

Super-B Quick Start



Objective

This document outlines the basic steps needed to run a Super-B plasma cutting CNC. It does not cover all possible topics and is intended as an introduction and training aid. CAD and CAM are not covered.

Contents

- Safety
- Key Points
- Quick Start
- Default Start Cycle Sequence

Safety

Please read the following important safety warnings before installing, operating, performing maintenance on, or troubleshooting your machine.

Do Not Attempt to Strong-arm the System

- Do not attempt to force the system to work while in a fault condition. Operating the system during a known fault could result in injury or death, damage to your equipment, and void your warranty. Pay attention to errors and faults reported by both the plasma generator and the Dynatorch CNC software.
- Do not attempt to bypass safety features. This could result in injury, death, equipment damage, and void your warranty.

Do Not Come Near or Touch the Dynatorch CNC Table While It Is In Motion

- Loose hair, clothes, and jewelry can get caught in CNC components, resulting in severe injury or death.
- If the ground or work lead fails while the machine is cutting and someone touches the table, they may become a path for ground resulting in severe injury or death from electrical shock.
- The machine could start or stop unintentionally if the operator is confused or distracted.
- Do not position or hold down material, or remove material, while the machine is cutting. This can result in a dangerous collision with the machine or electric shock.

Do Not Change Torch Consumables While the Plasma Generator Is On

- If the PIP (Parts In Place) micro-switch fails to detect that the consumables have been removed, the plasma start signal will not be disabled. This will allow the plasma to strike an arc. It's very unlikely, but removing the consumables could trigger a plasma start.
- The safety interlock inside of the plasma generator could fail even if the PIP (Parts in Place) micro-switch works.
- Putting your hands near the torch while it is off is dangerous because a static discharge, path to ground, or pressing a gas switch accidentally, could inadvertently tell the plasma generator to strike an arc.

Do Not Alter Machine Settings

- Altering machine settings can cause the machine to malfunction, which could inadvertently lead to someone getting injured or killed.

Safety

Use PPE (Personal Protective Equipment)

- Use gloves to handle material. Thin materials have surprisingly sharp edges and cut material may have slag on it sharp enough to cut you. Freshly cut materials are too hot to touch – consider using a gripping tool or magnet to remove hot material. Cut materials also have residue on them that are toxic.
- Wear steel toe shoes if you're loading and unloading materials. The side or edge of materials can crush or cut off toes and feet.
- Where welding glasses (shade 5 or greater) to protect your eyes from the arc. A plasma arc releases UVA, UVB, blue light, and infrared light with enough intensity to injure your retinas and can cause or accelerate cataracts of the lens. Safety glasses also protect your eyes from molten debris which can permanently damage your corneas.
- Never breath the fumes released by cut materials. Many of the chemicals released by cut materials can make you sick, dizzy (toxic psychosis), or cause COPD and cancer. Always use good ventilation and/or a mask to avoid breathing fumes.
- The torch is a gas jet and can produce a whistle that is capable of damaging your hearing. Use ear plugs if your close to the torch for prolonged periods of time.

Fire & Explosion Hazards

- To prevent an explosion, do not bring pure oxygen, acetylene, or hydrogen anywhere near the plasma arc or spark area.
- Do not put oil or grease on the torch, torch consumables, or materials under cut. This will catch on fire. Only use the oxygen compatible grease to lube consumable O-rings that is made by, and provided by, the plasma generator manufacturer.
- Do not cut wood or create a dust cloud of any kind near the plasma arc - the dust cloud will ignite.
- Do not have any combustible chemicals or materials, especially pressurized combustibles in spray cans, within a 35 foot radius of the CNC table.
- Do not place freshly cut hot materials on or near combustible materials.
- In older versions of the Dynatorch CNC software, if you intend to cut material that is equal to or greater than 3/8 (0.375) inches, you will need to switch the CNC software over to thick material cutting mode. Failure to do so can result in the torch and CNC catching on fire. If you do not have a 'Thick Material' button, your version of software automatically handles this.
- Have a fire extinguisher nearby.

Safety

Electric Shock Hazard

- Always ensure that the work-lead is electrically connected to the material before starting the plasma arc. If the work lead is not connected, there won't be a return path for the electrical energy coming out of the torch. If someone touches the table, and they happen to be a good path to ground, they will be electrocuted, resulting in severe injury or death. This will also damage and destroy the consumables, the torch, and the plasma generator.
- Check the ground rod once a month using the light bulb test as a part of routine maintenance. If the ground rod is not present, connected, or sufficiently conductive, the CNC will become an electrical shock hazard and you will experience EMI (Electromagnetic Interference) faults.
- The electrical energy coming from the torch is high voltage, high amperage DC (Direct Current), not low voltage AC (Alternating Current). This is not the same as being shocked by a small battery or electrical outlet. Being shocked by a plasma cutting machine is the equivalent of being hit by a lightning. High voltage DC electricity is far more dangerous than AC because a person can possibly break loose from an AC shock if they have enough will power and strength, or they may be punched by the shock instead of grabbed. This is not the case with DC. If you get shocked by DC, your muscles will be locked by the current and you will not be able to break free until the power source is deactivated. The victim is usually dead long before the power source is turned off.
- Never touch the CNC, or attempt to place, position, or remove material while the plasma arc is on.
- If you need to service your plasma generator, put the power switch in the off position, disconnect the plasma generator from the power source, pull the lever to the 'OFF' position on the disconnect, and wait 15 minutes before removing the cover of the plasma generator.

EMI (Electromagnetic Interference)

- Both the torch and plasma generator emit high frequency electromagnetic radiation when there is a plasma arc being generated.
- High frequency EMI can cause a pace-maker to malfunction, resulting in cardiac arrest. People who have a pace-maker should stay 30 feet away from the plasma generator and torch.
- High frequency EMI can cause neuro-stimulators to malfunction.
- High frequency EMI can damage hearing aids.
- The high frequency EMI can damage solid state electronics such as computers and phones, and can erase credit cards and key cards. Keep electronic devices away from the plasma generator and torch.

Safety

Trip Hazards

- There are numerous electrical cables and gas hoses connected between the gas source, computer, CNC, and control box. Be mindful not to trip over cables and hoses laying on the floor.

Always Home the Machine at Startup

- If the machine is not homed at startup, the software will not know where the gantry, carriage, or torch are in relation to the table. This can result in the gantry being commanded to come off of the table, which could injure or kill someone.

Know How to Stop the Machine

- In the event of an emergency, press the space-bar on the keyboard to disable the drives. This is not the same thing as an E-Stop, but will immediately stop motion.
- In the event of an emergency, press the emergency stop button; the big red mushroom button on the control box.
- Know where the plasma generator disconnect is located. In the event of an electric shock, pull the disconnect lever to the off position.
- Know where the breaker panel and breaker switches are for the CNC and plasma generator.

Do Not Attempt to Repair the CNC Or Plasma Generator

- Notify your supervisor and contact a certified Dynatorch technician for troubleshooting and repair.

Do Not Operate the Machine Under Duress

- Do not operate the machine if you feel emotionally distressed, sick, fatigued, or you are under the influence of a drug.

Why Do Accidents Happen?

- Accidents happen because of the combination of events, not one single event. For example, imagine that someone removed the work-lead (ground) from the table and forgot to put it back on, then someone else using the machine decides to remove material while the machine is still cutting because they assume the work-lead is still on – that person could be electrocuted.
- Operating equipment under duress (emotional distress, illness, fatigue, or drug use).
- Failure to operate equipment correctly (lack of training or a bad attitude).
- Failure to perform regular maintenance and inspections.

Key Points

Before you can properly install, operate, maintain, or troubleshoot your machine you need to know these important facts

- Keep the plasma generator at least 6 feet away from the computer. The EMI (Electromagnetic Interference) from the plasma generator will interfere with the Dynatorch CNC communication network.
- If your version of software does not have the cut chart feature, or your plasma generator is not supported by the cut chart feature, you will not be able to properly operate the machine without the operation manual that came with the plasma generator. It contains the cut charts you will need to properly adjust your plasma generator and Dynatorch CNC software settings. The proper plasma and Dynatorch CNC software settings can typically be found in the 'Operation' or 'Torch Data' section. Cut charts are organized based on amperage, material type, plasma/shield gas selection (if applicable), and material thickness. In addition to this, you should take the time to become familiar with the proper installation, operation, and maintenance of your plasma generator before attempting to use it.
- Always use the consumable stack listed by the cut chart precisely. Mixing and matching consumables will not work and can result in malfunction, equipment damage, and void your warranty. Go by the part number, not the amperage to avoid mismatching consumables. You may need a flashlight and magnifying glass to read the part numbers on the consumables.
- The plasma generator and CNC software values recommended by the cut chart are starting points. You will need to adjust these values to optimize cut quality.
- When you change material type or material thickness you will also need to change the plasma generator settings, torch consumables, and Dynatorch CNC software settings to match. This is also true if you select a different cutting method, such as expanded metal, gouging, normal cutting, a higher or lower amperage, and/or a different plasma/shield gas mix. You cannot use the same torch consumables and settings for any material, material thickness, plasma/shield gases (if applicable), or amperage. Doing so can prematurely age or damage torch consumables and risk damaging other components in your cutting system.
- Not everything on the Dynatorch CNC main screen is labeled. If you are curious what an unlabeled button or indicator is for, simply hover the mouse over it for 2 seconds to see a tool-tip (A white text box) pop up that explains what that button or indicator is.
- Currently, the Dynatorch CNC software does not have an onboard CAD program. Call Dynatorch and ask to speak with sales if CAD software is required.
- Currently, parts cannot be nested using the Dynatorch CNC software. Call Dynatorch and ask to speak with sales if nesting software is required.

Key Points

- G-Code is machine specific. It is very unlikely that G-Code for other CNCs will work with the Dynatorch CNC. Do not try to load G-Code from another machine onto the Dynatorch system.
- Many (but not all) of the machine settings (not to be confused with the operator settings in the main screen) in the Dynatorch CNC software are representative of the hardware in the drive system. Unless you have added, removed, or changed hardware in your drive system it is unlikely that you need to alter machine settings. Altering these settings can cause your machine to malfunction and may result in unsafe operating conditions and/or damage to the system. Machine settings are set to optimal and safe values at the factory. Do not alter machine settings without contacting technical support first or after having proper instruction and training. If you are instructed to make changes to the machine settings, always record the original settings before making changes. If authorized to change machine settings, only change one setting at a time in small increments for a safe and controlled test. Some of the machine settings should only be customized at setup or to change the setup, like those found in the 'Homing' tab. A few machine settings, like those found in the 'Holes' and 'Corners' tabs, are operational settings, and are useful for optimizing cut quality. Contact Dynatorch technical support for information and training.
- '.dxf' files are the universal file format for CAD drawings. G-Code is the instructions that the CNC requires in order to operate. G-Code can be derived from a '.dxf' using a converter program. You can use third party converter software, a post-processor add-on for third party software, or the onboard converter to convert '.dxf' files into G-Code. You will need either the '.dxf' file or the G-Code file to run the CNC. You cannot load and cut other file formats.
- Do not attempt to repair any component in your system without contacting tech support first.
- The first time you run the Dynatorch CNC software you will need to go into the setup for the onboard converter program and select 'OK' to apply the conversion settings. Otherwise, when you convert a .dxf into G-Code and try to run it, you will likely get an error telling you that the M02 or M30 codes are missing.
- The majority of malfunctions are the result of CAD drawings that are not optimized for CNC, improper conversion (omissions, inclusions, and incorrect values in the G-Code), insufficient air pressure and/or flow, improper plasma generator or CNC settings, incorrect or damaged consumables, non-existent or ineffective grounding of the CNC table, the drive system, and/or the computer, the plasma generator being too close to the computer, and electrical faults in the building wiring.

Key Points

- The Dynatorch software is primarily intended to provide precision motion control, automatic torch height control, ohmic sensing, and torch crash detection. However, in addition to this the Dynatorch CNC software also contains:
 - ▶ Help files for operation and performance optimization.
 - ▶ The 'Align' tool that allows the CNC to adjust to the rotation of the material on the table.
 - ▶ The 'Make it Fit' tool that eliminates the need for time consuming dry runs.
 - ▶ The 'Quick Cut' tool that provides speedy separation of the plate skeleton from usable material.
 - ▶ The settings library that stores common settings for fast setup.
 - ▶ Plasma Mate, a program that allows quick and easy generation of simple parts by modifying geometric parameters.
 - ▶ Tap Mate, a program that allows quick and easy common modifications to G-Code.
 - ▶ Code Wizard and Part Builder, two more programs that speed up and simplify G-Code generation for very simple parts.
 - ▶ An onboard G-Code editor (Or the option to use an external editor).
 - ▶ An onboard converter to translate '.dxf' files into G-Code (The CNC can only run G-Code).
 - ▶ Access to all machine settings.
 - ▶ Communication and friction diagnostics.
 - ▶ An event logger called 'Statistics' to help with troubleshooting.

Quick Start

- 01** Select a Cutting Process
- 02** Obtain or Create a DXF
- 03** Power Up
- 04** Launch The Dynatorch CNC Software
- 05** The Homing Procedure
- 06** Opening a Program to Cut
- 07** Install Torch Consumables
- 08** Set the Plasma Generator
- 09** Set Cut Values
- 10** Setting Feedrate
- 11** Thick Material Cutting (Legacy Version)
- 12** Set Hole and Corner Slowdown
- 13** Set Alignment
- 14** Jogging
- 15** Setting The Zero Position
- 16** The 'Make It Fit' Tool
- 17** Start Cycle

01 - Select a Cutting Process

Before doing anything, you should know what thickness of material you intend to use and then select your cutting process so that you will know what the resulting kerf will be.

Images are approximations - rely on part numbers only

Material: **Mild Steel** | Selected cut process: **104511**

Thickness: **3/16"**

220842	220857	220941	220953	220817
--------	--------	--------	--------	--------

Images of consumables for each part number are shown below.

Cut Process

Shielded	Amps	Feed Rate	PSI (25'/50')	Cutting Gas	Precautions	
104511	Yes	45	70.0	75 / 80	Air	Normal

The estimated kerf that will result from the selected cutting process can be found here.

Pierce Height	0.150	inches
Arc Start Height	0.150	inches
Cut Height	0.060	inches
Thickness	0.188	inches
Pierce Delay	0.5	sec
Time to Cut	0.0	sec
Voltage	135	volts
Amps	45	amps
Feedrate	70.0	in/min
Kerf	0.065	inches

Apply | Close

These values are RECOMMENDED settings from the Plasma manufacturer's catalog, fine tuning of these may be needed to obtain quality cuts

Kerf is the width of the cutting tool, in this case that's the plasma arc. The cutting process determines what the kerf will be. Kerf is not a setting - it is a result of the cutting process chosen. Every cut process results in a kerf diameter that may not work well, or at all, with a given drawing if the lines in the drawing are too close together.

The draftsman should be aware of what the kerf will be before they begin drawing so that they do not draw lines too close together. The cut process should be determined before drawing, tool-pathing, and converting a DXF.

If a DXF will not work with the kerf, then the drawing will have to be scaled up in size, altered, or another cut process chosen that will result in a kerf that works with the drawing.

If the DXF does not match the resulting kerf of the cut process chosen, and the operator chooses to cut it out anyway, the machine will likely lose arc transfer in various locations along the cut path, or the resulting cut will be very distorted.

The cut process includes the following control values

- Shielded, Unshielded, or FineCut (Hypertherm only)
- Amperage
- Air Pressure
- Arc Start Height
- Pierce Height
- Pierce Delay
- Cutting Height
- Time to Cut (Time Delay)
- Arc Voltage (Set Point)
- Feedrate

The cut process can be found using your [plasma operation manual](#) or by using the [cut-chart feature](#) in the Dynatorch CNC software (see Install Torch Consumables).

02 - Obtain or Create a DXF

You will need to obtain or generate a .dxf file before you can cut. Your options are:

- Obtain the .dxf file from your customer.
- Generate a .dxf file using a CAD program.
- Select a .dxf using Dynatorch part builder tools such as Plasma Mate, Code Wizard, or Part Builder. This option is for very simple parts. You cannot draw a custom part using the Dynatorch CNC software.
- Download a .dxf file from a website. (Caution: Many downloads are not actually CNC friendly even if they claim to be and will require some re-work.)

Once you have a completed .dxf file, you will need to modify it to make it “CNC ready”. This is when you define the toolpath by adding kerf compensation, lead-ins and lead-outs and lead gaps, specifying cut direction, specifying order of cut, and arraying a single part or nesting many unsimilar parts. After the tool path has been defined, the modified .dxf file can be converted into a Dynatorch G-Code file (.tap or .cnc file).

You cannot cut a .dxf file. It must be converted. Your options are to:

- Use a third party application which can be upgraded with the Dynatorch post-processor plug-in.
- Use the built-in Dynatorch converter ([click here for an instructional video](#)).

You do not need to specify certain parameters in the post process such as material thickness, pierce height, pierce delay, or set point. You will specify these values later in the Dynatorch CNC software. Other than cutting speed or feedrate, these values will be ignored by the Dynatorch post processor.

If the part was generated in Plasma Mate, the part can be saved as a ‘.dxf’ to be converted later or opened in the onboard converter. If the part is converted using the onboard converter, the part will automatically be loaded when converter is closed.

Typical Procedure

- 1 Create a DXF in CAD software**
- 2 Create a toolpath based on the DXF in CAM software**
- 3 Convert toolpath into Dynatorch G-Code (.tap file)**
- 4 Load TAP file into Dynatorch CNC software to cut**

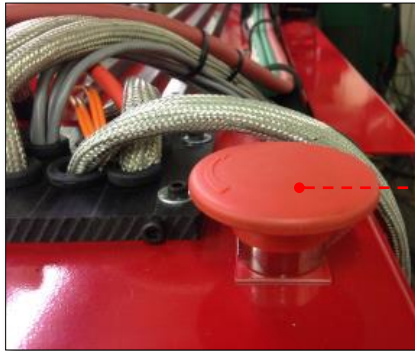
03 - Power Up

There are five systems that require power:

- Air compressor
- Air dryer
- Plasma generator
- Dynatorch Super-B
- Computer

The system needs to be powered up in a specific order to avoid communication errors. The electronics inside the drive box need to be powered up before powering on the computer and launching the Dynatorch software.

- 1 Turn on the air compressor and wait for the correct air pressure to be reached.
- 2 Turn on the air dryer.
- 3 Turn on the breaker for the plasma generator.
- 4 Pull the disconnect lever for the plasma generator to the 'ON' position.
- 5 Turn on the plasma generator.
- 6 Turn on the drive box by twisting and raising the large red E-Stop button. The E-Stop button is both the E-Stop and the power button.

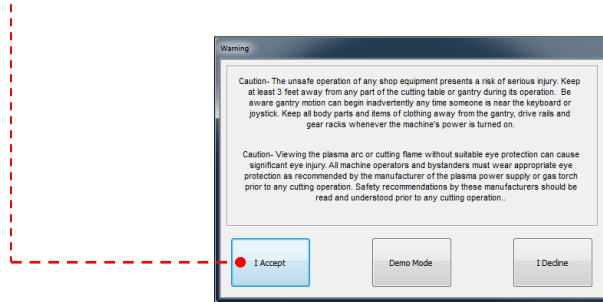


- 7 Turn on the computer.
- 8 Launch the Dynatorch CNC software by double left-clicking on the torch icon.

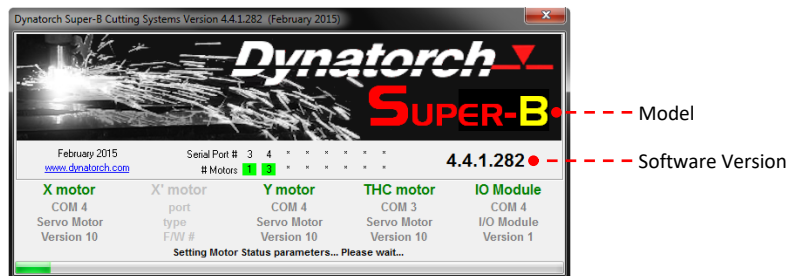
04 - Launch The Dynatorch CNC Software

Start the Dynatorch CNC Software.

If you agree to follow the safety guidelines, click 'I Accept' at the warning pop-up.

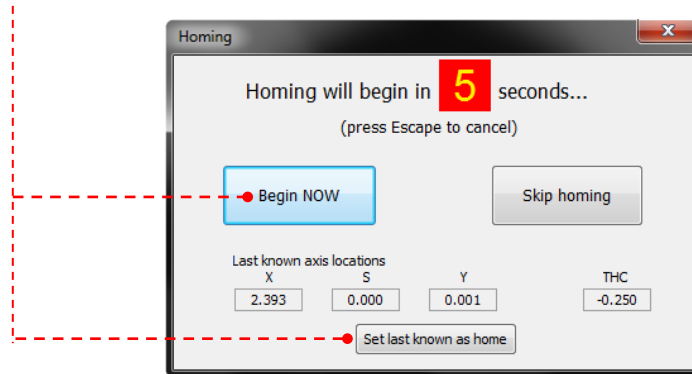


The Super-B software will automatically search for motors available.



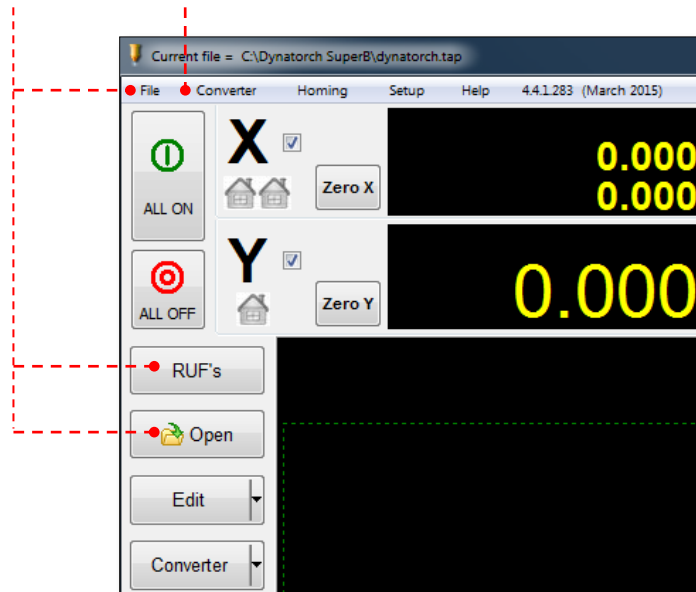
05 - The Homing Procedure

You will be prompted if you want to home the machine, skip homing, or home at the last known home position. It's recommended that you always home the machine at the beginning of the day, and return the gantry and carriage to the home position at the end of the day to speed up homing at the next power up. Note, if the machine is not homed the jog speed will automatically be locked at 10% to protect the CNC.



06 - Opening a Program to Cut

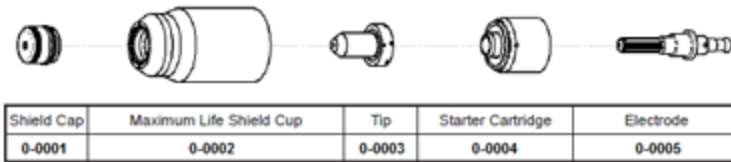
Load a '.tap' file (G-Code) by clicking on 'File' then 'Open G-Code', by clicking on 'RUF's' (recently used files), or by clicking on the 'Open' button. Alternatively, a '.dxf' may be converted in the Dynatorch converter and automatically loaded.



07 - Install Torch Consumables

Check all the consumables to ensure they match the amperage, material, material thickness that you will be cutting at. This information can be found in the [Dynatorch cut chart feature](#) and also in the [plasma operator manual](#).

- Shield Cup
- Electrode
- Swirl Ring
- Nozzle / Tip
- Shield Cap / Deflector



Cut Chart

To access the Dynatorch cut chart feature, click on the 'Cut Chart' button. Select a process to see the torch consumables.

Set Material Type & Thickness
Consumables

Images are approximations - rely on part numbers only

Material: **Mild Steel**

Thickness: **3/16"**

Selected cut process: **104511**

220842	220857	220941	220953	220817

Cut Process	Shielded	Amps	Feed Rate	PSI (25'/50')	Cutting Gas	Precautions
104511	Yes	45	70.0	75 / 80	Air	Normal

Cut Process & Amperage

Pierce Height	0.150	inches
Arc Start Height	0.150	inches
Cut Height	0.060	inches
Thickness	0.188	inches
Pierce Delay	0.5	sec
Time to Cut	0.0	sec
Voltage	135	volts
Amps	45	amps
Feedrate	70.0	in/min
Kerf	0.065	inches

Apply
Close

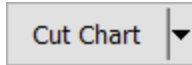
These values are RECOMMENDED settings from the Plasma manufacturer's catalog, fine tuning of these may be needed to obtain quality cuts

08 - Set the Plasma Generator

The amperage, mode, and output pressure can be found in the Dynatorch CNC software cut chart feature, or in the plasma generator's operation manual.



Your air plasma generator may be a different brand and model
This is an example only



To access the Dynatorch cut chart feature, click on the 'Cut Chart' button. Select a process to see the recommended output pressure.

Material	Mild Steel	Selected cut process	220842	220857	220941	220953	220817
Thickness	3/16"	104511					

Cut Process	Shielded	Amps	Feed Rate	PSI (25'/50')	Cutting Gas	Precautions
104511	Yes	45	70.0	75 / 80	Air	Normal

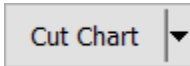
Set the plasma generator to the 25' pressure (75 in this example)

Pierce Height	0.150	inches
Arc Start Height	0.150	inches
Cut Height	0.060	inches
Thickness	0.188	inches
Pierce Delay	0.5	sec
Time to Cut	0.0	sec
Voltage	135	volts
Amps	45	amps
Feedrate	70.0	in/min
Kerf	0.065	inches

Apply Close

These values are RECOMMENDED settings from the Plasma manufacturer's catalog, fine tuning of these may be needed to obtain quality cuts

09 - Set Cut Values



To access the Dynatorch cut chart feature, click on the 'Cut Chart' button.

- 1 Select the material in the drop down box.
- 2 Select the thickness of the material in the drop down box.
- 3 Determine which amperage you will be cutting at and left-click on the process number.
- 4 Note the federate and kerf for this cutting process.
- 5 Left-click the 'Apply' button to set these control values.
- 6 Left-click 'Close' to close the cut chart window.

Images are approximations - rely on part numbers only

Material **Mild Steel** ▼

Thickness **1/4 in** ▼

Selected cut process **12024**

9-8232	9-8213	9-8208	9-8237	9-8245

Cut Process	Shielded	Amps	Feed Rate	PSI (25'/50')	Cutting Gas	Precautions
12024	Yes	40	45.0	75 / 80	Air	Normal
12132	Yes	60	80.0	85 / 90	Air	Normal
12259	Yes	80	100.0	85 / 90	Air	Normal
12372	Yes	100	85.0	75 / 75	Air	Normal
12458	Yes	120	160.0	80 / 80	Air	Normal

Pierce Height	0.160	inches
Arc Start Height	0.160	inches
Cut Height	0.160	inches
Thickness	0.250	inches
Pierce Delay	0.5	sec
Time to Cut	0.0	sec
Voltage	118	volts
Amps	40	amps
Feedrate	45.0	in/min
Kerf	0.066	inches

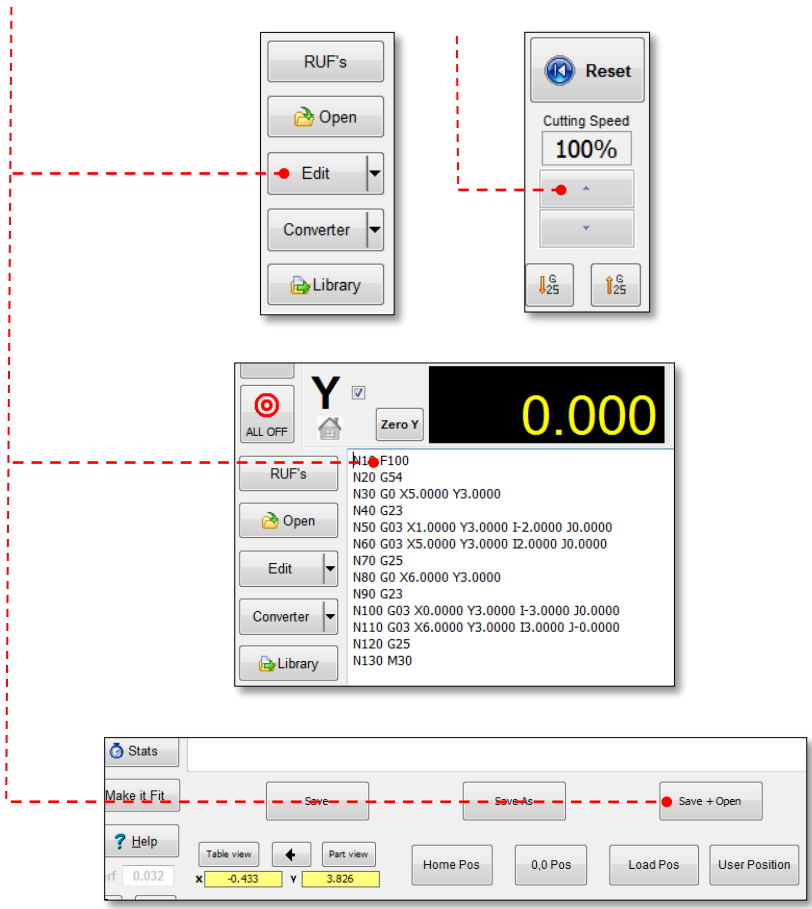
Apply

Close

These values are RECOMMENDED settings from the Plasma manufacturer's catalog, fine tuning of these may be needed to obtain quality cuts

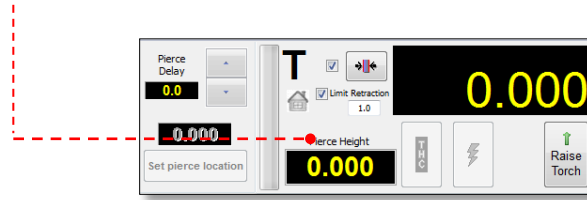
10 - Setting Feedrate

Set the Cutting Speed ('travel speed' or 'feedrate', refer to cut charts in the plasma generator operator's manual) using the cutting speed override for any value under 200 ipm and above 5 ipm, or open the G-Code using the onboard or external editor to set the feedrate above 200 ipm. If you are using the G-Code editor to set the cutting speed, locate the "F" code(s) in the G-Code and modify the value. Be sure to click "Save+Open" to save the update to the G-Code and load it. If the feedrate is above 200 ipm in the G-Code, make sure the cutting speed override is set to 100%.

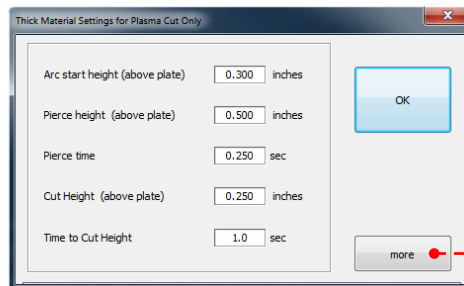


11 - Thick Material Cutting (Legacy Versions)

- If using thin metal settings, set the Pierce Height here (Do not confuse this with cutting height).



- If your cutting material that is 3/8" or greater, you should use thick metal settings instead of thin metal settings. To use thick metal settings instead of thin metal settings left click on the thick metal button and then right click to open the thick metal settings window. Click on the 'more' button to read a description of the thick metal cutting sequence.



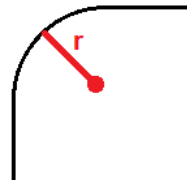
12 - Set Hole and Corner Slowdown

The CNC does not move at a constant speed when cutting. Feedrate or Cutting Speed is a speed limit, not a speed setting. The CNC attempts to accelerate up to the federate while cutting a straight path, and decelerates when changing direction or cutting a circle or hole. This slowdown is proportional to the amount of change in direction. The sharper the turn or the smaller the circle or hole, the more the machine slows down. This is necessary because the momentum and velocity of the plasma arc is not equal to the torch – the plasma arc has its own momentum and velocity. When the torch changes direction too quickly you lose control (position) of the plasma arc.

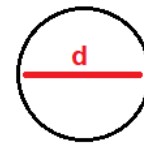
There are three conditions that trigger proportional slowdown; a sudden change in direction, a gradual or radial change in direction, and a circle or hole.



Sudden Change



Radial Change



Circle / Hole

When the feedrate is correct there is very little dross on a straight line cut. However, the feedrate could be correct, but the amount of slowdown could be too high or too low:

- Excessive dross or lost arc may occur while cutting corners if too much slowdown is applied (a low percentage value).
- Waves and radial distortions may occur while cutting corners if too little slowdown is applied (a high percentage value).
- Circles and holes will always have some amount of dross because of the amount of slowdown required to make a truly round hole with the least amount of bevel and taper. Lost arc can occur if the hole is cut too slow. If a hole is cut too fast, the circle or hole may not be round and the sidewall may have too much bevel or taper.

Feedrate is set by the G-Code or the cut chart feature (depending on settings in the cut chart setup tab). At any given point in the cut path, feedrate is modified by the Cutting Speed % and the percentage of slowdown.

$$\text{Current Feedrate} = (\text{Feedrate Setting} \times \text{Cutting Speed \%}) \times \text{Percentage of Slowdown}$$

The amount of slowdown that is applied can be set in the 'Corners' and 'Holes' tab in machine settings / 'Setup'.

It isn't always necessary to change the amount of slowdown, but a skilled operator must know how much the machine is slowing down, and how to increase or decrease this value. The percentage of slowdown is not automatically adjusted by the selected cut process. Percentage of slowdown is correlated with the amount and type of detail in a program, not just the material type, material thickness, or cutting amperage.

12 - Set Hole and Corner Slowdown (Continued)

To adjust the percentage of slowdown for circles and holes, left-click on 'Setup' in the toolbar, then left-click on the 'Holes' tab in machine settings.

When a path is short, we can assume it is cutting a circle. For instance, a 1/2" circle has a circumference of 1.57", if you add a nominal lead in of 1/4", then the total path length is 1.82". We define the path as anything between a G23 (start torch) and a G25 (stop torch) command. A typical g-code has multiple paths. Here we are ONLY interested in those individual paths that are short. We will treat them special as defined below because small holes (circles) will cut more precisely when they go slower. The graph to the right will demonstrate this principle.

Use APS (automatic proportional slowdown) for short path segments

Path lengths under 1.75 inches will be run at 40 % of the feed rate

Other lengths will be scaled proportionally up to a length of 3.25 inches

Above that, they run at the current porgrammed feed rate

Freeze THC during these short path cuts to prevent the torch from diving into the material

Current Feedrate: 100
Adjusted Feedrate: 41
Minimum Feedrate: 10

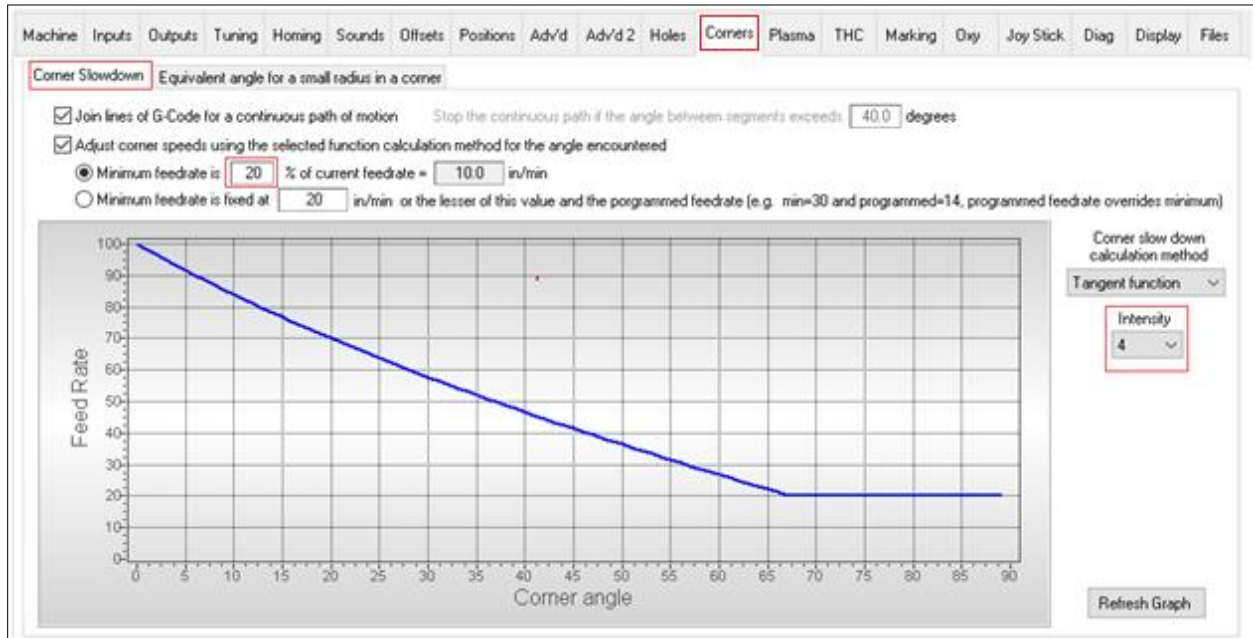
Path length: 1.8 ~Hole diameter: 0.54

In the example above:

- APS (Automatic Proportional Slowdown) for circles and holes is turned on.
- Any circle or hole with a path length under 1.75 inches will be run at 40% of the feedrate.
- Any circle or hole with a path length greater than 1.75 inches and less than 3.25 inches will be run at a percentage proportional to the path length.
- Any circle or hole with a path length greater than 3.25 inches will be run at the set feedrate.
- THC (Torch Height Control) will freeze when APS is being used to cut a circle or hole. This is to prevent the torch from crashing into the material. **Note:** When THC is frozen, you are cutting at pierce height, not cut height.
- On the right, the preview is set to a path length of 1.8 inches, which is equivalent to a circle or hole diameter of 0.54 inches. The current feed rate is 100 inches per minute, and the adjusted feedrate for this 0.54 inch diameter hole will be 41 inches per minute. The minimum feedrate that the machine could ever move is 10 inches per minute (set in the 'Corners' tab) – so the hole slowdown percentage cannot cause the machine to go below 10 inches per minute.

12 - Set Hole and Corner Slowdown (Continued)

To adjust the percentage of slowdown for sudden changes of direction, left-click on 'Setup' in the toolbar, then left-click on the 'Corners' tab, then the 'Corner Slowdown' tab, in machine settings.



In the example above:

- Automatic proportional slowdown is turned on for both sudden and gradual changes of direction.
- The “percentage of feedrate” method (recommended) is selected to determine the minimum feedrate. The minimum is important to prevent the CNC from slowing down so much that you lose arc transfer. In this example the minimum feedrate will be 20% of the feedrate x Cutting Speed %.
- An intensity of 4 is selected. The higher the intensity, the more aggressive is the percentage of slowdown. The lower the intensity, the less aggressive slowdown will be.

12 - Set Hole and Corner Slowdown (Continued)

To adjust the percentage of slowdown for gradual changes of direction, left-click on 'Setup' in the toolbar, then left-click on the 'Corners' tab, then the 'Equivalent angle for a small radius in a corner' tab, in machine settings.



Machine Inputs Outputs Tuning Homing Sounds Offsets Positions Adv'd Adv'd 2 Holes **Corners** Plasma THC Marking Oxy Joy Stick Diag Display Files

Corner Slowdown **Equivalent angle for a small radius in a corner**

Typically when two line segments intersect, the path disruption can be calculated by the vector direction of the two lines. However, when the two lines have a small fillet (arc) connecting them, the vector direction of the incoming line has no mathematical path disruption, [the vector directions are identical]. In these cases we change direction, possibly 90 degrees with no mathematical path disruption, but a huge mechanical path disruption. So when we come upon a small radius, we need to calculate it's angular equivalent... We do that here. If the radius is smaller than the limit listed below, then we calculate an angular displacement based on the size of the radius and the degrees of turn. Obviously a 1/16" radius turning 90 degrees will need to slow down much more than a 2" radius at 90 degrees, in fact a 2" radius [4" diameter] should not need to slow down at all. Remember this is designed for typical plasma cutting speeds.

Radius to equivalent angle calculation method
Logarithmic function
Intensity **2**
Arc radii less than **0.50** inches
will slow down based on the calculated method to their equivalent corner angle.
Refresh Graph

In the example above:

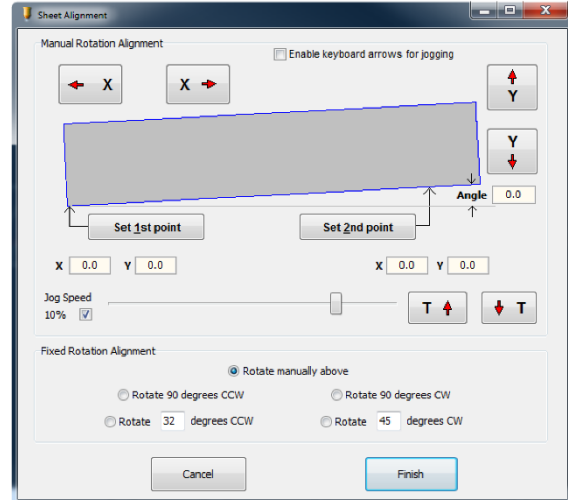
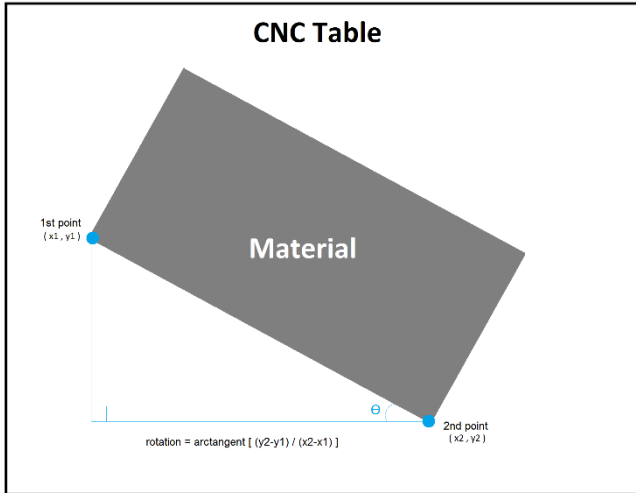
- The intensity is set to 2. The higher the intensity, the more aggressive is the percentage of slowdown. The lower the intensity, the less aggressive slowdown will be.

13 - Set Alignment



Click the 'Align' button to align cutting with the angle of the plate. This feature allows the operator to compensate for plate angel without having to move a large heavy plate and may not be necessary for smaller, lighter material.

Plate angel is calculated using two adjacent corner points on the plate. Line the torch and/or laser cross hair with the first corner of the plate and click "Set 1st point". Using the jog buttons on the menu, move the torch to the second adjacent corner and click "Set 2nd point". Once the two points have been determined, click the 'Finish' button at the bottom. The geometry in the viewport will be modified automatically.

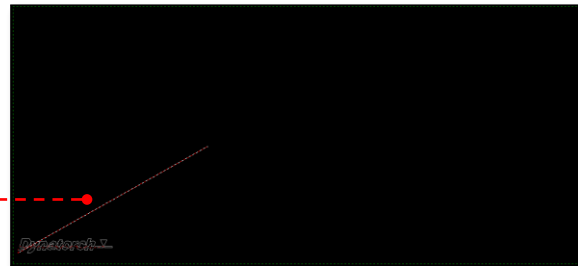
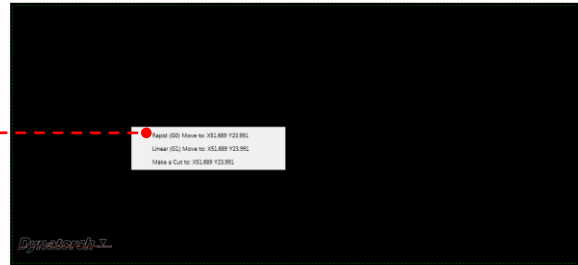


14 - Jogging

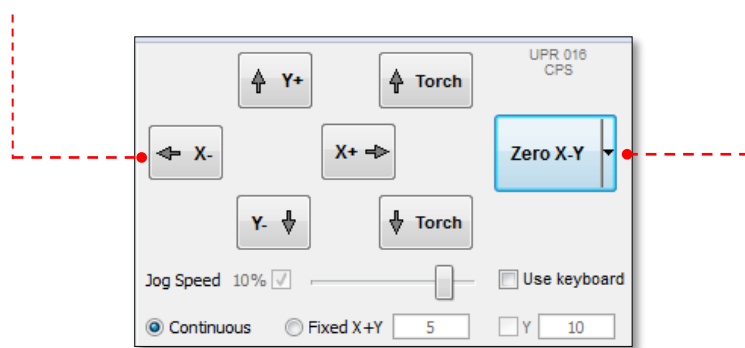
<edit>

15 - Set Zero Position

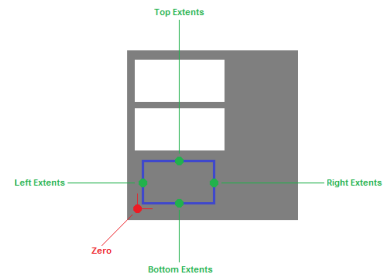
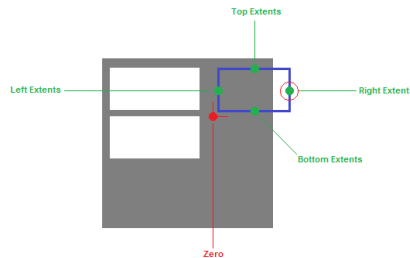
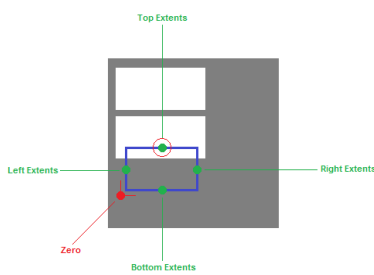
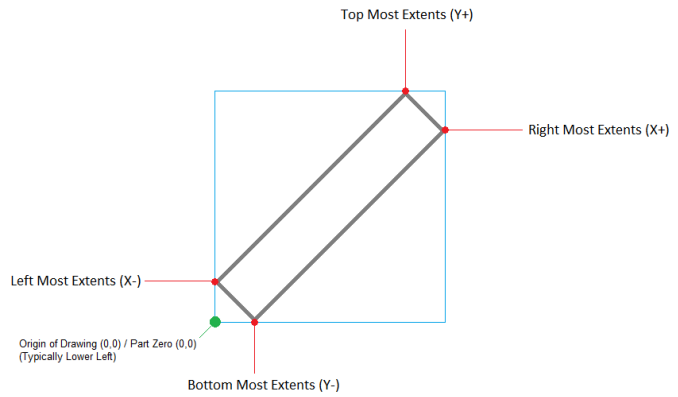
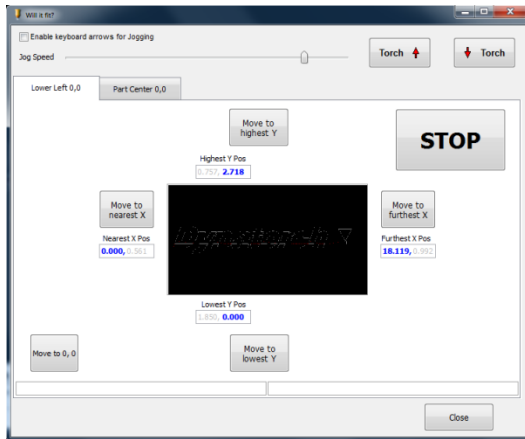
Right click in the general location within the view port that you want the torch to go. A pop up menu will appear; click on "Rapid (G0) Move to: X---,--- Y---,---". A red and white line will appear that represents the traverse path of the torch. The carriage and gantry will move to that location immediately.



Now using the jog buttons, (make sure 'Continuous' is checked) jog the torch to the desired start location and click the Zero X-Y button. Fixed jogging tells the CNC to move a specified number of inches or meters per mouse click. You can perform a fixed jog of the specified distance for both X and Y (X+Y) or you can use a separate distance for X and Y.

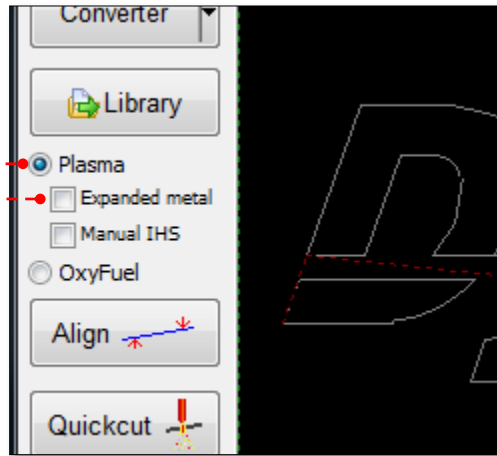


16 - Check Cut Area with 'Make It Fit' Tool



17 - Start Cycle

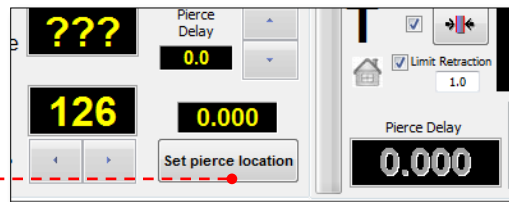
Check 'Plasma' mode.



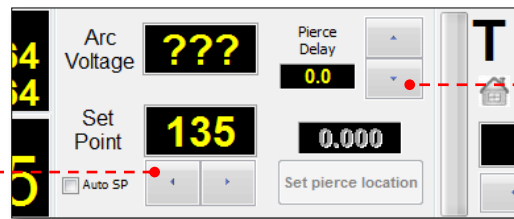
Check/Uncheck 'Expanded metal' if you intend to cut expanded metal. If cutting expanded metal be sure to also set the plasma generator to cut expanded metal. (Refer to plasma generator operator's manual)

Check/Uncheck 'Manual IHS' if you intend to use Initial Height Setting instead of pierce height.

If using manual IHS, jog the torch to the pierce height you want and then click the 'Set pierce location' button. Now click 'Raise Torch' to prevent accidental torch collision.



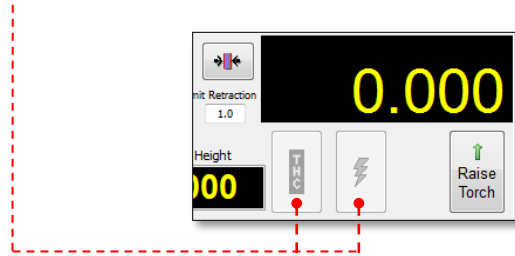
Set the Set Point ('Arc Voltage' in the cut chart, refer to plasma generator operator's manual).



If using thin metal settings, set the Pierce Delay here. (Refer to plasma generator operator's manual)

17 - Start Cycle

Enable the Tool (lightening button). Enable THC (THC button) if you want to use automatic torch height control, otherwise the torch will travel at the height it is currently set to.



Set up is complete. Put on your safety glasses, ensure that no one is too close to or touching the machine, and then click the 'Start Cycle' button.

