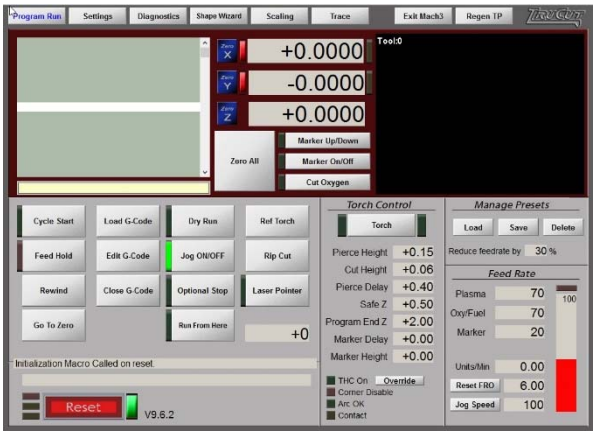
Operator Manual

SeriesOne and XT Machines

Machines shipped after 3/30/2017

TABLE OF CONTENTS



Code Window *1.1.1*

The code window in Mach3 displays the G-Code that is currently loading into Mach3. Use the scroll bar beside the code window to scroll the code up and down. To rewind a program to the beginning to cut the job, use the rewind button.

MDI Code Window *1.1.2*

The MDI code window allows the operator to enter a line of G-Code manually. For instance, if you were to type G00X10 into the MDI code window followed by the enter key, the machine would make a rapid move 10" in the X axis.

Position DRO's *1.1.3*

Position DRO's display the current X and Y coordinates. The Z axis DRO is only functional while the machine is either not cutting or the THC is disabled. While the THC is enabled, the Z axis DRO will not reflect motion commanded by the THC.

Zero Axes Buttons *1.1.4*

Use the Zero axis buttons to zero the machine position prior to each cut. Pressing the Zero All button zeros all axes. Pressing the blue zero button beside each DRO will zero just that axis.

Aux Outputs *1.1.5*

The aux output buttons control auxiliary functions including Marker Up/Down, Marker On/Off and Cut Oxygen On/Off (for machines equipped with oxy/fuel cutting. Pressing the button toggles the output.

Toolpath Display *1.1.6*

The toolpath display shows the file that is currently loaded to run. If after using Shape Wizard, you notice that the toolpath display on the Run screen did not update, press Regen TP located above the toolpath window. Rapid moves are displayed in red, G1 feed rate moves are displayed in blue and G2/G3 arc moves are displayed in magenta.

Cycle Start *1.1.7*

The Cycle Start button is used to start the job. To start a job, press Zero All then Cycle Start.

Feed Hold *1.1.8*

Feed Hold is used to pause a job. Feed Hold is a controlled stop. It does not stop the machine on a dime. A better option is to use Optional Stop.

Go To Zero *1.1.9*

Pressing Go To Zero will make the machine move back to the point where it was last zero'd (The last position Zero All was pressed).

Load G-Code *1.1.10*

Press Load G-Code to load the G-Code file. The G-Code file will have a .TAP extension. No other file type will work. The .TAP file loaded here should be created using SheetCAM software that came with

your machine. If you try loading a file and cannot find the file in the file dialog, it most likely because you have not created a .TAP file in SheetCAM or you are trying to load a .DXF (drawing) or .JOB (SheetCAM job) file.

Edit G-Code ***1.1.11***
Pressing this button opens the loaded G-Code file in Windows Notepad, where the program can be edited. Saving the file and closing Notepad reloads the program into Mach3.

Close G-Code ***1.1.12***
Pressing Close G-Code closes the file that is currently loaded.

Rewind ***1.1.13***
Pressing Rewind rewinds the program to the top. If the program is stopped mid-way for some reason and you want to restart the job from the beginning, you must press Rewind prior to re-running the program.

Dry Run ***1.1.14***
Press Dry Run to run a job with no Z motion or torch activity. This is a good way to reality check a program to see if it will fit on a sheet or just a good way to practice running the machine without wasting material.

Jog On/Off ***1.1.15***
This button enables jog mode. When enabled, the green LED beside the Jog button will be lit. Jog mode can be left enabled all the time.

Optional Stop ***1.1.16***
When enabled, Optional Stop will make the machine pause at the end of every cut. This is the preferred way to pause the machine as it is a controlled stop at a known location. If you press Optional Stop in the middle of a cut, the machine will finish the cut, raise to Safe Z then pause. To resume, press Cycle Start. As long as Optional Stop is active, the machine will pause after every cut. Turn off Optional Stop to resume normal operation.

Run From Here ***1.1.17***
This feature allows you to start a job in the middle of the job. For instance, if you have a plasma fault in the middle of a job, you can use Run From Here to re-start the job just after where the fault occurred. To use Run From Here, follow these steps...

1. *Make note of the program line number that the job faulted on.*
2. *Press Rewind. This will rewind the program to the beginning.*
3. *Using the scroll bar in the code window, scroll the program to the line number noted in step 1. The current line number will be displayed beside the Run From Here button. Once at that line number, scroll a little further until you see RUN FROM HERE START POINT. Make sure this like is highlighted in the code window.*
4. *Press Run From Here. The program will run through the code up to the line you selected in step 3.*
5. *Note the message above the Reset button. Press Cycle Start. One the Preparation Move dialog, press OK. The machine will move to the start point highlighted in step 4.*
6. *Press Cycle Start again to run the program from this point.*

Ref Torch ***1.1.18***
Pressing this button will make the torch go down, sense the material and go back up to the height set in Pierce Height. This is not necessary on every job. You only need to do this when calibrating the pierce height or checking that pierce height is still in calibration.

Rip Cut *1.1.19*

Rip Cut allows you to make straight line cuts in the X and Y axes. This is great for remnant cutting or just cutting off a sheet. In the Rip Cut dialog, enter a length. This can be a negative or positive number. -10 will go 10" in the – direction while 10 will go 10" in the + direction. Next, select an axis and a process. At the time of this writing, the only process supported is plasma. Oxyfuel and marker processes will be added. Enable THC is no longer used. To enable the THC, turn it off on the THC remote.

Torch (On/Off)..... *1.1.20*

This button controls the cutting torch. This is a toggle button. If the torch is off, pressing this button will turn it on. If the torch is on, this button will turn it off.

Pierce Height..... *1.1.21*

Pierce Height is the height at which the torch will retract off the material before it lights. For this to be accurate, the pierce height must be calibrated and stay in calibration. We recommend checking pierce height every now and then. See Chapter 5 – Calibration, Section 5.2.

Cut Height..... *1.1.22*

Cut Height is not used by our new height control. Cut height is controlled completely by the THC and is relative to the set voltage set on the THC remote.

Pierce Delay *1.1.23*

Pierce Delay is the amount of time in seconds that the machine will pause to allow the plasma jet to pierce all the way through the material. In most cases, up to about 1/8" material, this can set to 0 (zero). Other than that, use the pierce delay settings in the plasma manual or select the correct process from the presets under Manage Presets. See Section 1.1.32 in this chapter.

Safe Z *1.1.24*

Safe Z is the height that the torch will raise off the material while traveling from one cut to the next. For problem parts that tend to flip up in the grid, you can raise Safe Z to clear the parts.

Program End Z *1.1.25*

Program End Z is the height that the torch will raise off the material at the end of the program to travel back to zero.

Marker Delay *1.1.26*

This applies to machines with plasma marking and sets the delay between when the torch is fired and when the machine starts to move. 0.6 – 1.0 seconds seems to be the average.

Marker Height..... *1.1.27*

This also applies to machines with plasma marking and sets the height of the marking torch off the material.

THC On/Override (LED) *1.1.28*

This LED is lit when the THC has been enabled. The Override button allows the operator to disable or enable the height control. This is a toggle button. If the THC is enabled, pressing Override will disable it. If the THC is disabled, pressing override will enable it.

Corner Disable (LED) *1.1.29*

This LED is lit when the actual feed rate is not equal to the commanded feed rate. When the LED is lit, the THC is disabled. This prevents the THC from diving when the feed rate slows down for corners. It also automatically locks out the height control for small holes, which can produce better hole quality.

Arc OK (LED) *1.1.30*
This LED is lit when the plasma has established an arc transfer. It is triggered by a circuit inside the plasma power supply.

Contact (LED) *1.1.31*
This LED is lit when the contact switch on the floating head makes contact after the torch touches the material. You can lift the torch by hand until it touches the switch and see the Contact light come on. If this light becomes stuck on, the torch will not reference. Check the wiring to the switch or the switch itself.

Manage Presets - Load *1.1.32*
Pressing Load will open a dialog box listing the entire cut chart for the Hypertherm plasma. Select the process you wish to use, then press OK. Another dialog will appear prompting you to set the THC voltage on the THC remote.

Manage Presets - Save *1.1.33*
Pressing Save will prompt you to enter a name and save a new process with the screen parameters that are currently displayed.

Manage Presets - Delete *1.1.34*
Pressing Delete will show a list of processes. To delete a process, select the process to delete and press OK. A prompt will confirm that the process was deleted.

Manage Presets – Feed Rate Reduction *1.1.35*
The cut speeds straight from the cut charts are too fast most of the time. This feature allows you to set a reduction percentage for the feed rate. For instance, if you select a process that has a feed rate of 150IPM and you enter a reduction of 20%, the system will load a feed rate of 120IPM.

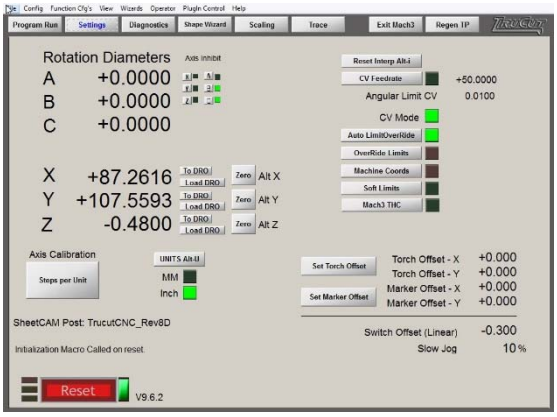
Feed Rate DRO's/Feed Rate Override *1.1.36*
Feed rate is set by entering a number in the feed rate DRO's. Each process has its own feed rate DRO. If you are running the plasma process and want to cut at 30IPM, enter 30 in the DRO beside Plasma.

Feed rate override can be used to change the feed rate on the fly. FRO is displayed as a percentage. For instance, if the feed rate is set to 200IPM and the FRO is set to 50, the feed rate would be slowed to 50% of 200, or 100IPM. Drag the FRO slider to change the FRO value. To reset it back to 100%, press the Reset FRO button.

Jog Speed *1.1.37*
The Jog Speed button toggles jog speed from 100% to a value entered by the operator. If you enter 10 in the Jog Speed DRO, this would lower the jog speed to 10% of the set rapid speed of 500IPM. Pressing the Jog Speed button would toggle between 100% and 10%. You can also use the "5" key on the keyboard keypad to toggle Jog Speed.

Laser Pointer *1.1.38*
Pressing the Laser Pointer button turns on the laser pointer mounted on the torch mount. It will automatically shut off if you E-Stop the machine. The LED beside the button reflects the state of the LED (on or off).

Note: If the machine is E-Stopped, the laser will shut off, but the LED beside the button will not. After clearing the E-Stop, pressing the Laser Pointer button again will re-sync the LED with the state of the laser.



Axis Calibration *1.2.1*

Axis Calibration is used to calibrate the steps per unit so that when the machine is told to move 1" it does move 1". If you notice that parts are coming out a little large or small, you should calibrate the X and Y axes. See Chapter 5 – Calibration, Section 5.1 for complete details.

Units *1.2.2*

This displays the units that the software is currently set to. This is set automatically in the GCode program. Do not change units here.

Auto Limit Override *1.2.3*

When enabled, this allows the operator to jog off a tripped limit at a reduced speed. If this is disabled, the machine will not jog off a tripped limit. Auto Limit Override should always be enabled.

Override Limits *1.2.4*

Allows the machine to jog off a tripped limit if Auto Limit Override is disabled.

Machine Coords *1.2.5*

Places the machine in machine coordinate mode. We do not use this mode. We always refeference work coordinates. Machine Coords should never be enabled.

Soft Limits *1.2.6*

Soft limits should never ne enabled. It is here only to verify that it is disabled.

Mach3 THC *1.2.7*

Mach3 THC must always be enabled when cutting. If disabled, Mach3 will not automatically sense Arc OK. Mach3 THC is enabled/disabled automatically in the GCode program. There is no need to do anything with it here.

Tool Offsets *1.2.8*

Tool Offsets can be set to zero or they can be set in reference to the laser pointer. If set to zero, you would position the torch at the point that you wish to start. If set in reference to the laser, you would position the laser at the desired start point and when the program runs, the tool offset is applied.

There are two tool offsets, Torch and Marker. If using the marker, Marker Offset must be set. It can be set relative to the torch or the laser. If you set the Marker Offset from the laser, then you must also set the Torch Offset from the laser as well. You must then use the laser to position the program start point. If you would rather not use the laser for program start positioning, you can set the Marker Offset from the torch and leave the torch offset set to zero.

For detailed instructions on setting Tool Offsets, see…

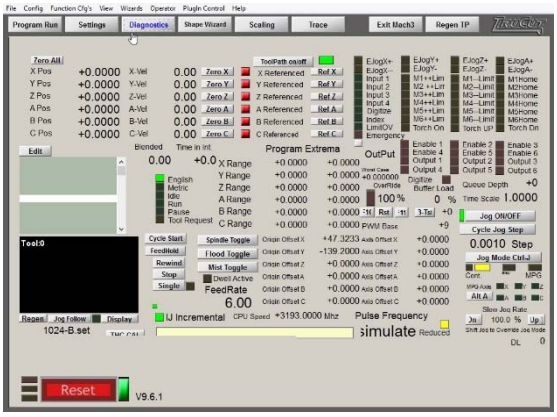
- *Chapter 3 – Basic Operation, Section 3.8 and…*
- *Chapter 4 – Plasma Marking System, Section 4.2*

Floating Head/Ohmic Switch Offsets *1.2.9*

These offset numbers represent the distance between where the torch touches the material and where the floating head switch makes contact. This number is usually around .5 - .6 and must be calibrated. See Chapter 5 – Calibration, Section 5.2 for detailed instruction on Pierce Height calibration.

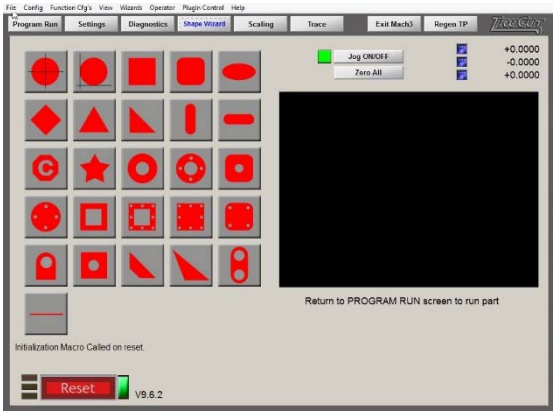
Slow Jog *1.2.10*

This is the same number displayed on the Program Run screen beside the Jog Speed button. It exists here for redundancy.



General Usage..... 1.3.1

The Diagnostics screen displays various LED's and DRO's useful for troubleshooting. This screen has nothing useful for routine machine operation. When remoting in on tech support calls we will many times look at this screen to troubleshoot an issue.



The part will show in the tool path display window.

General Usage *1.4.1*

The Shape Wizard is a part generator that creates various parts easily and quickly by simply entering a few parameters for the chosen part.

Part Buttons *1.4.2*

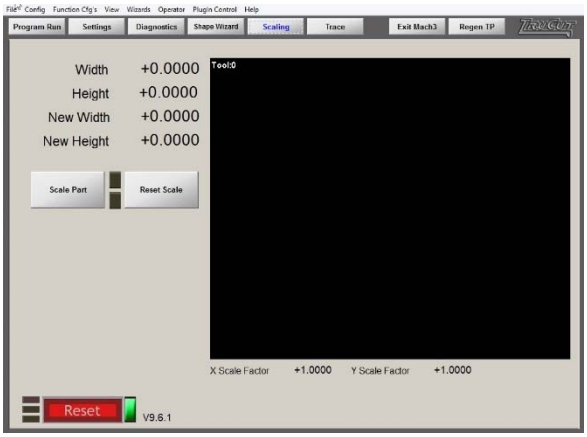
The Part buttons are used to select the shape you wish to cut. Each part will have a slightly different interface depending on the particular parameters of that part. Enter the part parameters on the dialog that that appears after pressing the part button and press OK to create the part.

Tool Path Display *1.4.3*

The tool path display window shows the part that was created.

Running a Shape Wizard Part *1.4.4*

To run the part shown in the tool path display window, you must return to the Program Run screen. If the part does not show correctly in the toolpath display window on the Program Run screen, press the Regen TP button above the tool path display window. This will refresh and regenerate the programsm in the window.



The Scaling screen can be used to scale a part up or down or to check the size of a part loaded in GCode. This method of scaling is not precise. For precise scaling, the part should the scaled in your CAD software. Also, keep in mind that when you scale a part, everything is scaled, including lead-ins/lead-outs and kerf widths.

For more information see Chapter 3 – Basic Operation, Section 3.8.

These DROs show the X and Y dimensions of the currently loaded part.

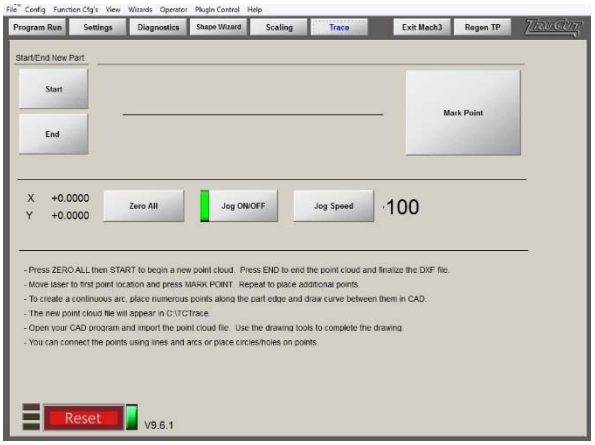
To scale a part, enter a new Width or Height. Enter only a Width OR a height, not both. The program will scale the part and determine a scale factor based on the original width or height vs the new width or height.

Press to scale the part.

Press to return the part to its original size, or 1:1 scale factor.

This is the calculated scale factor. 1 = original size. .5 = half size. 2 = double size.

Shows the current toolpath. Note that scaling will not make a noticeable difference in the tool path display window.



The Trace Wizard is a point cloud generator. It creates a DXF file that can be opened using your CAD software. This DXF file will consist of points placed at locations marked by the operator. You can use the point cloud file as a guide to complete the drawing by connecting the points using lines and arcs.

For more information, see Chapter 3 – Basic Operation, Section 3.9.

Press the Start button to initiate the point cloud file. Enter a name for the file when prompted.

Press the End button to finalize the point cloud file. A prompt will appear showing where the file was created

.

Press Mark Point to mark a point. A dialog will appear showing the coordinates at which the point was created.

Corner Anti-Dive Delay..... 2.2.5

Corner Anti-Dive monitors the machine speed and compares the actual speed to the commanded speed. If the actual speed is less than the commanded speed, Corner Anti-Dive locks the THC to prevent the torch from diving into corners. If we engaged this feature immediately, the THC would not enable until the machine reached commanded speed. Corner Anti-Dive Delay adds a delay so the THC can enable and drop to cut height in a timely manner. 1-2 seconds works well and should be at least ½ second more than Puddle Jump/THC On Delay.

Manual Height Control 2.2.6

This screen allows you to control the torch height manually while cutting. To use this feature, the THC must be turned off on the Home screen. This feature is particularly useful for those using an Oxy/Fuel torch where no voltage based automatic height control is available.

Basic Operation..... 2.2.7

For the height control to operate properly, you must first set the target voltage, or set voltage. This is done using the up and down arrow keys on the home page. There are three ways to determine a starting point for the set voltage.

1. *Open the Hypertherm manual that came with your plasma and look at the cut chart for the amperage and consumable you are using. Look under Best Quality Settings and enter the voltage from the chart on the THC remote.*

2. *Select the desired process from Manage Presets by pressing Load. This will load process parameters and display the correct set voltage for the process.*

3. *Turn off the THC from the home screen and cut a small sample part. Pay attention to the Tip Volts on the remote. Set the set volts to the same number. This will get the height close. After you start cutting you can dial it in.*

Once you have a starting point for the set voltage, you can dial it in closer while it's cutting. As you change the set volts while cutting, the height will respond instantly. ***If the height looks too high, lower the set volts. If it looks too low, raise the set volts.***

Once you have it dialed in, make a note of that number for the amperage and material you are running. After a few jobs, you'll remember the set volts, and if you notice it is off a little, you can adjust it on the fly without ruining the job. In many cases, the book settings are accurate and others they are not.

Pierce height has little effect on the cut height since cut height is controlled solely by arc voltage. If the pierce height is off when the cut begins, the height will move rapidly to the cut height determined by the set volts entered on the home screen when the THC enables.

Saving THC Settings 2.2.8

Any parameters that you change on the THC remote control will be automatically saved to non-volatile memory when you cycle the THC from on to off. If you power down without turning the THC off on the remote, the parameters that were last saved will be loaded on start-up.

NOTE: *The DL05 PLC EEPROM memory has a maximum limit of 100,000 writes. That means that after 100,000 save cycles, the PLC memory may become unreliable. Under normal, or even heavy use, this limitation should not be an issue.*

While the PLC is saving data, the remote display will show a ***PLC COM Timeout*** *error. This is normal.*

If you would like to limit the number of EEPROM writes to the PLC, just power down the controller without cycling the THC to off. This works great if you cut the same material most of the time.

CHAPTER 3 - BASIC OPERATION

STARTING MACH3 ..... 3.1

The start-up sequence for Mach3 is important. If Mach3 is started before the machine controller is turned on, it will generate an error. Also, if the machine controller is turned off before Mach3 is shut down, an error will occur. To avoid this, always turn on the controller first THEN start Mach3. When shutting down, shut down Mach3 first THEN turn off the controller.

If you make a mistake and see an error, shut the controller off, shut down Mach3 and start over using the correct sequence.

JOGGING THE MACHINE 3.2

The machine is usually jogged by using the keypad on the keyboard. Keys 4 and 6 jog the X (long) axis, Keys 8 and 2 jog the Y (short) axis and keys 9 and 3 jog the torch up and down. There is also a graphical jog screen that can be accessed by pressing the TAB key.

Note: If the machine does not jog via the keypad, check that jog mode is turned on (green LED beside the Jog button) and that the NumLock key on the keyboard is on. Some keyboard have a numLock and some do not.

LOAD A JOB 3.3

Press Load G-Code to load the G-Code file. The G-Code file will have a .TAP extension. No other file type will work. The .TAP file loaded here should be created using SheetCAM software that came with your machine. If you try loading a file and can not find the file in the file dialog, it most likely because you have not created a .TAP file in SheetCAM or you are trying to load a .DXF (drawing) or .JOB (SheetCAM job) file.

RUN A JOB 3.4

To run a job, Position the torch or the laser, depending how you have tool offsets configured, to a zero point on the material. Press Zero All then press Cycle Start. The program loading in the G-Code window will run.

PAUSE A JOB 3.5

Pausing a job can be done one of two ways…

1) *While the machine is running, press Feed Hold. This is a controlled stop. The machine will not lose position after pressing Feed Hold. To restart, press Cycle Start.*

2) *Press Optional Stop and let the machine finish the current cut. When it finishes, it will pause before moving to the next cut. This is the best way to pause the machine.*

SETTING FEEDRATE 3.6

The SeriesOne and XT machines do not use a hard coded feed rate in the G-Code. Instead, feed rates are set on the machine in Mach3. In the Feed Rate section toward the lower right hand corner of the Program Run screen, you will find three feed rates…Plasma, Oxy/Fuel and Marker. If running a plasma process, you would set the plasma feed rate. Oxy/Fuel and Marker feed rates would set speeds for those processes. The process is set up in SheetCAM.

To change a feed rate, you can type in a new one at any time, however, the new speed will not update until the next cut. To change a feed rate on the fly, use the red Feed Rate Override slider. The number above the slider represents the FRO percentage. IE: 100 means the the machine is running at 100% of the commanded feed rate. To change the FRO value, drag the slider up or down. To reset it back to 100%, press the Reset FRO button.

This section will go over setting tool offsets for the cutting torch. If you have the Plasma Marking System, see Chapter 4, Section 4.2 for more information.

Setting the laser offset means that you must use the laser dot to zero the machine before every cut. If you would rather just use the torch head itself as the reference point, skip this step and ensure that both Torch Offset and Marker Offset are both set to zero on the Settings page.

To set up the torch offset, follow these steps…

1) *Find a clean spot on a piece of material, drop the torch to 1/8" off the material and light the torch briefly piercing a small hole.*

2) *Press Zero All.*

3) *Jog the machine so the laser drops into the hole you just pierced.*

4) *Go to the Settings screen and press the button called Set Torch Offset. Leave Marker offset set to zero. If you have the Plasma Marker System, see Chapter 4, Section 4.2.*

This process sets the cutting torch offset relative to the laser.

Emergency Stop can be achieved one of two ways…

1) *Press the Reset button in Mach3 in the lower left corner of the screen.*

2) *Push one of the red Emergency Stop buttons on the gantry side covers.*

Either method will stop the machine. Note that after an E-Stop, the machine will most likely lose its position. The faster it was moving when it was E-Stopped, the more likely it will be that position has been lost. If you just want to pause the program and it is not an emergency, use Optional Stop instead. See Chapter 1, Section 1.1.16 for more on Optional Stop.

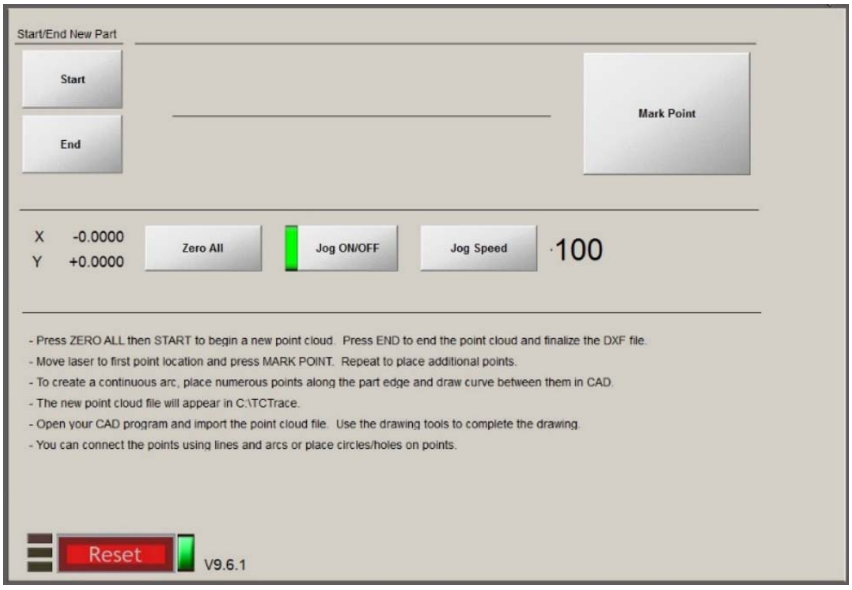
To scale a part, navigate to the Scaling screen. Height and width display the current size of the job. To change that size, enter either a New Width OR a New Height. Do not enter both a new width and new height. Enter only one and leave the other set to zero.

To scale the part, press the Scale Part button. The part will scale to the size entered. The scale factor will be displayed below the tool path display window. To revert back to a 1:1 scale, press Reset Scale.

Note: When a part is scaled, two yellow LED's will flash beside the X and Y DROs on the Program Run screen.

The Trucut Tracing Wizard is a utility built into the controller that allows the operator to place points around a part on the table. This is called a point cloud. The operator would place points at key locations on the part, such as a corner, a hole center, etc.

The Tracing Wizard is accessed by pressing the TRACE button at the top of the Mach3 screen and brings up a screen like the one below.



Instructions are included on the Trace screen, but we'll go over the process here step by step.

1) *Press Zero all. This aligns part zero to the current machine location. The trace will still work if you miss this step, but zero will be offset, which can be corrected in your CAD software.*

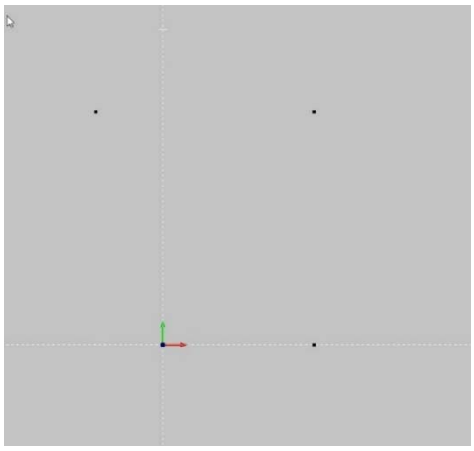
2) *Press START and choose a name for the new point cloud file. Do not enter an file extension. For instance, enter "FILE1" not "FILE1.DXF". The point cloud file will be saved as a DXF file and the extension will be automatically added.*

3) *Move the laser or camera, if so equipped, to the first point on the part and hit MARK POINT. This creates a point at the current location.*

4) *Move to the next point and repeat. Repeat this process until the part has been laid out.*

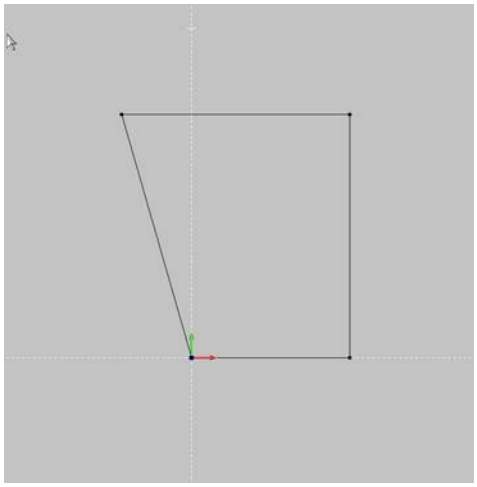
5) *Press END. This writes the file footer and closes the DXF file. The new file will be in C:\TCTrace. This completes the Mach3 portion of the Trace Wizard. We will now open the point cloud file in BobCAD.*

6) *Start BobCAD, or your preferred CAD program and open the point cloud file you just created. It will look something like what is pictured below.*

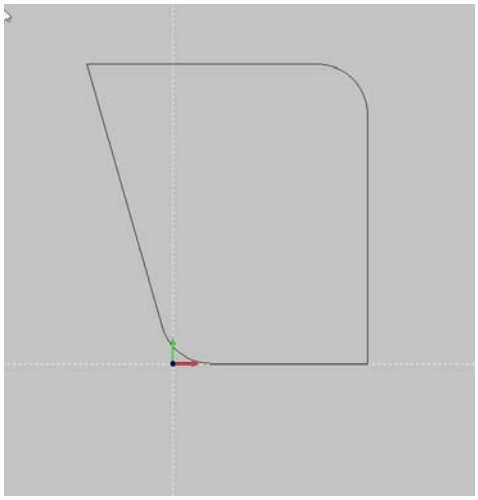


You can now use the points as a guide to complete the part in CAD. Connect the points using lines and arcs.

You can also mark larger holes using three points, then in CAD, crate a circle through 3 points. To make a hole using a single point, place the point at the center of the hole, then in CAD, create a circle on that point.



You could end up with something like this.



Now you can add details such as corner radii, like this. After the part is drawn, the points are no longer needed, so you can delete them.

Once the part is done to your satisfaction, save it as a DXF file, import into SheetCAM and cut.

Limit switches are there to prevent the gantry, lifter carriage or Z slide from traveling beyond its limits. Limit switches are configures normally closed in series so if there is a problem with the wiring and the loop opens, a limit will trigger. If this happens, you must track down the offending wire(s) with an ohm meter. Limits can be disabled, but this is not recommended.

Note: If you disable limits and the Z axis over-runs the limit, the coupler connecting the Z axis motor to the leadscrew can be damaged.

*This may not be as intuitive as one may think. Many of the DRO's on the Mach3 screen, such as feed rates, cut height, pierce height, etc. require that you enter a value. To enter a value, you must first click on the DRO so it highlights, then enter a value. After entering a value, you **MUST** press the Enter key to lock in the value. If you navigate away from the DRO without pressing Enter, the value will revert back to what it previously was.*

CHAPTER 4 - PLASMA MARKING SYSTEM

How It Works 4.1

Plasma marking has been around for years but only available on expensive hi-definition plasma machines. All of that changed when Hypertherm came out with their Powermax 45XP. The 45XP can operate at very low current setting (as low as 10A) and still use a contact arc start method.

Our plasma marking system includes a new Powermax 45XP with a machine torch. The marker torch is mounted beside the cutting torch on an air lifter. When the marker operation is set up in SheetCAM, commands are programmed to lower the marker torch for marking and raise it for cutting.

The marker torch must be set up in Mach3 with an offset. This offset either….

1) *Tells the software the X and Y offset of the marker torch from the laser, or…*

2) *Tells the software the X and Y offset of the marker torch from the cutting torch.*

If you set a marker torch offset relative to the laser, then you must also set a cutting torch offset relative to the laser. You will then use the laser to set your zero before running a job.

If you set a marker torch offset relative to the cutting torch, then you would leave the cutting torch offsets at zero and use the actual cutting torch position to set zero before running a job.

Setting Tool Offsets 4.2

to set the marker tool offset, first figure out if you want to use the laser all the time to reference zero on the machine. If you do, follow these instructions…

1) *Find a clean location on a piece of material and jog the torch to that location.*

2) *Lower the cutting torch to about 1/8" off the material, press Zero All and press the torch button briefly to pierce a small hole.*

3) *Make sure the position DRO's are zero'd out. If they are not, press Zero All now.*

4) *Jog the machine so the laser dot falls into the hole you just pierced.*

5) *Go to the Settings screen and press the button called Set Torch Offset. This will set the offset distance between the laser and cutting torch.*

6) *Repeat steps 1 – 5 but instead of firing the cutting torch, use the marking torch. Note that when doing this with the marking torch, the torch may not pierce all the way through. Just put the laser dot on the mark.*

7) *On the Settings screen, press Set Marker Offset. This will set the offset distance between the laser and the marker torch.*

If you don't want to use the laser at all, follow these steps…

1) *Lower the marking torch by pressing Marker Up/Down, light the marking torch by pressing Marker On/Off and make a visible mark.*

2) *Press Zero All.*

3) *Raise the marking torch by pressing Marker Up/Down again. Jog the machine so the Cutting torch is centered on the mark you made with the marking torch.*

4) *Go to the Settings screen and press the Set Marker Offset button. This will set the marker torch offset relative to the cutting torch.*

The cutting torch and marking torch must be set up so that when the marker torch is in its down position, it is about 1" below the cutting torch. To do this, you simply move the torched up or down in the spring mount.

The photo to the left shows the marker torch in the up position.

This photo shows the marker torch in the down position. To raise and lower the marker torch, press the button called Marker Up/Down.

CHAPTER 5 - CALIBRATION

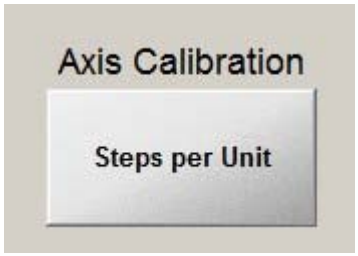
CALIBRATING X AND Y AXES 5.1

Calibrating the axis will be necessary if one or both axes are not cutting the length that they are commanded. Follow these steps to calibrate the axis

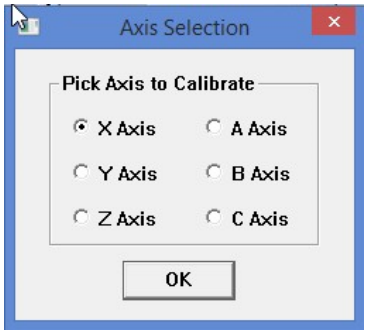
1) *Jog X axis so the gantry is 10" from the end of the rail.*

2) *Press "Zero All" on the Program Run screen.*

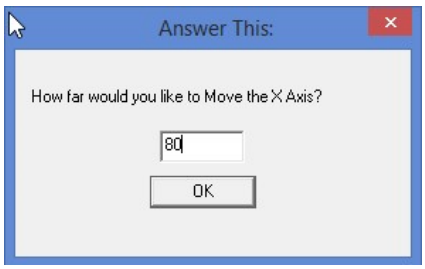
3) *Go to the Settings screen and press the button labeled "Steps Per Unit" under Axis Calibration.*



4) *Select the axis you wish to calibrate.*

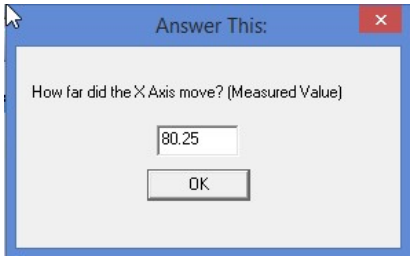


5) *Enter the distance you would like the X axis to move. The longer you move, the more accurate the calibration will be. 80 works well for a 4x8 machine and 100 works well for 5x10 and 6x12. Press OK.*

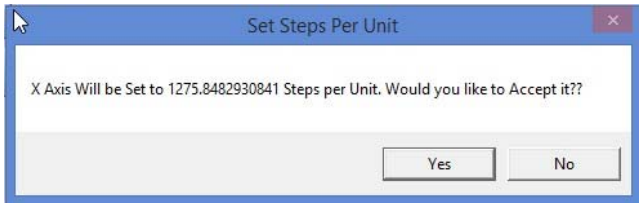


6) *If the axis is calibrated correctly, it will move to the distance you told it to move. Re-measure from the same points that you previously measured and note that measurement. The actual distance moved till be this measurement minus 10". If you told it to move 80" and you are now measuring 90.25", you moved 80.25".*

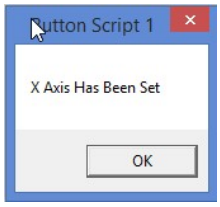
Enter the distance the gantry actually moved. Again, this will be measured distance – 10".



7) *The system will now calculate the required number of steps per inch/mm to move exactly the distance commanded.*



8) *Mach3 confirms that the axis has been set.*



9) *Return to the Program Run screen and re-start Mach3. This writes the values to the Mach3 configuration. Once Mach3 has been restarted, you can repeat the same process on the Y axis. 3" works well as a starting point on the Y axis.*

Repeat steps 2 – 9 on the Y axis. 40" works well for a 4x8 machine. 50" works well for 5x10 and 6x12. Restart Mach3 again after the Y axis has been set. You must restart Mach3 after each axis.

In order for Pierce Height to work properly, the software needs to know the distance that Z travels after it touches and makes contact with the floating head switch. To do this, follow these steps….

1) *Check that the floating head switch is connected and working properly by lifting up on the torch so it makes contact with the floating head switch mounted above the floating head. When this switch makes contact, the yellow LED labeled Contact should come on.* ***If it doesn't, do not continue. Automatic referencing will not work.***

2) *Set Pierce Height to .25" and find a piece of ¼" material you can use as a feeler gauge. We use a piece of ¼" cold rolled. A ¼" drill bit works well also.*

3) *Jog the torch over some fairly heavy material that will not distort when the torch touches off to it.*

4) *Jog the Z down to about ½" off the material and press Zero All.*

5) *Press "Reference Torch" on the Mach3 screen.*

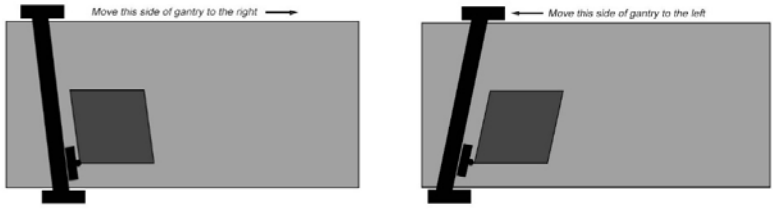
The torch should go down, touch off and come back up to .25". Use your ¼" feeler gauge to check the torch height. If it is too low, go to the Settings screen and increase Switch Offset.

For Instance, If the height is less than .25 and the current switch offset is -0.55, enter -0.65 and repeat the above process. ***Note that there must always be a – in front of the number.*** *If it is still not right, re-adjust the Switch Offset and repeat. This a trial and error process, but the good news is that once it's done you shouldn't have to do it again.*

To square the machine, you first need to cut a square and determine if it is square or skewed. If it is skewed, the machine is out of square. To square the machine, you will loosen the belt tension on one side of the machine. With the tension loose on the belt, you can skip teeth on the timing belt to move this side of the gantry fore or aft depending on your test square.

Measure the cut square corner to corner or put a framing square on it to determine which direction to move the gantry side. Again, for coarse adjustment more than 1/16" - 1/8", loosen the timing belt tension and skip a tooth. For fine adjustment, loosen one side of the drive chain and tighten the other. For instance, loosen the rear chain bolt 1/2 turn and tighten the front bold 1/2 turn.

See the graphic examples below. The skew in these images is exaggerated for graphic purposes. You will not be able see the skew by eye.



XT machines are driven independently on each side of the gantry. These machines have squaring stops on the rear of the table. To square the gantry do the following…

1) *Ensure that the racks are engaged.*

2) *If the machine is on and Mach3 is running, Jog the machine to the rear of the table close to the rear limit trigger pin. If the machine is not running and Mach3 is shut down, you can disengage the racks, move the gantry to the rear of the machine, then re-engage the racks.*

3) *Exit Mach3 and turn the machine controller OFF.* ***Ensure racks are engaged****.*

4) *Pull the gantry rearward until it rests against the squaring stops.*

5) *Turn the machine controller ON and start Mach3.*

6) *The machine will be resting on the rear limit. You should be able to job off the limit.*

Similar to the XT machines, the XTP machines have squaring stops at the rear of the table with each side of the gantry driven independently. The difference is in the controller. The XTP is equipped with the Victor iCNC control. To square the XTP, do the following…

1) *Jog the machine rearward close to the rear limit trigger pin.*

2) *Press Enable on the controller (just below the power button). This will disable the drives.*

3) *Pull the gantry rearward against the squaring stops.*

4) *Enable the drives and jog off the limit switch.*

CHAPTER 7 - SUPPORT

EMAIL 7.1

Email support is available at rcarlisle@trucutcnc.com.

PHONE 7.2

Phone support is available 9am – 6pm, Monday – Thursday and 9am – 12pm on Friday.
Call (615) 290-6260.

REMOTE 7.3

Remote support is our preferred method and in some cases, is the only way we can diagnose a problem. Remote support requires a high speed WiFi internet connection. All machine come set up ready to connect to your WiFi internet. Once connected, we can remote in for training and support.

On newer machines with the LCD THC remote, raise the set voltage on the remote. It set voltage it too low, the torch will drag. You can find set voltage in the manual for your plasma unit. It is usually called arc voltage in the manual. You can also select the desired process from the Presets. You will be prompted with the correct arc voltage setting. See Chapter 1, section 1.1.32.

On older machines without a remote, height is determined by the Cut Height setting on the Program Run screen. For cut height to be accurate, pierce height must be accurate. After the machine starts moving, a voltage sample is taken and the arc voltage for the THC is set to that number. The height that the torch is at when it starts moving it the height it will stay at. If the torch is dragging, make sure that there is an air gap under the torch when it starts moving. If it is dragging when it starts, that's where it will stay.

On newer machines with the LCD THC remote, If the torch is cutting too high, lower the set voltage on the THC remote. You can do this on the fly. As you lower the set voltage on the remote, the torch will go down.

On older machines without the LCD THC remote, this can be caused by the plasma arc voltage not stabilizing fast enough. This usually means a fault in the plasma unit or the torch. There is no way to lower the torch at this point.

Before getting into any troubleshooting, make sure the THC is not functional by cutting a circle from the Shape Wizard. While it's cutting, put your finger on the silver motor coupler on the Z lifter. You should feel movement. Even if the material is flat, you should feel slight movement. If you feel movement, the THC is working correctly. If you feel no movement, follow these steps….

1) *Make sure THC is turned on from the remote. From the home screen on the remote, press the On/Off button.*

2) *Make sure Set Voltage is set correctly.*

3) *Open the controller box and look for the DL05 PLC. This will be on the left side just below the large power supply. There should be two small LED lights lit on the right side with two other LED's flashing rapidly. The two on the top indicate power is getting to the PLC. The two below indicate that the remote is communicating with the PLC.*

   If you don't see any LEDS on the PLC, check the two white circuit breaker toward the top of the controller. One is labeled C2 and one is labeled C20. C2 feeds the Rhino power supply for the PLC. Both breakers should be showing a red window.

   If both breakers are ON, then the Rhino 24V power supply. It should have a steady light. If the light on the Rhino is flashing, this indicates a short circuit. Call for further assistance.

   If the Rhino power supply is showing a steady light and the PLC is still not showing power LEDs, the PLC may be faulty. Call for further assistance.

4) *Check that the THC has not been disabled in SheetCAM. THC can be disabled by setting specific operation variables.*

5) *If none of the above helps, call for further assistance.*

Torch dives at the end of a cut ***8.1.4***

This caused by increased arc voltage at the end of the cut when the torch travels over a gap or into a hole where there is no material. To reduce this effect, go to the Anti-Dive Sensitivity screen on the THC remote and lower the number. We default this number to 12. Lowering it will reduce the diving effect but will also make Anti-Dive more sensitive. On more extreme changes in height, anti-dive may enable when you don't want it to. If this happens, you can increase Anti-Dive Sensitivity or increase Motion Sensitivity by one.

THC will not track rapid changes in height ***8.1.5***

If the THC is crashing into the material rather than tracking it up, either the THC is not working (See this chapter, section 8.1.3) or the height is changing faster than the THC can respond. To increase the THC response time, go to the Motion Sensitivity screen on the THC remote and increase the number. Do not increase more than one number at a time. Increasing this too much may cause the THC to become unstable.

Note: Extreme changes in height often require Motion Sensitivity settings of 5 or 6.

If the torch is tracking up fine, but not tracking the material back down, go the Anti-Dive Sensitivity screen on the THC remote and increase the number. This will make Anti-Dive less sensitive and should fix the problem.

THC tracks material going up, but not down ***8.1.6***

This is usually because Anti-Dive Sensitivity is set too low (too sensitive). Go to the Anti-Dive Sensitivity screen on the THC remote and increase this number.

THC is not acting right and none of the above helped ***8.1.7***

The THC relies on stable arc voltage from the plasma. It is not uncommon for a fault in the plasma to cause this voltage to be inconsistent…or non-existent. This can be determined by checking voltages coming in from the plasma, but this is beyond the scope of this guide. Please call for further assistance.

Are the axis DRO numbers moving?

If yes…The machine is likely not connected to the motion controller. See this chapter, section 8.2.1.

If no…Check that the LED beside the Jog On/Off button is lit. If not, press the Jog On/Off button. Also, if your keyboard has NumLock key on the keypad. If it does, NumLock must be turned on. Another way to jog is to press the TAB key to expose the graphical jog control.

Ensure that you are entering the correct speed in the correct Feed Rate DRO on the Program Run screen. If you changed a feed rate in SheetCAM, it will not be reflected on the machine. The SeriesOne and XT use the feed rate entered on the Program Run screen.

Check the following…

1. *There is a setting toward the bottom of the Feed Rate controls called Jog Speed. This a percentage. Pressing the Jog Speed button will toggle between a user setting and 100%. If the machine seems slow, make sure this is set to 100%.*

2. *Go to the Diagnostics screen and make sure there are no limits triggers. Triggered limits will appear in the upper right corner as yellow LEDs. If a limit is triggered, jog speed will be limited. If you do have limits triggered, check all three limit switches for free movement.*

This is usually caused by trying to start a job somewhere in the middle of the program. If you stop the job for any reason and want to start over, make sure you press Rewind to bring the program back to the top of the G-Code file.

This is caused by the Pierce Height calibration being off. See Chapter 5, Section 5.2 for detailed instruction on calibrating Pierce Height.

For instance, you cut a 10" square and the square measures 10-1/16". This is usually caused by the steps per unit being incorrect. The machine need to be calibrated. This is quick and easy. See Chapter 5, Section 5.1 for detailed instructions on calibrating the X and Y axis.

Also make sure that the lead and offset were placed on the correct side of the part. For instance, if you lay out a 10" square expecting to get a 10" square, but you accidentally use in inside offset, the square will be smaller than 10".

This is almost always a plasma fault. First determine if the LED lights beside the torch button stay on when the plasma shuts off. If they do, the plasma shut off on its own and when it does this, it will show a fault number in the LCD window. Those faults are in the plasma manual. The most common Hypertherm faults are…

- *0-12…Low input gas pressure*
- *0-20…Low gas pressure*

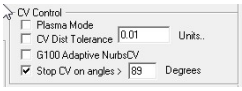
- *0-21…Gas flow lost while cutting*
- *0-22…No gas input*
- *0-30…Torch consumables stuck (Replace swirl ring)*
- *0-50…Retaining cap off*
- *0-51…Torch is on at power-up*

If you are sure you are not getting a plasma fault and this is still happening, please call for further assistance.

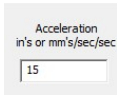
Square corners are cutting round 8.3.8

This is caused by the Mach3 CV (constant Velocity) engine trying to maintain the commanded federate through a sharp corner. There are three ways to correct this…

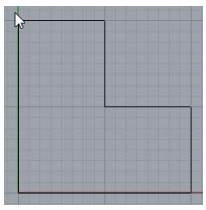
1. *Go to Config -> General Config and check the box labeled Stop CV on Angles. The angle value defaults to 89, which means that CV will be disabled on angles of 89 degrees or greater. You can enter any angle here that you wish. This will force Mach3 to switch to Exact Stop mode for these corners and cut them square.*

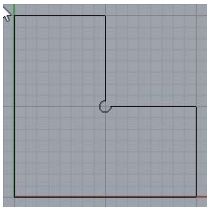


2. *Go to Config -> Motor Tuning. Click on the X axis and increase the Acceleration value. An increase of 5 or less is usually sufficient. After changing the value, press Save Axis Settings. Click on the Y axis and repeat. It is important that the X and Y axes parameters match.*



3. *Redraw your part to eliminate sharp inside corners. Outside corners are fine because SheetCAM rolls the tool around the corner, but inside corners are a simple segment to segment transition. Your original part probably has inside corners that look like what is pictured to the left.*

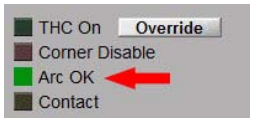




Redraw the part so you inside corners look like this. This will force Mach3 to cut into the corner. It also makes for clean breaks (bends).

The torch lights but the machine does not move 8.3.9

For the machine to move there must be an ArkOK signal from the plasma unit. This signal tells the Mach3 that the arc is established and it is OK to begin moving. There is an LED labeled Arc OK in the Torch Control section of the Program Run screen. This LED must be lit for the machine to move. If it is not lit, check the following…



- *Change consumables. This should be the first course of action for any cut related issue.*
- *Make sure the plasma is making a good ground to the table AND the material.*

- *Check the pierce height. Excessive pierce height will cause this. If in doubt, lower the pierce height and try again.*

- *If the above does not solve the problem, please call for further assistance.*

The torch moves to the starting point, references, but does not light 8.3.10
This is usually caused by a fault in the plasma unit. The Hypertherm plasma units will show a fault number on the LCD screen on the front panel. The most common fault is 0-30. This indicates that when the torch tried to light, the electrode and nozzle were shorted together. A worn swirl ring is the most likely cause

To recover from this fault, all you need to do is…

1. *Cycle the torch off by hitting the Torch button so the LEDs beside the button are off.*

2. *Cycle power on the plasma unit to clear the fault.*

3. *Re-light the torch by pressing the Torch button. If the fault is cleared, the torch will light and the job will continue. If the fault persists, change all consumables, including the swirl ring. If the fault persists, call for further assistance.*

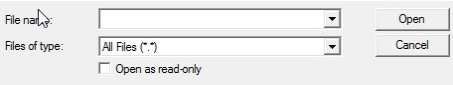
When I zero the torch and start the program, the torch makes unexplainable moves..... 8.3.11
There are three things that will cause this…

1. *There is a tool offset set when there should not be or the tool offset entered is incorrect. To fix this, enter 0 (zero) for all 4 tool offsets. This will set the X and Y offsets for the cutting torch and marker torch to zero. For more information on tool offsets, see Chapter 3, section 3.8.*

2. *You forgot to press Zero All before running the program. Start over and Press Zero All, then Cycle Start.*

3. *The program was created in SheetCAM with an offset from zero. If a Shape Wizard part runs fine, but a part programmed in SheetCAM does not, this is likely the problem. In SheetCAM, go to Job Options and make sure there is no offset. Re-generate the job (TAP file) and try it again.*

If you try loading a file and cannot find the file in the file dialog, it most likely because you have not created a .TAP file in SheetCAM or you are trying to load a .DXF (drawing) or .JOB (SheetCAM job) file.

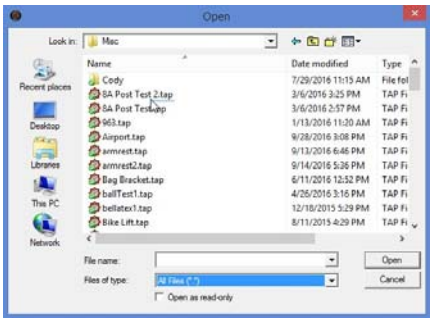


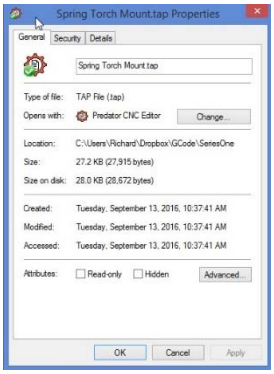
When you import a DXF file into SheetCAM, the SheetCAM job file usually defaults to the same as the DXF file. When you run the post processor in SheetCAM, that file usually defaults to same also. The result of this is that you'll have 3 files with same name but different file extensions. For instance, if your DXF file was duck.dxf, you will probably end up with duck.dxf, duck.job and duck.tap.

Mach 3 can only read TAP files. If you try opening a file that you know you created, it is likely that you are not looking where SheetCAM saved the TAP file or the file dialog is set to filter out TAP files. If this is the case, clear the File Name text box and set Files Of Type to All Files (.*). See image above.*

The file dialog will list all of the files available. Make sure you are loading a TAP file. There are two ways you can tell…

***1)** If your file dialog is set to view file details, it will look like the one pictured. If yours does not look like this, you set it to look like this by right clicking in the white space in the dialog, then clicking View and selecting Details. In Detail view, the file type will be shown to the right side to the dialog, as shown in the dialog pictured.*





***2)** Highlight the file in the file dialog, right click on the file name and select Properties. The file type will be displayed in the Properties window.*

There are a couple of things that will cause this…

1. *Sometimes when you think you are loading a G-Code file, you are actually loading a SheetCAM job file or a DXF drawing file. Look at the file type in the file open dialog and make sure you are opening a TAP file.*

2. *You may have created the TAP file in SheetCAM, but failed to creat an operation for each layer in the job. Each active layer must have an operation. If you fail to apply the operation(s) and*

run the post processor, it will create a TAP file, but the file will only contain the file header and footer. When you open this file in Mach3, it will appear to load, but nothing will appear in tool path display window.

The program loads, but does not run properly 8.4.3

This is usually because the wrong post processor had been selected in SheetCAM. Open SheetCAM and go to Options -> Machine -> Post Processor. Make sure the TrucutCNC post has been selected as the Post Processor. The actual post name will vary depending on the age of the machine, but it will always have Trucut in the post name.

A corrupt Mach3 configuration can also cause this. See Chapter 9, Section 9.1.3, Installing the Trucut Screensets.

All the files you will need to re-install Mach3 are on the flash drive that came with your machine. Follow these steps to re-install Mach3….

1. *Plug the Trucut flash drive into the computer. A list of folders and files will appear. Click the folder called Mach3 to open the folder. Double click the Mach3 setup file.*

2. *A large splash screen will appear with a dialog box. Click Next.*

3. *The next dialog is the Mach3 license agreement. Click I Agree… then click next.*

4. *The next dialog allows you to choose an installation location. DO NOT change this. It should be C:\Mach3. Click Next.*

5. *The next dialog allows you to select packages to install. Nothing should be checked here.*

6. *The next dialog allows you to create custom profiles. DO NOT press these buttons. We will install the Trucut profile in the next step. Click Next.*

7. *You are now ready to install. Click Next to complete the installation.*

8. *Continue to section 9.1.2, Installing ESS Plug-In.*

The ESS (Ethernet Smoothstepper) requires a plug-in to function with Mach3. Follow these steps to install the plug-in…

1. *If the Trucut USB flash drive is not plugged into the computer, plug it in now.*

2. *Double click on the folder called ESS to open the folder. Double click again on the folder called ESS_v10r2d1d. Right click on the file called ESS_v10r2d1d.m3p and click Copy.*

3. *Go back to the main ESS folder and double click on the file called ESS_v10r2d1d.m3p. A message will pop up saying that the plugin was installed. And the ESS_v10r2d1d.m3p file will disappear.*

4. *Still in the main ESS folder, right click in the white space and click Paste. This will paste the ESS_v10r2d1d.m3p back in the ESS folder in case you ever need it again.*

5. *Continue to section 9.1.3, Installing the Trucut Screensets.*

This step installs all our custom enhancements to Mach3. Follow the steps below….

1. *If the Trucut USB flash drive is not plugging into the computer, plug it in now.*

2. *Close Mach3 if it is open.*

3. *The setup files are self-extracting zip files called Trucut Setup – S1 and Trucut Setup – XT. Double click the file that corresponds to your machine. A dialog will appear. Click Unzip. All files will be extracted to where they need to be.*

4. *Click Close and continue to section 9.1.4, Setting the Computers Ethernet Port.*

Setting the Computers Ethernet Port 9.1.4

The ESS communicates with the computer via the Cat5 Ethernet port using a dedicated static IP address. To set the static IP address, follow these steps…

1. *Open Windows Control Panel.*

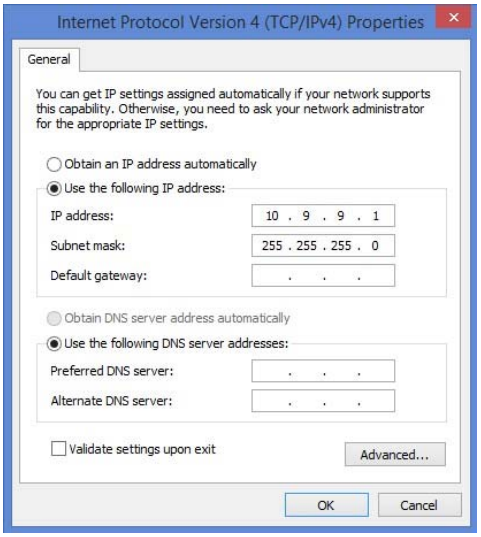
2. *Click on Network and Internet.*

3. *Click Change Adapter Settings.*

4. *You should see an Ethernet and a WiFi connection. Right click on the Ethernet connection and select Properties.*

5. *From the list of items, select Internet Protocol Version 4 and press Properties.*

6. *Set the port properties as per the image below.*



NOTE: After making this change, connection to the internet will not be possible via the Cat5 Ethernet port. Use WiFi to connect to the internet.

Continue to section 9.1.5, Installing Mach3 License.

Installing Mach3 License..... 9.1.5

Follow these steps to install the Mach3 license….

1. *If the Trucut USB flash drive is not plugged into the computer, plug it in now.*

2. *Close Mach3 if it is open.*

3. *On the Trucut flash drive, click on the folder called Licenses.*

4. *The Mach3 license file is called Mach1Lic.dat. Right click on this file and click Copy.*

5. *Navigate to the C:\Mach3 folder. Right click in white space in this folder and select Paste. This will copy the license file into the C:\Mach3 folder.*

6. *Open Mach3 and you should see your name at the top of the screen. If you see "Demo", you did not copy the license file into C:\Mach3 properly.*

All Mach3 configuration setting are stored in a file and this file can become corrupt. If this happens, Mach3 may open to a blank screen or the program may just not operate properly. The best way to fix this is to re-run the Trucut setup file. See this chapter, section 9.1.3, Installing the Trucut Screensets.