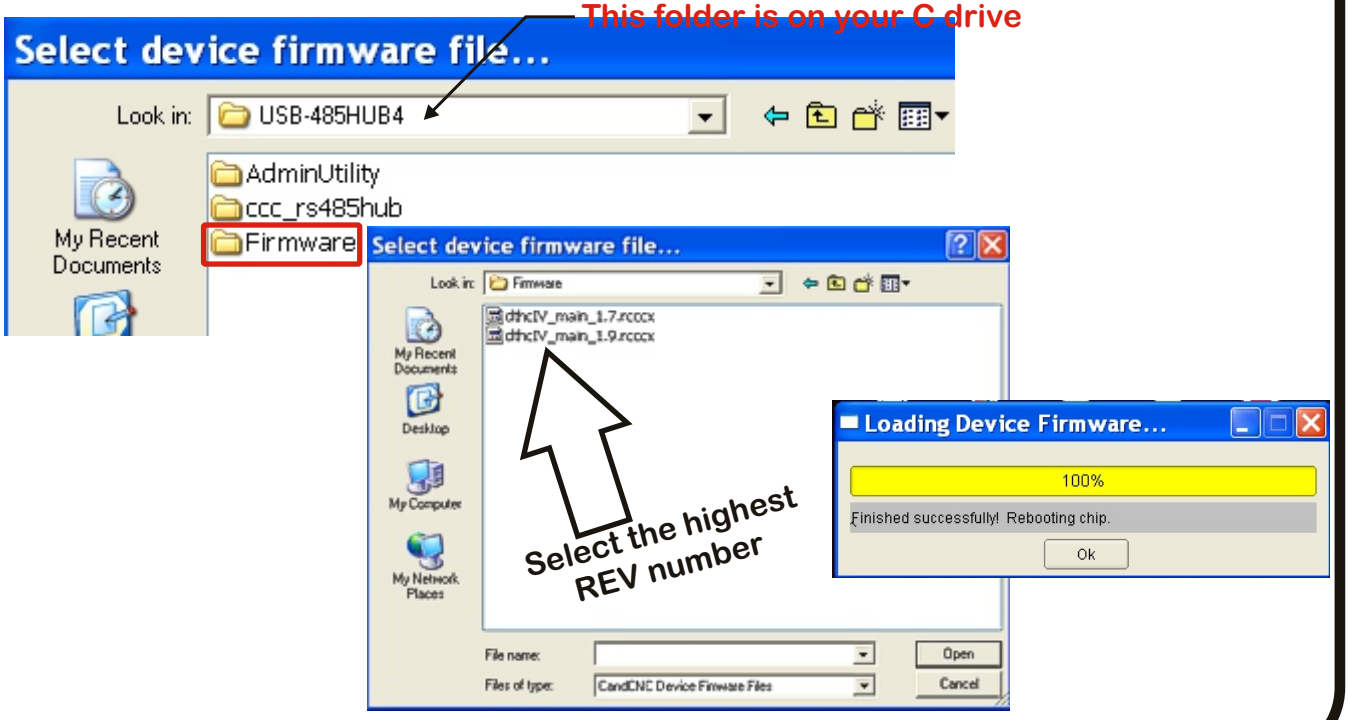


C



Updating a DEVICE Firmware is done by first selecting the device (Highlight with cursor) in the Devices List then hitting the ***Update Device Firmware*** button. You will get a File Explorer window. Navigate to the **USB-485HUB4/FIRMWARE** Folder on your C drive.

After an install or update you will see a list of Device firmware files . Select (left click) the one for the device and the highest REV number and hit the OPEN button. You will see a Loading Device Firmware window with a progress bar. When it says *Finished successfully* it will reboot the hub. Click OK and close Hub Admin and reopen it and the new device rev will be listed



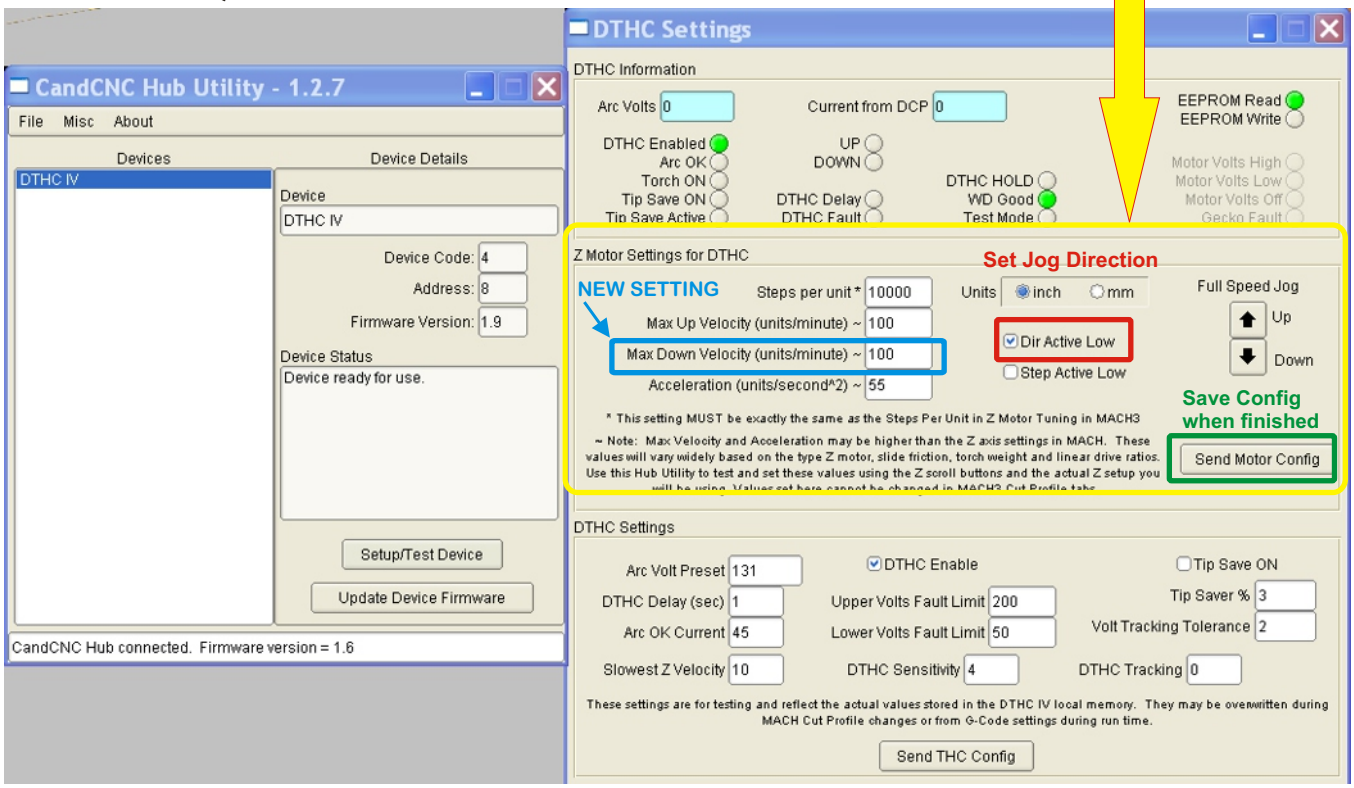
HUB UTILITY
DTHCIV SETTINGS

C

DTHC INFORMATION is direct feedback from the DTHC IV module and indicates the current status of the unit . You cannot make changes or do inputs from this section. Since you cannot run CommandCNC and the Hub Utility at the same time these settings are primarily used for off-line diagnostics.

Z DTHC SETTINGS. **This section is the most important because it sets the base motor settings for the Z motor under DTHC control.** You will find that the Max Velocity and Acceleration *could* be much higher than your settings in MACH for the Z motor tuning. HOWEVER **we suggest that you set the velocity setting to the same as the value in the MACH Z motor tuning.** (about 100 to 120 IPM in this example) It has come to our attention that some Z's are simply not designed to move at the RPM required to get the higher speeds. It also makes initial setup and trouble shooting more difficult since everything moves and happens MUCH faster. The settings here are for a stepper system with a 620 oz-in motor running a 5 TPI leadscrew. Jog the Z UP and DOWN using the Full Speed Jog buttons on the screen. The motor should move smoothly **AND IN THE RIGHT DIRECTION**. The DIR Active Low is a checkbox that sets which direction the Z moves under DTHC control. If want to test at higher speeds , Move the settings up and test using the Z motion Jog buttons on the screen. The motor should move smoothly and rapidly both up and down. On loss of steps (or faults on a servo system) back the numbers off by 25%. They will effect the Gain and Tracking settings for the DTHC. Save any changes using the Send Motor Config Button

THIS IS THE PRIMARY PLACE TO SET THE Z TUNING FOR DTHC IV . A MINIMUM INSTALL REQUIRES YOU PREFORM THIS ONE FUNCTION IN THE HUB UTILITY.

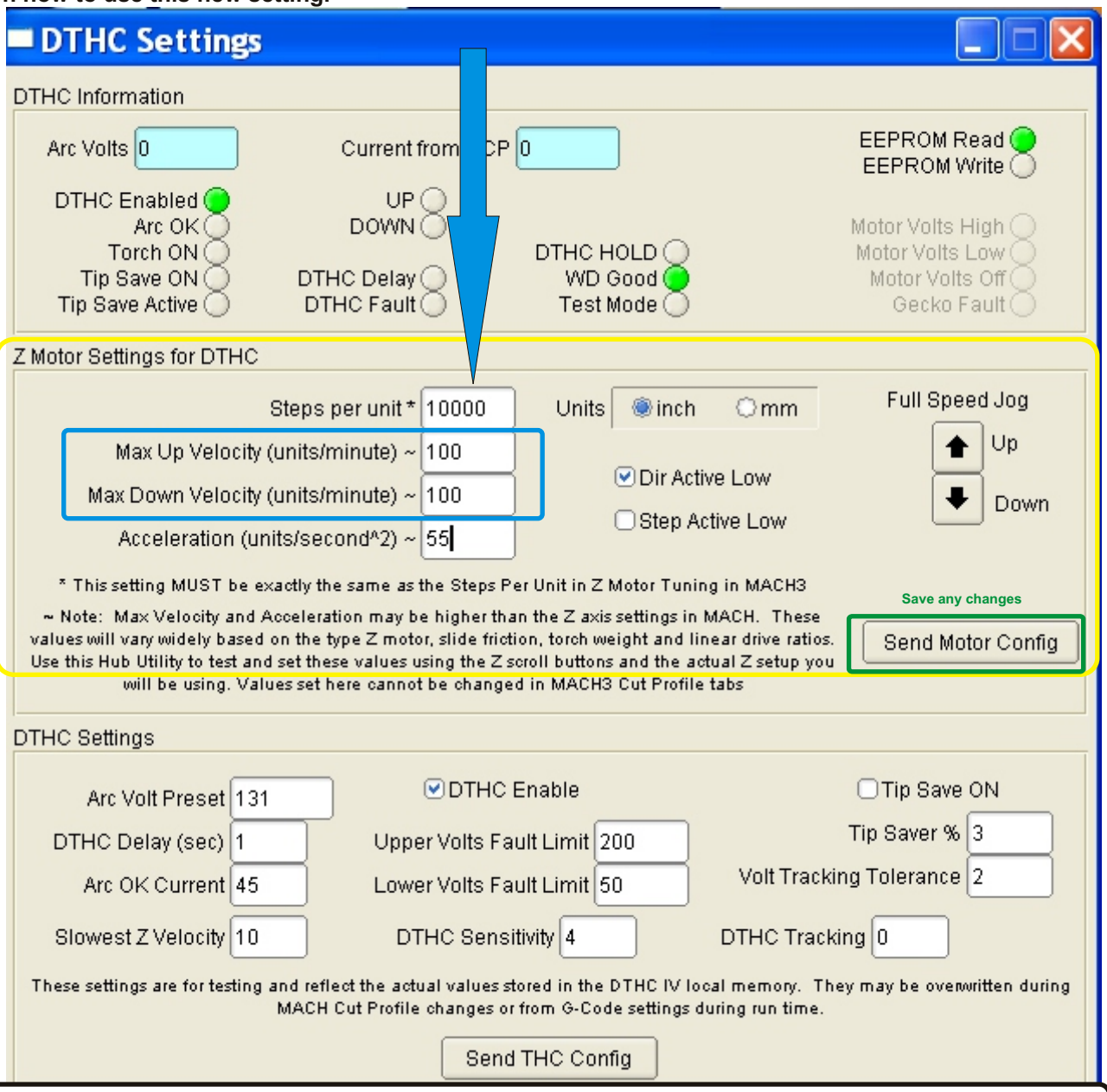


DTHC SETTINGS. These are the same as using the CUT PROFILE when MACH is running and serve to be able to test the RS485 communication to/from the DTHC IV and to test the DTHC IV memory and interface. In normal cutting these settings are provided by the Cut Profile and/or the DCC values from the G-Code at run-time. The Cut Profile in MACH now allows multiple levels of settings and is a better place to do the setup and testing under MACH.

C

Z axis tuning for proper operation:

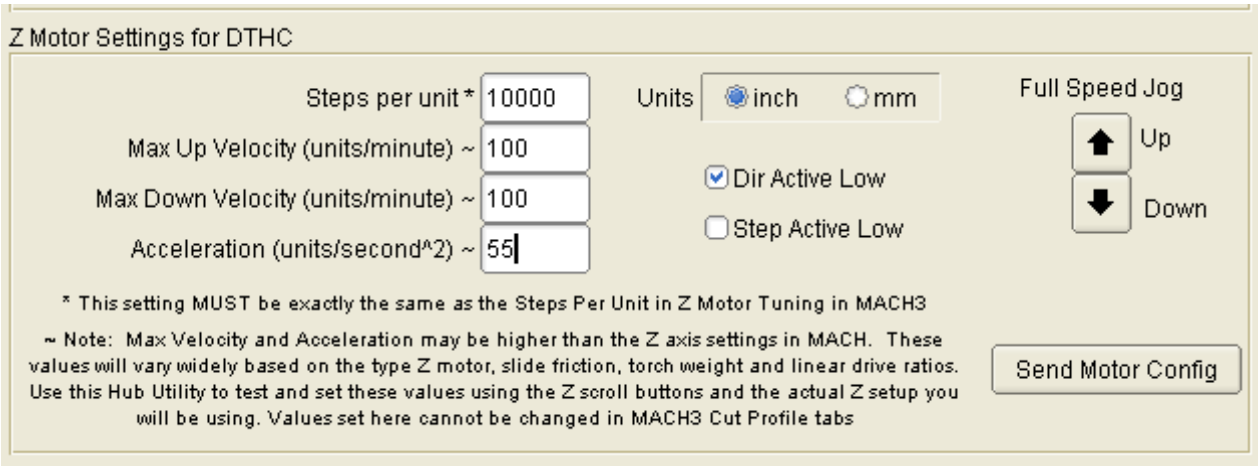
The Z motor Settings in this section determine the base settings of the Z axis WHEN THE DTHC IV (not MACH) IS MOVING THE MOTOR. When the Z is being moved by MACH from G-Code it uses the tuning set in the MACH3/Config/Motor Tuning for Z. When it is under control of the DTHC it uses the values in this section. Essentially you have two "controls" sharing the same motor but using their own rules. **The one setting that MUST be the same is the Steps per Unit. NOTE NEW IN DTHCIV FIRMWARE REV1.9 THE MAX UP VELOCITY and MAX DOWN VELOCITY ARE NOW SEPARATE SETTINGS See next page on how to use this new setting.**



The other parameters are set by starting out with the same velocity and acceleration as the Z motor tuning in MACH3 and using the Full Speed Jog Up and Down buttons to test the Z motion under DTHC control. **NOTE: If your motion is opposite the buttons, change the setting of the Dir Active Low checkbox. Failure to do this will cause the DTHC to do really wild motion in the opposite direct as it should. For initial setup and testing you should keep the Velocity and Acceleration settings for the DTHC low. In most circumstances these settings will work well on cutting flat stock with normal warping. Later** You can raise the velocity and acceleration to increase performance for cutting really thin material or corrugated.

C

THIS SECTION OF THE HUB ADMIN DTHC SETUP IS THE MOST IMPORTANT to get acceptable DTHC action during a cut.

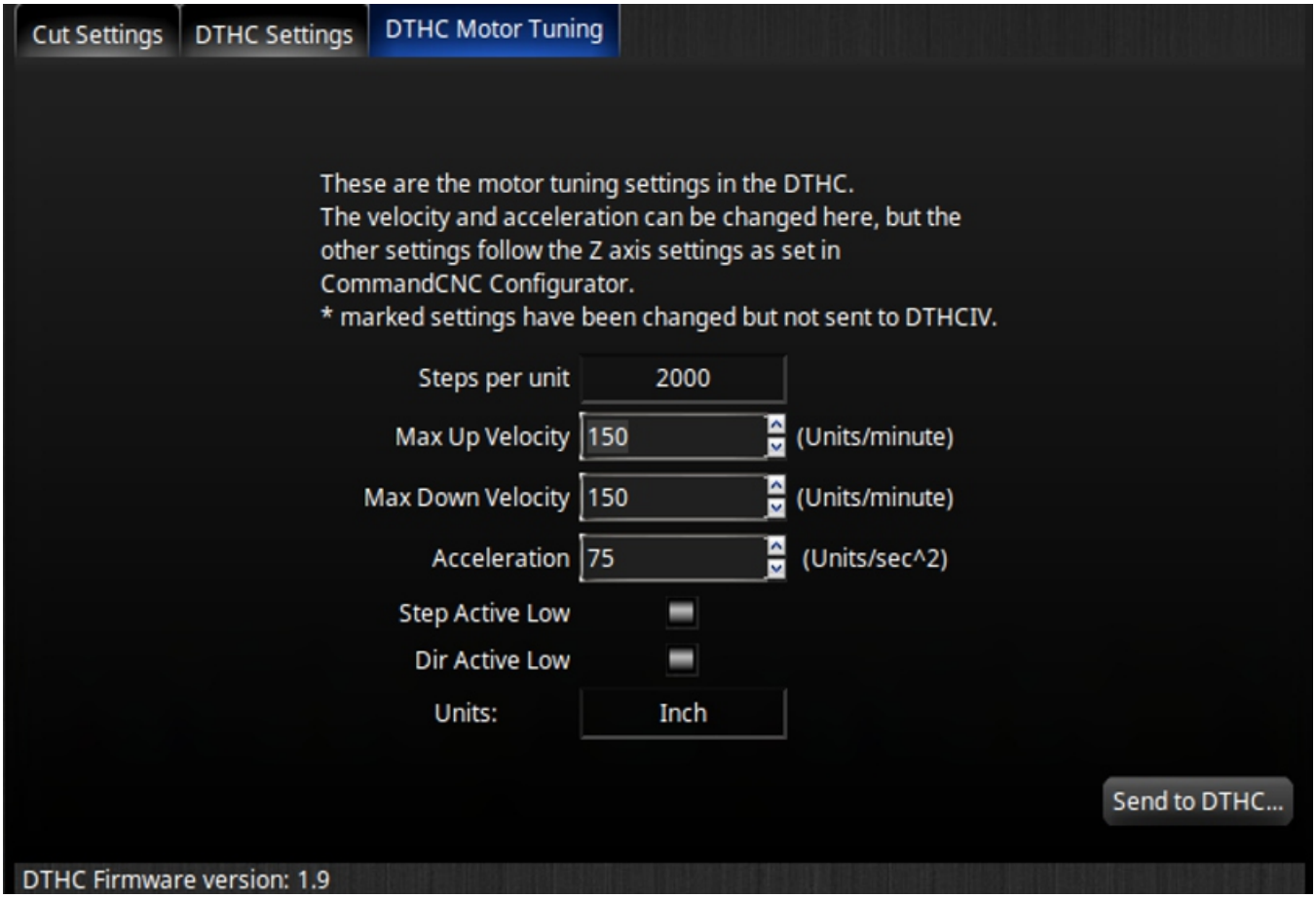


In the previous pages the settings that effect Z motion under DTHC control have been listed. IT IS IMPORTANT to understand that the motor tuning in MACH APPLIES TO NORMAL Z MOVES WHEN THE **DTHC IS NOT ACTIVE**. This is anytime it is turned off in the screen or from the G-code. All moves for touch-off , pierce height and plunge to Cut Height, are handled by G-Code and CommandCNC via normal Motor tuning. THE SETTINGS ABOVE ARE ONLY USED WHEN THE DTHC IS ENABLED. ACTIVE AND NOT IN A DTHC DELAY PERIOD. That would be after the torch fires and after the code enables the DTHC (usually when the torch reaches the Cut Height) AND the DTHC is turned on in code AND after the DTHC delay time. TUNING IN THE SCREEN ABOVE ONLY EFFECTS HOW THE Z MOVES (even which direction) when the DTHC has control of the torch axis (Z)

ABOUT THE MAX VELOCITY UP and the MAX VELOCITY DOWN.

in prior versions of the firmware and utility before REV 1.9 you only had the choice of *Max velocity* for Z along with *Acceleration* and the *Steps per Unit*. In this version we have split the UP velocity and DOWN velocity into two separate settings. This allows you to better tune the motion of the DTHC. Cutting most sheet and plate material it is generally better to have a FAST UP and a SLOWER DOWN response. It helps with voltage spikes that sometimes happen while cutting, changing direction or slowing down. The exception to this would be if you are cutting corrugated or perhaps diamond plate where both up and down need to be fast. If your DTHC is cutting good with the old settings just make UP and DOWN the same speed. If you have occasional torch dives during a cut (after the pierce and DTHC Delay time) you should move the DOWN velocity to ½ the Up velocity as a test. even as low as 20% of the UP velocity might be used on some material.

DTHC Motor Tuning. These are the same as using the Hub Admin application with the exception of you cannot change the Steps Per Unit. The UNITS (of measurement) or the Step & Dir Active low. Use this screen to fine tune the UP and Down Velocity and Acceleration AFTER you have done the Primary Motor Tuning in Hub Admin and have tested the motion using the Fast Jog Buttons in Hub Admin. This screen in CommandCNC allows you to quickly make changes if needed for a different kind of cut. The Settings in DTHC Motor Tuning in CommandCNC reflect the CURRENT settings and if changed , override the settings done previously in Hub Admin.



MODULES SETUP

DTHCIV MODULE

PWM MODULE

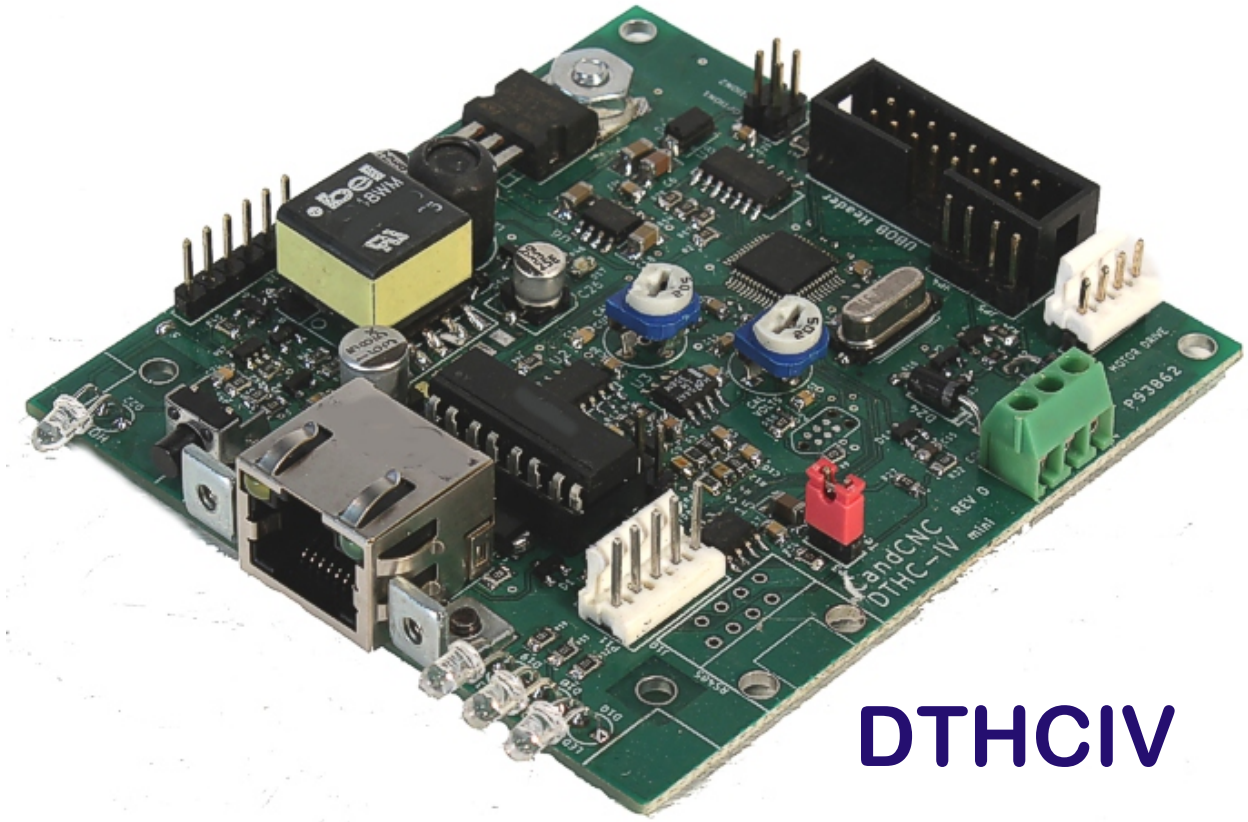
RAV-01/02 VOLTAGE DIVIDER (OPTION)

THE FOLLOWING SECTIONS DEAL WITH SETTING UP EACH MODULE.

IF YOU HAVE A CandCNC Product you may not need to use one or more of these sections since some products have theses modules already installed, tested and calibrated.

If you have a BLADERUNNER AIO DragonCut or any MP3000-DTHCII based product (Plazpaks) than YOU CAN SKIP THE SECTION ON INSTALLING AND SETTING UP THE DTHCIV Module.

D



DTHCIV

In most circumstances the User can skip this section. The DTHCIV is already installed and tested and only needs to be consulted if the user needs to

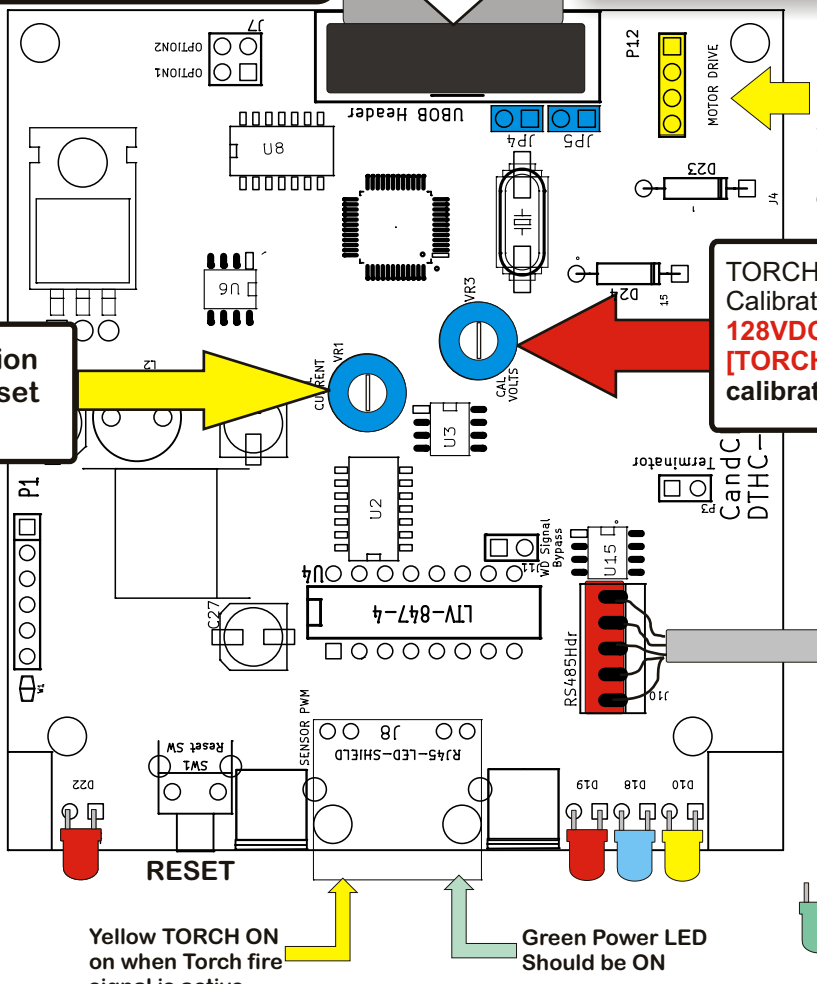
DTHCIV MODULE

D

DTHCIV EXPANSION MODULE. Located above the UBOBIII card in most CandCNC products. Ribbon cable connecting DTHC down to UBOB card may cover adjustment pots. If so, gently move it out of the way. Do not unplug the ribbon cable or the card will be disabled.

16 pin header (plug) for DTHC to UBOB Cable

All LINUX BladeRunners , Plazpaks and MP3600-DTHCIV Units have this module already installed and IT IS CALIBRATED. DO NOT ADJUST POTS.



Four wide cable is for UBOB cards prior to REV15.

Set JP4 and JP5 ON (jumpered) for REV15 UBOB . OFF for earlier versions.

DCP Calibration POT Factory set for 100A test

TORCH VOLTS Calibration **SET FOR 126-128VDC INDICATED [TORCH VOLTS] Unit is calibrated at the factory.**

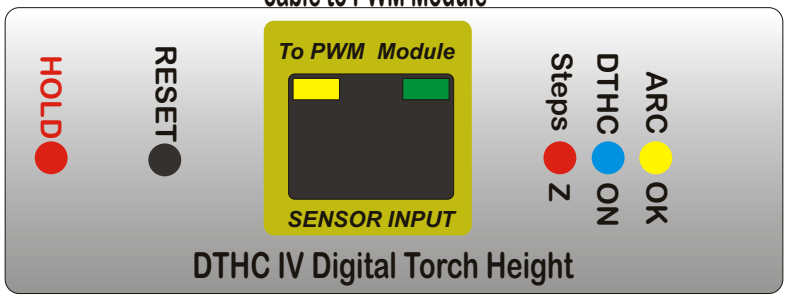
FRONT PANEL RS485 JACK

RESET

Yellow TORCH ON on when Torch fire signal is active

Green Power LED Should be ON

RJ45 connector for CAT5 cable to PWM Module



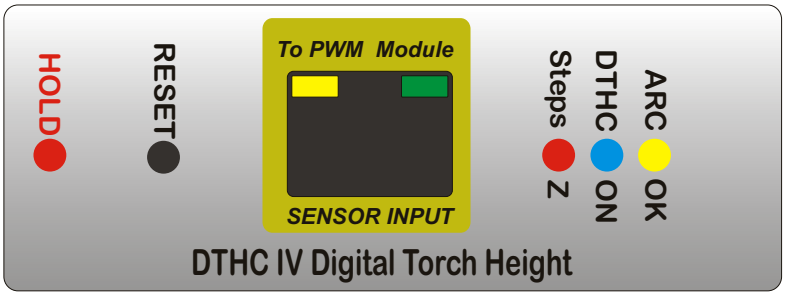
PRELIMINARY TESTING THE DTHC MODULE:

The only preliminary tests of the DTHCIV module are for power getting to the board and to test the RESET button and check for the initializations sequence on the front panel LED's.

1. Power up the BladeRunner/MP3600/Plazpak and watch the front LEDs on the DTHCIV. They should turn on one at a time from right to left starting with the ARC OK LED, then all LEDs should flash twice. This indicates the microprocessor in the DTHCIV is live and working.

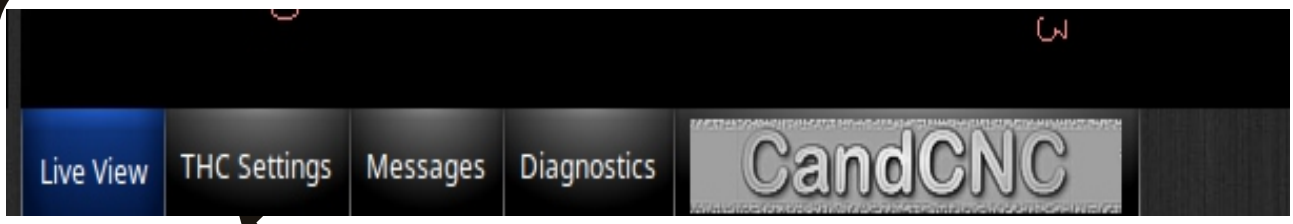
2. **An integral part of the DTHCIV communications to and from MACH are dependant on the C3Bus USB to RS485 4 port Hub. A lot of testing and DTHCIV settings are done via the HUB UTILITY through the C3BUS.**

Front Panel Indicator Identification



SIGNAL NAME	COLOR	FUNCTION (What it means)
ARC OK	YEL	Shows the torch is on and that there is a valid arc so automated Cutting can begin. Signal comes from Plasma (PWM Module) or from DCP-01 current reading.
DTHC ON	BLU	The DTHC can be turned on and off in software and from the screen. It shows when the DTHC is active.
STEPS Z	RED	Shows Z motion activity from both DTHC and MACH control
RESET	Button	RESET button resets the DTHCIV processor and does an initilization Pattern for self check
HOLD	RED	Indicates the DTHCIV has issued a HOLD command back to MACH to prevent motion
RJ45 RT	GRN	DTHCIV POWER INDICATOR On all of the time
RJ45 LFT	YEL	TORCH ON. Shows torch FIRE signal from MACH3

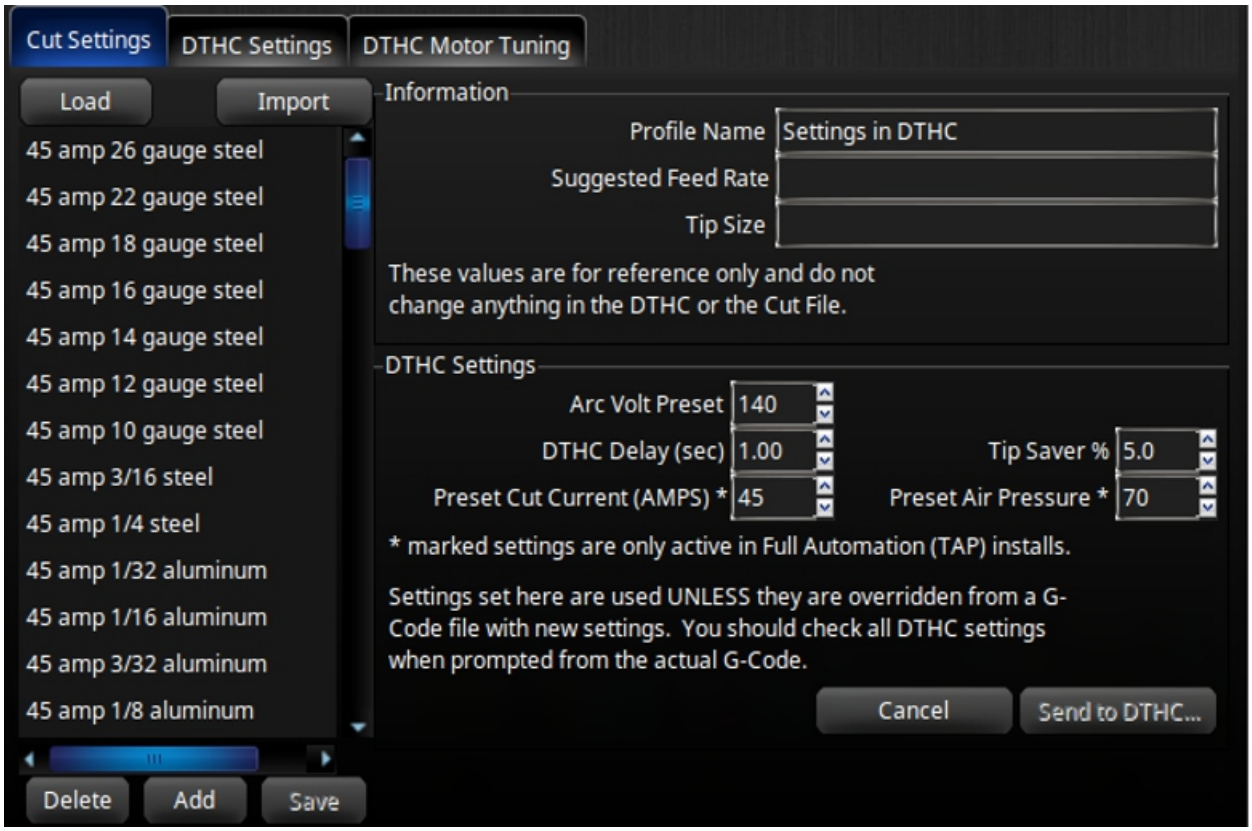
CUT PROFILES
Local THC Settings



How to access the CUT PROFILES (Stored Settings) in CommandCNC

Use the Menu Tabs at the bottom of the Upper Right window and select the THC SETTINGS button. You will have the following window appear and the Cut Settings tab will be open.

NOTE: Cut Profiles are LOCALLY stored Presets for DTHCIV settings . It is NOT the same as the Automatic Settings that come from SheetCAM and OVERRIDE local settings when the G-Code is loaded .

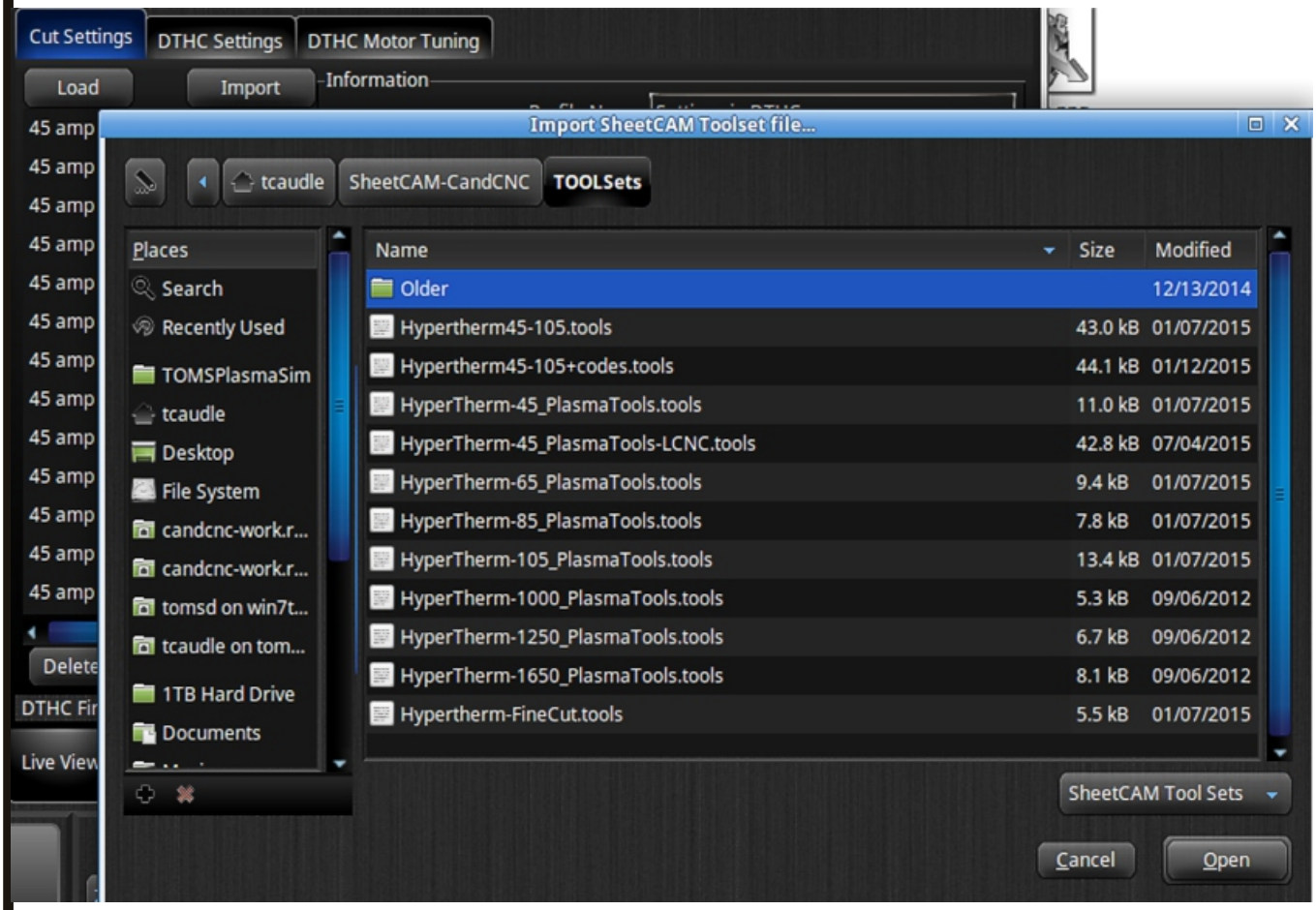


WARNING: Unless you are in DEMO (simulated) MODE, If you do not have the C3BUS Hub setup and working and the DTHCIV shows up in the device list, THIS BUTTON DOES NOTHING!
Without communication between CommandCNC and the DTHCIV you cannot read or change settings.

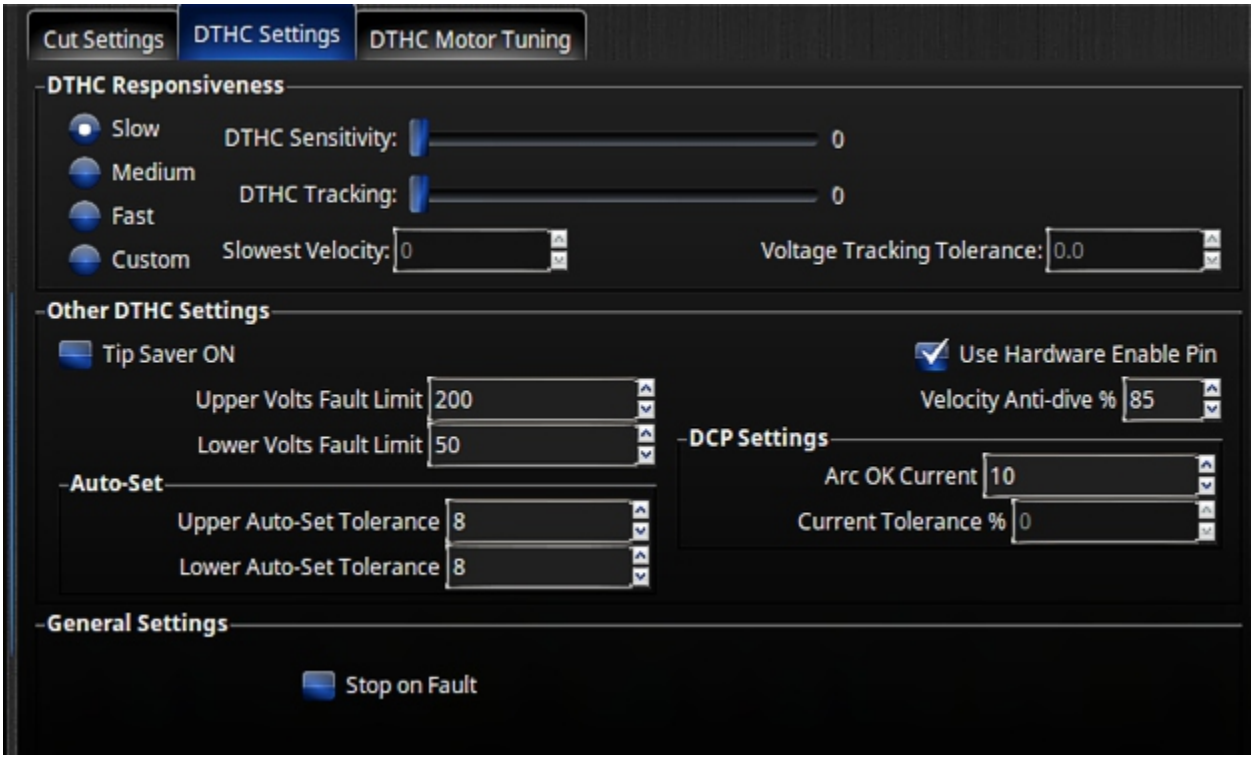
D

Adding/changing Toolsets by importing SheetCAM defined toolsets.

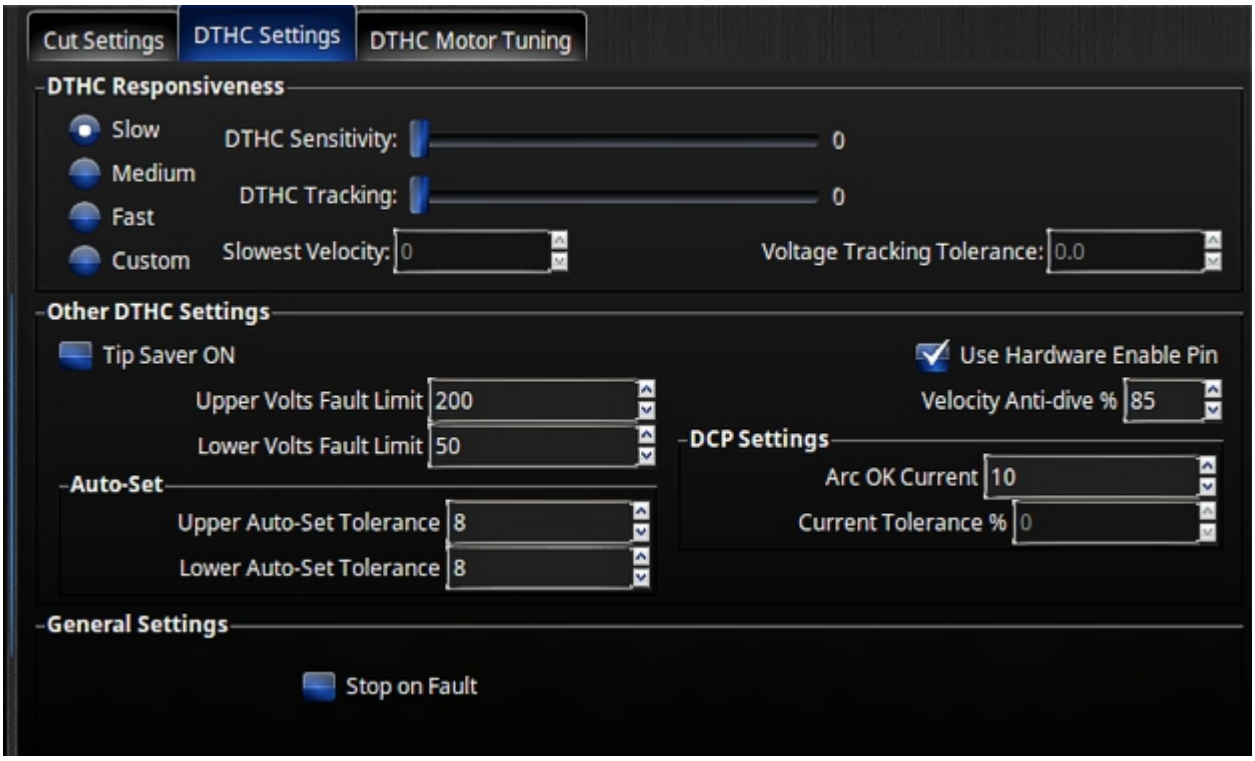
The LOCAL CUT PROFILE can import toolsets from SheetCAM toolsets. They are stored in the local Folders under <user>/SheetCAM-CandCNC/Toolsets. This does not add those toolsets to SheetCAM and any changes made locally DO NOT get saved back to the SheetCAM toolset .



If you elect to use the internal Cut Profile rather than the Automated settings from SheetCAM You can still use the pre-defined toolsets for SheetCAM. Click the IMPORT tab and use the screen to naviagte to the folder that holds the SheetCAM toolsets and you will be presented a list of toolsets to choose from. Double Clicking a toolset will load it into the Cut Profile and allow you to select those preset values and /or edit and change any value/ NOTE: Once you make a change to a toolset in CommandCNC it DOES NOT make the change in the master file you imported from. CommandCNC imports a COPY of the toolset and you can then use it locally and save and reload it locally.



DTHC SETTINGS TAB opens up a screen with 3 sections. Each one has specific functions that will be covered in the following pages. The DTHC Settings effect the Operation of the DTHCIV and are global and not by Cut Profile (previous TAB). These settings remain in effect until they are changed in this menu.



SELECTING DTHC RESPONSE PRESETS

There are four DTHC Response presets for different types of cutting conditions. There are three for common Presets and one for Custom

Slow Preset: Is for general flat sheet cutting of material that does not tend to warp more than about a ½"rise over 2 inches or more of length. Slow does not mean the motion is slow but rather it slows down the response to voltage change during a cut and dampens over-response to brief spikes or dips in arc gap (arc volts).

Medium Preset: Is for cutting thinner material that can warp more intensely and allows response to cut up to about a 30 deg rise at rates up to about 200 IPM XY rates*. Select this Preset if you cut 16ga and thinner and you see a lot of warping and vertical changes.

Fast Preset: Is for very aggressive cutting on severe warps, corrugated or at very high cut rates. This setting will function to cut up to a 45 deg slope at 180 IPM and smaller slopes at 350 IPM*. It will cut thin corrugated. (Some other DTHC settings may need to be adjusted.)

* It is important to understand that the above settings and suggestions are based on typical leadscrew type Z mechanisms with average **pitch** of .2 " (5 mm) for stepper and .1 " (2.54 mm) for servo. The settings may need to be adjusted for courser or finer pitches. Use the CUSTOM setting if the presets do not product smooth response.

SETTING DTHC RESPONSE USING CUSTOM Settings

If you select CUSTOM in the Preset Tuning buttons the **DTHC Sesitivity** and **DTHC Tracking** will no longer be grayed out and can be used to custom tune the Z for a given condition. You may find that custom tuning is needed if your Z mechanics are either very course or very fine.

DTHC Sensitivity determines the response RATE (how quickly the Z responds to small changes in the ARC VOLTS). Setting the DTHC Sensitivity higher will force the DTHCIV and the Z to increase the rate that it responds.

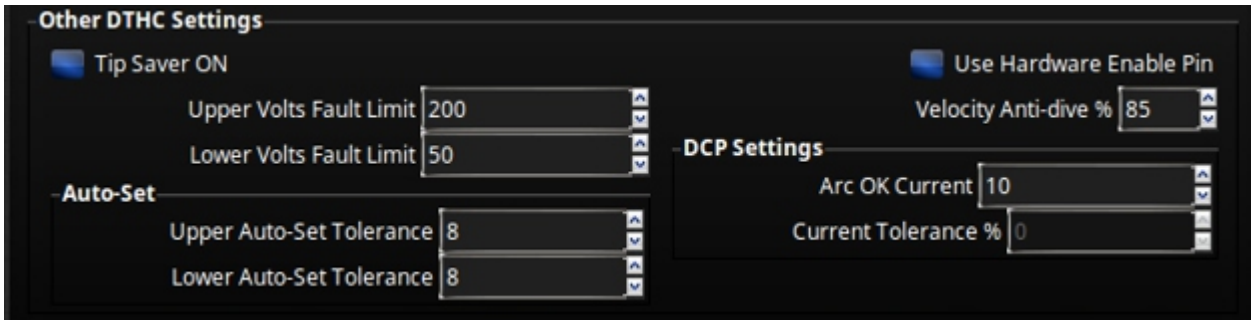
DTHC Tracking determines how close to the PRESET VOLTS the DTHCII will try and hold the changes (dampening). The resolution of the DTHCIV is 1/16 VDC. It uses a unique non-linear averaging algorithm developed specifically for Torch Height Control. Typically the two settings are interactive. If you increase one you may need to decrease the other or vice versa. Too much DTHC Tracking will cause Z oscillation (the Z will move up and down rapidly as it cuts). Zero tracking is a valid setting in some circumstances. **When making changes to either setting, do so in small increments**. Once you have a good setting make note (write it down) for both values. Over time we will develop numbers for different configurations of Z mechanics and motors and make them available in a chart.

Slowest Velocity (Default value = 10) is the slowest speed in IPM that Z will travel while under DTHC control. It the DTHC moves based on an error voltage this is the MINIMUM speed it can move at. (it moves faster the greater the error). It effects the lower end of the response curve to give better response to small amounts of change.

Voltage Tracking Tolerance. This is the voltage range that the DTHC works across from minimum to maximum velocity. It is the allowable error. This works in combination with the DTHC tracking value to establish the response curve of the DTHC. The value is in VOLTS of error and the default 1 volt setting is appropriates for most applications.

D

OTHER DTHC SETTINGS. Located in the Center of the DTHC SETTINGS TAB the 4 input boxes allow for changes in the Voltage Fault (volts out of range error code trigger) and two inputs for use if the DCP-01 is being used. (NOTE: the ARC OK CURRENT needs to be set to some value above 0 or it will false trigger on systems that DO NOT use the DCP-01).



Upper Volts Fault Limit: Set this value to about 10% below the open circuit voltage of your plasma torch (voltage when fired in the air). It will prevent DTHC signals from moving the torch too far down in the event the torch runs off the material or hits a void.

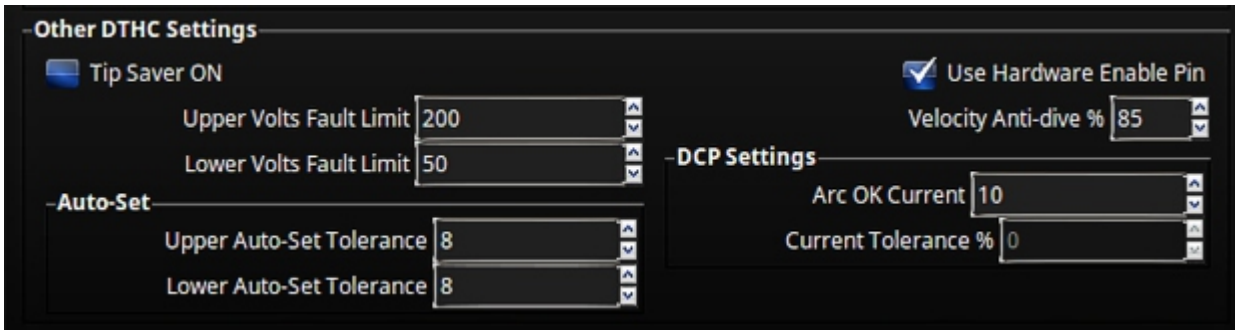
Lower Volts Fault Limit: Set this value to a number that triggers a fault if the voltage is below the normal cutting voltage of the torch with the consumables you use.

If you change consumables that have different settings you may need to adjust the parameters or you can set the lower limit to reflect the lowest voltage used with ANY type consumable.

Arc OK Current: This setting is used when the plasma unit you are using DOES NOT HAVE an ARC OK (OK to MOVE; Arc Xfr; TRANSFER, Etc) signal. The UNIVERSAL CONNECTION KIT from CandCNC has the DCP-01 Digital Current Probe that reads Arc Current (amps) flowing in the Workclamp lead and is used to trigger the ARC OK signal. If you have the DCP-01 connected and in use either in the Universal Connection Kit or as part of the TAP (Advanced Interface Kit) you should set the Arc OK value to about ½ of the lowest cut current you use. *It is important to understand that the ARC OK can be triggered by EITHER the hard signal reading coming from the plasma or from the DCP-01 reading current greater than the Arc OK Current setting*.

Current Tolerance: This setting is a percentage of the indicated Cut Current and works in conjunction with the ARC OK CURRENT and has to have the DCP-01 to monitor the actual cut current. It triggers a current fault and there are General Options that control what happens if a current fault uses the PRESET CURRENT value (not on this screen).

OTHER DTHC SETTINGS. Located in the Center of the DTHC SETTINGS TAB the 4 input boxes allow for changes in the Voltage Fault (volts out of range error code trigger) and two inputs for use if the DCP-01 is being used. (NOTE: the ARC OK CURRENT needs to be set to some value above 0 or it will false trigger on systems that DO NOT use the DCP-01). Colored Text indicates NEW or moved settings in release 0.8.5 or later



Tip Saver ON (checkbox) This enables/disables the legacy Tip Saver . Recommended setting: Unchecked(off) if you are using Velocity Anti-Dive (VAD). Note: this moved from DTHC front panel settings

Use Hardware Enable PIn (checkbox). This enables the direct high speed DTHC ON/OFF. Used in conjunction with VAD. Must be ON to use VAD

Velocity Anti-dive % (DRO) The 5 the combined feedrate must drop below for the DTHC to turn OFF then back on when it reaches 100% The deafult setting of 85% works well for normal cutting and detailed cuts on thinner material. This also works as

Upper Volts Fault Limit: Set this value to about 5% below the open circuit voltage of your plasma torch (voltage when fired in the air). It will prevent DTHC signals from moving the torch too far down in the event the torch runs off the material or hits a void.

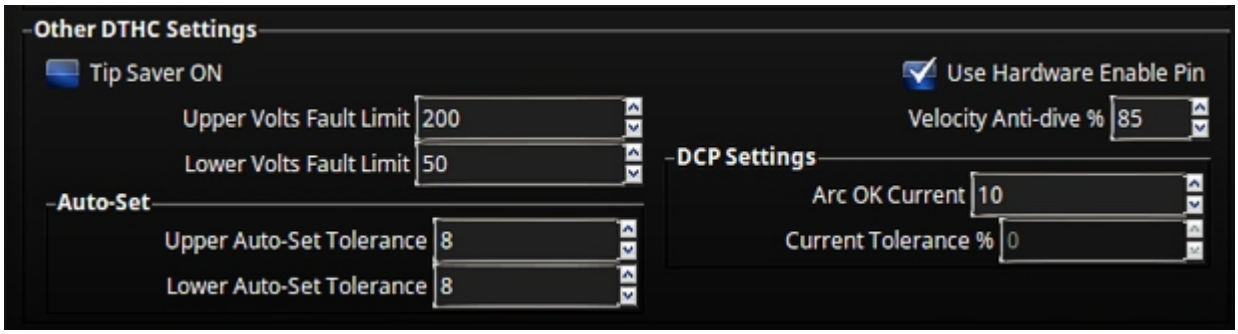
Lower Volts Fault Limit: Set this value to a number that triggers a fault if the voltage is below the normal cutting voltage of the torch with the consumables you use.

If you change consumables that have different settings you may need to adjust the parameters or you can set the lower limit to reflect the lowest voltage used with ANY type consumable.

Arc OK Current: This setting is used when the plasma unit you are using DOES NOT HAVE an ARC OK (OK to MOVE; Arc Xfr; TRANSFER, Etc) signal. The UNIVERSAL CONNECTION KIT from CandCNC has the DCP-01 Digital Current Probe that reads Arc Current (amps) flowing in the Workclamp lead and is used to trigger the ARC OK signal. If you have the DCP-01 connected and in use either in the Universal Connection Kit or as part of the TAP (Advanced Interface Kit) you should set the Arc OK value to about ½ of the lowest cut current you use. *It is important to understand that the ARC OK can be triggered by EITHER the hard signal reading coming from the plasma or from the DCP-01 reading current greater than the Arc OK Current setting*.

D

OTHER DTHC SETTINGS. Continued



Current Tolerance: This setting is a percentage of the indicated Cut Current and works in conjunction with the ARC OK CURRENT **and has to have the DCP-01 to monitor the actual cut current.** It triggers a current fault and there are General Options that control what happens if a current fault uses the PRESET CURRENT value (not on this screen). This is not used with plasma systems that have an internal ARC OK (Ok to Move/Transfer) signal

General Settings Global Settings that effect all operations

Stop on Fault When checked this causes the motion to stop when there is a fault. Typically a fault from the DTHC is a voltage out of range fault and stops the DTHC moving the Z up or down but does not stop motion. Recommended setting is UNCHECKED.

SETTING UP THE THC SENSOR PWM MODULE

IF you already have a working PWM Module with your DTHC II of either REV number and you are just upgrading to the DTHC IV you can skip this section of the setup

PWM MODULE
REV 21 and later

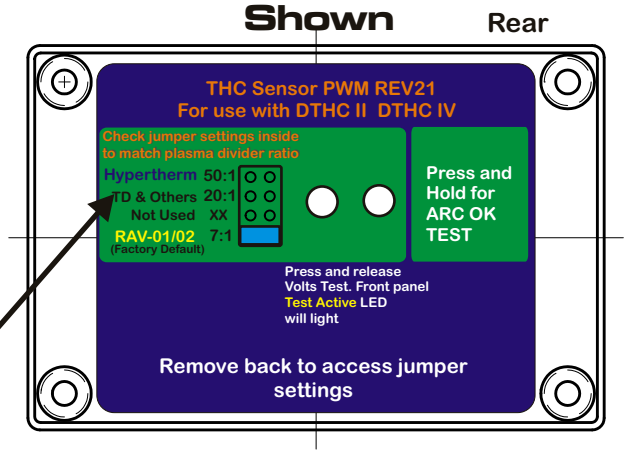
The THC SENSOR PWM MODULE (PWM Module) has been redesigned to fit into a smaller case. The function and settings are EXACTLY THE SAME. If you have a working REV 20 or sooner module your DO NOT need to upgrade it to a newer REV 21. This manual is written for using either module and the setup and testing is the same . Certain indicators and button locations have been moved.

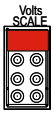
THIS MUST BE DONE TO MATCH YOUR INSTALL UNITS ARE SHIPPED TO MATCH THE RAV-01 (7:1) RATIO. USE THE CHART BELOW IF YOU ARE USING THE MODULE WITH A PLASMA WITH AN INTERNAL VOLTAGE DIVIDER. SEE THE SECTION ON CONNECTING A PLASMA WITH AN INTERNAL VOLTAGE DIVIDER TO DETERMINE TE PROPER RATIO

To set PWM SENSOR ratio divider option:

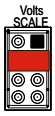
1. Remove rear cover of THC SENSOR PWM Case
2. Determine the proper setting for the type setup you have.
3. Set the small option jumper to match.

NEW MODEL REV 21 Shown





50:1 Scale (Hypertherm and some others) Recommended for Hypertherm with CPC

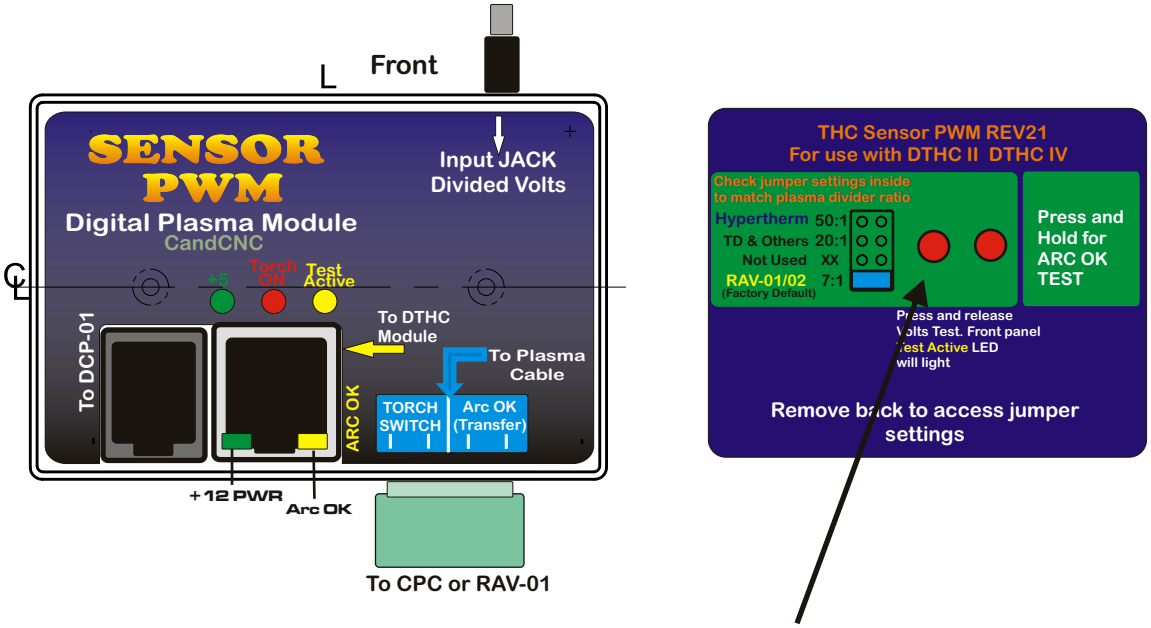


20:1 Standard on some Thermal Dynamics, others

7:1 Divider (from CandCNC RAV-01 Card) DEFAULT

IMPORTANT:
The Hypertherm 45 has an internal voltage divider. It cannot be changed from the 50:1 ratio. Set your PWM Sensor jumper to the 50:1 position if you are using the MIC-01 Direct Connect Cable.

THC SENSOR PWM MODULE TEST/CAL



THC PWM Module Rev 21 (Plasma Pickup Module) can be put in TEST/CAL mode without removing the card from the case.

- Turn the case over and look on the back. There are two recessed test buttons find the button marked VOLTS.
- One push puts the unit in test/cal mode. The Test Active on the front LED will flash. You should see 126 to 128 volts displayed on the TORCH VOLTS in the MACH screen
- A second push will take the unit out of test/cal mode and the LED will stop flashing and the TORCH VOLTS reading will return to Zero.
- Once in TEST mode you can calibrate the DTHCIV module to display the correct TORCH VOLTS on the screen. See the DTHCII module section to in the front part of this manual to find the calibration points. The THC SENSOR PWM is factory calibrated. Do not try to set the calibration if the test voltage is MORE THAN A FEW VOLTS OFF THE TARGET OF 126. It will not fix problems with reading the torch volts or if it displays wrong when the torch is fired!

The calibration is based on a simulated PWM from the circuit ***so if the prescale divider setting is wrong the calibration will show correct but when the torch is fired the voltage will be wrong.***

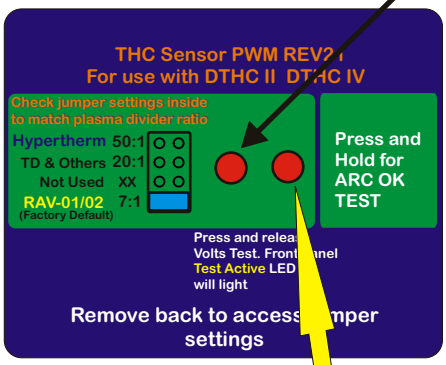
THC SENSOR PWM MODULE VOLTS TEST/CAL

Depress and release TEST button to test PWM circuit back to DTHC II. Test LED on front will flash and DTHC screen in MACH should display a voltage. Calibrated units (see calibration section) should display 126 to 128 volts during test.

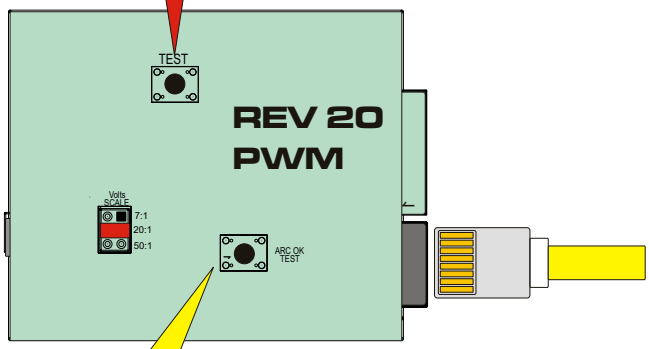
CandCNC THC SENSOR PWM Card REV 20

REV 21 PWM

CandCNC PWM SENSOR Card REV 21



Case not shown for clearity



CHECKING ARC OK SIGNAL BACK TO DTHC II CARD:

1. Remove case bottom. (rev 20 version)
2 With RJ45 (UTP cable) connected to DTHCII and unit powered up, depress the ARC OK TEST BUTTON on the card. The ARC OK LED on the MACH screen should light. The ARC OK test LED on the front of the case will light.

NOTE: rev 21 module has rear access holes for test buttons but the back must be removed for setting the JUMPERS

Screen may be different

THC SENSOR PWM MODULE VOLTS TEST/CAL

Before you make connections to the plasma unit you may want to do some testing to confirm proper operation of the THC SENSOR PWM with the DTHC II/ IV Module.

☑ **Connect the THC SENSOR PWM (or PWM MODULE REV 21) to the RJ45 (CAT5) connector on the DTHCIV Module.** **Be careful! Do not connect the CAT5 from the DTHCIV module to ANY OTHER RJ45 jack. It has voltages that can damage other modules (like the USB to RS485 4 port hub or the Ethernet jack on the controller or your PC**

☑ **Start CommandCNC using the Desktop Icon Plasma Profile** and make sure you can come out of RESET and that the CP (Charge Pump) LED on the front of the UBOB/MP3600/ BladeRunner is ON.

☑ **On some full controller products you have to have the Motor DC on to come out of RESET.**

☑**Click on the TORCH icon on the screen.** You should see the LED above the TORCH button on CommandCNC turn on and there will be a click in the THC Sensor PWM card and the small LED on the front labeled TORCH ON will light. That indicates the TORCH ON relay is working.

☑The next check is to confirm the ARC OK circuit is working. **Follow the instructions for the THC SENSOR PWM section and open the back and use the ARC OK Test button.** The ARC OK LED on the MACH3 Screen should light. If it does you can proceed to the actual hookup of the THC SENSOR PWM Module to your plasma unit.

☑**If any of the tests fail make sure you have the cables firmly attached and that they are the correct type.**

All cards are tested at least twice and most three times before they leave the factory. It's unusual for a THC SENSOR PWM to be bad or fail in no load testing. If you have checked all of the connections, cables and MACH setup and you still cannot get the THC SENSOR to work contact us at 903-364-2740 or via e-mail at Tom@CandCNC.com

NOTE: Some Larger (>100A) plasma units or older smaller models use various methods to start the initial ARC. Most common is HF (High Frequency) start. HF Start presents several challenges. It uses the concept that higher frequency waves travel through air (and arc) easier than DC voltage. The HF is normally combined with a higher voltage and it starts an ARC that the plasma uses to ignite the air. Once the arc fires, if a conductive part is close, the arc will transfer to the material. The HF start causes a lot of noise and current spikes. The other form of High Voltage start is the CD (Capacitor Discharge) method. It is basically a high current version of an Automotive ignition system. Up to 30,000 volts can be generated. If the THC Sensor is not protected, the high voltage and high frequency can cause component failure on the card or (worse) in the THC unit and even burn the board. The THC Sensor PWM (REV18 and up) is protected from HF and most High Voltage start circuits.

MAKING CONNECTIONS TO YOUR PLASMA CUTTER

1. You need the proper CandCNC Plasma Connection Kit to make the interface to your plasma cutter.

2. Different types of plasma cutters can be used. Some have everything needed terminated to one connector (e.g. Hypertherm 45/65/85/105/125) with rear CPC. Others may have the same or no external connectors. We can only cover a few specific types.

NOTE: Some plasma cutters use a manual contact start (TAP start) when the end of the torch has to touch the metal then be retracted to start the arc. These types of cutters WILL NOT WORK for plasma CNC cutting.

A note about HF start plasma cutters and the DTHCIV.
The DTHCIV is highly isolated from the plasma side and the noise from an HF start plasma will not enter the controlls thought the PWM module or the DTHCIV. **Most problems from HF start are from having a common connection to the control side (motor power supply, PC or monitor, etc)** The table and plasma chassis needs a good local earth ground (rod) close to the table. You need to separate the Plasma cutter and its cables as far as possible from the controls. You should NOT tie the control side to the same ground rod as the plasma or table is connected to. The AC safety grounds between the two systems need to connect at one common point (only) and as physically separated as possible. In severe cases you may need to provide a separate clean ground to the control side or even place the control equipment in a shielded metal enclosure NOT connected to the plasma ground. Even earth grounded wires with high currents running through them will induct some noise into anything tied to the same ground. Just a few volts of noise on a shared ground can disrupt the operation of some PC's. If you fire your torch and the PC locks up , MACH3 stops responding, or the controller does weird things, you need to start separating the source of the noise (the plasma cutter and the torch leads) away from the PC control side and break any LOCAL common connections...even grounds.

The smaller Hypertherm and other modern brand units use a low noise method called “blowback arc start”. The electrode is mounted against a spring that keeps it pushed against the inside of the nozzle as long as air is not flowing. When the unit is triggered the ARC starts a few milliseconds after the current starts to flow in the electrode circuit. As the air flows it pulls the electrode away from the nozzle and creates an ARC. That is used to ionize the air and start the plasma.
The DTHC IV can be used with all types of plasma units. The HF units tend to be very noisy and some even have large amounts of RFI. The total isolation of the DTHC IV circuit from any low level (PC logic) including any common ground, stops any conducted noise. The internal circuits are protected from RFI with proper layout and careful attention to bypass components on all active circuits.

Hooking Up Your Plasma Machine to the MP3600-DTHC IV/BladeRunnerAIO DTHC IV LINUX

NOTE: IF you have a Hypertherm with a rear “CPC (14 pin) round connector you can skip this section.

CAUTION: Portions of this install may include opening your plasma cutter machine and attaching wires. **MAKE SURE THE UNIT IS UNPLUGGED PRIOR TO REMOVING ANY COVER(S) OR MAKING ANY CONNECTIONS**. Plasma units have HIGH VOLTAGES present that can be dangerous or lethal. IF YOU ARE NOT EXPERIENCED WORKING WITH HIGH VOLTAGES, DO NOT ATTEMPT TO INSTALL THIS OR ANY OTHER DEVICE INSIDE YOUR PLASMA UNIT YOURSELF. SEEK PROFESSIONAL HELP.

In order to control your plasma unit, there are three main connections that need to be made to the plasma unit itself. All of the following operations are to be done with the power disconnected from your plasma unit.

You should determine which type install you will need for your plasma.

There are 3 questions that need to be answered:

1.) Does your plasma unit have an internal ARC OK (dry contact) signal or one on a standard CPC connector? If not then you will need to purchase and install the Digital Current Probe Option (DCP-01). it is available separately or as part of the Universal Connection Kit

2.) Does your plasma unit have an internal voltage divider (Automation Interface) with a ratio of 20:1 or 50:1 ? If not you will need to purchase and install the Raw Arc Volts divider card (RAV-01). It is available separately or as part of the Universal Connection Kit

3.) Are you using a hand torch or machine torch? If using a hand torch (even with a unit that is setup for automation, you will probably need to tap into the TORCH SWITCH wires from the hand torch to fire the torch remotely (from the computer). There is a page on how to do that from either the RAV-01 (if you already have it because of #2 above) or directly from the THC SENSOR PWM connector.

If you have a Plasma unit that needs the RAV-01 card you will need to install that card in your plasma unit or have it done. See the RAV-01 card section for instructions and warnings. If you are using the DCP-01 for ARC OK, there is an addendum at the end of this manual on installing and testing the DCP-01.

NOTE: IF you have a Hypertherm with a rear "CPC (14 pin) round connector you can skip this section.

1. Most plasma units have connection terminals where wires from the torch or panel connectors attach to the internal PC Boards. The terminals provide a convenient place to do your connections. Use crimp-on spade or round terminals to attach the wires to the terminal strips. Make sure the new wires you install do not touch adjacent metal objects. On some machines there may be more than one set of small wires and are used for sensing tip shorts and other conditions. **To identity the correct pair for the Torch Switch** use an ohmmeter or continuity checker across each pair while you manually push the torch head button. When you identify the pair make note of where they attach. Use #22 to #18 stranded wire (twisted pair) to connect between the two screw terminals on the THC Sensor PCB marked "Torch Switch" to the two switch terminals in the plasma unit.

There is no polarity. NOTE: IF your unit has noise filter chokes from the torch switch wires up to its internal logic card, it is recommended you place the two wires to the RAV-02 PCB on the other side of the chokes from their torch head connection (end closest to the internal logic card).

2. If your unit has a Raw Arc Voltage connection point (i.e. like the Hypertherm 1000 series), you will need to use their manual and suggestions as to how to connect to the two points and run those wires to the RAV-02 card. Just make sure you use wire that has insulation rated for at least 400 V. Small signal wire like telephone wire (UTP) is not rated that high and can arc to nearby components. **The RAV-02 card is designed to take the full tip voltage and divide and filter it. Open circuit full tip voltage can be as high as 300VDC in some machines.**

3. If your plasma unit does not have a designated tip voltage measurement point, you will need to locate a place inside the unit where you can get one wire onto the work clamp lead and another on the heavy lead(s) that connects to the torch tip (electrode).

a. Note: some machines like the Hypertherm 380 do not have a single heavy wire to the Torch tip and instead have a set of parallel smaller wires that all terminate into one connector. In the case of the 380 the WHITE wires are the tip volts negative.

b. You can identify both locations by visually tracing the two leads as they come into the box. You should find several locations/terminal strips that have connections to these two points and you can use those for your sense wire connections. Use unshielded stranded.

c. Make a connection between the locations you have identified that tie directly to the two leads (workclamp and torch tip) to the two "TIP Volts" terminals. Make sure that these wires are routed where they cannot come into contact with hot or moving components. The TIP VOLTS inputs on the RAV-02 card have a polarity. The + side is the Workclamp and the - side is the Electrode

5. If you are using the CandCNC Digital Current Probe (DCP) with a plasma that does not have an ARC OK signal , the DCP01 provides a current feedback to CommandCNC and the ARC OK trip point is set and provides a software based ARC OK

6. If you have a plasma unit that DOES have an Arc Good signal and you have the DCP module, either signal will trip the ARC OK.

Note: The term Arc Good is interchangeable with Arc Ok , Arc Xfer and OK to MOVE.

7. NOTE: IT IS ESSENTIAL that the chassis of the plasma table have a good earth ground. Refer to the suggested grounding section of the diagrams and provide for a good earth ground close to the table. A safety ground back to a breaker panel many feet away may be a good ground for AC frequencies (60hz) but poor for higher frequencies like plasma noise.

DISCLAIMER AND LEGAL NOTICE

The following section covers the installation of a Voltage Divider card inside the plasma unit. There are dangerous and possibly lethal voltages present in a plasma power supply/unit. ALWAYS UNPLUG THE UNIT FROM THE AC POWER BEFORE REMOVING ANY COVERS. ALWAYS REPLACE ALL COVERS AND SAFETY BARRIERS BEFORE TURNING THE POWER BACK ON.

You do any install of a card inside of your Plasma Power Unit AT YOUR OWN RISK. If you do not wish to do the procedure either find a person qualified to do so OR contact us for options.

CandCNC/Fourhills Designs (hereafter referred to as “CandCNC”) nor any of its resellers or agents will be responsible for any damage to any plasma unit or the loss of any income resulting from using any of our electronics or using our instructions written or verbal to connect to any electronics. While we take care to provide accurate and concise information, we will not be responsible for any damages to equipment, personnel, or surrounding equipment, structures or land resulting from the direct or indirect use of our products.

The entire liability of CandCNC or any of its agents or employees is to replace or repair products provided by CandCNC. Under no circumstances will we be liable for any damages or loss exceeding the value of the actual products provided by us regardless if the products are used as described and in the proper manner. All CandCNC products carry a warranty that covers repair or replacement ONLY. Any labor, travel expense or costs to replace a component or product outside the CandCNC factory is NOT COVERED by warranty.

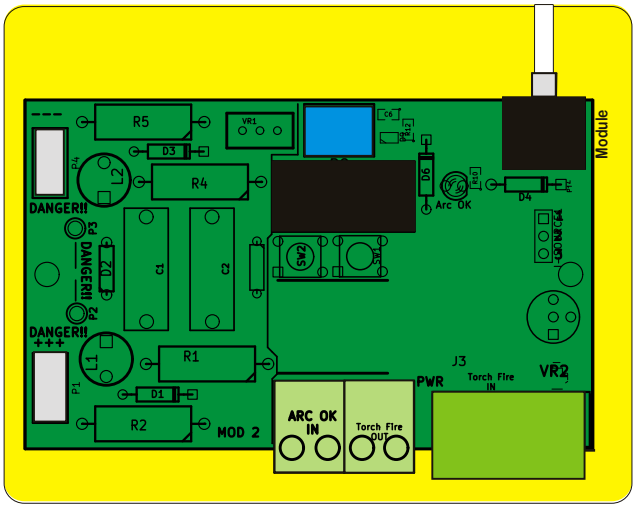
If you do not accept the terms of this notice DO NOT OPEN OR INSTALL THE RAV-01 CARD. Return the card for a full refund and seek an alternative way to sense the voltage.

CAUTION: Some plasma units use a very high voltage spark (Capacitor Discharge or CD) arc starting system. While the RAV-01 card is protected from high voltage inputs CD type systems can cause arcing in the connecting wires or to nearby components. If you have a CD start unit and do not have experience working with high voltage systems SEEK PROFESSIONAL HELP to do any install.

E

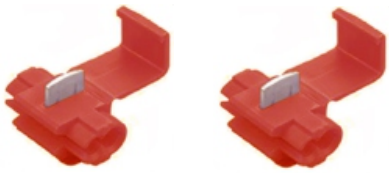
For Plasma Units WITHOUT Internal Voltage Divider
NOTE this card comes in most of the Connections Kits including the UNIVERSAL CONNECTION KIT

RAV-02 Raw Arc Volts CARD



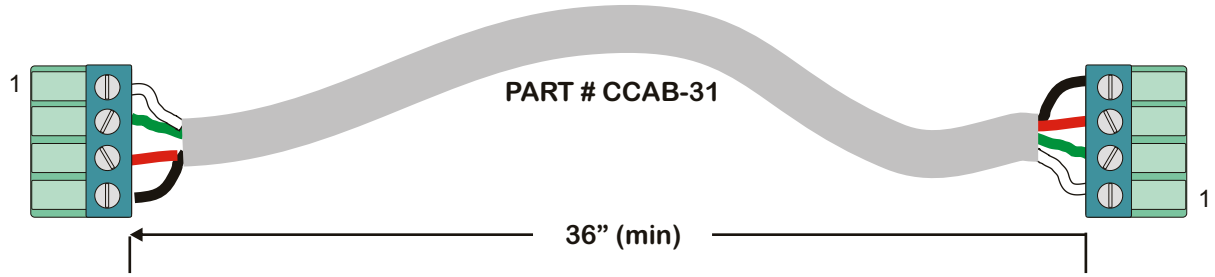
Plexglas Insulator Base (safety shield)

SHIELDED ANALOG CABLE
36" Nom
Part# SAC-01



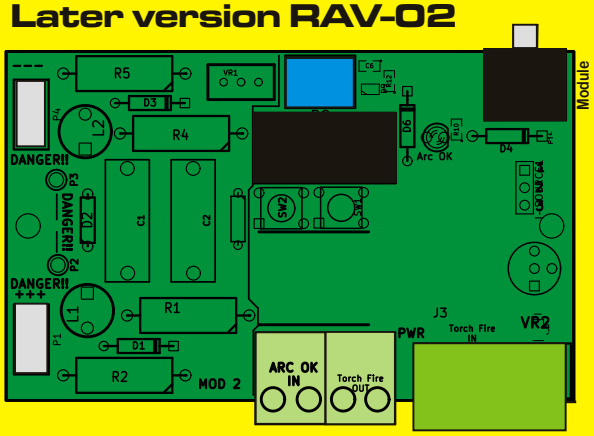
ScotchLOC™
IDC Splices (RED)
2 pieces

RAV to THCSensorPWM Interconnect



E

Later version RAV-02



RAV-02 Raw Arc Volts CARD

IMPORTANT: When making any connection inside the Plasma Unit, disconnect the unit from the AC Line (unplug it). Do not open the case with power on the AC line. THERE ARE DANGEROUS VOLTAGES present in the unit anytime it is connected to an AC source EVEN IF IT IS TURNED OFF.

IF YOU HAVE AN INTERNAL ARC OK SIGNAL: Connect one wire to Terminal 4 and Terminal 3 above (left two terminals)

IF YOU **DO NOT** HAVE AN INTERNAL ARC OK SIGNAL: You will need to purchase a DCP-01 Digital Current Probe.

TORCH SWITCH Terminals.
If you have a plasma unit with a Machine Torch you will need to locate the two torch fire connection points. in some units it is called START. IT is a REMOTE Fire set of wires.

IF YOU HAVE A HAND TORCH you will need to identify the two Torch Switch wires that come from the Hand torch and

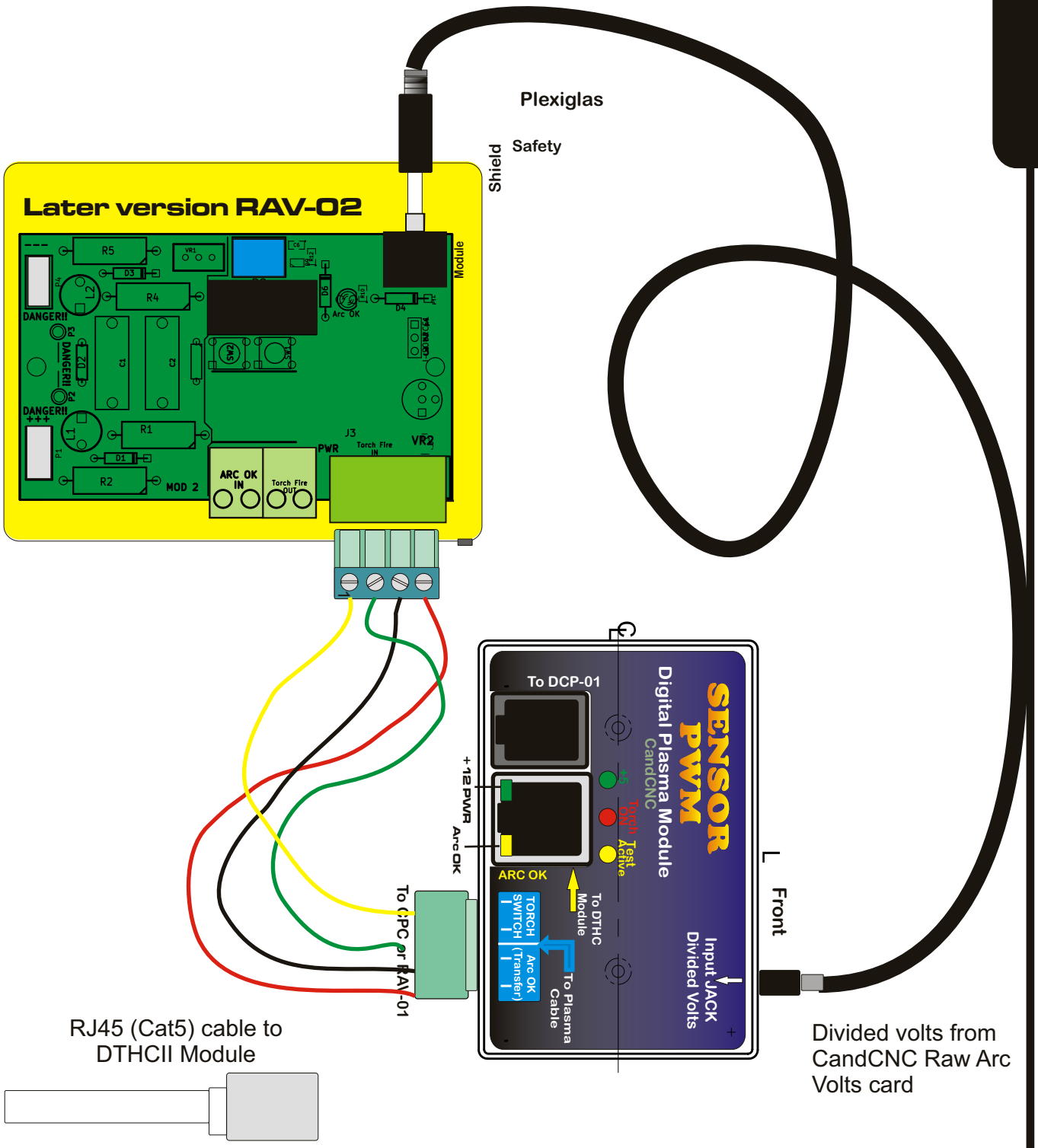
RAV-02 PREFACE: General Information

The RAV-02 can be used several ways so it can get confusing about making the connections. The primary use of the RAV-02 is to take Raw Arc Volts (undivided volts and divide it by a precise ratio so it can be used by the THC SENSOR PWM module to generate a digital signal to send volts data to the DTHCII Digital Torch Height Module. In some circumstances that is all that you will use the card for. Examples are on the Hypertherm 1000/1250/1650 series and the Older Thermal Dynamics “A” series. Both types have a rear CPC connector (14 pin) that has the Torch Fire and ARC OK signals available and are used with the CandCNC MIC-02 interface cable so ONLY the Shielded Analog Cable (SAC-01) is used between the RAV-01 and THC SENSOR PWM.

On other types of Plasma cutters that have no external connectors for ANY of the required signals the RAV-01 can become a connection board for accessing those signals internally and passing them to the THC SENSOR PWM Module via the CCAS-01 4 wire cable. The following pages will cover how to first use the RAV-02 as a voltage divider card in a unit that does not have an integrated voltage divider. If you have a plasma unit with no external connectors than the next section shows how to use the RAV-02 to fire the torch from the software. Finally if your plasma has an internal access to the Torch fire for a mechanical torch and an internal ARC OK (OK to MOVE). REMEMBER: the RAV-02 is just a pass-through (place to make connections) for ARC OK and TORCH FIRE. It does not sense or provide those signals.

There are added pages on some Plasma Cutters and how to connect them; sometimes there is more than one way. If there is a way to access the proper signals externally than that method with a cable is recommended. In each case the objective is to get the signals in and out of the THC SENSOR PWM Module.

RAV-02 Mounted inside Plasma UNIT.
CandCNC RAW ARC VOLTS Voltage divider card



THC SENSOR PWM MODULE
Mounted outside plasma unit

Connections for PWM MODULE REV21 are the SAME

For Plasma Units that have NO external connector for the required signals OR do not have an internal voltage divider.

Mounting the RAV-02 Card inside your Plasma Cutter: Do any mounting or modification with the Power cord DISCONNECTED to the plasma unit

The mounting location for the card will vary from one type/brand of plasma unit to another. Pick a location and mount the RAV-01 using the Plexiglass shield using small self tapping screws (not furnished) or adhesive Velcro strips. Mount the card at least 1 inch away on all sides from any internal cards, terminals or bare wires. The standoffs of the card are insulated and so the card can be mounted with the plexiglass shield against the chassis. There is high voltage present when the torch fires at the end of the card where the RAW ARC VOLTS (tip Volts) is connected. Keep that end of the card away from ANY conductive object closer than 2 inches.

Once you have the RAV-01 Card mounted in a safe location you will need to make provisions to connect the two low voltage cables (CCAB-31 and SAC-01). You need to provide holes on the rear or side of the unit (BE CAREFUL DRILLING METAL IN YOUR UNIT. TINY SHAVINGS CAN FALL ON PARTS THAT COULD SHORT.) Ream the holes smooth or drill them oversized and use a rubber grommet to protect the wires. Clean up with a magnet or blow the cabinet out with air. One hole needs to be able to pass the diameter of the SAC-01 Plug. The other needs to pass the diameter of the cable for the CCAB-31. NOTE: The 4 wide plug on the CCAB-31 cable is removable. You will need to remove the end that passes into the enclosure to fish the wire through for the RAV-01 Connection. BE SURE TO FOLLOW THE COLOR CODE TO REATTACH THE CONNECTOR. THE COLORS ON BOTH ENDS WILL GO ONTO THE SAME TERMINAL/PIN NUMBER. Connect the two cables where the diagram shows to the RAV-01 Card.

Making the connections

A WORD ABOUT ARC OK SIGNALS: “ARC OK” is our terminology of a signal coming back from the plasma unit that signals the unit has fired the torch and has a valid arc. Most plasma units made for manual cutting DO NOT HAVE an ARC OK (also known as ARC XFR; TRANSFER; OK to MOVE and other similar terms). It is a circuit that closes (relay contacts or sometimes called “dry contacts”) on a valid arc. Units that do have the signal will have it labeled, on a connector or covered in their manual. IF YOU HAVE DOUBTS, assume the unit does NOT have ARC OK and follow the guidelines below.

IF YOUR PLASMA HAS AN ARC OK CONNECTION POINT: There will be two connection points. Run a small gauge (18-24 ga) pair of wires from the internal ARC OK points to the two screw terminals on the edge of the RAV-01 Card. This wire carries no high voltage or current. Follow the diagram on Page ____. There is a board labeling program on some cards where the white printing is WRONG! USE THE PICTURES IN THIS MANUAL to make the connection. Make sure the connection is secure and that no stray wires are outside the opening of the screw terminals.

IF YOUR PLASMA DOES NOT HAVE AN ARC OK CONNECTION POINT: Turn to the section on the DCP-01 Digital Current Probe option. You will need to purchase and install this option to provide a reliable ARC OK signal to your DTHCII system.

Finding the correct connections.

The first signal you need to identify and locate is the Raw Arc Volts (Raw Tip Volts). This is the voltage between the Electrode and the Workclamp. The Hypertherm 1000 thru 1650 series have two spade terminals (J15 and J16) that are for easy connection of Raw Arc Volts. On later models (45/65/85/105/125) WITHOUT the internal voltage divider, the location of the Raw Arc Volts is not as obvious but they have Field Service Bulletins where they give detailed directions on finding the Raw Arc Volts. Our manual covers connection to the PowerMax 1000, 1250 and 1650 as well as the PowerMax 45 and PowerMax 65/85/105 with the optional CPC connector.

https://www.hypertherm.com/Xnet/library.jsp/null is a search page where you can enter your model number and then search the FSBs. The files are in PDF format.

For other brands of plasma units or a model not designed to be automated the search for connection points may be a little more difficult but not impossible. The key is the leads going to the torch cable. On most plasma units you can locate these signals by opening the unit (POWER DISCONNECTED!) and visually tracing the wires coming from the plasma torch. The Workclamp will be connected to a stud or terminal inside and is pretty easy to identify. It is the POSITIVE (+) side of the circuit. The Electrode side goes up the plasma cable to the torch head. It will be one heavy wire or a series of smaller gauge (12ga or larger) stranded wires of the same color and they will all connect to the same electrical spot (bus) inside the plasma. In a lot of units these wires are all solid WHITE in color but do not use color as your clue. Some plasma manufacturers provide block level schematics in their use or service manual that give wire colors (and in some cases terminal numbers and locations).

WHEN YOU HAVE LOCATED THE WORKCLAMP AND ELECTRODE WIRES IN THE UNIT:

Using a two conductor wire (18-22 ga) [not supplied] with insulation rated to 400V or more crimp on two ring or fork terminals. USE WIRES OF TWO DIFFERENT COLORS and long enough to reach the RAV-02 Card using an indirect route (give yourself extra wire). Run the first color wire (red or the brightest color) to where the WORKCLAMP attaches. Normally that will be a heavy bus bar with other smaller wires attached. If it is a single large stud you will need a ring terminal that will fit over the stud. That will be your positive (+) wire.

Use the other wire color and run a connection using a ring or fork terminal to where the ELECTRODE wires attach.

Carefully route both wires from their connection points over to where they will attach at spade inputs on the edge of the RAV-01 Card. Keep the wires away from other high voltage wires or components on the circuit board. Use nylon wire ties to secure the wires to other wire bundles or to the chassis. DO NOT WIRE TIE THEM TO COMPONENTS ON THE PC BOARDS. DO NOT USE LOW VOLTAGE WIRES LIKE THOSE USED FOR WIRING PHONES OF NETWORKS.

The next internal signal you need to locate is the TORCH SWITCH. If you are connecting to a unit with a Hand Torch you will need to find and tap into the two wires coming from the torch switch in the hand piece. How will you know? First the wires will be smaller and different colors than the ELECTRODE or PILOT ARC (more about that later) wires. In most cases there will be four smaller wires. Two will be the torch switch and two will be the PIP or CIP (Consummables-In-Place) wires. Once again the manufacturers documents can be of service here identifying colors and even connection points. If you do not have the manufacturers service information with schematics and cannot find them on-line you will need to do a little detective work to identify the torch switch wired.

The first thing your should do is get an Ohmmeter, and with it set to low ohms, short the leads together and make sure the meter shows the change and displays low (close to zero) ohms. If your meter has a “squaker” continuity tester position then use that as a tone indication. Clip across two of the four smaller wires. If you get no reading or tone (or an OV or OL indication) the circuit is open. Activate the torch switch on the hand piece and if the reading goes to a low value of ohms (<100) or the tone sounds, it is the switch contacts. Confirm the reading by pushing the torch switch several times. Keep testing wires until you find the pair that changes the meter. Note the colors. Use your meter to test the other wires. You may well find a pair that causes the meter to go to low ohms as soon as you touch them, but working the torch switch WILL NOT change the meter. Those are NOT the torch switch pair. Once you have identified the Torch Switch pair study the diagram on page ____ and using the two ScotchLOC connectors slide one over each of the two wires connect them to the Torch Switch pair.

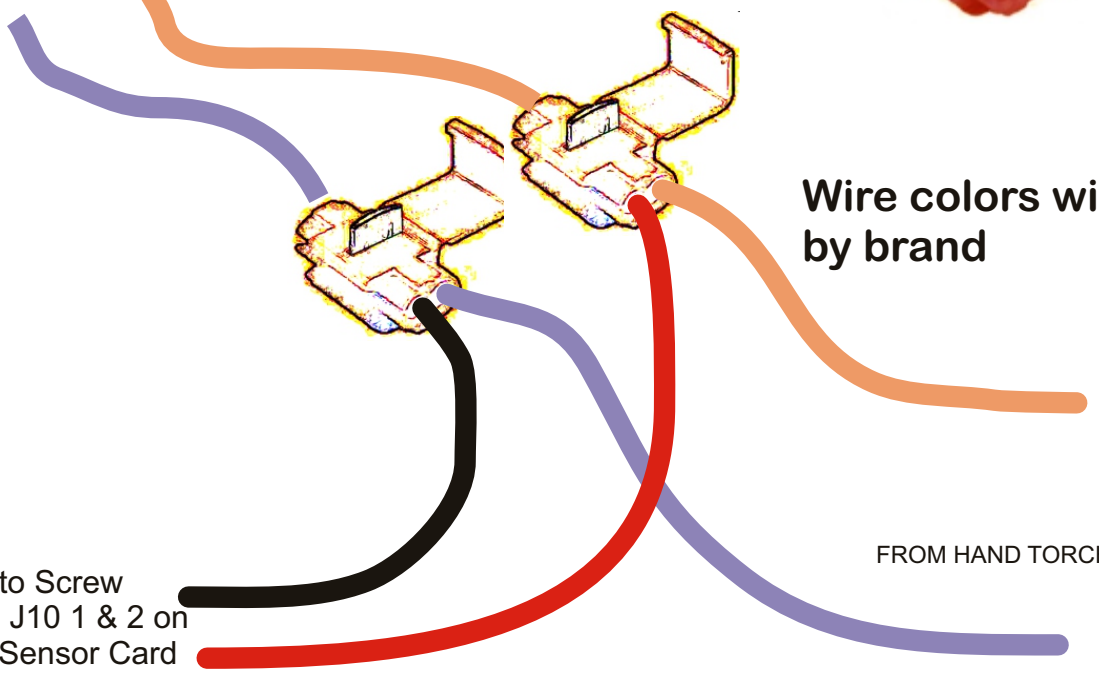
That concludes the internal connections you will have to make for your unit. Make sure all leads are insulated and away from possible physical damage. Double check to make sure there are no loose connections and that you have attached/ re-attached any wires mentioned in the above guidelines.

Replace all covers and safety devices on the plasma unit and plug the plasma unit into power with the unit switched off. Turn the unit on, and make sure the unit works correctly in manual mode. (i.e. cut a piece of metal by hand). If you have a machine torch manually fire the torch from the Torch On button in MACH3

NOTE : Some models of plasma cutters that have a CNC connector (CPC connector) have a safety lockout to prevent firing the hand torch remotely. The listed connection will fire the torch the same as pulling the trigger. Both the trigger and the software can fire the torch. There may be a way to bypass the lock-out and use the CPC connector to fire the torch. Contact us via e-mail for other possible solutions on plasma cutters that have a CPC connector.

Use ScotchLoc IDC Splices (RED) to tie Torch Switch output on THC Sensor (J10) Screw Terminals. Locate Orange and Violet wires at J10 in the PowerMAX box and tap each wire as shown. To test short two screw terminals on J10 THC Sensor and torch should fire (Plasma Unit on)

CONNECTING HAND TORCH TO THC SENSOR CARD



Wire colors will vary by brand

FROM HAND TORCH CABLE

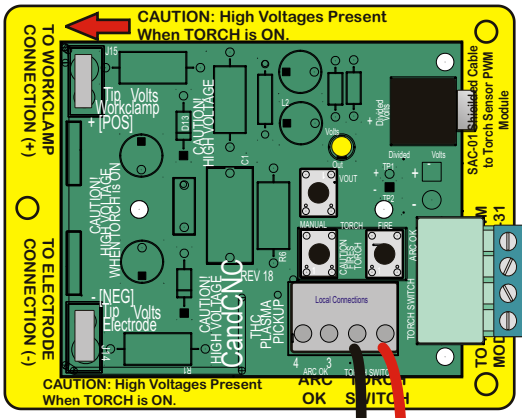
Connect to Screw terminals J10 1 & 2 on the THC Sensor Card

IMPORTANT: When making any connection inside the Plasma Unit, **disconnect the unit from the AC Line (unplug it). Do not open the case with power on the AC line.** THERE ARE DANGEROUS VOLTAGES present in the unit anytime it is connected to an AC source EVEN IF IT IS TURNED OFF.

USING THE RAV-01 TO FIRE THE TORCH
(for plasam units with no external connector to fire the torch or with a hand torch)

NOTE the RAV-01 Card has been updated and designated the RAV-02. Its functions and connections remain the same but some locations for connectiosn have been moved. Any reference to the RAV-01 on new systems should be replaced with the RAV-02

RAV-01 Raw Arc Volts CARD

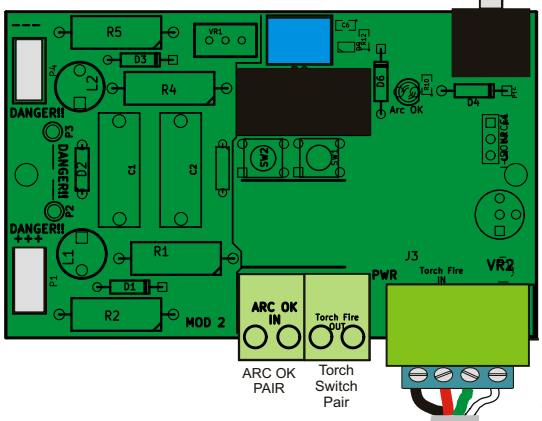


PART # CCAB-31

Torch Fire wires see previous page for connecting to Hand torch models without CPC connectors

Divided Volts OUT to PWM Module 3.5mm stereo plug

Later version RAV-02



Electrode (-)

WORK clamp (+)

ARC OK PAIR

Torch Switch Pair

PART # CCAB-31

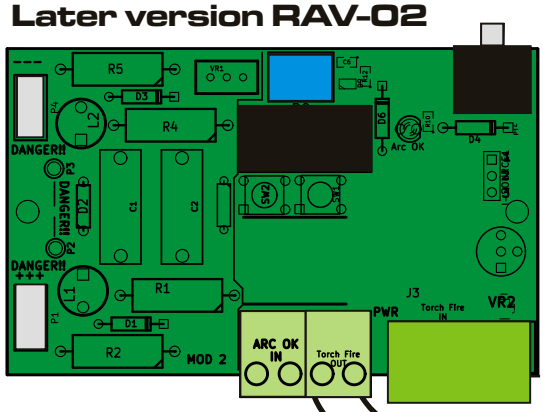
Connection to THC SENSOR PWM Module

The RAV-01 card has been redesigned but the connection points are the same and similar in location.

USING THE RAV-01 TO FIRE THE TORCH
(for plasam units with no external connector to fire the torch or with a hand torch)

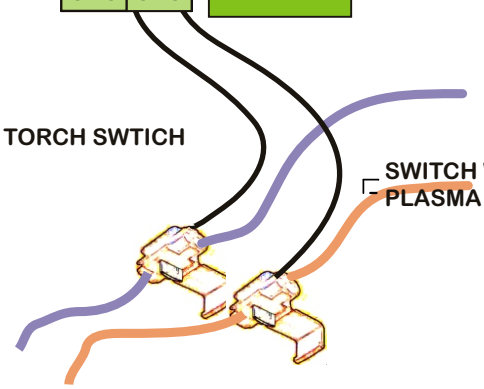
RAV-02 Raw Arc Volts CARD

Later version RAV-02

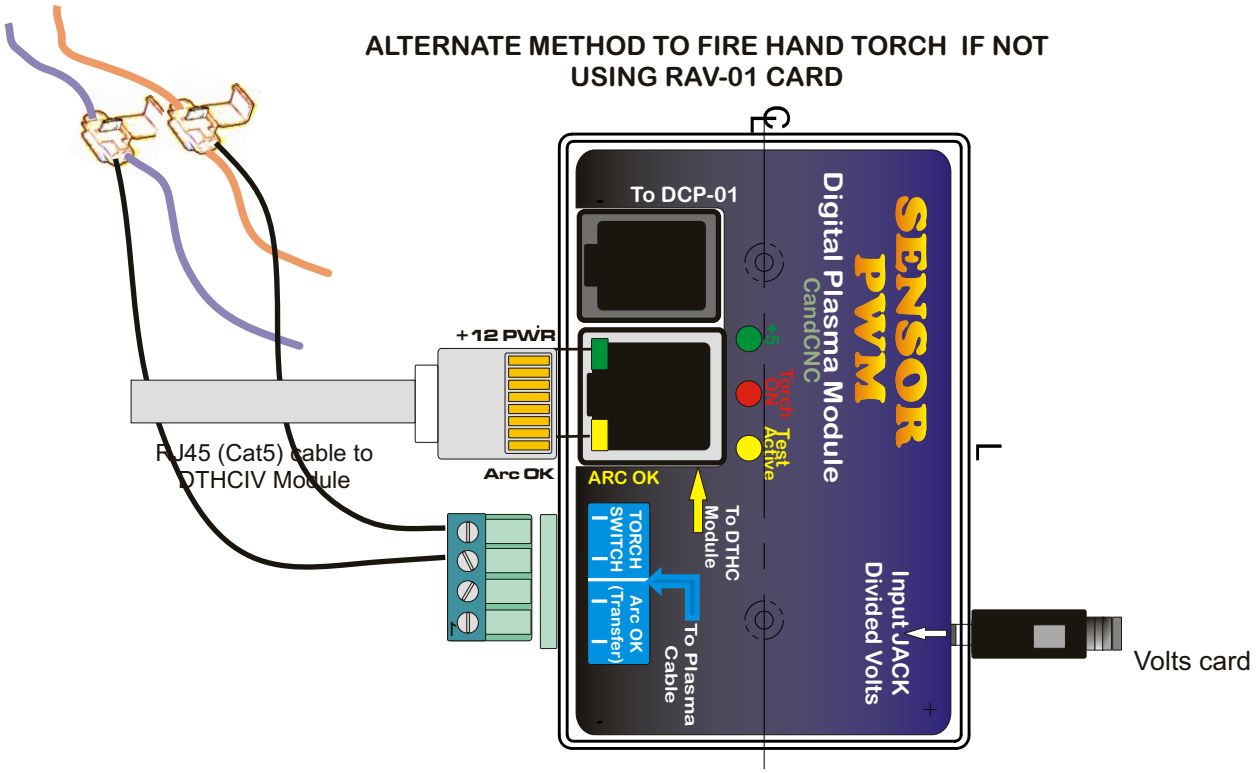


CONNECTING TO HAND TORCH SWTICH WIRES FROM RAV-01

SWITCH WIRES FROM PLASMA HAND PIECE



ALTERNATE METHOD TO FIRE HAND TORCH IF NOT USING RAV-01 CARD



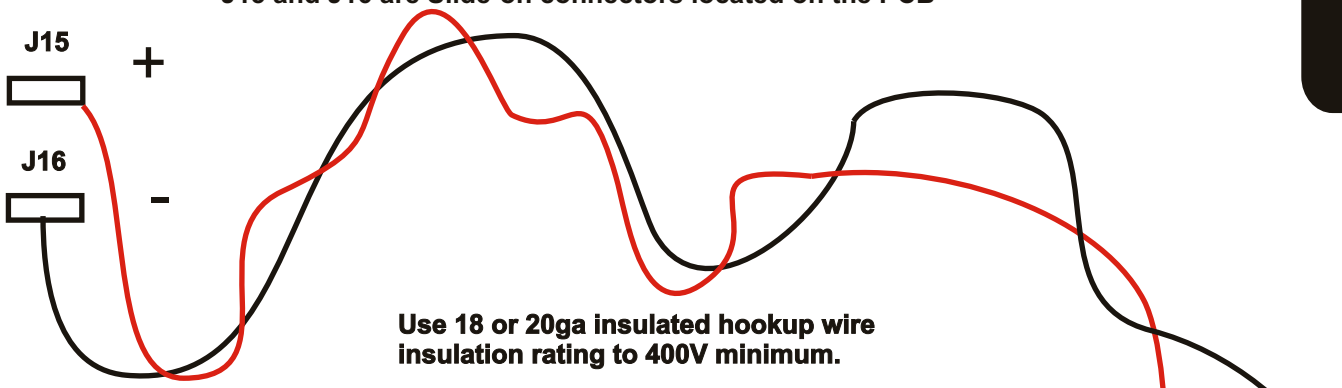
Use the same two wires on the 4 pin connector for the PWM Module REV 21

F

This is for units with a hand torch that do not have the rear CPC connector

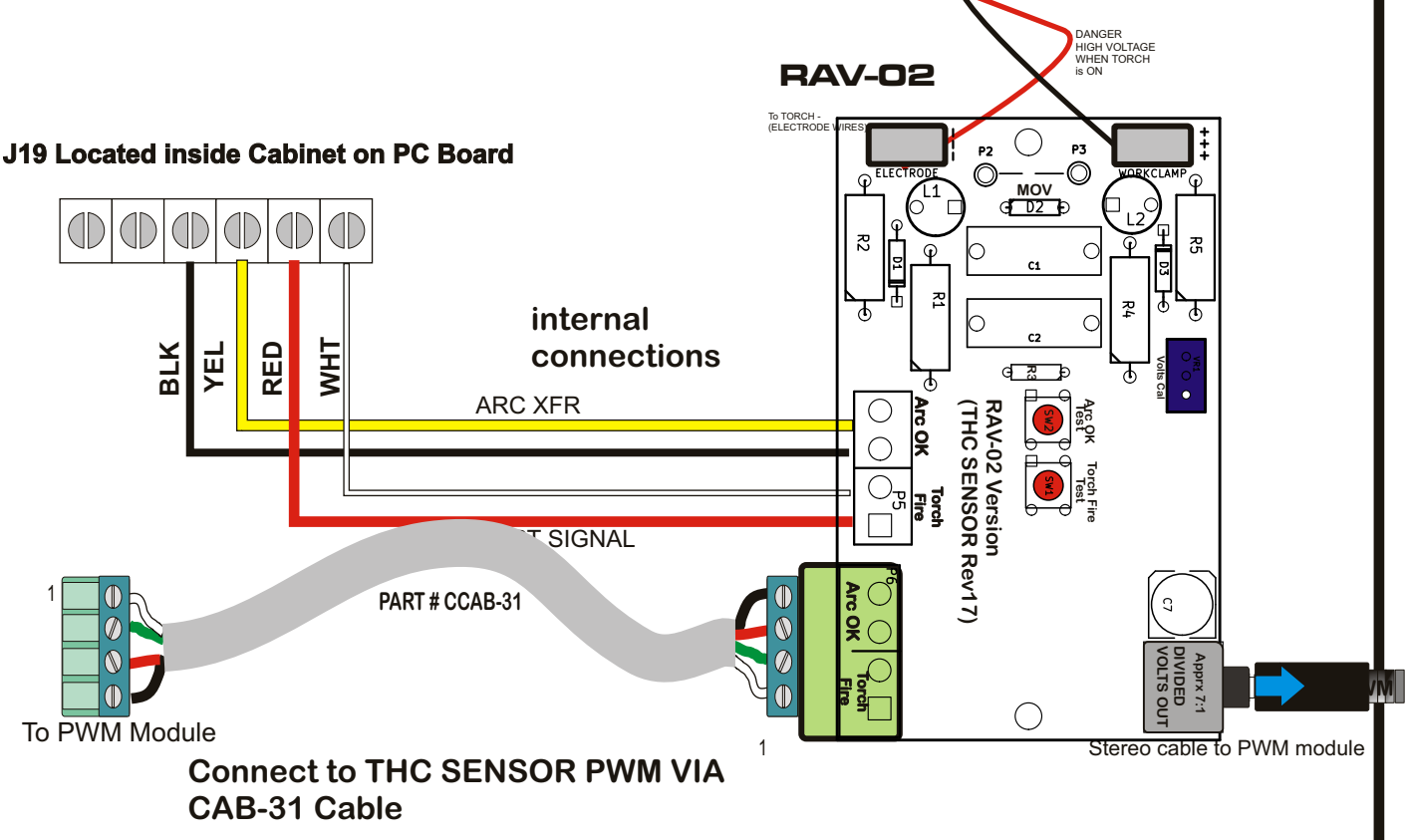
TIP VOLTS CONNECTION

J15 and J16 are Slide-on connectors located on the PCB



Use 18 or 20ga insulated hookup wire insulation rating to 400V minimum.

J19 Located inside Cabinet on PC Board



Connect to THC SENSOR PWM VIA CAB-31 Cable

F

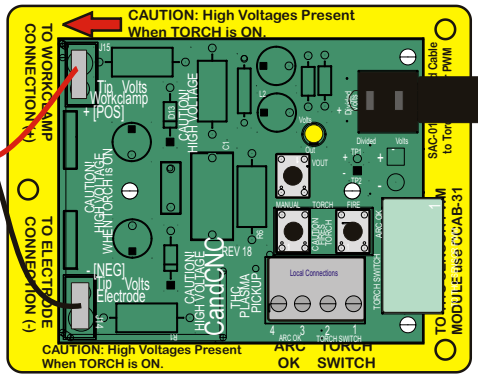
Raw arc volts connection points vary from one model to another. Refer to manufacturers manuals for locations.

+

-

Use 18 or 20ga insulated hookup wire. Insulation rating to 400V minimum.

OPTIONAL RAV-02 MOUNTED INSIDE PLASMA



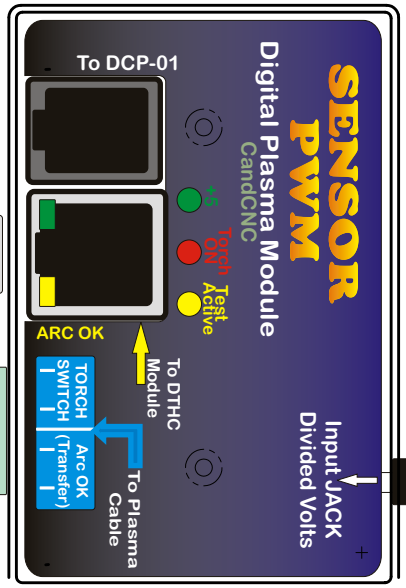
Route shielded cable through rear bulkhead

METHOD TO CONNECT TO UNIT WITH MACHINE TORCH USING REAR CPC CONNECTOR FOR TORCH FIRING AND ARC OK

TO CPC CONNECTOR ON REAR OF PLASMA

MIC-02
For units with CPC but NO Internal Voltage Divider

RJ45 (Cat5) cable to DTHCII Module



Divided volts from CandCNC Raw Arc Volts card

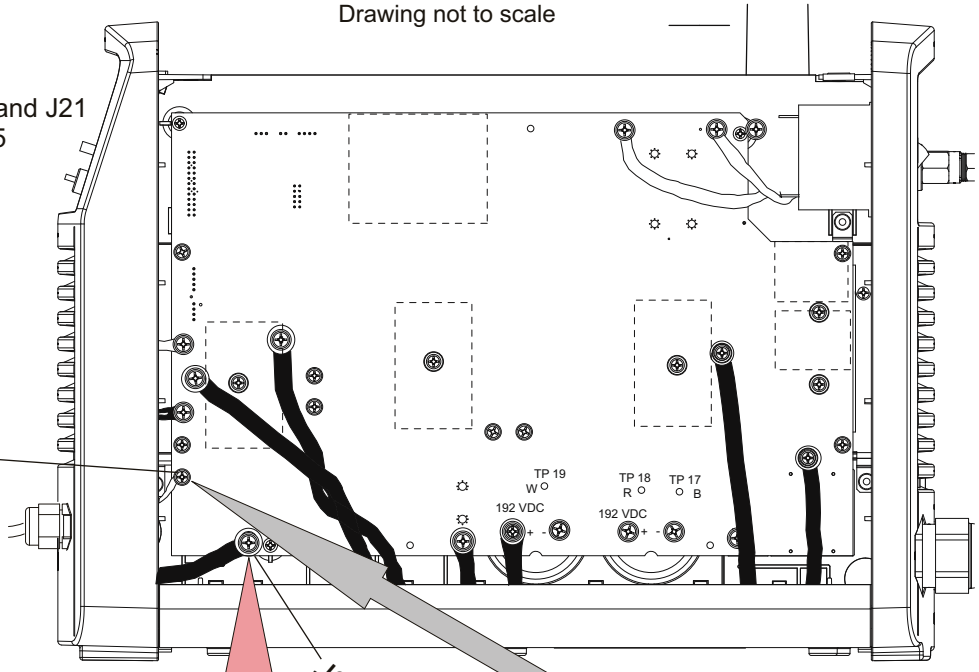
HYPERTHERM PowerMAX 45
Connecting THC SENSOR CARD

NOTE: THE Hypertherm 45 comes stock with a rear CPC connector and a 50:1 internal voltage divider. The preferred connection method is to use the MIC-01 cable kit from CandCNC.

SIGNAL NAME(s)	POWERMAX REF	RAV-01	NOTES	WIRE Type
TORCH SWITCH; START - MECH TORCH	PINS 3 & 4 J12	Screw Term 1 & 2	For use with mechanical Torch ONLY.	18-22Ga stranded low voltage insulated
TORCH SWITCH HAND TORCH	ORG & VIO wires at J10	Screw Term 1 & 2	Parallel taps where Torch Cable Plugs into J10 in PowerMax	18-22Ga stranded low voltage insulated
ARC VOLTS; RAW TIP VOLTS	J19 & J21 inside Powermax45	J14 (neg) , J15 (pos)	This is NOT the 50:1 divided voltage at J12. See detailed instructions	18-22 PVC stranded , insulted, twisted pair min 600V rated insulation
ARC OK ; ARC XFR	PINS 12 & 14 J12	Screw Term 3 & 4	Dry Contact ouput from PowerMAX (no voltage)	18-22Ga stranded low voltage insulated

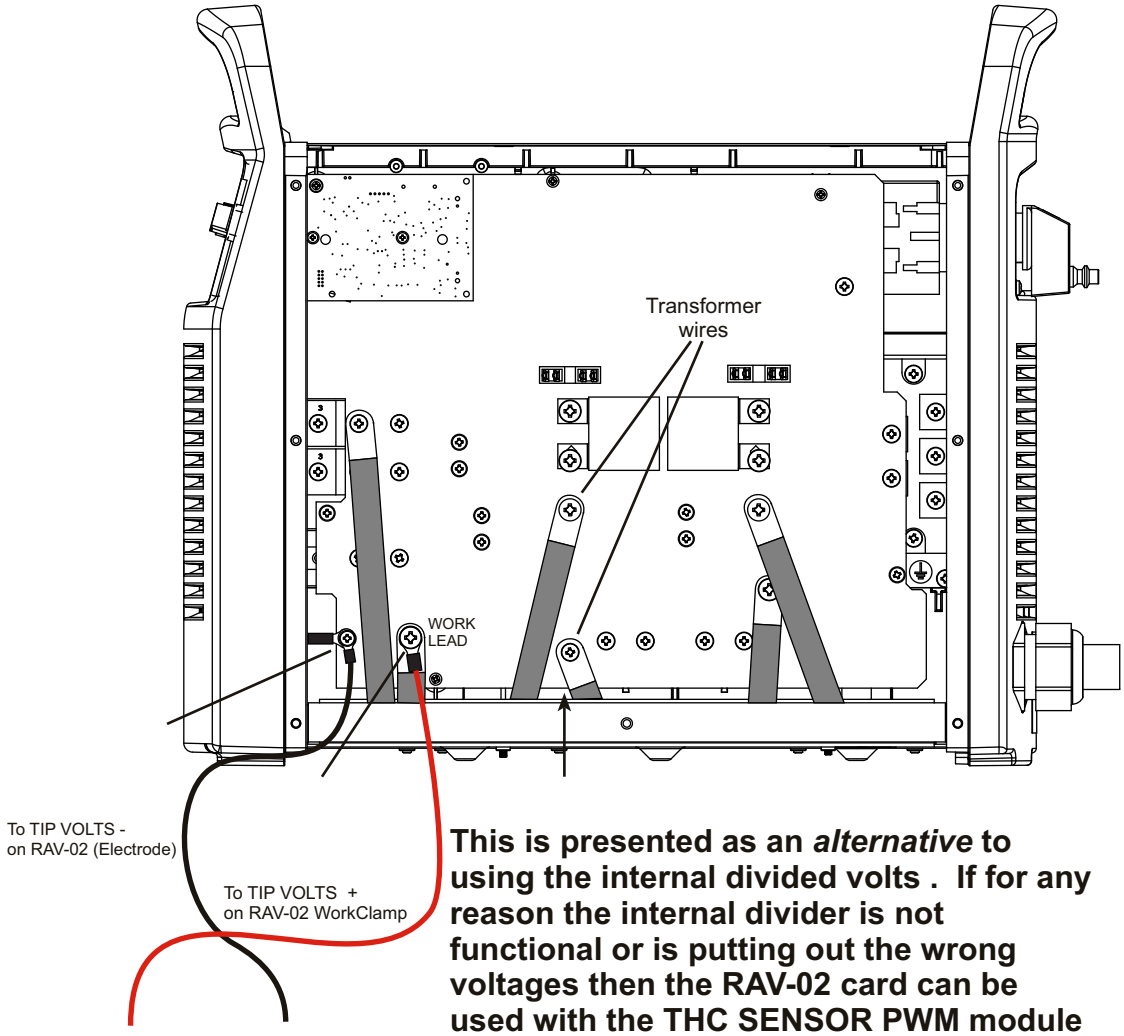
Drawing not to scale

LOCATION OF J19 and J21
Inside PowerMAX 45



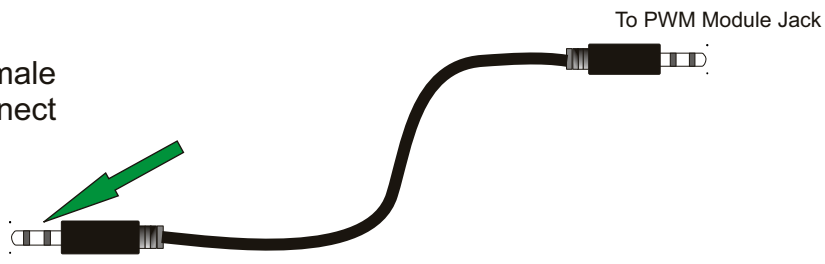
CAUTION: Make sure wires and terminals do not touch anything but the screws for J19 and J21. HIGH VOLTAGE IS PRESENT WHEN TORCH IS ON.

ByPassing the 65/85 internal voltage divder using the RAV-02 Raw Arc Voltage Divider card



This is presented as an *alternative* to using the internal divided volts . If for any reason the internal divider is not functional or is putting out the wrong voltages then the RAV-02 card can be used with the THC SENSOR PWM module

Use CPC cable to access START (Torch on) and TRANSFER (Arc OK) Use male to male Stereo cable to connect Divided Volts to PWM . Set PWM ratio to 7:1



Interfacing using Circular Plastic Connector on Plasma Units with Automation Features

NOTE: The presence of a CPC connector on a Plasma Cutter does not necessarily indicate that it has all (or any) of the signals for direct connection to the DTHC II system. The diagrams and examples listed in the following pages are for specific units that have CPC connectors and a pinout that is consistent with the cables we supply. Several import Plasma units have added CPC (CNC interface) connectors but they do not have the same pinouts and require custom cables. We have not been supplied with any sample units and the manuals are confusing so interface support from CandCNC will be minimal.

As of this published date the following units have CPC connectors with the correct pinout:

Hypertherm 1000/1250/1650 (no divided arc volts)
Hypertherm 45 (stock with rear CPC connector and voltage divider)
Hypertherm 65/85/105/125 (CPC option with divided arc volts)
Thermal Dynamics 52/82/102 series (option for connector and Automation Interface)
Thermal Dynamics A60/A80/A120 (CPC connector standard; Arc Volts divider is an OPTION on older units)
Thermal Dynamics Automation 151 with optional TD Automation Interface Card

There may be other models and brands with CPC connectors using the same pinouts. Check with you plasma authorized reseller or service representative for technical information about any automated connectors and pinouts.

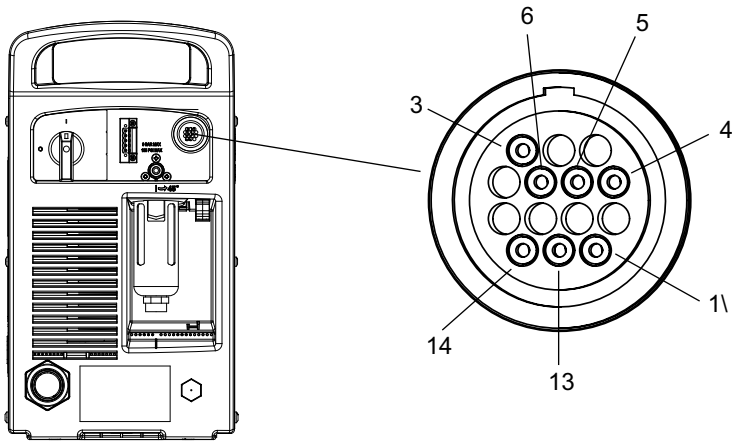
On units that have the CPC connector but DO NOT have the Arc Volts Divider, you will need to purchase an arc volts divider. The RAV-01 is an ARC VOLTS divider card that is available from CandCNC.

On units with a machine torch you can elect to make all of the connections via the RAV-01 card of use it only for divided Arc Volts and the MIC-02 cable to connect to the rear CPC for the TORCH FIRE and ARC OK.

CandCNC has a whole series of Connection Kits designed to connect to the units listed above or by using our *Universal Connection Kit* virtually any other plasma cutter.
Visit http://www.CandCNC.com/PlasmaHT-Connect-Kits.html for Hypertherm Kits or http://www.CandCNC.com/PlasmaOther-Connect-Kits.html for other brands of plasma cutters.

For hand torch models you **may not be able to fire the hand torch via the remote START signal**. In that case you will need to find and tap into the Torch Switch wires coming from the hand torch. It is suggested that for Hand Torch installs even if the unit has a CPC interface, that all connections to/from the THC SENSOR PWM Module be made via the RAV-01 and it's supplied cables.

HYPERTHERM
Rear CPC Connector Pinout



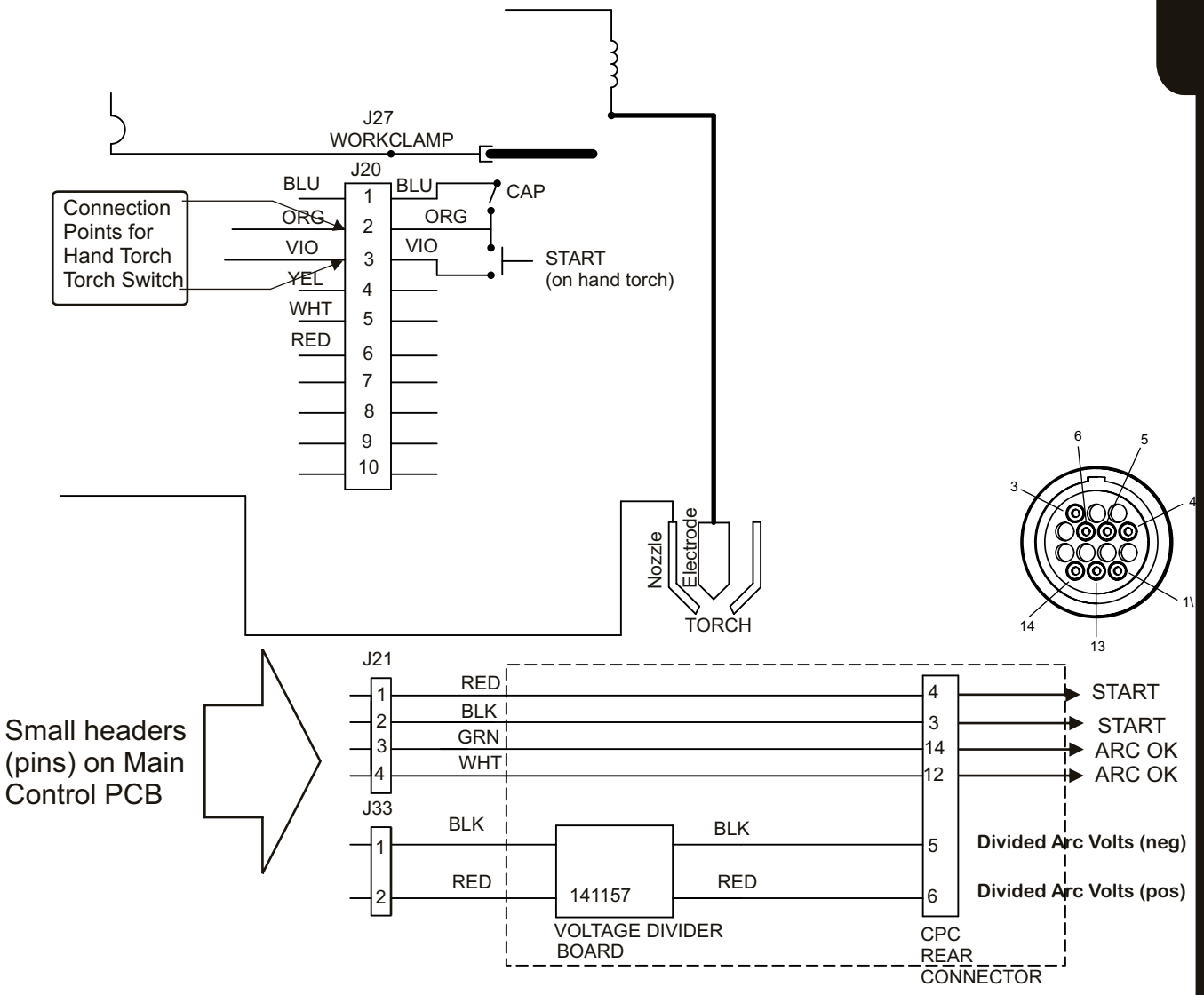
Refer to the following table when connecting the Powermax65 or Powermax85 to a DTHC II torch height controller with a Custom CandCNC Machine Interface Cable

Signal	Type	Notes	Connector sockets	CandCNC Cable wires
Start (start plasma)	Input	Normally open. 18 VDC open circuit voltage at START terminals. Requires dry contact closure to activate.	3, 4	Green, White
Arc Transfer (ARC OK) (start machine motion)	Output	Normally open. Dry contact closure when the arc transfers.	12, 14	Red, Black
Ground	Ground		13	Shield
Voltage divider	Output	Option. Not on all units	5 (-), 6 (+)	Red (-), White (+)

Note: Wire colors for CandCNC Hypertherm CPC Interface cables are different from the wire colors for a Hypertherm CPC interface cable

F

HYPERTHERM 45/65/85/105 Machine Interface Connection points



As long as your PWM module is mounted within about 48 inches or less to the CPC connector the factory default of 50:1 divider ratio is recommended. Only change the setting if you are having problems with torch noise causing erratic voltage readings.

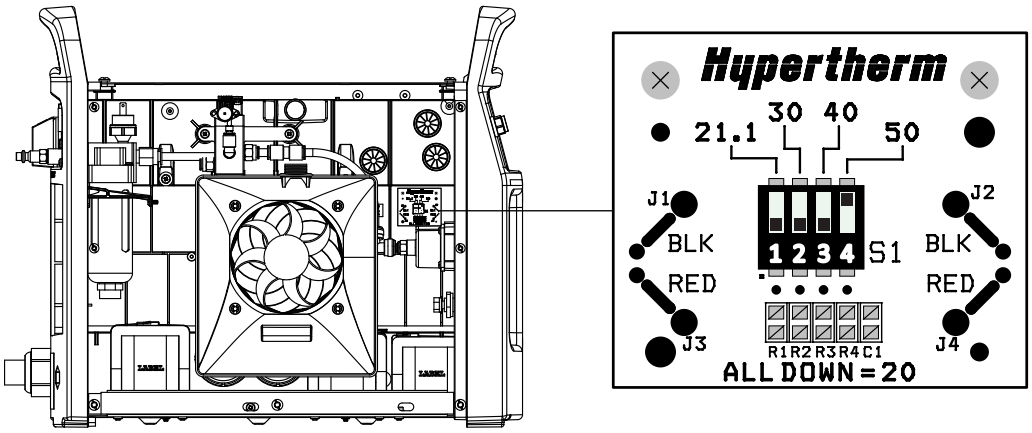
TORCH SETUP

SETTING THE 5 POSITION DIP SWITCH FOR USE WITH CandCNC DTHC IV TORCH SENSOR PWM

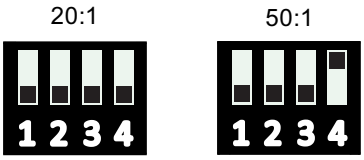
To change the factory preset voltage divider from 50:1 to the 20:1 setting

1. **Turn OFF the power supply and disconnect the power cord**
2. **Remove the power supply cover**
3. **Locate the voltage divider DIP switches on the left side of the power supply**

Note: The figure below shows the default setting (50:1) with the number 4 switch up

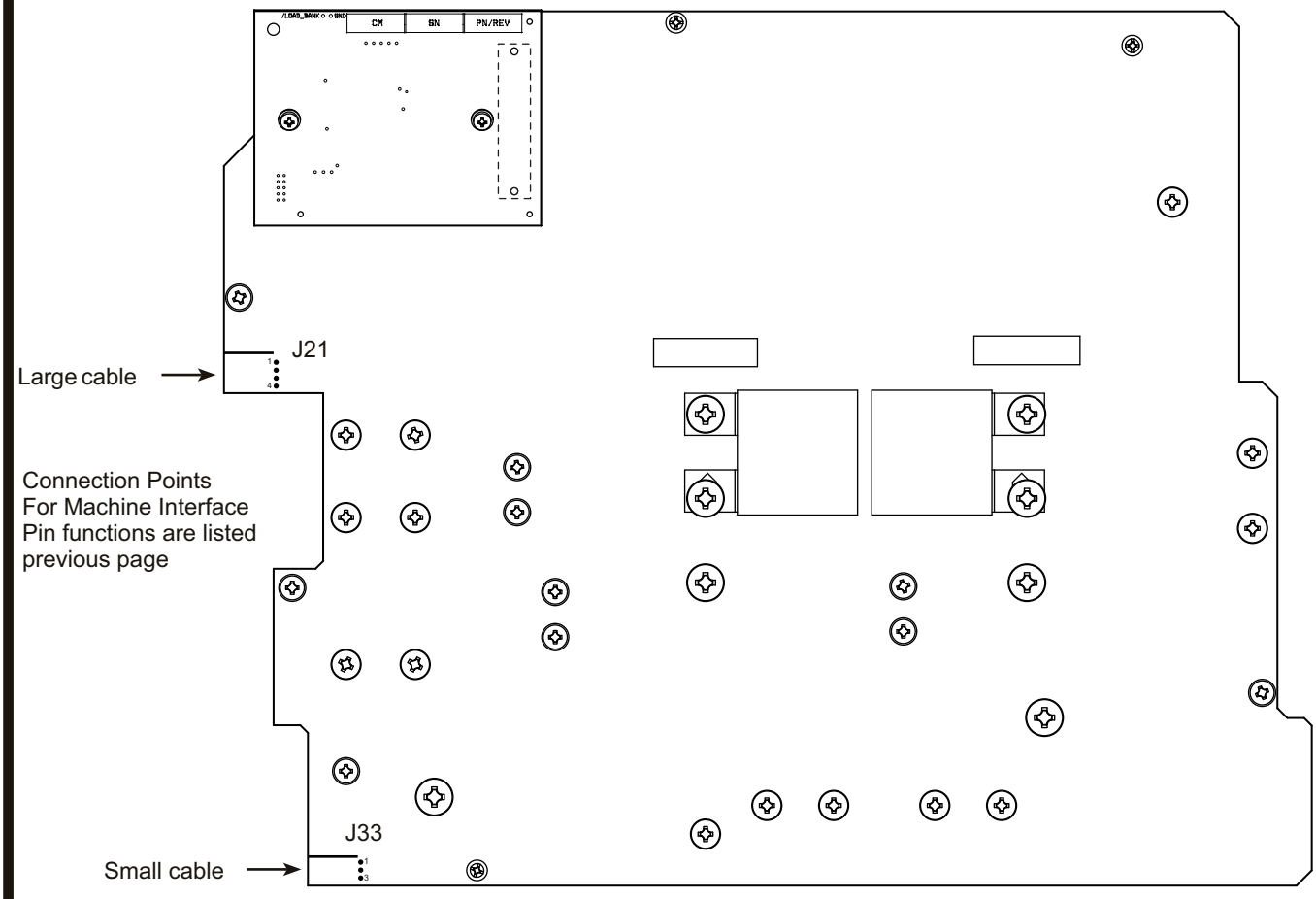


4. **Set the DIP switc**



Note: The Hypertherm document has additional switch settings for other divider ratios but ONLY the 20:1 or 50:1 ratios work with the DTHC IV and the THC SENSOR PWM module. The divider setting inside the THC SENSOR PWM case (bottom removed) is changed to work with a 20:1, a 50:1 and a 7:1 (CandCNC Raw ARC Volts divider card) input.

Physical Location of J21

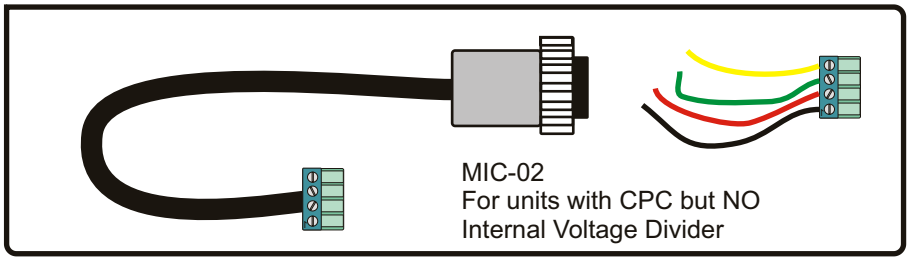


MIC-01 is for use with Hypertherm models 45, 65, 105, and 125 that have the CPC option installed. The CPC is a round plastic connector on the rear of the units CPC comes with Machine Torch option or can be ordered separately. See the previous pages for the location of the receptacle.

MIC-01
Shown with THC Sensor PWM Module

Connects THC Sensor PWM Module directly and picks up:
START (remote start)
TRANSFER (Arc OK)
Divided Arc Volts

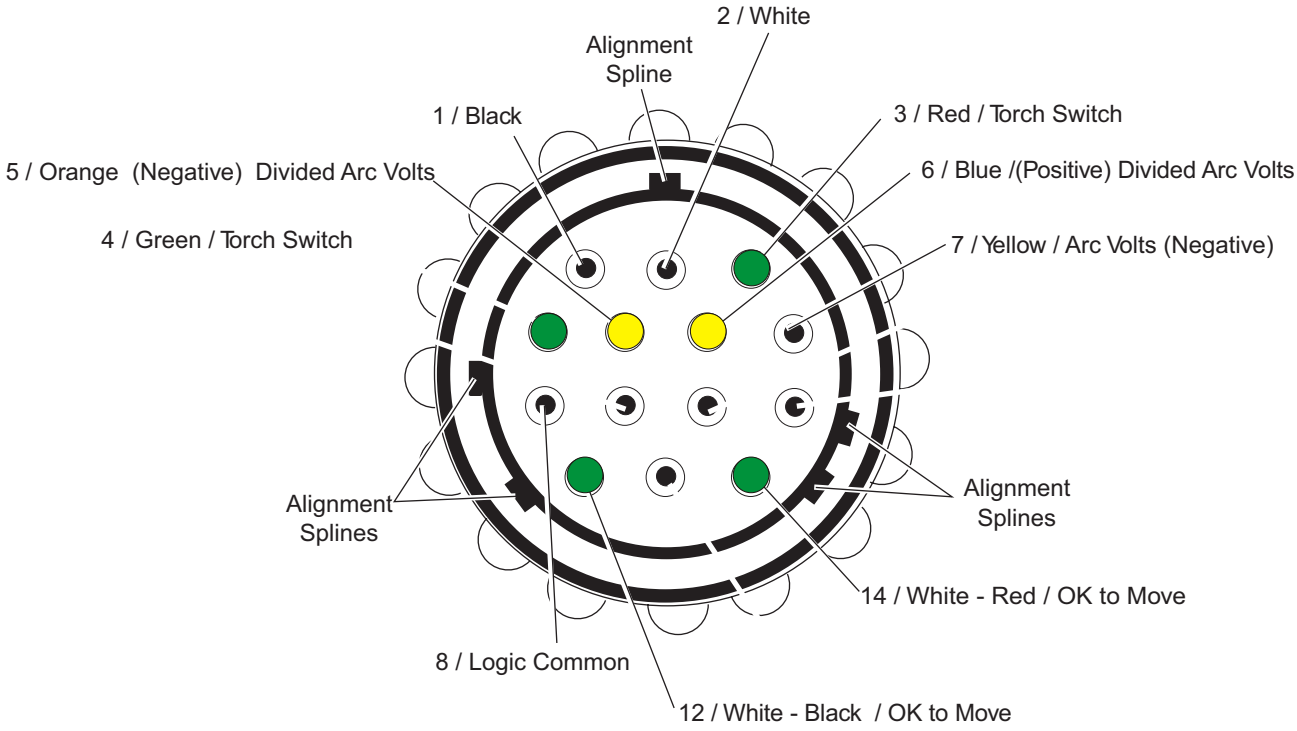
Cable comes prewired to plug into the CandCNC THC Sensor PWM Module and the standard CPC plug on the back of 45, 65 and 85 units that have that option. See the previous pages for recommended settings.



MIC-02
For units with CPC but NO Internal Voltage Divider

F

CONTROL CABLE PIN - OUT DIAGRAM



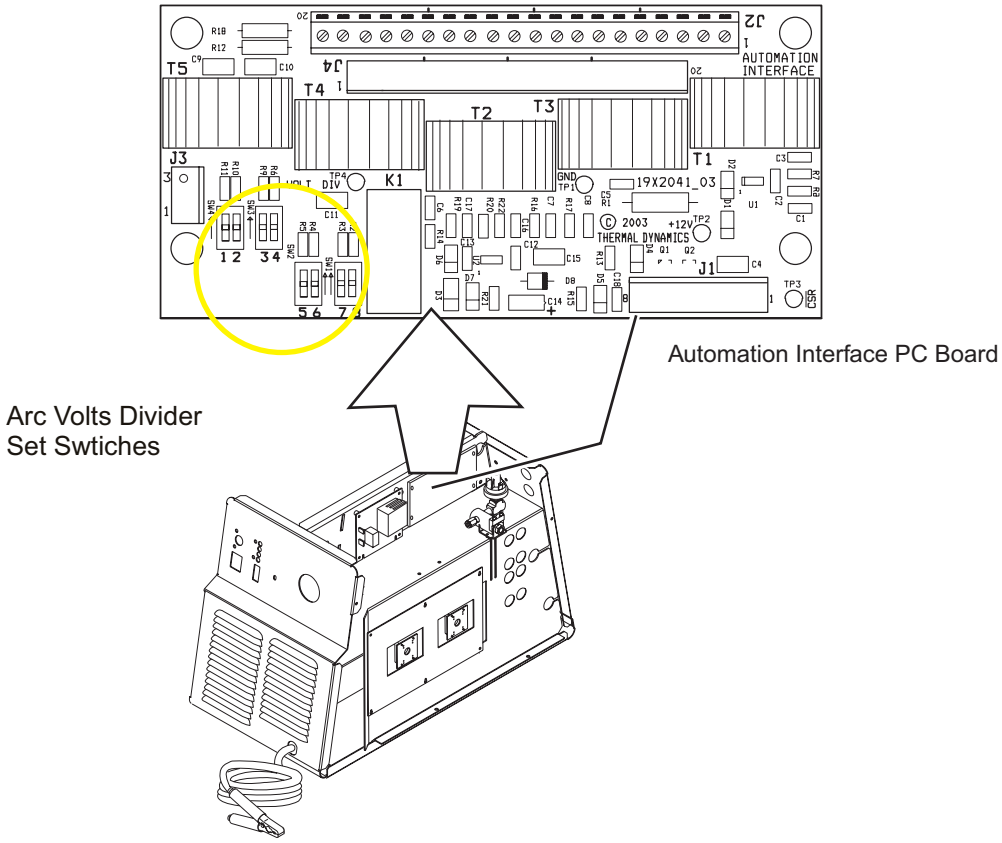
Use CandCNC part # MIC-01

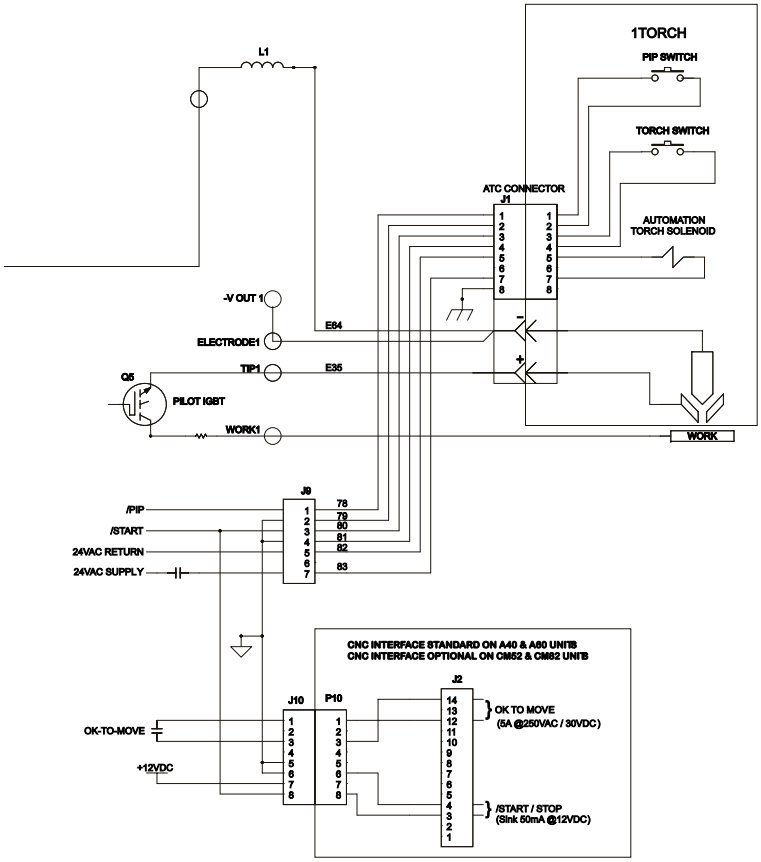
APPENDIX 7: INTERFACE PCB SWITCH SETTINGS (MOST COMMON SETTINGS)

Automation Interface Switch Setting Chart - Common Voltage Divider Output Settings											
SW 4		SW3		SW2		SW1		Volts Out for 100vdc In	Volts Out for 200vdc In	Division Factor	
1	2	3	4	5	6	7	8				
0 = DOWN = OFF, 1 = UP = ON											
Factory Default Settings Suitable for Thermal Dynamics SC-11 Standoff Control:											
0	0	0	0	0	0	0	0	6.00	12.00	16.3:1	✗
Other Common Settings:											
0	0	0	1	0	1	1	0	5.00	10.00	20:1	✓ Preferred Setting
0	1	0	1	0	0	0	1	3.3	6.6	30:1	
1	1	0	0	0	0	0	0	2.5	5.0	40:1	
1	1	1	1	1	1	1	1	2.0	4.0	50:1	✓
0 = DOWN = OFF, 1 = UP = ON											

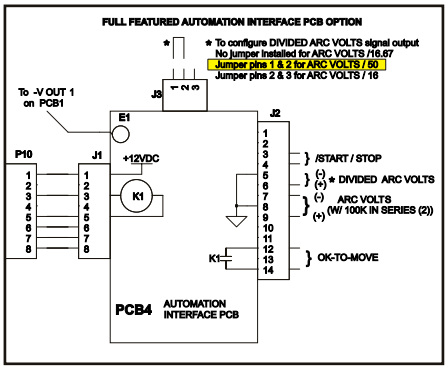
Match THC Sensor PWM setting to this

Thermal Dynamics
AUTOMATION INTERFACE PC BOARD
SWITCH LOCATIONS





Automation Interface PCB (Arc Voltage divider) is an OPTION



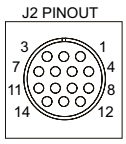
Use the drawing to determine if your Thermal Dynamics Plasma cutter has the rear panel CPC connector and/or the complete Automation Interface (provides Arc Voltage divider). Some 52/82/102 units may have the rear connector and the automation interface. All “A” series units have the rear panel connector with the two signals. Your A series MAY have the Automation Interface Option as well.

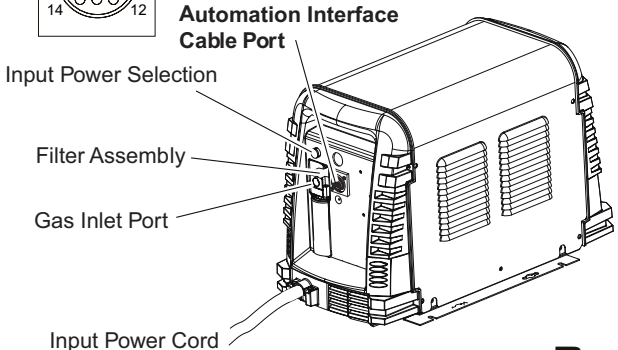
Once you have determined the options you have installed then contact CandCNC to determine what options you may need to purchase.

COMPATIBLE CABLES:

For units with CPC rear connector (only) order a MIC-02 cable

For units with CPC rear connector AND Automation Interface PCB installed order a MIC-01 cable



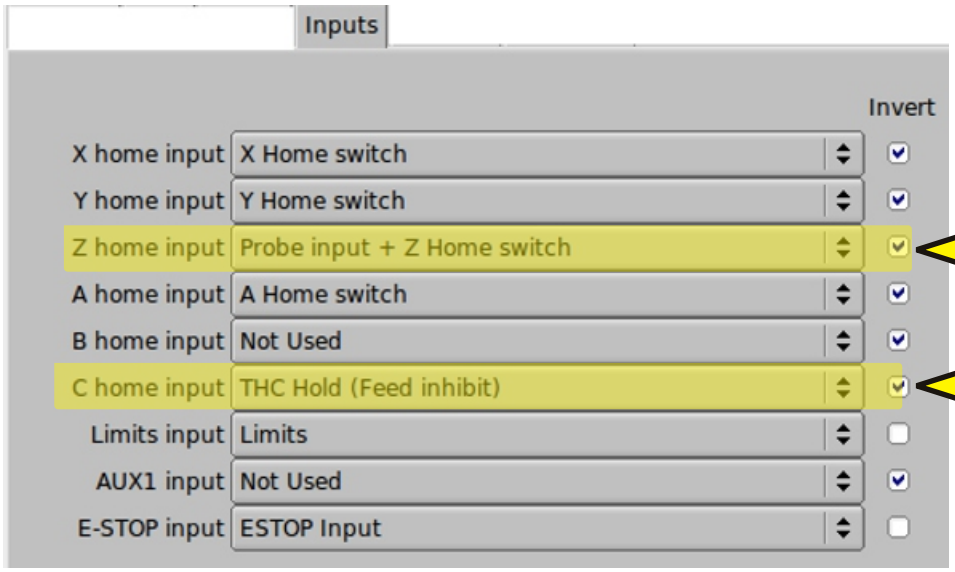


SETTING UP AND PLASMA CUTTING WITH THE DTHC IV SYSTEM

The following pages will take you through the initial setup and testing for CNC Plasma Cutting. There are several critical setups that if skipped WILL create bad results. Please take the time to go though the sections and perform the tests. If at any point your setup fails a test STOP! It is indication that something is wrong and you cannot just ignore it and force it to cut properly.

SETTING UP YOUR Z AXIS FOR AUTOMATIC TOUCH-OFF

✪Make sure your Z is calibrated. So when you move it 1” by the DRO it actually moves exactly 1”. If it does not, find the axis setup section in the manual for your control (BladeRunner AIO User Manual “Motor Tuning Section” for the BladeRunner products) and run the axis setup and calibration. All axis MUST have the correct Steps per Unit setting in CommandCNC.



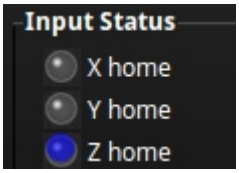
Setup the inputs you have. If you do not have XY or other HOME s then set them to “None”.
The Z home input has to have an input signal for Plasma to do the TOUCH OFF. Set it as shown.

The C home input is used in plasma as the THC HOLD input and MUST be set as shown.

✪Open the **Configurator** and in the Z AXIS tab set the **Home Switch Offset** to 0.00. Save and Exit

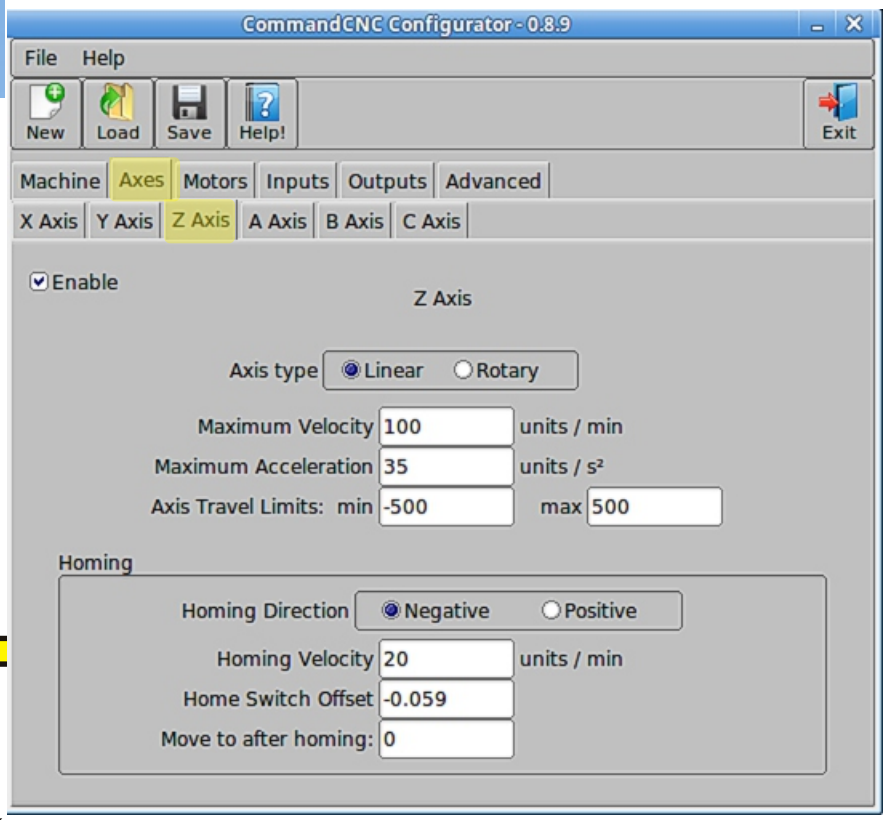
✪Check to make sure your Z HOME is working. **Make sure CommandCNC is out of RESET**. Open to the DIAGNOSTIC Tab and watch the Input Status for the Z home input while you manually trip the Z switch on the Floating Torch Holder. It should light the Z Home LED (only)

.



✪ Position the Z above a piece of flat metal and hit the HOME Z button next to Z DRO
✪ When the Torch Tip hits the metal the Z will continue to move down until it trips the switch. The Z should stop.
✪You may want to lower your Jog % in the CommandCNC Window to 10% or less to slow down the manual jog rate. Use the slide bar for **Jog Speed** (lower left) Carefully jog the Z up using the keyboard hotkey (default is Page Up Key) until you can slip a piece of paper under the torch tip. Write the value in the Z DRO down.
✪Perform the move again and confirm the value. Once you have several readings within .005 then write down the Z DRO reading.
✪The value you have is the Net Switch Offset and will be used as the NEW VALUE in the **Z AXIS Home Switch Offset**. (see page 71 for Configurator screen).
NOTE a “Home” move runs to the switch/sensor and STOPS. It then backs up and moves the Z up to the zero location as defined by the Home Switch offset. Once you program in a Home Switch Offset it will lift the Z and place it so the tipis just touching the top of the material

Sample Numbers: Your values will be different



IF YOUR SCREEN DOES NOTHAVE THE TAB MENU then you need to upgrade your CommandCNC version

Home Switch Offset in CommandCNC is set using the Configurator Application. Open the config (Profile) you are setting up and put in the Switch Offset value into Home Switch Offset. The value you use is determined by running the test sequence by using the HomeZ button and recording the offset. Your value WILL be different . Note: if you are using the FT-01 Feather Touch Ohmic Sensor then the value will be zero (0).

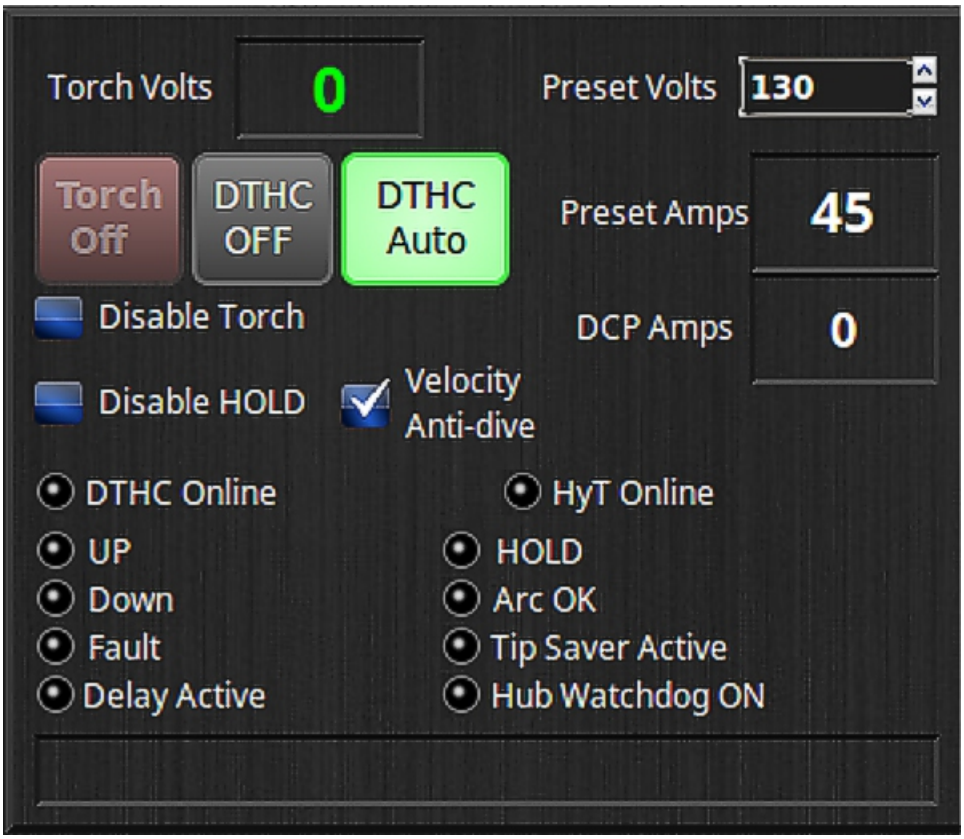
THIS PAGE BLANK

Cut Quality Factors

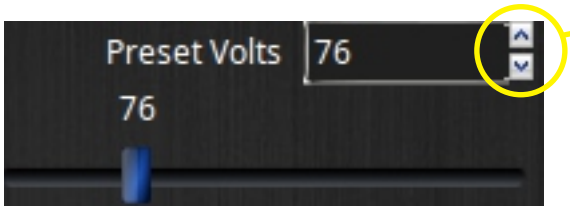
Cut quality in plasma is a function of several factors:

- Clean DRY air
- Sufficient and consistent air pressure (typically 65 to 80 PSI)
- Good consumables
- Accurate Pierce Height (Initial Height Sense aka “touch-off”)
- Proper Cut Height (Proper adjustment of THC voltage)
- Correct Pierce Delay
- Proper Feedrate (cutting speed)
- Proper settings of the DTHC Parameters
- Correct Cut Profile Settings for Span and Tip Saver

If your cut edges are flared in or out, check all but the last factor. If your MP3100-DTHCIV (BladeRunner DTHCIV models) does not respond fast enough you may need to open the DTHC Settings tab and either select the next higher preset or to use the custom tuning. WARNING: The Acceleration in the Z DTHC motor tuning (different than the Z motor tuning in MACH3) is critical to good performance from your DTHCIV. You need to set that tuning via the HUB UTILITY and the DTHC Settings in that location.



PRESET VOLTS. Preset volts **is probably the most important function** since it sets the "target" ARC VOLTS voltage you want to run at. It sets the voltage point that the DTHC IV uses to decide if the torch needs to move UP (to increase volts) or DOWN (to decrease volts). If the TORCH VOLTS from the torch is correctly calibrated than the PRESET VOLTS should

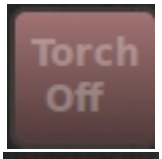


UP ARROW Button: The UP arrow increases the PRESET VOLTS by one full volt per click, in essence raising the torch and increasing the gap. It increments the PRESET VOLTS DRO and **automatically sends the value to the DTHC**. This can be useful since you can change the PRESET VOLTS while cutting which will adjust the CUT GAP at the TORCH. Sometimes a volt ot two of "tweek" can improve the cut. At the end of the cut the Current Settings will remain (for the next cut) and they become the new "Current Settings" in the memory of the DTHC.

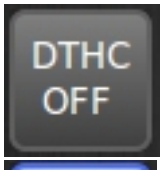
DOWN ARROW Button: Performs the same function as the UP ARROW but decrements the PRESET VOLTS DRO (value). It has the same effect as lowering the torch and decreasing the Tip Volts.

SLIDER BAR Preset Volts Adjust. Does the same thing as using the UP / DOWN arrows on the DRO but is a more course adjustment for making quick and wider changes. You can adjust with Slider Bar then fine tune with UP / DOWN arrows to get single digit adjustments. You can also type in the actual value in the DRO and hit the ENTER key on the keyboard. NOTE: If you are using the DCC options in SheetCAM and the proper POST, Preset volts will change automatically when you load the G-Code based on the settings in SheetCAM for that Job.

TORCH VOLTS DRO (readout) Displays the actual ARC VOLTS at the plasma as reported by the DTHC IV and sent back to CommandCNC via RS485 (C3BUSS HUb) You MUST have RS485 communication before the value will display. If there is no voltage showing and the torch is fired you need to check that the DTHC ONLINE is ON. This voltage should be close to what the cut chart calls for when DTHC is OFF and should be close to the RESET VOLTS with the plasma cutting metal at the proper feedrates and correct AMP setting on the plasma.

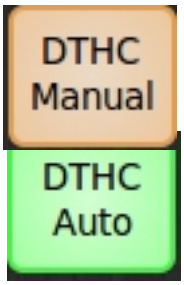


The TORCH ON/OFF button controls turning the Torch ON/OFF. Under normal cutting conditions (e.g. running from code) the Torch ON/OFF is controlled from the software. You have the option of turning the torch on or off manually using the button. The Torch button will fire the torch anytime CommandCNC is out of RESET. The TORCH LED above the button shows the status of the Torch output. When it is illuminated, the signal is being sent to the MP3000 to turn on the torch. Anytime the Torch is on (or should be on) through manual OR software activation that LED should be on.

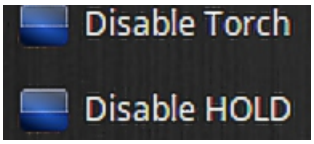


DTHC ON/OFF BUTTON: Unlike the older DTHC and DTHCII, the Torch Height is no longer part of the control program . The DTHC ON/OFF turns off the signals from the DTHC IV (stops moving the Z and does not send the UP/DOWN indicators via RS485 to CommandCNC3. The DTHC IV when coupled with the DTHC IV POSTs for SheetCAM use DYNAMIC DTHC control so the DTHC IV is controlled from the G-CODE and by conditions set by the CAM operator. Several things are automatic. See the addendum on SheetCAM TNG for more information. The operator may override the DTHC from the code but the code may turn it back on depending on the conditions. If you want to run out no DTHC IV you need to make sure you use a TOOL (plasma tool) in CommandCNC with the DTHC disabled.

TORCH HEIGHT CONTROL section (continued)



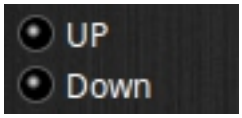
DTHC MANUAL/AUTO BUTTON: Sets whether the DTHC is enabled/disabled from G-Code (Auto Mode) or from the screen button, When enabled the DTHC uses whatever parameters are in the currently displayed settings. You can put the DTHC in Manual mode while you cut and take over control of the DTHC ON/OFF. You can turn the DTHC off with the DTHC In AUTO Mode but it will automatically tunr itself back on on the next cut.



Disable Torch. This prevents firing the torch and is used to run a “dry” cut to test the motion without firing the torch. This also disables the DTHC since th e torch never fires.

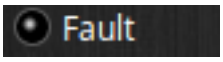
Disable HOLD (checkbox) This is a new feature of CommandCNC release 0.8.9 that turns off the HOLD motion when the torch fires with the DTHC OFF. In a prior release the HOLD function was changed from being ON when the DTHC is active to working anytime the torch is fired and it waits for the ARC OK. This prevented the motion taking off on a manual cut before the torch had established an ARC. It was discovered that SOME circumstances require that the motion start before the torch fires or the arc is sensed . This box when checked will let the motion happen as soon as it is called or the torch fires, regardless of the ARC signal . The setting is temporary and turns back off when you unload the current Config. In normal operation this should be left OFF.

DTHC ONLINE Indicator. This screen LED shows if there is valid communication between CommandCNC and the DTHC IV. All communication is via the C3BUS hub . **It MUST be on to change settings in the DTHC or to see the Torch Volts and UP/Down indicators on the screen. Torch will not fire if there is no communication between the DTHC and the 4 port Hub.**

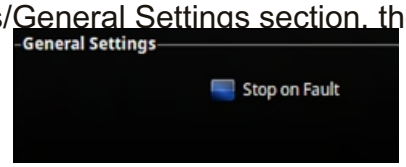


UP & DOWN LED indicators. These two screen LEDS in CommandCNC show the actual UP and DOWN commands CommandCNC is receiving from the DTHC Module. As the torch cuts and with the DTHC active, you will see the UP and DOWN LEDs change. It will tell you at a glance that CommandCNC is getting the proper signals and coupled with the TIP VOLTS DRO show you the activity of the DTHC. In the DTHCIV the the UP and DOWN signals come into CommandCNC via the C3BUS (RS485) since the DTHC IV does not use “hard” inputs into CommandCNC

TORCH HEIGHT CONTROL section (continued)



The Fault Indicator. Shows if there has been a Fault at the DTHC **usually from a voltage fault being** outside the Min or MAX settings values
A FAULT will not turn off the torch or stop motion it simply stops the DTHC from moving. Note: If you have the Stop on Fault checked in the DTHC Stettings/General Settings section, the motion will STOP



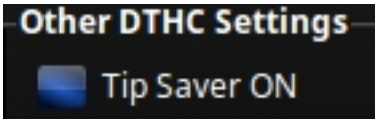
Delay Active Indicator. This screen LED shows when the DTHC Delay (as set by the DTHC Delay value) is active. It is normal to see this come on at the beginning of a cut and go then go off. The indicates the DTHC is not sending any signals. DTHC Delay is used to allow the voltage spike and possible tip saver errors during and right after a Pierce Cycle. The Timer starts as soon as the DTHC goes active (unusually when the torch reaches the defined CUT HEIGHT)

HOLD indicator: The DTHC can issue a HOLD to the motion engine based on DTHC settings. This prevents motion if the torch is not ready to cut. It is normal for the HOLD to be on when the torch has fired but before there is an ARC OK or there is a FAULT at the DTHC

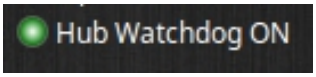
ARC OK Indicator. The ARC OK is an integral part of our cutting system. It is a signal that tells CommandCNC you have a fired the torch and it has a valid arc and you are ready to cut. (See HOLD indicator description). It also detects the loss of arc and CommandCNC will stop movement . If you do not get Arc OK the DTHC IV will not release motion (HOLD will Stay active) and the DTHC will not start processing data to send UP and DOWN commands. Some plasma units have a signal (normally “dry contacts) which are basically relay contacts with no connection to the internal circuit or voltage. On other machines you may have to order the optional DCP-01. It’s a REQUIRED SIGNAL..

Tip Saver Active. Shows the status of the Tip Saver (turns on when the tip saver is tripped) This only functions if you have enabled the Tip Saver in the DTHC Settings tab and the Other DTHC Setttings panel

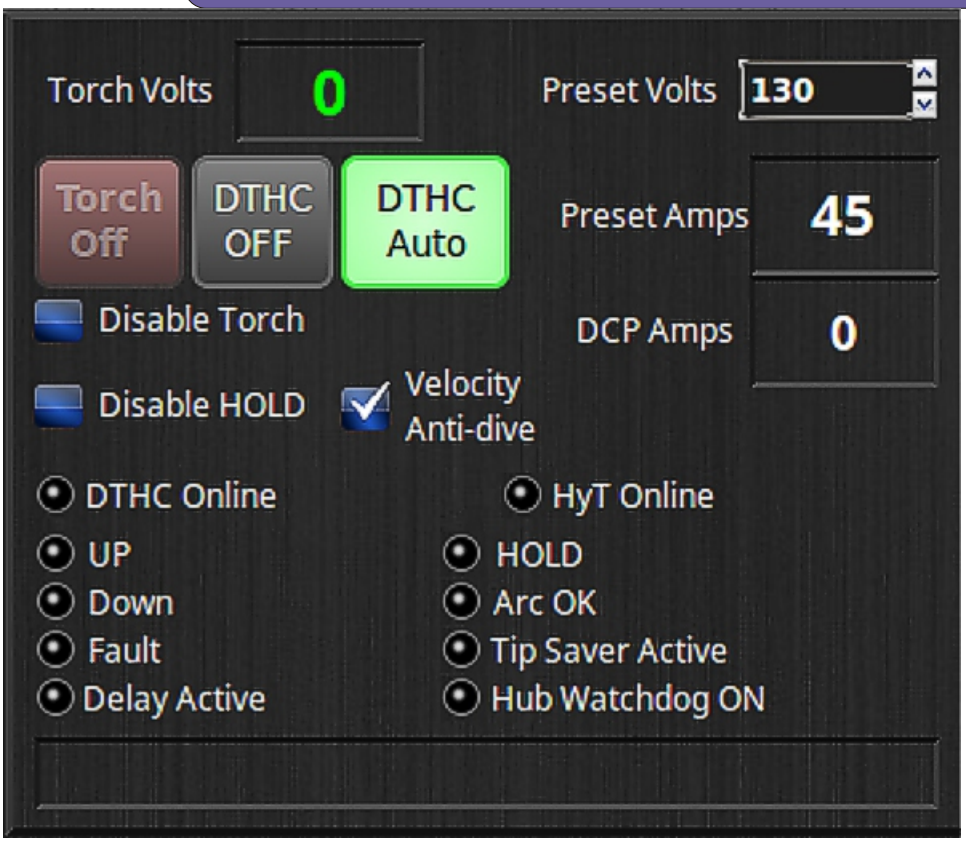
NOTE ABOUT TIPSAVER:
Since the release of the VAD the use of the TIp Saver has been discouraged. It does not work well with a high speed DTHC and is obsolete



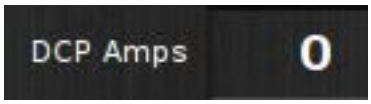
Leave Tip Saver off



Hub Watchdog ON Indicator. Because the RS485 4 Port Hub is an integral part of the DTHC system by providing communication between CommandCNC and the DTHC it has a “watchdog” signal (generated by the hub) when the hub has valid communication to CommandCNC. In the event the hub loses that valid communication the watchdog signal will go away and none of the connected modules (including the DTHCIV module) will be able to work. While the DTHC ONLine shows if the DTHCIV module is talking to CommandCNC and indicates there has to be valid RS485 communications , if it is OFF then a lack of Hub Watchdog

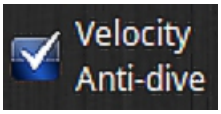


.**Preset Amps** is the "commanded" AMPS called from the CUT Profile Or the dynamic value in the G-Code. **Unless you have the HYT-Connect RS485 option and a serial RS485 port on your Hypertherm this does nothing. This is DISPLAY ONLY and shows you the AMPS that CommandCNC is sending to the Hypertherm. The values come from a preset input (next page) or from G-Code dynamically**



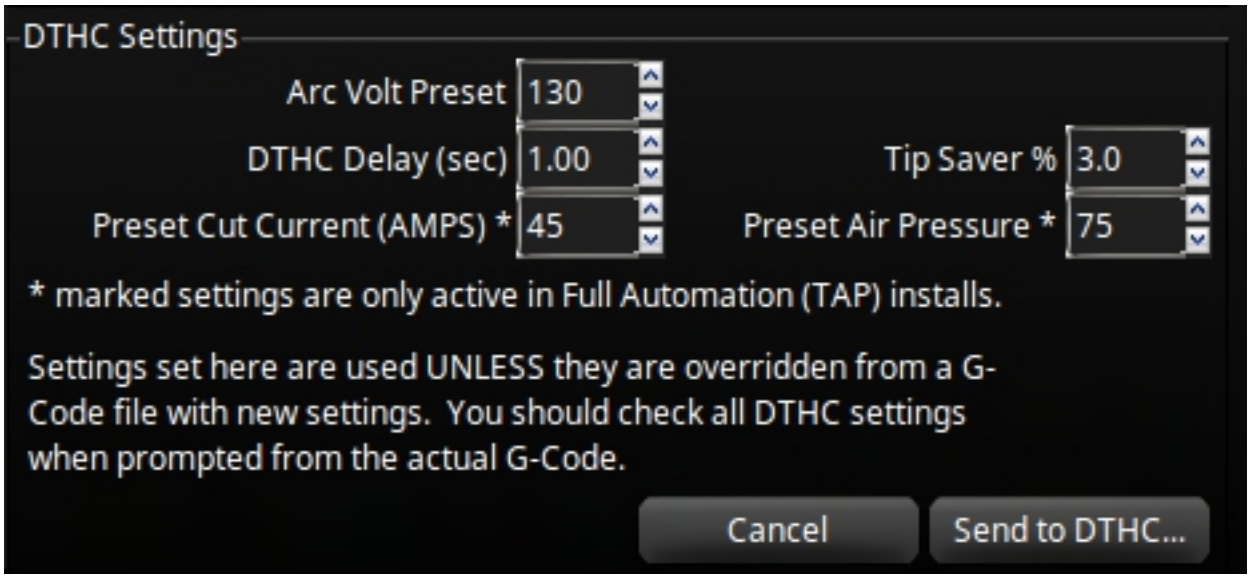
DCP AMPS is an actual readout of the plasma current AT THE CUT in AMPS. **You MUST HAVE the DCP-01 installed and working to get a value in this DRO.**

HyT Online. Indicates that CommandCNC is communicating with the Hypertherm RS485 Serial Port. If the LED is not ON then the Preset AMPS or the Preset Air Pressure will not work. **You MUST have the optional RS485 serial port on your Hypertherm and have it connected to the 4 port hub.** The HyT-connect software is an option



Velocity Anti-Dive. This function is a new and important feature. VAD is based on detecting the slowdown in the feedrate that occurs naturally as the torch approaches a corner or slows down at the end of a cut. The velocity in CommandCNC is handled differently than most controls. There is a setting that tells LINUXCNC how far it can deviate from the toolpath. It gives preference of tracking (accuracy) over feedrate, so it does not round corners or cut across a turn to try and keep velocity as high as possible. **As a result VAD virtually eliminates all corner and end of cut diving. The Settings in DTHC SETTINGS tab/ Other DTHC Settings determine the VAD actions. the Use Hardware Enable Pin MUST be checked. The default 85% works for most cutting. The higher the number the more sensitive VAD is.**

To use VAD you must: Have DTHC Rev 2 or higher (or a REV1 with the MOD (green sticker). Contact the factory if you have a rev1 that has not been modified



SETTING DTHC PARAMETERS BEFORE A CUT

PRESET CUT CURRENT: This DRO is an INPUT (will take a value) and is used ONLY if you have a full TAP™ compliant Plasma cutter (example Hypertherm 65/85/105/125 model WITH RS485 serial port.) This function REQUIRES you have our HyT-Connect RS485 SIMKit connected through the C3BUS to the Hypertherm RS485 serial port which allows remote setting of the CUT CURRENT. This value can be set from this DRO or from G-CODE using the updated DTHC IV POSTS for SheetCAM TNG.

% of Preset Amps: This is an informational DRO (display only) and shows what the current PERCENTAGE of the PRESET AMPS the torch has been commanded (typically from a value sent from the G-Code). This ONLY WORKS with the full TAP™ setup where cut current can be controlled from software dynamically.

DTHC DELAY: The amount of time the DTHC waits to take over after the torch fires. The number will vary depending on the POST used. Newer posts (like the DTHC-HYT- has Auto DTHC Delay TAP_SoftPierce+Marker-rev#.scpost has auto DTHC delay so the DTHC does not take over until AFTER the pierce and plunge to cut height. The normal setting for the DTHC delay is in the 1 to 3 second range. The Auto DTHC delay values should be from .2 to .5 seconds.

IMPORTANT: If you use SheetCAM and the correct POST for LINUXCNC (LCNC) for plasma you can select settings from the Tool table for a cut and it includes ALL of the settings above. They are embedded in the G-Code and load automatically when the code is loaded and DYNAMICALLY set the parameters. The auto settings OVERRIDE the settings you have in the Cut Profile Screen

You must have the DCP-01 hooked up and working to use the CUT CURRENT list below.

USING CUT CURRENT FOR TROUBLESHOOTING A BAD CUT.

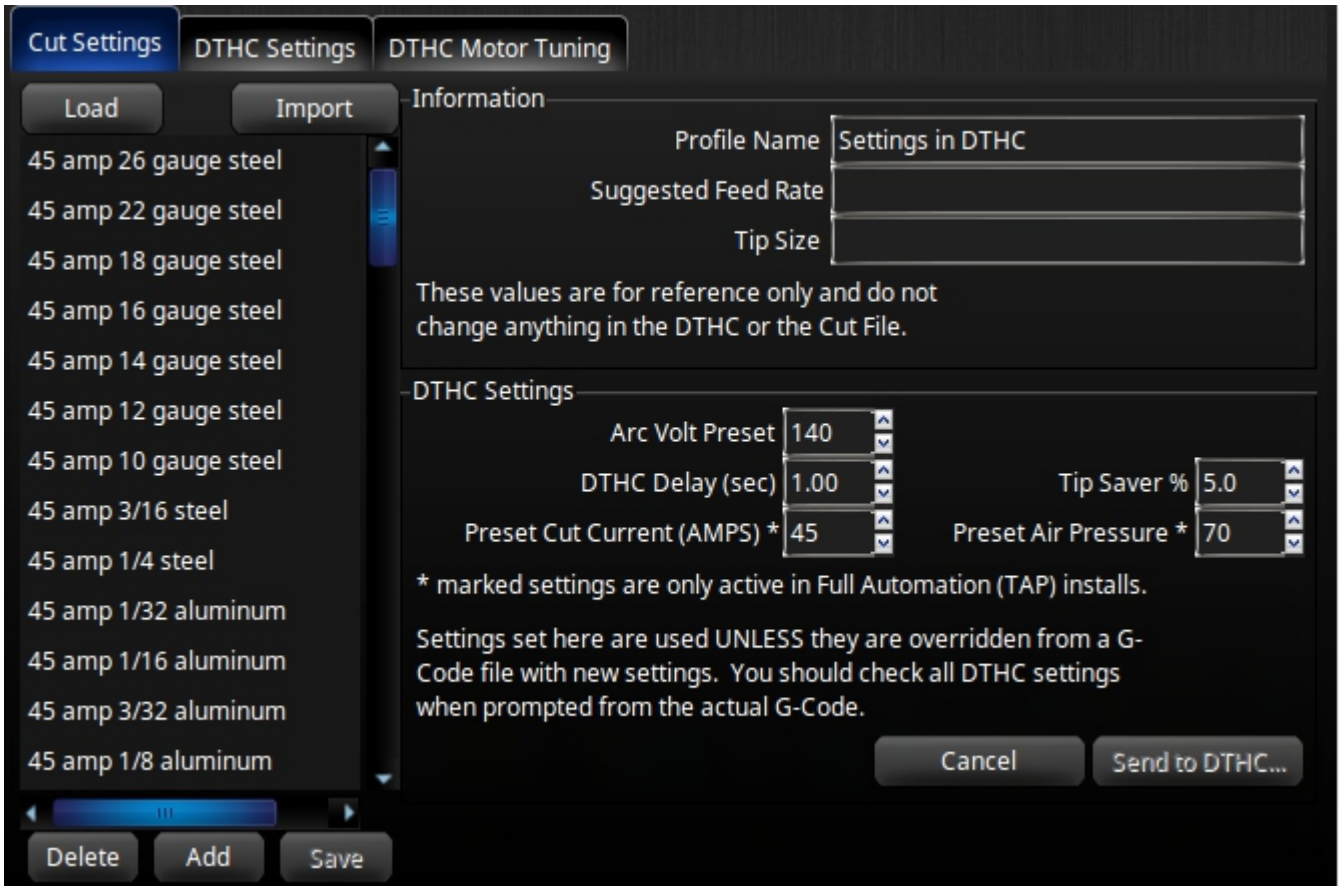
The CUT CURRENT can be a valuable tool to help diagnose and fix poor cutting or loss of arc and other annoying problems. Not all cut issues are from improper current but a good plasma cut cannot occur if the cut current is too far out of the specified value. The whole process of Torch Height Control depends on the current being constant and the value set on the plasma unit.

If you are experiencing problems cutting with the DTHC and it has passed all of the self tests, then setup and make a manual cut at a constant height (the recommended Cut gap) and watch the CUT CURRENT DRO. While cutting it should be close to the value you have set on the plasma machine front panel (dial value). If it is not, here is a list of things to check:

CUT CURRENT TOO LOW	Bad Workclamp Connection (either end) Workclamp not on material Current Setting on Plasma Unit wrong Plasma Current not calibrated to knob
CUT CURRENT VARIES DURING CUT	Bad Workclamp Connection (either end) Bad material (rusty/dirty) Worn defective consumables
CUT CURRENT TOO HIGH	Current Setting on Plasma Unit wrong Constant Current circuit in Plasma Unit not working

CUT SETTINGS BUTTON: Shows the settings in the DTHC memory being used by the DTHC. The DTHC uses a real time processor to process the torch volts and send the proper signals to CommandCNC. The processor stores the settings in NVRAM (non-volatile ram). Changing any setting in the Settings Group or the General THC Settings and using **Send to DTHC** saves the profile (writes it to the DTHC processor RAM. You must close the window to be able to access the CommandCNC screen and to move the machine (JOG) or RUN g-code. **Cancel** cancels any changes you have made before you exit.

The DTHC module "remembers" the settings you used last even if you power everything off. The Cut Profile values are sent to the DTHC module (via an RS485 connection from the PC to the UBOB and up to the DTHC module). The "Current Values are what is in the DTHC memory. Values are transferred when you hit OK NOT when they are just displayed. ONLY the CURRENT SETTINGS values are what the DTHC uses during cutting. If you pull up a profile and do not transfer it to the DTHC module



PROFILE LIST BOX : Shows a list of all saved profiles. Any profile can be selected and those parameters will be transferred to the Current Settings screen. You can add new profiles using the **Add Button**. To delete an entire profile, highlight the profile by clicking on it and hit the **Delete Button**. If for any reason the DTHC module has lost communication with CommandCNC, the settings on the screen ARE NOT SAVED to the NVRAM and opening the screen again to display CURRENT SETTINGS will show the old settings. Check to make sure the THC ONLINE LED is on.

Adding Profiles. As you do cutting on your table using the DTHC Digital Torch Height Control you will be able to choose optimized settings for each type of cutting you do. You can edit and save an existing Cut Profile or add a new one of you own. Since there are variables that change from one machine to another, the sample values may or may not be usable in your environment. It's best to start out with the default settings and use the Tip Volt Preset recommended by your plasma manufacturer. **If your plasma machine has no documentation or recommended cut charts then go to the Plasma Setup section in this manual and use the Initial Setup and calibration methods to establish a base line for building your own charts.** The two most critical components are the feedrate (set in the G-Code and CAM program) and the Tip Volt Preset. Since both values vary between machines, it's best to run a series of tests. Even the cut gap (distance from the tip to the material) and the gap volts (actual tip volts) varies from one plasma manufacturer to another. *Example: A Hypertherm G series calls for .063 (1/16) cut gap and a tip volts reading of 140VDC on 10Ga material with a 40A tip. A Thermal Dynamics unit uses a wider cut gap (about .1 to .12) and lower tip volts (about 110VDC) for the same material.* Other machines will vary. It's best to develop your own values for your machine over time and store them in the Cut Profiles. NOTE: The Cut Profiles are stored in a file named default.cutprof located in the main lhome/<user>/linuxcnc/<profile neme>/folder. It's a good idea to back up your CommandCNC settings , backup the *home/<user>/linuxcnc/configs folder* Restoring values from a backup copy can save hours of frustration.

IMPORTANT INFORMATION: If you use the DCC features available with the DTHCIV when you use SheetCAM TNG and our custom POSTS that expand the Tool Table settings you can store your Cut Profile directly in that JOB file and the settings will be changed **automatically** when you run the code. It eliminates the need to have the operator remember to change to another Cut Profile or to enter new settings at the console. You DO NOT have to be running a special brand of type of plasma to use DCC. It is totally between the Cut Profile settings and the DTHC IV.

The following section on the setup and testing of the DTHC IV Is CRITICAL to the success of having a working table.

CandCNC strives to provide a detailed manual for all setup requirements and it is very important that the manual is read and followed. The DTHC IV in your system has been tested and calibrated at several points along the production process including a final system test of all the products together that you ordered. It is not setup for your specific CNC table/hardware or plasma cutter.

It is imperative that all steps are followed to make sure you are successful in your electronics setup. Without baseline readings it becomes much more difficult to analyze problems and affect cures.

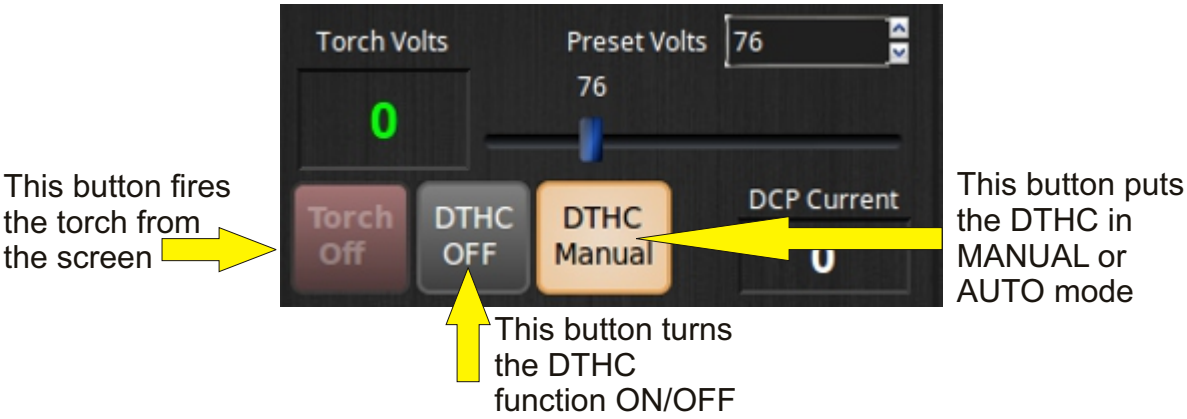
Doing a startup test using a manual cut ....DO NOT SKIP THIS TEST!

To establish that the DTHC is working and to find the best Torch Volts (Preset Volts) setting and initial cut height parameters, you should make a series of cuts dome with the torch off and some with it on. Follow the steps below

1 **Generate a cut file consisting of long straight cuts**. You need enough time to watch the screen indications (DRO readouts and LED's). Use the commandCNC posts file that comes with this install for SheetCAM. Simply draw one or more 16 to 20 : straight lines in your drawing or CAD program. Bring them into SheetCAM and use a plasma tool of the right type for the plasma and material you will be cutting. Set the first start point at X 0 and y 0 /

2. Setup the table with a flat piece of mild steel (12 ga or thicker). Setup the torch with the correct consumables for Automated Cutting. Use the smallest rated NORMAL tip for the material you are cutting. Set you plasma to the amperage for that tip. DO NOT START WITH Fine Cut consumables or try to cut thin material to run these tests!

3, Put the DTHC in MANUAL Mode and turn it off The DTHCIV is DYNAMIC in the fact that it is turned ON and OFF via G-code commands which can override the DTHC ON/OFF button In the current Screens we have provided a way to put the DTHCIV in MANUAL mode (meaning it will ignore On/OFF commands from the G-Code) **You need to have the DTHC Auto/Manual button in MANUAL mode.**



4. **Move to an uncut spot on the plate and zero your X and Y DROs**. Zero Z manually by jogging down slowly and just touching the top of the material . Zero the Z

5.**Turn your plasma torch OFF**

6. Start the code. Watch all of the first moves. Confirm that:

a. The torch goes down, touches the material, stops then raises up to the **CORRECT pierce height.**

b. Watch the LED above the Torch Fire button. When it comes on, look at the TORCH LED on the PWM module to make sure it comes on.

c. Watch to see if the torch plunges quickly down to the proper CUT HEIGHT

d. Watch to see if the torch starts to move in XY at the right height. You can STOP the code at any point and rewind and staart over

IMPORTANT! If ANY of the above moves are wrong or at the wrong height **STOP!** Go back to the section in the DTHC IV manual on ***Setting Up the Z for Automatic Touch-off.*** **At this point you should have your touch-off moves and distance setup and calibrated.** IF YOU DO NOT or do not have a floating torch holder you will not be able to do automated cutting of thinner material.

7. The next test is to establish that the TORCH VOLTS feedback from the torch is correct

a. TURN ON the Plasma cutter

b. Run the same exact test as before (with DTHC on MANUAL and off) and have the torch cut a straight line. The Z should not move up or down during the cut. You should get a good cut .

c. As the cut is made watch the indicators on the DTHC screen in CommandCNC. The ARC OK indicator should be ON. The TORCH VOLTS will display the actual volts at the cut. It should be close to the recommended volts the torch manufacturer calls for. Watch the voltage and pick an average and make note of it. If the voltage is way off from recommended, check the current setting on the Plasma unit. Check the tip size and for excessive wear. Replace consumables if necessary. Make sure you have a GOOD Workclamp connection to the Material (not just the table or cut grid). If the voltage is out of spec by more than 10% go back to the PWM install and setup in the front of the manual. It is EXTREMELY important that the feedback voltage is working properly. If it is not STOP! The DTHC will not work if the voltage is incorrect.

d. Write down the TORCH VOLTS number if it is close to the cut chart number

8. This test starts the testing of the DTHCIV .

a. Move the material or the torch position and re-zero to cut in fresh material

b. On the screen in the PRESET VOLTS readout enter the value you wrote down in 7d above and hit ENTER on the keyboard to lock it in.

c. Start the new cut with the DTHC in MANUAL and DTHC OFF .

d. After the touch off and pierce cycle and the torch starts to cut in XY for a couple of inches turn the DTHC ON/OFF to ON.

e. The torch should remain at the height and stabilize. The TORCH VOLTS voltage should stay within 1 or 2 volts of the PRESET VOLTS. Watch the UP and DOWN indicators on the screen they should be active.

IF THE MOTION STOPS AFTER YOU TURN ON THE DTHC YOU ARE NOT GETTING ARC OK SIGNAL

f. If the torch slams down on the material or raises up you need to go back to the DTHC Settings page in the Hub Admin (page 20 - 24) and check the Z settings and run the test using the Fast Jog buttons on the screen.

If the manual test cuts work and you can turn the DTHC ON during a cut and it starts to function correctly THEN its time to progress to cutting with the DTHC in AUTO mode. To do that you need to change the POST processor you are using to one taht supports the automatic settings. Download and install the SheetCAM Support Installer file from the Downloads section of the www.candcnc.net website. Run it on the computer you use for SheetCAM. It will put a folder on the C drive named ***SheetCAM-CandCNC.*** there is a sub-folder named POSTS. there will be several POSTS. You need to open SheetCAM and OPTIONS/MACHINE/POST PROCESSOR and IMPORT the POST from that folder into SheetCAM. Use the post ending in 11d to 11m (the highest one there) Also in the same folder there will be a PDF file on setting the POST options by using the EDIT POST in SheetCAM.

It's best to start off with larger simple shapes like squares (4 to 8 inches) circles and maybe a star. Generate the G-code and turn the DTHC AUTO/MANUAL to AUTO. The code will handle turning it on and off during the cut. After you have mastered cutting some simple shapes go on to reading about the CUT RULES in SheetCAM. The most important one is a rule built from the "Before End" rule that allows you to turn off the DTHC before the end of cut and avoid any end of cut dives from the voltage spike as the torch slows down at the end.

IMPORTANT CONCEPT: **Torch Volts reflects the Arc Gap (distance between the tip and the material with the torch cutting.** It is what the DTHC uses to measure height. Because the ARC Gap is small it takes very little voltage change to indicate a relative major gap change. Changes of .020 in a gap of .063 are significant! The Preset Volts (Target Volts) tells the DTHC what you want that gap voltage to be. It is not a magic radar that senses the height. The manual test is to establish that the ARC Volts are indeed close to being correct with no adjustment of the torch that would change them. It also establishes a valid setting for the DTHC Target Volts. If you get a number from TORCH VOLTS that is more than 10% out of the recommend volts (on average) or you do not have a chart to work from, then USE THE TORCH VOLTS setting you get from the manual test AS A BEGINNING POINT FOR THE TEST WITH THE DTHC BUTTON ON!

IMPORTANT!

If you have skipped the manual cut test or did not get consistent results from it, your DTHCIV is NOT GOING TO FIX THE PROBLEM AUTOMATICALLY. Setting a random value or using the values in the cut chart and expecting it to just work is a formula for failure.

IF YOU CANNOT GIVE US THE RESULTS OF THE MANUAL TEST, PROVIDING TECHNICAL SUPPORT WILL BE DIFFICULT of not IMPOSSIBLE.

TESTING THE DTHC IV WITH AN AUTOMATED CUT FILE

If you have the Floating Torch Holder setup (auto touch off) then you should have it already setup with the correct values so that it will touch off, raise the Z up to the top of the material and reset the Z to zero (from the g-code). The automated test cut file needs to be generated from a POST that is for CommamdCNC. If you have SHEETCAM, select **any post that is CandCNCPlasmaLLCNC-rev14.scpost or higher REV.** Those posts **ONLY WORK** WITH THE FLOATING HEAD setup. IMPORTANT: YOU CANNOT use previously generated files for the MACH version.

1. Generate a series of basic shapes in your CAD/DRAWING. Process them in CAM to define the cut parameters. In SHEETCAM you define certain parameters for plasma cutting in the TOOL you use to generate the G-Code.

2. Use the DTHCIV settings you derived from your manual tests. Make any adjustments to the PRESET VOLTS one volt at a time using the UP arrow or DOWN arrow buttons on the DTHC screen in MACH.

3. Remember that each tip (nozzle) size and material type/thickness needs different settings to cut properly. The PRESET VOLTS does not set an absolute height, it just defines a height **under specific conditions.** The CUT PROFILES is a Stored Settings Feature that lets you enter and store various parameters for different types of cutting.

4. One of the most frequent mistakes made is either having the current setting on the plasma unit wrong for the nozzle you are using OR forgetting to clip on the workclamp. The DCP-01 will detect those type of conditions and warn the operator.

Note to SHEETCAM (and SheetCAM TNG) users

We have provided special Posts for CommandCNC and the DTHCIV to be used when generating output from SheetCAM. It has an automatic "touch-n-go" feature that reads the traveled distance and once it exceeds 500mm (about 20 inches) a Z reference is performed *just prior to the next pierce*.

You should use the posts named **CandCNCPlasmaLLCNC-rev14.scpost** (or highest rev number) This post is intended for use with CommandCNC and DTHCIV with a floating head setup.

The SheetCAM-CandCNC_Install.exe (Support Install) off the CD is located in the MasterInstaller[root]/SheetCAM-CandCNC-Support folder or as a download off the Candcnc website. It adds in new POST processors and a full toolset for most Hypertherm Plasma Cutters. Please read the addendum on SheetCAM TNG for an overview of new features these new POST take advantage of.

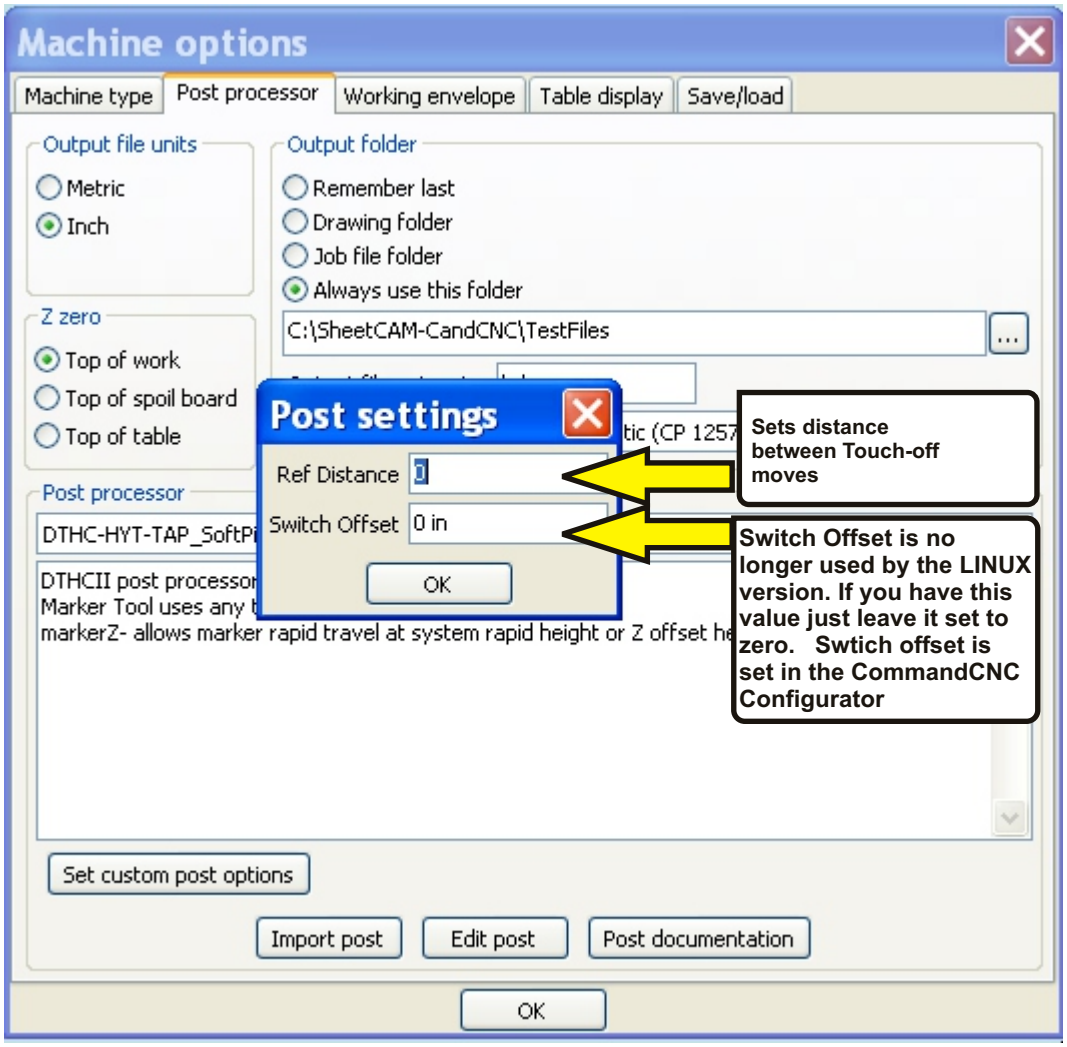
A note about SheetCAM TNG for Windows and SheetCAM TNG for LINUX; You can use either version to generate G-Code for CommandCNC. Because CommandCNC is based on LINUXCNC the G and M codes vary so you cannot use code that was generated for MACH or other CNC control systems. SheetCAM LINUX will open job files made in SheetCAM Windows .

Switch Offset is the distance past the top of the material that Z travels when using the floating torch holder. **It is no longer set in the POST or via the *Custom Post Options***

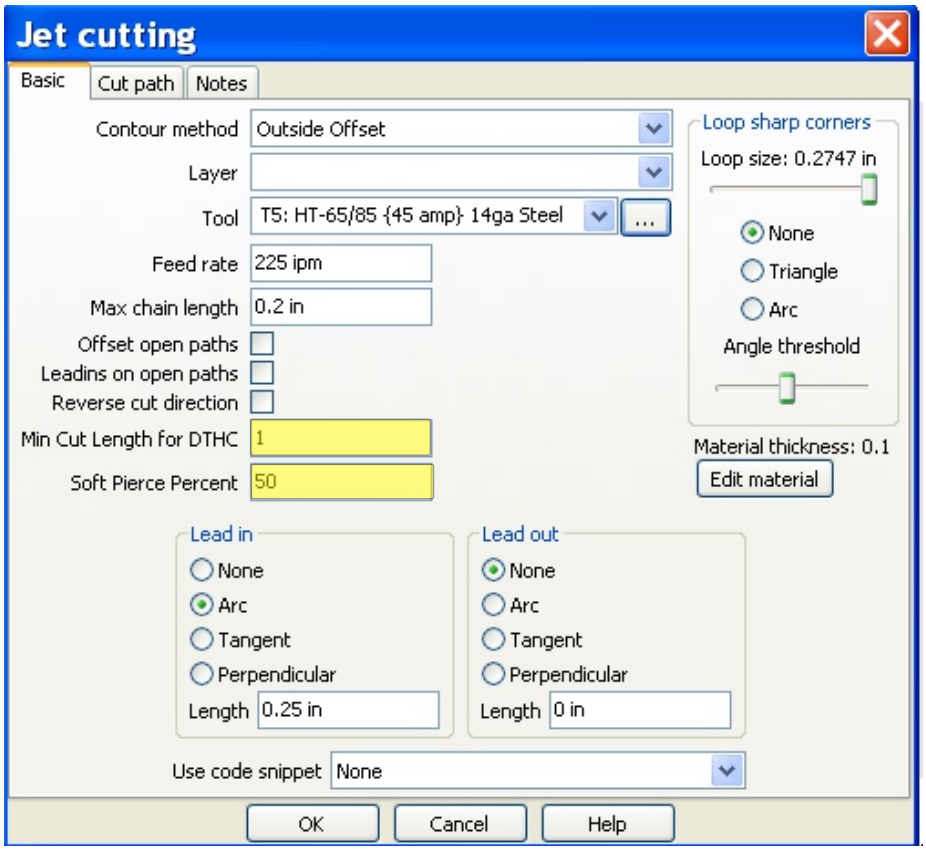
Refdistance is the distance you let XY travel before you do the next touch-off sequence. You can change how close (and how often) that sequence happens by raising or lowering the value. Scale in this context is 25.4. Refdistance in in mm so in the example above the actual distance is 254 mm (about 10 inches). That is the combined distance of both X & Y movement. On thin material that may need a touch off before every pierce that the number to 0. Save the POST with another name and select it when doing your final CAM post to G-code.

NOTE: Latest CandCNCPlasmaLLCNC-rev14.scpost coupled with SheetCAM TNG 6.0.14 (released 4/7/15) or later, has the refDistance options as POST parameters. if you have the following screen in the Options/Machine/Post Processor window set the parameters there.

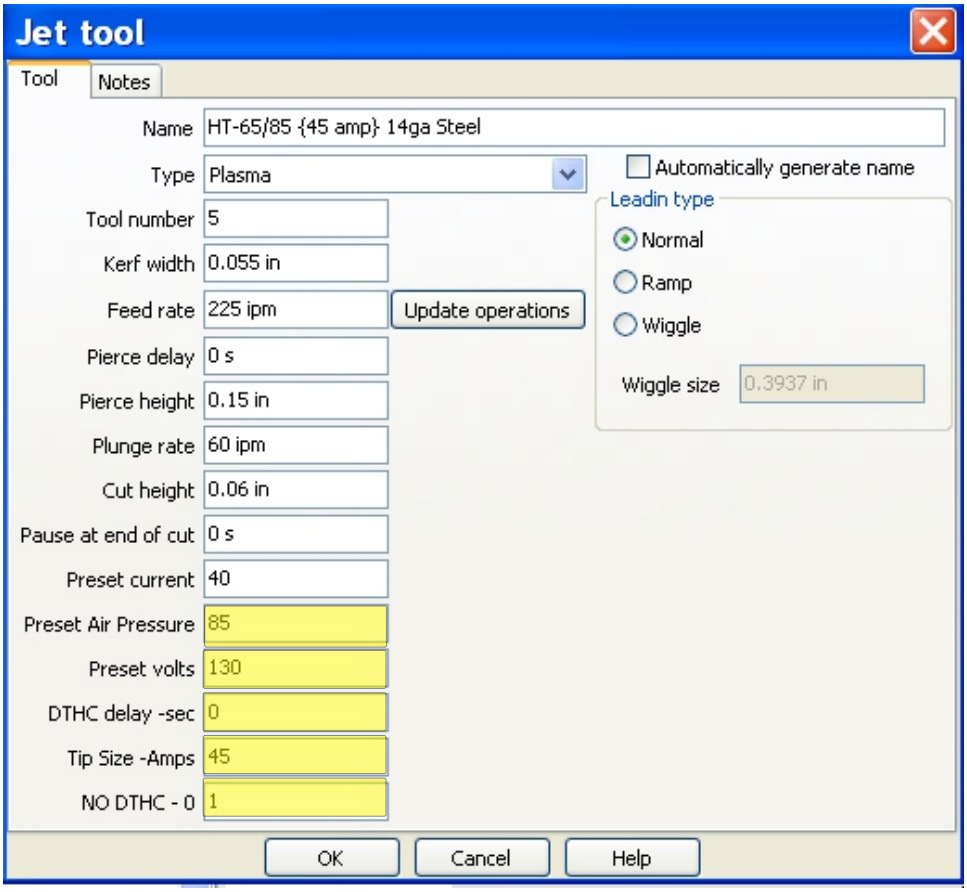
This new option no longer requires you to manually edit the POSTs you use to add in the variables but you must do it for EVERY post you use.



For non-SHEETCAM users.
You need to have a CAM program that offers a LINUXCNC Plasma Post with the automated touch off. If you want to do your own G-Code programming or adapt an existing CAM program you will need to open and study the “G-Code Quick Reference “file located in the “CNC” folder on your LINUX PC (installed with CommandCNC). To see what codes are used and when its advised to go ahead and load the demo version of SheetCAM and use the CandCNCPlasmaLLCNC-rev14.scpost SheetCAM post to process a simple job with a square and circle. SheetCAm is limited as to the number of lines it will process in DEMO mode so the file needs to be simple. Printing out the G-Code it generates will give you a roadmap of LinuxCNC specific G-Codes.



NOTE: Not all the parameters you see will appear on your screen. It depends on which POST you are using because the POST defines added tool parameters and other features (like Cut Rules) . If you do not have the added (highlighted yellow) parameters you should make sure you have the latest POST processors from CandCNC and then use the OPTIONS/MACHINE /Post Processor and import post processor to load and use the POST.



Kerf width: Sets the width offset used for inside or outside cuts. If you don't know the value go back to your manual cut you made and measure the width. Note: width will vary based on: Nozzle size used and age of consumables, Arc gap (height), and feed rate of the cut. You can expect an AVERAGE width that will remain fairly close if you hold the listed variables to a narrow range.

Feed Rate: The recommended feed rate for this material and settings. This is a default value and can be changed when you build the OPERATION to match the material you are cutting.

Preheat: Not used for plasma.

Pierce Delay: Zero for material thinner than .187. The Pierce Delay is a total of this setting, PLUS the time it takes the ARC OK to light, PLUS the time it takes the torch to PLUNGE (at the Set rate) from Pierce Height to Cut Height. Excessive pierce delay can result in voltage spikes that will "confuse" the DTHC and cause the TIP saver to lock on or the torch to plunge. Whatever the recommended pierce delay is on your chart make sure you subtract the cycle time of the numbers above.

Pierce Height: The recommended pierce height for your plasma. Usually 2 times the recommended cut height on material thicker than .125 (3mm). This defines how far the torch will lift above the material after a touch off.

Plunge rate: How fast the Z moves down from pierce height to cut height. This value should be close to the normal Z VELOCITY (motor tuning value in MACH3 for Z), not the Z velocity set in the DTHC IV tuning described earlier.

Cut Height: The beginning cut height before the DTHC takes over after its programmed delay (default 1 sec). This is the normal "stand off" distance the plasma torch manufacturer recommends for automated cutting.

Pause at end of cut: A pause after the torch is turned off from MACH to let the arc die out and voltage to go to zero. Recommended ½ sec to 1 sec.

ADDED PARAMETERS (based on POST Processor used in SheetCAM TNG)

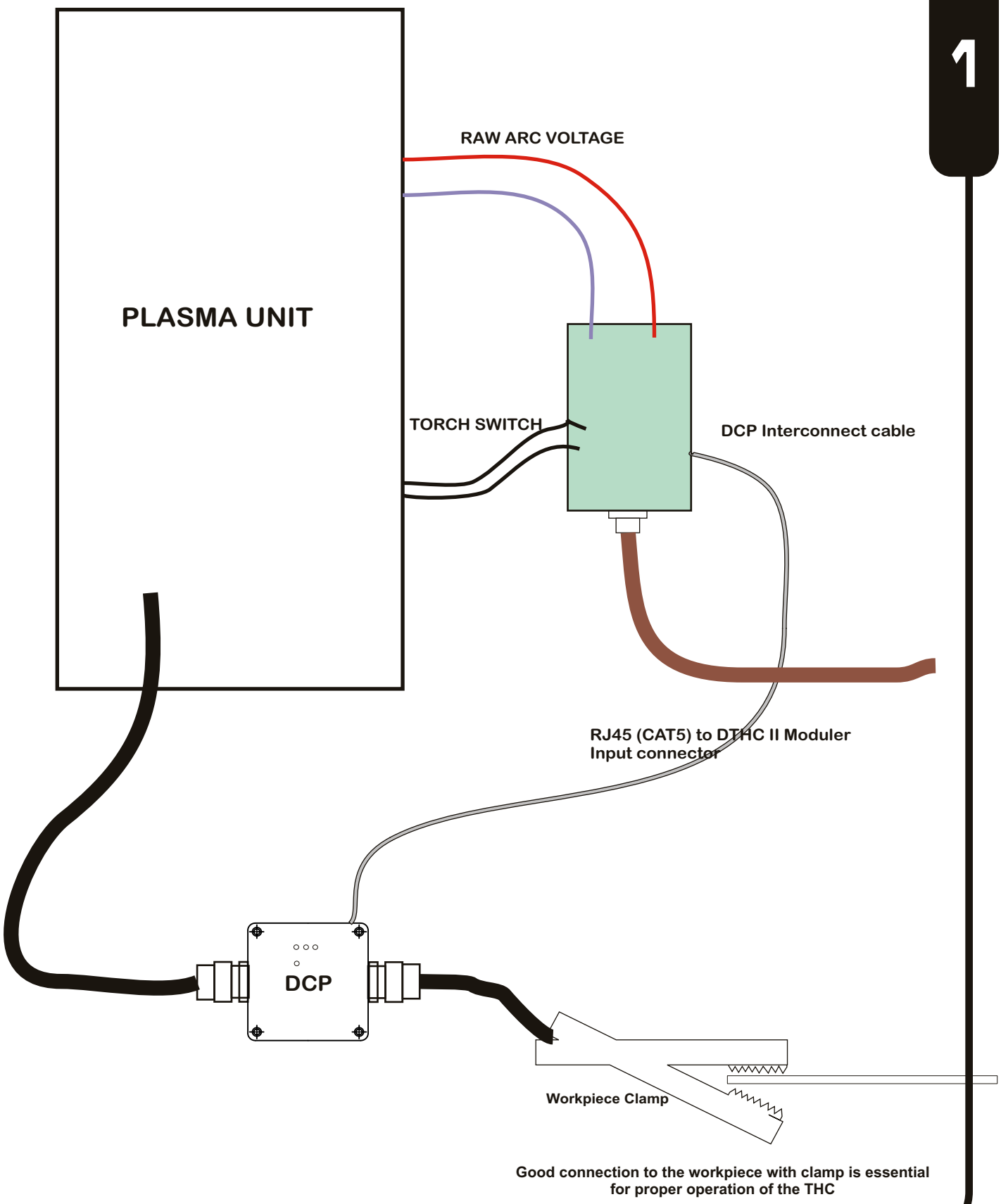
Preset Current. The cut current the plasma will be set to begin the cut. You MUST HAVE A PLASMA that has the ability to be remotely controlled from the tool table settings via G-code. That requires the RS485 serial communications option on the plasma cutter AND the CandCNC "TAP" setup (HyT-Connect RS485 SIM Kit). **Just because these parameters may appear in your tool table they will do nothing unless you have the full TAP (Total Automation Plasma) options fully implemented.**

END OF

DTHC IV -SETUP/INSTALL MANUAL FOR COMMANDCNC PART 1

ADDENDUM

1. DCP-01 Digital Current Probe Install & setup
2. Grounding Practices for plasma noise suppression
3. Troubleshooting Charts
4. FT-01 Feather Touch (revised)
5. Installing and using HYT-Connect RS485 serial Option for Hypertherm
6. SheetCAM DCC Settings and Cut Rules
7. Hypertherm ToolSets
8. POST Processor - Setting POST Options in the POST
9. Peck Piercing using Plasma Torch
10. Plate Marker for Center Punch
11. Routing with CommandCNC
12. Oxy-Fuel Configuration in Commandcnc


PLASMA UNIT
RAW ARC VOLTAGE
TORCH SWITCH
DCP Interconnect cable
RJ45 (CAT5) to DTHC II Moduler
Input connector
DCP
Workpiece Clamp
Good connection to the workpiece with clamp is essential
for proper operation of the THC

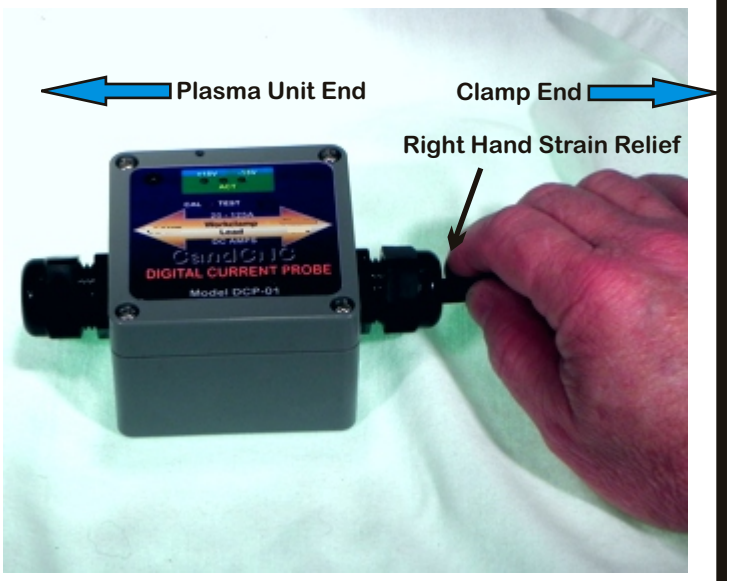
Physical install and hookup

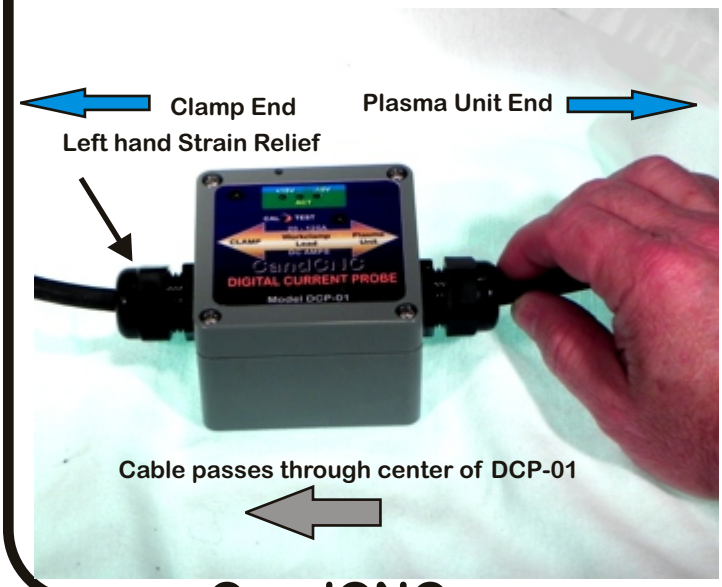
DCP-01 Digital Current Probe uses the existing THC SENSOR (rev 14 or above) and slips over the Plasma System Workclamp lead wire to provide DC amp feedback to the DTHC and to the MACH screen. The following series of photos shows the DCP being hooked up to a plasma cutter.

Start by removing the ground clamp from your plasma work clamp. If it has a large ring terminal on the end where it attaches to the clamp you will need remove the ring terminal and replace it with a new one after you push it through the DCP unit.

Loosen the cable strain relief/clamps on each end of the box by twisting the retainer nut counterclockwise viewed from the end. The strain relief has an internal collar that clamps down on the wire the more it is tightened.

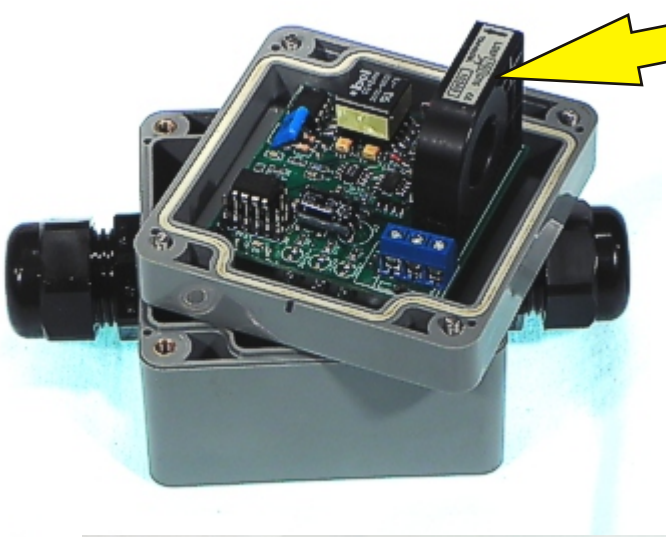
Straighten the wire as much as possible and start feeding it from the right hand side.





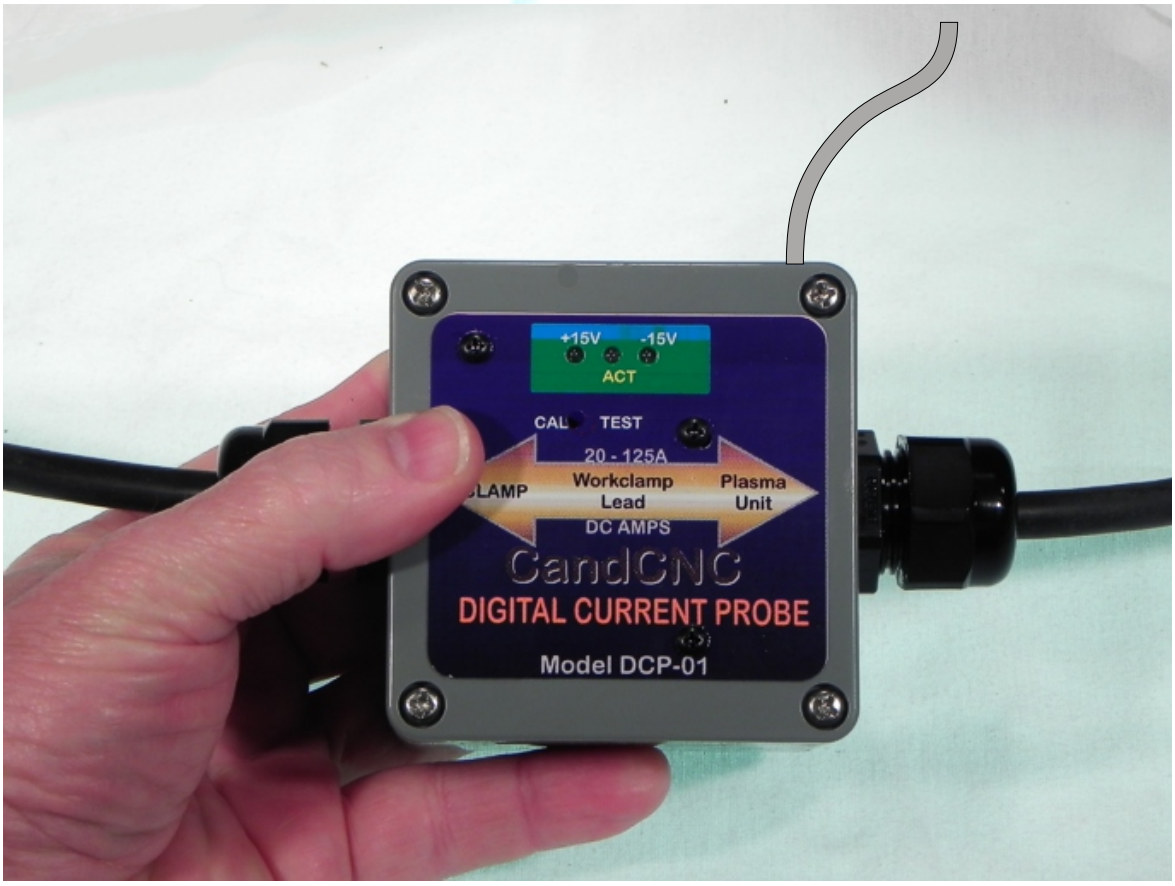
Push the wire through gently. If it hangs try rotating the wire and pushing but do not force it. There is a round hole in the hall probe inside the box that the wire has to pass through. It is located close to the right side of the unit. If you have problems getting the wire to go though pull the wire out and remove the 4 screws holding the top and remove the top and the PCB with the Hall Probe (see next page). Thread the wire though the right side strain relief and pull enough through so you can thread the Hall Probe on the card and then over and out of the left side strain relief.

Digital Current Probe
Model DCP-01



Workclamp Lead Wire goes through Hall Probe center as shown. Top shown flipped 180 degrees.

NOTE: Your DCP-01 unit will ship with an interconnect cable to connect it to the PWM Module. If you remove the top cover pull carefully to prevent breaking or disconnecting the interconnect cable. If it comes loose it fits in a RJ11 jack on the board. The cable is a standard 6 conductor flat DATA cable (straight through wiring) not a Phone cable



When you have the Workclamp lead wire threaded through the box, hand tighten the outside nuts on each strain relief until it is tight around the cable. It is important that each end is sealed to keep out plasma dust and smoke. Position the DCP-01 along the cable close to the plasma unit and in a place it will not get stepped on, crushed or can be dragged across the floor if you decide to move your plasma unit or use it manually in the shop. The enclosure is sealed and rugged but it can be damaged by excessive abuse.

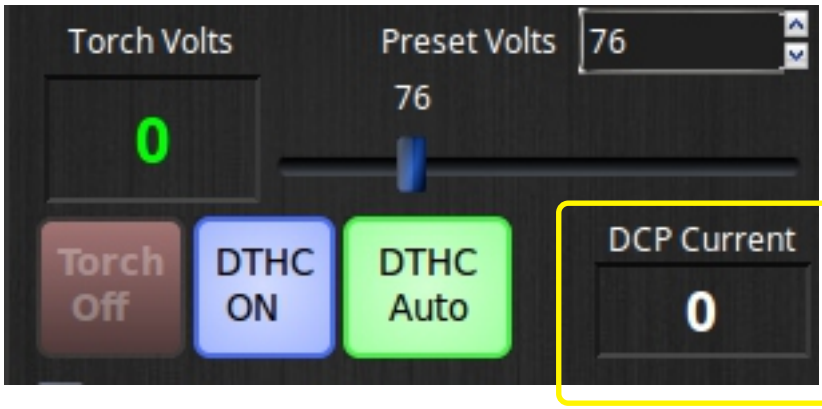
SETUP, TESTING and CALIBRATION of the DCP-01 DIGITAL CURRENT PROBE

Do the following steps in order:

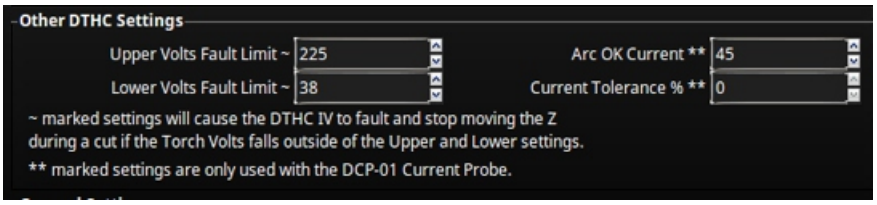
1. Connect the DCP-01 interface cable (RJ11 flat silver) to the THC SENSOR PWM front panel jack labeled DCP. You should get a positive “click” as the cable is seated. Removing the cable should require releasing the locking pin.

2. If you are connecting up your DTHC II and THC SENSOR PWM for the first time, make the connections to the THC SENSOR PWM Module and the CAT5 cable back to the DTHC II module front Panel connector on the MP3000-DTHC or the BladeRunner Dragon Cut. Do this BEFORE you make the Tip Volts (Arc volts) connection or the TORCH SWITCH. You can run some tests on the THC SENSOR Card and the DCP without having the plasma unit turned on or the Tip Volts present.

4. Start MACH3 on the controller PC and load the profile and screen set for the DTHC. If this is the first time you have used the DTHC and CommandCNC you should have had CommandCNC loaded and setup. If you do not, STOP! Go back to the MP3000-DTHC or the BladeRunner AIO manual and first setup CommandCNC and get your table moving and the proper CommandCNC profile loaded for your system.



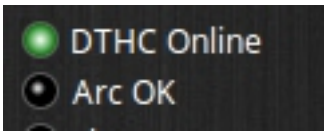
Readout of DC Cut Current when using the DCP-01 Digital Current Probe. This will not display a value if you do not have the DCP-01 connected and working. It's important that you have gone through the DTHCIV setup in this manual FIRST and that you have confirmed that it is working properly.



5. Power up the MP3000-DTHC or the BladeRunner (or your UBOB Builders Kit + DTHC) so the DTHC module has power. The first thing you should see is that the +15 LED and - 15 LED on the front of the DCP lights up. The ACT (activity) LED will NOT be on.

6. Bring CommandCNC out of reset. On any unit with our ESP smart power controller (including BladeRunners) you MUST have the DC power to the motors on to come out of reset. At this stage you should have CommandCNC setup, running and know how to come out

7. Make sure that the DTHC module is communicating with CommandCNC. The DTHC ONLINE LED should be ON (green) **When you activate the DTHC II Self-test the TORCH VOLTS should change from 100 to 150 and the THC UP and THC DOWN LEDs (and ARC OK) will alternate off and on.** The TORCH VOLTS comes from the DTHC **across the PC serial port connection to CommandCNC. Without the serial communications you will not get Torch Volts and you will not be able to see TORCH AMPS.**



PAGE 6

8. Once you have determined the DTHC passes self-test then locate the small hole in the front of the DCP unit (marked CAL and TEST). You will need a small screwdriver or probe tip (paperclip?). You will feel the button click. Push it once to turn on the TEST/CAL function. When it is active the ACT LED (yellow) will flash.

TO TURN OFF TEST/CAL MODE on the DCP-01 push the recessed CAL/Test button once and the ACT LED should stop flashing and the TORCH AMPS readout should return to 0

Calibration/Test Button (recessed)

9.. While you are in TEST/CAL mode (LED blinking) the TORCH AMPS should display a value. If you are installing a DCP on an existing product with a DTHC the calibration could be off so the number you see could be anything from ___ to ____. If your unit displays 100 amps as shown then you continue on to final testing and CUT PROFILE Setup. IF YOUR CURRENT IS NOT 100 you will need to proceed to the CALIBRATION SECTION.

CALIBRATION of the DCP-01
Use this section any time you put the unit into TEST/CAL and the displayed value in the TORCH APMS readout is NOT 100 as shown.

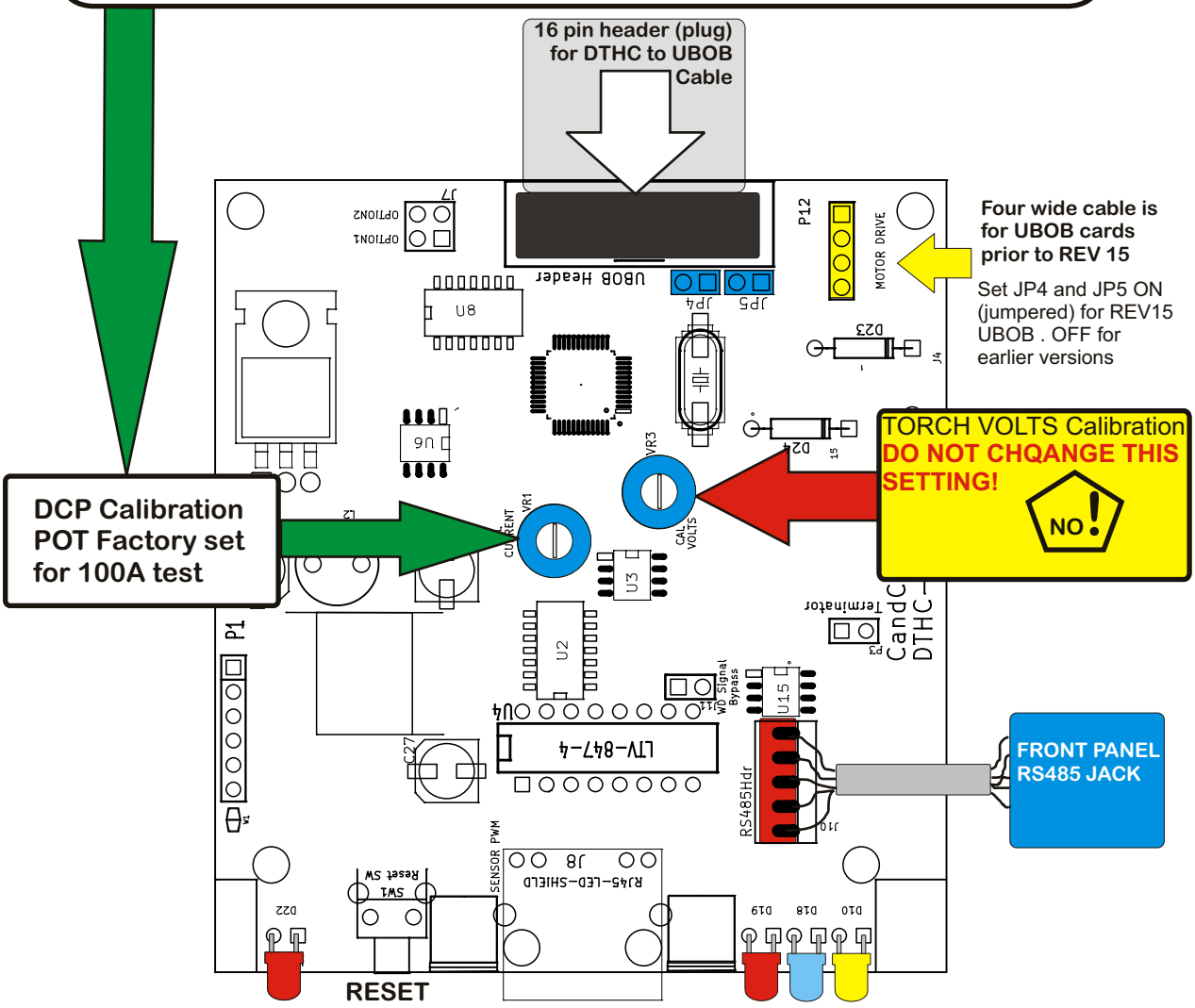
1. To calibrate the DTHC module to the DCP-01 you must have access to the top of the DTHC Expansion Module. Refer to your product manual to identify and access the DTHC module. It is the small PCB card behind the panel on all CandCNC units where the CAT5 cable from the PWM Sensor plugs in. In most cases all you have to do to gain access is remove the top cover (MP3600-DTHC) or the front panel (BladeRunner AIO). Use the photos below to find and identify the DTHC card and the correct adjustment point for the DCP. CAUTION: there are two identical pots (variable resistors) in the card. One is the DCP calibration pot. ***The other is the TORCH VOLTS calibration pot and is set at the factory DO NOT ADJUST THE WRONG POT. IF YOU ACCIDENTLY CHANGE THE TORCH VOLTS (wrong pot) you will throw your DTHC unit out of calibration and you will need to run the TEST/CAL sequence on the THC SENSOR PWM module and reset the displayed volts. NOTE: SOME later rev's of the DTHC II module may not have the arc volts calibration pot. If it does not exist then ignore the above procedure.***
Check the photos. Study the board orientation and MAKE SURE you are adjusting the pot. This calibration should only have to be done once so take the time to do it right.

DCP Calibration (Cont)

1. DTHCIV EXPANION MODULE. Located above the UBOB III card in most CandCNC products. Ribbon cable connecting DTHC IV down to UBOB card may cover adjustment pots. If so, gently move it out of the way. Do not unplug the ribbon cable or the card will be disabled.

2. Using the diagram below and with the DCP in the TEST/CAL mode (LED flashing) adjust the DCP calibration pot while watching the TORCH AMPS DRO in the CommandCNC screen. Adjust the pot until the value displayed is 100 AMPs.

Your calibration procedure is complete!

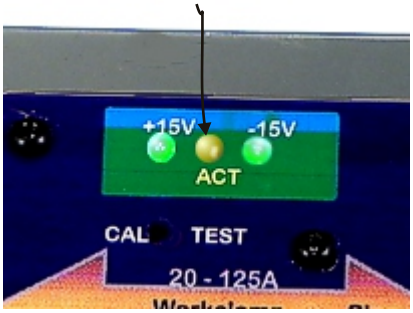


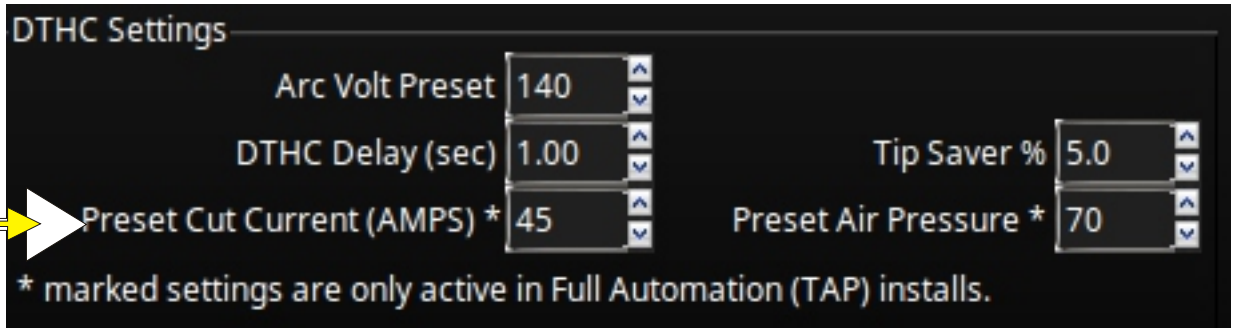
OPERATION of the DTHCII with the DCP-01.

The primary purpose of the DCP-01 is to give the operator real time feedback of the actual Cut Current. Using the settings in the Cut Profile you can set fault points (based on a percentage of the variance from the current preset value) to warn the operator if cut current is too high or too low. The ACT LED on the front of the DCP-01 (when not in Test/Cal mode) will light up anytime the detected current is above 20A. It's just a visual indicator that the DCP-01 is reading current. If the LED lights when you fire the torch and have a valid arc and you do not see the Current displayed on the TORCH AMPS DRO on the CommandCNC screen then go back through the test and calibration section. If you see cut current out of range or get a Current Fault then check the work clamp connection, the current setting on the plasma unit, the consumables, the Cut Current setting in the Cut Profile (stored settings) and determine why the current is not what it should be.

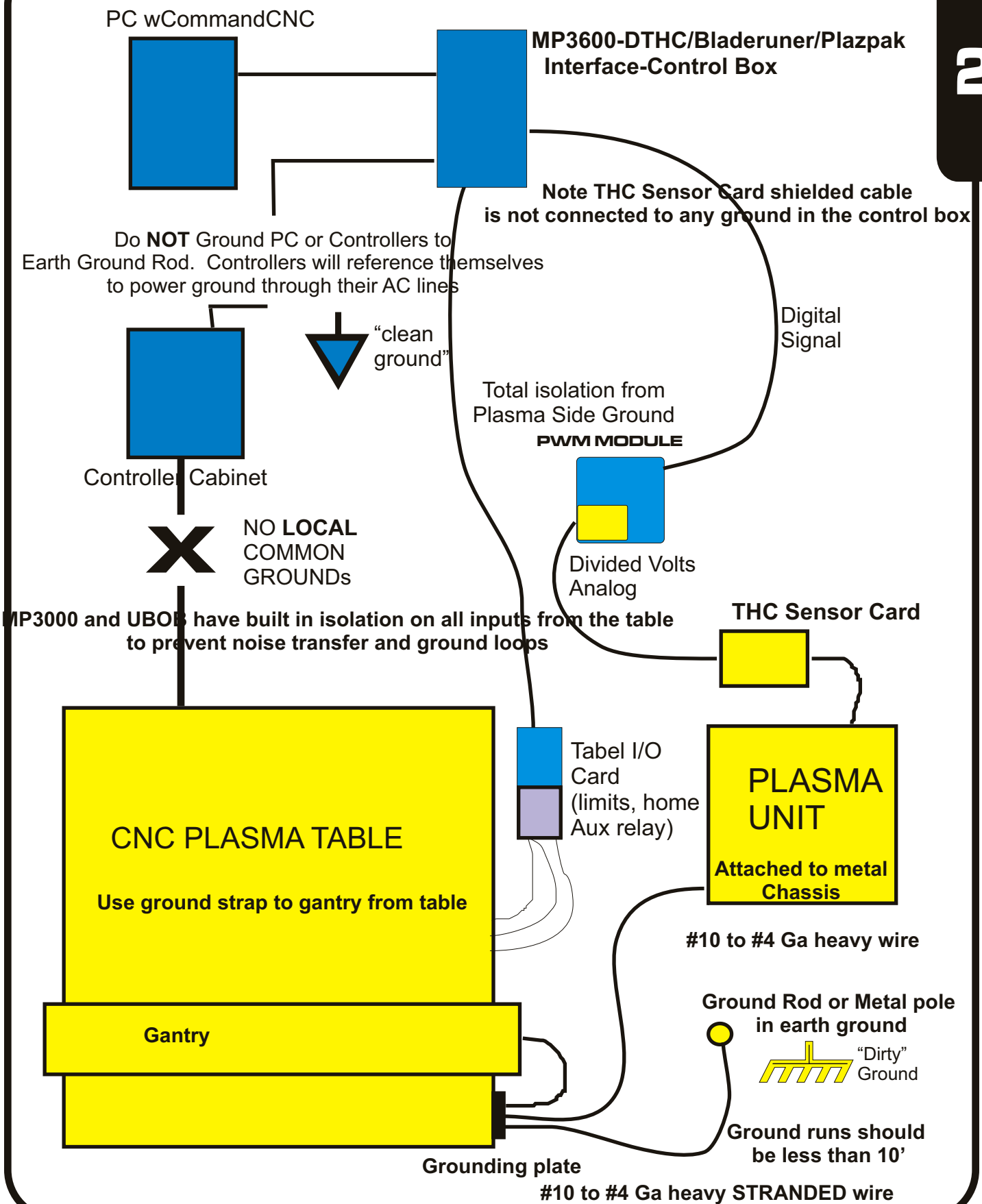
It's important to understand that the Preset Cut Current(AMPS) does NOT *set* the cutting current UNLESS you have the full TAP options with a Hypertherm models 65 thru 125 and the Hyt-Connect RS485 SIM Kit Option. Only the manual adjustment on the plasma unit sets that value. The DCP-01 just tells you what the cut current REALLY is at the cut and tells you if the value is not what you have set in the specific Cut Profile you are running.

This LED comes on any time detected current is 20A or greater





Grounding practices to reduce noise and increase safety



1. DTHC does strange things after a pierce (TIP SAVER locks on or tip plunges to the metal.)	✍ DTHC settings are wrong (wrong PRESET VOLTS). ✍ Pierce delay too long ✍DTHC Delay in CUT PROFILE too SHORT.
2. UP and DOWN (LED's) not coming on (no Z movement).	✍Arc OK not working. DTHC not working (run self-test). CommandCNC inputs not working.
3. UP & DOWN LEDs work but no Z movement.	✍DTHCIV Z motor tuning wrong ✍
4. Z moves but erratically or loses steps.	
5. UP or DOWN LED is on but Z stops moving (won't go further down or up). Torch won't cut low enough or high enough even though the UP or DOWN is on and THC Button is ON	✍Z motor tuning wrong. **Max or Min Volts setting is wrong** ✍. NOTE: MAX and MIN settings are in "units". If y
6. Z DRO does not agree with actual height at the end of a cut.	✍Pierce height in G-code (from CAM) is set wrong. Touch-off values (switch Offset) are wrong. Z is losing steps during a cut (see #6)
7. Torch pierces at wrong height.	✍
8. Torch cuts too Low/High. UP and Down and Z is working.	✍PRESET VOLTS is wrong for the material, tip and feed rate you are using. Preset needs to be adjusted in 1 volt increments and in the same direction as the error. Raise PRESET value to raise the torch. Lower PRESET VOLTS value to lower the torch. DO NOT GUESS AT A VALUE! Run the manual cut test to establish the proper PRESET VOLTS value OR run the SMART-KUT option (one time) if all of the pierce heights and beginning cut height are correct.
9. TIP SAVER comes on and stays on (it is normal for the TIP SAVER to flash on/off especially at the end of a cut or if the feed rate has slowed down. Only make changes if the cut quality is poor.	✍TORCH VOLTS is above the PRESET. Turn off the TIP SAVER or increase the percentage in the CUT PROFILE and try the cut. If it holds the correct height then either increase the THC Delay in the cut profile (NOT CommandCNC) or increase the Tip Saver percentage in the CUT PROFILE.

10. Torch oscillates wildly UP and DOWN while cutting.	✍ Torch is overshooting. Use the DTHC Settings tab in the CommandCNC Cut Profiles section (page 49) and lower the DTHC Tracking and DTHC Sensitivity until the oscillations stop.
11. Torch slowly rises UP while cutting	✍Check Torch Volts reading. IF it is above the PRESET VOLTS then the DOWN LED should be on. Also see the TIP SAVER locking on issue.
12. DTHC works fine sometimes and then does not other times using the same parameters.	✍Possible noise problem. Make sure you have your table and plasma well grounded with a local ground rod. Try not to run your PC and Controller out of the same subpanel as the plasma. keep the grounds for each side separated locally.
13 Torch runs for a while then shuts off while cutting or does not stay lit after the pierce.	✍Check the Indicator LED on the THC SENSOR card for the TORCH ON (relay) It should remain ON steady. If the torch still turns off (with it on) then the problem is with a connection (torch switch terminals or a problem at the torch.

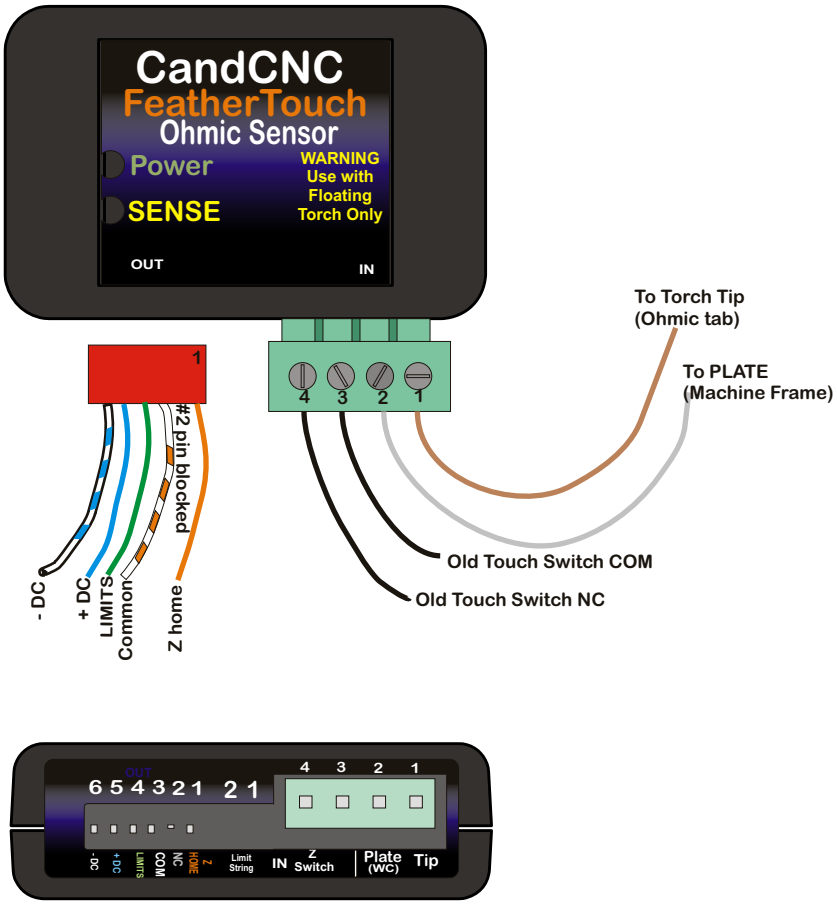
Do's and Don'ts

DO run the manual cut test and record the conditions you see including the LED indications.
DO make sure you understand that the setups for touch-off, pierce height and cut height are working correctly.
DO have the results of any testing ready for the support person.
DO take the time to understand the basic concepts of how an ARC VOLTAGE THC (like the DTHC) works.
DO realize there are external conditions and torch problems that can effect the DTHC.
DO follow all of the grounding rules for the plasma. HF start units need extra attention to proper grounding for reliable operation.
DO understand the relationships of Torch Volts, Torch AMPS, feed rate and air pressure/quality in plasma cutting.

DO NOT assume there is an electronics problem until you have eliminated ALL possible problems of incorrect settings and things like consumables and air.
DO NOT attempt to run plasma without the proper CommandCNC Profile (XML) loaded. Must be a CandCNC profile or a copy from our profiles.
DO NOT call for support with vague descriptions or having not tested to a point.
DO NOT make changes in the CommandCNC config to the base profile. Use a Clone copy and then only make one change at a time. The pin settings (mappings) are complex for our

FT-01 FeatherTouch Ohmic Sensor for use with CandCNC BladerRunner Dragon-Cut , BladeRunner Ether-Cut, MP3000-DTHCII, MP3100-DTHCIV MP3500 and Plazpak.

REV 5 Manual
1/15/14



This manual contains new connection suggestions for the FT-01 based on previous manuals. If you already have an FT-01 installed and working you should review this section and determine if you should make changes to the installation

IMPORTANT INFORMATION:

It has been brought to our attention that with the old touch-off switch wired to the table I/O limits input as per the previous manuals, the switch does NOT stop motion if the Z is a **homing move** and the FT-01 fails to sense. In this manual we have changed where the Z safety switch is connected so it now functions as an E-stop rather than a limit. The Green wire is now installed in the 5 wide connector

Connecting up the old TOUCH-OFF switch (Z Safety Switch)

1. You should check your Z safety switch and make sure it can be wired as Normally Closed. That means there is conduction between the COM and the NC terminal when the switch is not activated and it OPENS (no conduction) when the switch is activated (Tripped). Use an ohmeter or continuity tester to check the switch. THE E-STOP INPUT SIGNAL IN CommandCNC CANNOT BE CHANGED to make it work with a Normally open. There are other switches in the EPO circuit (in series).

2. When you strip jacket off the UTP cable to expose the individual wires take off enough so you can make the green LIMIT wire about 3 to 4 inches longer than the others.

3. Strip back about 1/4” of insulation off the green wire and apply a .250 Crimp-on terminal to the exposed wires.

4. Remove the factory supplied jumper wire across the EPO (E-Stop) tabs on the Table I/O card.

5. Temporally plug the LIMITS wire to the EPO terminal as shown in the illustration. Measure so the remaining wires in the UTP will reach the header fo the 5 pin connector and cut them off.

6. Unplug the Limits wire and strip each wire in the UTP cable so about 3/16” is exposed and insert them in the screw terminal openings in the 5 pin connector as shown. It is important that the insulation is off back far enough that the exposed wires are making good contact with the metal contacts in the screw terminal but that the bare wires do not stick out far enough that they can touch each other. Poor wiring on this plug is the leading cause of problems with the FT-01 not working.

7. Once you have the 5 wide plug wired insert it into the header on the board as shown and reconnect the Green Limits wire over to the OUTSIDE EPO TAB (closest to the edge with the other TABS) tab as shown.

To test the EPO part of the install power everything up and take CommandCNC out of RESET. IF you cannot get CommandCNC to come out of reset and you have an “External E-STOP Event “ error flash in the diagnostics screen than the Z safety is either not wired as Normally Closed or there is a wiring problem with the 5 wide connector.

Connecting up the Z Safety Switch (CONT.)

8. To test if the EPO is still working temporarily short across the two EPO pins and make sure you can come out of RESET . In systems with an ESP or ESPII power supply from CandCNC (BladeRunners, Plazpaks,) the Motor DC poser MUST be ON before you can come out of RESET.

9. You MUST have the jumper in place on the FT-01 module of LIMIT STRING or you must have a string on Normally Closed switches into these inputs.

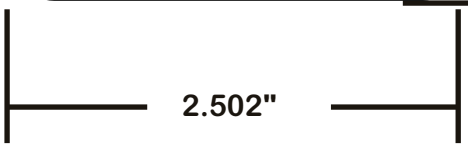
3. The previous “LIMIT” switches on your system will become E-STOP switches when wired in through the FT-01. If want to keep them as LIMITS you will need to wire the two ends of the string with one end tied to any of the Table I/O common TABS (inside row) and the other end into the old LIMITS input (center terminal) on the 5 pin connector. You will need to setup you LIMITS as before using the X++ (X limits) input.

4. It is highly recommended that you get your system moving and cutting BEFORE you add the FT-01 into the mix and start changing the E-Stop connections.

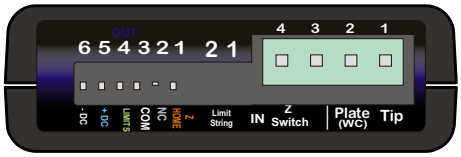
FeatherTouch™ Ohmic Sensor
for
CandCNC BladeRunner. Plazpak

For REV1 & REV2 Units

MANUAL RELEASE 5



If your lable shows TIP as 2 and Plate as 1, it is WRONG. Use this

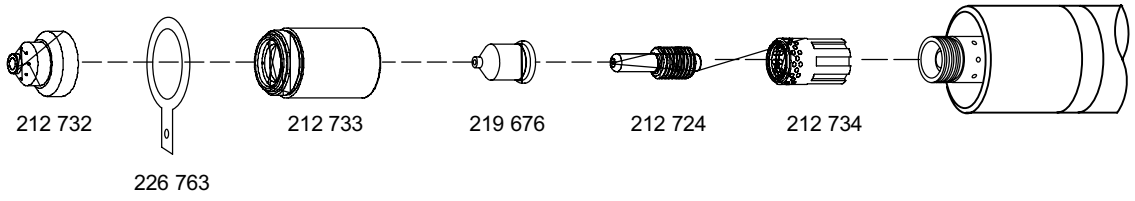


The CandCNC FeatherTouch Ohmic Sensor Module is designed to work with all existing CandCNC plasma controls including all MP1000-THC, MP3000-DTHC/DTHCII, BladeRunner Dragon-Cut, and Plazpak systems. It uses an active circuit to sense the tip of the torch touching the plate (material to be cut). It is totally isolated from the normal inputs to the Table I/O. The inputs on all CandCNC interface and BoB products have always been isolated from the PC ground to both prevent spikes and surges from harming sensitive port inputs on the PC and to reject noise that might come through sharing a dirty ground. The Table I/O inputs all use the same input common and it is designed to “float” and not be attached to a circuit that has a ground connection to the table (i.e. switches and devices not connected to the table electrically). If you allow one side of the Table I/O circuit to be connected to the table side ground then the isolation is partially or completely defeated. Noise from plasma cutting is exponentially higher than from routing or milling setups and needs to be considered in any input or sensor feed to the controls.

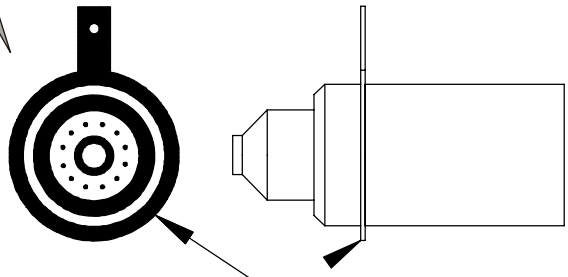
The Ohmic Touch circuit is operated from its own stand-alone power source (9 to 14VDC wall plug) and its output is opto isolated so it can maintain the integrity of the noise canceling and isolated inputs of the Table I/O. It’s sensor input is surge and voltage protected so normal plasma start and run voltages do not effect it. It offers a fairly low sense impedance and filtering to prevent false triggering.

Here is a list of rules to follow to get good results:

1. Always use the Ohmic Touch with a backup sensing system that will stop down motion if the sensor fails to work. This can happen on dirty, oily, rusty or painted metal. The Floating Torch Holder (mechanical Touch-off) acts as both a mechanical shock absorber and a limit switch to protect the torch from damage. DO NOT RUN WITHOUT A BACKUP MOTION LIMIT.
2. Keep the tip of the nozzle on your torch clean and free from trash and slag. If you start to have contact problems on material that is rusty or dirty, keep a spray bottle of water handy and wet down the surface for better conduction.
3. Be careful to observe polarity when hooking up the Inputs to the Torch Tip and Plate (ground) terminals. A reversed connection can result in a false signal and possible damage if left connected wrong.
4. The DC power terminals are marked for polarity. Do NOT reverse connect a source of DC or you will damage the module. If in doubt, meter it out! The + must go in the side marked + on the label.
5. Make sure to go though the hookup and calibration to test the proper functions of the Ohmic Touch and the backup limit to stop the motion.
6. **Do not attempt to use Ohmic Touch with unshielded consumables. It won’t work. Some consumables may need special shields (like the Hypertherm Fine Cut with a special Ohmic shield).**
7. The Ohmic Touch is designed for use with automated plasma cutting using controls from CandCNC. We cannot support other uses or interface to outer users systems..

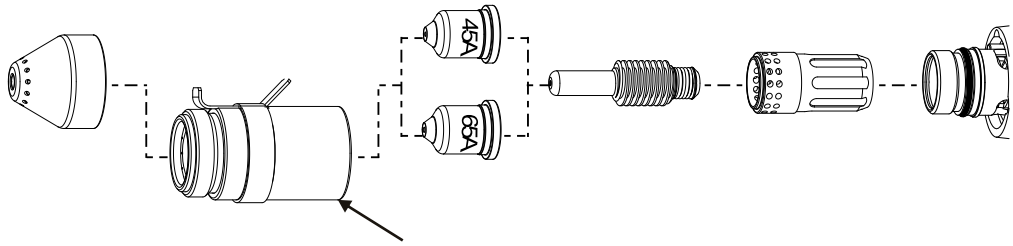


Note: Your ohmic sensor for your torch may be different than those shown here. The objective is to have a connection to the shield at the tip of the torch and it is not connected to the body of the torch or touching the frame of the torch holder.



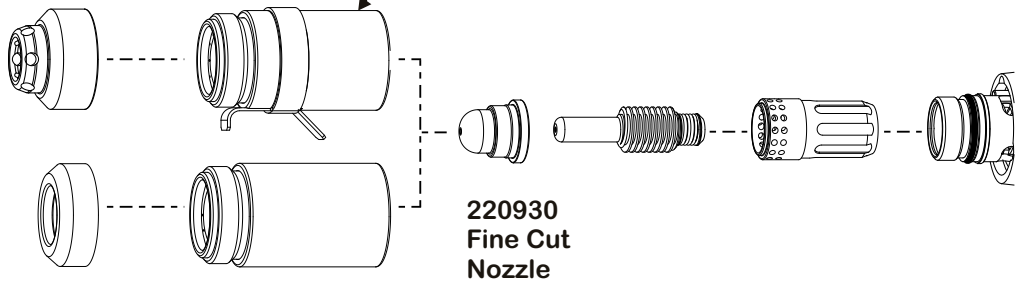
Shield sense tab provides feedback to a compatible torch height controller before starting the cutting process. Place the shield sense tab between the cup and shield.

Newer 45/65/85/105 consumables for Machine Torch



Ohmic Retaining Cap
Hypertherm Part # 220953

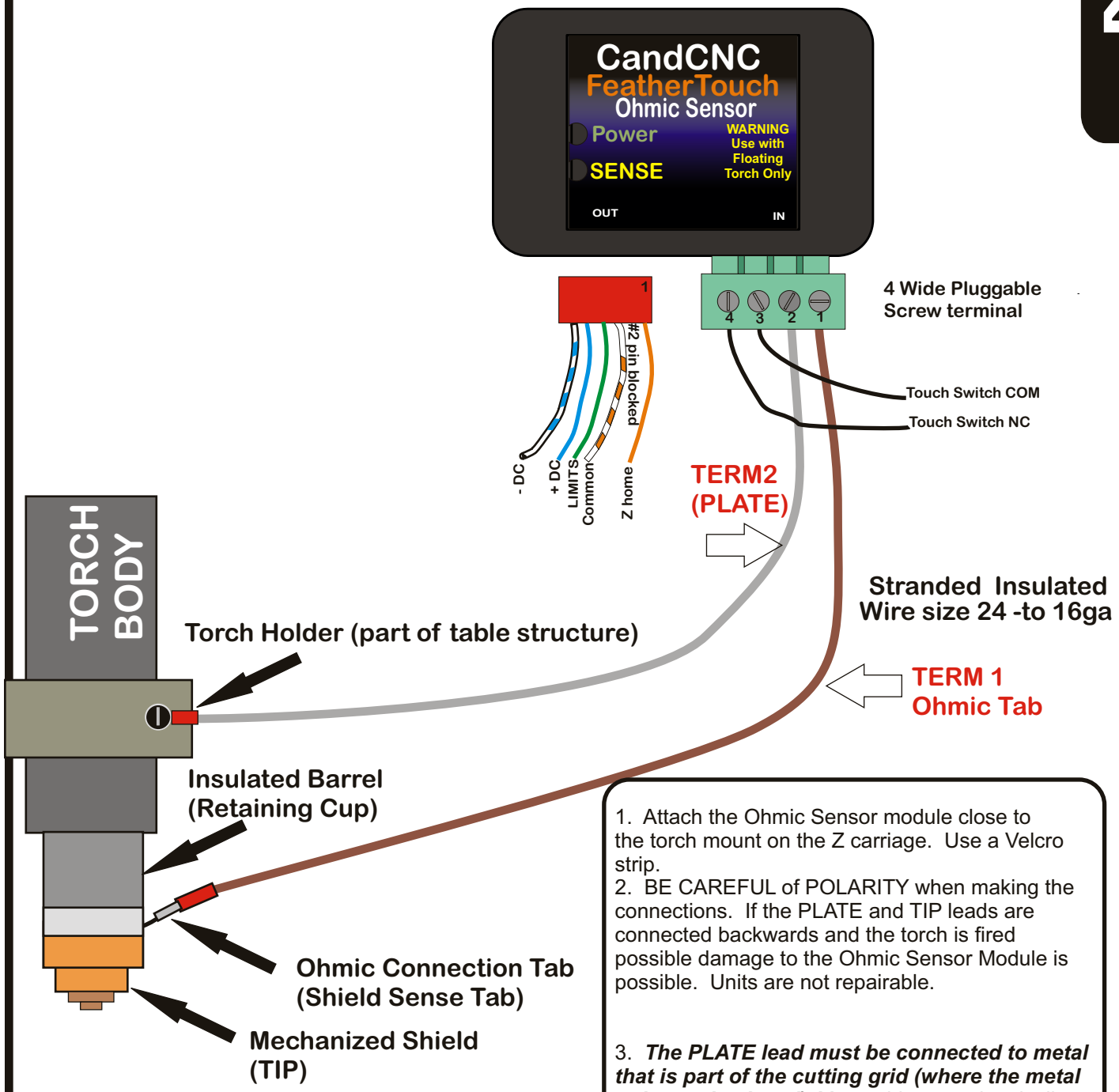
FineCut Consumables
Ohmic Shield
Hypertherm Part # 220948



220930
Fine Cut
Nozzle

Normal Fine Cut uses unshielded ring. Will not work with Ohmic Touch

OPTION: FT-01
FeatherTouch Ohmic Sensor

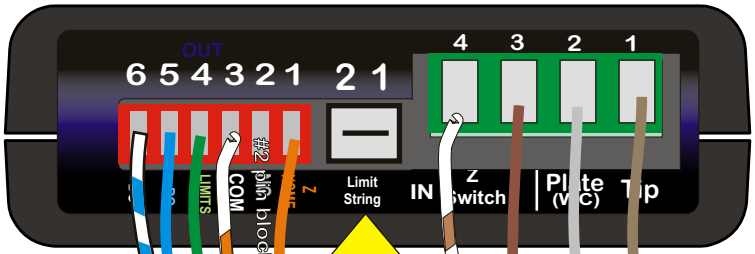


Requires special Ohmic Consumable
See your torch vendor for ordering.

1. Attach the Ohmic Sensor module close to the torch mount on the Z carriage. Use a Velcro strip.
2. BE CAREFUL of POLARITY when making the connections. If the PLATE and TIP leads are connected backwards and the torch is fired possible damage to the Ohmic Sensor Module is possible. Units are not repairable.

3. ***The PLATE lead must be connected to metal that is part of the cutting grid (where the metal to be cut is placed)*** Most tables are electrically connected between the gantry and table parts and the cutting grid. If your table has a separate table for cutting make sure it is attached to the gantry superstructure and torch holder via a strap or you will have to extend the PLATE wire (Term 2) down to the actual plate you are cutting.

OPTION: FT-01
FeatherTouch Ohmic Sensor



Jumper MUST be in place on installs that do not have a string of LIMITS already setup. See the instructions for more details.

These Wires are not part of the cable bundle and are not supplied.

Shipped Jumpered
See text to use as part of string of limits

NO
COM
NC

UTP CABLE

Existing Touch-off Switch on Floating Torch Holder wire as NORMALLY CLOSED

Table I/O REV 6 - 8 versions
Earlier versions will work as well

+ DC

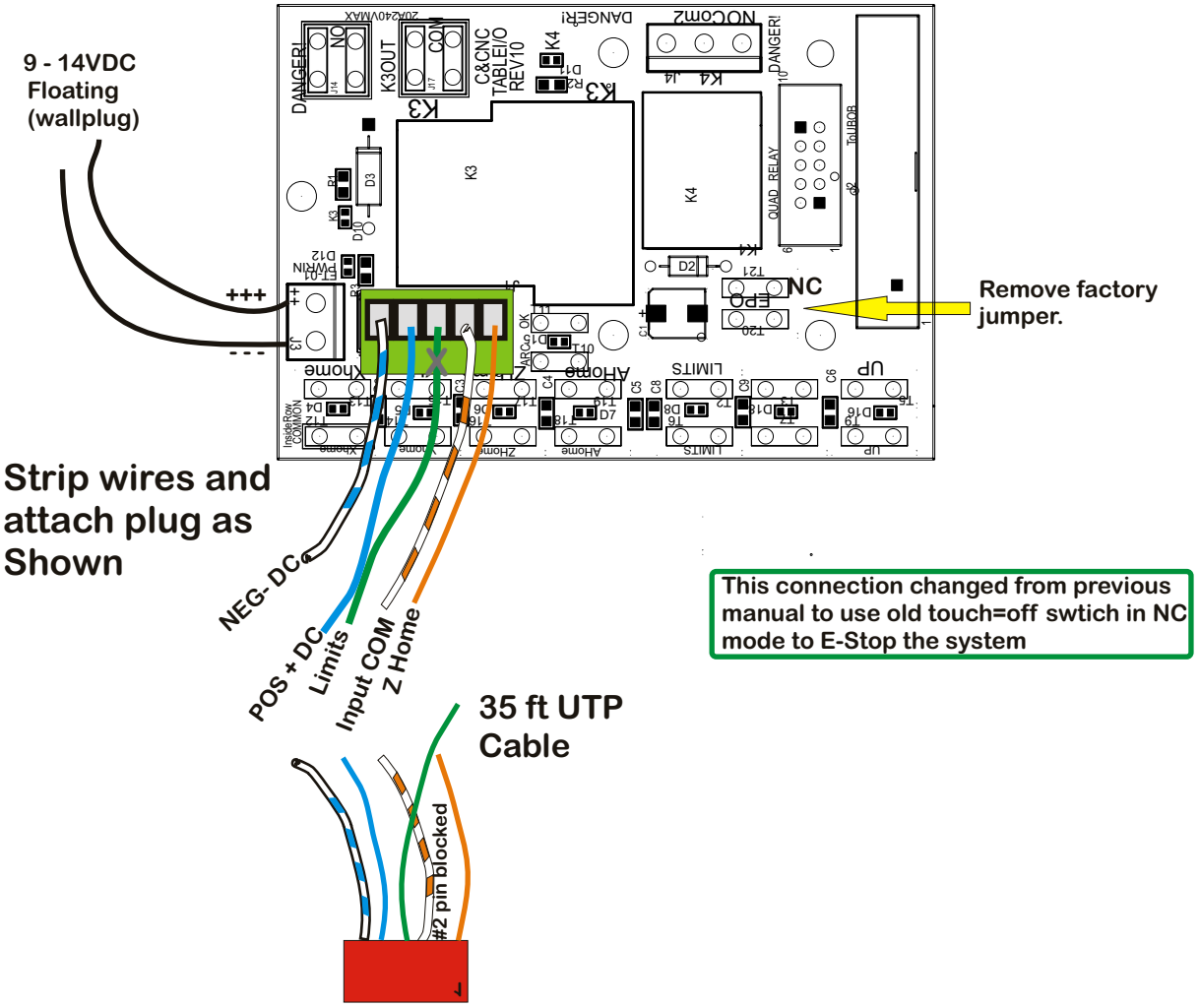
- DC

Table I/O Card Located inside BladeRunner and newer Plazpaks

NOTE: The indicators on the Table I/O card will light when the signal is active (ON) between the input terminal and the common terminal across from it (inner row). It will give you visual indication that the signal is getting to the inputs and that the Table I/O is connected to the UBOB card in the controller.

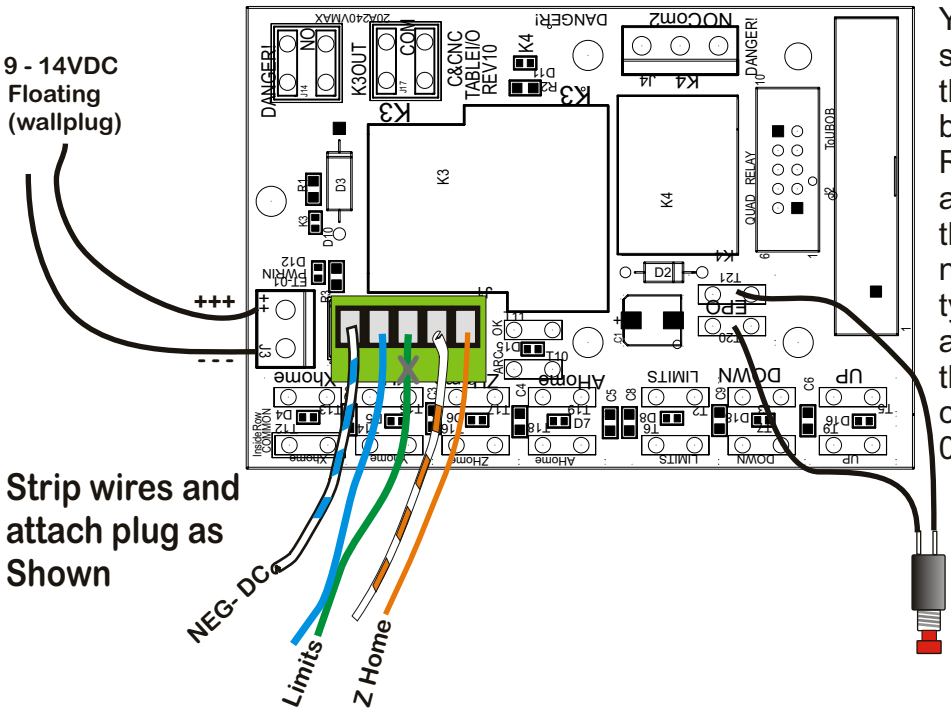
TABLE I/O REV 12 w/ Ohmic Sensor Interface Card

REV 12 Table I/O was released to production 2/15/14



Shown Above: New Table I/O REV 10 table I/O card. This card is a redesign. The PORT 2 inputs are removed (not used). Jack for Ohmic Sensor is added along with the floating power input for the Ohmic Sensor Module. It uses the same cable as the previous REV 8 model with the Ohmic Sensor Interface Card. The normal Port 1 inputs and outputs are the same as the previous Table I/O cards although the exact position on the cards has changed. The card is smaller and easier to get into tight spaces. The 35 ft UTP cable is shipped with the 6 wide IDC connector attached and the other end of the cable unterminated. The 5 pin Mini Pluggable Termial is shipped un-attached to the cable so the cable is easier to thread down exiting cable routes. Cut the cable to length then strip the wires on the unterminated end and carefully wire the wire colors as shown. It is important that you get the wires oriented as shown. The 5 pin and jack are keyed so it only fits one way. Plug it in the jack first to determine the orientation. NOTE: In the cable there are 8 wires but only 5 are used. The White/Green wire and the Brown and White/Brown are NOT used. Cut them off. If you have enough length you can use them for the wires to connect your Tip and Material (plate) wires and/or your old touch-off switch.

BY-PASS OPTION SWITCH FOR E-STOP
(not included in FT-01 kit)



You can setup a bypass switch and locate it close to the operator so you can bring CommandCNC out of RESET and manually jog the axis off the switch to clear the condition. The switch needs to be a MOMENTARY type so it won't be left on and defeat the E-STOP from the Z Safety and any LIMITS connected through the FT-01.

You will need to make up a 2 conductor wire and switch to fit the two .250 EPO tabs on the Table I/O The type of switch used for the Bypass is not critical as long as it is Normally Open and closed when you push it. When you release it should return to Normally Open. Most pushbutton switches are configured this way. Because the circuit is NC most of the time noise on the Bypass Switch or wires has no effect .

NOTE: This can be used on any Table I/O version shown if the EPO is used for other E-STOP inputs.

Dual switch probing (touch off).

Before this release of CommandCNC; for those who had a FeatherTouch Ohmic sense setup, if the Ohmic touch failed to detect the metal because it was rusty or for any other reason, the floating head switch would be tripped and would put your machine into E-Stop. This caused many sore heads and lost hair.

Fortunately, there is now a better solution! With a simple change to your wiring, and a change in your config, the touchoff can now use the floating head switch for the touch-off if the ohmic sense misses - and it will automatically apply the switch offset if the floating head switch is the one that tripped.

Note: if you do not wish to make the changes listed below, your system will continue to operate as before.

In order to use this feature you will need to change how the switches are wired into the Table I/O card. The basic setup is that the floating head switch wire (green wire) should now be connected to the Z Home input on the Table I/O, and the ohmic sense wire (orange wire) should now be connected to an unused input - preferably the AUX1 input. Image 1 shows the green connector that plugs into the Table I/O card. The green wire from the floating head switch now goes into the position at the edge next to the white/orange wire, and the orange wire from the ohmic sense is pulled out of the connector and has a .25” shove-on connector crimped onto the end. This configuration will be the same for all versions of the Table I/O cards which have the connector for this plug.

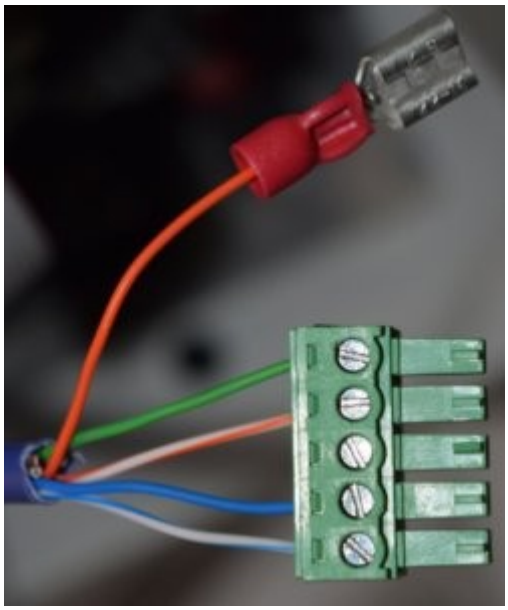


Image 1: FT-01 plug at Table I/O card

Image 2 shows a Table I/O card rev 12.2 with the connectors plugged in. Another change is that now that the floating head switch is not part of the E-STOP chain, you must put a jumper across the EPO tabs on the Table I/O card as shown in the image below with the blue wire. ***If you do not jumper across these tabs, your machine will not be able to come out of E-STOP!***

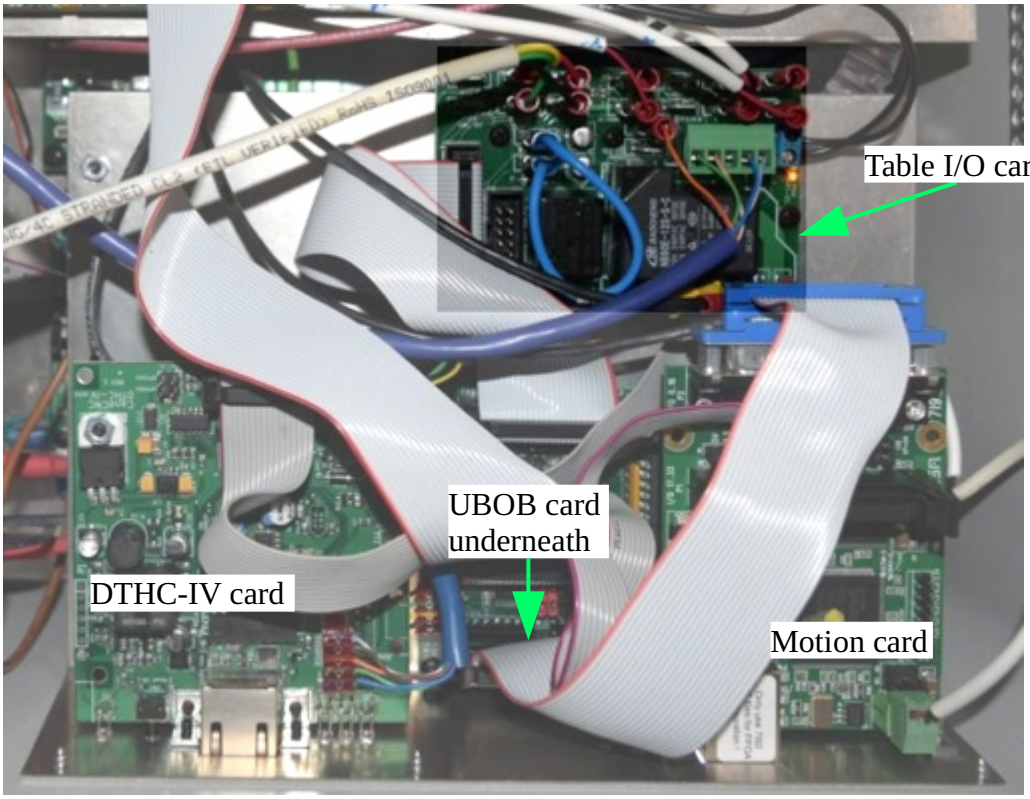


Image2: Plate assembly with UBOB, DTHC-IV, Table I/O, and Motion cards

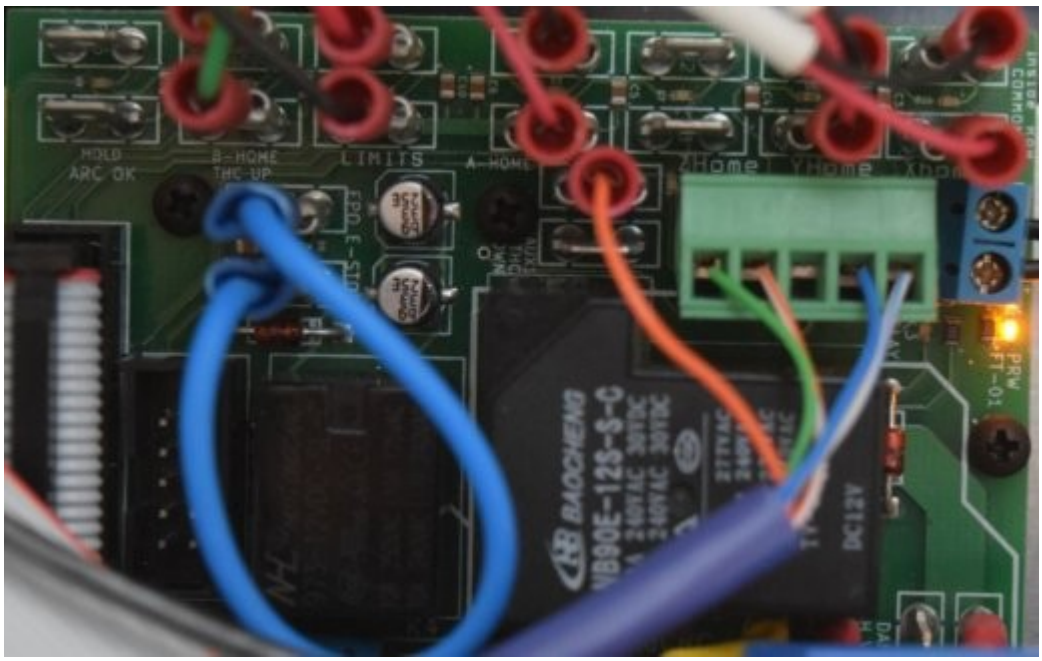


Image 3: Rev 12.2 Table I/O card showing new FT-01 connections

After making the connections as shown above, you will need to change some settings in the Configurator:

1. "Z Home input" should now be set to "Z Home switch"
2. "AUX1 input" should now be set to "Probe input"
3. The "Z Home input" Invert checkbox should be **un-checked**. This is because the floating

head switch was (or should have been!) wired as normally closed (NC), so when used as a home switch again the signal is inverted. If you wish to re-wire the switch as normally open (NO), then you would check the Invert checkbox.

4. You will need to determine the switch offset for the floating head switch and enter it in the "Home Switch Offset" entry on the "Z Axis" tab in the Axes page of the Configurator.

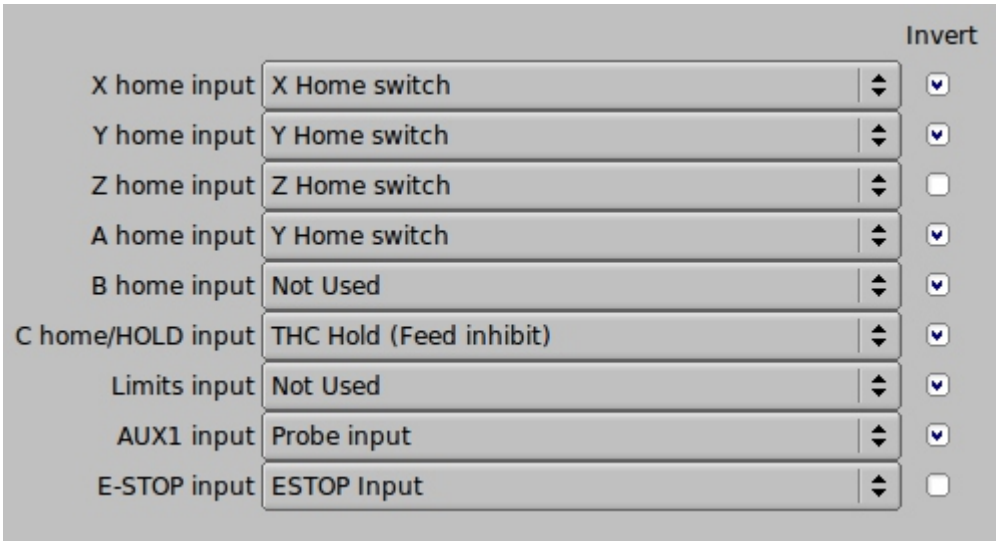
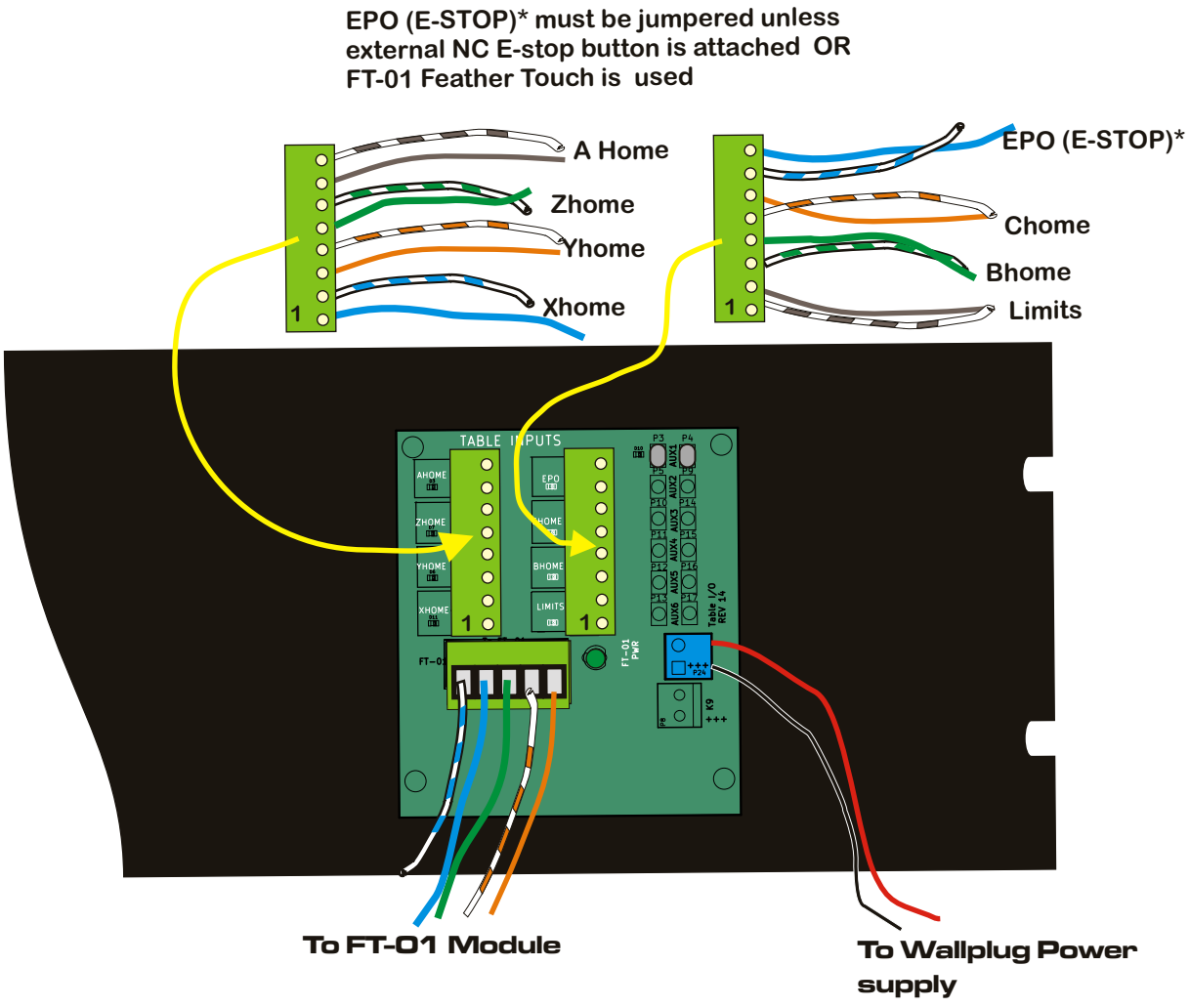


Image 4: Sample Configurator inputs

FeatherTouch Ohmic Sensor (New CoreCNC Table I/o

SETUP AND TESTING:

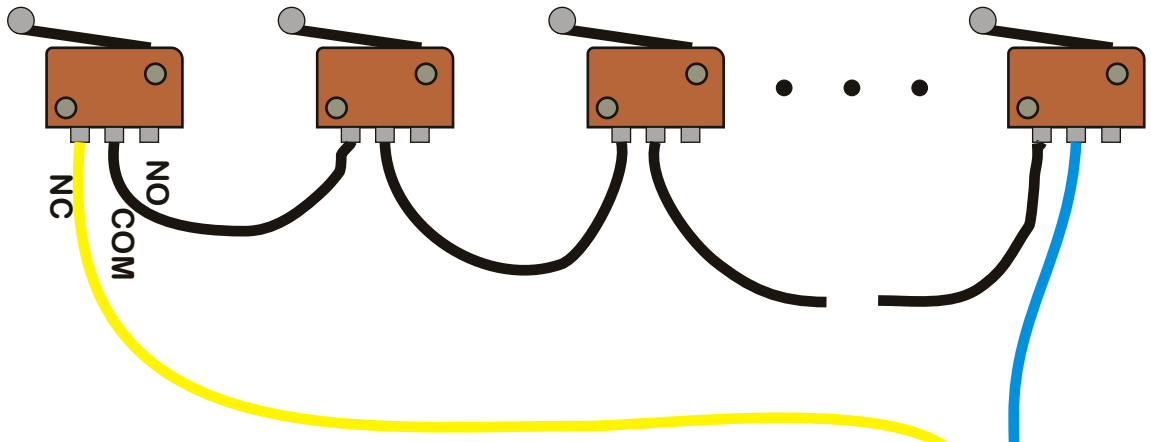
1. Make the connections between the Ohmic Sensor and the Table I/O card as shown on page # . . Provide DC power to the module. Modules ship with a DC wall plug power supply that should be used unless you have access to a source of FLOATING (ground neg side is not tied to any other circuit). See the section power hookups and options.
2. Make sure the wire jumper is in place at the location indicated. The jumper is applied at the factory and only should be removed if you are going to integrate the LIMIT for the touch off in with existing limits on your table (see section on hooking to existing limit string.
3. Use the section on hooking up the touch-off switch (not part of the Ohmic Touch kit) on your Floating Torch Holder.
4. Plug in the power to the Ohmic Sensor. Confirm that the power LED (Green) is on. If it is not, unplug it immediately and locate the cause of the lack of power to the module using a DVM.
5. Check the settings on your limits input. The limit input is port 8 pin 11 in most UBOB based systems. (new Ether-Cut system is an exception) At least one of the ++ or – signal (usually X--) needs to be Enabled (green check) if you are using it for just the touch off. It has to be set differently than if you are using other limits in a normally closed string.



FOR SETUPS THAT HAVE EXISTING SEPARATE E-STOP ON OTHER AXIS USING a NC STRING OF SWITCHES: (if you do not have this setup then disregard this page)

Note: Your switch pinout may be different

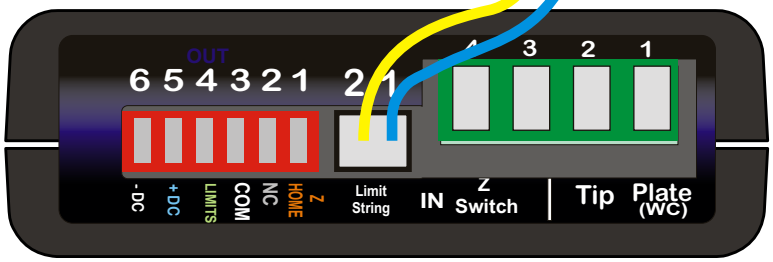
SEPARATE E-Stop SWITCHES (Normally Closed)



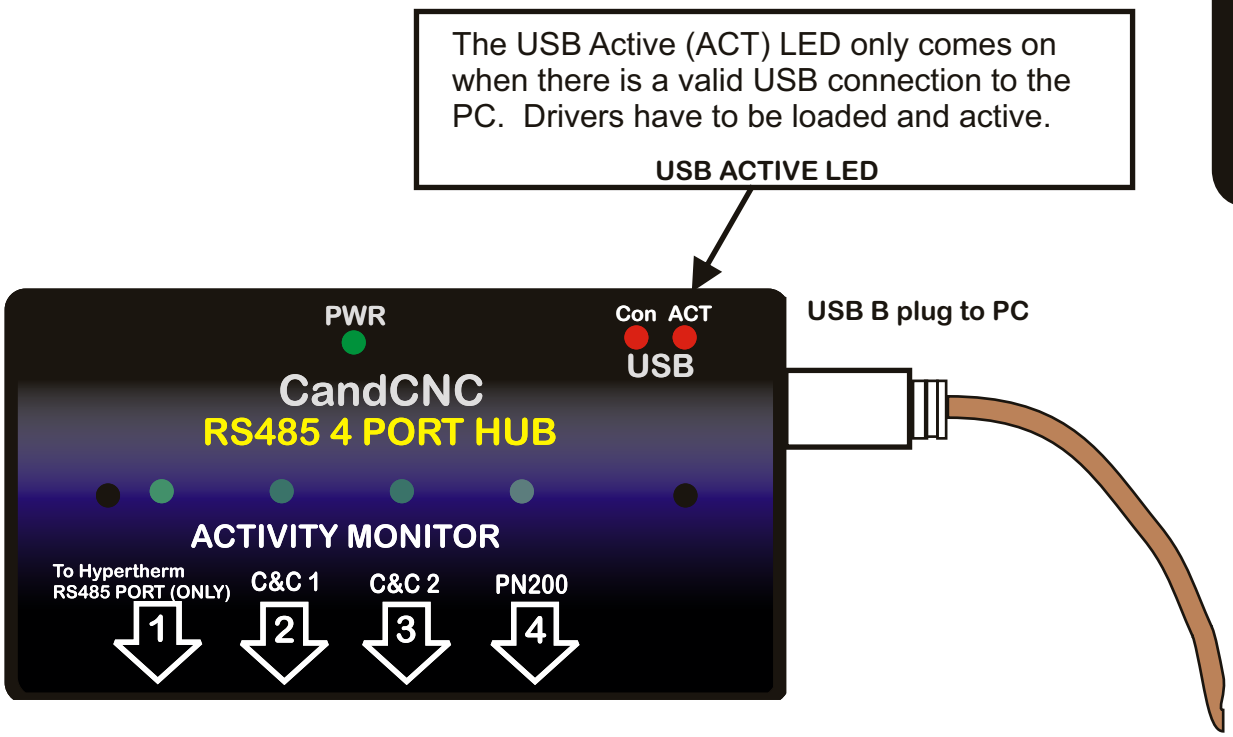
THIS SETUP PUTS THE LIMITS in the STRING INTO the E-STOP circuit and they operate as part of the ESTOP and no longer just limits. They will NOT be ignored during a Homing Move.

Beginning of EStop String

End of ESTOP String



1. Take Limit string loose from Table I/O card
2. Remove jumper on limit string
3. Put two ends of limit string into the ESTOP STRING terminals
4. Connect the LIMITS OUT wire (white in this manual) as shown to LIMITS input terminal.



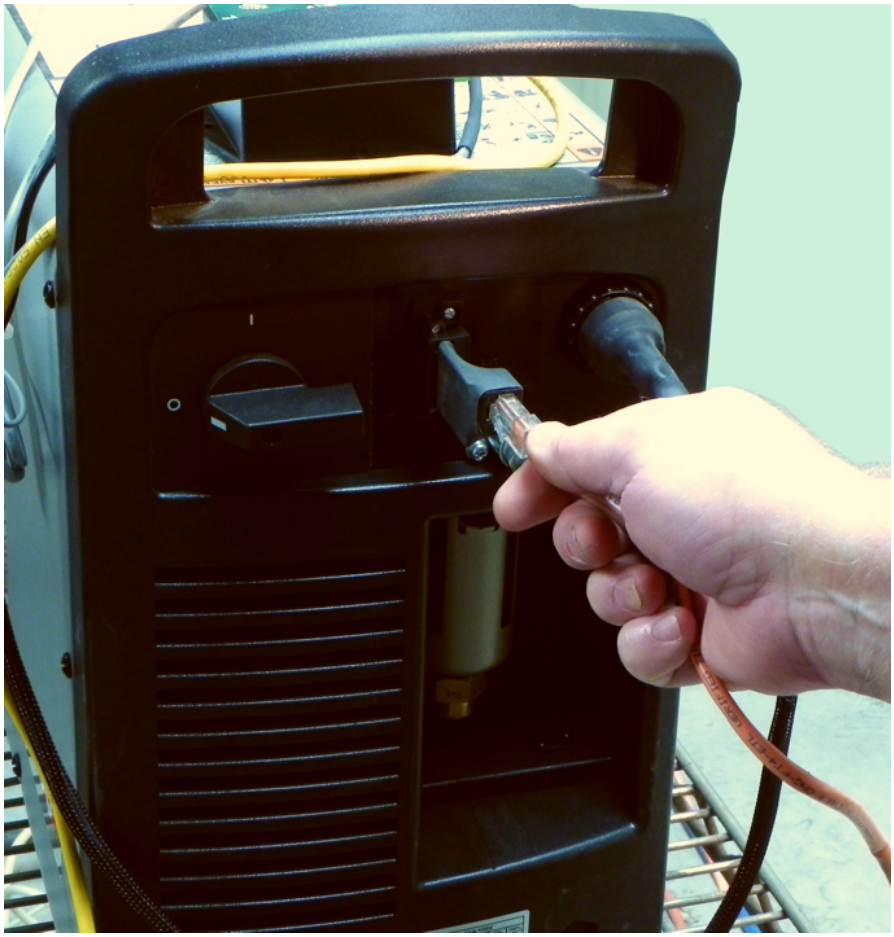
INSTALLING CandCNC RS485 Devices to the USB-RS485 4 PORT HUB

The USB-RS485 4 PORT HUB has an advanced processor than can communicate with several RS485 devices. RS485 is a robust and noise-immune communications standard used in industrial electronics for years. Because of its differential signal methods it is unaffected by external or ground based noise and reliable communications of several hundred feet are common. RS485 is a multi-drop topology meaning there can be multiple devices on the same pair of wires as long as all of the devices operate at the same speed (BAUD RATE) and have a unique address. Since USB is a common port on most PC's it is a logical choice for communications that do not depend on precise timing.

You will note that the USB-RS485 4 Port Hub has four in dependant channels ad each channel can talk to multiple devices, Because of different Baud Rates or special signals the 4 port hub has two special jacks:

1. **Hypertherm RS485 Port**. This channel runs at a much slower speed and can only talk to a Hypertherm Plasma Cutter equipped with an RS485 port (optional) and through our HyT-Connect RS485 interface. If you already have the HyT-Connect RS485 SIM Kit installed and have the older RS485 module you need to unplug the existing setup and plug the RJ45 (Cat5) cable FROM the port on the rear of the Hypertherm into the jack 1 marked "To Hypertherm Rs485 PORT (Only)

Plugging RS485 Hypertherm into RS485 PORT on back of Hypertherm 65/85/105



See HyT-Connect Manual for details on installing and using the RS485 remote control for the Hypertherm and the DCC (Dynamic Cut Control)

2. **PN200 Hand Control.** The PN200 hand control uses normal high speed baud rates but the special “dead man” E-Stop to the EPO of any CandCNC controller requires a special jack. The PN200 MUST be plugged into the PN200 jack to be able to use the E-STOP safety (recommended). The PN200 has to be the last device in a group of devices. The PN200 will worked plugged inot other jacks on the 4 PORT HUB (except JACK 1) but you cannot use the E-STOP option. You can also use the PN200 jack on the hub for other RS 485 devices if you do not have a PN200. While the PN200 would work with other devices on the channel that have loop through (pass through) jacks the E-STOP to EPO DOES NOT PASS THROUGH OTHER DEVICES

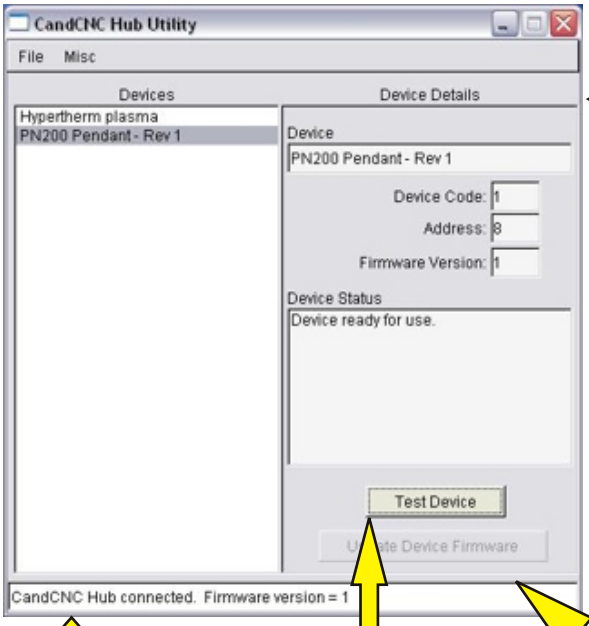
3. **Other CandCNC RS485 deivies**. At the time of this manual, the only released RS485 devices are the PN200 and the Hypertherm RS485. There will be other devices as new revisions of our other cards are released over the next year. *There will be termination settings that will matter when devices are cascaded (hooked together on the same channel). The END DEVICE in the string has to be terminated. The PN200 is an “End Device” (has to be the last in the string) as is the Hypertherm RS485. In both those cases the termination is already in place.. No action needs be taken if the PN200 is the only device (as recommended) on it’s channel and/or the Hypertherm cable is connected as the only device on it’s channel.*

MANAGING THE HUB

During the install the CandCNC Hub Utility was added and an ICON was placed on your desktop.

Click on th icon to open the Hub Utility. 

The following screens show the CandCNC Hub Utility displaying information about different devices connected to it. The Hub is “smart” and automatically detects any compatable device connected to it. Review the screens that follow and understand what each section o fthe screen is used for.



DEVICES:

Shows a list of connected devices. As a device is connected and sensed by the hub it will appear in the list. If you connect a device and it does not show up immediately,

DEVICE DETAILS

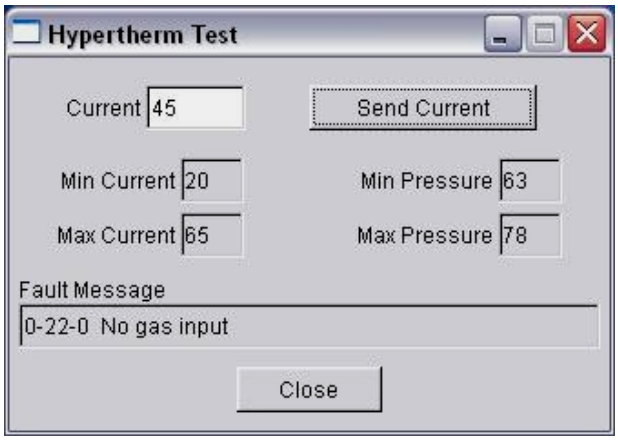
Provides the:
Device Name
Device Code: Each CLASS (type) of device has a different Device CODE Codes can be from 1 to 99.
Device Address: The unique address of a device within a class (values from 1 to 8).. Devices of the same type must have a different Device Address. Future cards will have address jumpers to allow multiple cards of the same type to work on the same hub.
Firmware Version: This displays the current

Hub Connected
(to USB on PC) Displays current firmware of the HUB. This MUST show connected status before any other of the functions will display

TEST DEVICE
Action button that opens a new dialog window. You must have a device highlighted in the Devices List to

Update Device Firmware.
This allows you to update each module with new firmware. New firmware will be designated by a REV number. Firmware updates will be available for down load in a special download section of the CandCNC website and on the Yahoo CandCNCSupport Forum

DEVICE TESTING: Hypertherm RS485 VIA HyT-Connect RS485 SIM



HYPERTHERM TEST

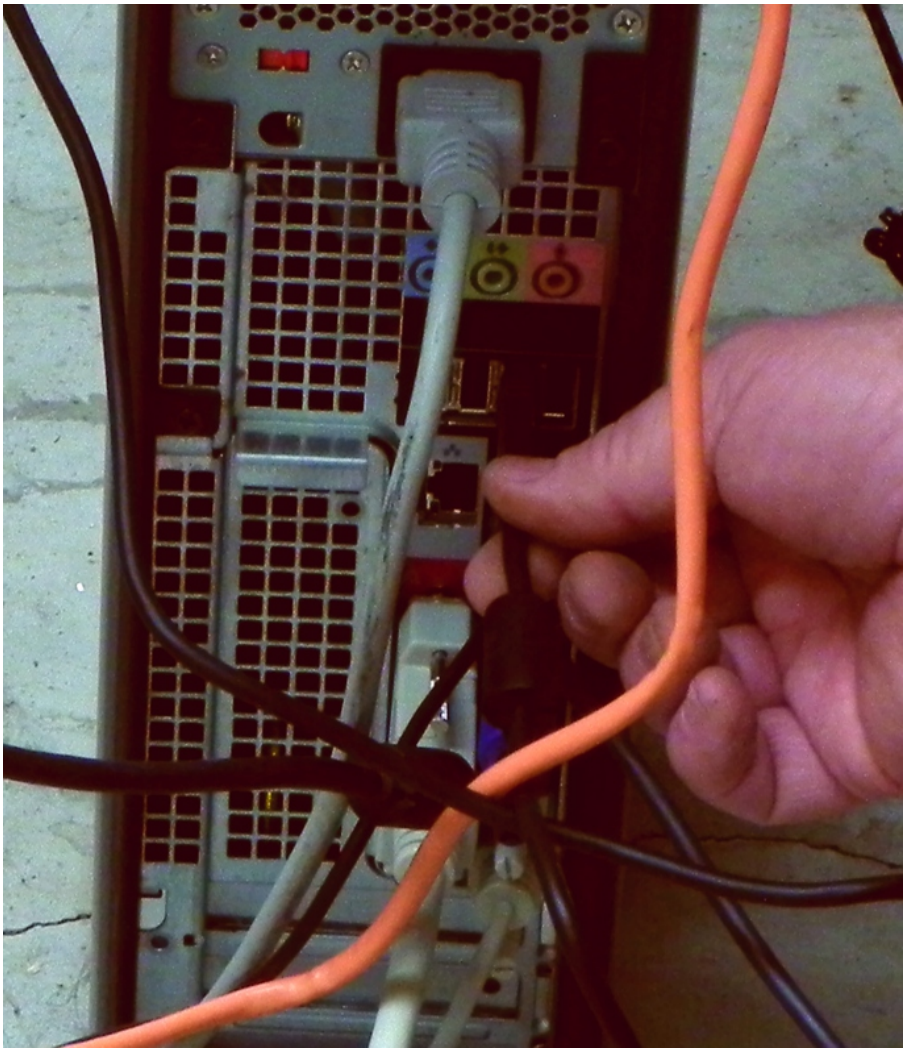
Shows connection and communication to/from the Hypertherm RS485 Port tot he USB-RS495 4 Port Hub. Any fault message will display (in this case air was not hooked up and it shows a "No Gas Input error)

To test you can input a Current in AMPS) and send it using the **Send Current** button. the LCD display on the Hypertherm should show the new value. Other values are tose read from the

9. With the module under power (green power LED will be on) locate the USB A to B cable provided with the kit. Connect the "B: (square) end of the cable to the plug in the end of the USN to RS485 module. It is keyed and will only go in one way.

NOTE: Photo is of older single port RS485 hub. See previous sections for connection points

10. You may want to secure the USB to RS485 module to the top or side of your PC. The best method is to is a strip of adhesive backed Velcro on each side so the module can be up out of the way but easily removed if needed

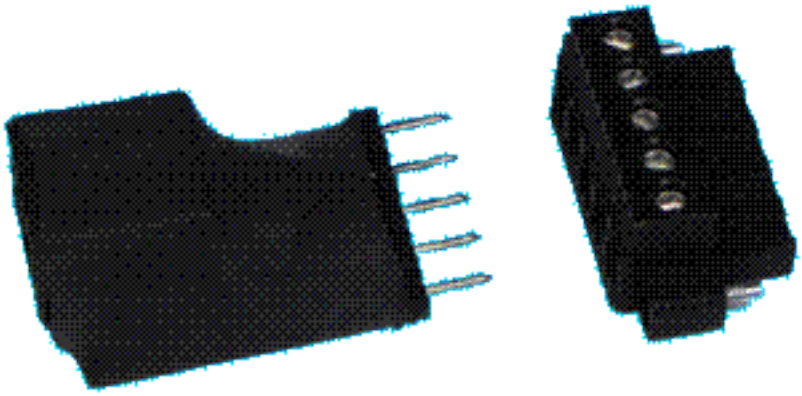


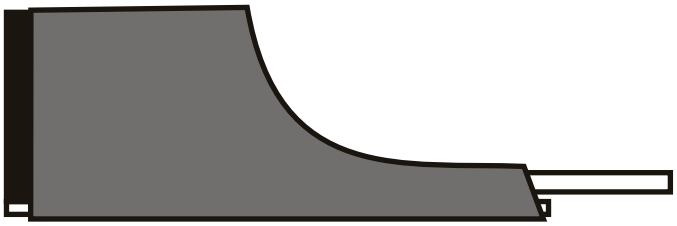
11. Find an open USB jack on your PC and plug the "A" end of the USB A to B cable into that jack.

When you plug the powered USB into a USB jack on the PC and if Windows is running it will pop up a "New Hardware Found" box, but it will install the device automatically (provided you have done the software install on pages ## thur ## in this manual).

At this point you may want to go back and verify that the settings in the COM ports on the PC.

Note: If you have the drivers loaded and the correct COM port is designated in the ccc_comm Plugin, then you will start to see activity on the ACT USB LED of the USB to RS485 module.



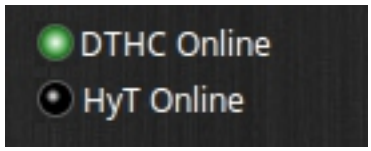


Locate the 65/85 Serial Plug Adapter in the kit. If you need to remove the adapter card from the 5 pin Eurostyle plug for any reason use the diagram to connect it correctly. Plugging it in to the Eurostyle plug upside down and connecting it to the USB to RS485 module could cause damage.

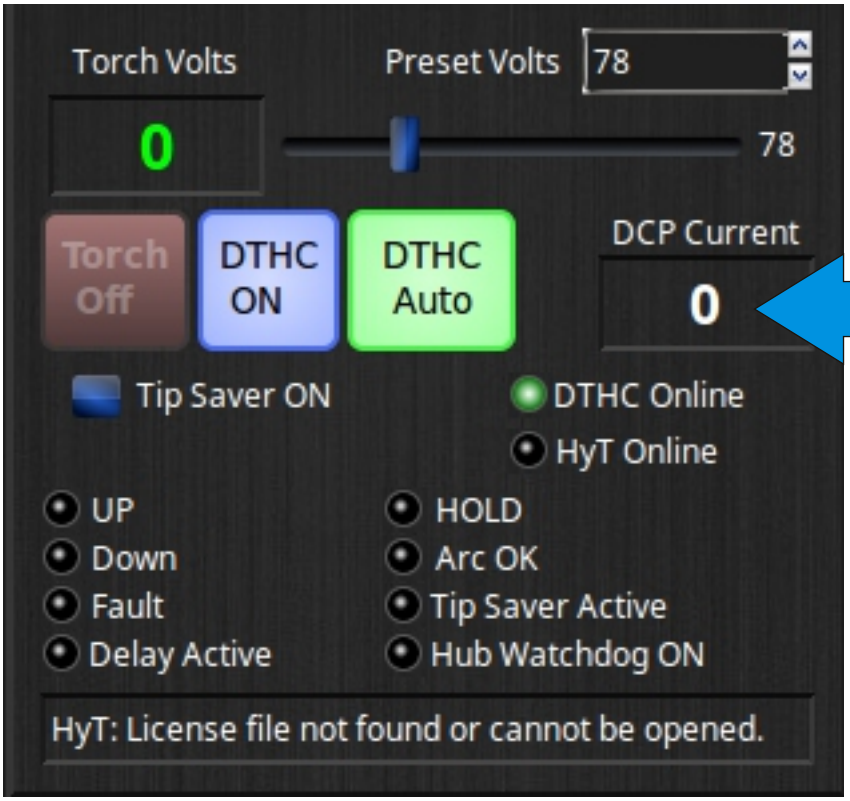
With the 65/85 serial adapter module as shown , locate the matching receptacle on the rear of the Hypertherm Plasma unit. The plug and receptacle are keyed so it only will go in one way.. Insert the plug into the receptacle until it seats against the flanges. With a small screwdriver tighten the screws on each end. DO NOT OVER TIGHTEN.

NOTE: If your plasma unit does not have the 5 pin vertical receptacle on the back then your unit does not have the optional RS485 Serial Interface Kit installed. If you have the kit (Hypertherm part # 223036) use the file at: https://www.hypertherm.com/Xnet/library/library.jsp?file=HYP109750

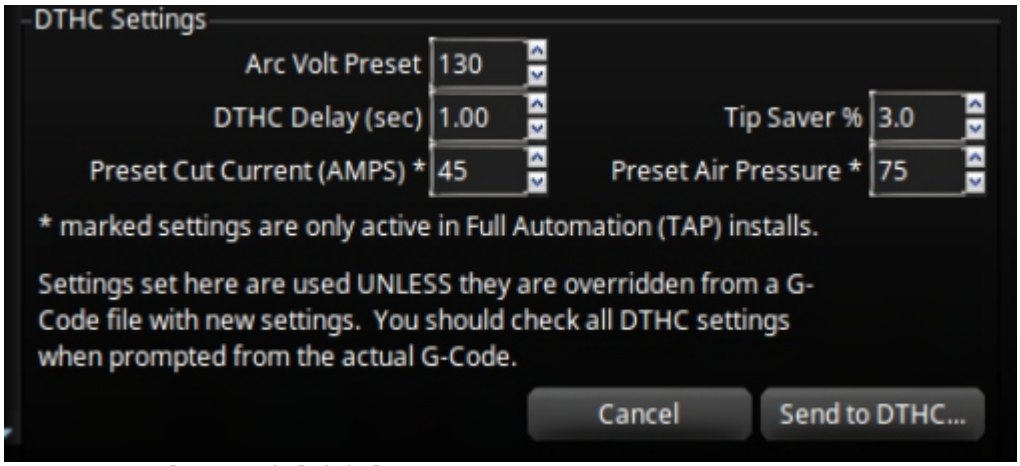
Once you have the 65/85 Serial Adapter Card installed and secured take the opposite end of the UTP cable from the USB to RS485 module and plug inot the open RJ45 jack on the end of the 65/85 Serial Adapter Card as shown in the photo.



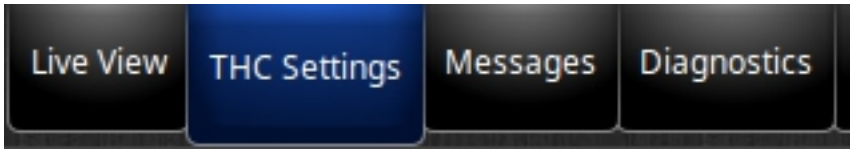
Note in the CommandCNC screen there are two status LEDs for confirming the communication to the DTHC and the HyT-Connect RS485 SIM. When you have a valid RS485 communication with the Hpertherm 65/85 the HyT ONLINE LED will turn GREEN and stay on. It means the both the physical connection and the software



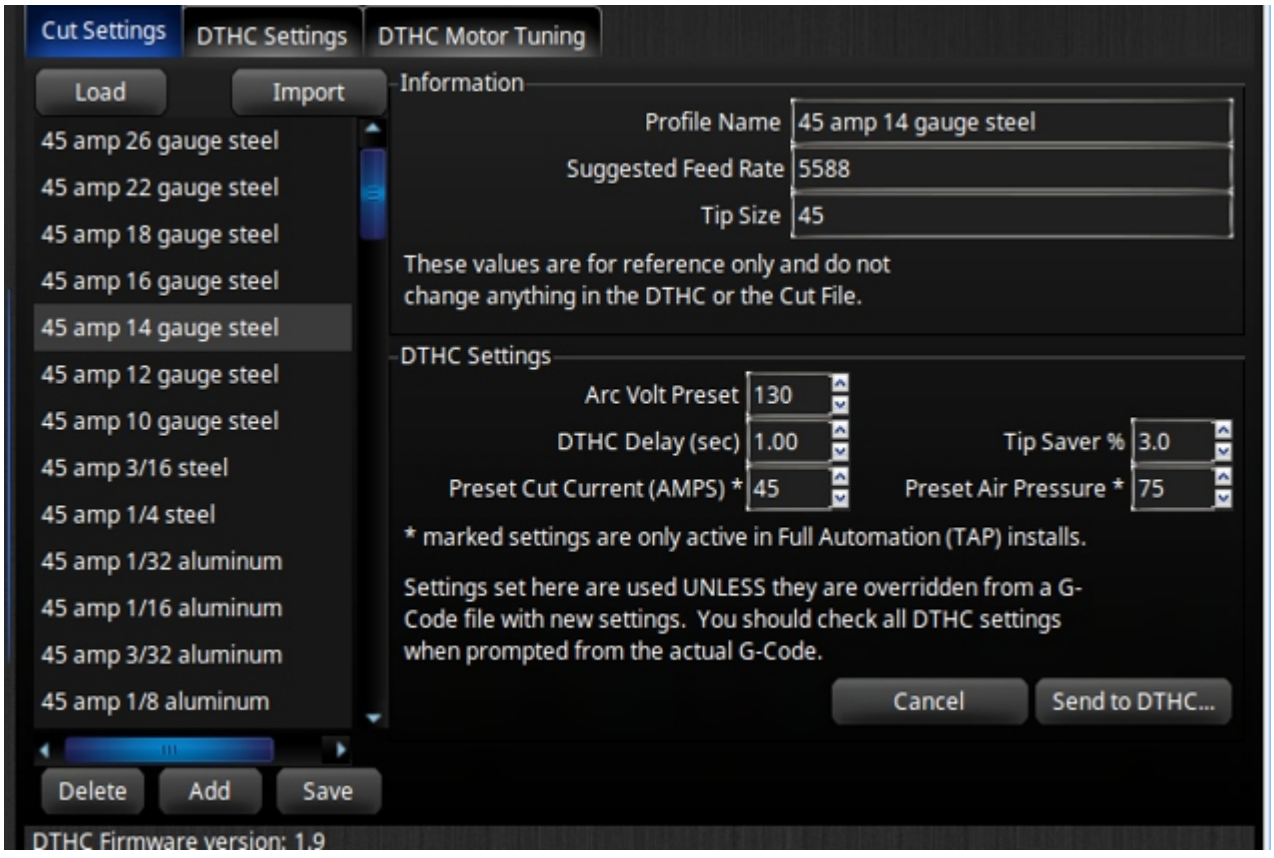
TORCH AMPS DISPLAY is ONLY ACTIVE IF you have the DCP-01 Digital Current Probe installed and working. It is not part of the HyT-Connect RS485 and is not required for it's operation but it does provide an actual confirmation of the cutting amps.

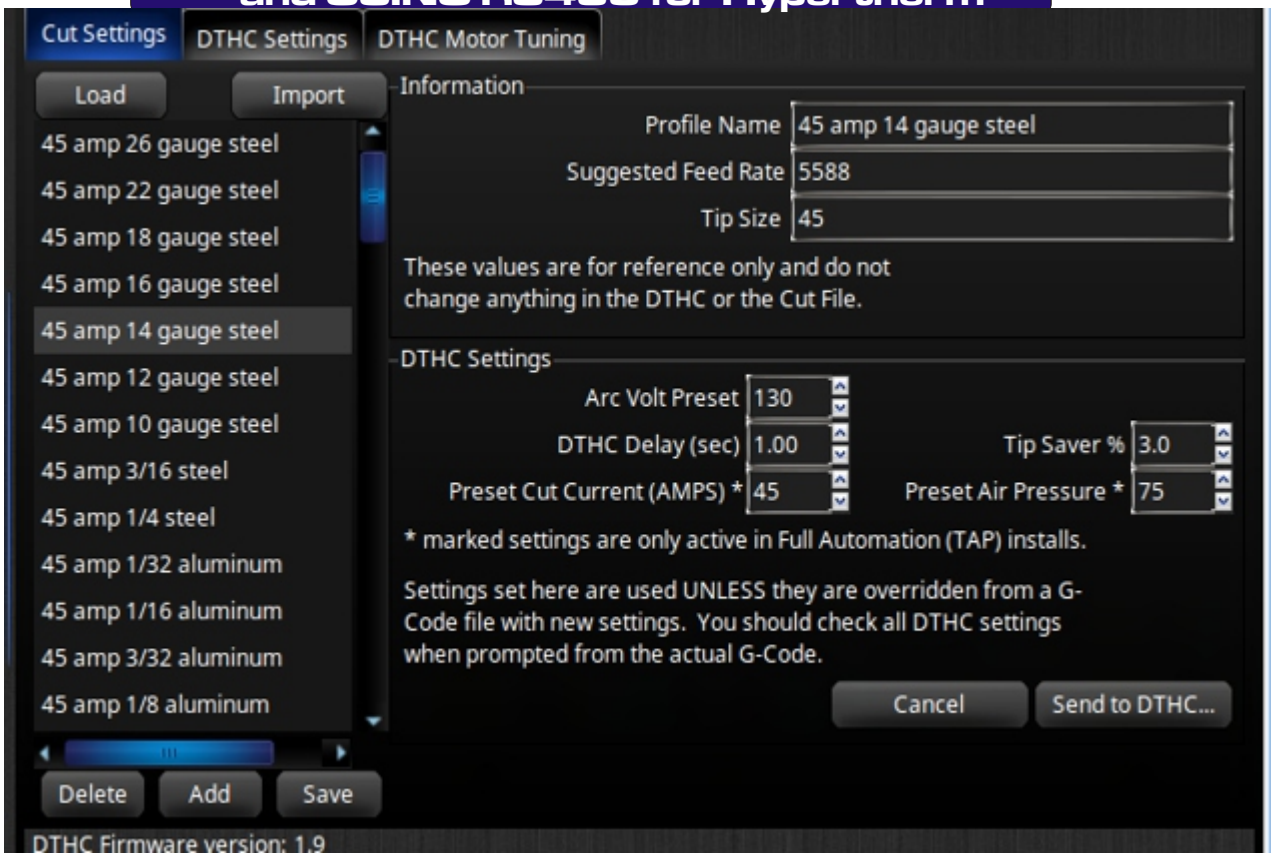


Shows the AMPS the Hypertherm is **set for** when the HyT-Connect RS485 is connected and working. You cannot change it in the DRO...you have to open the CUT PROFILE and make the changes in the CUT Current setting. **IT DOES NOT show** the actual current that is happening at the cut. It is the same as reading the dial on the plasma cutter.

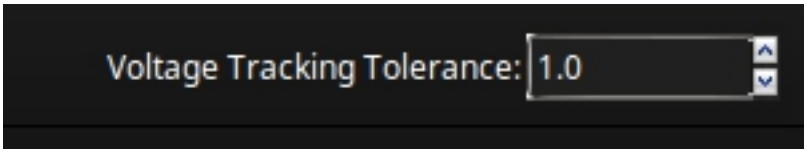


To open the CUT PROFILES dialog window shown above click on the SETTINGS(Cut Profile) button on the Program Run screen.





OTHER DTHCIV Settings:
The **Arc Volt Preset** sets the PRESET VOLTS (target volts for the DTHC to use to compare to the actual TORCH VOLTS from the cut and to use to adjust the ARC GAP (height) as it cuts.
The **DTHC Delay** sets the length of time in seconds the DTHCIV delays reacting to the change in voltage variations in volts after the torch fires. It delays the FAULT signals for 1 sec AFTER the duration of the primary delay. It should be longer for thicker material and should allow enough time for the torch to fire, and for the torch to start cutting at intital cut height and for a normal cut to start. Default is 1sec but raising that lengthens the delay .



Voltage Tracking Tolerance (in DTHC Settings tab) Sets the sensitivity of the ARC VOLT PRESET. It creates a Dead zone of volts where the control is in balance. The setting is in volt increments so 1.0 is one volt. That translates to the control in the setting shown below would be in balance (not sending UP or DOWN commands) as long as the TORCH VOLTS reading was between 141 and 139. Normal setting is 1.0 On some material that value can be set tighter or more loose to change the reaction of the DTHC. Each volt represents about .015 to .020 of arc gap so settings too high will result in poor DTHC response.

ARC OK CURRENT. The “trip point” for ARC OK signal to turn on (sends ARC OK to MACH). YOU MUST HAVE THE DCP-01 Digital Current Probe providing the actual Cut Current before this value has any function. The Hypertherm 65 and 85 have the ARC OK (TRANSFER) signal available and that is normally used as the ARC OK signal. The two signals are in parallel so EITHER will trip the ARC OK.

AIR PRESSURE. This is an HYT-Connect RS485 SIM feature ONLY. IT shows the pressure that has been set at the Hypertherm in PSI. There is no feedback value (actual air pressure indicator) so the value is the same as reading the value on the LCD.

NOTE: This section may be outdated if you have upgraded to the newer DCC (Dynamic Cut Control) version.

Using the Hypertherm 65/85/105 Air Plasma Units with the CandCNC DTHCII & HyT-Connect Dynamic Cut Control (DCC) Upgrade to Provide Advance Cut Features

CandCNC Offers:

- High speed microprcessor based DTHCII 4th generation control.
- Wide range of Motion Controllers with integrated Torch Height Control
- Single cable connections with no internal modifications to the plasma.
- Digital noise immune plasma side pickup for accurate readings from 50:1 voltage divider. Total isolation of all signals.
- Only vendor that offers low cost Digital Current Probe to read and display the actual current at the cut in AMPS.
- Remote communication and control capability using industry leading RS485 to USB.
- All interface is via custom Operator Screens.
- Unlimited Stored Settings uses Hypertherm cut charts. Settings for 45, 65 85 and 105.
- Dynamic Control of Plasma Cut Current and Air Pressure.
- Readout on the screen of any plasma faults.
- Advanced THC features including Anti-dive (tip saver) and options to Stop on Fault and Retract Torch on Fault.
- No-pause THC enable/disable during a cut.
- DTHC settings from G-CODE job file using SheetCAM.
- Only low cost system to offer industrial level features.
- First vendor to offer Dynamic Cut Control including Feedrate reduction, Cut Current reduction, THC ON/OFF, in real time, in a low cost system.

Hypertherm offers:

- Industry leading air plasma vendor with advanced torch technology.
- Low cost air plasma with CNC connection options and internal voltage divider.
- Only vendor to provide advanced communications and control via industry standard RS485 in smaller plasma cutters (65 to 105 size).
- Industry leading support. Help when you need it most.
- Strong industry reputation for reliability and quality.
- Built in the USA.

OPTION NUMBER	DESCRIPTION
1	Any plasma cutter.
2	DTHCIVDigital Torch Height Control
3	DTHCIV Digital Torch Height Control with REV 1.9 or higher firmware
4	Hypertherm model 45/65/85/105 plasma cutter w/CPC port
5	Hypertherm model 65/85/105 plasma cutter w/CPC and optional RS485 port
6	CommandCNC for LINUX
7	SheetCAM update/install to SheetCAM TNG Development version 6.0.14
8	Hub Admin Application
9	CandCNC Advanced Connection Kit for Hyperthem 65/85/105
10	SheetCAM for LINUX

WHAT OPTIONS YOU NEED FOR ADVANCED FEATURES

FEATURE:	DESCRIPTION	REQUIRES::
STORED SETTINGS (Basic)	Library of stored settings for the DTHCIV.	**OPTION1 OPTION 2:**
Real Time setting of Preset Volts, Preset Amps and THC Delay	Allows setting of certain stored settings from the main operator screen in CCNC instead of Popup	**OPTION1 OPTION 2 OPTION 7:**
Cut Current settings to Hypertherm from Screen	Allows setting of Cut Current (Torch AMPS) from the screen (while cutting or from a Cut Profile.)	**OPTION 2 OPTION 5 OPTION 8**
Preset Volts setting from G-CODE	Allows CAM operator to define certain plasma presets to use at run-time. Preset Volts overrides current Cut Profile settings for	**OPTION 2 OPTION 6 OPTION 7**
Preset Volts AND Preset AMPS setting from G-CODE	Allows CAM operator to define certain plasma presets to use at run-time. Presets override current Cut Profile settings for DTHCII	**OPTION 4 THRU OPTION 9**
No-Pause DTHC ON/OFF	Allows CAM operator to define THC ON and THC OFF commands While cutting with no motion pause. Manual insertion or via G-Code	**OPTION 3 THRU OPTION 8**
FULL DYNAMIC CUT CONTROL (DCC)	Allows CAM operator to define ACTION POINTS in the contour to automatically insert Feedrate changes, Cut Current Reduction, DTHC ON/.OFF.	**OPTION 3 THRU OPTION 10**

HYPERTHERM PART # CROSS REFERENCE

AT THIS TIME: Hypertherm is the ONLY air plasma manufacturer that has seen the opportunity to provide advanced cutting features and remote commination to the smaller 65 to 100 A air plasma market. The HyT-Connect technology from CandCNC was developed with the assistance of Hypertherm Enginnerig and is exclusive to their machines.

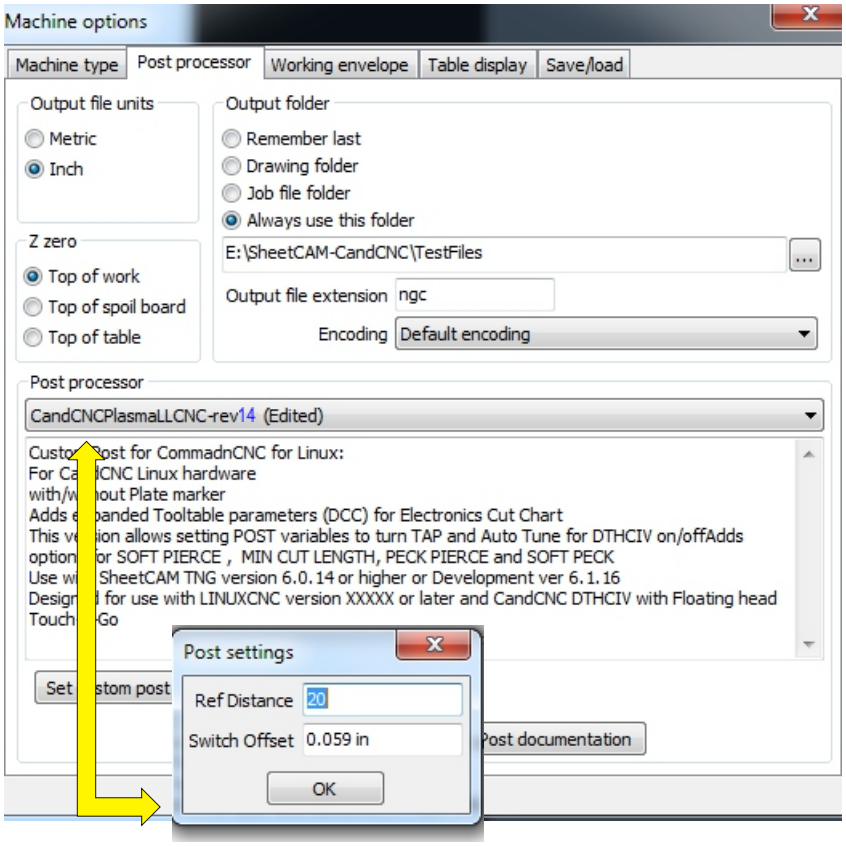
Hypertherm MODEL #	With CPC (CNC) port only	CandCNC Part for interface	With CPC + RS485 serial	CandCNC Part for interface	NOTES
45 No Torch	088013	MIC-01 cable	N/A	N/A	Comes stock with CPC
45 Mechanized	088034	MIC-01 cable	N/A	N/A	RS485 Serial Option not available
65 No Torch	083266	MIC-01 cable	083267	Advanced Connection Kit	CPC & Voltage divider is an option
65 Mechanized	083294	MIC-01 cable	087105	Advanced Connection Kit	CPC & Voltage divider is an option
85 No Torch	087104	MIC-01 cable	087105	Advanced Connection Kit	CPC & Voltage divider is an option
85 Mechanized	087132	MIC-01 cable	087139	Advanced Connection Kit	CPC & Voltage divider is an option
105 No Torch	059731	MIC-01 cable	059732	Advanced Connection Kit	CPC & Voltage divider is an option
105 Mechanized	059380	MIC-01 cable	059386	Advanced Connection Kit	CPC & Voltage divider is an option
CPC-voltage Divider Kit	228696	MIC-01 cable	These parts are for Hypertherm units that are not ordered from the factory with the listed options. These kits can be added in the field to the listed models. Model 45 cannot be used with RS485 option.		
RS485 Serial Interface Kit	228539	Advanced Connection Kit			

Mechanized model part numbers shown above are for North America voltages and have the Duramax full length mechanical torch with 25 ft leads. If you need other configurations please contact you Hypertherm reseller or Hypertherm Tech Support for the appropriate part numbers.

OLDER HYPERTHERM MODELS

None of the Older Models listed have the option to add the RS485 Serial so will not do full DCC control with Dynamic Cut Current

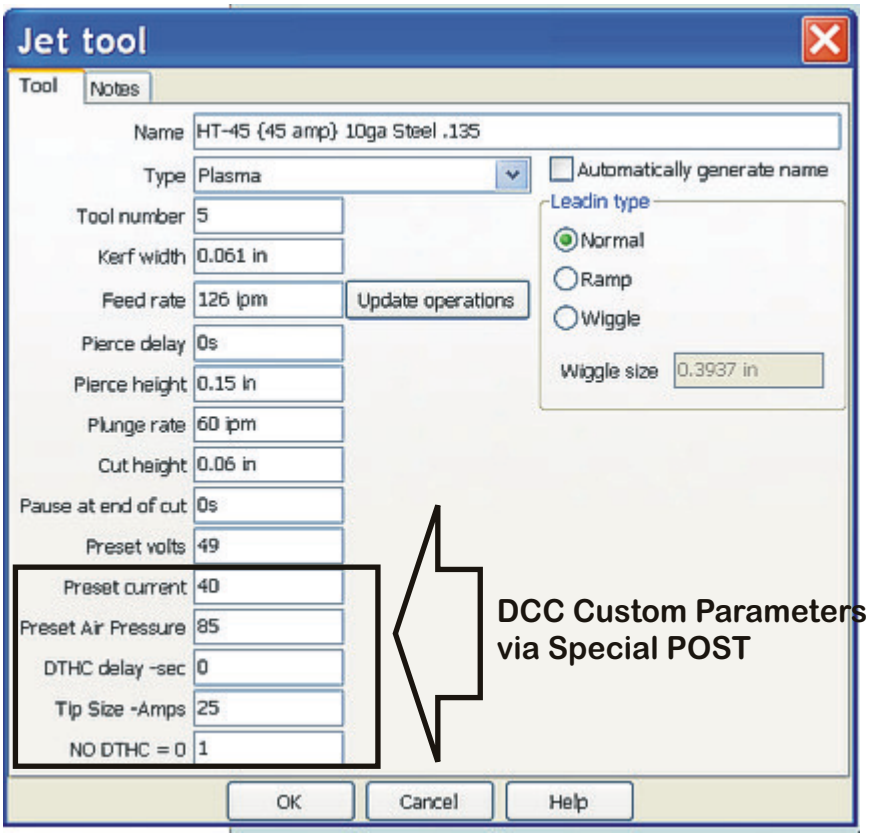
MODEL	CandCNC Interface	NOTES
600, 900, 1100	HyT-Connect RETRO1 (Universal Connection Kit)	Has all cards and modules for connection to any plasma
1000, 1250, 1650	HyT-Connect 1000 (Semi Automated	Includes cabel for CPC connecter (standard) and voltage divider card



refDistance: Is the distance in MM you allow of XY motion (total) before doing a Z REF (touch off) default is 500 mm (20 inches)

switchOffset. Set Using Configurator Application in CommandCNC as HOME SWITHC OFFSET in Z /

DCC CUSTOM PARAMETERS for DCC



NOTE: Three new Custom TOOL parameters added using the specified **MP3000-DTHCII-DCC+scriber** POST. If you do not have the input boxes shown you either do not have the correct SheetCAM version running or you have not selected the correct POST to generate code in SheetCAM. for the equipment you have.

Not All of the parameters are added as shown and listed below. The values of PRESET CURRENT and Preset Air PRessure only appear if you are using the **CandCNCPlasmaLLCNC HYT+scriber This post is for users that have the Hypertherm RS485 Option installed and the CandCNC HyT-Connect RS485 Sim Kit (included in the Advanced Connection Kit for the 65/85/105)**

KERF WIDTH. Based on the nozzle orifice you are using. The width of the cut used for calculating offsets for inside or outside type cuts. Actual kerf width can vary based on the feedrate, current setting, air pressure and age of the consumables.

PIERCE HEIGHT This is the height above the material the arc is started and and a pierce of the metal is started. On material thicker than 063 (1.5mm) that height is normaly 2X or more the normal Cut Height. It is done to prevent molten metal form splashing back and fouling the nozzle (tip).

Pause at End of Cut. Sets the end of cut delay. The time between when theTORCH OFF signal and the Z lift at the end of a cut. Most plasma arcs needs from .2 to .5 seconds to die out

FEEDRATE. The speed in IPM the job will be cut at. This does not set rapid speed and feedrate can vary based on the ability of the machine and the toolpath. Plasma cutting has recommended feedrate values for each type of material and current setting

PLUNGE RATE The speed in IPM you move from Pierce Height to Cut Height. The slower this rate the longer the delay is before the torch reaches proper cutting height and horizontal motion starts. Excessive delay will cause the starting hole to grow in size and the ARC VOLTS to start to climb. Typical rates are from 1./2 to 3/4 of the Z max velocity as set in motor tuning

PRESET VOLTS. Sets the value for PRESET VOLTS if you are running a DTHCII system and the DCC drivers. Overrides a PRESET VOLTS value from the Cut Profile when code starts. It provides a way to set the parameter at the time the job is defined. If the value is set to 0 the value is ignored. This value will be picked up in the G-CODE and displayed in the G-Code comments.

PIERCE DELAY. The amount of time the motion is paused AFTER the torch fires but BEFORE the plunge to Cut Height. Total delay is a sum of all delays before XY motion has started. Be sure to reduce the pierce delay so the plunge time (from pierce to cut height) is included

CUT HEIGHT This is the height the torch moves to begin the cut. Normally the recommend ARC GAP from the plasma manufacturer. Sometimes called Beginning Cut Height it defines where the torch is above the material when a horizontal cut is

PRESET CURRENT. Sets the value for PRESET AMPS (Cut Current) if you are running a DTHCII system and the DCC drivers. If you are running a HyT-Connect RS485 SIM install with a compatible plasma it will automatically set the plasma cut current. If you are using the DCP-01 the Preset Amps also sets the trip point for a Current Fault. This set the BEGINNING cut current in the code. That value can be changed during a cut but is the Default value if the override is canceled

TIPSIZE. The AMP rating of the consumable tip (Nozzle). It is stored in the G-Code and just reminds the operator the check for the correct tip. It has no other function.

NO DTHC. Overrides ALL DTHC signals for THIS TOOL so the DTHC will be turned off for the entire run of this tool if the value is set to "0". Default is 1 so DTHC IS on

DTHC DELAY; Sets the total delay time from when the torch fires until the DTHC takes over Z control. Time should allow cut to start and motion to reach full speed.

PRESET AIR PRESSURE : Sets air pressure in PSI on Hypertherm's with RS485

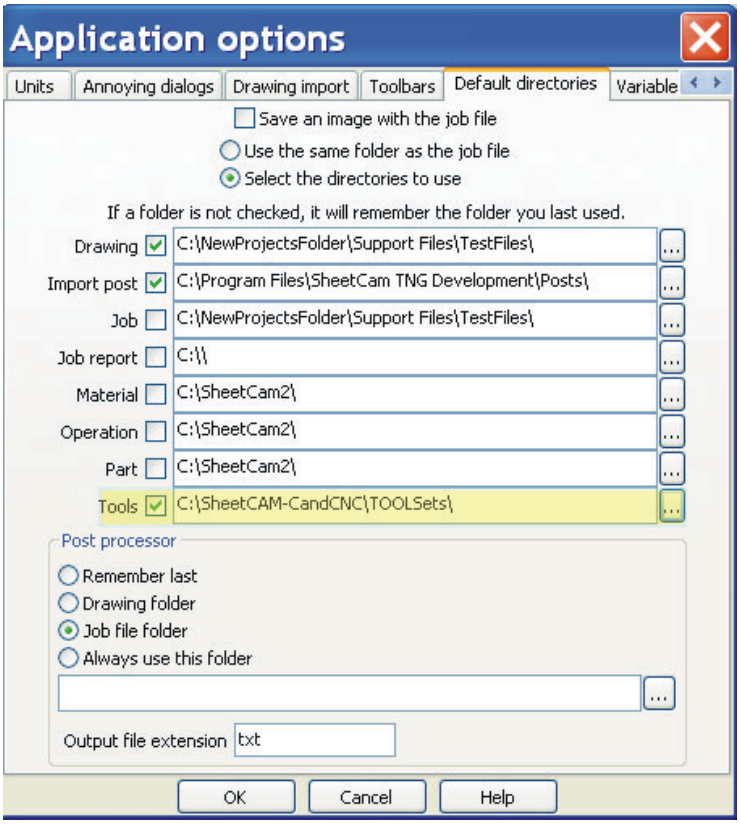
Yellow Highlights indicate DCC values. ORANGE Titles indicate values only available to RS485 equipped Hardware

The following pages will show you how you can use a special version of SheetCAM to take advantage of Dynamic Cut Control. If you have the full DCC set of options (including the RS485 options with a Hypertherm 65/85/105) than you can do some interesting cut control during a cut. There are two parts of the DCC for SheetCAM. The first is the ability to define PRESETS (same as calling up a CUT PROFILE. The previous page shows the custom parameters added to the tool table (toolset). If your setup will use Preset AMPS than that value can be used to setup the default job Cut Current. The Preset Volts and DTHC Delay values will work with any DTHCII based system and the DTHC ON/OFF function will work if you have a DTHCII REV1.4 or higher rev level card.

Setting up a new toolset

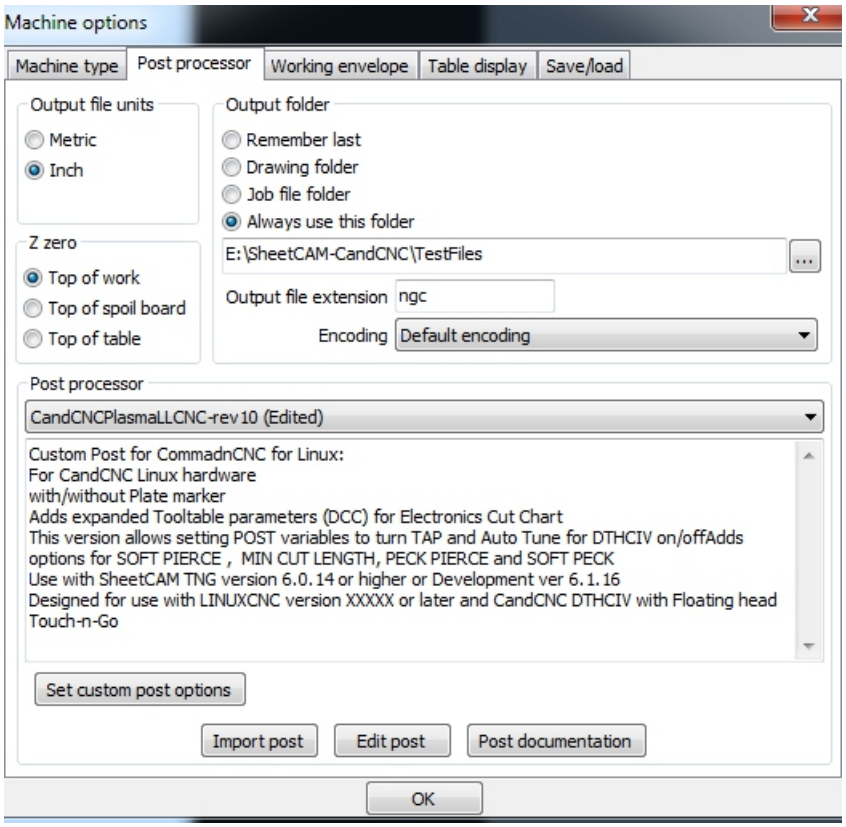
? Open SheetCAM TNG Development and make sure you have applied the license using the Help/Install License File

? Open OPTIONS/Application Options/Default Directories

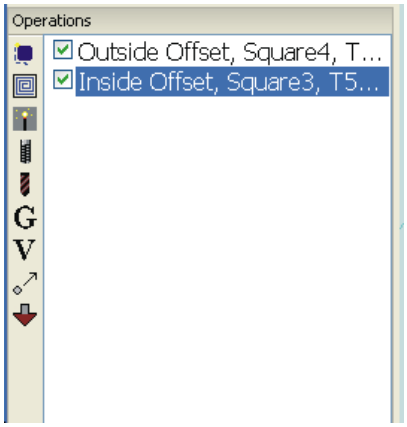


? Once you have the Default Directory set for Toolsets (other directories are optional) than you can have access to the custom toolsets we have provided for the Hypertherm 45/85/85/105. You also have toolsets for older 1000, 1250 and 1650 and you can build your own tool tables in SheetCAM (or make chages to ours) and store them under new names. Since you can now do the major parameter setting for plasma cutting in the G-Code you no longer need to match the CUT PROFILE in MACH to the job.

With SheetCAM setup and before you import a drawing (DXF, SVG) to process you should setup the environment. The setup of the Machine Options is essential. The values for Working envelope and Table display is setup for your specific table and covered in the SheeTCAM instructions. The important parameter for the DCC is to select the correct POST PROCESSOR before you start building Operations and using tools. Open the Post processor tab



? Select the Operation you wish to add the Action Points to in the OPERATIONS list on the lower left of the screen. When you select an operation it will highlight the contours associated with that operation and any custom operations like start points, and Action Points will show. Only one operation can be selected at a time and the other operations wil not show special operations, The Action points DO show on the contours as“X” points .

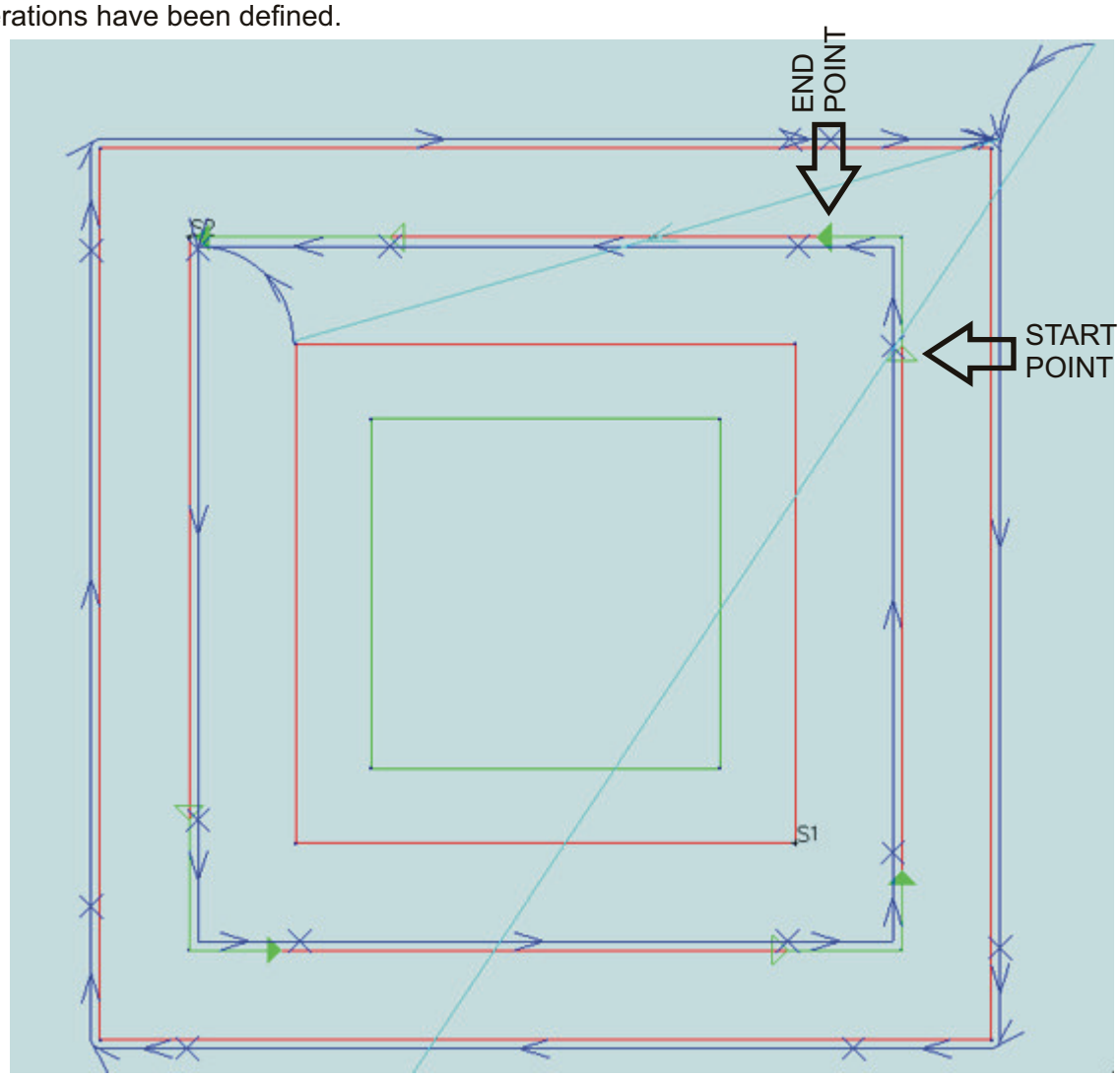


OPERATIONS WINDOW lower left of screen

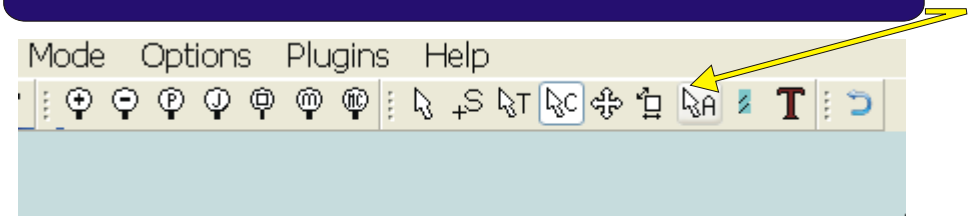
This example is not intended to be a tutorial on how to use SheetCAM and how to build OPERATIONS and assign TOOLS to a LAYER to build an operation. You should already know how to do that,

IMPORTANT NOTE: With the advent of Velocity Anti Dive that has corner detection and Auto DTHC off on slowdown, the need for complex or numerous Cut Rules in SheetCAM has become greatly reduced. In most circumstances you need no cut rules at all. The cut rule for small circles SHOULD NOT BE USED to control the DTHC ON/OFF. In the OPERATIONS section of a JOB in SheetCAM you will find a setting “Min Cut Length for DTHC” with an input box (default value will be 1 inch). You should use that setting and the TOTAL length of the largest hole PLUS the leadin. Example: 1” circle would work with 3.5 “

1. To understand how action points work open the TestCut-Basicshapes-APTest.job file in SheetCAM using FILE.Open Job and go to the C:\ SheetCAM-CandCNC\TestFiles Folder and load that job. You will see a job setup with several basic shapes that we use for test cuts. There are a series of squares and in the job file the outside two sqaures have been put on different named LAYERS and using the Hypertherm 45 Toolset two operations have been defined.

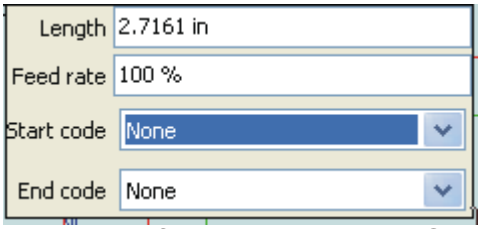


This example is not intended to be a tutorial on how to use SheetCAM and how to build OPERATIONS and assign TOOLS to a LAYER to build an operation. You should already know how to do that,



.

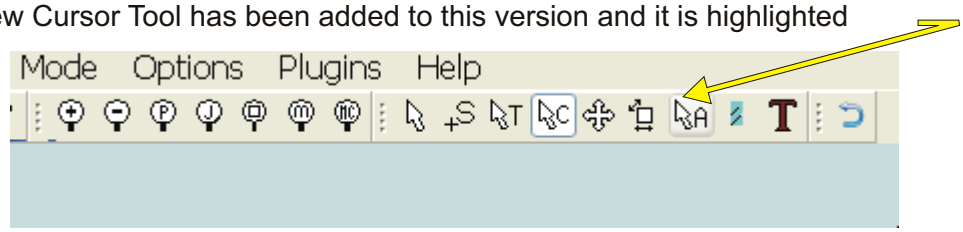
? Go to the Cursor Menu and select the Action Points arrow cursor.
? Click on the Contour where you want to set the points the direction (Start and End arrows will show as a hollow (START) and solid (END) arrow.
? The width of the span from START to END will be the last value used. Do not worry if it is not the right size for what you want.
? Note the direction of the Action Points will match the cut direction of the contour.
? Move the Cursor around the contour (you can move around corners or features which is handy) and left click to set the Action Points. If you move you mouse over the START or END Coursor and pause (no click) you will see a popup selection box like the one below.

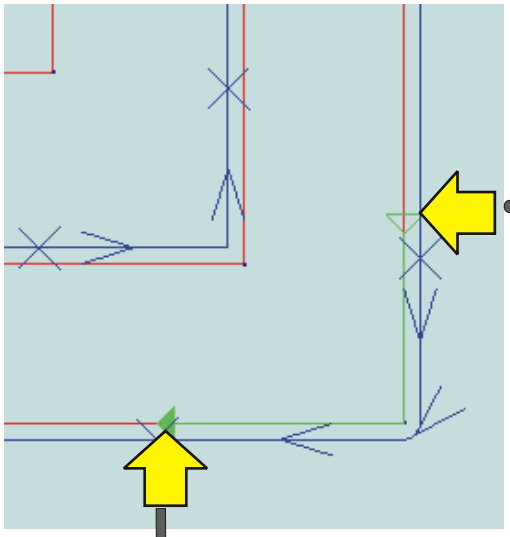


The same box pops up regardless if you hover over the START or the END Arrows. (may change in later releases of the software). Each field in the box is used to set a Action to happen at that point.

Feedrate	Start Code	End Code
this value sets a PERCENTAGE of the normal feedrate you have definedfor this OPERATION (slected at the time you build the OPERATION. This can let you define a slowdown point and will slow the commanded feedrate to that percentage. **The END value for FEEDRATE ALWAYS returns the feedrate back to the original value** **Do not use this to set the feedrate of your whole operation...it is temporary**	This allows a “Code Snip “ to be used at that point. We have included a large set of canned codes The button allows a selection of different codes. Using a special code system the DCC can turn the DTHC on or off and (in the same code), set a PERCENTAGE of the normal PRESET AMPS (Cut Current) for the Hypertherm units with the RS485 option.. Other “actions” can be defined in the toolset. If you change a value via a CODE then it stays that value until you change it back	This allows a “Code Snip “ to be used at that point. It is used to cancel any Start Code action. The values DO NOT return to default automatically EXCEPTIONS: The Feedrate is always returned to the last running value before the reduction if used. The DTHC is automatically turned back on at the beginning of the next cut. If you have set the tool to have no DTHC (No DTHC= 0) that OVERRIDES the automatic re-set of the DTHC

? In the toolbar located at the top there are a series of Cursor Tools. A new Cursor Tool has been added to this version and it is highlighted



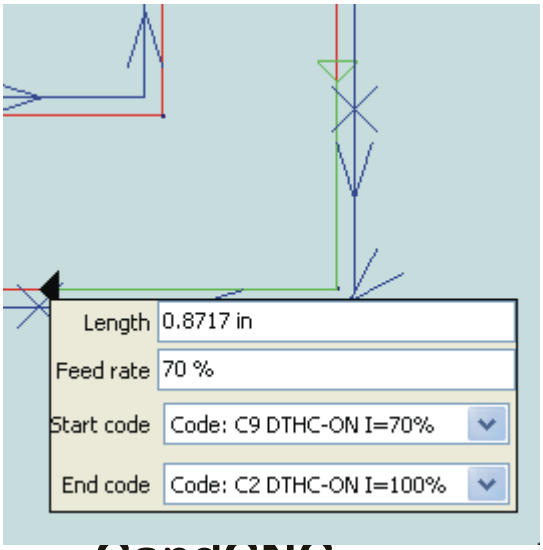


START ACTION POINT

THE TWO POINTS TOGETHER MAKE UP AN ACTION POINT**PAIR**

END ACTION POINT

To move/position the START (Hollow) and END (solid) ACTION POINTS after they are set, simply have the Action Point Cursor selected and move over an arrow, click and hold the left mouse button and drag the arrow along the contour and release to reset it at the new locations. You can do this with any arrow on the contour if there are multiple Action Point pairs



If you “hover” over an action point with it unselected for abut 1 sec the Parameters window will popup and you can see the LENGTH of the Span between the points (this number will change if drag the points) and Both the Start Mode and End Mode Code Snip names. In this example we have define a set of action points and selected a canned code from the Hyperthem-45_Plasma Tools toolset. The code reduces the feedratte at the START arrow to 70% of the normal feetrate for this operation (from 225 IPM to 157 IPM) and leaves the DTHCII active (ON) and reduces the cut current to 70% of the defined PRESET AMPS. The END Code leaves the DTHC active and takes the cut current back to 100%

Special Notes about the Action Tool:

1. The action tool "remembers" the last settings (Length, Feedrate and START and End Codes) so when building a cut file if you have several points you want to treat the same way (like corners) you can select the Action Point Tool, left click along an active contour, drag the action PAIR, and release and they will have the same settings as the last pair you used. This can let you build a multi-pair operation very quickly. If there are other points you want that are different you can come back and add them in after the repeat points are put down.
2. At this time you can drag a set of Action Points around a START point on a contour and it will perform the END action BEFORE it performs the START Action. This may cause some confusing code. This is a bug that will be fixed in a later release.
3. You cannot select the lead-in as part of the Action Point area.
4. The Feedrate and Cut Current are REDUCTION percentages so you cannot define values > 100% to INCREASE the base level Feedrate or Cut Current.

BUILDING CUSTOM CODE SNIPS FOR DCC

We have provided most of what you will need in the toolsets loaded during the install. It is recommended that if you want to build a new toolset you use one of the provided toolsets , edit it and save it off as another name in the same location. That way the existing The CODE Snips can be retained..
To build a CODE SNIP of your own remember the following.

1. It has to be a valid G-code command. IF you don't know what you are doing in G-Code BE CAREFUL!.

2. The Action Point processes that G-CODE at the exact spot in the toolpath shown on the screen. Some codes can cause the motion to hesitate while that line of code is processed. Calling a MACRO (M code) is an example.

SPECIAL S Codes used by the DTHCII:

10 - 19. The first digit (1) turns the DTHC off. The second digit sets the percentage times 10. Example: S!5 would turn the DTHC off (stop Z axis UP and DOWN commands) and set the Cut Current to 50% of the normal PRESET AMPS NOTE: The letter "I" in elecronics is "Current " measured in AMPS	**20 - 29**. The first digit (2) turns the DTHC ON. The second digit sets the percentage times 10. Example: S!5 would turn the DTHC off (stop Z axis UP and DOWN commands) and set the Cut Current to 50% of the normal PRESET AMPS

Continued next page

SPECIAL ## Codes used by the DTHCII:

LINUXCNC Prefix Code is M67 E0 M##

310 - 3300. The first digit (3)Indicates this is a volts Preset value. The next 3 digits set the actual value (in volts).
Example: S3117 send 117 volt value to the PRESET VOLTS DRO and transfer that value to the memory of the DTHCII. This is done using the value defined in the Toolset parameters window for Preset Volts/ This is automatically put at the beginning of the Code to set the values BEFORE cutting starts. The POST translates the value sent into the correct code. Wile it is possible to send a new Preset value during a cut using this code it is NOT RECOMMENDED since it changes the actual job (tool) preset value and has to be reversed out by changing it back in a code snip that could change with every tool

410 - 4200. (Hypertherm RS485 equipped models only) The first digit (4)indicates this is a cureent (amps) Preset Value. The next three digits sets the actual value (in AMPS)
Example: S445 would send 45 amps to the PRESET AMPS DRO. IF you have a Hypertherm plasma cutter with the optional RS485 port and the CandCNC RS485 SIM Kit that value would be loaded into the Hypertherm automatically. This is put at the beginning of the Code to set the values BEFORE cutting starts. The POST translates the value sent into the correct code. While this code could be used in a Code snip to send a specific value and cahnge the Cut Current DURING a cut it is nuch better to use the PERCENTAGE reduction in the 10 or 20 type codes.

50 - 520.9. The first digit (5)Indicates this is a DTHCII Delay (in seconds) value. The next 2 digits plus decimal sets the actual value.
Example: S51.5 sends the value of 1.5 seconds to the DTHCII delay DRO and transfers that value to the memory of the DTHCII. This is done using the value defined in the Toolset parameters window for DTHC delay-sec This is automatically put at the beginning of the Code to set the values BEFORE cutting starts A value of ZERO (50) is ignored by the control and it uses the stored Setting from Cut Profiles for the DTHC Delay. **NOTE: a value of 0.5 is VALID and shows a fractional (½) second.**

60 - 699.
Sets the number of volts (in volts) the Preset Volts can be *changed* DURING a cut. This is an INCREASE that in value to change in the event of a feedrate reduction. Example: A Code Snip with the code S65 would raise the PRESET VALUE by 5 volts. So a PRESET of 117 would raise it to 122. This has the effect of canceling the down motion of going into a feedrate slowdown.
CAUTION: be careful with this tool. Each plasma cutter will behave somewhat different and moving the PRESET VOLTS too far can have ugly consequences. NOTE 699 cancels the command and reverts back to normal PRESET value (in this case 117) Be SURE to cancel and increase on the END point.

70 - 7199 (Hypertherm RS485 equipped models only)
Not currently used at this time. Will be used to provide variable current piercing (ramped current during a pierce cycle)

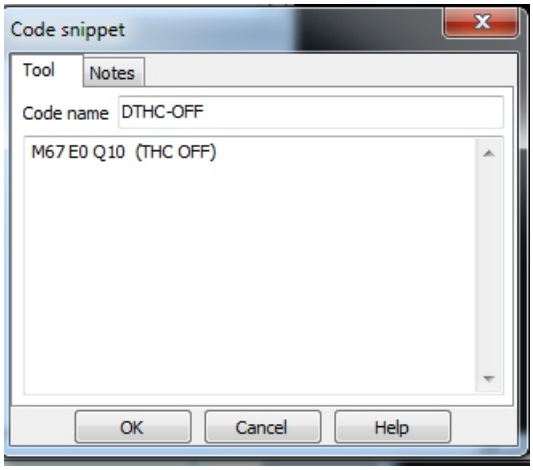
NOT USED

80 - 8199 (Hypertherm RS485 equipped models only)
Sets the AIR PRESSURE in PSI for cutting. This setting does nothing on Plasma Cutters that do not support remote setting of air pressure

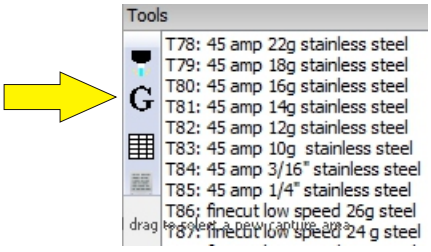
NOT USED

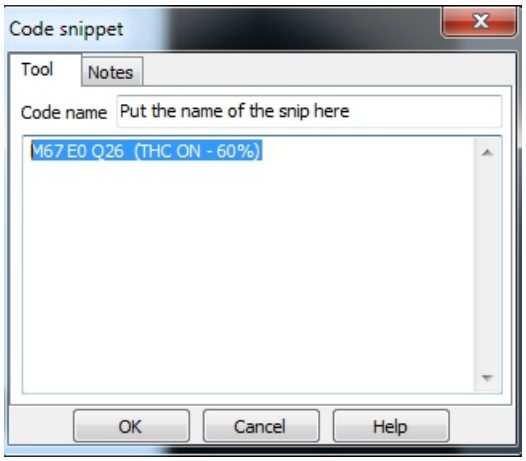
900
Turns ON the DTHCII signals (typically used to reset the DTHCII to ON after it has been turned off by a 999 code. This is a GLOBAL command meaning it overrides any other DTHC off.

999
Turns OFF the DTHCII signals This is a GLOBAL command meaning it overrides any other DTHC on. Command. It is set in the Tool Table as the DTHC OFF parameter. It disables the DTHC signals ***for that tool.*** It will override a 10 or 20 command as far as the DTHC being turned on/off



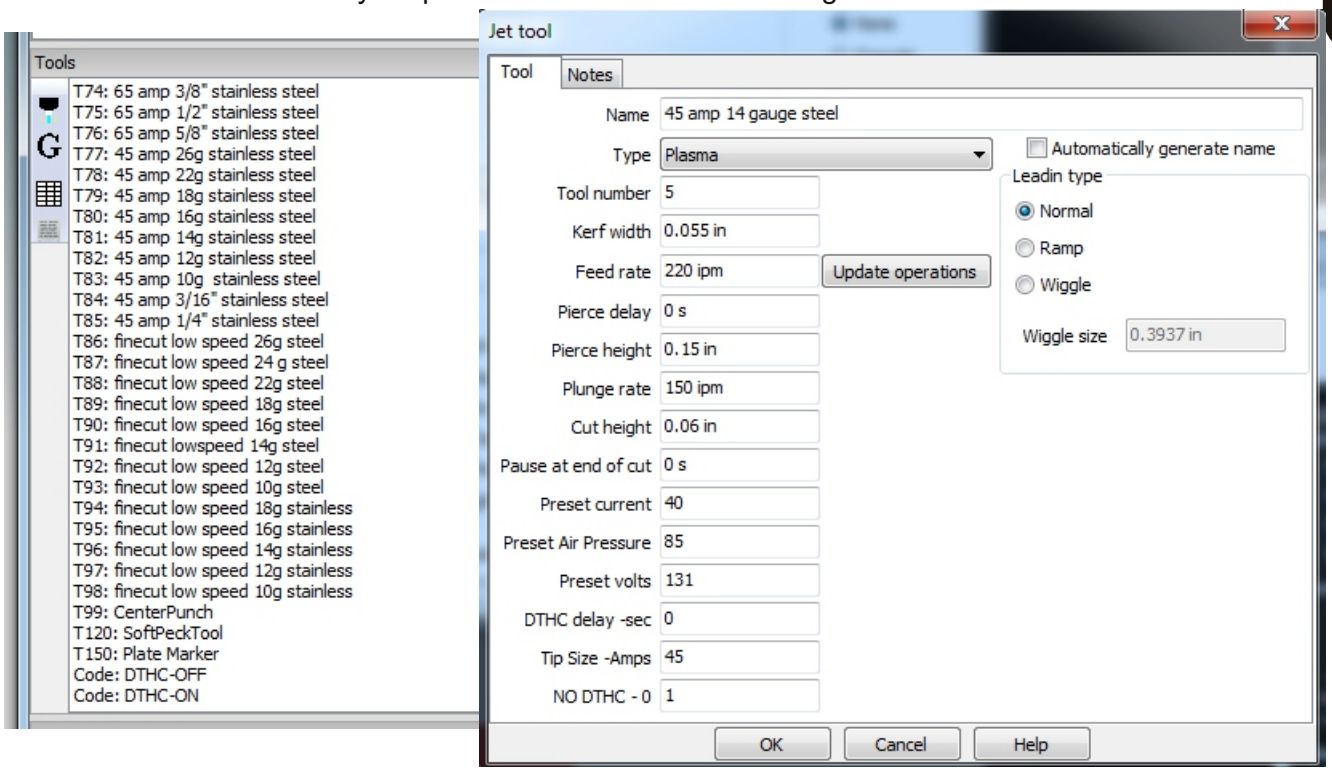
Setting up Toolset Codes for Cut Rules involves opening the toolset and then adding a “G” code snip and naming it . It will show up in the tool list then





In the sample G snip the “Q” number is the call to the special codes listed on the prior pages. The “2” means turn the DTHC ON and the “5” means set the cut current to 50%. A “20” means turn DTHC ON and set cut current to 100%. If you do not have the RS495 option with the Hypertherm RS485 serial port then the Cut Current number does nothing. The “2” portion of the code will turn on the DTHC.

? Once you have the new SheetCAM TNG REV 6.0.14 (or higher) loaded and setup and the access to the new toolsets, use the FILE/Open Toolset menu and find the toolset that matches your plasma. You will see something similar to the screen below



The screen above is a toolset for a Hypertherm 45-105 and we have selected the 45A 14Ga Steel setting. Note that the tool definition window shows a range of parameters that are set. Anny value can be changed by the user and **if you save the toolset before you close SheetCAM** that value will be stored in that toolset for futire use. For DCC the three lower values will allow you to store the DTHC settings for that material.

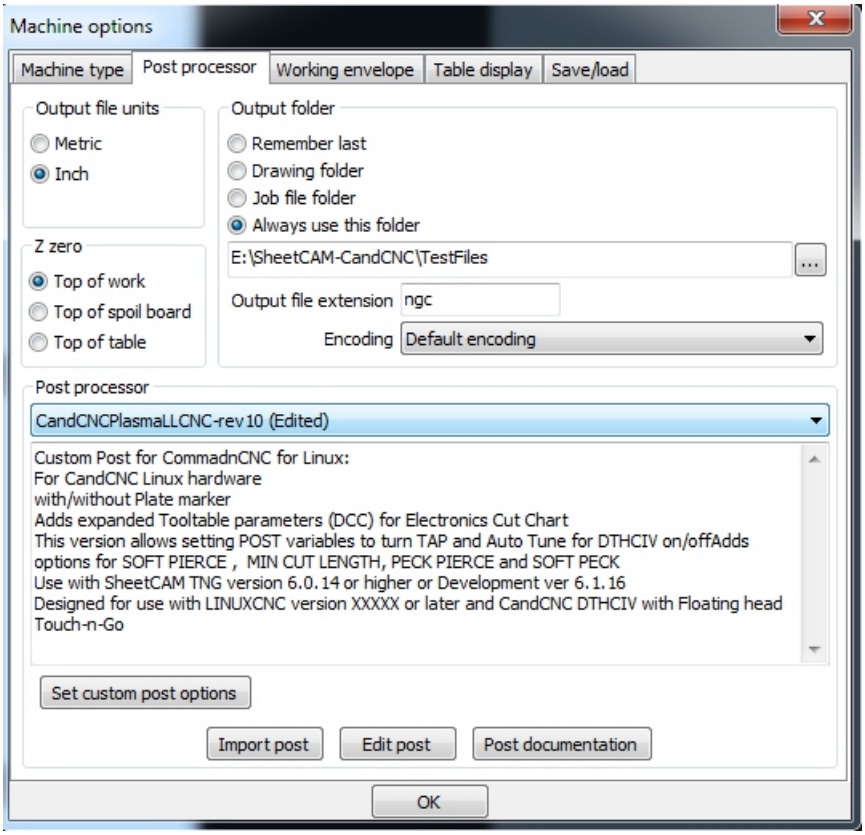
1. The DTHC has numerous settings that can be changed by the user. Most of the settings will remain at the default (like the SPAN VOLTS, ARC OK Value and Min-MAX values for faults.
2. The DTHCIV module has its own processor and memory independent of control or the application. The CUT PROFILES show the "Current Settings" (Current meaning the settings in the memory) and that is what the DTHCIV uses to cut from. When you make a change it gets loaded into the DTHCIV memory via a SERIAL (COM) which why you MUST have the DTHC ONLINE indicator ON so the DTHCII talks to the PC. The changes canbe made via the cut profile of the specific DRO or buttons on the screen. See your DTHCII USer Manual for a more in depth explaination.
3. When you load a g-code generated by this version of SheetCAM and the special POST included with this version than it will display the DTHC parameters as information text in the G-code (Window in CommandCNC Screen).

What this POST provides:

- Correct G-Code syntax for LINUXCNC (CommandCNC) systems
- The option to **run with TAP** (Total Automatic Plasmas for use with Hypertherm RS485 option and HyT-Connect RS485 option'
- The option to **use a plasma "drill" tool** to center mark (peck start) drill holes (gouge
- Has "**Set Custom Post Options**" for Ref Distance; Switch Offset
- **Minimum Cut Length** setting keeps DTHC OFF for short cuts and small circles
- Supports SheetCAM Job (global) and Operation level **Cut Rules** like corner slowdown, and DTHC off before end of cut.
- Supports **SOFT PIERCE** on Hypertherm's with RS485 (Cut Current remote set) including during a "peck start"
-

NEW FEAURES FOR SHEETCAM POST

This is a reference manual on using the new CandCNCPlasmaLCNC-rev10.scpost (or later rev). It covers options that you can set in the POST to make it behave in a certain way. To change these options you will need to open the specific post with a text editor or use the Edit Post option in the OPTIONS/Machine/Post Processor tab



NOTE: The first time you load or import a POST You may not see the Set Custom Post Options button. To see all of the buttons, close the Options menu in SheetCAM and reopen the Post Prcocessor tab again

--**********

--*** Set these values up to suit your machine ***

-- Values set HERE are in MM. If the value can be set in SheetCAM it is in the UNITS (MM or Inches) you have in SheetCAM
--**********
refDistance = 25 --this is the distance between each torch reference in MILLIMETRES.(can be set from POST Options button in UNITS)
lineNumber = 0 --set this to 1 if you want the G-Code to have assigned line numbers on each line. Default is 0 for LinuxCNC
defaultSwitchOffset = 4.00 -- default touch0ff swtich offset in MM (uses this value ONLY if the value set in the Set custom post options is zero)
minLength = 0 -- default minumum length of cut for DTHC operation (set dynamically)
dthcTune = 1 -- set this to 1 if you are using the new DTHCIV and want variable tuning of Z response from code
fullTAP = 1 --set this to 1 for TAP via Hypertherm RS485 (requires options)
plasmaDrill = 1 --set this to 1 to use a plasma for a drill tool . Set it to 0 to use a drill or router bit
warnings = true -- set this to false to turn off the Check Parameters warnings on a toolchange
verbose = true -- set this to false to turn off the G-Code comments (except code snips)
-- make sure you have the PROBE input in LINUXCNC Input Signals enabled and mapped to the probe switch input
refFeed = 508 --this is the reference feed rate in mm/min NOTE used only with (G38.2 probe touch off)
startCode = " M3 S100"
noTouchOff = false -- set to true if you want to run in simulation without touchoffs. Normal setting would be false

-- do not change the following values unless you are told to do so by an experienced LinuxCNC user or vendor of your controller or table
toolTrack = .254 -- this is the blending tolerance in MM (deviation from toolpath) the Path Blending allows. Lower numbers favors tight tracking over the defined feedrate (slows down)
toolTolerance = .0254 -- this is the linear tolerance for multiple nodes in the same toolpath. Lower numbers cause nodes out of line by less than the toolTolerance amount to be a single line
--**********
--Scriber X,Y,Z offsets in MILLIMETRES. Do not use inches here even if you want inch code
--For the marker use the scriber Z as the offset from Z zero you want the Z to move to during a scribe.
--Change these values to suit your scriber setup
scriberX = 10
scriberY = 10
scriberZ = 10
--scriber axis. Leave this as nil if the scriber is fixed to the same axis as the torch
scriberAxis = nil

--this value when set to "true" will set the Z to lift to rapid height between paths when using the marker tool.
--if false it will not lift the Z between paths but will at the end
markerZ = true
drillZ = .250

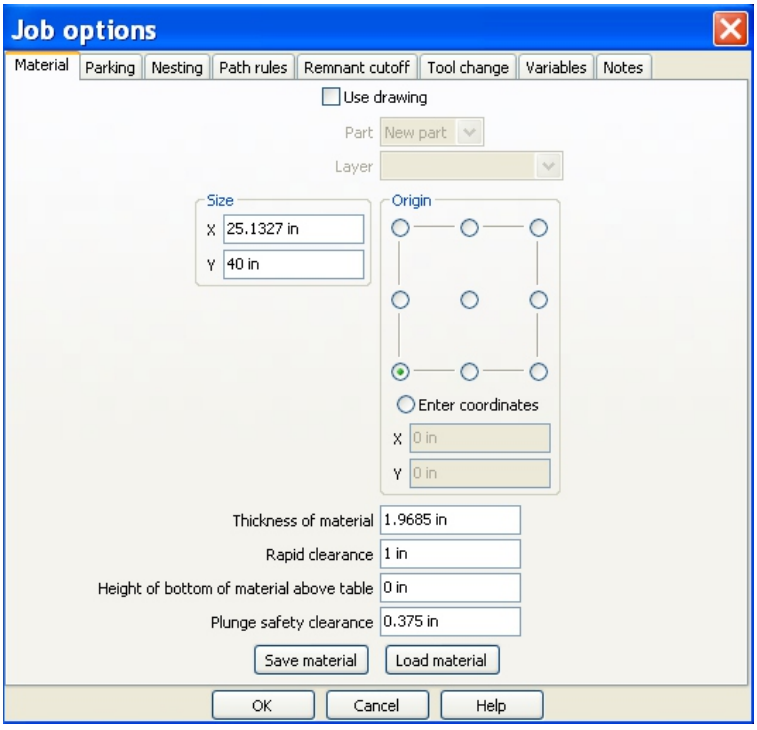
--these codes are for the DCC extensions and turn the DTHC on/off if you have DTHCII REV1.4 or DTHCIV REV1.6 or higher firmware. Used with
--ALL DTHCII and DTHCIV units
dynthcOnCode = " M67 E0 Q20"
dynthcOffCode = " M67 E0 Q10"

statthcONCode = " M68 E0 Q20"
statthcOffCode = " M68 E0 Q10"

NOTE about using the G38.2 with the Ether-Cut LINUX. The LINUXCNC codes do not use the G28.1 exclusive to MACH3 that does an axis specific touch off. Instead LCNC uses G38.2 probe command, The G38.2 looks like this

G38.2 Z- #### F####
Z-#### is the distance the Z will travel down looking for a probe swtich. If it travels past that point it will stop and not continue down but the system will report an error. The number is negative because it is below Z zero. It is automatically set to the negative number of the **Plunge Safety Clearance** setting in SheetCAM (Options/Job Options)

F#### is the feedrate in units as defined by the value in the POST Options as “refFeed” (in mm/min) The default of 508 mm/min is about 20 IPM. In the example below the Plunge Safety Clearance is set to .375” The would mean the Z will rapid down to a point .375 ABOVE 0 and move down at the speed of 508mm/min (20IPM) until it hits the touch-off switch (or the Ohmic sensor trips) OR it has gone -.375 BELOW the Z zero and will stop.



The **PLUNGE SAFETY CLEARANCE** setting in SheetCAM (Options/Job) still sets the slowdown point above the current zero. Before that it rapids down using the full velocity to the height you set in the plunge safety clearance.. **The two settings that are used in plasma are : Rapid Clearance and Plunge Safety Clearance. None of the other settings are used in plasma**

fullTAP = 0 --set this to 1 for TAP via Hypertherm RS485 (requires options)
This is to use **T**otal **A**utomated **P**lasma cutting. It turns on the ability to use the extended remote settings on a Hypertherm plasma cutter that has the OPTIONAL RS485 Port. You Must have the CandCNC HyT-Connect RS485 SIM Kit (or Advanced Connecton KIt) in order to implement this. If you select this option you get Preset Current and Preset Air Pressure added to the Tool Table for Plasma tools in SheetCAM TNG.
NOTE: *If you implement this option and DO NOT have the above listed hardware installed and setup the settings will appear and get put in the resulting G-code but they are ignored. It will not cause problems other than operator confusion.*

warnings = true -- set this to false to turn off the Check Parameters warnings on a tool change
Several users have requested the ability to turn off the warring at the beginning of a cut or after a tool change that pauses operation and prompts the operator to “Check the Settings”. Setting this to “false” will cause the code to NOT pause and the prompt is not displayed. If you use the same tool and settings most of the time and seldom change anything including the Tip Size (in AMPS) then turning this off will stop an annoyance some have reported. We recommend you leave this set to “true”

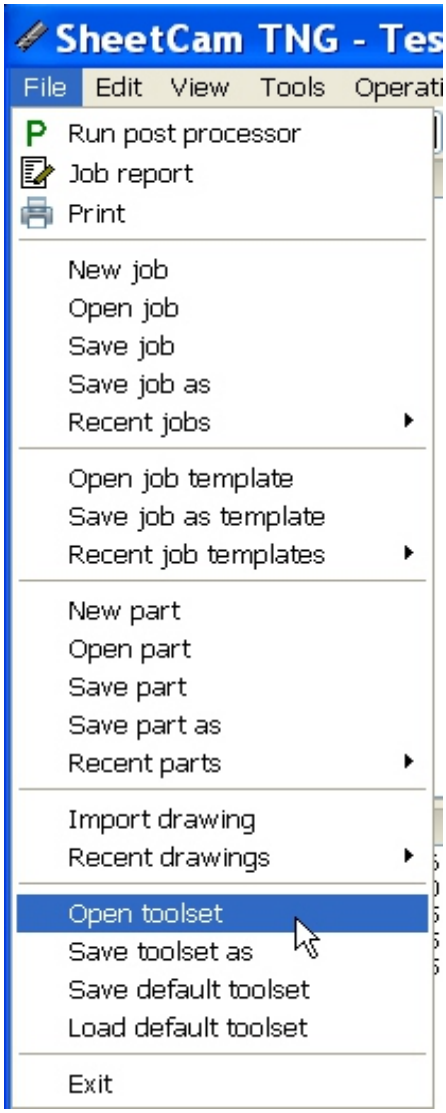
verbose = true -- set this to false to turn off the G-Code comments (except code snips)
This when set to false, turns off the verbose comments you get (within the parentheses) in the GCode. It helps in troubleshooting to have the comments so you can follow the g-code and does not cost any time or speed but some may want to set this to “false” to clean up the G-code scroll

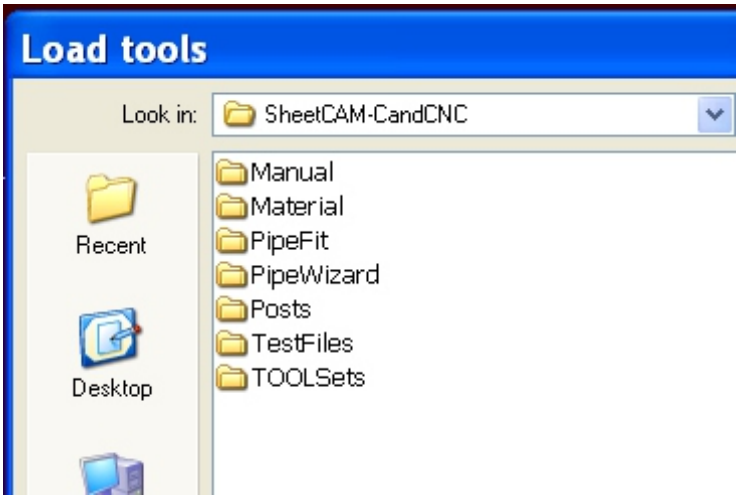
noTouchOff = false **T**his a special setting that is used when you want to generate a code that does not have the probe touch offs . It leaves out the touch off moves and allows you to run the code in SIMULATION in CommandCNC without getting all of the error messages on the probe not hitting on a touch off. Normally you would leave this set to “false” so the touch-off moves are in the code to do a noraml cut. If you

```
--************************************************
--***          End of settings                 ***
--************************************************
```

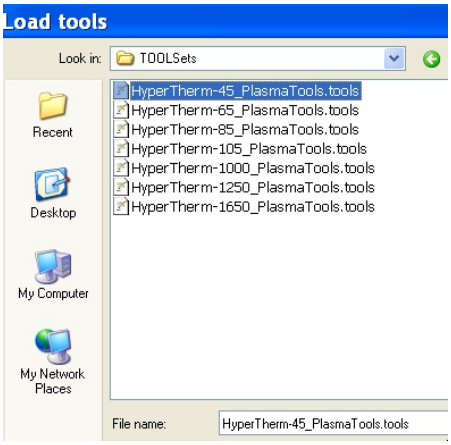
Most of the new POST options show up in SheetCAM as either a Tool Table (tool parameter) or in the Operaton Window. The difference is that tool parameters are for the whole job as long as that tool is selected. A tool is selected for an Operation but most operations in a job will use the same tool. An Operation is created when you select a TYPE of operation (i.e. Jet Cutting, Drilling, Contour, etc) from the Operatons Menu Within each Operation Type there are various tools you can select. The tools you can apply to an Operation are from the ToolSet you have loaded in SheetCAM.

LOADING A TOOLSET IN SHEETCAM TNG



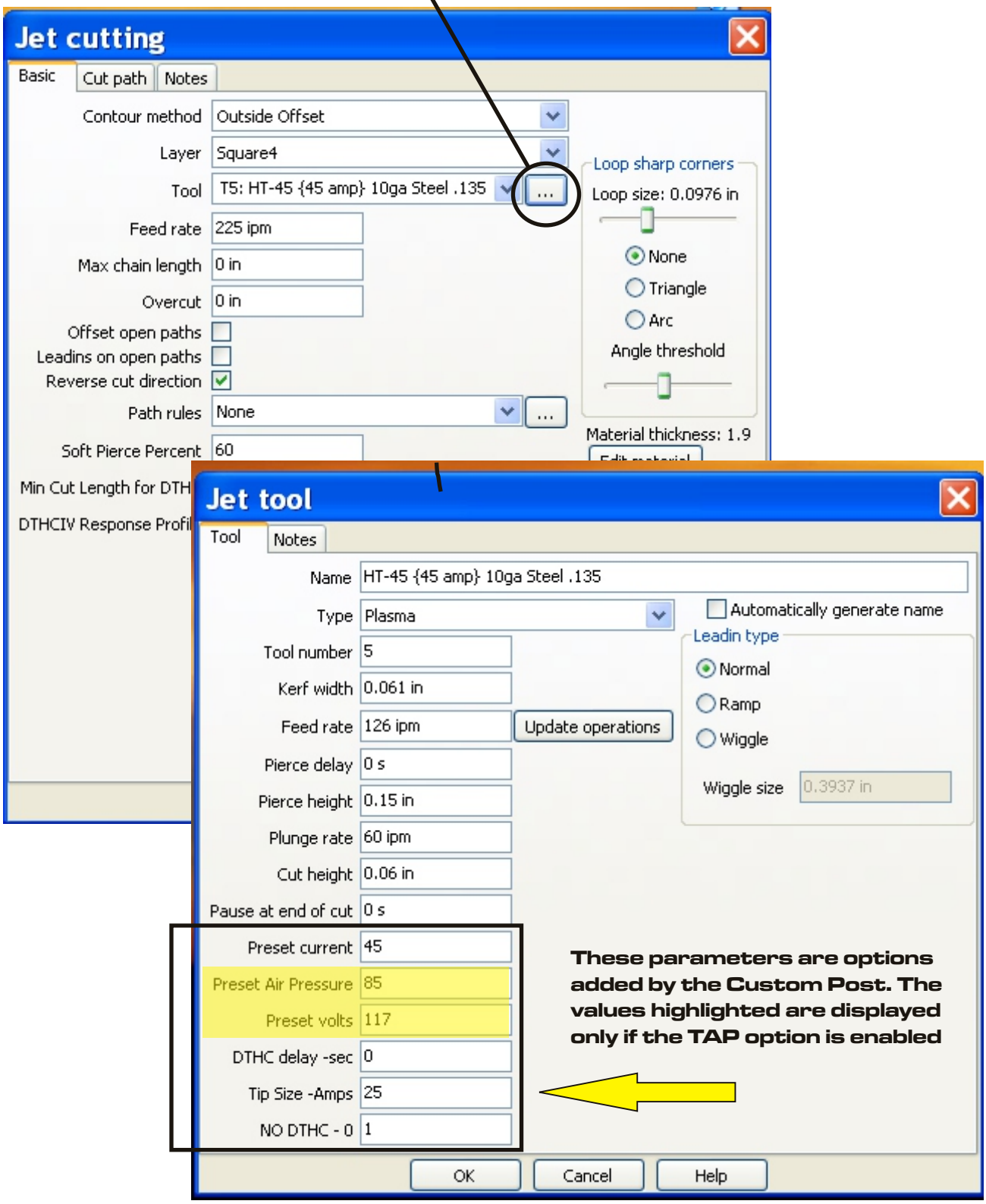


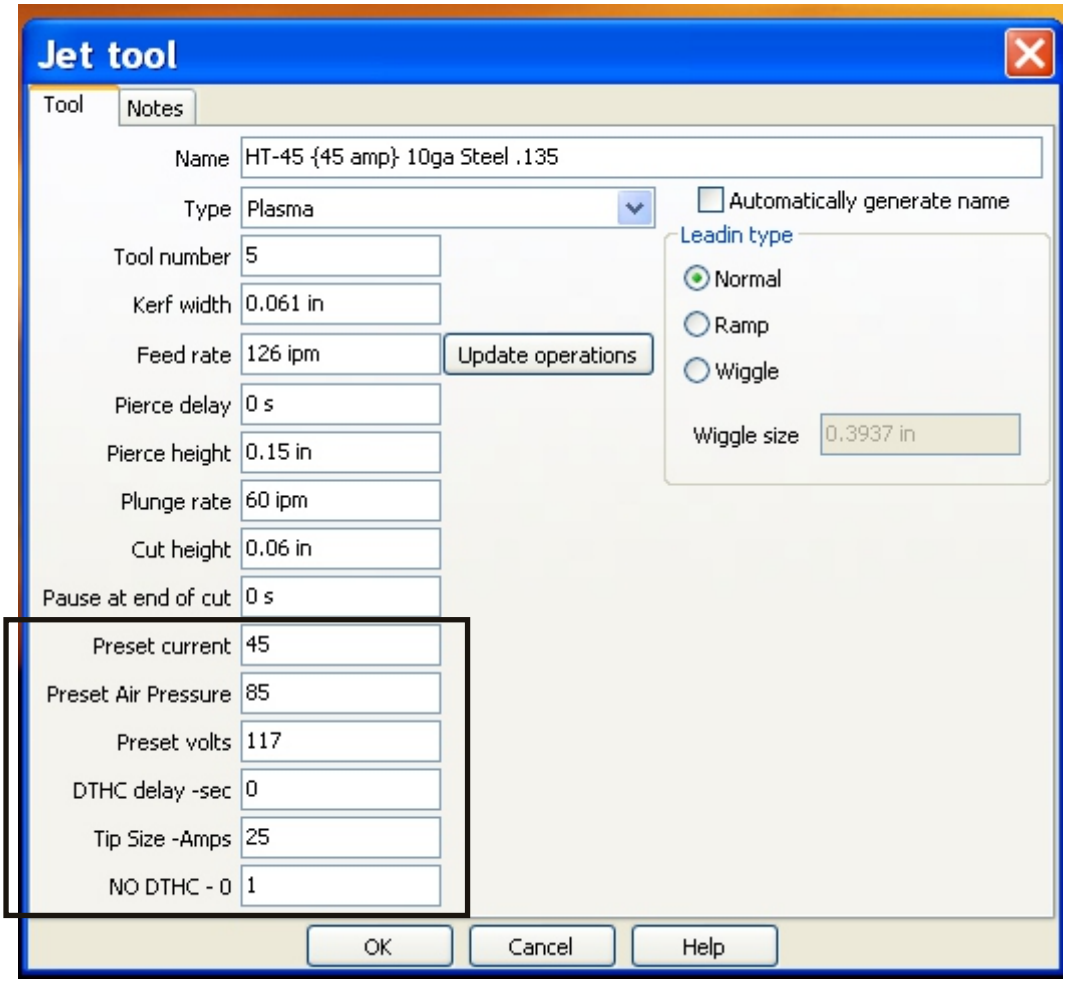
Open toolset will open a sub menu with a File Browser (Load Tools) It will allow to navigate the folder structure of the PC . As part of the SheetCAM Support Install from CandCNC the SheetCAM-CandCNC Folder is placed on your C:\ drive. It has several sub folders that hold custom Posts, and toolsets. Open the TOOLsets folder by double clicking the folder.



Using Options in SheetCAM

Once you have processed the contours and assigned layers in SheetCAM TNG you then define an Operation that will use a specific Contour Method, Layer and Tool. The Tool you select will appear in a popup window if you click on the button next the Tool Drop Down box

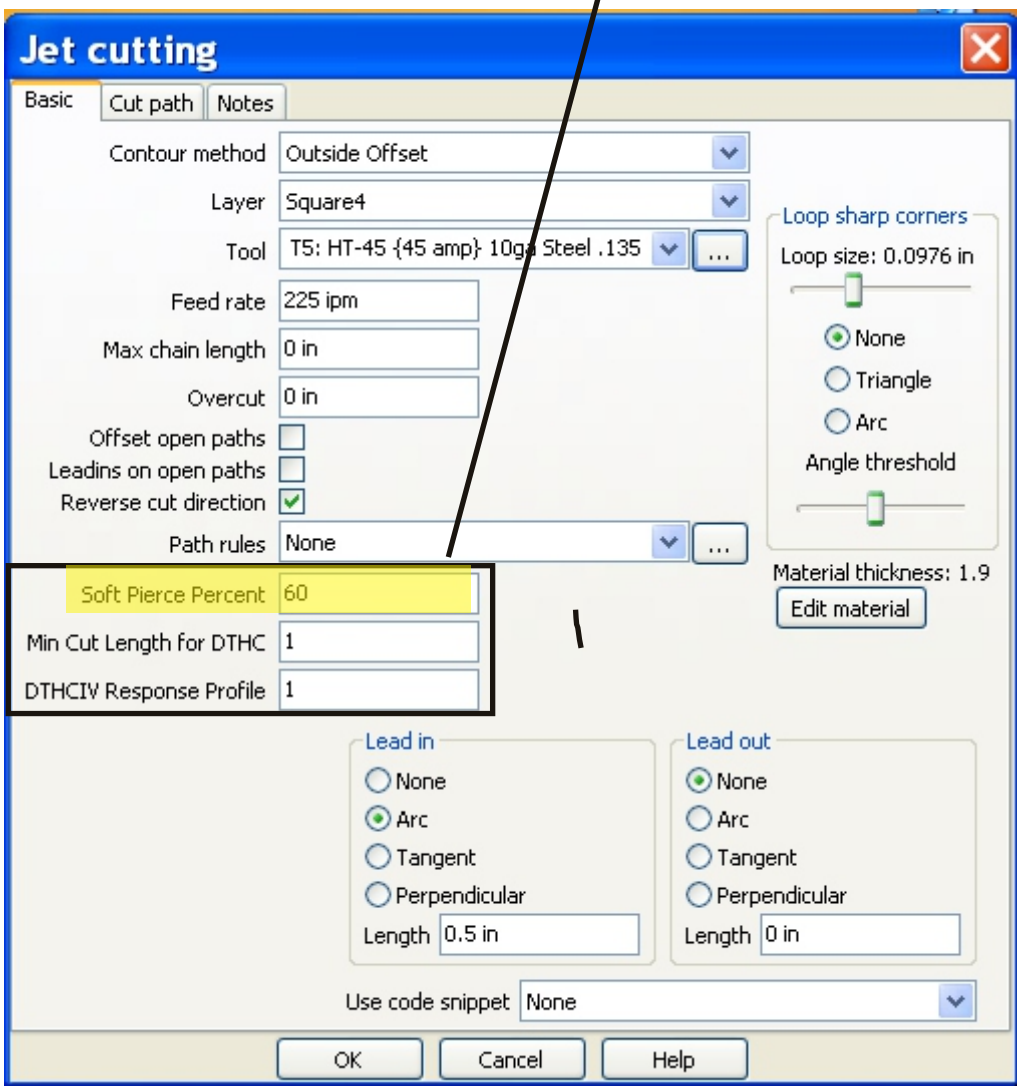




Preset Current . Only active if you have a TAP enabled plasma that allows remote control of the cut current (AMPS). It requires a plasma with an RS485 port and the CandCNC HyT-Connect RS485 SIM Kit. It will still transfer the value into the Preset Current DRO in the Screen flyout but it cannot change the actual cut current on a plasma not equipped with the optional port. It won't hurt to have the value in the ToolSet and helps remind the opeartor to change the current setting on manual models
Preset Air Pressure. Works only with TAP enabled plasma cutters
Preset Volts Works with any plasma cutter and auto-loads the **Preset Volts** into the Preset Volts DRO. This is the target value for Torch Height gap. This is one of the base settlings for all plasma cutting. Get the proper value from the plasma manufacturer or from testing
DTHC delay - sec The time the DTHC waits until it takes over. In previous POSTS this was measured from the time the torch fired and included the (variable) plunge time and pierce delay. **In this POST the timer STARTS when the torch reaches the CUT HEIGHT so is a more predictable time** and is typically between 0 and 1 sec. If your torch dives into the cut right after the pierce than increase this value to stop the premature dive. The DTHCIV needs the more predictable time.
Tip Size - Amps. This is strictly a reference value and is used as a REMINDER to the operator to change the nozzle size and has no affect on any physical characteristic.
NO DTHC - 0 . This setting turns off the DTHC FOR THIS TOOL ONLY. This allows you to define a tool as being for non-THC cutting. This setting has been largely outdated by using the **Min Cut Length for DTHC** and /or **Cut Rules** in SheetCAM. To set a tool to NOT use the DTHC you must make this value 0. Be sure to turn it back on (set to 1) unless you save it as a new tool.

Using Options in SheetCAM

Once you have selected the Tool and set the parameters there are several Options that will appear on the Main Operations window. Depending on the Options you choose in the POST edit you may see all or none of the added options in the box below

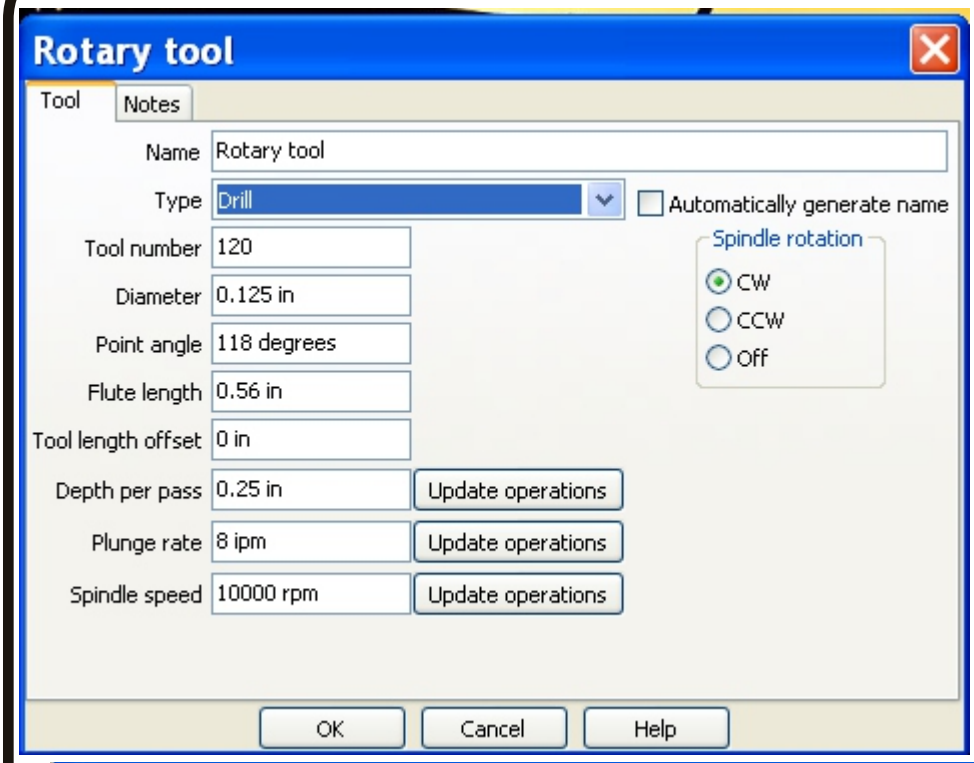


Soft Pierce Percent (only displayed if TAP option is selected in the POST). This sets the Cut Current to be used for piercing and is a percentage of the Preset Current. It lowers the current and slows down the plunge rate so the pierce is slower and less intense. Testing shows major improvements in consumables life using Soft Pierce Percent in the 60 to 70% range. You MUST have a TAP capable plasma and the matching RS485 interface hardware and software to use this feature

Min Cut Length for DTHC Displayed on most screens. This feature is handy to control the action of the DTHC for smaller cuts. It looks ahead at the cuts in the operation and any cut length less that the value does NOT tunr on the DTHC. It works on any shaped object: holes, shapes, lines and includeds the leadin. You can invoke this rather than setting a Cut Rule for small holes and keep the DTHC turned off for smaller holes and cut lines

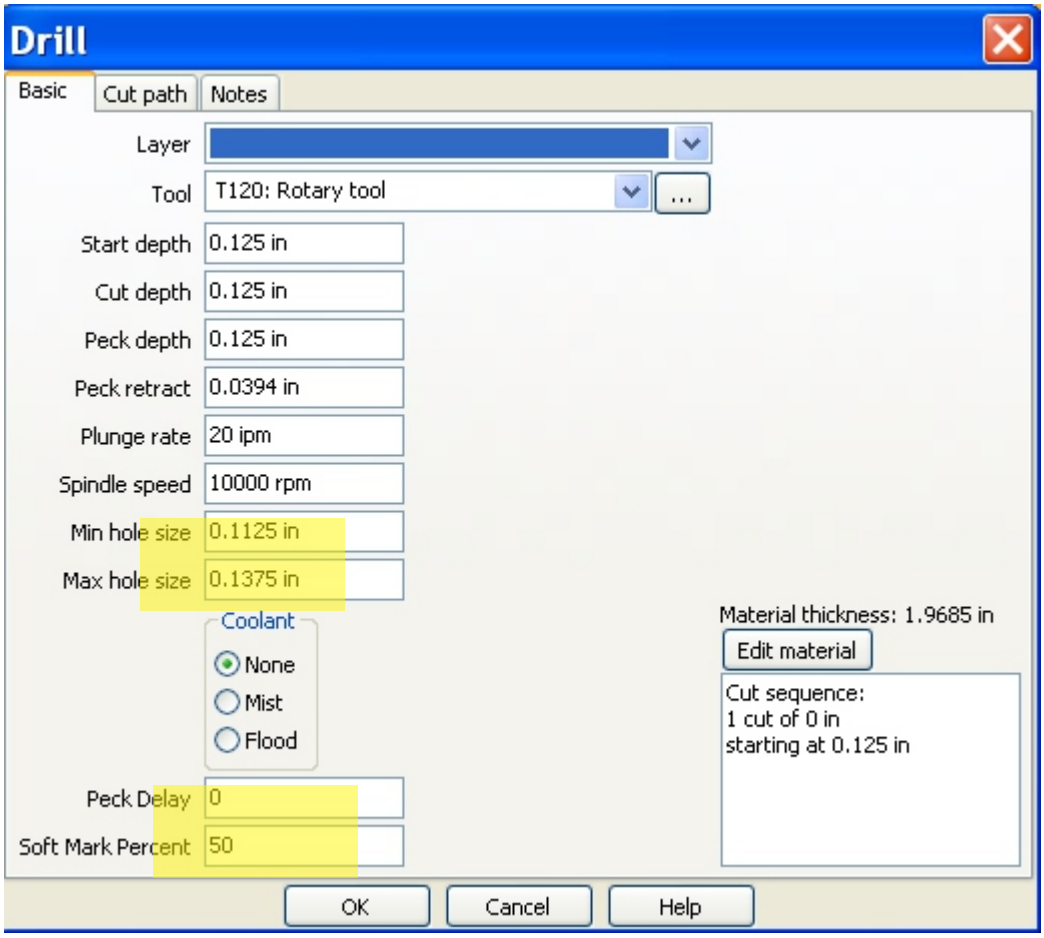
DTHCIV Response Profile Shows if you have a value above 0 in the dthcTune = option in the POST of have entered a value in the box above zero. To see it the first time you must have the Post value above 0. As stated this setting is not fully implemented yet.

Using the Peck Pierce for Center Drill



Defining a Rotary Tool

To use the Peck Pierce for marking center holes to drill you must add a Rotay tool to your ToolSet. The essential values are the TYPE (drill) the diameter and a spindle speed above 0 the other values are not used in Peck Pierce. When you do you drawing to do hole center make the hole you want to center a smaller constant size (I.e. .125) so when they are processed in SheetCAM the torch will auto center and peck the hole. **If the actual hole size does not fall in the Min and Max hole Size it will not peck the hole**



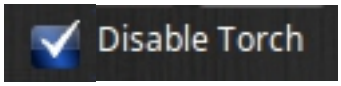
Define a DRILL Operation . Make the *Start Depth* the height ABOVE the plate you want to mark from . Set the *Cut Depth* and *Peck Depth* to the same value so you only have one Cut sequence (material thickness does not matter) Make sure the Min Hole Size is smaller thatn your actual hole size in the drawing and max Hole size is larger

Peck Delay (in decimal seconds) is how long the torch fires. .2 to .5 sec is suggested

Soft Mark Percent. Thsi work ONLY with a plasma that has remote adjustable Cut Current and is a percentage of the existing Cut Current to do the Peck Pierce

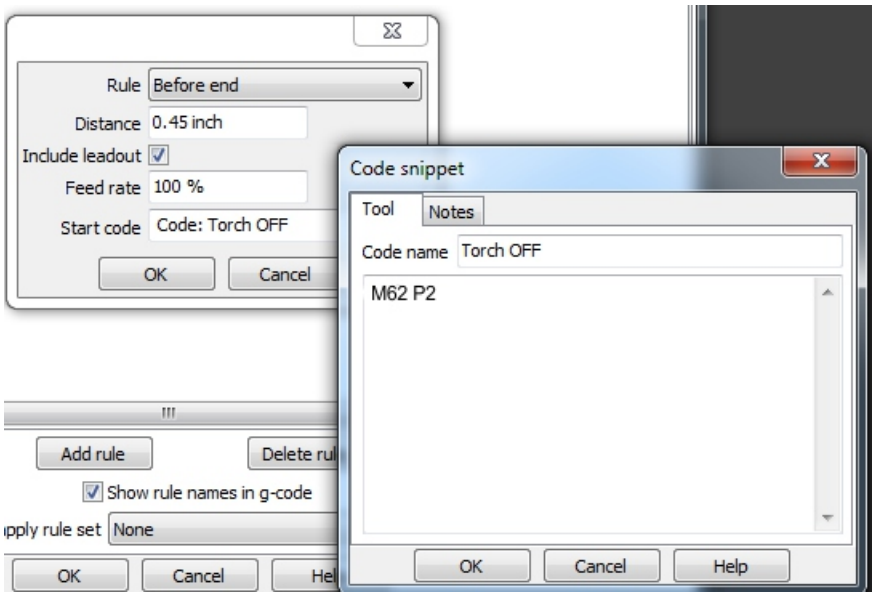
NOTE: Save ToolSet or Save Toolset As if you want to keep the new tools you define

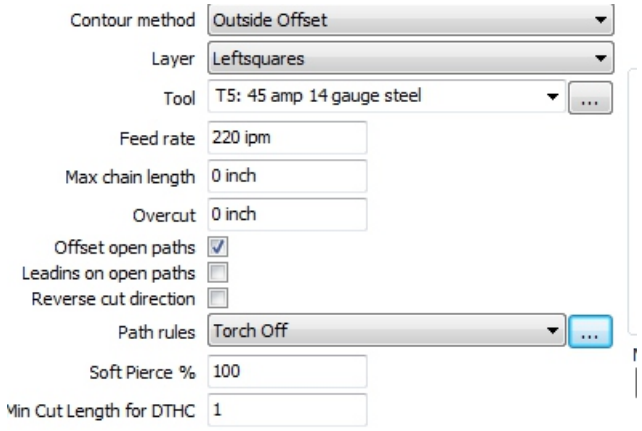
A plasma leaves a divot at the end of cut because the signal to stop the arc (torch off) is at the end of motion (torch has stopped moving) and the action is not instant, so the flame can persist long enough to burn a larger hole than the kerf width. To avoid this you either need to overcut the end, use lead-outs or to turn off the torch BEFORE it reaches the end of the cut. In CommandCNC you can do this prior to the M5 that normally turns off the torch by setting a “Before End” cut rule and make the distance something about .4 sec (its in distance so “time” changes with feedrate) before the end. The code to use for the Torch OFF while it’s in motion is:

M62 P2 

The M62 P2 turns ON the Disable Torch (which turns off the Torch and DTHC). The Disable Torch turns OFF at the end of a cut. You still need the M5 to end the cut.

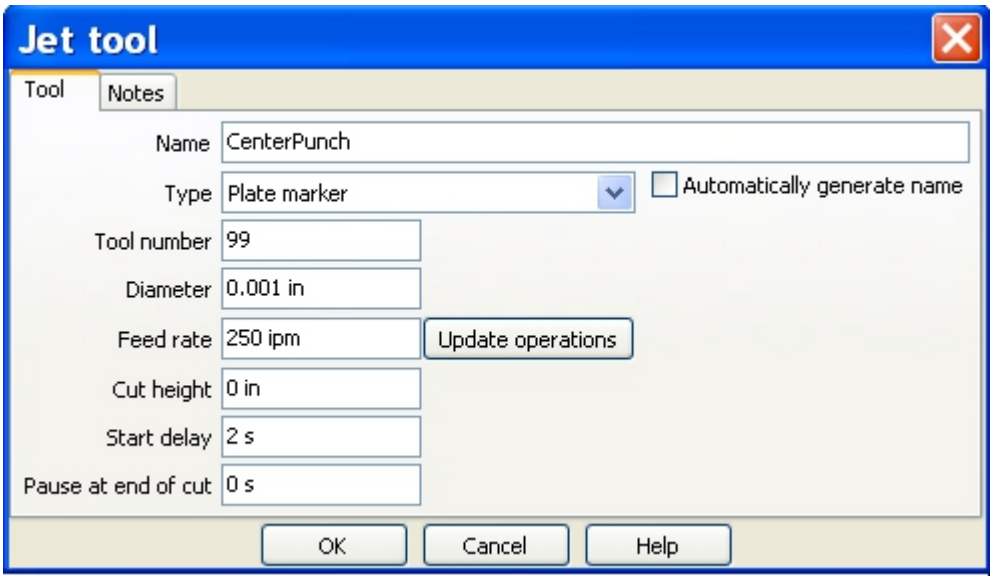
You will have to play with the distance in the Cut Rule to get it to turn off before the motion is stopped but not before the arc has reached the end. You will find there is some latitude in the time but it will change if the feedrate is a lot different because things happen slower or faster based on the speed of the cut. There is also a finite delay for the time it takes for the software to turn off the output.





To make it happen in an Operation you do not define it as global in the Path Rules. You simply select the rule from the list.

If you want the rule to run on every operation then you set the Always apply Rule Set to the Rule name in the Job Options/Path Rules



Defining a Center Punch

1. Setup a new Jet Tool
and use TYPE Plate Marker
2. Se the NAME to (exactly) **CenterPunch (same caps and no spaces)**

Your drawing should have either tiny dots or hole with center point . If you use CorelDraw or Inkscape holes do not have a defined center point in the drawing. You can place a dot (Period) and snap to the center and change it to an object from text.

Center punching is a Plate Marker process and you select that tool with the above NAME and place the points to be center punched on a SheetCA< layer and apply the CenterPunch tool. A start delay is acually the DWELL time and needs to be a second or two to peck the spot well.

Things to Know About CandCNC_LCNCRouter_REV2 POST:

1. Written for a full 3 axis (XYZ) setup
2. Uses a seperate Custom Tool Table in SheetCAM with just Rotary tools. Do not use the P Plasma Tools.
3. Z zero is considered the top of the material (Not the table top)
4. There are settings at the top of the POST (options) that have default values. You can see and edit them in SheetCAM by using the *Options/Machine/Post Processor* and the *Edit Post* button. **In most circumstances you should leave the default values**

```
--************************************************
--*** Set these OPTION values up to suit your machine ***
-- Values set HERE are in MM. If the value can be set in SheetCAM it is in the UNITS (MM or Inches) you have in SheetCAM
--************************************************
toolChangeX = 254 -- set distance in mm from 0 for toolchange event
toolChangeY = 254 -- set distance in mm from 0 for toolchange event
toolChangeZ = 76.2 -- set distance in mm from 0 for toolchange event
touchOff = false  -- set to true if you have a toollength touch-off device
toolLengthOffset = 0  -- set this to the Z offset for the toollength touch off device
refHeight = 25.4 -- this is the height above zero that you use for a manual height to use with a gauge block
touchOffSpeed = 254 -- set the touch off speed in mm/min
```

What the options mean:

toolChangeX, **toolChangeY** is a value to set the move-to position for changing tools. If you set those to 0 it will remain in the position where it stopped
NOTE:DO NOT MAKE *toolChangeZ* less than 1" (25.4mm) . It should be GREATER than the *refHeight* number . Default toolChangZ is 3.00" (76.2mm)

touchOFF = false (true) When this is set to "true" you have told the POST you have a touch off sensor attached and working. A touch off sensor is a device that can sense when the end of a tool touches a sensor plate or mechanical switch. (not provided) If set to "false" it assumes there is no sensor and instead will slowly move the Z down to the *refHeight* value and allow the operator to use a Gauge Block to loosen the tool collet and end of the tool to the *refHeight* *NOTE:* **You should leave this set to "false" unless you have installed and tested a working tool touch off device. Instuctions for installing and testing will be with the device you purchase.**

refHeight = 25.4 sets the distance above (initial) zero You can use other values than the default 1" (25.4mm) to match a gauge bolck you have but small values will possibly cause tool damage. The move the *refHeight* makes is ½ the *touchOffSpeed*

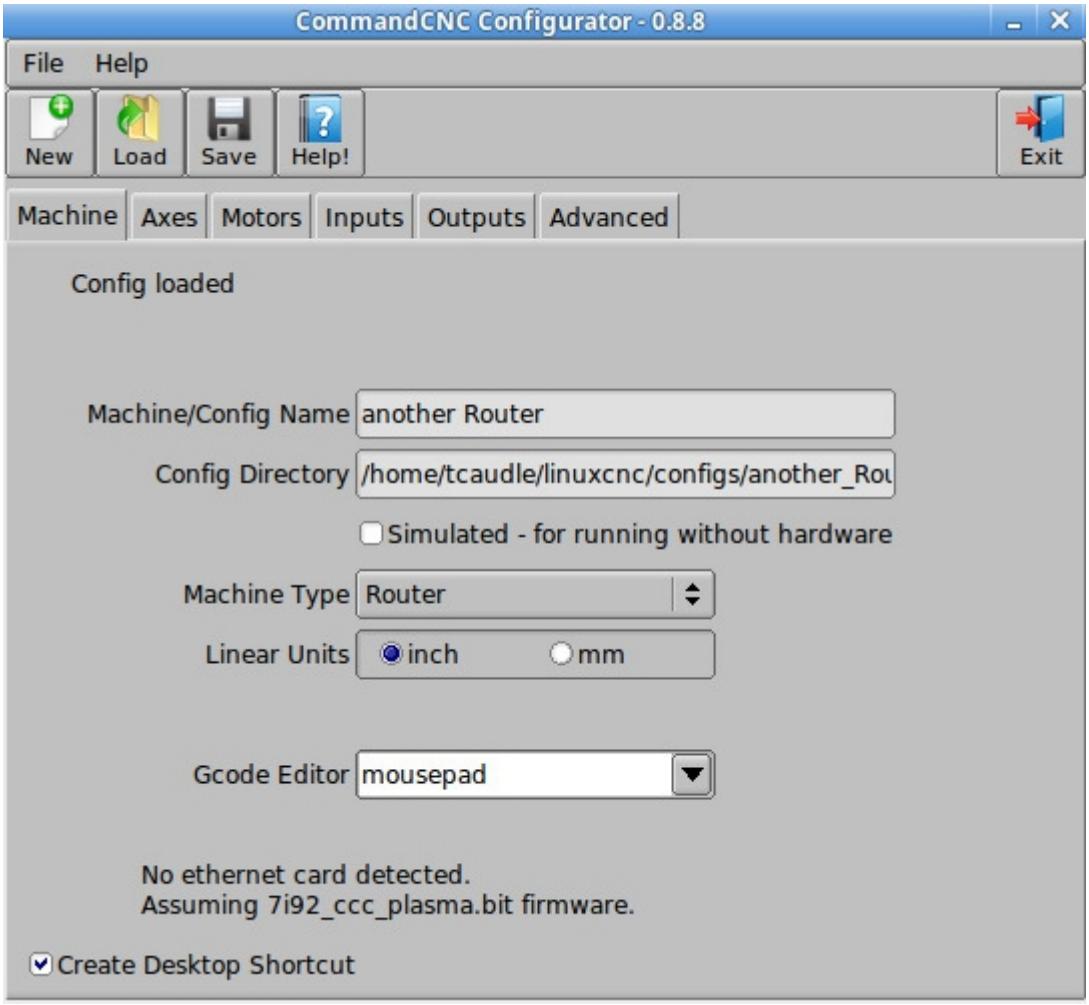
toolLengthOffset. Defines the offset (distance above work zero) where the touch -off device triggers. THIS IS ONLY USED IF YOU HAVE A touch off device and you have **touchOff = true**

touchOffSpeed. Sets the Z speed for doing either a touch - off on a device (see touchOff) or the movr to r*efHeight* to do a manual tool set. DEFAULT is 254mm/min (10IPM)

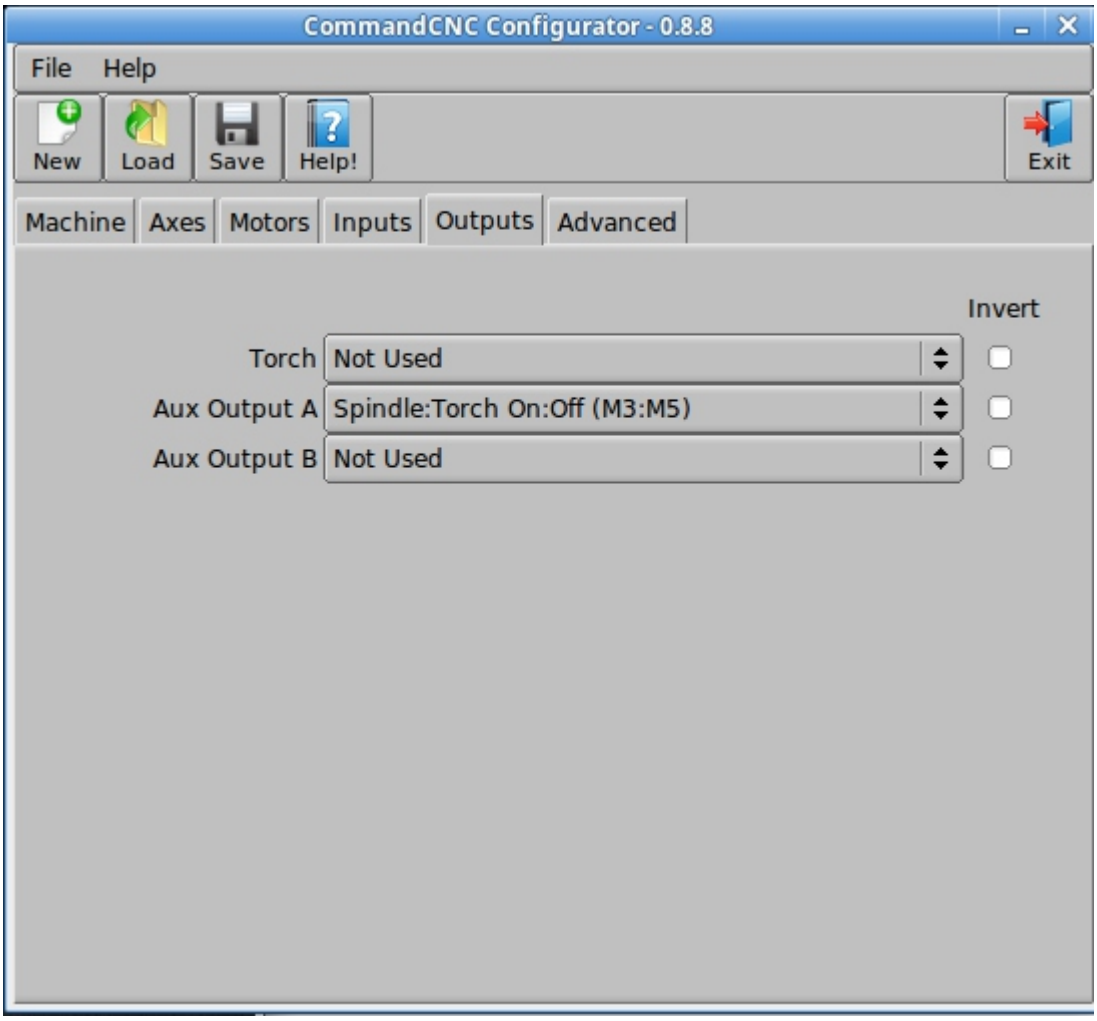
In most circumstances you should leave the default values

NOTES about setting up a Router Config in CommandCNC using the Configurator

1. All of the Axis, Motors and Advanced settings should be the same as a working plasma Config. You can click the NEW button and enter a new Config name and the Select the Item to Clone. When you do that, the Machine/Config name will show the new name and you are working on that Config
2. For a Router, change the **Machine Type** to ROUTER
3. Leave the **Create Desktop Shortcut** checked



4. You need to set the active output for M3 and M5 to either the *Aux Output A* or the *Aux Output B* . These are power relays on the Table I/O card and are wired to the AC Outputs on complete controllers. **The Torch signal is Not USED**.



These are the major settings you will need to change to use a router with a CommandCNC system.

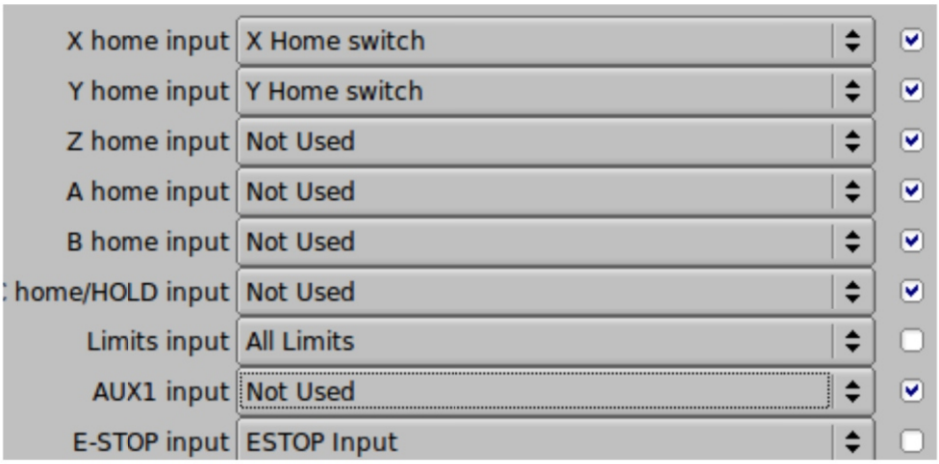
SAVE then EXIT the Configurator and it will create a desktop ICON for your new Router Config.

NOTE:
You may want to go back into the new Config (using Configurator tool) and slow down some of the motor tuning. If you are doing mostly light carving and taking small Z increments higher speeds are essential BUT for contour cutting to a depth or cutting pockets in denser material you rarely need more than 100 IPM . For routing harder materials like aluminum sheet you will need lower speeds and some form of cooling like mist and/or air. You rarely need more than 100 IPM . For routing harder materials like aluminum sheet you will need lower speeds and some form of cooling like mist and/or air.

Using the CandCNC_LCNCRouter_REV2 POST

Once you have used the POST in SheetCAM to generate G-Code, open the Router Config you generated in CommandCNC OPEN the G-Code file (preferred format is .ngc) and you should get a toolpath display (if you are in LIVE VIEW) .

1. **If you have X and Y homes on your table** then perform an X and a Y Home by pressing the buttons. In the example below we have left X and Y homes on but have disabled the other inputs except Limits and Estop. If you do not have Limits then set it to Not Used as well as Leave E-Stop ON!
Even if you have a Z home switch on your Router you need to turn it off (set to Not Used in



2. If you do not have Home switches or do not want to use them then open the Configurator and in the INPUTS tab set each axis to NOT USED, DO THE HOMING even if you do not have home switches defined .To set homes with homes defined then manually jog the table to the spot you want to have X0 and Y0 and press the HomeX and HomeY buttons.

.

That will zero the X and Y DRO;s and remove any offsets. Then perform a Zero X and Zero Y the Configurator)

To Home the Z axis: Put the material you are going to route onto the table. Mount the rotary tool you are going to use in the chuck or collet and carefull jog the Z down unitl the end of the tool just touches the surface (top of the material.) Hit the Home Z button and then the ZeroZ button. This zeros the Z to the end of the tool. You need to do this when you start the machine or when you change material (blanks)

Once the axis are homed you can move to the XY corner of the blank (if it's not at X0 Y0 or if you have not configured the job in SheetCAM using a Machine Zero then moving the parts to the absolute position from Machine zero .) When you locate the WORK X zero and Y zero you can click the Zero X and Zero Y buttons. DO NOT HOME there unless you do not have XY homes and want that position to be both WORK and MACHINE zero.

Bring the machine out of E-STOP and press the Aux Ouput A (or B) button where you have your router plugged to and check to make sure you can turn it on and off.

Click the RUN and it will cut the file. **See the section on PARKING about defining the ending location at the end of a cut**

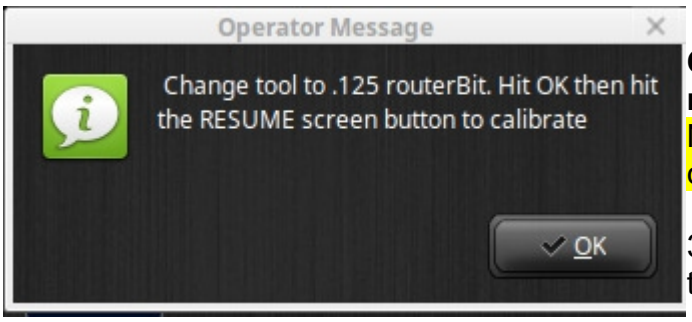
IF YOU HAVE MORE THAN ONE TOOL (number) DEFINED IN THE JOB:

The machine will make the first cut(s) with the first tool (tool 01) in the first operation regardless of the tool number in SheetCAM. The POST counts tool CHANGES and assumes you have the first operation and tool setup and with Z zero at the end of the tool.

At the end of the cuts using the first tool :

1, The Router Output will turn off . IMPORTANT : **If you are not using the Aux outputs to turn your router on and off you will have to do that manually** . DO NOT SKIP THIS STEP because just pressing the RESUME button will result in a possible problem.

2. The Z will pickup to the *toolChangeZ* height and it will RAPID (full velocity) in XY over to the XY poisition for the toolcahnge that you have defined in *toolChangeX* and *toolChangeY* If you have chnged those values to 0,0 it will NOT move to 0.0 but simply stays where the last cut ended

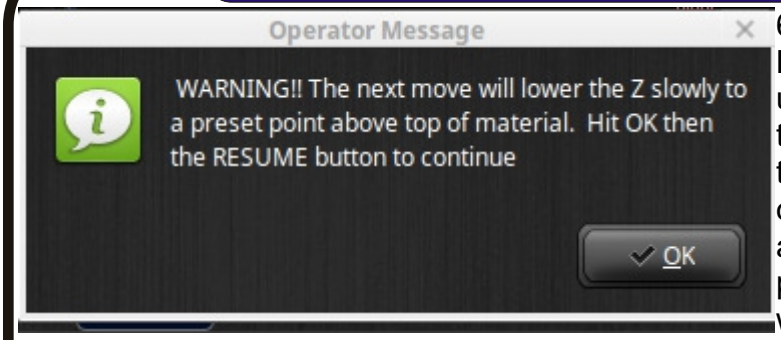


ONCE AGAIN: Make sure the router/spindle is OFF!
NOTE: when the machine is in PAUSE you cannot jog any axis.

3. Change the tool and do not completely tighten it down.

4. Hit the RESUME button

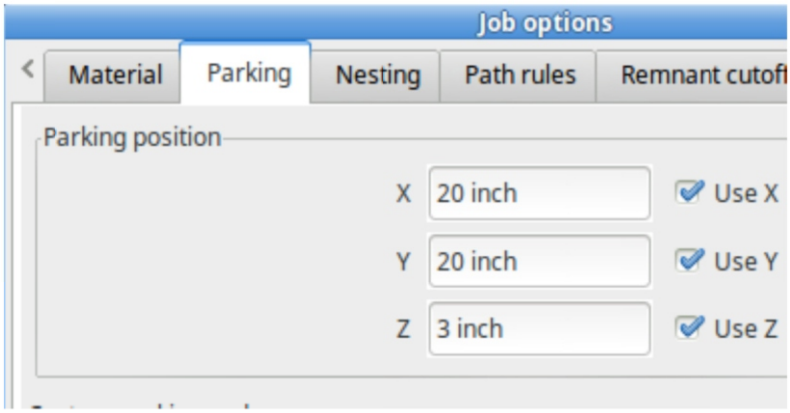
5. The Z will move SLOWLY at ½ the *touchOffSpeed* until it hits the *refHeight* distance.



6. Loosen the tool and slide your Gauge Block under the tool and adjust the tool up or down until the tip of the tool rests on the top of the Gauge block. The simplest thing to use is a machinists “1-2-3 block” or a piece of material machined to an accurate thickness. A piece of hard plastic or wood with a accurate depth works well. If the Gauge is not 1” (25.4 mm) then set the r*efHeight* in the POST to whatever the Gauge thickness is. Do not use thin material as a gauge block.
7. Tighten down the tool and **remove the gauge block.**
8. After setting the tool and removing the Gauge Block from the surface, hit the RESUME again. The machine will rapid back at full velocity and drop/raise the Z to Rapid Height (set in SheetCAM)
9 . The cuts defined using tool 02 will cut.
10 . If you have additional tools defined they too will do steps 1 thru 9 above.

NOTE If you STOP a job with multiple tools and use the **Run from Line** to start in the middle of a cut you may not get a toolchange for that cut. Always try to go back to the the previous M5 to get the next tool change.

Using the PARKING OPTION in SheetCAM



In SheetCAM *OPTIONS/ JOB OPTIONS/* Parking tab you will find parking options for XY and Z. By default they are turned off. If you leave them disabled (unchecked) then at the end of a cut the axis **will return to XY (MACHINE) zeros**. Z will remain at Rapid Height.

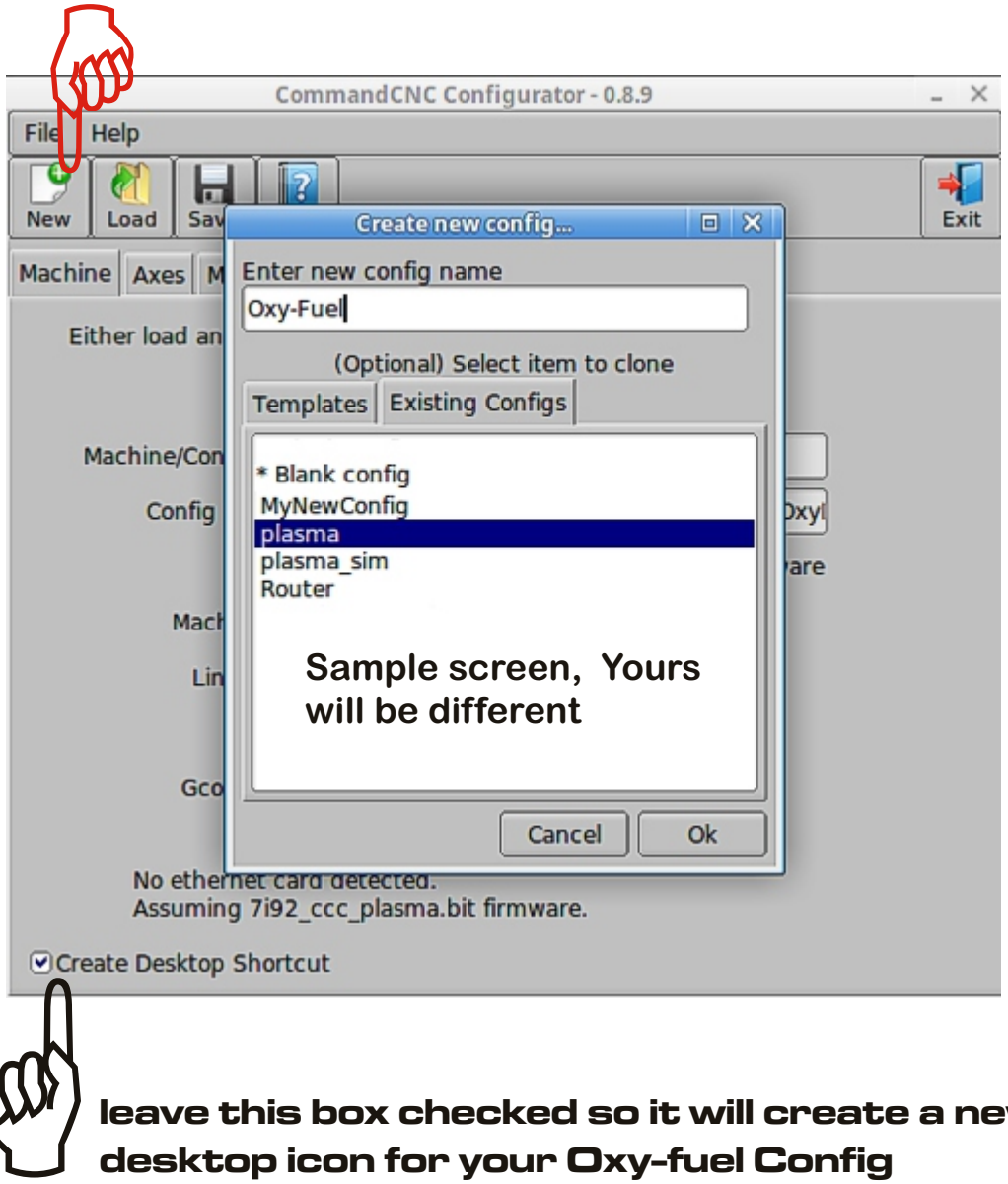
If you enable both X or Y and put in values greater than 0 the machine will return to those positions (as in the example above the machine will raise Z to 3” and move XY to 20, .20 (up and over) at the end of a cut. I

NOTE: Z will always use the HIGHEST of the Rapid Height or the Parking Z height if it is enabled

Since this is an option in SheetCAM the settings must be made prior to running the Post Processor and generating code.

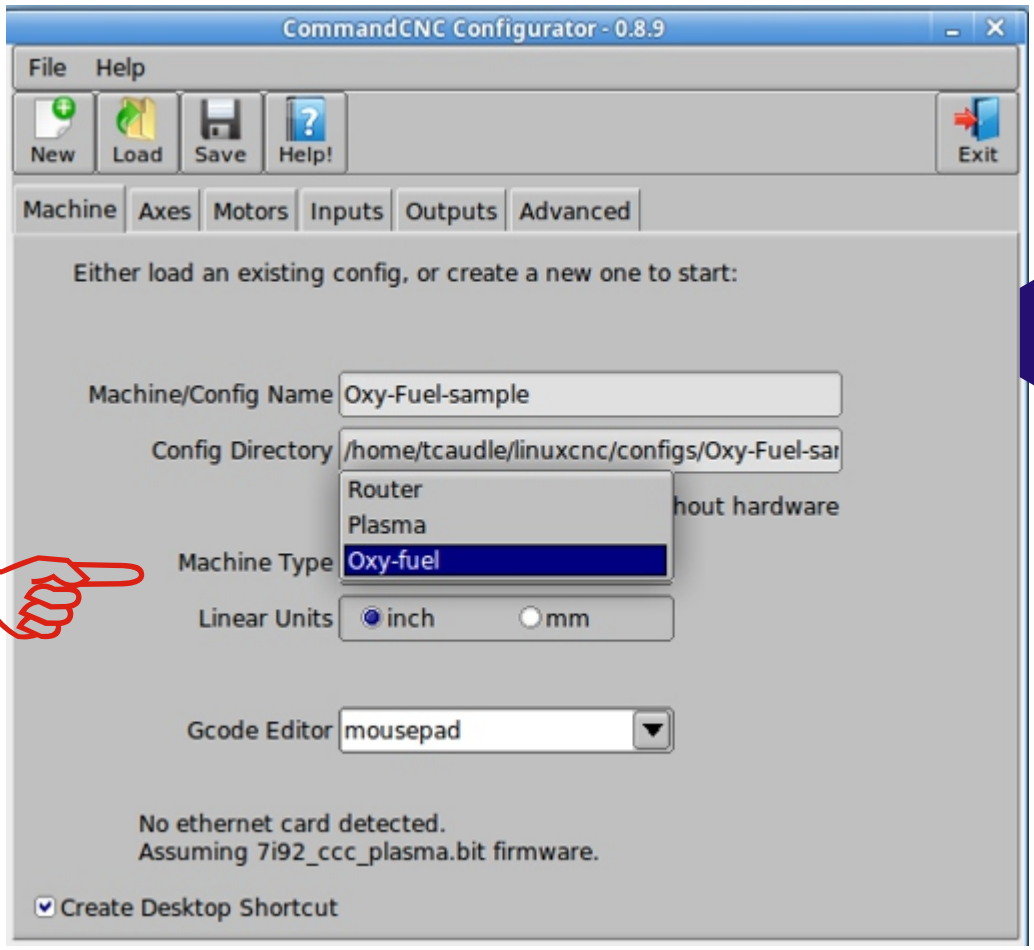
To setup an Oxy-Fuel Config you need to make a NEW Config by opening the Configurator and select on the Machine Tab the New Button. YOu will get a popup window that will display your current Configs. Enter a new config name in the top Input Box and highlight a working plasma Config to Clone. This will transfer all of your motor tuning, axis information and the inputs and outputs. We will be making some changes to several of the settings in the following pages.

A

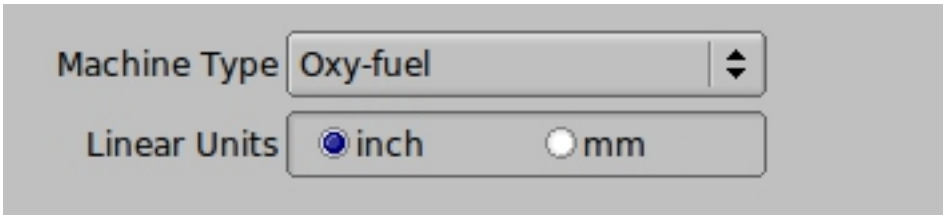


leave this box checked so it will create a new desktop icon for your Oxy-fuel Config

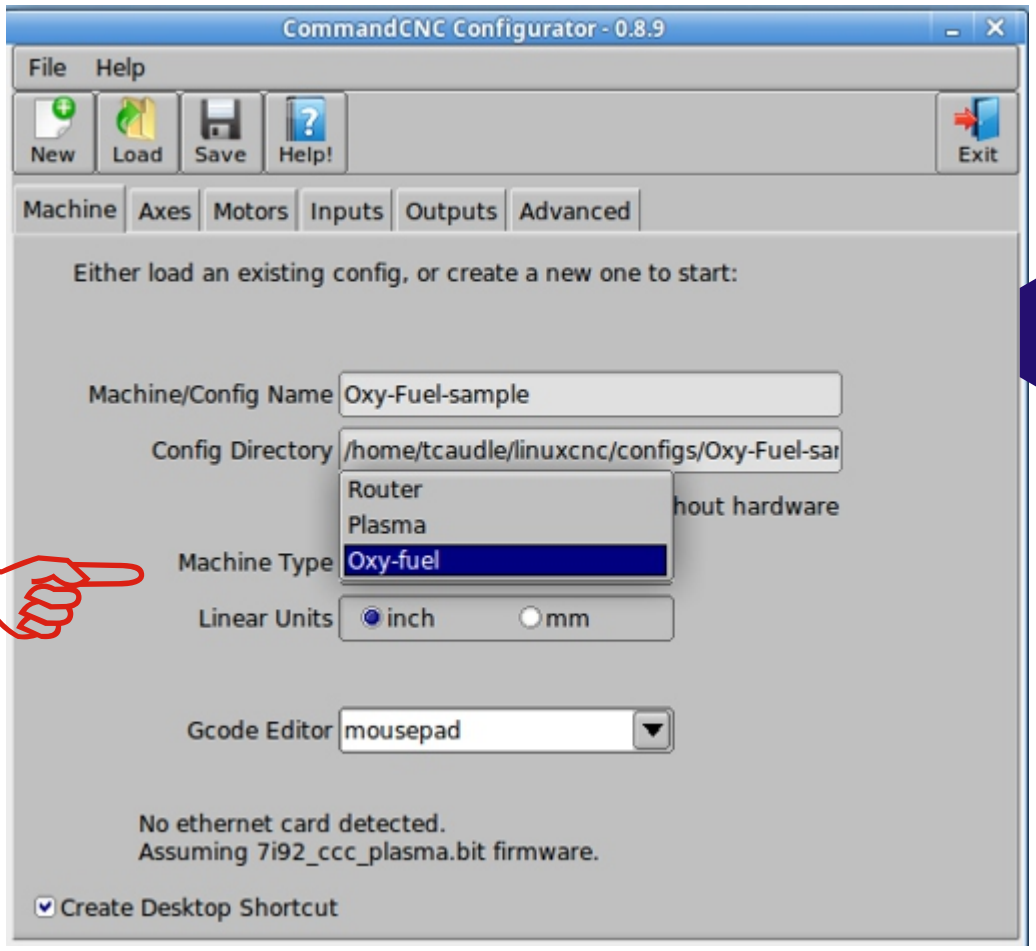
After you have cloned a Config , in the Machine Type select Oxy-Fuel
That should then display in the Machine Type box



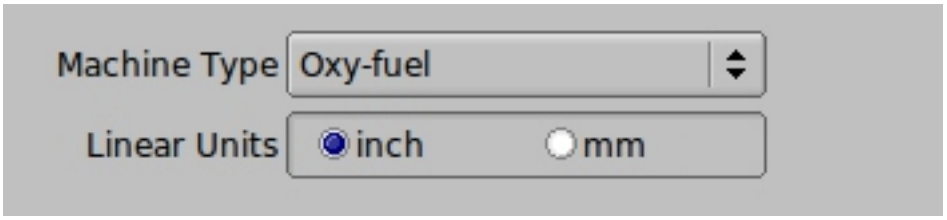
B



After you have cloned a Config , in the Machine Type select Oxy-Fuel
That should then display in the Machine Type box



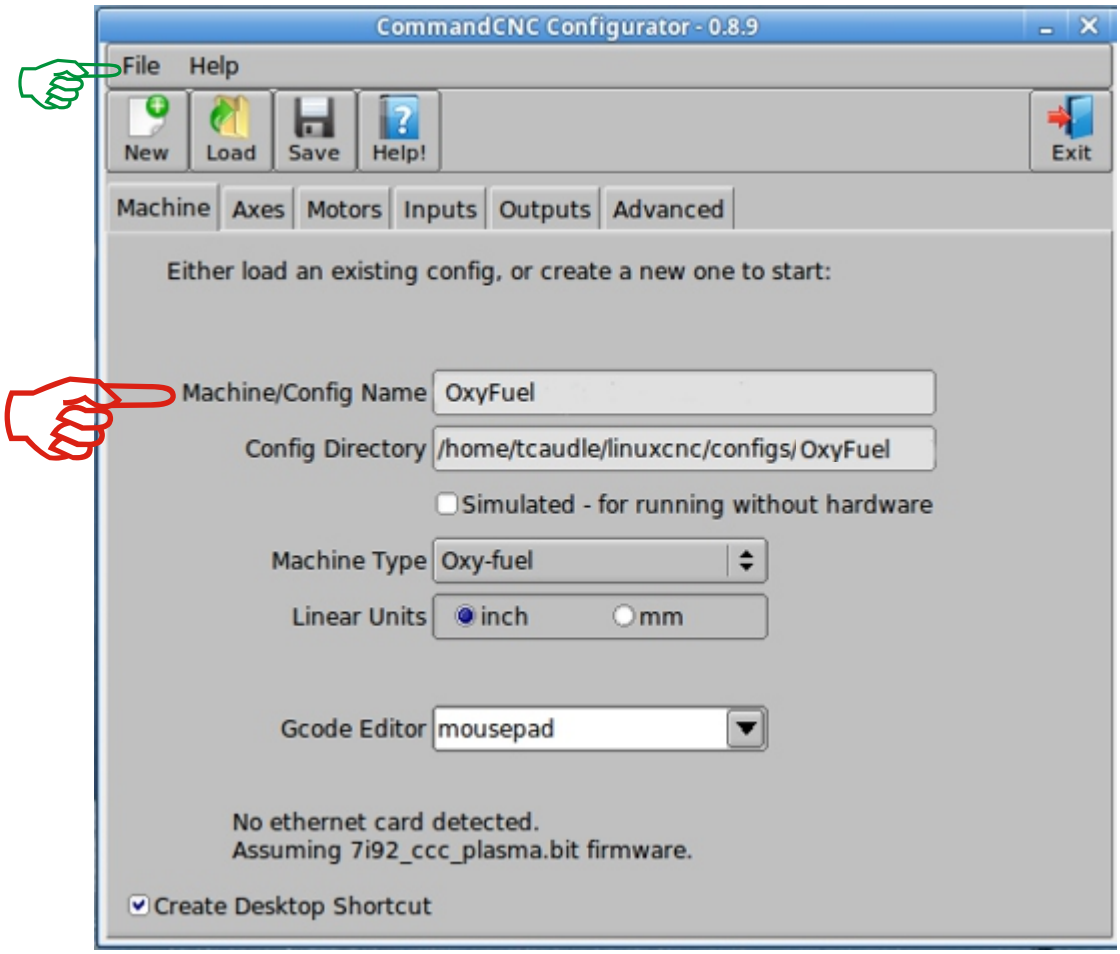
C



D

At this point you should SAVE the new Config and EXIT. You should have a new ICON on the desktop

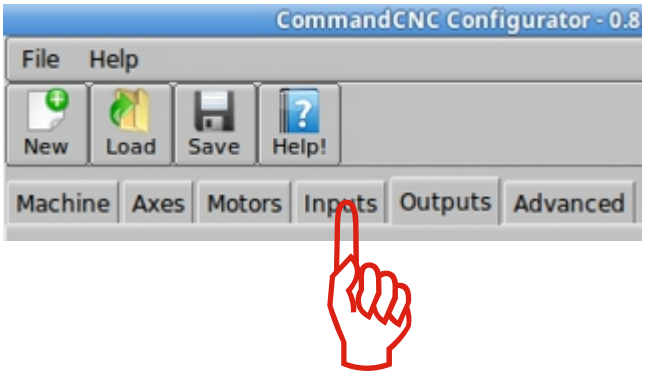
Using the Configurator, open the new Config you just made. The Configurator remembers the last Config you had open so you should see the new Config name you chose in the Machine/Config Name. If not use the File/Open from the top menu and Open the Config from the Popup window.



Oxy-fuel support

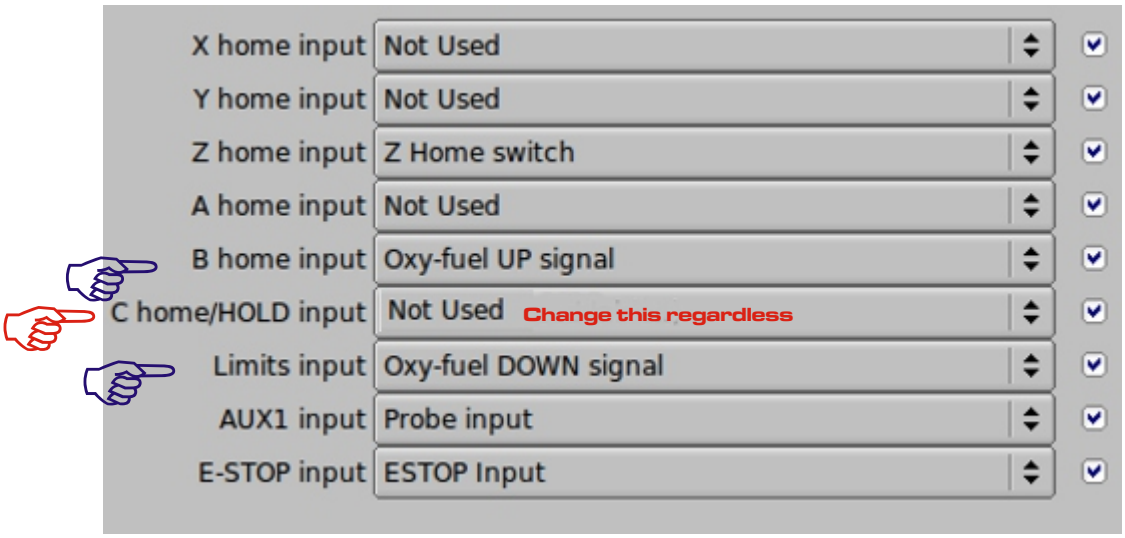
There is now a new machine type called Oxy-fuel. A configuration using this machine type allows you to manually adjust the Z height while cutting with an oxy-fuel torch. To move the Z, the "torch" output must be on. The "u" key will move Z up, and the "d" key will move it down.

Open the INPUTS Tab on the New Config



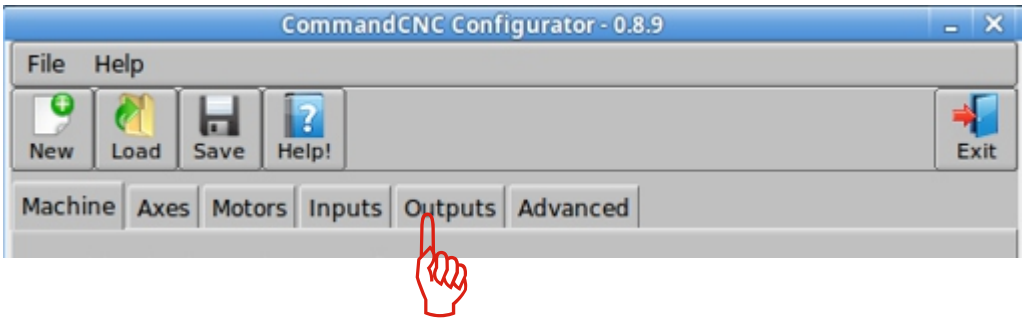
F

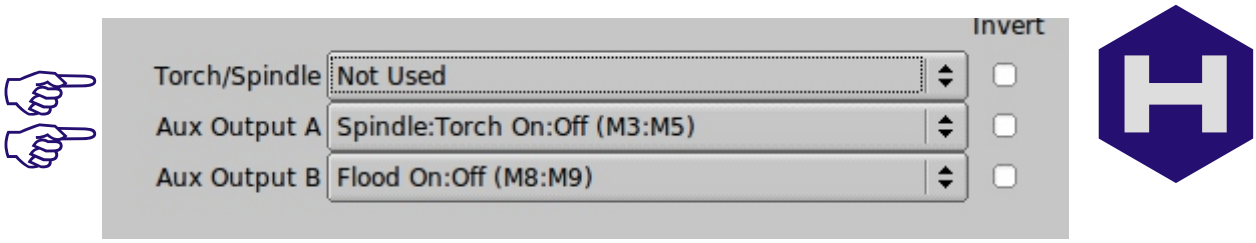
Change the following settings in the INPUTS



G

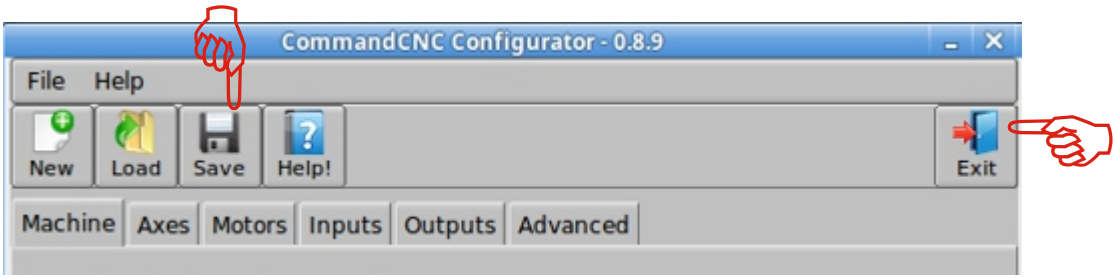
NOTE: These settings are to allow an external pair of switches (or one SPDT Center-Off switch) to be used to make a small "pendant" to control the Torch up and down away from the keyboard. Its recommended you make these changes even if you do not plan on having an external set of switches. In the example the Limits and BHome input are used, but any unused input will work.





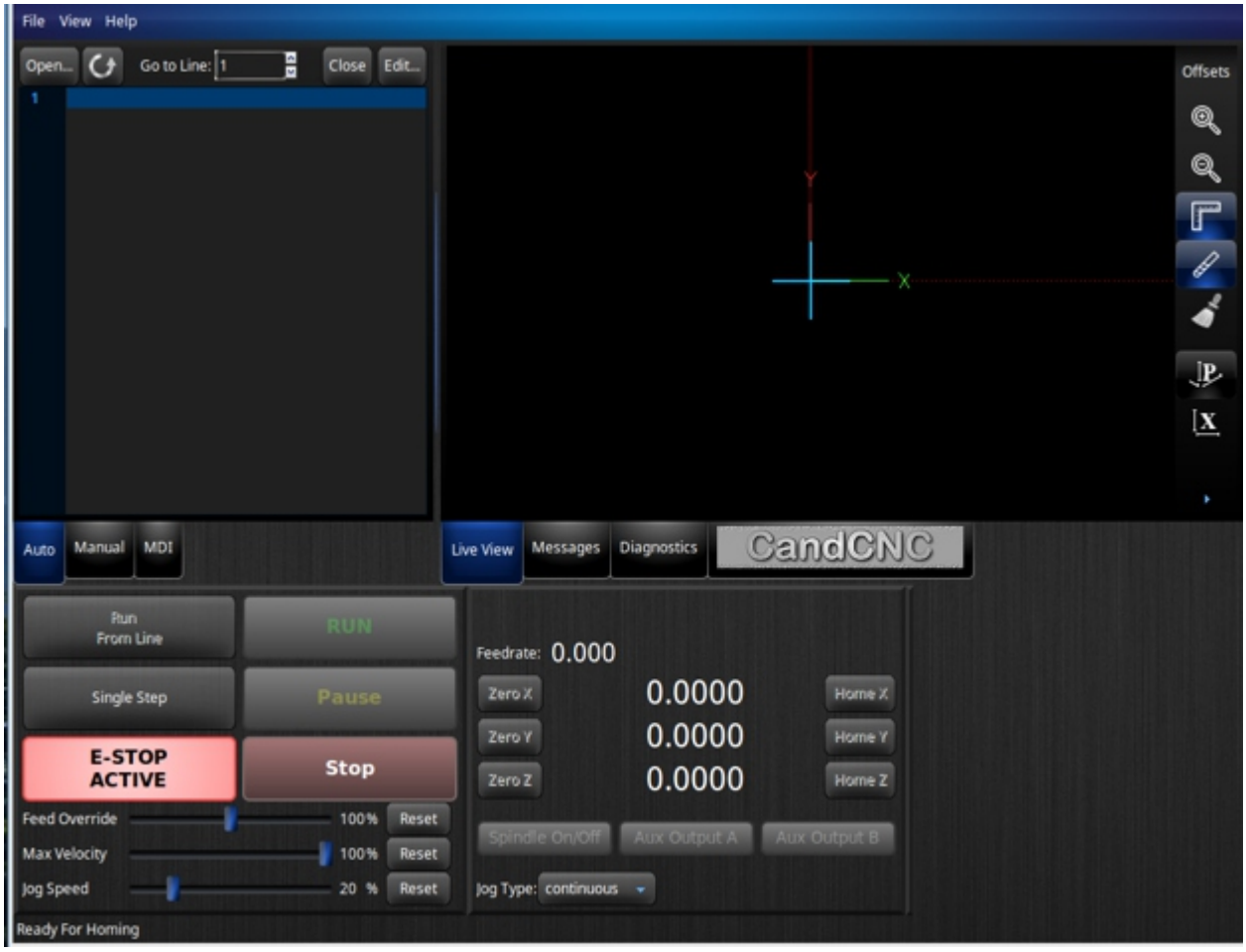
The full oxygen for cutting will typically be wired into Aux Output A or B – this picture shows a sample setup using A.

At this point you should SAVE the new Config and EXIT.



Oxy-fuel support

There is now a new start ICON called Oxy-fuel. A configuration using this machine type allows you to manually adjust the Z height while cutting with an oxy-fuel torch. **To move the Z, while running code, the "torch" output must be on. The "u" key will move Z up, and the "d" key will move it down.** To move the Z when the torch is off (jog Z) use the normal Page UP and Page Down keys



NOTE: It is beyond the scope of this Addendum to explain in detail how to setup and cut using an oxy-fuel torch or to wire in a solenoid/valve to let CommandCNC turn on the Oxygen Cut flow. The AUX outputs are connected to the AUX AC input so they switch 120VAC power to the Outputs. Any external valve or solenoid should be rated for 120VAC. An AC "snubber" should be used across the coil of either an external relay or solenoid to prevent arcing and kickback onto the AUX relays in the system.

END OF DOCUMENT