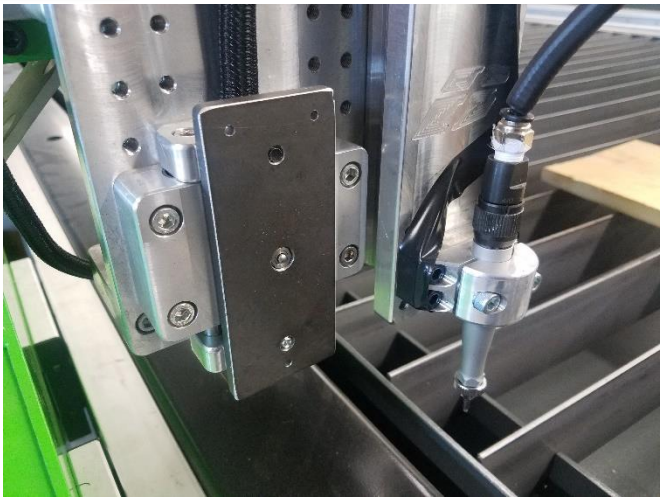
Quick Start Guide

563-380-1535 www.bosstables.com 1658 St Hwy 9 Decorah, IA 52101

1- ANY DAMAGE call immediately 563-380-1535
2- Unstrap computer stand and remove from bed. 2 people may be needed to lift and place on ground.
3- Place slats in bed, they are cut to length and can be placed into bed without a certain order. The slats are curved so that straight cuts will not fall directly onto the top of the slats.

4- When your slats need to be replaced the easiest way is to purchase 3” X 1/8 steel strap and cut to length.
5- Remove the torch from wrapping and place on magnet mount. The machine will not come out of E-Stop without the torch in place on magnetic mount. The Torch has an ohmic Clip installed, you will need to connect the black covered wire to the clip. To let the machine find the height of the steel.

6- There is a E Stop Switch located on the steel plate used for the magnetic breakaway, the switch has to be depressed to allow the machine to come out of E Stop. (You need to have the torch mounted to remove E-Stop.)



7- Ensure that the E-Stop buttons are not De-pressed as the machine will not come out of E-Stop. There are 2 located on gantry and 1 on the computer stand.

8- Level table. There are leveler feel located on the bottom of the legs. The table does not need to be perfectly level but the table will not drain properly if the drain is higher than the cutting area. The drain is located on the side of the controller. There is a black cover/ grate covering the “Drain”. You will not need to remove the cover for any reason unless parts or debris in interfering.

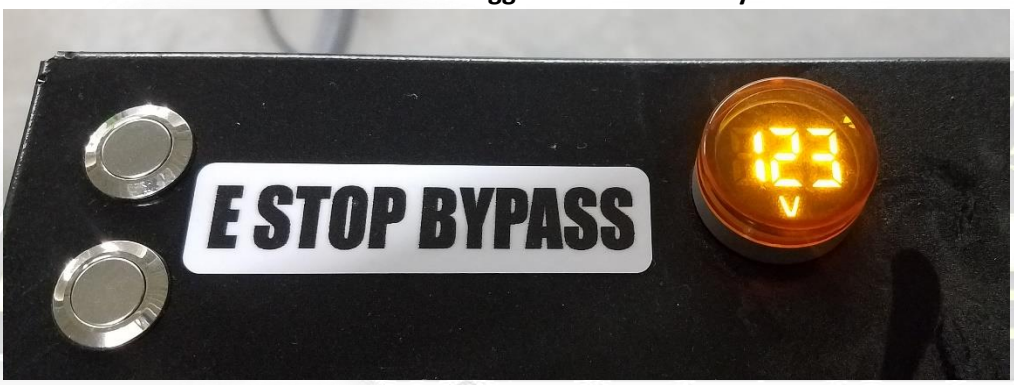
9- Close the valve located on the bottom of the table. **Very Important as you will dump cutting fluid on floor if valve is not closed.**

10- Dump the BOSS Tables cutting fluid into the table anywhere in the cutting area. Then begin filling with water. Use any water available if it is clean. The 3/4 valve located on the control end of the table is the water/air valve(Left in picture). The valve needs to be open to allow air to escape and water to fill the lower tank. The table is full when the lower tank is full. Water level can by observed with a flashlight shining through the grate. The water level will rise in the lower tank and is full when the water level meets the floor of the cutting area. (the bottom of the grate) If over filled it will not hurt the machine and you can just let the extra water evaporate or utilize the table drain located underneath the table. This will take some time as the table holds approx. 300 gallons.

11- Air pressure supplied to tank is applied from control end of the table. The smaller valve is used to supply air into the tank. The larger ¾ valve need to be closed to allow the air to push the water into the cutting area. If you table has a pneumatic scribe the small valve pointed down supplies air pressure to scribe. The air pressure will need to be supplied to hold the scribe up in the travel position.

12- Plug in computer. Computer is in a small box, the mount is on the right side of the computer stand. Slide the computer onto the mounting fixture and plug in the monitor, mouse, keyboard, white usb plug (from HUB), usb hub, and cat 5 cable green (from controller). The extra cord can be pushed into the hole that the white and green wires are ran through.
13- Mount monitor the mounting screws are located in the back of the monitor
14- Place key board and mouse on stand and place extra cord in hole next to computer mount.
15- The computer stand has a surge protector located in the control cabinet. But another level of protection at your breaker box is recommended.
16- If customer is supplying their own torch the torch cable will need to be fixtured in the cable track. Open the cable track and lay in torch lead. Ensure the torch lead has proper clearance for z axis travel. The torch should be approximately fixtured 4 inches from the bottom of the z axis. The torch can be adjusted as needed to cut larger items such as 4 inch pipe ect.
17- Ensure that the table is cleared off and all personnel is out of way.
18- Press power button located on computer. The light will illuminate blue when turned on.
19- Ensure that home screen is shown. From there you can explore the computer if wished. But the settings are not to be changed unless specified by BOSS Tables. Feel free in future to create folder and save DXF, job and other files on computer. But it is not advised to troll the internet for free DXF and or other personal needs. Think of it more as a controller than a computer.
20- The power button on the computer will not turn on the motor drivers and motors. The motor driver/power button is located on the right side of computer stand toward rear of cabinet. The power button can be locked out and tagged if needed to stop use.

21- When the button is depressed for motor power the voltage indicator / power indicator light will illuminate to show that power is supplied to motors. **If your going to be working around or on top of the table the power switch will need to be locked out and tagged to ensure safety.**



22- The motor and drive power will need to be powered on to supply power the MP 3700 located in the cabinet. The MP3700 is the large box located near top of fixture board. The MP 3700 is the controller.

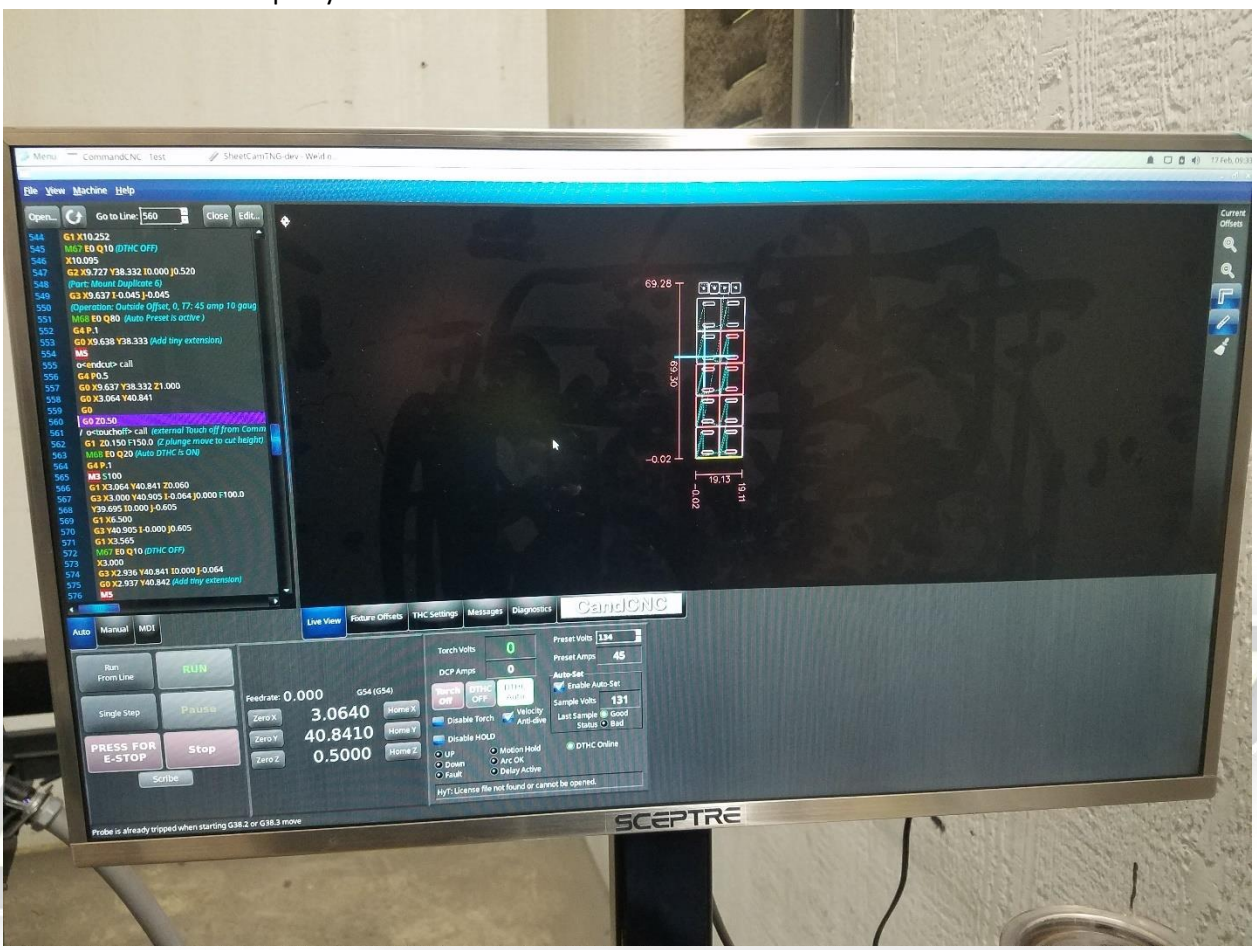
23- If the command CNC interface is opened without the power supplied to motors and controller you will receive a no communication error. If present, just close the dialog box. Turn on power to the motor and controller. Wait 5 seconds and re open command cnc interface(new or rusty steel).

24- Command CNC Interface is labeled similar to “New Steel” and or “Rusty steel”. They are located on the desktop or home screen.

25- “New steel” is used for steel that is in good condition and will utilize the “feather touch” operation. “Rusty steel” is used for material that is rusty, coated, painted, or has some sort of inhibitor on surface. The feather touch works very similar to an ohm meter. Once the feather touch senses the top of the material the torch

home is set. If the rusty steel cutting interface is opened the torch will use the microswitch and offset the travel distance of the torch movement to operate the switch.

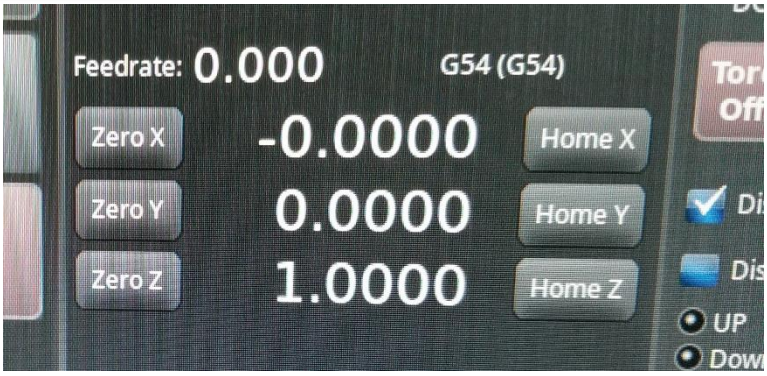
26- Once the interface is open you will notice lots of buttons that will be talked about later.



27- Your machine will open with the E-Stop button in lower left illuminated. You will need to “lift” E-Stop to operate the table. Click on the e stop button to remove E stop. If the button does not change check for proper torch placement and that none of the E Stop buttons are depressed.

28- You can now jog your table around using the Arrow keys for left, right, up, and down. Use the PgUp and PgDn located above the arrow keys to travel the Z axis up and down.

29- **You need to home the machine.** To home the machine jog the table to the lower left side of the table. Do not ram the gantry into the stops. Stop 4 inches before the X and Y limit switch. Then select the Home X and wait till it touches the home switch. Then Select Home Y and wait for it to touch off the Home switch. After the gantry has been homed with x and y axis a symbol will appear to let you know that your machine has been homed.



30- The Z axis will be homed when the material loaded is to be cut.

31- Granted that you followed the steps your machine is now set up and ready to accept Code to cut steel.

32- If you know how to use sheet cam you can skip the sheetcam portion. But if your not familiar with sheet cam please review sheetcam documents and watch videos located at www.bosstables.com

33- This is very important for cut quality and operating your plasma.

- You can change the amperage of your power unit, this allows you to operate at a lower amperage and cut thinner material. More amps for thicker material.
- The tools are pre-loaded for Hypertherm in sheetcam. Pick appropriate tool in sheet cam and match the consumable tip size and amperage on power unit.
- Sheetcam tool/ consumable tip/ setting on power unit must match.

Hypertherm®

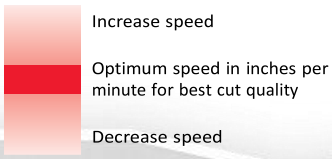
Powermax® machine-side reference guide

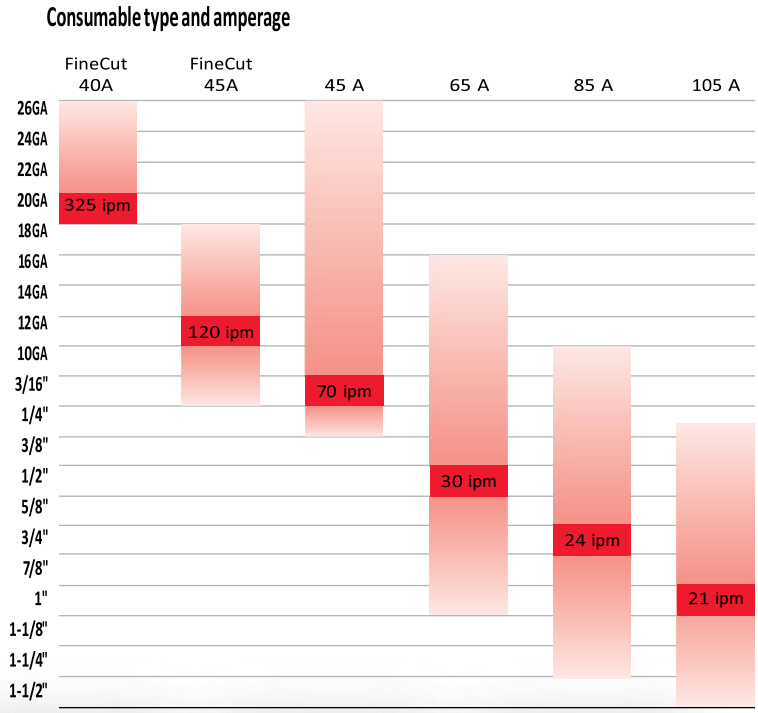
For mechanized applications with Powermax65/85/105 systems

Step 1

Select appropriate consumables and amperage

- Once you have determined the thickness of the metal to be cut, use the chart to determine the appropriate consumable type and amperage setting for optimum cutting. The chart provides general thickness ranges for cutting of mild steel. Refer to your Operator Manual for detailed speed and thickness measurements.





Step 2

Install consumables

Mechanized torch consumables

This Powermax machine-side reference guide is a supplement to your Operator Manual and includes examples of edge cut quality and consumable wear. Always refer to your Operator Manual for detailed safety and operating instructions.

- Use the chart on the right to install the appropriate consumables. Make sure the power is OFF before installing and changing consumables.

Mechanized consumables – engineered for the most productive mechanized cutting

Gouging consumables – designed for your toughest metal-removal jobs

CopperPlus™ consumables – an optional long-life electrode when cutting metal 1/2-inch thick or less **FineCut® consumables** – optimized for high-quality cuts on thin metal – for a clean edge and a narrow kerf **Unshielded consumables** – ideal for cutting in hard-to-reach areas and for the best arc visibility

		Shield/ deflector	Retaining cap	Nozzle	Electrode	Swirl ring
Mechanized	105 A 85 A 65 A 45 A	220993 220817	220854	220990 220816 220819 220941	220842 or 220777	220994 220857
Mechanized, ohmic	105 A ohmic 85 A ohmic 65 A ohmic 45 A ohmic	220993 220817	220953	220990 220816 220819 220941	220842	220994 220857
Unshielded	105 A 85 A 65 A 45 A	220955	220854	220990 220816 220819 220941	220842	220994 220857
Gouging	105 A 85 A 65 A	220798	220854	220991 220797	220842	220994 220857
FineCut	45 A 45 A ohmic	220955 220948	220854 220953	220930	220842	220947

Powermax105 – Use consumables up to 105 A

Powermax85 – Use consumables up to 85 A Powermax65 – Use consumables up to 65 A

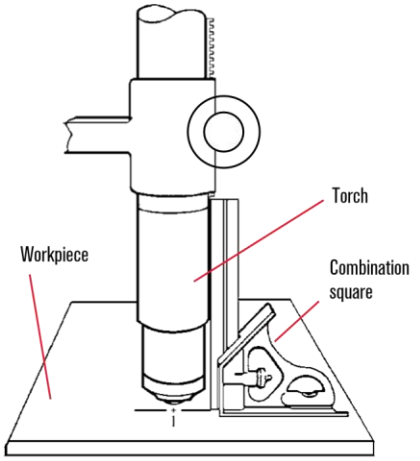
BOSS TABLES.COM

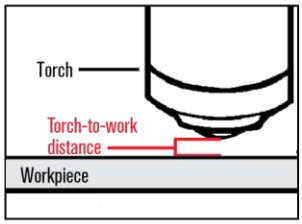
Note: For customers using the serial interface, steps 4 through 7

Step 3

Verify that the torch is square and adjust torch-to-work distance

- Set up your torch so that it is perpendicular to the workpiece, in order to achieve a square, vertical cut. Use a combination square to ensure the torch is square from the front and side of the torch.
- Set the proper torch-to-work distance. Use the diagram on the right as a reference.
  - Proper torch-to-work distance is very important for the plasma cutting process. Always refer to your operator manual cut charts to determine the proper cutting and piercing heights.

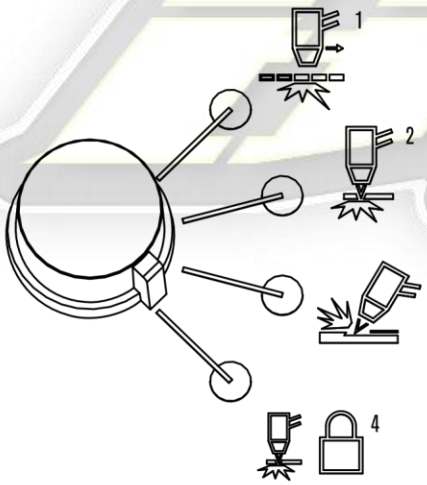




Step 4

Set the mode

With CNC controls enabled, some settings, such as gas pressure, may be disabled at the power supply.



1. Continuous pilot arc

Expanded/punched metal

Mode switch

2. Non-continuous pilot arc

Plate/sheet metal

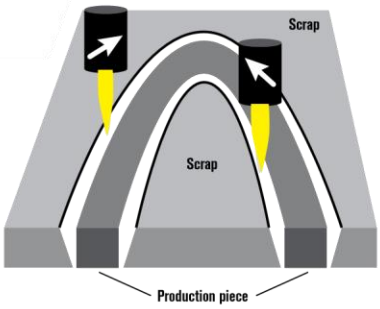
Mode switch

3. Gouging

Mode switch

Note: Verify correct torch direction when cutting plate/sheet metal.

Due to the swirling action of the plasma gas, one side of the cut will always have more bevel angle. This is called the “scrap side” of the cut. The “good side” is on the right as the torch is traveling away from you. Refer to the picture on the right.



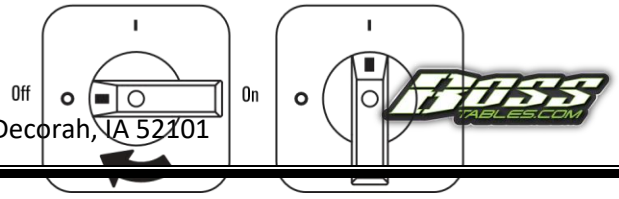
Step 5

Turn on the power

- Position the power switch to ON as shown.

Note: The ohmic 15-35 automatic and will only operate when needed.

- The power switch is found on the back of the system.



may be done at the CNC (Computer Numerical Control) rather than the power supply.

Non-continuous pilot arc Torch lock* **Mode switch**

Refer to your Operator Manual for instructions

***Not intended for Mechanized cutting applications**

Step 6

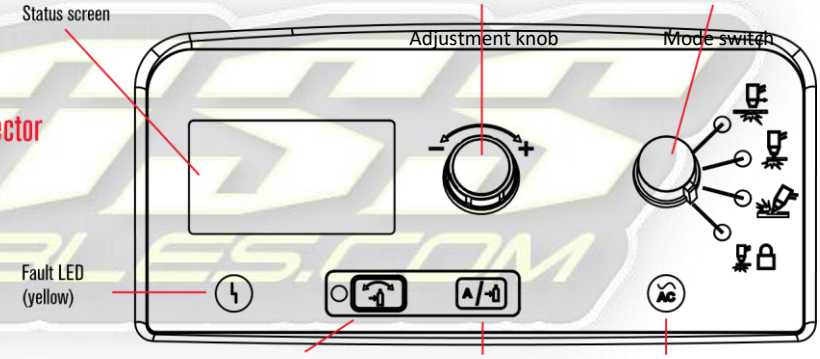
Set your amperage

Automatic/manual pressure setting mode selector

The selector switches between automatic and manual mode. In automatic mode, the power supply automatically sets the gas pressure based upon the torch type and lead length and the adjustment knob sets only the amperage. In manual mode, the adjustment knob sets either the gas pressure or the amperage. This LED is illuminated in manual mode.

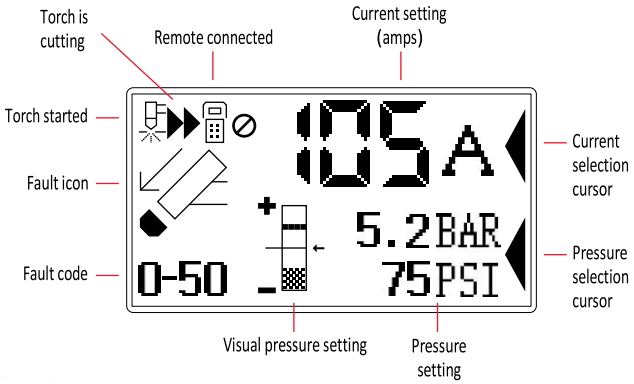
Note: Manual mode should be used by experienced users who need to optimize the gas setting (override the automatic gas setting) for a specific cutting application.

See Section 4 in your Operator Manual for adjusting your system in manual mode.



Step 7

Check your status screen to ensure that there are no fault codes to troubleshoot



Automatic/manual pressure setting Current/gas Power ON LED
mode selector selector (green)

563-380-1535 www.bosstables.com 1658 St Hwy 9 Decorah, IA 52101

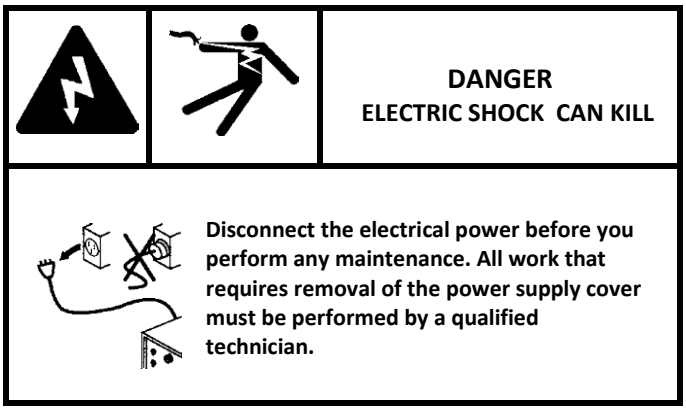
Warning/Fault codes (refer to operator manual)	
0-12	Low input gas pressure: warning
0-13	AC input unstable: warning
0-19	Power board hardware protection
0-20	Low gas pressure
0-21	Gas flow lost while cutting
0-22	No gas input
0-30	Torch consumables stuck
0-32	End of consumable life
0-40	Over temperature
0-50	Retaining cap off
0-51	Start/trigger signal on at power up
0-52	Torch not connected
0-60	AC input voltage error
0-61	AC input unstable: shutdown
0-98	Internal communication failure
0-99	System hardware fault – service required

See Section 4 in your Operator Manual for a full list of fault icons and Section 5 for basic troubleshooting.

Step 8

Begin cutting

Maintenance schedule



Every use:

Inspect the consumables for proper installation and wear.

Check indicator lights and fault icons. Correct any fault conditions.

Every 3 months:

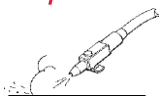
Replace any damaged labels.

Inspect the trigger for damage. Inspect the torch body for cracks and exposed wires. Replace any damaged parts.

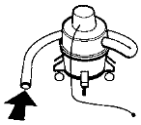
Inspect the power cord and plug. Replace if damaged.

Inspect the torch lead. Replace if damaged.

Every 6 months:



Or



Clean the inside of the power supply with compressed air or a vacuum.

Hypertherm, Powermax, and FineCut are trademarks of Hypertherm Inc. and may be registered in the United States and/or other countries.

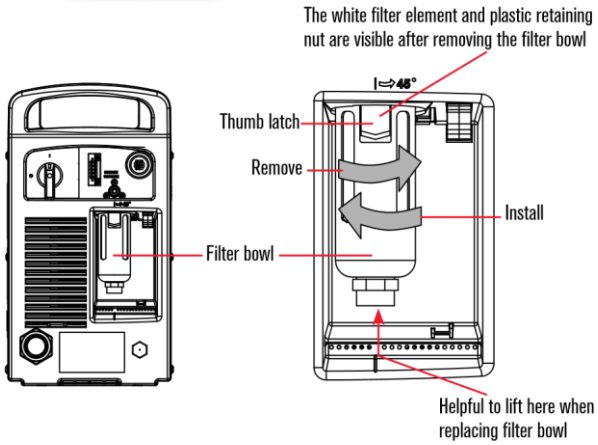
Environmental stewardship is one of Hypertherm's core values, and it is critical to our success and our customers' success. We are striving to reduce the environmental impact of everything we do. For more information: www. hypertherm.com/environment.

©1/2018 Hypertherm Inc. Revision 2

894380

Replace the gas filter element

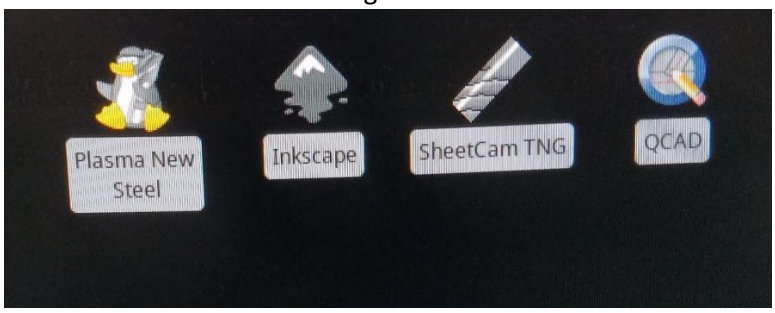
1. Turn OFF the power, disconnect the power cord, and make sure the gas supply is disconnected.
2. Position the rear of the power supply so the removable gas filter bowl is easily accessible.
3. Grasp the filter bowl with your right hand.
4. Push down the thumb latch and rotate the filter bowl approximately 45 degrees to the right.
5. Pull the filter bowl straight down to remove. You can see the white filter element and retaining nut.
6. Unscrew (counterclockwise) the plastic retaining nut that secures the filter element.
7. Replace the dirty element with a new element (part number 011092. Reinstall (clockwise) the plastic retaining nut to finger-tight only.
8. Insert the filter bowl with the thumb latch positioned approximately 45 degrees to the right of center. This is the same orientation in which the filter bowl was pulled down and removed.
9. Vertically align the filter bowl (with metal guard) and firmly push the filter bowl up to the top of the receptacle to seat the bowl. It is helpful to lift the bowl with your left index finger under the nut on the bottom of the bowl.
10. Once the bowl is seated properly, rotate the bowl 45 degrees to the left until you hear the thumb latch click into place.
11. R econnect the gas supply hose to the power supply and check for leaks.
12. R econnect the electrical power and turn ON the power switch.


The white filter element and plastic retaining nut are visible after removing the filter bowl
Thumb latch
Remove
Install
Filter bowl
Helpful to lift here when replacing filter bowl

WARNING

Sheet Cam Quick Manual (Videos are on website www.bosstables.com)

1. Sheetcam is the drill bit looking icon on home screen. Double click to open.

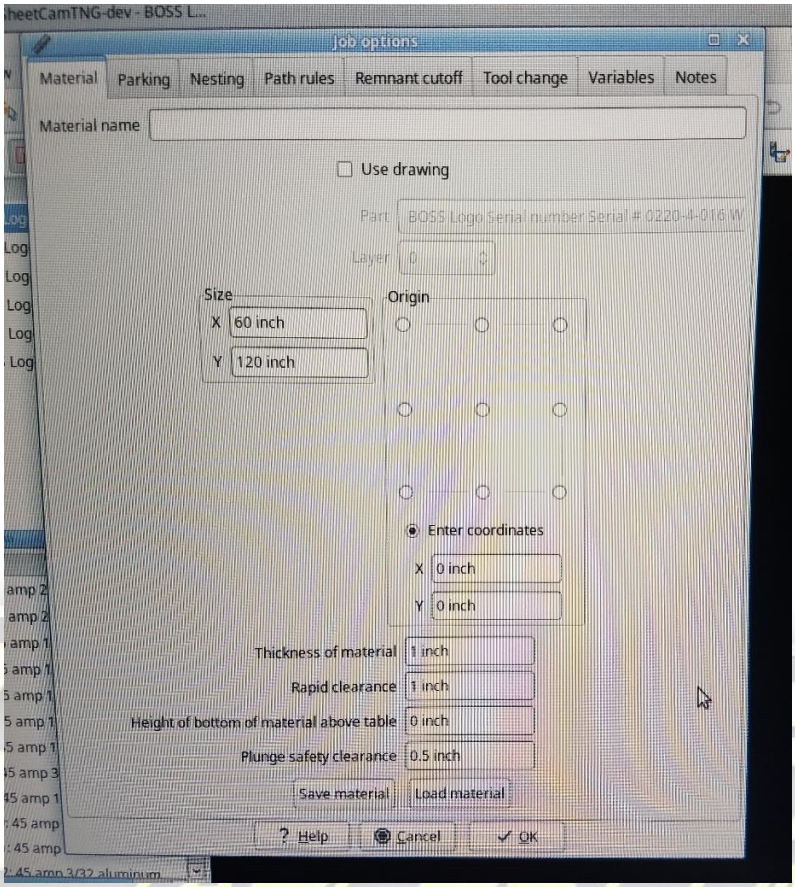


2. Go to Options located at top of screen and select Job Options. In the material tab you will select the size of material your using. EX- 60” in X / 120” in Y box.

The box 9 dots will create the 0 X 0 Y origin Lower left is the most common selection.

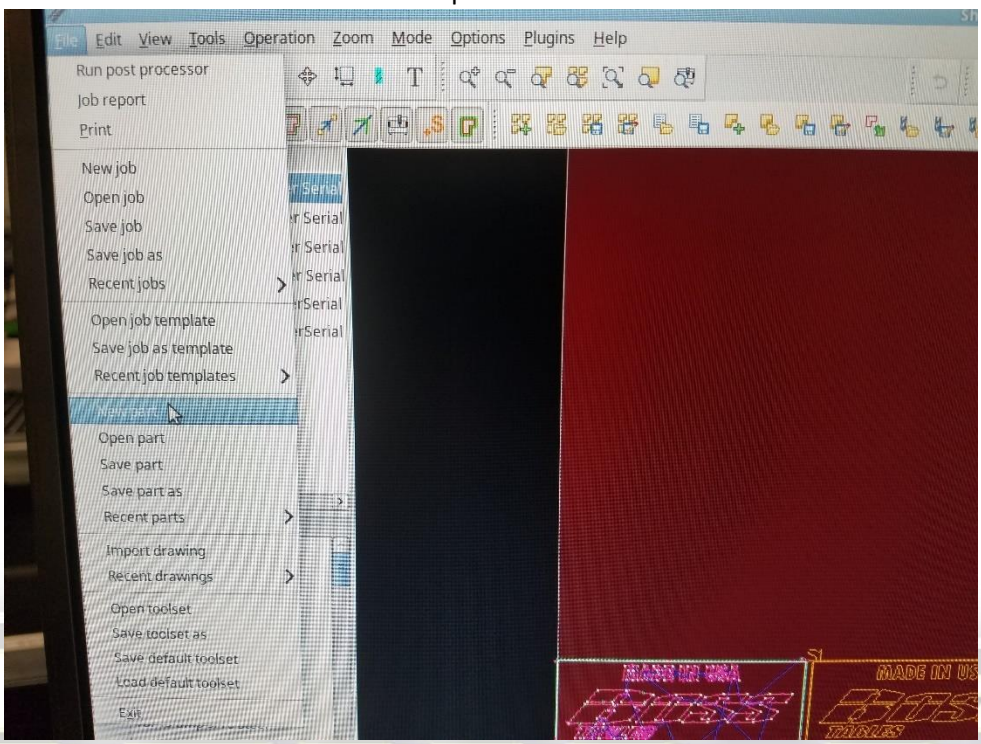
- You can select other options such as thickness of material (not used for plasma).
- Rapid clearance is the travel height of torch 1” is a good number
- Height bottom material is not used

- Plunge safety clearance is the distance above last touch off that it will slow down and touch off. .25-.5”

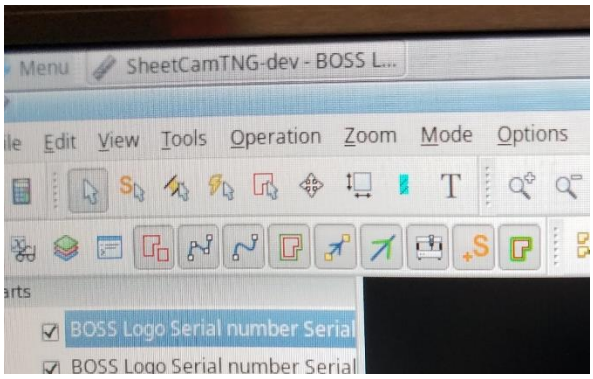


3. The size of the red area will change to the selected size in the Job options dialog box. This is you material area that you can nest parts on. You do not need to set this every time but if nesting a complex job it’s a very valuable thing. Click ok and you will return to main screen.

4. To Import a part Part Click- File – New Part- then find your dxf that your cutting out. Once selected double click or select open.

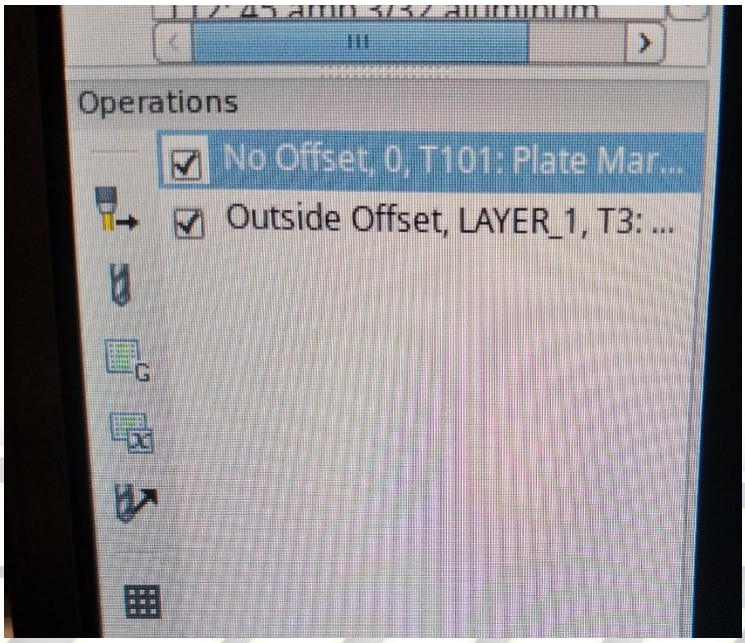


5. A box will open you will need to select Inch if not highlighted already by black dot. The dots on the drawing position will allow you to select where on the job you would like to drop your part. Lower felt will import to the lower left ect…. Points for drilling may be checked if needed. Click ok and your part will be imported into sheetcam.
6. From there you can continue to bring more parts in as needed. You can also right click on the Parts listing box in upper left and select New part for a faster import. Sheetcam will remember you last selection for drawing imports and remain the same until they are changed.

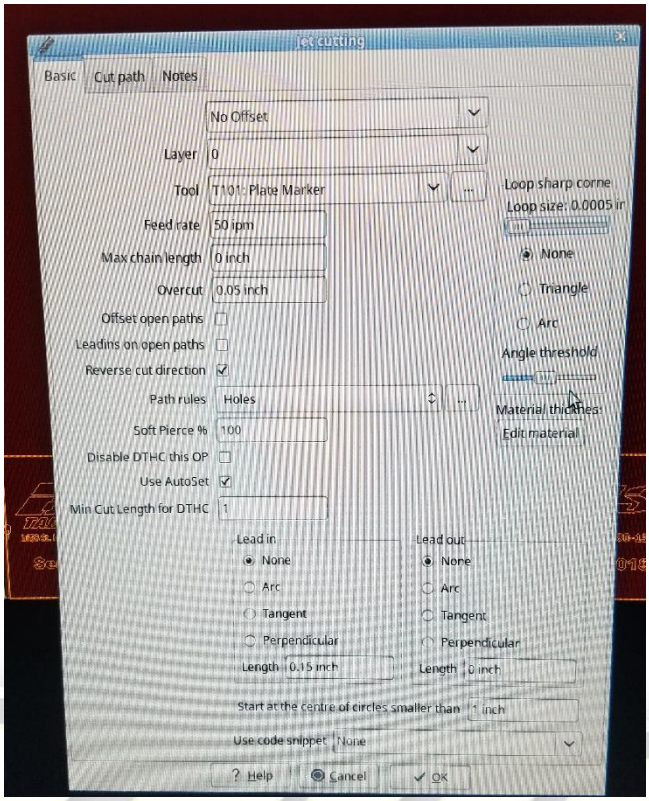


7. Notice the Tool bar located at the top of the screen.
    - Curser symbol is what you will use when creating tool paths and selecting things (if you clicking on things an nothing happens select the Cursor symbol)
    - S will change the start location of SELECTED PART
    - Tabs Will not use
    - Action will not use
    - Red Box is Edit contours. For changing the offset and or creating different tool paths for peck pierce, scribe operation.
    - 4 arrow button is for nesting parts on you material. (Needs to be selected to nest parts.
    - Drill bit icon is for a simulation of the tool paths created.
8. Right click on middle of screen to find the measure tool (Super handy)
9. Rotate the DXF files with the < and > symbols. Hold to spin in circle.
10. Mirror parts in the bottom right with Mirror X and Y check boxed.
11. Right click and select duplicate or multiple duplicate to make more of same part. Select copy and you will need to create a new toolpath for the part. Array parts will allow you to create rows and columns with amount you choose of each. Part spacing can be adjusted to needed size. (Be careful of lead in and lead out as they might land on top of part.) Change stat position by selecting S on tool bar and correct tool path in lower left if more than one toolpaths. And click on part to change start path.
12. If nesting a thousand parts at once give sheetcam a chance to keep up with you. It may be advisable if creating a complex job to save your progress as to not loose your Job. To do so Select File-Save Job. Name accordingly and save in a good location.
13. Creating tool paths can be done before or after nesting parts.

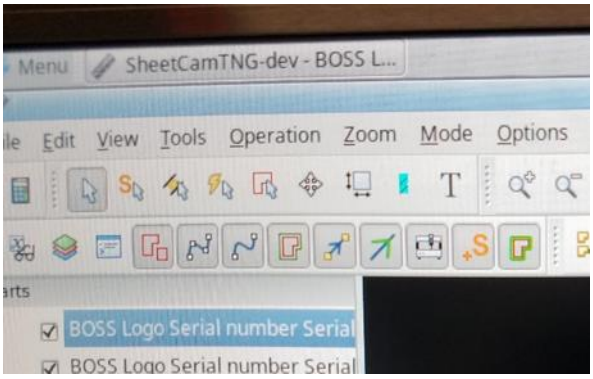
14. To create your tool path highlight the part in the upper left menu. Then select the torch icon on the lower left in the operations menu. The tip icon with an yellow arc. (You will not have operations as shown until they are created, that is what we are working on now). **The tools section located in the middle left side is not how you create and operation or cut anything. It is there as a reference for tool speed, feed ect…..**



15. From there you will be brought to a dialog box where you will select the operations and steel thickness you desire.



16. On the jet cutting page at the top is your offset settings. Outside offset will cut on the inside of the interior features and the outside of the outer features. Inside offset is the opposite and no offset will cut directly on the lines. No offset is very handy if the part drawn by customer or operator is too small. Move the holes that are too small to a new layer and generate toolpath with no offset. But if done second you will need to drag the operation to the top of the part operations. The idea is to cut or scribe the inside features first, then cut the outside of the part as to fixture the part with the larger plate that it is being cut from.
17. Layer is the features that you have selected. To create a new select the layer tool red box with cursor and click to select/drag a box over features/hold Ctrl and select more than one.



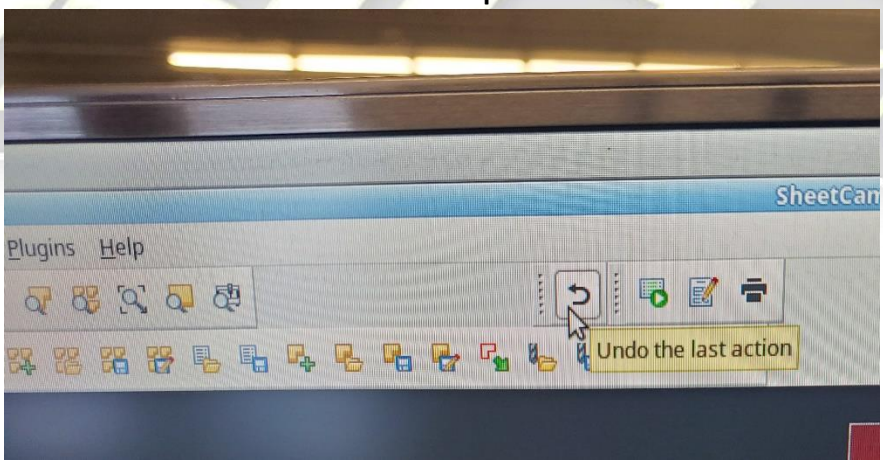
18. Select the features you want to change and right click and select move to new layer. Rename layer accordingly to you needs Ex. Drill/no offset/ or dead layer.
19. Selected layers can also be deleted if needed right click and select delete.

20. Back in the jet cutting operation you will need to select you tool. Your tool is chosen off what size of tip you would like to run and the size of steel your cutting. TIP! the slower you cut and the smaller your tip size is the more defined your parts will be. The larger you tip size the faster you can cut allowing you more production. **Please reference your owner's manual of your plasma for the proper tip tooling and cutting parts. VERY IMPORTANT TO HAVE THE CORRECT PARTS INSTALLED IN TIP TO MAINTAIN CUT QUALITY.** Select the proper tip size which is also in correlation with you amperage. Example if you using a 65 amp tip you need you plasma cutter set to 65 amps. Failure to do so will result in consumable failure or poor cuts.
21. Your feeds, speed, pierce delay, torch volts, ect will all be adjusted automatically. It can be reviewed by clicking on the dots(…) located on the right of the tool selection. Changes can be made accordingly if needed and the default tool set can always be uploaded again.
22. Max chain length will cut from part to part without piercing (not very often used)
23. Offset open paths and lead in on open path can be check depending on needs.
24. Overcut is the distance past the start of the cut you would like to troch to travel while still cutting.
25. REVERSE CUT DIRECTON IS ALWAYS CHECKED this is done to maintain cut quality because the air out of the torch is swirled in a tornado like motion. In doing so it will held the arc maintain a straight kerf on inside and outside features.
26. Path rules can always be left as HOLES.
    - Holes sets the feed to a set percentage (50%) or choice of user from the chosen tool path. EX. 45 amp 3/16 steel cuts at 63 IPM. When the holes rule is applied the cut in the hole will travel at 31.5 IPM on the inside of the hole. This will greatly improve the kerf (taper) quality of you holes.
    - When the holes rule is selected the ECO Anti dive is applied as well when selected (defaulted)
    - ECO Anti Dive turns the DTHC off in corner to prevent the torch from diving. No matter how fast your table is you must slow down for corners.
27. Soft pierce is used with an HYT Kit (option used for thicker steel mainly)
28. Disable DTHC will turn your Digital Torch Height Control off
29. AutoSet will automatically adjust you cutting volts in the cut interface. leave check as its A nice feature and can be disabled later if needed. (explained later)
30. Min cut length for DTHC should be left at 1" for most operations unless a custom tool is needed or very thick steel is being cut.
31. Lead in and Lead out will allow you to pierce the material outside of the part with the selected distance. Lead In will pierce and travel toward feature as selected with choices of None, Arc, Tangent, and perpendicular. Arc is the most common choice. Lead out will do the same but finish the cut outside away from feature. Your lead out generally can be very small or not used at all.

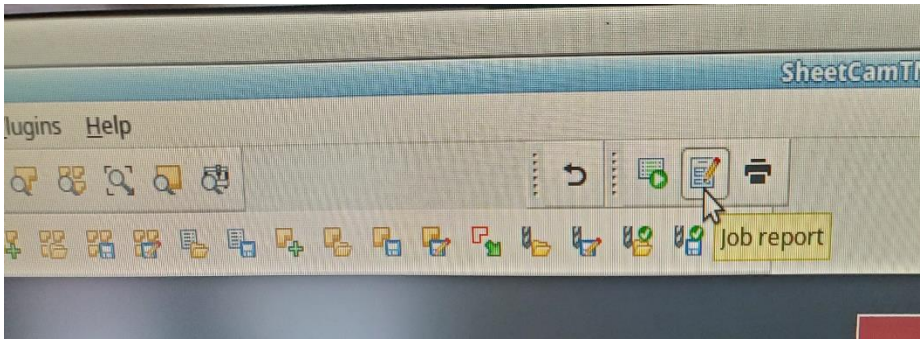
32. Use code snippet is seldom used as the HOLES and Anti Dive are very effective
33. Select OK and you will see the tool path generated.
34. Follow step 14-33 to generate tool path for next part/s that being said once you have the correct tip/amp and size of steel selected you can just hit ok and sheetcam will remember the last operation set in Jet cutting making the process very fast.
    - Select part
    - Gernerate operation needed (generally same as last)
    - Hit ok and select another part.
35. You have you tools paths created and your part nested (lower Left as that's your 0-X 0-Y location) -Start with something simple please-
36. From here you need to create your G Code Click in upper right corner File- Run Post Processor. This is what generates that fancy looking code you thankful you don have to right by hand anymore.
37. Create a file name that is associated with you part and save it in a location that you know the location of and will be able to find again. Do not be afraid to create new files and save accordingly. (generally saved in documents or desktop)
38. Moving on to the Cut interface as your now a pro at sheetcam -Congrats-

More on SheetCam

1. **There is a Back button located at the top of the screen**

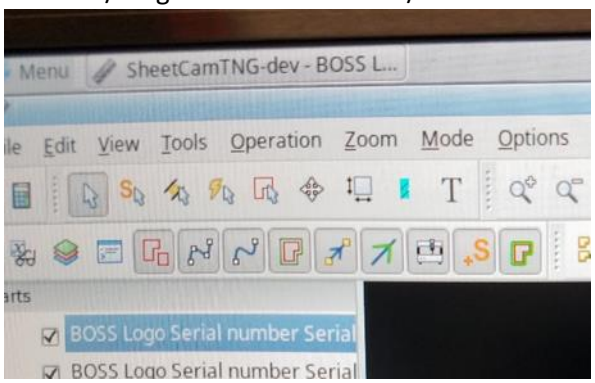


2. **There is a Job report that can be used for more information**

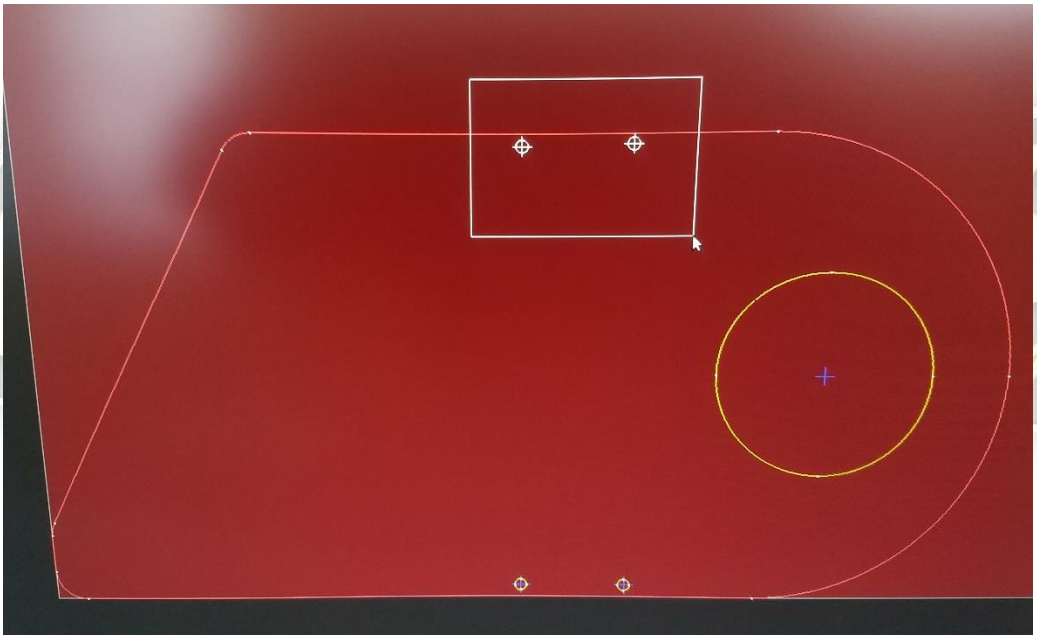


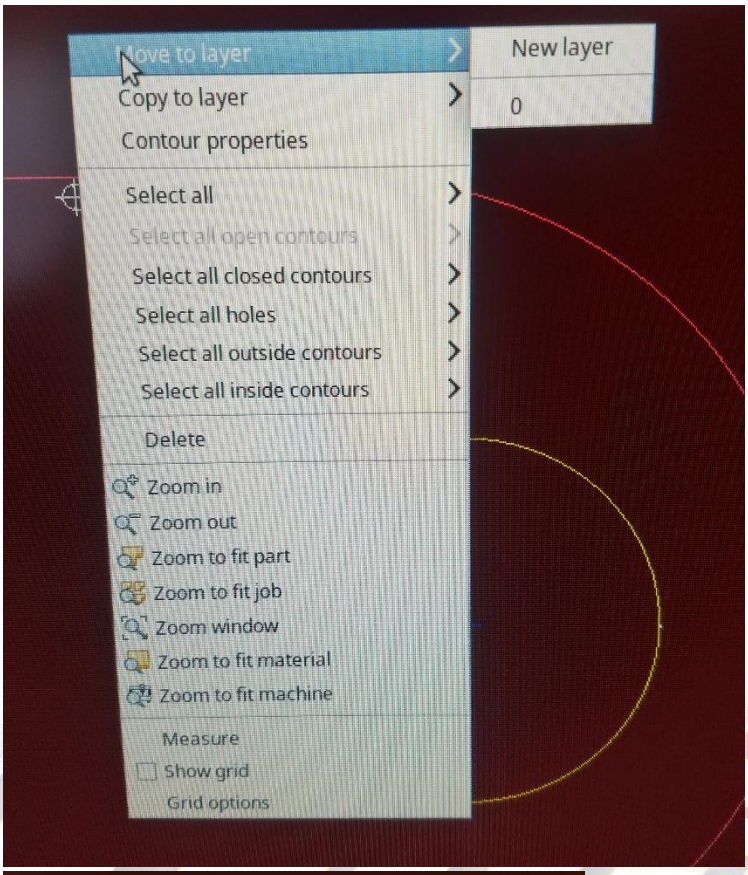
3. How to Utilize the Peck Peirce

- Layer is the features that you have selected. To create a new select the layer tool red box with cursor and click to select/drag a box over features/hold Ctrl and select more than one.



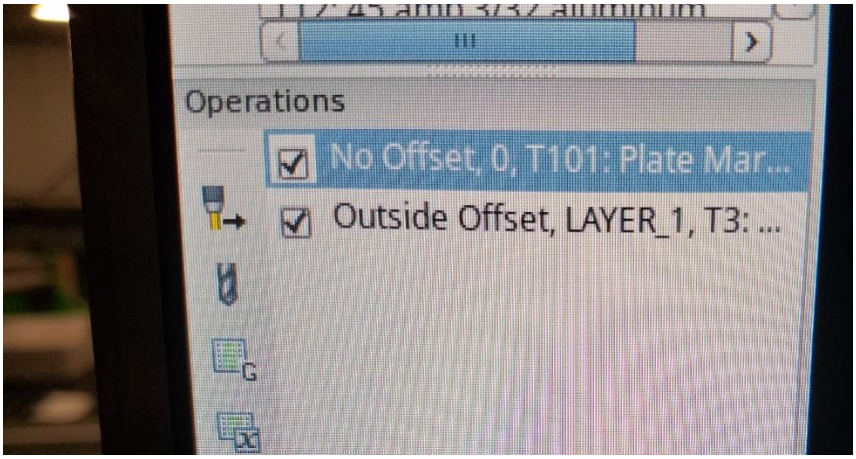
- Select the features you want to change and right click and select move to new layer. Rename layer accordingly to you needs Ex. Drill/no offset/ or dead layer.
- Selected layers can also be deleted if needed right click and select delete.



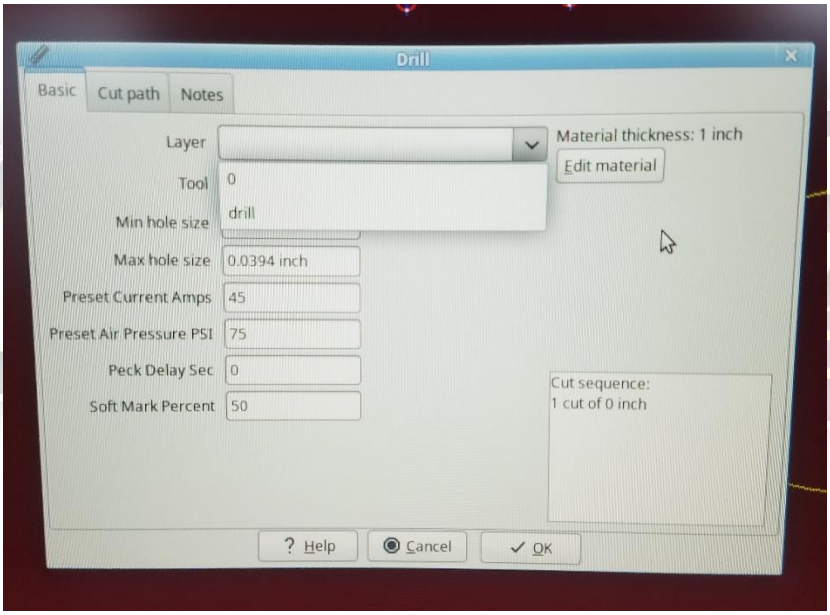


- You have Created your new layer. From here you can either not cut the layer, use the just cutting operation and change the offset, or use for peck pierce locating

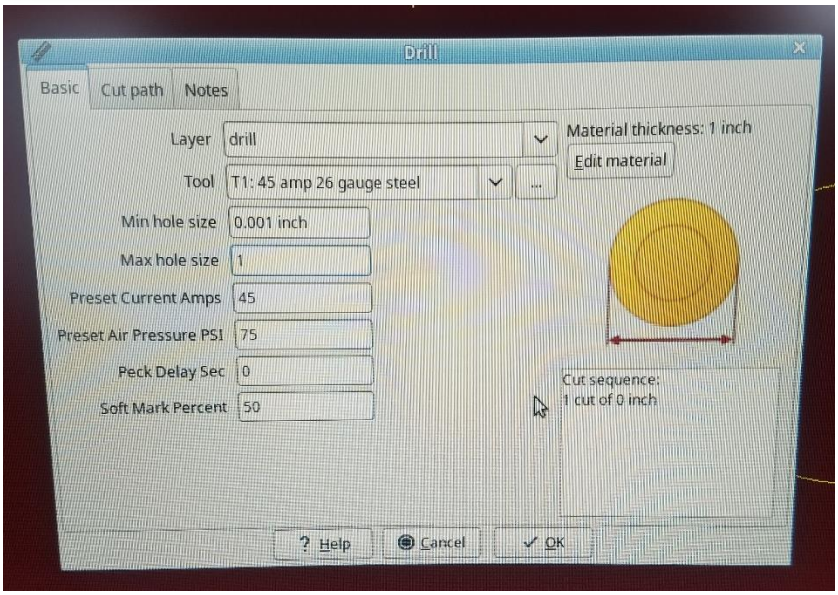
- Bottom left on sheet cam there is a drill bit logo select the drill bit located just under the torch or jet cutting operation button.



- You will now be brought to a screen here you can select your layer, Use the layer you have created or made to locate your drill locations.



- What I do to reduce the size of the hole is always operate the peck pierce with T1

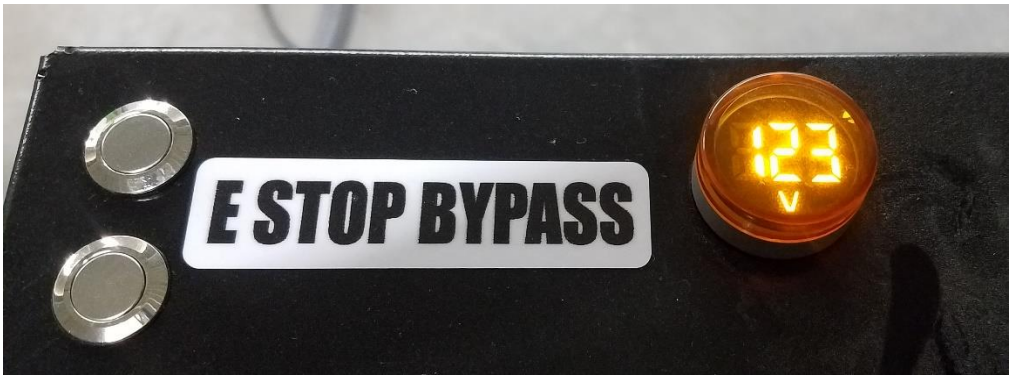


- Ensure that the Min and Max Hole size will fit the appropriate selection. NOTE you can only locate and peck holes. If you have a slot sheet cam will not readily find the center.
- Hit ok and notice the S1,S2,S3, that represent you starting points. They will just be a cross hair as there is no tool path.

Starting with Candcnc

1- **Ensure that the table is cleared off and all personnel is out of way.**
2- Press power button located on computer. The light will illuminate blue when turned on.
3- Ensure that home screen is shown. From there you can explore the computer if wished. But the settings are not to be changed unless specified by BOSS Tables. Feel free in future to create folder and save DXF, job and other files on computer. But it is not advised to troll the internet for free DXF and or other personal needs. Think of it more as a controller than a computer.
4- The power button on the computer will not turn on the motor drivers and motors. The motor driver/power button is located on the right side of computer stand toward rear of cabinet. The power button can be locked out and tagged if needed to stop use.

5- When the button is depressed for motor power the voltage indicator / power indicator light will illuminate to show that power is supplied to motors. **If your going to be working around or on top of the table the power switch will need to be locked out and tagged to ensure safety.**



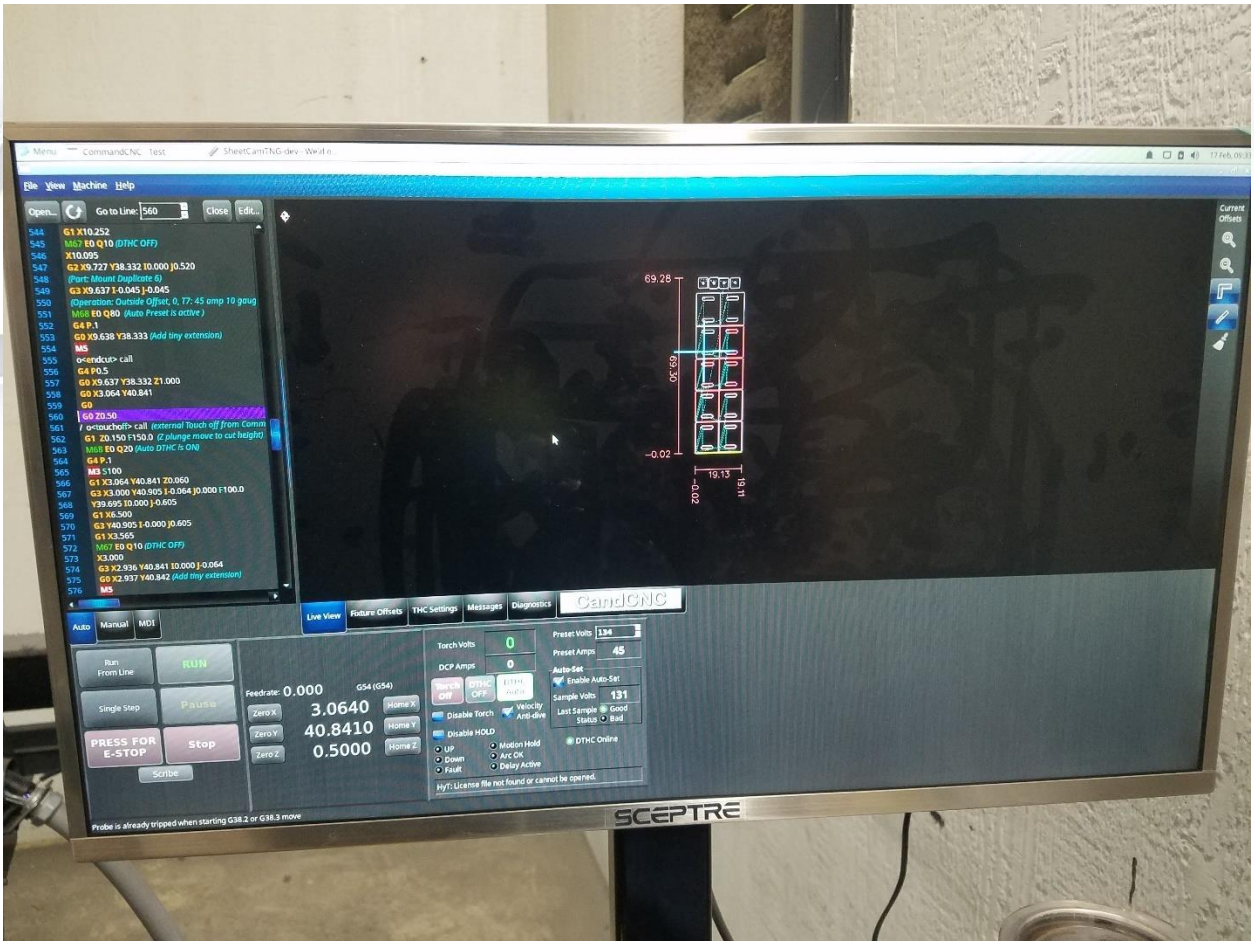
6- The motor and drive power will need to be powered on to supply power the MP 3700 located in the cabinet. The MP3700 is the large box located near top of fixture board. The MP 3700 is the controller.
7- If the command CNC interface is opened without the power supplied to motors and controller you will receive a no communication error. If present, just close the dialog box. Turn on power

to the motor and controller. Wait 5 seconds and re open command cnc interface(new or rusty steel).

8- Command CNC Interface is labeled similar to “New Steel” and or “Rusty steel”. They are located on the desktop or home screen.

9- “New steel” is used for steel that is in good condition and will utilize the “feather touch” operation. “Rusty steel” is used for material that is rusty, coated, painted, or has some sort of inhibitor on surface. The feather touch works very similar to an ohm meter. Once the feather touch senses the top of the material the torch home is set. If the rusty steel cutting interface is opened the torch will use the microswitch and offset the travel distance of the torch movement to operate the switch.

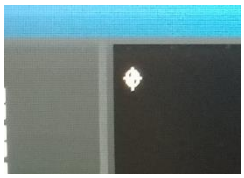
10-

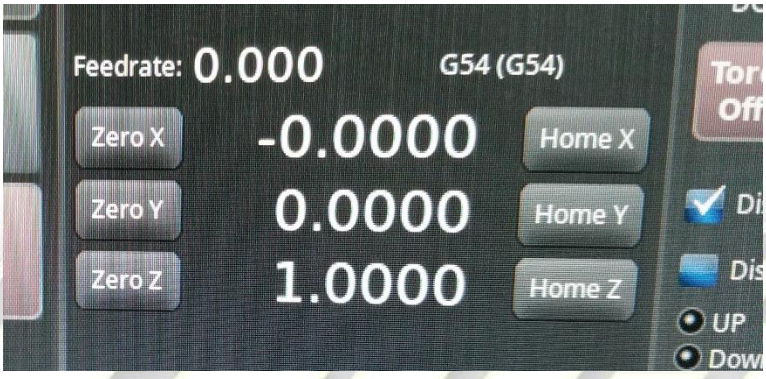


11- Your machine will open with the E-Stop button in lower left illuminated. You will need to “lift” E-Stop to operate the table. Click on the e stop button to remove E stop. If the button does not change check for proper torch placement and that none of the E Stop buttons are depressed.

12- You can now jog your table around using the Arrow keys for left, right, up, and down. Use the PgUp and PgDn located above the arrow keys to travel the Z axis up and down.

13- **You need to home the machine.** To home the machine jog the table to the lower left side of the table. **Do not ram the gantry into the stops**. Stop 4 inches before the X and Y limit switch. Then select the Home X and wait till it touches the home switch. Then Select Home Y and wait for it to touch off the Home switch. After the gantry has been homed with x and y axis a symbol will appear to let you know that your machine has been homed.

 Located in Upper left side of generated image.



Do not worry if there are still number is not the XYZ at this point your looking for triple 0's later…

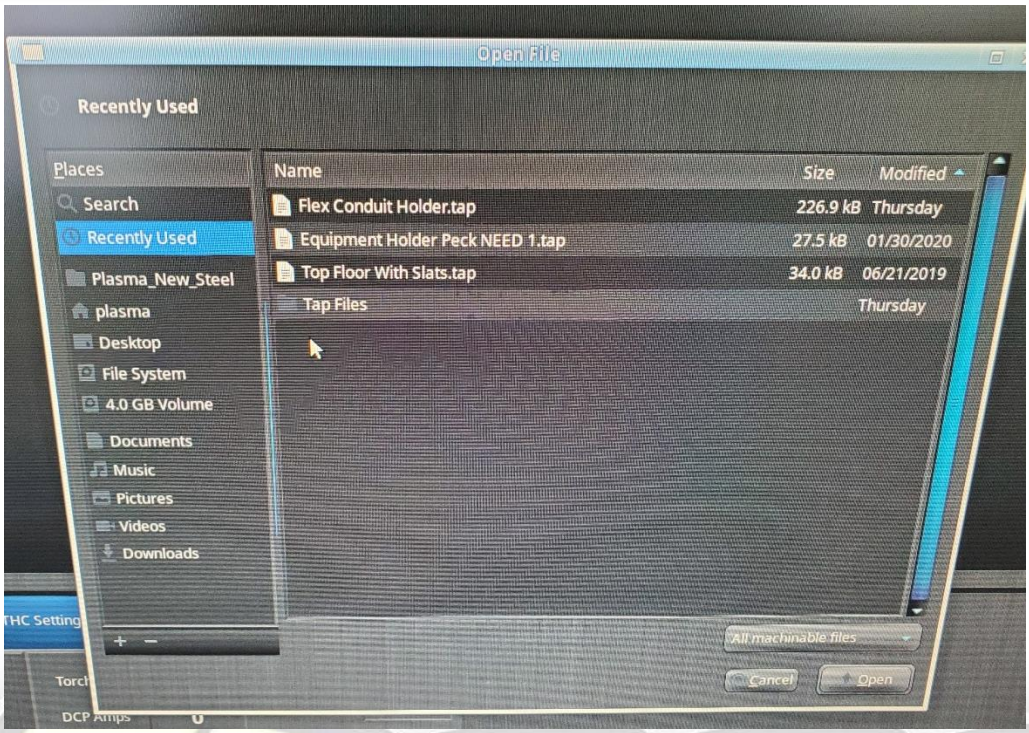
14- The Z axis will be homed when the material loaded is to be cut.

15- Granted that you followed the steps your machine is now set up and ready to accept Code to cut steel.

16- If you know how to use sheet cam you can skip the sheetcam portion. But if your not familiar with sheet cam please review sheetcam documents and watch videos located at www.bosstables.com

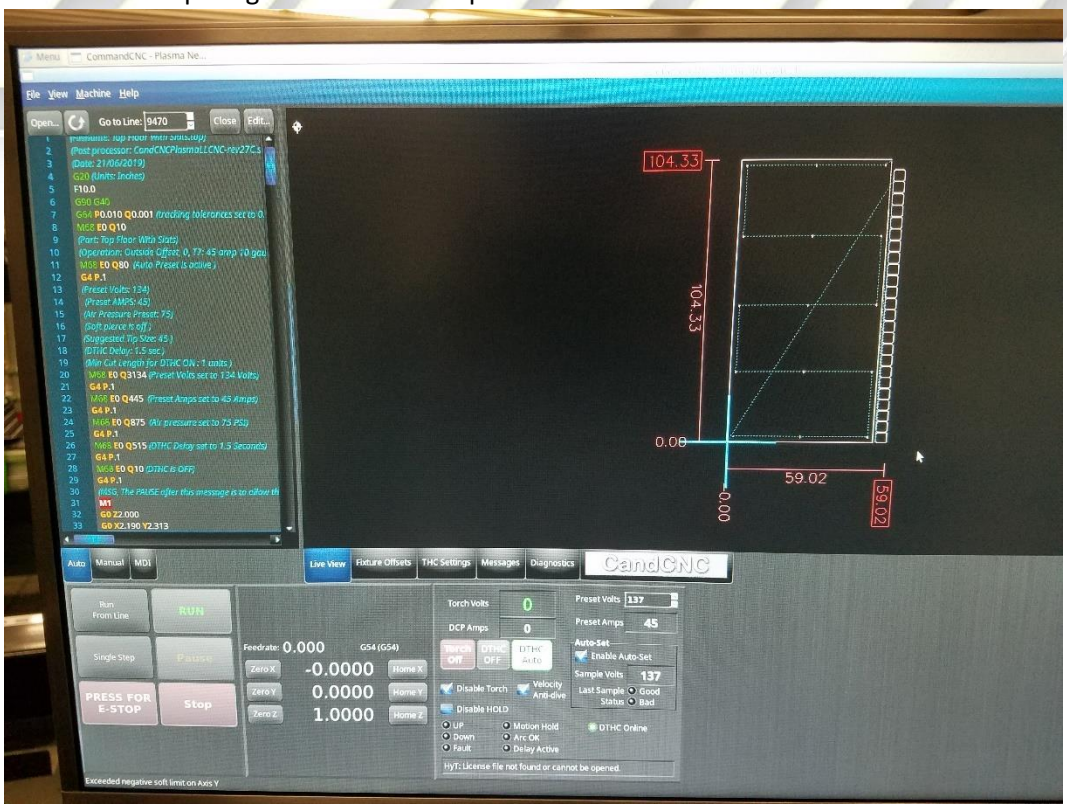
Operating the Cut Interface

1. You have your operation created in sheetcam and are ready to start cutting. Upper left corner click File-Open Find your file you saved from running your post processor and select open or double click.



You May select the Recently used tab for faster access to you files

2. The part will then be given a visual representation in the larger black portion of your screen. This is the tool path generated not the part itself.

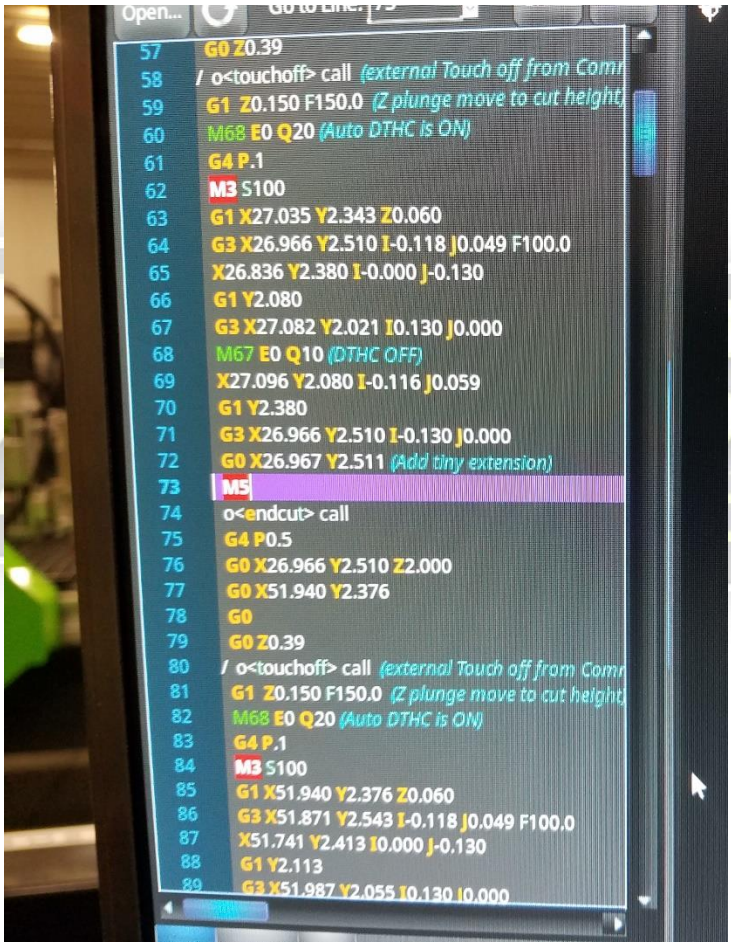


3. The GCode will be produced in the left side of the screen. There you can see the operations and tool chosen in sheetcam. This is very important as it tells the controller what the speeds and feeds are ect.
4. You have homed your machine but your image is located in a random location. You will need zero you part location.
    - **Home sets the machine parameters but it does not orientate the part on the machine. Once the machine is homed you will not be able to run the machine into the stops or run it off of the table.**
    - **Zero sets the part location located in the lower left of the screen oriented off of the torch. Travel your torch to any location and hit zero x and zero y to see the part move on the screen.**
    - **Bring torch to the lower left of the steel to be cut and hit Zero X, Zero Y the part will be located off of the torch tip. If you nested your part in sheetcam 4 inches away for the lower left corner your part will be offset 4 inches from the tip.**
5. Notice in the image above the upper most and right most numbers are red and outlined. The part is nested on the machine too close to the edge or is located off of the machine. If not homed do so now and zero the torch correctly oriented by the steel on the table.
6. Your part is now nested on the steel to be cut. If desired depending on allotted room travel the torch around on the screen and verify that the part or parts are going to fit in allotted sheet. Used the arrow keys and the screen as a visual reference to see where the torch will travel.
7. If your going to cut outside of the allotted material by .25" then you can move the appropriate x and or y axis to a -.25 and rezero the part on the material. Shifting the parts nested by -.25. if you simply do not have enough room you will need to renest the parts on sheet cam or resize the part.
    - **If you re-nest your parts you have essentially changed the g code needed to cut he appropriate job. You will need to run the post processor again (we advise not to save over the existing file but create a revision file number as a reference that you have indeed loaded the new job EX--- Test.tap will now be Test2.tap**
    - **Tip- you thought the steel was 16 gauge but there is 14 on the table you do not need to start over. Open sheetcam change the operation to 14 on ALL the parts and RE-Run the post processor to change the cutting operation.**
8. Everything looks good? You have told the machine where X and Y are oriented but you now need to zero the Z height. You can home z in any location as tong as its on the material your cutting. Hit Home Z and the torch will travel down and touch off of the steel. If the Z values does not change to 0.0000 hit Zero Z and you ready to cut.
9. Your part fits and your ready hit Run in Lower Left and you will be prompted to check settings. Hit ok and check settings. To start hit resume and your now cutting steel.
10. The torch will travel to the initial pierce point and touch off the steel again. Finding the height of the steel and offsetting the pierce height. Torch will fire and drop down to cut height and continue on path wrote by sheetcam.
11. **Do not stare at the arc as it will damage your eyes.**

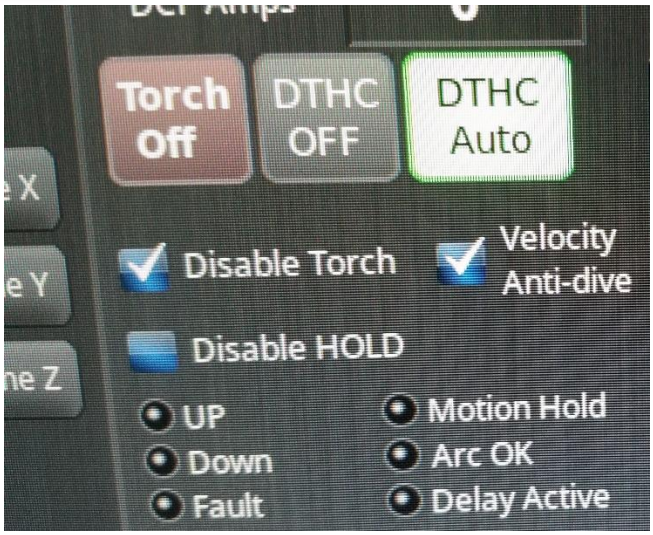
Operating the Cut interface like a pro

Your up and running cutting some steel but whats all the other stuff for?

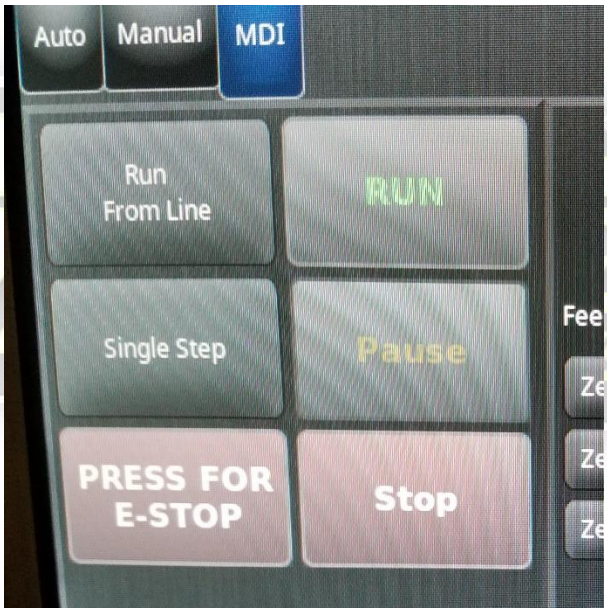
1. How to restart a cut
    - There are lots of reasons you will need to restart a cut. Ran off the steel? Poorly nested parts, collision with part that tipped up and dislodged the torch, consumables are worn, lost your ground, or just quit cutting.
    - Clear any obstructions or issues that have stopped the operation.
    - Hit stop on the lower left corner. Click on the last M5 as shown below. The M5 is the end cut of the last feature.



- You will need to disable the torch momentarily while it runs through the G-Code traveling to where the arc stopped.



- With the torch disable button checked as shown.
- Click Run from line (NOT RUN) if you hit run the G-Code will start from beginning.



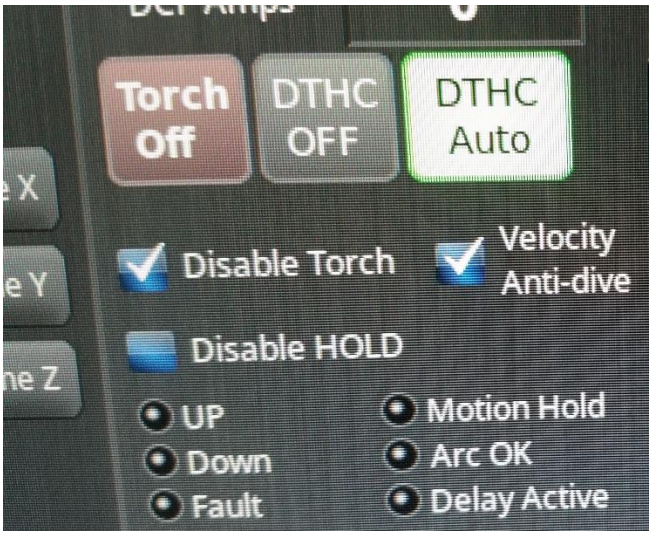
- Torch will travel to last M5 and come back to current feature, touch off, (the torch will not fire if disable torch is selected) and travel on the same part that it was running on. 1" to ½ inch before where the cut stopped uncheck Disable torch. The Torch will pause, fire, then continue with the cut as if you never had an issue.
- **STOP- M5- DISABLE TORCH- RUN FROM LINE- ENABLE TORCH-**

2. Move torch after there is a collision. The torch has become dislodged or the upper torch travel limit switch has been activated. The Machine has E Stop Activated and you can not get e stop off.

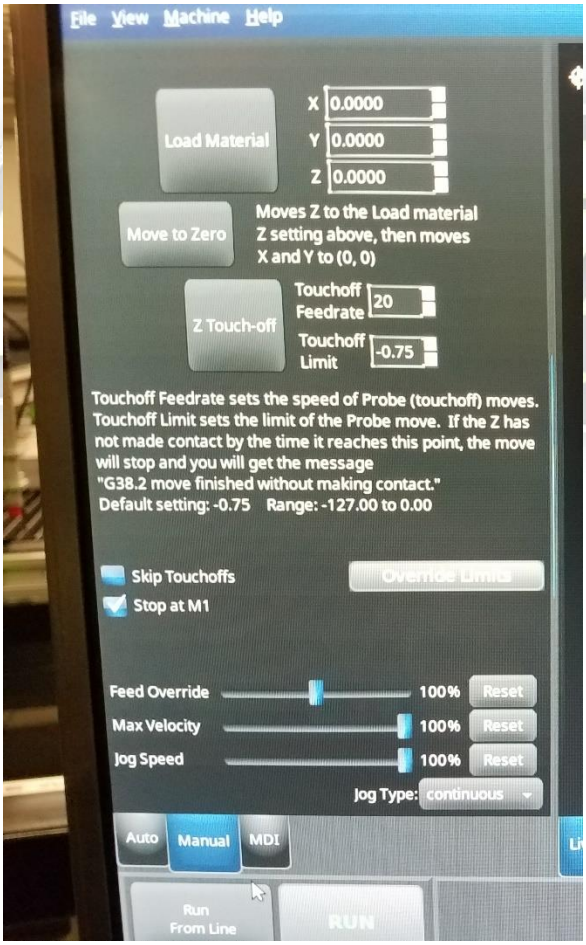
- **Press E-Stop Bypass and hold** you need to hold as the buttons are momentary switches that are only operated by holding them down.
- **While holding the Bypass down** click the E-Stop button to remove.

- **Travel the torch away using the arrows or PgUp While holding the Bypass down.** After traveled out of the way of obstruction. Place torch back onto magnetic torch mount. And remove E stop again by clicking on it.

3. You can hit Pause at any time while the torch is not cutting and simply hit resume to continue cut.
4. When the torch touches off steel and fires the motion hold is maintained until the arc ok signal is given by the power unit. If the Disable hold box is checked the torch will fire and not wait for the Arc Ok signal starting without properly starting an arc. Potentially leaving a bit of material that was not cut.
5. The Disable hold is useful in trying to “cheat” you machine. Such as staring off the steel, you might have left the ground off and it pierced but never continued on. Now there is no steel directly under the starting point. Check Disable hold and it will not wait for the signal and just start cutting!



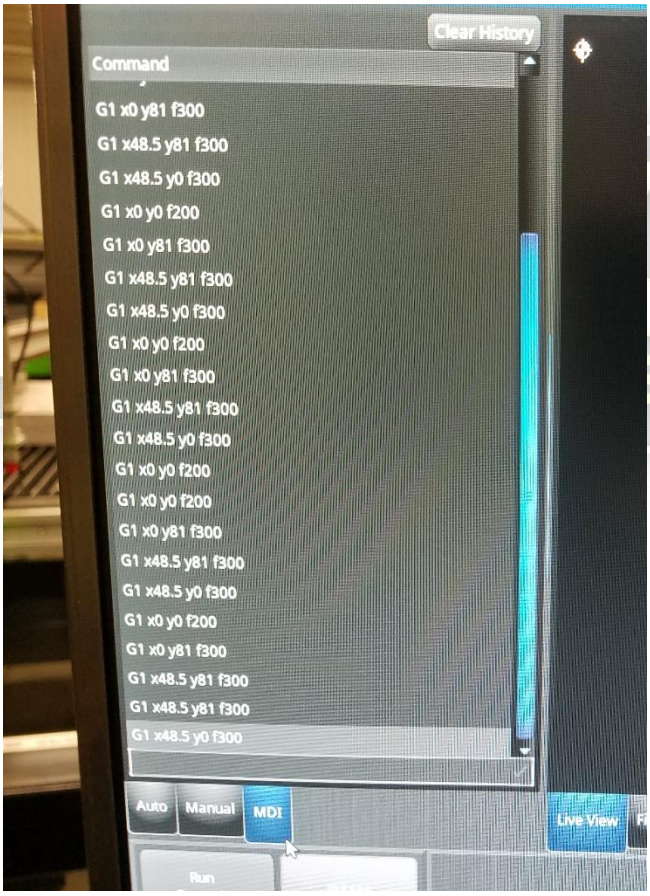
6. Pictured below is the Manual Tab.



- Load material when selected will have torch travel to the set location (Notice the Z is 0.0000 and it will probably hit the table or material as it will travel to 0.000 enter 2 for Z axis as a safe number.

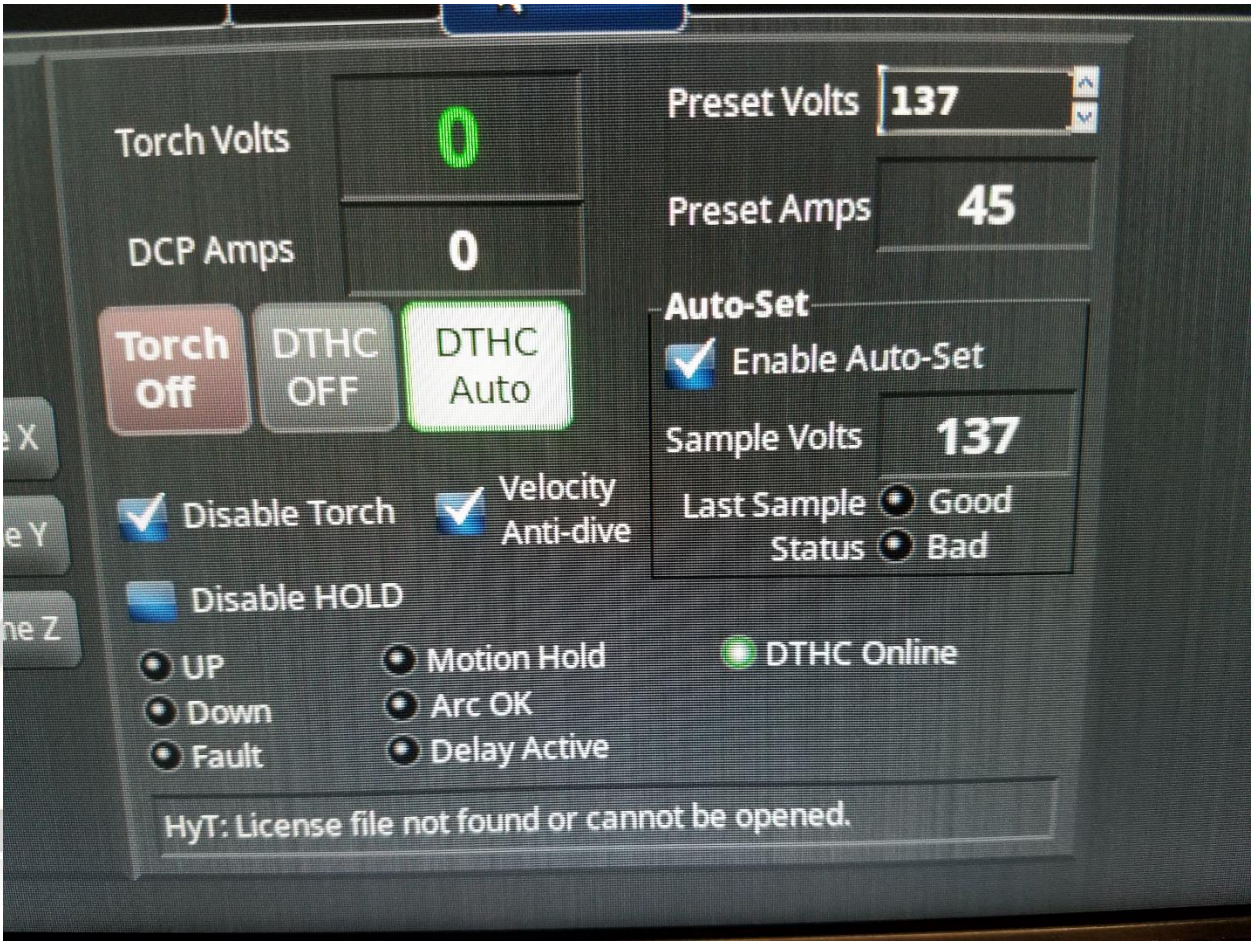
- Z touch-off does as described in picuture and can also be adjust in sheetcam under the jet cutting screen (that is where the number initially come from when creating a tool path)
- Skip touchoffs will allow you to cut faster as it will not touch of material and just move to pierce height and fire the torch and continue onf if the arc ok is given. If utilized your material needs to be very flat an not warped.
- Feed override will slow down of speed up cut in an effort to improve cut quality. 15% and over will turn off DTHC
- Max velocity will set the maximum speed you machine will run at (even you rapid speed)-----WHY? Because you can set the travel speed very low to allow time between cuts as not to overuse the machine according to duty cycle.
- Slowing the jog speed will allow you to dial in your torch placement by slowing dow the movement initated by the arrow and up down keys.

7. MDI tab- The MDI Tab allows you to wright simple G code by hand and move machine manually (NC)



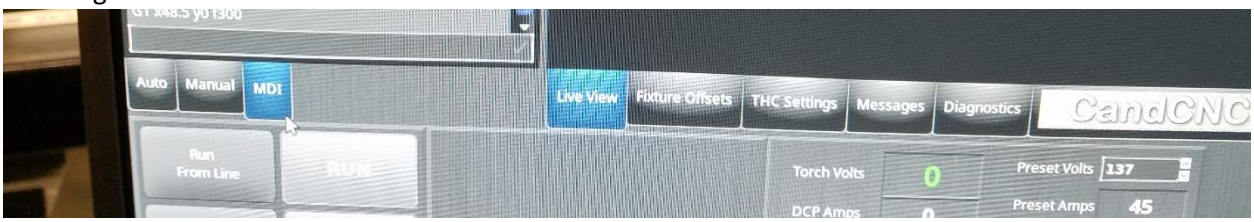
8. Torch volt is what sets the digital torch height. Your preset volts are what the machine is set at. You need to change the volts with the machine paused or stopped as it will cause and error. The green value is what the torch volts are reading from the power unit during the cut. Measuring the cutting height is performed by letting the machine start the cut, wait till the torch volts have a steady reading and hit the stop button. Use a caliper and measure a piece of material or drill

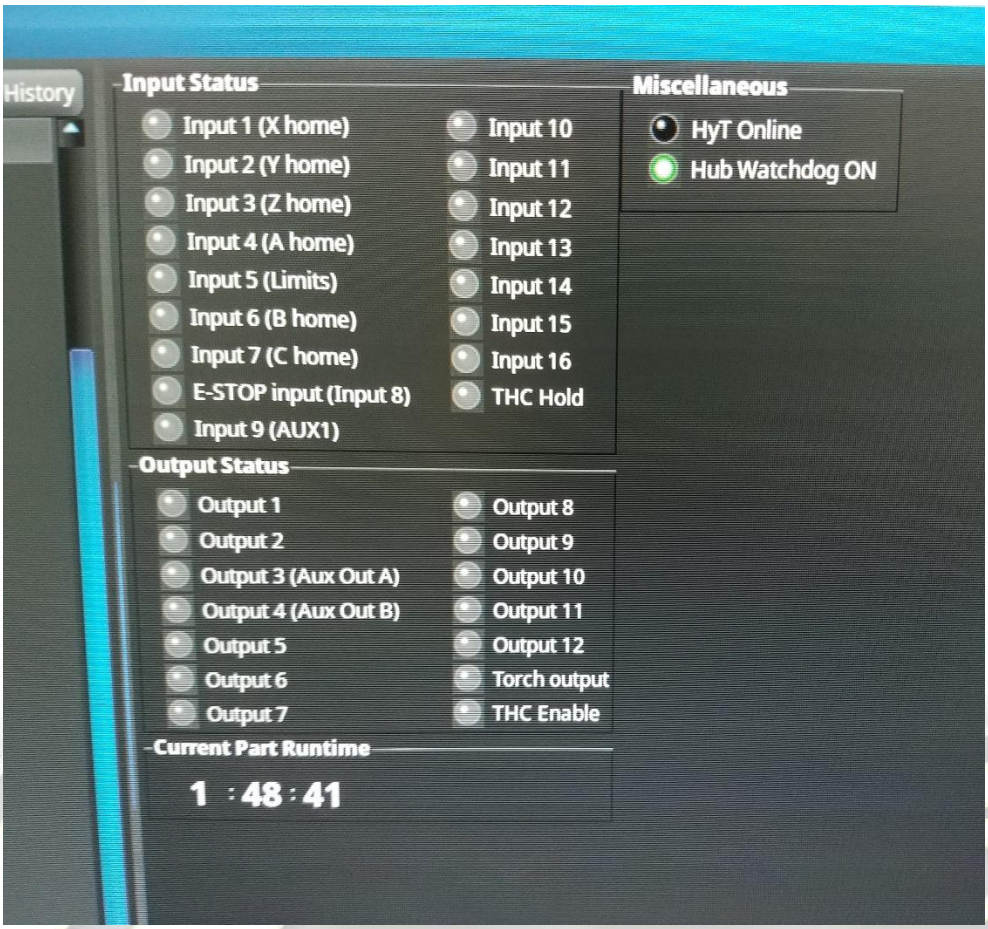
bit that .060 thick and slide under the torch tip. If the torch needs to be raised simply adjust the torch volt higher. Lower? lower the volt located in the preset box. Auto set will use the machine travel height to adjust the auto set volts and inform you that the last sample was good or bad. The settings here will generally not need to be changed.



HyT: License not found is normal unless you purchased the Hyt connect kit.
Hyt will automatically adjust the volts on the plasm cutter.

9. The Diagnostic box is located at the bottom of the screen





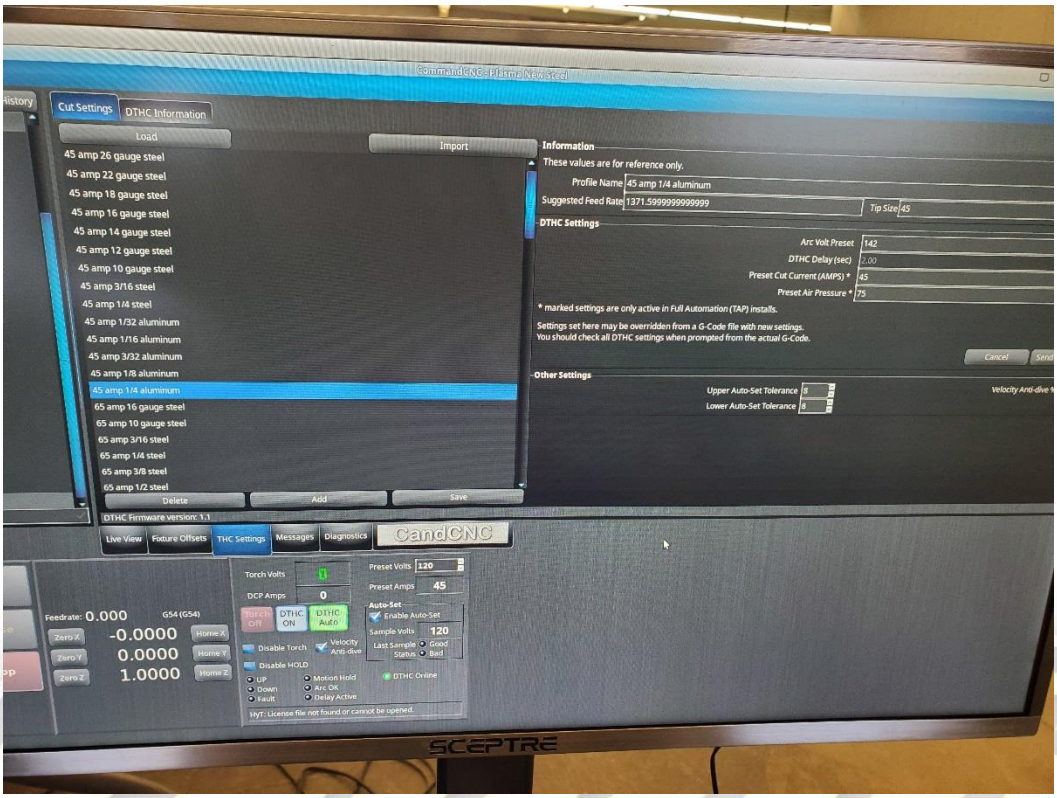
Once the diagnostics box is opened you can review the inputs and outputs. Inputs are listed as shown

1. X Home
2. Y Home
3. Not Used
4. A Home is YA or right side of the table Y axis
5. Computer stand E Stop
6. E-Stop located on the gantry Y and YA Axis
7. Break away Sensor
8. Torch Limit Switch
9. Z Height for feather touch and Z homing switch.

Outputs are used to control features such as Pneumatic Scribe, Oxy Torch, and Router operation.

Notice the Current part runtime, this will time you job from when you hit RUN. If you hit Run again it will restart the timer.

10. You can adjust the cut settings directly from CandCNC Used the THC Setting located in the dmddle of the screen



11.