Arclightcnc.com   **866-222-2154**

Arclight Dynamics Table Operations Manual

Version 1.1

10/17/13

<u>**Maintenance**</u>

<u>**Table Rails:**</u>

Keeping your table rails clear of large debris will greatly decrease the likelihood of a positioning error. The rails simply need to be wiped down every now and then with a light oil or WD-40. The table is equipped with rail scrappers that will remove larger debris, but if you are knocking off slag and welding right next to the table this can lead problems. **If you are cutting lots of aluminum you will need to pay more attention to your rails as this produces aluminum chips that can easily get smashed into the rails and rough up their smooth surface.**

<u>**Z-axis:**</u>

Your Z-axis is the part on your table that requires the most attention and maintenance. Your **Z-axis** has a small slide, where the torch attaches. This slide only moves a small distance during normal operation of the table and must move smoothly. This slide allows the **Z-axis** to sense the top of the steel plate and trigger the micro switch that allows the table to set it's initial pierce height correctly. **So if this slide does not stay lubricated it can get sticky and lead to improper pierce heights**. This slide has two slide blocks with grease zerks on each for lubrication. These should be lubricated once a month.   The area around the micro switch should also be kept clean and blown off with compressed air every so often. This will prevent debris from interfering with the operation of the switch. In addition to this it is helpful to lubricate the **Z-axis Acme screw** with a dry lubricant.

Your table is equipped with a **Magnetic Release Torch Holder** that will allow the torch to breakaway in the event of a collision, thus preventing damage to the torch. To re-attach the torch, place the mount carefully up against the back plate and allow the mount to snap into the magnets. Your torch's angle can be adjusted in the **X-axis** direction by loosening the four bolts that hold the mount to the slide and angling the torch. The torch can be adjusted in the **Y direction** by loosening the two bolts right below the torch clamp.

<u>**Water Table/ Anti-corrosive:**</u>

Your water table must have some form of anti-corrosion agent mixed into the water to prevent rust. **We recommend a cutting solution produced by FMT and sold by Fastenal, called Sharp Cool Premium Synthetic. This mixed into your water in a 10% solution does a nice job preventing rust**. **There are many other products on the market that will work equally well**.

<u>**Cleaning the Table and Slats:**</u>

Depending on the amount of use and thickness of metal you cut, you will need to clean out the table every 2 to 6 months. **If you wait too long the table can fill up with slag which can decrease cut quality**

and interfere with proper voltage feedback. To clean the table; drain the water out of the table, remove the slats and remove all the debris. The slats can be dropped on a concrete floor and much of the slag with pop off them. You can then scrape the slats with a steel flat bar. It is not required that the slats be perfectly flat, just have the high points knocked off. Depending on what you are cutting you can usually reuse your slats many times, getting one to two years of use out of them. Often you can get more use out of them by simply flipping them over. Slats can be replaced by shearing an 11 gauge sheet into the proper width and then 2 ½" strips.

Control Cabinet filter:

Your control cabinet has a filtered fan air input that maintains positive pressure inside the cabinet, preventing dust from getting inside and it is also essential for cooling the electronics. These filters can be found online through various retailers. The size of the filter is a **10"x10"x1"** and is not commonly found in hardware stores.

Powering the table on:

To power the table on follow the steps below,

1. Turn on the **power strip** inside the control cabinet. This will turn on the power to your black control box, but it will leave the motor power off. This control box also has it's own power switch on the underside by the front right corner , however you can just leave this switch on and power the unit on and off with the power strip.

2. Power on the **computer** and let it get fully turned on.

3. Double click on the **Mach 3 Loader** and then select the appropriate profile. You must select the correct profile otherwise the table may not run correctly.

4. Once mach 3 has loaded up turn on the **motor power** by hitting the green button on the control box. This will lock up the stepper motors. I**t is very important to remember that you should never plug in or unplug the motor cables from the control box with the motor power on**, only do this with the motor power off. So is it very important that you ensure the cables are fully plugged in before powering on the motors, loose cables can cause shorts that will damage the motor drives.

5. Now that the motor power is on you can click on the **reset button** in the bottom right corner of the mach 3 screen and it will stop flashing. If the motor power is turned off Mach 3 will go into reset mode. **Once Mach3 is out of rest mode you will be able to jog the table**.

Jogging:

Jogging is done by using the arrow keys on the keyboard. Right and left arrows control the x axis and the up and down arrows control the y axis (gantry). **The page up and page down keys control the z axis.**

Practice moving the table around using the jog keys. You can run all three axis's at the same time if you like. **Remember that all of the limits on the table are hard stops.** So if you jog to a limit the table won't stop, The switches on the front of the table are homing switches only.

Fast Jog:

While jogging using the direction keys you are in **slow speed** jog mode, this is 30% of full speed. If you hit and hold down the **shift key** and then jog you'll be in **full speed** jog mode, which is 700 inches per min (IPM).

Step mode Jog:

If you hold down the keyboards **control button** and jog you will be in step mode. The default setting is . 01", so every time you hit a arrow key that axis will move .01"

Jog Control:

At the top of Mach3 you will see three page options, **Setup/Testing, Settings, and Diagnostics.** The main page (which you are on) is called **Program Run page**. These pages provide different options and we will be talking more about them later.

Right now click on the diagnostics page, this is where you will find the Jog control options.

Your jog control option are located in the bottom right of this page and this will be the main reason you use this page.

At the bottom of the Jog mode box you will see **Slow Jog Rate**, this shows the percentage of the full speed that the table is set to move in slow jog rate . You can use the arrows to raise and lower that rate or type a number into that box by clicking on it. If you type a number into then box the widow will turn pink, you need to hit enter for this setting to stick (this is the same the rest of Mach 3). So by changing this rate you will notice that the speed at which the table jogs, changes. To the bottom left of this jog control window you'll see a box with **"blended"** written above it.

When you jog this widow will display the speed you are moving at in inches per minute. If you adjust the **slow jog rate** you will see the speed change.
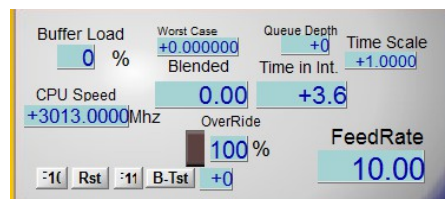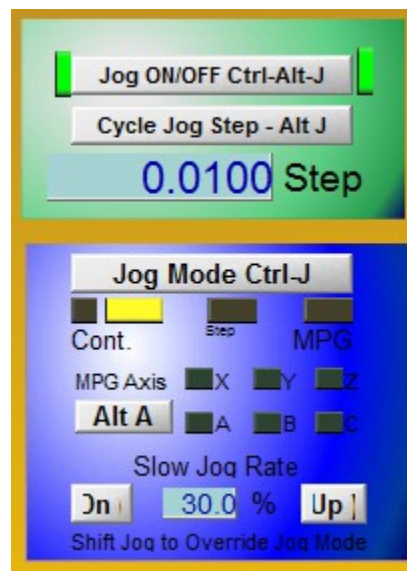
Manual cuts:

A manual cut is simply turning on the torch and jogging the table to cut across a plate. This is useful for cutting up your scrap plate. The first step to doing a manual cut is to set you jog speed correctly. You will need to set the jog speed based on what material and what amperage you are cutting at. Cut speed charts are located in your plasma cutter manual. Once you have determined the speed you should be cutting at (based on thickness, amperage, and using shielded tip) you will need to make your table jog at that approximate speed. This is done by changing the **Slow Jog Rate** percentage until you see that the table is jogging at the right speed in the **"blended"** read out.

Once you have the speed set right, click on the **Program Run page** at the top left of the **Diagnostics page**. Now when you are back on the **Program Run page,** jog the table to the edge of the piece of steel you want to cut off. Position half on half off the sheet with the torch tip about 1/8" above the plate. Now click on the **Torch on-off button** to fire the torch and then start jogging in the direction you want, at the same time the torch fires. You then jog off the edge of the plate to complete the cut and jog back over the plate to decrease the splashing. Lastly click on the **Torch on-off** button to turn off the torch. **While manual cutting the THC will not be active so you may need to tap the z axis up or down while cutting to maintain close to the right height.** The most important part is that the torch doesn't touch the material.

DROS (Digital Readouts)
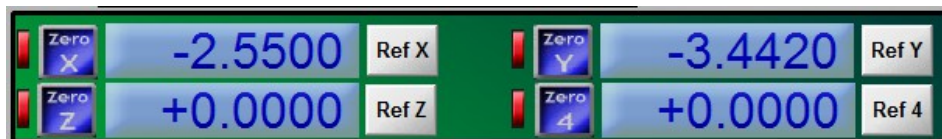
These are located in the top left of the **Program Run** page of Mach 3 (main page). These indicate the position of your X, Y, and Z axis's relative to where the table was last zeroed out. As you jog the table you will notice the DROS moving positive and negative. As you jog the gantry away from you (towards the back of the table) this will move the y axis in the positive direction, and jogging back towards you is in the negative direction. Likewise if you jog the X axis to the right its moves in the positive direction and to the left is the negative direction. The z axis moves positive if you move it up and negative if you move it down. You won't have to use the 4 axis DRO , so just ignore it.

Zero axis button:

To the left of each axis **DRO** you will see a **zero button** that will zero out the axis when hit. This will be used at the start of each plasma cut to zero out the X Y and Z axis's. **This is a very important step to making a cut and is not to be missed.**

Reference button:

To the right of the axis **DROS** you will see a **reference button**. We will be using this to reference the axis's up against the home switches at the front and left sides of the table.

1. Use the page up button to make sure the Z axis is raised above the table surface, at least a inch or so.

2. Ensure that there is nothing on the side rails that will get in the way of the gantry moving to the front of the table.

3. Click the **Reference Y axis button** , and the table with move the gantry slowly toward the front home switches and stop.

4. Click the **reference X axis button** , and the X axis will move toward the left side of the table.

5. When the Y and X axis's have been referenced the table will be at the **machine zero**.

Squaring Gantry with the y axis home switches.

Your table's gantry is designed to have some play in is motion so it can be adjusted for square cuts. The motors on each side of the gantry hold the gantry square and if they are powered down and the gantry is moved or bumped it will go out of square. When the Y axis is homed it drives each side of the gantry independently untill it hits the home switch on the front of the table. This will square the gantry if it is out of square and the motors will hold it square while they are powered up. **So it is best practice to home the y axis each time the machine is powered up.**

The home switches are adjusted at the factory to square the gantry properly, but if they get moved you may need to re-adjust them so the table cuts square. This process entails making small adjustments to the position of the home switches forward or backwards depending on which way the gantry is skewed and then re-homing the Y axis to make the adjustment. Then you will need to make a cut to see if the adjustment worked.

G-code display window.

Mach3 displays the g-code in this window . To load up a g-code use the load code button. The tables desktop should have a g-code file on it from testing that you can open up, (.TAP file). Once you have a code loaded you will be able to use the scroll bar on the side of the code window to move up and down through the code. You must use the
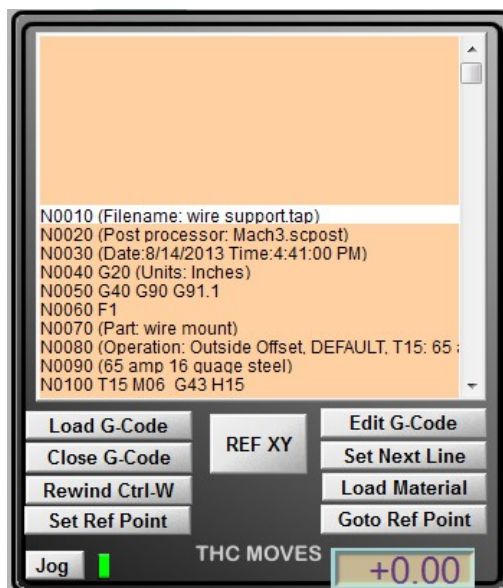
scroll bar to navigate, you wont be able to click on a line. While using the scroll bar to move up and down through the code I recommend using the up and down arrows at the top and bottom of the scroll bar to move one line at a time. Because if you have a large file loaded up moving the scroll bar a small distance can move the code a long distance and it is easy to lose your place.

The white box in the center of the g-code window indicates which line of code Mach 3 will read next. **It is important to remember that Mach will not read the code above that line.**

Load code/ close code buttons

Below the code window you have the load and close code buttons, these will load and close g-codes.

Rewind Button:

The rewind button simply rewinds the code to the beginning of the code.

Set Ref Point button:

The Set Ref point button will save the current X Y and Z coordinates shown in the DRO windows. This position is saved on the settings page and one position can be saved at a time.

Goto Ref button:

The **Go to Ref point** button will move the X Y and Z axis's to the saved reference point. This should be done with care, as if you have zeroed out your axis's since setting the reference point the table will not move to the right spot and it may crash into the hard stops. We suggest that if you are moving to a known reference point , you jog close to where you think the point is and then use the **Go to ref point** button.

Jog button:

Clicking this button will simply disable jogging. So if you loose your jogging capabilities check that this button hasn't been accidentally turned off.

REF Y X button:

This will reference your x and y axises up against the home switches. However it has one major difference, the X and Y axis's will move at full speed, at the same time till they reach the last set zero point, then from there they will progress slowly till they hit the home switches. For this reason we suggest that you only use the individual Ref Y and Ref X buttons by the **DROs**.

Edit G code:

This will pull your loaded g code up in **Notepad** and allow you to make changes to it. We however recommend you make changes in Sheetcam and let it re-write the code. But it is very useful for reading

your code in a larger format than the code window. The beginning section of your g-code contains lots of useable information and sometime you will want to check it.

Set Next line button:

This tells Mach3 to start reading the g code on whatever line of code is currently listed in the white line, in the center of the g-code window. This is used to fast forward and rewind the code. This will be discussed in further in the **Error Recovery section**.

THC Moves:

This window indicates how much distance the Torch Height Control has adjusted.

Feedrate Override:

This window is used to modify the speed of a cut while it is running or before you run it. The feed rate of a cut is written into the g-code, therefore this is the only way to modify it without redoing the code in **Sheetcam**. By using the up and down arrows you can increase or decrease the feed rate by 10% a click, up to a max of 300% or down to a minimum of 1%. Take note, if the red bar above this is flashing, it means the feed rate is overridden and even if you load up a new g-code it will still run at the overridden percentage. If you are running a g-code at a overridden federate and you hit the stop button it will pull the federate back to 100%. Using the reset in the feed rate override window will pull the federate back to 100%. When you run a new code, the feed rate will not adjust to the current speed until the g-code is started and Mach3 reads the feed rate in the code.
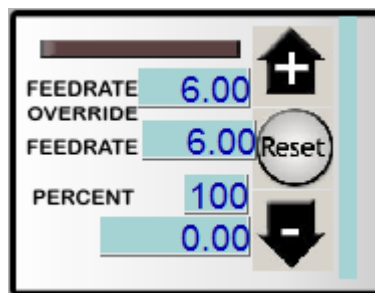
There is a range of speed at which you can cut different metals at set amperage's. The feed rates listed in the Sheetcam tools and plasma cutters' manual are ballpark starting figures and they may need to be adjusted to maximize cut quality. For example you can often improve the quality of a small hole in thick steel by decreasing the federate by 20-40%, this may decrease the bevel of the hole. This can be done on the fly by using the **feed rate override**. At other times you may want to speed a cut up so you get done with your cut faster; if the cut quality is not as essential.   We encourage you to play with the feed rates during a cut so that you can learn the effects of speed changes on the cut quality.

Auxiliary Outputs:

This window controls the two relay controlled outlets on the bottom of the black control box located in your control cabinet. If you feel on the bottom left of the black box you will notice a power cable and behind it, two standard outlets. These outlets are controlled by these output buttons. These can be use for controlling anything from fans to lights, or pumps. Max Amperage is 15 amps. If you turn on a output and then run a g-code the output will turn off at the end of the cut.

Loading up g code

To load a g-code you need to use the **load code button**. If you haven't gone through the section on how to generate g-code in **Sheetcam** you will have a sample g-code on the desktop of the tables computer. This code is a **.tap file** and was used to test your table. Use the **load code button** and navigate to the desktop and select one of the .tap files. Take note that Mach 3 will also open up a .DXF file if you open them by mistake, but obviously it won't run it.

After the code has loaded you will see the tool path display in the table display window. If you don't see the tool path click the **regen toolpath button** below the table display window.
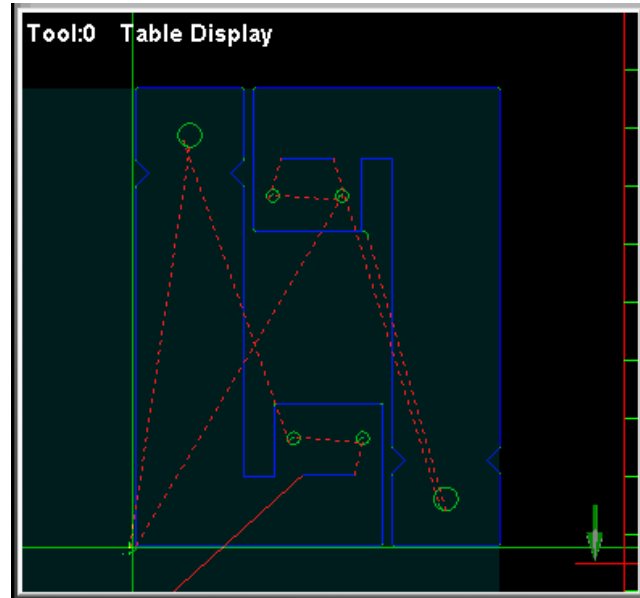
Features of the Tool Path display

1. Blue lines indicate cut lines

2. Green lines are also cut lines , but they are arcs.

3. Red dotted lines indicate rapids in between cuts

4. The arrow on the right will indicate the movement of the z axis while the table is cutting

5. Once you've jogged to where you want to cut and zeroed out your X, Y and Z axis's the green cross will snap to the lower left corner of your part. This indicates the position of the torch and it will follow the torch around the part on the display if you jog the x,y axis's. This is an excellent way to check to see if you have enough material to cut your part out by jogging around to the edges of the part and see if you are still on your material. Note if you've zeroed out and then jogged your X, Y axis, you don't have to move back to the zero position, The table still knows where it is.

6. This display can be panned by holding down the right mouse button and zoomed with the scroll wheel.

7. If you use the left mouse button the display will rotate in 3D. However this is not that useful for plasma and to get back to the flat position use the **Regen Toolpath button**.

Navigating the g-code

Use the scroll bar on the right side of the g-code window to move down through the g-code. As code that commands motion is placed in the white line the line on the display will light up showing you where in the code you are. Spend some time becoming familiar with the code and the what type of motion it commands on the table display.

Dry Run: (running the table without cutting)

This is an excellent way to get comfortable running the table without cutting up a bunch of steel. (But I highly recommend that you get a full sheet of 16-14 gauge steel to practice on and plan on cutting it up into scrap before you attempt more expensive cuts.)
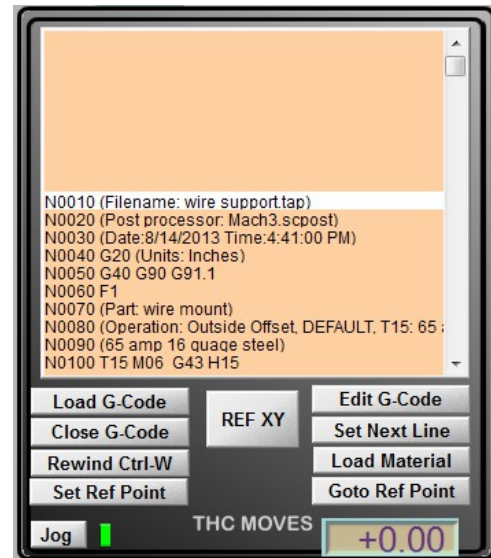
1. Make sure your plasma cutter is turned off

2. You must have material on the table for the torch to touch off on, it doesn't have to be metal, it could be plywood. But it should be flat!

3. Turn off the THC (Torch Height control), This is the Blue button in the middle of the screen that says THC on /off, Click it so it is not flashing green. If the THC is not turned off the table will move to the position of the first pierce and stop, waiting for the arc ok signal (more on this later). The DTHC on/off button can be ignored for this.

4. Position the torch over top of your material, ensuring you have enough room for the table to run through the part without running off the material or running into a hard stop.

5. Zero out the X Y and z axis's, position the z axis 1/4" above the surface of the material when you zero.

6. Verify that nothing is on the rails that may get in the way of the motion, and then hit the run button.

   Mach will run through the first part of the code and adjust the preset voltage and DTHC delay to match the code and then at the bottom of the mach screen you'll get a message to check your setting and then hit run. For this dry run just hit run again. Your machine will run through the cut without cutting.

A one thing you should notice while it is moving through the cut is the touch off sequence. This is the movements the z axis makes before the start of each cut. A dry run is a good chance to observe this key sequence.

## Touch off sequence in detail:

It is important to understand the machine touch off sequence and how it senses the top of the plate. Below is listed the order of events in a touch off.

1. Table moves to the pierce location at 1.5"

2. Z axis drops down to height set in **Plunge safety clearance**, this is usually .75" to 1.25" (this is set in the **Sheetcam job options**)

3. The Z axis then starts "feeling" for the plate slowly. Once the torch touches the plate it triggers the z axis home switch. Note, if there isn't anything for the torch to hit it will continue to move downwards into the water till the micro switch is triggered by the torch tip hitting something. This might end up being the bottom of the pan and if the torch fires it will easily put a hole in your water pan. So if you incorrectly zero out your table and it attempts to touch over the water hit the **e-stop.**

4. Mach 3 then zeros out the z axis

5. Mach moves up the **switch offset** distance (this is the throw distance of the z axis home switch). At this point the torch should be sitting right at the top the plate, not above it, not resting on it.

6. Mach 3 then zeros out the z axis again and moves up to **pierce height**. This distance is set in the **Sheetcam tools** and is usually .15" for Hypertherm plasma cutters.

7. Mach 3 now fires the torch and waits for the **Arc OK signal** and allows the **pierce delay** to elapse.

8. Mach 3 Drops down to cut height and starts the cut.

This sequence of events has to happen properly for the table to pierce correctly. Also the dry run can be repeated and you can practice modifying the feed rate.

Testing the plasma:

Turn on the plasma and position the torch above a scrap piece of steel (under 10 gauge thick) and set the Torch about ¼" above the surface of the of the steel. Click the torch on /off bottom located in the center of the Mach 3 main screen , this will fire the plasma . Fire the plasma for 1-2 second and then click the button again to turn it off, this test ensures that you have the plasma properly hooked up. While you have the plasma firing, check to see if you have proper voltage feedback by looking at the "Torch Volts" window at the top, center or the Mach 3 screen. This should register around 130- 200 volts.
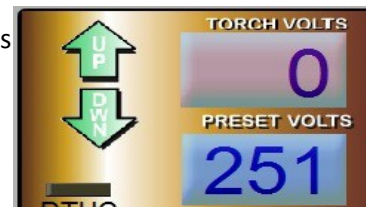
First cut:

After you have competed the initial dry run , you will than cut the part.

You have already zeroed out the table when you did the dry run , so unless you modify the DRO's by zeroing them , the machine will still know where to start the cut. Follow the steps below to set up for the cut.

1. Turn on plasma cutter (it is advisable to read the section on plasma setup and air supply)

2. Verify that the plasma is set to the right mode (plate cutting) and set to the correct amperage, with the right tips installed.

3. Make sure you have adequate air supply to the plasma cutter (125 PSI to 130)

4. Check to make sure nothing is on the rails of the table that would get it the way of the table's movement.

5. Verify that the g-code is rewound to the beginning and looks correct on the tool-path display window.

6. Ensure that you have enough room on your plate to cut this piece out. this can be accomplished by jogging the x and y axis around the plate and you will notice that the green cross hairs on the toolpath display window follow the movement. So you can watch the cross hairs till they get to the edge of the part and if your torch is still over steel, you have enough room in that direction.

Next you will hit the run button and Mach 3 will run through the first part of the g-code and set the Preset volts and the DTHC Delay to the values set in the g-code. A message will appear at the bottom of Mach telling you to check your DTHC settings and hit run to continue. Now you will want to check the preset voltage against the values listed in your plasmas manual to make sure they match. Finally hit run again and your table will start the cut.

Torch Height Control (THC)

The torch height Control monitors the voltage feedback from the cut and uses it to maintain the proper cut height. The torch height control is very effective at allowing the the torch to follow bends in the steel and warpage. It is not however, fast enough to cut items like corrugated steel unless it is cut very slow. The torch height control monitors the actual voltage at the cut and compares this to the preset volts, it will then raise or lower the torch to increase or decrease the voltage to match the preset voltage. As the torch is pulled away from the material the voltage increases. As the torch is moved closer to the material the voltage decreases.

The ideal cut cut height is determined by the plasma cutter manufacturer. For Hypertherm this is usually .06" or about the thickness of a nickle. The tools in Sheetcam have all the preset voltages set in the toolset for the Hypertherm cutters. It is important to understand that you set the Preset volts in order to achieve the proper cut height, .06". Many factors can effect the voltage feed back while cutting and so you may need to make adjustments to the preset volts to achieve the correct cut height.

Factors that can effect voltage feedback

1. Rusty or coated material

2. Differences in steel manufacturing process can lead two identical sheets cutting at different voltages. Different alloys of steel will have different voltage feed backs.

3. Low air pressure will decrease the voltage

4. Positioning of the work clamp and the quality of the connections can change the voltage feed back by 10+ volts and effect the cut quality. Don't be afraid to move you work clamp around and clean it's connections.

5. Damaged or worn consumables

6. Changes in cut amps and nozzles

7. Increasing or decreasing feed rates

## Torch Volts window

This displays the actual voltage at the cut.

Preset Volts window

This sets the target voltage that the THC attempts to achieve. This is usually set by the g-code when the run button is hit for the first time. You can type voltages into this window, but you must hit enter for it to change. You can also use the green Up and Down arrows to increase or decrease this voltage while the table is cutting. Raising the voltage will make the torch run higher off the table. Lowering the voltage will make it run closer. You will need a welding hood or shade 5 glasses so that you can take a close look at the cut gap while cutting. You may need to increase or decrease the voltage by +5 volts before you can see a difference with you eyes.

DTHC Delay (digital torch height control delay):

This delays the torch height control from turning on during the pierce. This delay is in seconds. When the plasma pierces the voltage spikes up. If the THC is allowed to be on during this spike in voltage, it will automatically adjust the torch height downwards. This will cause the torch to pierce the material and then drop down hitting the plate, and possibly damaging the shield and nozzle. So it is critical that the DTHC delay is set correctly.

This is programed into the tools in sheetcam and will be programed into the g-code if the correct tool is used. On thinner material one second is usually where this is set. This becomes more critical when you start cutting thicker material which takes longer to pierce. On thicker material it is better to have a longer delay than a shorter one, so do hesitate to increase this to 3 or 4 seconds. There is little
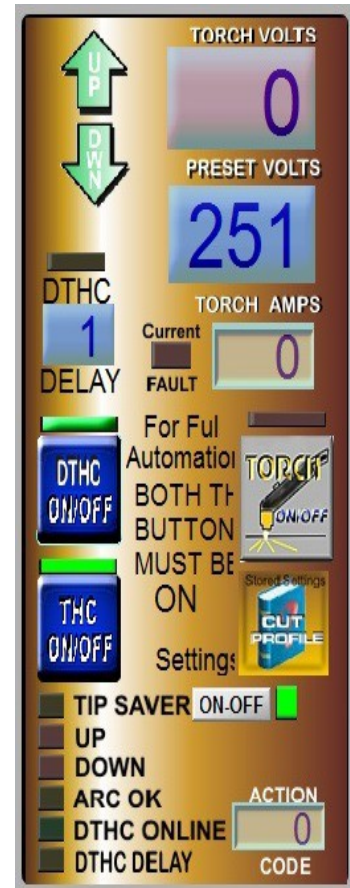
downside to this, as on thick material your travel speed is slower; therefore in that 4 seconds time you won't usually need the height control.

DTHC On/ Off

This is not a button, it simply tells you if the THC has been turned on or off by the G-code. If it is green the THC is on, if it is red it is off. This maybe red at the start of a file but will turn on once the cut is run.

THC On/ Off

This is the manual override for the Torch height control. It can be turned off before a cut or at any point in time during a cut. Flashing green means the THC is on. This is turned off during a dry run and while re-cutting lines.
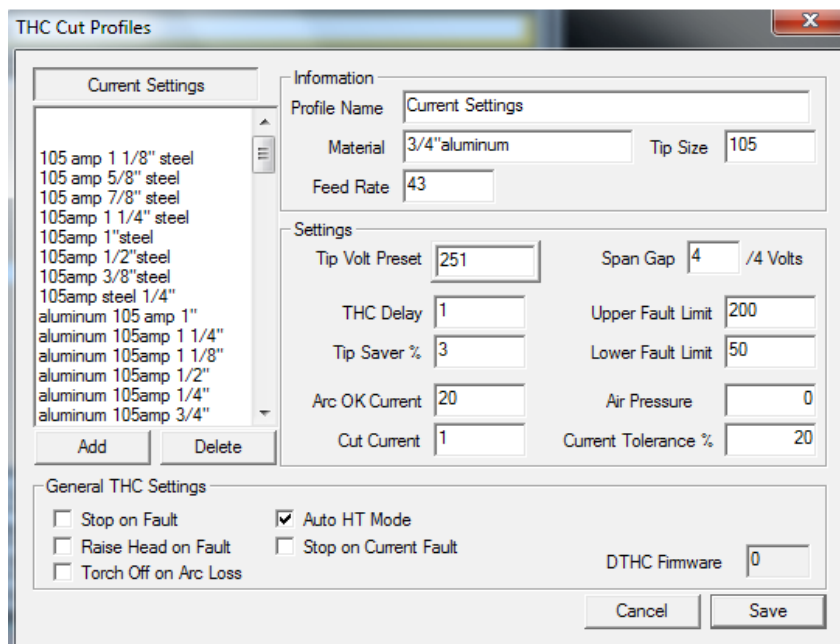
Torch amps and Current fault

Our standard system does not use these features, so you wont need to be concerned with them.

Cut profile window

This displays the settings held in the memory of the Torch Height Control card. Our current system sets the preset volts and DTHC delay in the g-code so this option does not need to be used. The information at the top including Name, material, feed rate and tip size is just information, **the feed rate can not be changed here**.

In the setting section you can change these. However it is not recommended that you change any of these factory settings.

Tip Saver

This is a anti-dive feature that prevents the torch height control from diving down during spikes in voltage. The percentage of this is set in the cut profile window and it is usually 3%. Basically if the Torch Volts spike 3% over the Preset volts, the the Tip Saver will lock the downward motion of the Z axis, preventing the THC from adjusting down into the plate.

Up and Down Indicators

These show you if you are receiving up and down commands from the THC.

Arc OK signal

This signal comes from the Plasma cutter and tells Mach 3 when the main arc has thrown and when to start motion. With the THC on the table will wait for this signal before motion is started. With the THC off the table will not wait for this signal before it starts motion. If the THC is on and the arc is lost for any reason during a cut the table will pause it's motion.

DTHC Online

This indicates that you have communication between the computer and the control system and should always be green.

Feedholds

Pausing during a cut

To pause the cut you'll use the feed hold button. There are two way the table can be stopped, a "Feedhold" and a "E-stop".   Both stop the table , but there is a important distinction between the two.

-A e-stop (using the e stop button or the stop button) stops all motion and turns off the torch abruptly, this can cause Mach3 to lose position and you will most likely not be able to finish the cut. So E-stops should only be used in emergency conditions.

- A feedhold allows Mach3 to finish moves it has already commanded and therefore the table will not lose position. When you feedhold the table will not stop right away. After you have feedheld during a operation it is a simple matter to continue the cut. So if you can always feedhold the table. Additionally you will always want to feedhold cuts during transfer moves (when the torch stops and moves up and over to another cut), if you feedhold on a cut Mach 3 will not turn off the torch, you'll have to do it manually. You also will not be able to easily start the cut back up.

Procedure for a Feed hold:

1. While the machine is doing a transfer move, as soon as the torch shuts off and starts to move up, hit the feedhold button .

   The table will pause. (If you need to check the consumables you can move the z axis up or down)

2. Hit the stop button, this will dump the buffer of motion that the table has in it's memory and prevent unnecessary motion.

3. Hit the run button and the table will continue cutting out the part.

You will want to become very comfortable using the feedhold to stop the machine when you have part tip ups. Adding additional slats to your table, decreasing the slat gaps to 1 ½" can help greatly in minimizing tip ups.   Slots for this spacing are provided and additional slats can be produced by shearing 11 gauges to the correct length and width.

Error Recovery

Machine stops at the pierce

This is usually due to a misfire of the plasma unit. This can be caused by bad consumables, low air pressure, bad air quality, or consumable placement. Follows these steps to restart the table.

1. Machine has stopped and Mach3 shows the torch is still trying to be fired and run button is green.

2. Hit the stop button, this will turn off the torch and pull mach out of the run mode and also clears mach's buffer of memory; preventing unnecessary motion on restart.

3. Check the plasma cutter power unit to see if there are any faults, if there are fix them accordingly.

4.   Rewind the G code to the previous M05 code that is above the current line using the arrows in g-code window. (Place the M05 in the white line) The M05 code commands Mach 3 to turn off the torch, and so represents the end of the last cut. Additionally, the M03 code commands Mach to turn the torch on, and the code in between these m codes is the transfer and touch off sequence. So we always want Mach 3 to have the chance to read through the transfer/ touch off code before it fires the torch.

5. Click the "set Next Line" button below g code window.

6. Click "Run"

7. The machine will move the z axis up to 1.5" and then down and go through the touch off sequence and fire the torch, continuing on with the file.

Practice this sequence till you have the steps memorized. To practice this error you can load up a code, run it and after one of the cuts (while the table is transferring to the next cut) turn off the plasma cutter units power. This will simulate a misfire on the pierce. You can then go though the sequence listed above to practice.

It is important to understand that Mach 3 Reads everything below the white line in the g-code window. So if the command you need is above that white line mach 3 won't read it. The M03 and M05 tell Mach 3 to turn the plasma torch on and off respectively.   So these are used as landmarks in the g code telling us where to start from. If you have a error on a cut or at the start of the cut you can rewind past the M03 to the M05 and use the set next line button to tell Mach to start from here. The reason we recommend you go past the M03 and start from the M05 is because you want Mach3 to read the code in

between them. This code is the transfer move and the touch off sequence (torch drops down and touches off the plate). And by allowing Mach 3 to read this before we restart a line or rewind a line, we ensure Mach 3 is in the correct position and can go through the touch off sequence to get the correct pierce Height.

Recovering from an error in the middle of a cut

If while the table is cutting there is a fault or error the plasma will shut off and mach 3 will pause. This errors can be caused by bad consumables, low air pressure, voltage faults, and poor work clamp connections. Follow the steps listed below to recover this error and finish the cut.

1. Hit the stop button, this will turn off the torch and pull mach out of the run mode. It also clears mach's buffer memory and stops unnecessary motion on restart.

2. Check the plasma cutter power unit to see if there are any faults, if there are fix them accordingly.

3. Rewind the G code to the previous M05 code using arrows in g-code window.

4. Hit "Set Next Line" button below g-code window.

5. Turn off the THC Button (Bottom THC button). If the THC is left on while recutting the line the voltage will not be correct and the torch will run closer to the plate than it should, so THC should be off while cutting over lines that have been already cut.

6. Hit Run, Machine will return to the start of the line that had the error. It will proceed through the touch off sequence, fire the torch and start cutting the line.

7. When the torch approaches the spot where the cut stopped you can turn the THC back on (with the bottom THC button) once the torch hits new material.

This sequence can also be practiced by turning off the power on the torch in the middle of a cut.

Fast forwarding and rewinding through the code

You can use a very similar process to fast forward past part of a cut or rewind back to part of a cut.

1. Determine where in the code is the cut you want to move to (forward or back).

2. Move to the previous **M05** (end of the cut before it)

3. Hit **Set Next Line** and then hit the **Run.**

4. Your table will move back or forward in the code and begin from there.

The Set Ref button is located below the g-code window. Click this save the current DRO readouts on the Settings Page. This feature can be used in many ways, most important of which is to ensure that you can recover a cut after a E-stop. To use this feature follow the directions listed below.

1. Prior to setting a large cut or nest use the Ref X and Ref Y buttons to bring the table to the Machine zero. Be careful that the torch does not strike the material stops while doing so.

2. Zero out the X Y and Z axises.

3. Now move the torch to the location on the material where you are going to start your cut.

4. Now hit the Set reference point button, This saves your current X Y and Z positions in a field on the settings page. By saving these coordinates this ensures that if we ever loose steps due to a e-stop or the gantry hitting something that stops it's motion, we can than get back to the start of the cut accurately.

5. Once you have moved to where the cut will start, zero out the x y and z axises as you normally would.

Now if you have a e-stop of the gantry stalls and you loose steps you can now use the steps below to regain those lost steps and get back to the zero point of your cut.

1. Raise your z axis up to give it clearance and use the Reference y and X buttons to move back to the Machine zero.

2. Zero out the X Y and Z Axises.

3. Jog you table close to where you think your start point was, within 5"

4. Hit the Go to reference point button, and you table will move to the saved location, which is your part zero.

5. Zero out your X Y and Z axises and now fast forward though the code till you get to the spot you had a problem at and use the Steps for fast forwarding though the code to start from there.