WHC Halo User Manual

www.bosstables.com 563-380-1535 www.info@bosstables.com

Boss Tables Warranty Program

Boss Tables strives to build quality equipment and is including with the purchase of a NEW plasma table a 2 year "bumper to bumper" warranty. Warranty replacement parts will need to be cleared with a factory authorized representative. Upon conformation you will be issued a new part or service rendered to alleviate the issue. Upon speaking with a representative from the "factory" (Resellers are not permitted) the part will be released as soon as possible and overnighted (if requested) at the expensive of Boss Tables. The RETURN shipping will be the responsibility of the customer. The customer will need to return damaged good in 5 to 7 business days.

"Bumper to bumper" includes-

All components on the machine directly sold to the consumer from Boss tables or an authorized reseller. Computer and controller components are included for the 24-month program. Frame and structural problems that are attributed to "nominal use"-see below. Hypertherm plasma power supply units are warranted by the manufacturer for 36 months from purchase date from our distributor. Hypertherm plasma torches are warranted for 12 months from purchase date from our distributor. Electronics including limit switches, E-stop switches, wire, and sensors are warranted for 24 months. Accessory package electronic and mechanical components are warranted for 24 months. Routers are warranted by the manufacturer. The warranties are only valid to the to the original purchaser. Damage from nominal use that is not "negligent"-See Below.

"Bumper to Bumper" Does not include-

Damage to the machine or components from mishandling while in the customers possession.
*Including- "Negligence"

- When machine arrives, the customer drops the machine or damages while installing.
- Improper installation of electrical components. Example- supplying the wrong voltage or single vs 3 phase where applicable.
- The customer loads an excessive amount of material. (Center legs or engineering / up fit is required for the cutting of anything greater than 2-inch-thick mild steel. With the approximate weight being 4000 lbs. (anything greater than 4,000 lbs material weight please call and confirm.
- Lightning and or power issues related to the customers location either caused by nature (Act of God) or power provider.
- Fire and or flood related incidents are not covered.
- "Kicking" of sheet clamps from material or dropping the material onto the bed of the cutting machine.
- The slats or the sacrificial cutting surface is not covered under the warranty.

- Any form of damage due to corrosion or galvanic or electrostatic corrosion is not covered by this warranty. It is the customer responsibility to treat the water to ensure that no corrosion is taking place in tank.
- The warranties are only valid to the to the original purchaser.
- Lifetime remote support is only valid to the original purchaser.

By purchasing a Boss Tables CNC Table or other machine you are committing to the warranty agreement and will have all the benefits of the warranty agreement. The starting date of the warranty program is when the machine leaves the possession of Boss Tables. The time will expire after a period of 2 years or 24 months.

Table of Contents

Safety

Users

Read and understand this manual thoroughly before operating the machine.

Supervisors

It is very important that a safe and appropriate working environment is provided for this Boss equipment and in compliance with applicable federal and local industry standards. It is imperative that programmers, machine operators and maintenance personnel be trained adequately in the use and care of the equipment. These employees should receive the proper instruction in order to have a complete understanding of the operation of this machine before beginning to program, operate or service it. Careful programming and debugging of new programs are essential for successful operation of this machine. Use program Stop Codes to stop machine motion for operator removal of parts or scrap. Never allow operators to place any part of their body into the machine while the machine is active. Ensure that all personnel understand the function and use of EMERGENCY STOP button.

Maintenance Personnel

Only qualified personnel should make repairs on this equipment. Use caution and follow Boss Tables procedures when working on the machine. Be sure to observe the following guidelines:

1. Before performing maintenance or repair, turn the power OFF and follow lock out/tag out (zero energy shutdown) procedures. Also, follow any lock out/tag out procedures applicable to your specific plant requirements.
2. Wear safety glasses and other personal protective equipment as required by applicable federal, local industry, and plant safety program standards.
3. Wear proper clothing. Do not wear watches, rings, jewelry, or loose-fitting clothes.
4. Read and review the manual carefully.
5. Be familiar with the operation of the machine.
6. Practice preventative maintenance. Inspect the equipment regularly and repair or replace worn components and tooling.
7. Always replace safety guards and other safety devices removed for service and make sure that they are fully functional before operating the equipment.
8. Never remove, jumper out or bypass a safety device to permit machine production.
9. Never place yourself in a hazardous situation to observe a problem and ask someone else to operate the machine. This could be a very dangerous and life-threatening situation.

Safety

Operator

This equipment has been designed with operator safety in mind (when used under normal operating conditions). The user must always be alert to the possibility of dangerous situations. Always exercise care and caution. Report any minor problems immediately, so that they can be corrected before becoming major difficulties. Only qualified personnel should make repairs on the machine.

1. Be familiar with the machine.
2. Be alert to the significance of the various warning indicators and be conscious of the functions of pushbuttons and other controls. Use the controls properly. Review and understand the operation of the EMERGENCY STOP function and the CYCLE STOP function.
3. Never operate the equipment unless it is in good working order.
4. Wear safety glasses and other personal protective equipment as required by applicable federal, local industry and plant safety program standards.
5. Wear proper clothing. Do not wear watches, rings, jewelry or loose-fitting clothes.
6. Avoid all moving parts of the machine or workpiece when setting up or operating the equipment. Never reach into the machine while it is active. Use the EMERGENCY STOP or CYCLE STOP function to stop machine motion.
7. Recognize and avoid unsafe operating conditions.
8. Maintain a clean work area. Avoid accidents by keeping work areas clean and neat.
9. Never leave the machine in an unsafe condition.
10. Never leave a machine running unattended.
11. Never remove or bypass safety devices.
12. Report any unsafe conditions, personal injury or machine problems immediately to your appropriate supervisor(s) and safety manager(s).
13. Never operate the machine with someone within a hazardous area.

Water table use

Keep the operator's body and clothing dry. Do not stand, sit, or lie in/on any wet surfaces when using this equipment. Never work in a damp or wet area without proper insulation against electric shock. Disconnect main power before servicing the torch, power supply or service connections to the plasma arc system, or any part of the machine bed. Wear adequate personal equipment (overalls, gloves, safety boots etc.) when operating the machine. Remove or secure articles of clothing, such as ties and loose sleeves, which may catch or be drawn into moving machinery.

Eye Protection

LENS SHADE:

- Arc Current AWS (USA) ISO 4850
- Up to 100A No. 8 No. 11
- 100 – 200A No. 10 No. 11 – 12

- 200 – 400A No. 12 No. 13
- Above 400A No. 14 No. 14

Medical treatment facilities and a qualified first aid person should be available for immediate treatment of flash burns to the eyes and skin. It is recommended that the cutting area be prepared in such a way as to minimize the reflection and transmission of ultraviolet radiation. Walls and other surface areas should be painted in dark colors to reduce reflection. Protective screens or curtains may be installed to avoid unnecessary ultraviolet transmission.

 **Warning**

The plasma arc cutting process produces rays that can burn eyes and skin. Always wear eye protection with appropriate lens shades.

 **Noise**

The noise levels generated during plasma arc cutting may be as high as 105 decibels. This depends on the distance from the machine, arc, plasma torch nozzle design, gas velocity, material type, and plate thickness. Boss Tables recommends that each user check the sound levels in his own shop under normal operating conditions. Based on those findings, provide adequate ear protection to all personnel who must work near the machine, in accordance with applicable local, state, and federal industry standards. Noise levels that can cause discomfort or damage to hearing will vary greatly from one individual to another. We recommend that ear protection be furnished to any worker who requests it, regardless of applicable industrial standards or tested noise levels. Exposure to noise from the cutting process can damage hearing. Wear appropriate ear protection when operating the machine or when working in the proximity of the machine.

 **Safety Devices**

Plasma arc units are provided with certain safety interlocks designed to prevent equipment damage and/or personal injury. Never short out or in any way attempt to defeat the safety interlock devices. All exposed electrical connections must be covered with the proper insulation material. Safety devices must be regularly checked for proper operation and replaced immediately if found to be inoperative.

 ***Warning***

Never attempt to operate the plasma unit with any of the power supply covers not in place. This is extremely hazardous to the operator and any other person in the area. It also prevents the equipment from properly cooling critical components and could result in equipment damage.

Safety

 ***Risk of Electric Shock***

Plasma cutting equipment uses high open circuit voltages to initiate the plasma arc. Normal load voltages are higher than experienced with other types of welding equipment. Extreme CAUTION must be exercised when operating or servicing this equipment.

Input Connections

A wall mounted line isolating switch, fused as required by local electrical codes, must be fitted as close as possible to the plasma arc power supply.

 ***Danger***

Always verify that ALL electrical supplies are isolated before undertaking any service or maintenance work. The machine may have more than one electrical supply.

 ***Warning***

Plasma arc can cause injury and burns. Verify that no person is in the proximity of the plasma torch at any time and that the plasma system is switched on. Serious burns and or electrical shock hazards exist, even when the plasma cutting system is not active.

 ***Warning***

Frequently inspect the cable for damage or cracking of the cover or sleeve. ***Bare wiring can kill***! Replace damaged cable immediately.

Grounding

Be sure all ground lugs are of adequate size to carry the rated current load. Make all connections tight to avoid resistance heating. Connect the material grid of the worktable to a good earth ground.

Boss CNC Pro Plasma Tables require a dedicated earth ground that is isolated from all other electrical systems.

Fumes and Air Contamination

Proper precautions must be exercised to prevent the exposure of others in the vicinity totoxic fumes that may be generated while plasma cutting. Certain chlorinated solvents such a perchloroethylene and trichlorethylene will decompose under ultraviolet radiation to form phosgene and other gasses. Care must be taken to avoid the use of these solvents on materials being cut with plasma arc cutting equipment. Containers of these solvents and other degreasing agents should be removed from the

immediate area around the plasma arc. Metals coated with or containing significant amounts of lead, cadmium, zinc, mercury or beryllium can produce harmful concentrations of toxic fumes when the plasma arc cuts. Adequate local exhaust ventilation must be used, or the operator must be supplied with special equipment to guarantee a supply of fresh air such as a respirator or air supplied helmet. Metals coated with materials that emit toxic fumes must not be cut unless:
1. The coating is removed prior to cutting.
2. The area is adequately ventilated.
3. The operator is supplied with fresh air breathing equipment.

Air Contamination

The plasma cutting process generates large quantities of hot metal dust and fumes that would be hazardous if uncontrolled. The gases listed below either are produced normally during plasma arc cutting or can form under certain conditions.

Ozone

Ozone is produced by the reaction of the plasma arc's ultraviolet radiation with oxygen in the air. Uncontrolled, excessive levels of ozone can constitute a hazard. When there is proper venting to the outside and the machine's internal ventilation system is functioning properly, there is adequate control of ozone during torch cutting.

Nitrogen Dioxide

Nitrogen dioxide gas is produced when nitrogen and oxygen in the air pass through the electric arc. A hazard may exist if uncontrolled, excessive levels of nitrogen dioxide are formed. With proper venting to the outside, the machine's internal ventilation system is adequate to control nitrogen dioxide during torch cutting, if the system is functioning normally.

Acetyl Chloride

Acetyl chloride gases form in the air surrounding the plasma arc when the airborne vapors of chlorinated solvents or degreasers decompose upon being exposed to the ultraviolet radiation of the arc. A hazard may exist if uncontrolled, excessive levels of acetyl chlorides are formed. A pungent aroma like chlorine bleach is the first sign that these gases are being produced. Shut down the plasma arc cutting system immediately if you detect the acetyl chloride odor. Do not resume cutting until you locate and control the source of the vapors. Various cleaning solvents and vapor degreasers contain chemicals that decompose rapidly when exposed to ultraviolet radiation. If the solvents, cleaning solutions, or vapor degreasers used in the shop contain any of the following chemicals, do not use them near the plasma arc cutting system:
1. trichloroethylene
2. trichloroethane
3. perchloroethylene

4. perchloroethane
5. trifluoro-trichloroethane (fluorocarbon-113)

These chemicals also decompose into small amounts of the toxic gases phosgene and chlorine. You will notice the acetyl chloride odor long before phosgene or chlorine levels become harmful. The vapors can decompose up to several feet away from the arc. Do not use or store chlorinated solvents, cleaning solutions, and vapor degreasers close to the machine, where the vapors can enter the torch-cutting area.

Note: It may prove advisable to provide separate ventilation for the solvent/degreaser storage area.

Metal Fumes

Metal fumes are produced when the plasma arc vaporizes the metal. A hazard may exist when uncontrolled, excessive levels of metal fumes are produced some vaporized metals form toxic gases. These metals may be in their pure metallic state, in an alloy, or in a coating such as paint or plating. Metals that are known to produce toxic fumes include beryllium, cadmium, lead, manganese, mercury, and zinc. Beryllium products require particular care, because their fumes are highly toxic. If there is proper water level, there should be adequate control of metal fumes during torch cutting.

Metal Dust

Metal dust is formed as metal vaporizes during torch cutting. A hazard may exist when uncontrolled, excessive levels of metal dust are produced. If there is proper water level there should be adequate control of metal dust during torch cutting. Fire, Explosion, and Burn Prevention, all combustible materials must be removed from the immediate cutting area to at least 35 feet away. Appropriate fire extinguishing equipment must be available in the immediate cutting area. After cutting, be sure to allow the metal to cool sufficiently before handling or before allowing contact with combustible materials. Never plasma cut empty containers that have held toxic or potentially explosive materials. Those containers must be thoroughly cleaned according to national standards prior to cutting or welding. Never plasma cut in an atmosphere that contains heavy concentrations of dust, flammable gas, or combustible liquids.

Hot Surfaces

Assure that the bed is free of obstructions and no person or articles of clothing are in the proximity of moving parts when the machine is in operation. This safety precaution also applies when the machine is manually moved and when the plasma system is off.

Warning

Since plasma arc cutting produces hot metal, sparks, and slag, precautions must be taken to prevent fire or explosions.

Safety

Warning

Components may remain hot for a considerable period of time. Always wear gloves to remove components and scrap from the bed.

Heat Affected Zone

Plasma arc cutting creates a Heat Affected Zone (HAZ) around the cut edge of the workpiece. Until the hot edges cool, the HAZ will burn an unprotected hand severely.
1. When removing produced parts or skeletons from the machine, operators should wear heat-resistant, gauntlet-type gloves.
2. The torch and cutting slate bars become hot during torch cutting. Avoid contact with these components unless you are wearing heat-resistant gloves.

Sparks

Sparks form as the plasma arc torch vaporizes metal. These sparks are tiny droplets of extremely hot molten metal and are a possible fire hazard. The volume of sparks formed and the area over which they are scattered depend on several variables. These variables include the type and thickness of the material being cut, the cutting current, and the feed rate. Where practical, keep all combustible material at least 35 ft. away from the plasma arc work area. Where this is not practical, protect all combustible materials with close fitting, flame proof covers or shields. Protect wooden or other combustible floors by covering them with sand or installing fire-resistant shields. Shield any wall openings, floor openings, cracks, ducts, or conveyors within 35 ft. of the torch to prevent sparks from passing into adjacent areas.

Burn Prevention

High intensity ultraviolet and infrared radiation is produced by the plasma arc and is of similar intensity to typical high current welding arcs. This radiation is damaging to the eyes and skin. As the operator comes closer to the torch, the level of exposure increases rapidly.

Cutting Aluminum on a Water Table

When cutting aluminum with plasma over a water table, hydrogen bubbles form. These bubbles can accumulate and get trapped beneath the workpiece causing an explosion in the presence of an ignition source (such as an arc ignition). Do not leave workpieces on the table if it is not being cut at that moment. For example, do not leave a sheet on the table overnight. Cut aluminum, and all materials, in a well-ventilated area. Use a fan to increase air circulation beneath the workpiece on the plasma table. Make sure the fan is turned on for several minutes prior to initiating an arc on the plasma table.

Allow at least one inch of clearance between the aluminum plate and the water level. Use pointed slats to improve air flow. Consider installing an aerator, circulator, or filtration system if cutting consecutive sheets of aluminum, anything that agitates the water to break up the bubbles of hydrogen. Clean out fine aluminum particles from the water table after each use. Never lower water into tank after cutting, allow 24 hours. Keep water levels full to avoid gas buildup in the tank.

Light and Radiant Energy

When it is necessary to look directly at the arc for diagnostic purposes, do so briefly. Use shade #10 welding glass (for up to 200 amps) or shade #12 (for 200 amps). During operation, use a shade not less than #8. Ultraviolet rays and other radiant energy reflected off the workpiece can produce sunburn. Therefore, when plasma arc cutting is being performed, anyone working within 25 feet of the arc should wear an approved, protective full-face mask, a long-sleeved shirt, gloves, and long pants. Shield personnel at nearby workstations from accidental exposure to radiant energy using non-reflective, fireproof enclosures, open at the top and at floor level to allow air to circulate freely. The pilot arc in the plasma cutting systems is initiated and stabilized by a high voltage signal. This signal can create electromagnetic interference. As with any equipment that can create such interference (e.g., microwave ovens and TIG welders), people who have implanted heart pacemakers must exercise caution when working near the equipment. Boss Tables recommends that a person with a pacemaker who works near where plasma arc cutting is being performed should wear a Holter monitor for one day of work to record the existence of electromagnetic fields. A qualified doctor should review the recorded data with the pacemaker manufacturer to determine whether the worker can safely continue working in the area on which the study is based.

Compressed Gas Equipment

Gas cylinders should be mounted securely to a wall or other stable supporting device. Compressed gas cylinders must be handled and used in accordance with appropriate national safety standards. Never use a cylinder that is physically damaged or leaks. Never move or transport a cylinder without the protective valve cover in place. Never use a gas cylinder or its contents for any other purpose than that for which it is intended. Never lubricate cylinder valves with oil or grease. Never allow electrical contact such as welding arcs with cylinders. Never expose cylinders to excessive heat, sparks, slag, or open flames, which may cause rupture. Never use hammers, wrenches or other tools to open stuck valves. Send these cylinders back to the supplier.

Pressure Regulators

All regulators used to operate plasma equipment must be maintained in proper working condition. Faulty equipment can cause equipment damage or operator injury. Faulty equipment must be serviced at the manufacturers designated facility by trained repair technicians. Never use a regulator for any other gas than that for which it is intended. Never use a regulator that leaks, excessively creeps, or is physically damaged in any way. Never attempt to lubricate a regulator with oil or grease.

Hoses

Gas hoses used for plasma arc cutting systems adhere to the following color coding:
Red……Acetylene
Green……..Oxygen
Black…….Inert gases and air
Replace any hose that is damaged by physical abuse or from sparks, heat or open flame.
Lay hoses out straight to prevent kinks. Coil excess hose and place out of the way to prevent loose connections, or other damage. Keep hose lengths to a minimum to prevent damage, reduce pressure drop and prevent possible volume flow restriction. Please refer to national standards for more information on hoses.

Additional Safety Information

The general safety information presented in this chapter does not constitute a complete list of safety instructions for any particular configuration of the Boss CNC Plasma Table. Specific equipment being used by the customer and its particular application in the customer's factory may require supplementary safety information.

Note: It is the responsibility of the customer's company to make sure safety information covering the equipment being used and its particular application is available to personnel operating and maintaining the equipment and is read by them.

SAFETY STANDARDS PUBLICATIONS

It is recommended that companies using the kind of equipment covered in this manual consult the applicable Safety Standards publications available from the agencies listed below:
OSHA
Superintendent of Documents
U. S. Government Printing Office
Washington, DC 20402-9371, USA
Tel: (202) 512-2457

ANSI
American National Standards Institute
11 West 42nd Street
13th Floor
New York, NY 10036-8002, USA
Tel: (212) 642-4900
Fax: (212) 398-0023

NFPA
National Fire Protection Association
P.O. Box 9101
1 Batterymarch Park
Quincy, MA 02269-9101, USA
Tel: (617) 770-3000
1-800-344-3555
Fax: (617) 770-0700
SAFETY 27

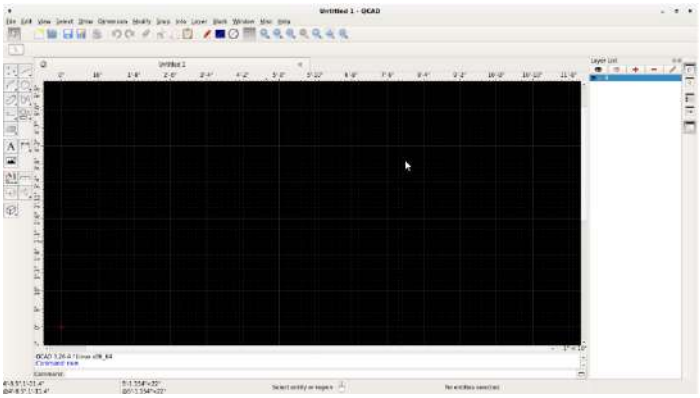
Powering up

Your WHC table's controller is located on the home side (MVP) or YA side (WHC) of the table. The main power toggle switch is located on the side of the controller.

Power on the computer located on the cabinets left side. Your table will now be on and ready to run.

QCAD Basics

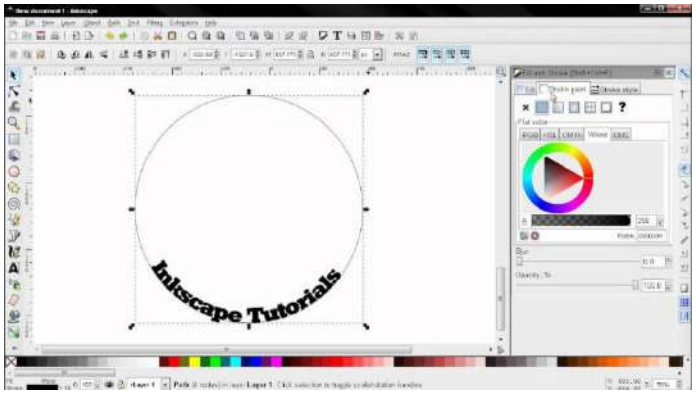
QCAD is a free, open-source application for computer aided drafting (CAD) in two dimensions (2D). With QCAD you can create technical drawings such as plans for buildings, interiors, mechanical parts or schematics and diagrams. QCAD works on Windows, macOS and Linux. For information and tutorials visit www.qcad.org/en/



Inkscape Basics

The design process may begin by doodles on a napkin, a sketched mindmap, a photo of a memorable object, or a mockup in software which really wouldn't work to complete the project. Inkscape can take you from this stage to a final, professional-grade design format which is ready for publication on the web or in physical form.

If you are new to the process of creating vector graphics it may feel different, but you will quickly be pleased by the flexibility, and power Inkscape offers. Vector design is often the preferred method of image creation for logos, illustrations and art which require high scalability. The Inkscape application is used across a wide variety of industries (marketing/branding, engineering/CAD, web graphics, cartooning) and individual uses. For information and tutorials visit www.inkscape.org

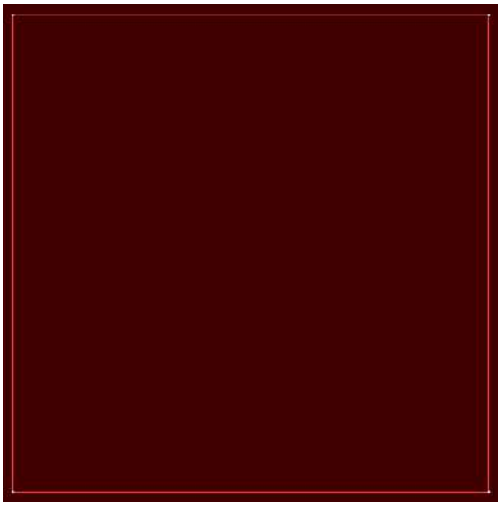


Creating drawings

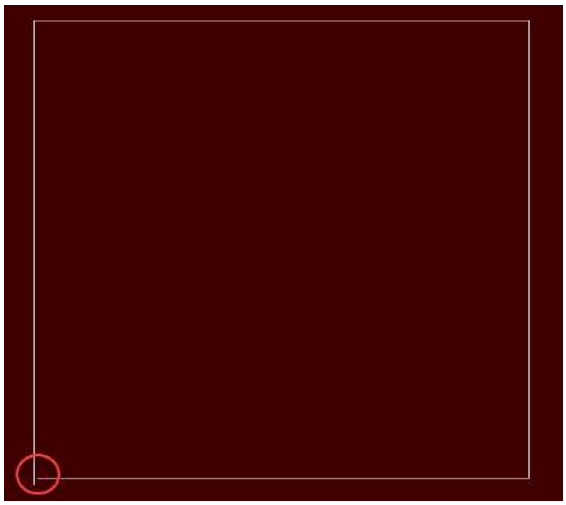
In order to create usable drawings in either a CAD or art application certain fundamental principles **must** be adhered to otherwise SheetCam will not be able to manipulate the resulting file. Outlines must be properly closed. A closed shape is any shape where the lines completely enclose an area. A square is a simple example of a closed shape. It has an inside and an outside, so SheetCam needs to work out which side you want to cut. A simple line is an open shape as there is no inside or outside therefore SheetCam treats it differently. So, what is meant by 'properly closed'? Let's take a simple example. You draw a rectangle using four lines. However, two of the lines don't quite meet. On the screen it looks fine unless you zoom in very close. When you load the drawing into SheetCam it will recognize that two ends do not meet and it will assume that the shape is open so you can't cut it out. All open lines will be shown on the screen in white. If your lines are very close (as defined under 'Options/Application options/Drawing import tab/Import link tolerance') SheetCam will automatically move the ends so that they touch.

Note: This should not be used to compensate for poor drawings!

Properly drawn files will open in SheetCam showing outside areas in red and inside areas in yellow.



A correctly closed square



An incorrectly open square

Sheetcam

SheetCam is a feature packed CAM package. SheetCam is suitable for milling, routing, plasma, waterjet, laser and oxy-fuel cutting. SheetCam accepts data in the form of DXF files (CAD drawings), HPGL files (line art), SVG files and converts them into useful toolpaths for cutting. SheetCam will allow the nesting of parts and has features for copying, duplicating, rotating and mirroring parts to reduce wastage to the minimum. SheetCam can also allow for parts that are not aligned perfectly along the machine axes by aligning the drawing to the actual part. SheetCam will show cutter paths, rapid moves, layers etc. and the part can be rotated in three dimensions in the view panel to check for errors prior to machining.New users are sometimes confused as to the 'correct' way to get SheetCam to manipulate their drawings/files. The information below is intended to be a guide to the optimal steps in the process from drawing to compiled G-code. There are normally three or four steps in any CAD/CAM process:

1. Create your part/design in either a CAD package or an 'art' application like Inkscape or Corel Draw.

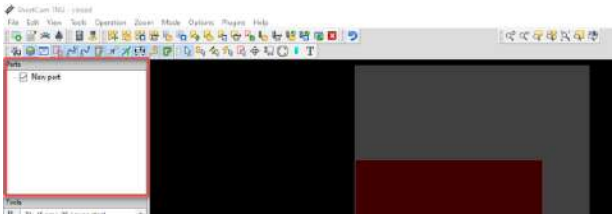
2. Open/import the drawing as a DXF file into the CAM program and set up your tooling sizes and cutting operations. Once these are set run the 'post processor' to generate the required G-code.

3. (Optional but very worthwhile) Run the generated G-code in the CNC simulator

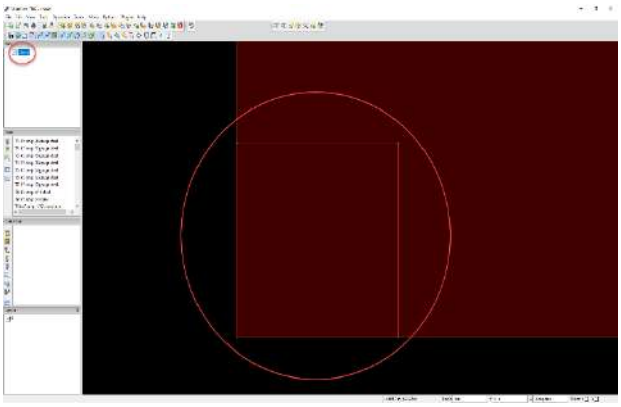
4. Once the code has been verified open the G-code file in your CNC control package and run the program to create the part.

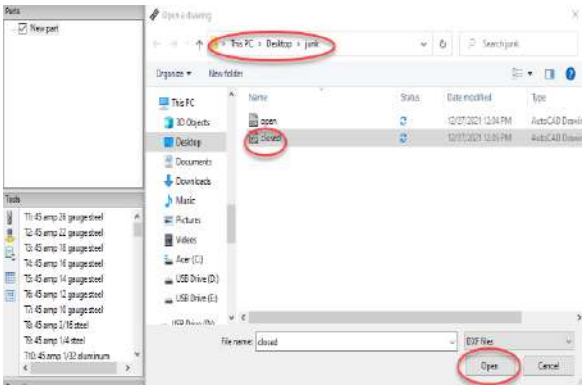
For more information and tutorials visit www.bosstables.com/cnc-machin-training-center

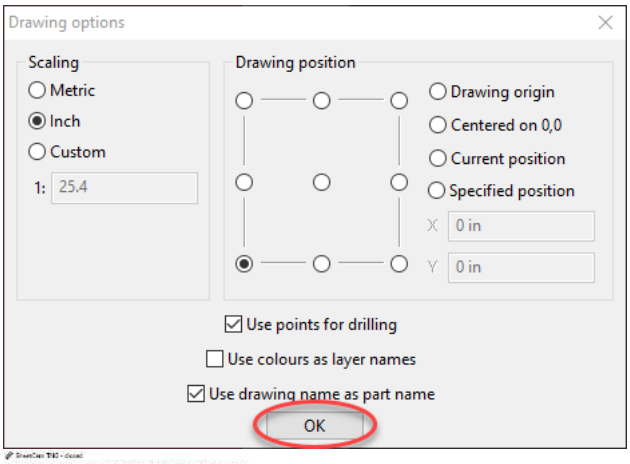
Loading a New Part



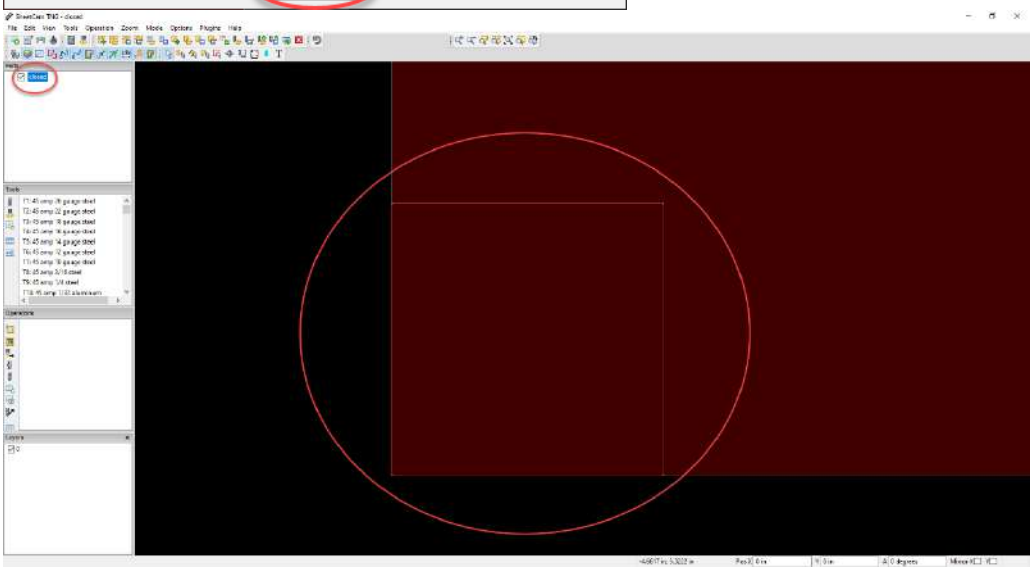
Loading a file on SheetCam is a simple process. Multiple files may be added and are controlled by clicking on the selected part. To load a new part into SheetCam right click in the parts box and select ***New Part.*** Select the location of the file and click ***Open***. A drawing options window will open, select where on the material to place the file and click ***OK.*** Your file will then be visible on the materials screen.







It is important to select the correct scaling when loading a file. Most will be defaulted as inch. With the file loaded the parts box will populate with the file you have selected to load. Generally, the drawing position window will load the correct settings for your file.



Layers

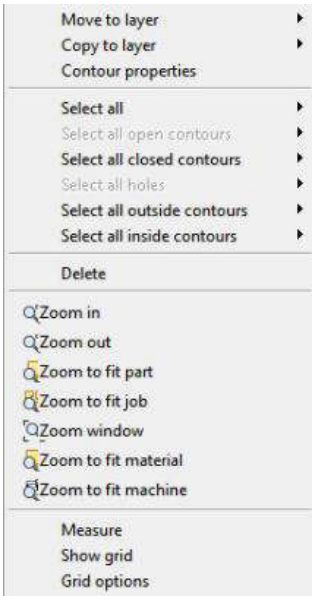
Edit contours  Use this function to move or copy contours to different layers. Click on a contour to select it.

Hold the ***Control*** key to select multiple contours.

Note: Hold down the ***Shift*** key to scroll the screen with the mouse. Double-click anywhere in the 'view' panel with the left mouse button to return to the 'plan' or overhead view.

Note: This option can also be activated by clicking on the contour button in the toolbar.

Right click options 'Right clicking' while using the 'Edit Contours' selection tool will bring up the following menu screen.

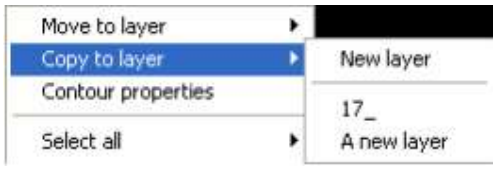


Move to layer 'Hovering' over or clicking this menu item will open the following pop-out dialog box.

Choose either a 'New layer' or an existing layer. This will move the selected contour(s) to the chosen layer.

Choosing 'New layer' will open the following window.

Enter a new name for the layer and click ***OK*** .



Copy to layer 'Hovering' over or clicking this menu item will open the following pop-out dialog box.

Choose either a 'New layer' or an existing layer. This will copy the selected contour(s) to the chosen layer (but leave the original contour(s) intact on the original layer).

Choosing 'New layer' will open the following window.

Enter a new name for the layer and click ***OK***.

Contour Properties Clicking this function opens a dialog box that shows how many contours you have selected and which layers they are on.

Select options 'Hovering' over or clicking on any items in this menu area will open the following pop-out dialog box.

Select all This will select all contours from the selected layer(s).

Select all open contours This will select all open contours from the selected layer(s).

Note: If there are no open contours this option will be 'greyed' out. Select all closed contours This will select all closed contours from the selected layer(s).

Note: If there are no closed contours this option will be 'greyed' out. Select all holes This will select all open holes from the selected layer(s).

Note: If there are no holes contours this option will be 'greyed' out. Select all outside contours This will select all outside contours from the selected layer(s).

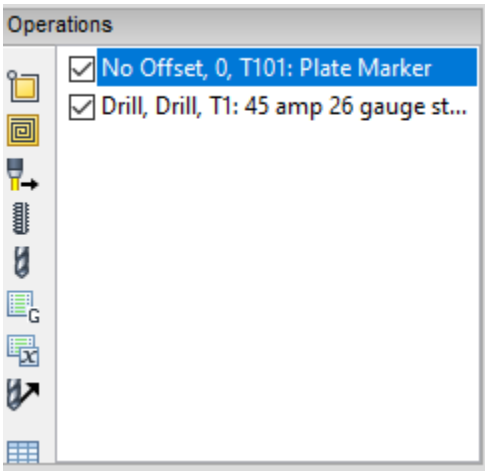
Note: If there are no outside contours this option will be 'greyed' out. Select all inside contours This will select all inside contours from the selected layer(s).

Note: If there are no outside contours this option will be 'greyed' out. Zoom controls Clicking on any of these items will zoom the view as required. Measure To use the measure tool click on the point you want to measure from.

Peck Pierce (Drilling)

Peck pierce is a useful tool for drilling and tapping material without hardening the material. It will briefly fire the torch creating a divot or center punch. To create a drilling operation, move the desired part of the drawing to a new layer. Select the layer to be drilled. In operations select the drilling operation. Select the layer you used for the drill and use the lowest amperage with the lightest material and select ***OK***. The toolpath will now be created as a dot in the middle of the selected drawing.

Scribing



Like drilling, the parts that will be scribed must be moved to a new layer. Select the new layer and go to jet cutting in operations. Use no offset and the layer to be used in scribing. Next select part 101, Plate Marker. Lead ins and out are not used and should be set to none, click ***OK***. In operations move the scribed operation to the first operation as it will need to be the first toolpaths used before cutting. To do this simply click and hold on the scribe operation and move it to the top in operations.

Nesting

This feature allows you to arrange multiple parts on the same workpiece.

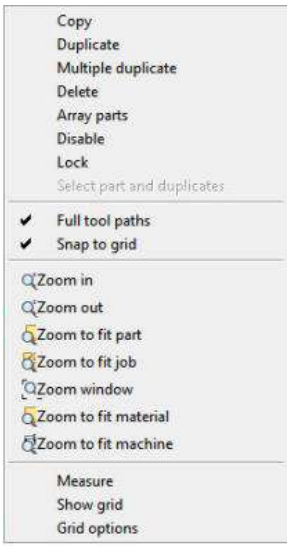
Click on the part to select it then drag it to the new position. You can also copy and duplicate the part with this function.

TIP: 'Copy' and 'Duplicate' have different functions as follows:

- Copy: This will make a complete copy of the drawing and operations. The copy will be completely independent of the part it was copied from.
- Duplicate: This will perform a 'step and repeat' of the original drawing and operations. Any changes made to the original will be reflected in the duplicate.

Note: Hold down the ***Shift*** key to scroll the screen with the mouse. Double-click anywhere in the 'view' panel with the left mouse button to return to the 'plan' or overhead view.

Note: This option can also be activated by clicking on the button in the toolbar.



Right click options 'Right clicking' while using the 'Nesting' tool will bring up the following menu screen.

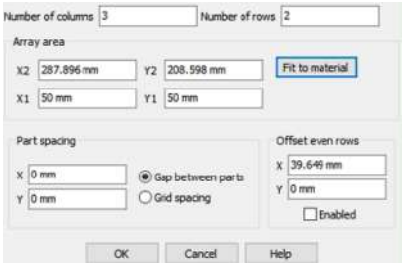
Copy Click on this function to copy the part. A part consists of a drawing with its associated operations. Copy will make a complete copy of the drawing and operations. The copy will be completely independent of the part it was copied from.

Duplicate Click on this function to duplicate the part. Duplicate will perform a 'step and repeat' of the original drawing and operations. Any changes made to the original will be reflected in the duplicate.

Delete Clicking on this function deletes the selected item.

Array parts Clicking on 'Array' will open the following dialog box.

This creates an array of duplicates (step and repeat). Select the part(s) you want to array using 'Ctrl-click' or drag a box to select multiple parts then 'right-click' and select 'Array'.



Number of columns/rows Enter the number of columns and rows required. A matrix of parts will be generated with the number of columns and rows specified.

Array area Use the boxes in this section to specify the size of the array area. X1 and Y1 are the bottom left corner and X2 and Y2 are the upper right corner.

The boxes can be used in two ways:

1. If you enter a value in the columns/rows boxes above or change the part spacing or offset rows then the boxes show the area the array will cover.

2. If you directly enter a value the number of columns/rows will be updated to fit as many parts as possible into the specified area.

Fit to material The 'Fit to material' button sets the area to the size of the material and then updates the number of columns/rows to suit.

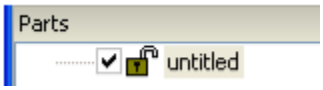
Part spacing If 'Gap between parts' is selected then X and Y represent the space between the duplicates.

Note: To calculate the spacing the parts are assumed to be rectangular, with the rectangle being the smallest that will completely enclose the part. For some oddly shaped parts you may well end up using a negative X or Y to minimize wasted material.

If '**Grid**' is selected, then X and Y represent the horizontal and vertical increment. For instance, if X is 2" then the duplicates will be placed every 2" in the X axis.

Offset even rows If you are laying out irregularly shaped or circular parts then simply placing them on a regular grid would be inefficient. Instead, you can offset alternate rows to pack them in more tightly. If an array does not come out the way you want it to you can always undo (Edit->Undo or Ctrl-Z) the operation. **Enabled** Clicking the 'Enabled' check box enables the 'Offset even rows' function. Dimensions can be left in the boxes and used as required by simply toggling the 'Enabled' check box on or off.

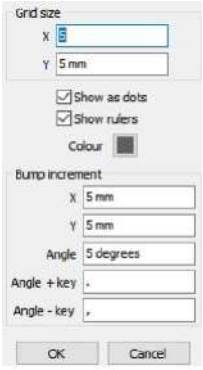
Disable/Enable A disabled part will not be cut. Using the right-click menu is the same as using the check box next to the part name.



Lock/Unlock When in nesting mode the part tree shows a padlock symbol next to the part name (see below). This symbol indicates if the part is locked or not. Locked parts cannot be moved while in nesting mode. Lock/Unlock appears in the right-click menu in the part tree and the main graphics windows in nesting mode.

Snap to grid Clicking on this item toggles the 'Snap to grid' function on or off.

Grid options:



Grid size If you set the grid size to be greater than zero (0) when you move the part it will always 'snap' to the nearest multiple of the grid size.

Snap to grid - check the box to enable 'snapping'.

Bump increment When moving a part, you can 'bump' it a fixed amount using the arrow keys. 'Bump increment' sets the amount each 'bump' moves the part.

Zoom controls Clicking on any of these items will zoom the view as required.

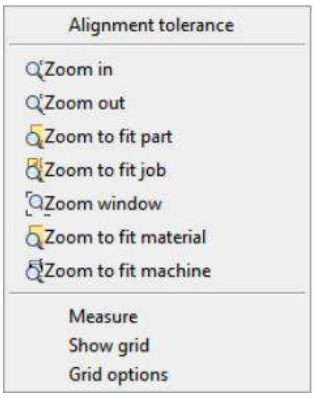
Measure To use the measure tool click on the point you want to measure from.

Align Click on this function to align the drawing in SheetCam to the work on your machine. For example, imagine you have a component or piece of material (here after called 'the job') that you have clamped to the table. Instead of carefully positioning 'the job' to line up with the drawing/machine axes you can clamp 'the job' roughly in place then line the drawing up to match it. Pick two points on 'the job' that are spaced as far apart as possible and measure their coordinates by jogging the machine and using an edge finder or lining up a pointed cutter by eye. Click on ***Align*** and then click on the first point on the part in the view window and the following dialog box will appear.

Enter the actual coordinates of this point and click ***OK***. Now click on the second point on the part in the view window and enter the coordinates of that point and click ***OK***. The drawing will be moved and rotated to suit 'the job'.

Note: Hold down the key to scroll the screen with the mouse. Double-click anywhere in the 'view' panel with the left mouse button to return to the 'plan' or overhead view.

Note: This option can also be activated by clicking on the button in the toolbar.



Right click options 'Right clicking' while using the 'Align' selection tool will bring up the following menu screen.

Align tolerance Clicking on this item will bring up the following dialog box:

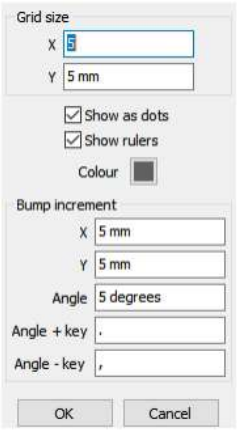
Enter the required value and click 'OK' to accept.

Zoom controls Clicking on any of these items will zoom the view as required.

Measure To use the measure tool click on the point you want to measure from.

Show grid This toggles the grid display on or off.

Grid options:



Grid size If you set the grid size to be greater than zero (0) when you move the part it will always 'snap' to the nearest multiple of the grid size.

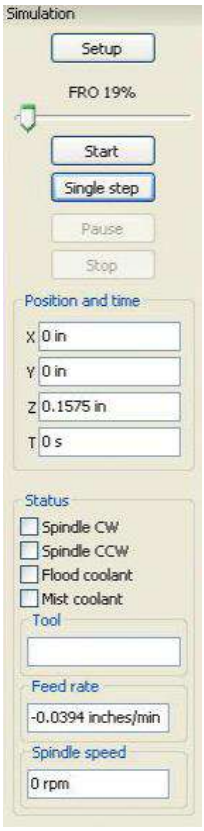
Show as dots Shows the grid as a matrix of dots.

Show rulers Turns the rulers on.

Bump increment When moving a part, you can 'bump' it a fixed amount using the arrow keys. 'Bump increment' sets the amount each 'bump' moves the part.

Run simulation

Clicking on this item will run the 'Simulation plugin'.



Note: Hold down the key to scroll the screen in 3D with the mouse. Double-click anywhere in the 'view' panel with the left mouse button to return to the 'plan' or overhead view.

Note: This option can also be activated by clicking on the button in the toolbar.

Setup Clicking on the 'Setup' button will open the following dialog box:

Special options:

Update rate Enter the required update rate in the box. Note: Increasing the update rate makes movement smoother but uses more CPU power.

Rapid feed Enter the required rapid feed rate in the box.

Maximum FRO Enter the maximum feed rate override (FRO) amount in the box as a percentage.

Tool polygon detail Use the slider to control the tool polygon detail. Note: Increasing the polygon detail makes the tool outline smoother but uses more CPU power.

Tool colours Click on a box next to an item to open the colour picker window then select a new colour.

Tool diameter:

- **Scale size** Enter the scale size as a percentage.
- **Minimum size** Enter the minimum size as a dimension. Machine type Select the machine type using the drop-down menu.

Enabled This check box toggles the plugin on or off.

Edit macro Clicking this button will open an editing window to allow editing of the plugin.

Help 'Help' opens the help file at the relevant location.

FRO Slider Move the slider to control the amount of feed rate override (FRO). The further to the right you move the slider the faster the simulation will run.

Start Clicking the 'Start' button will start the simulation.

Single step Clicking the 'Single step' button will step through the simulation one move at a time.

Pause Clicking the 'Pause' button will pause the simulation.

Stop Clicking the 'Stop' button will stop the simulation.

Position and time The four boxes show the current location of the tool in the X, Y and Z planes and the time taken to get there.

Status A check mark in a box indicates that specific item is in use at that time.

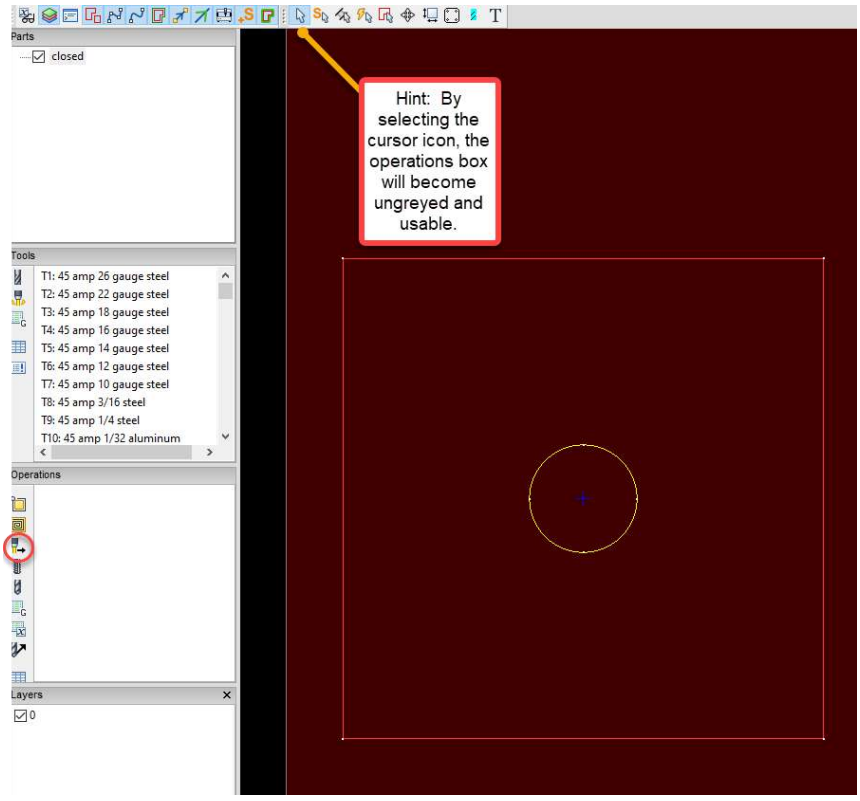
Tool This box displays the current tool in use.

Feed rate This box displays the current feed rate.

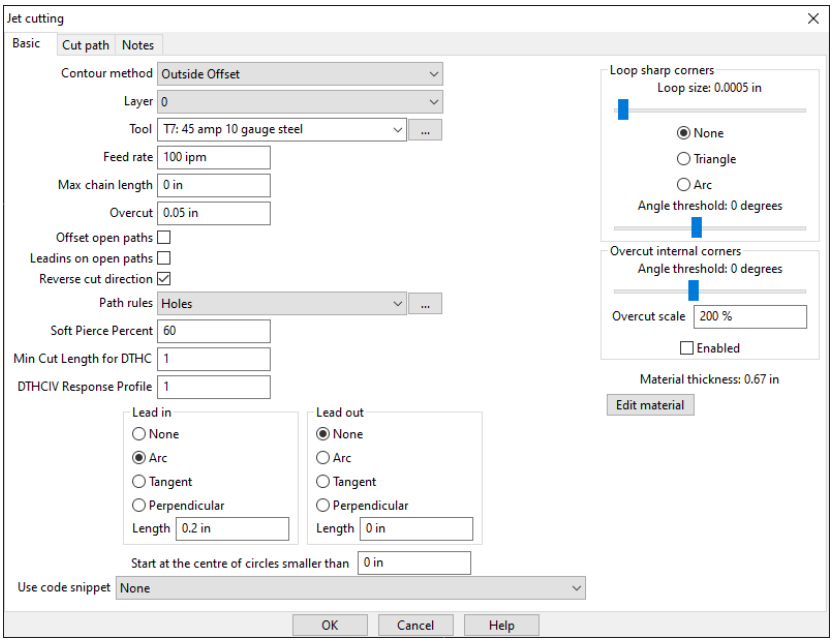
Spindle speed This box displays the current spindle speed.

Creating a Toolpath

With the cursor selected in the tool ribbon, select the jet cutting tool in operations.



This will open your jet cutting menu where setup can begin for creating a proper toolpath.



Contour method Select the contour method required using the drop-down menu.

Note: Outside Offset puts the cut line on the outside of the drawn line, Inside Offset on the inside and No Offset is centered. Selecting Outside Offset will automatically set outside lines to outside offsets and inside cuts to inside offsets.

Layer Select the layer you wish to apply the contour to using the drop-down menu shown.

Tool Select the correct tool using the down menu shown. ***Important! Only select the tool and material that match what you are using! Eg 45 amp shielded consumables, Power unit at 45 amps and in this case 10 ga mild steel.***

Edit Clicking on the button to the right of the 'Tool' box (the one with three dots) calls up the 'Tool Edit' dialog box.

Feed rate Enter the required feed rate here. Max chain length Specify the maximum length allowed for chain cuts in this box. SheetCam will then use this value when it calculates the chains between parts. Piercing wears consumables so if possible SheetCam will try to cut from the end of one outline to the start of the next. The maximum chain length defines the maximum distance to cut. If the distance is further than this value, the tool will lift and move to the next cut start as normal.

Offset open paths Open paths are always offset to the right of the start point. That is, if you were standing at the start point looking down the line the offset would be to your right. To reverse the side of the offset, move the start point to the other end of the line. Inside/outside offset make no difference as a line doesn't have an inside or outside.

Note: If you have climb cut selected the cut will start at the opposite end of the path to the start point. I know this is counter intuitive, but it is the only way to maintain the 'offset to the right' rule.

Reverse cut direction Check this box to reverse the cut direction.

Loop sharp corners When plasma/flame/water jet/laser cutting the exit point of the jet lags behind the entry point. This causes rounding of sharp corners. To get around this problem the corner is cut in a loop instead. Set up a simple jet cutting job on a square and you will see how this works. Triangle loops take up the least space but are relatively slow to cut due to the sharp corners. Arc loops are smoother to cut but take more room. If an arc loop would overlap an existing cut, SheetCam tries a triangle. If the triangle still won't fit, the corner is not looped.

Note: Loops will be left out if they would overlap an existing cut path.

Loop size The slider adjusts the size of the loop used above. This is an arbitrary function and can only be established using trial and error methods as it is dependent upon flame/jet size on your machine. Set the slider, make a test cut and adjust as required. Use the radio buttons to select the type of loop.

Edit material Clicking on the 'Edit material' button calls up the 'Material' dialog box.

Lead in Select the type of lead in required using the 'radio' buttons and enter the size in box.

Note: Size refers to the length of the lead in.

Lead out Select the type of lead out required using the 'radio' buttons and enter the size in box.

Note: Size refers to the length of the lead out.

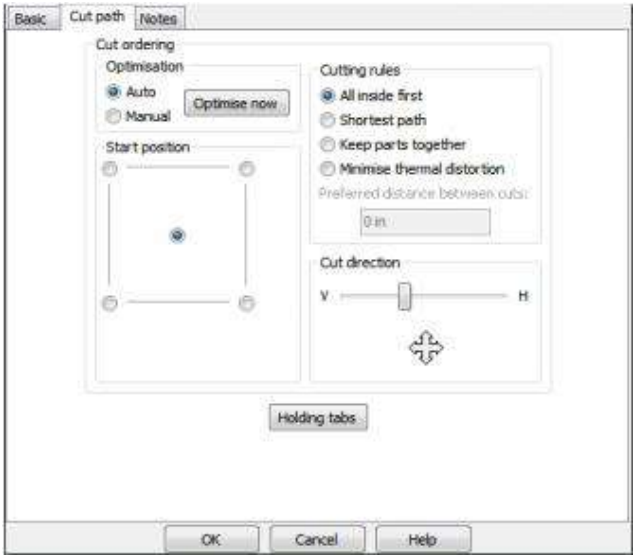
Use code snippet Select the required code snippet from the drop-down menu.

Note: The code snippet needs to be previously defined using the 'Tools/New code snippet' function. The snippet is inserted into the code at the start of the cut just before the cutter plunges to depth.

Help 'Help' opens the help file at the relevant location.

Note: This dialog box can also be accessed via the 'Create a new plasma cut operation' button located in the left-hand vertical toolbar.

Cut path tab:



Note: All units based on the selection in the 'Options\Application options\Units' menu.

Cut ordering / Optimisation

Auto/manual Select auto or manual path optimization using the radio buttons.

Note: If 'Auto' is selected the cut sequence is calculated automatically. SheetCam tries to minimise rapid moves while adhering to the rules shown under ‘Cutting rules’ below. If 'Manual' is selected, you can manually set the cut sequence (by editing the start points).

Optimise now This button optimizes the cut path immediately. This is useful to see the effect of any changes or to create a starting point for manual editing.

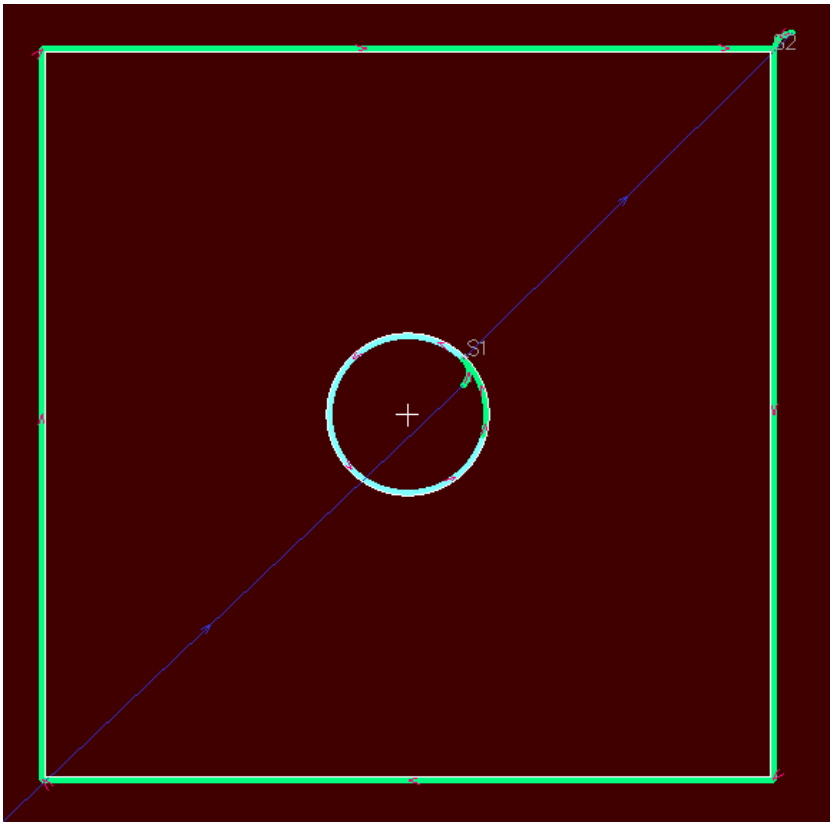
Start position This is the point where SheetCam assumes the cutter is when it starts calculating the paths. Paths nearest the start point will be cut first while complying with the above rules.

Cutting rules:

- All inside first Inside contours are cut first then outside. This is the way SheetCam always used to work. Useful for plasma or for milling/routing when you are cutting all the way through.
- Shortest path This option uses the shortest possible route between contours. This is the one you would use for most milling/routing jobs where you aren't cutting right through.

- Keep parts together Like all inside first, it cuts inside then outside. If your drawing contains more than one part, then each part is cut out completely before moving on to the next. This is useful for plasma, where heat distortion can cause problems if you cut all the insides of all parts then cut all the outsides. By the time you get to the last part heat distortion of the sheet can result in the inside not lining up with the outside.
- Minimise thermal distortion This option tries to avoid cutting outlines that are close to other recently cut outlines. This helps to spread the heat over the sheet and minimize distortion. Increasing 'Preferred distance between cuts' controls the minimum distance between adjacent cuts. Increasing this value spreads the heat more evenly but increases overall cut time because of the extra rapid moves between cuts.

Cut direction This adds a bias to the path optimization. For example, if you set the slider toward horizontal (H), SheetCam will prefer to rapid left/right rather than up/down.



After clicking ***OK*** in the jet cutting operation the toolpath will be created on your drawing. If all looks correct click ***File / Run Post Processor***. Your G code will be written and you are now able to move to the CNC portion.

Halo CNC

Halo CNC Controller is a robust CNC plasma cutting software. Offering its users a full array of cutting options with a simple, easy to use interface.

Halo CNC Controls

Power Powers on/off the CNC controller.

Home X,Y,Z,All selection allowing user to home individual axis or by choosing ***ALL*** the ability to home all axis.

Jog Override:

- **Low** 0-50 ipm manual rapid speed
- **Mid** 50-250 ipm manual rapid speed
- **High** 250-500 ipm manual rapid speed

Rapid speed slider fine tunes the manual rapid speeds.

Note: It is recommended new users use a slower manual rapid speed to avoid collisions, possibly damaging the machine's stops.

Y+ moves the gantry forward on the y axis

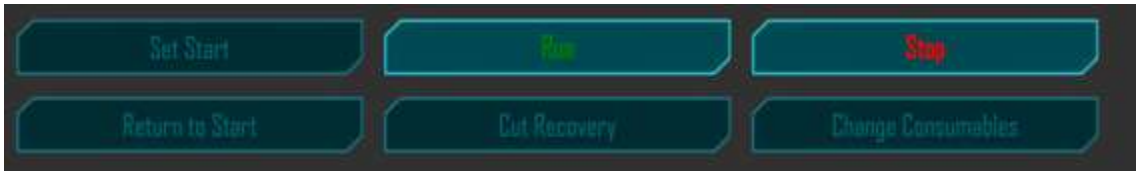
Y- moves the gantry backwards on the y axis

X- moves the z axis left on the x axis

X+ moves the z axis right on the x axis

Z moves the z axis up and down

Note: Machine can be driven using a keyboard by holding ***Shift*** and by pressing the corrilating ***Arrow*** keys. Holding ***Shift*** and pressing ***Page Up / Page Down*** will raise and lower the z axis.



Set Start Zeros the X,Y and sets the start point prior to cutting.

Run Begins the cutting operation.

Stop Ends or pauses the cutting operation.

Return to Start Moves the X,Y to the original zero or set start position.

Cut Recovery Opens a window allowing operator the ability to finish or recover from a broken cut.

Change Consumables Raises the Z axis allowing operator to replace or check cutting torch parts.

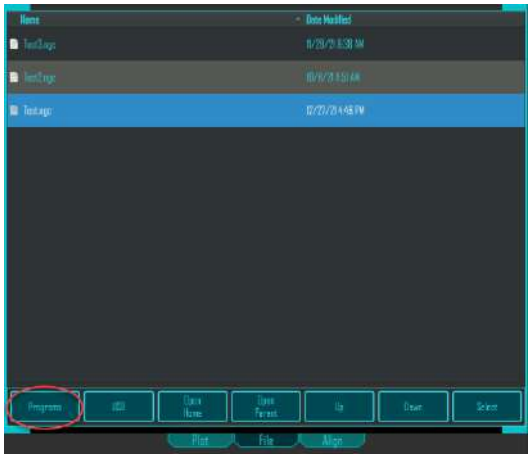
Plot Interface:

Allows operator a 2D or 3D view of loaded file. User can also zoom in and out with the +,-. The X crosshair shows the location of the torch head. X,Y,Z locations are shown in the upper left corner. The broom allows you to clear recent or previous travel in the plot screen.

File Interface:

The file interface is used to load .ngc cut files into Halo CNC.

Programs Easy access to recently used files. Selecting Programs will bring up cut files that have been recently used or modified.



USB Reads the attached USB drive for viewing files stored on an external drive or volume.

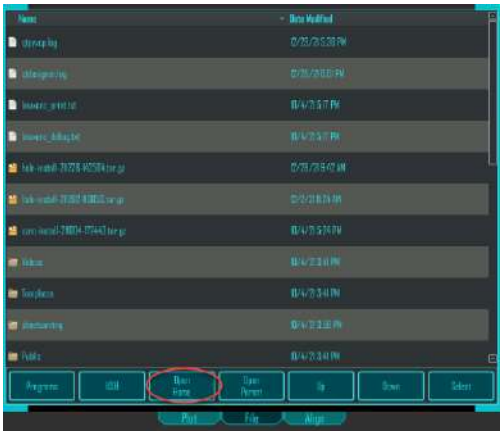
Open Home Opens the home drive of the computer. Allowing the user to navigate through various files on the hard drive.

Open Parent Returns to previous file extension. Used mainly as a back button.

Up Moves selection up.

Down Moves selection down.

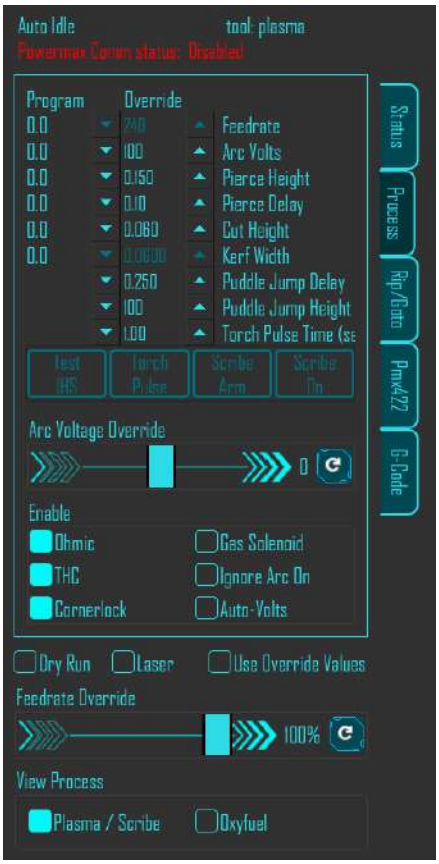
Select Access selected files.



Status: Valuble screen that allows the operator to see in real time any errors or curent system functions as well as arc voltage.

- **Drive** Not used.
- **Joint** Not used
- **Limit +** Used when homing. Stop switch engaged
- **Limit –** Used when homing. Stop switch engaged
- **Cycle Start** Signifies torch on.
- **Arc Transfer** Plasma to material exchange.
- **THC Enabled** Torch height control on.
- **THC Active** Torch height control is activly in use.
- **Corner Locked** Stops THC in corners to prevent torch crashing.
- **Ohmic Contact** Ohmic material sense is in use or stuck.
- **Force Contact** Mecanical switch material sense is in use
- **Torch Collision** Torch has broken away from magnetic holder.
- **Plasma Gas Sol.** Air on through plasma
- **Scribe Arm** Not used
- **Scribe On** Scribe is activated
- **Arc Volts** Arc voltage control uses feedback from the plasma system to measure the voltage between the electrode (negative) and the plate (positive). (some call this “tip volts”, which is incorrect as the torch nozzle is known as the “tip”, yet it is the electrode and plate that the voltage is referenced from) At a given cut speed and fixed torch to work distance this voltage remains constant. Arc Voltage should be close to zero while idling.
- **Reset** Resets axis or joint faults.
- **Bypass** Allows user to bypass estops due to a torch collision.

Note: If a estop is tripped 2, 3 or 4 will be indicated to the top right of the screen.



Process:

Program Override The operator can view the cutting numbers as setup in SheetCam as well as making alterations to the cut settings.

Arc Voltage Override Allows operator to adjust the output voltage +/- 10

Enable: Sets on/off in check box.

Test HIS: Tests the initial height.

Torch Pulse: Manually fires the torch.

Scribe On: Manually turns on scribe.

- **Ohmic** Ohmic resistance touch off.
- **THC** Torch Height Control
- **Cornerlock** Kerf line crossing delay
- **Gas Solenoid** Oxy preheat
- **Ignore Arc On** Ignores torch on
- **Auto Volts** Allows automatic adjustment to voltage accounting for worn consumables.

Dry Run Allows torch table to “ghost” cut without the torch being fired.

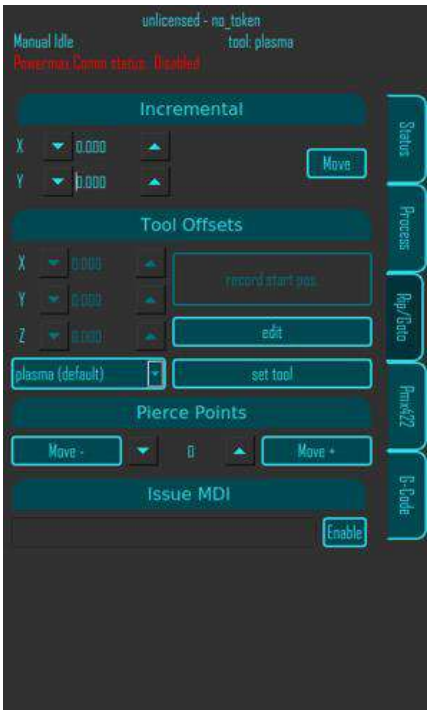
Laser Turns on/off laser line marker

Use Override Values Allows alterations to the set cutting numbers

Feedrate Override Slows torch speed

View Process Selects between plasma/ scribe or oxy torch

Rip/Goto



Incremental Manually moves the torch to a operator set position from the starting point.

Tool Offset Sets the offsets of various tools on the z axis. ***Note: Preset settings from factory, do not change!***

Issue MDI Code line the when entered will move the torch to a set area. ***Example:G53 F200 X0 Y0***

Set Point Allows user to rip cut a section of material. By lining up on the edge of the material input the amount of inches that the torch needs to cut in either the X or Y axis. Click Rip to Point to fire the torch and make the cut.

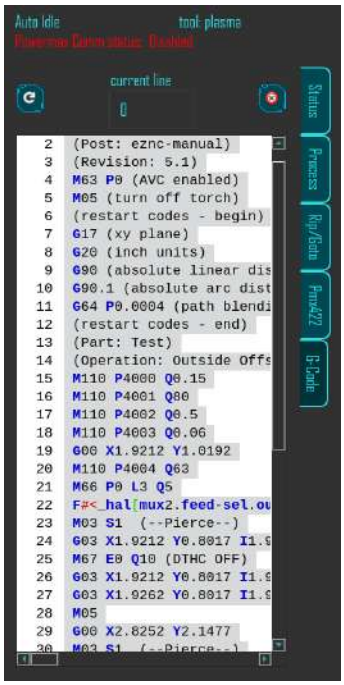
G Code Allows operator the ability to see the g code in real time.Also allows operator to start from a selected line of code. Plasma communication is also visible as well as tool selection.

Reset Restarts g code at line 1.

Clear Program Deletes g code.

Note: Line 14 allows user to check offset, amperage and material size programmed into the selected g code.

Note: Generally you will not need to interact with the g code, but it is there as a tool if needed.



Basic Cutting

Begin by opening Halo CNC on your desktop.

Click on the power button to activate the controller. A grayed, dim power icon means the controller is off. A green colored icon means the controller is now on.

Your machine will need to be homed. Homing the machine sets the parameters of the table thus negating the machine driving past its given coordinates and avoiding damage to the table. Homing must take place every time Halo CNC is opened. To home your machine, begin by driving the machine to the lower left corner, stopping close to 6 inches from the stop switches. The Gantry should be parked at the end closest to the control cabinet. Now move the X axis to the left 6 inches from the stop switch.

With the Y and Z axis 6 inches from the switches click the ***All*** button in the ***Home*** section. This will home the X,Y and Z axis.

Opening Cut Files:

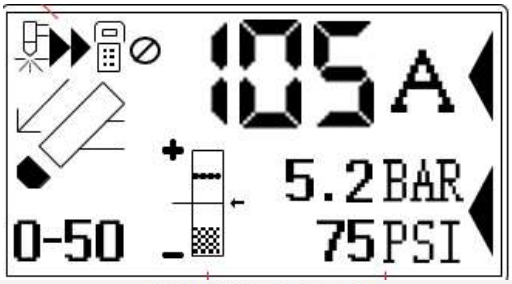
Once Halo is running and homed begin by:

1. Select ***File*** on the interface.
2. Choose the .ngc file that is to be cut.
3. With the file selected click ***Select***.

Consumables and Power Unit Settings:

While in SheetCam the cut settings were made. **To make a proper cut and reduce damage to the consumables you will need to take the following steps**. Ensure the consumables in the torch are the correct type that match the amperage you had set up in SheetCam.

Next move to the power unit. A power restart will be required after removal and replacement of the torch's consumables. Power cycle the knob on the back of the unit to do this. After power is restored look on the front panel, the set amperage will be shown on the front led screen. That amperage will need to match the set numbers in SheetCam.



With your part loaded into the Halo Interface and settings correct you can move to the next step. From here:

1. Check for correct part to be cut.
2. Check for proper settings in ***Process*** and line 14 in ***G-Code***
3. Drive the machine to the corner of the material to be cut.
4. Click on the ***Set Start*** button to zero the torch in the area to be cut.

Making the cut:

Now that the part is loaded into Halo and the torch is zeroed out on the material, the process of making a part can begin. Do a quick check of the setup on the screen. If everything looks correct click ***Run.*** Your machine will rapid over to the first cut, touch off and begin cutting.

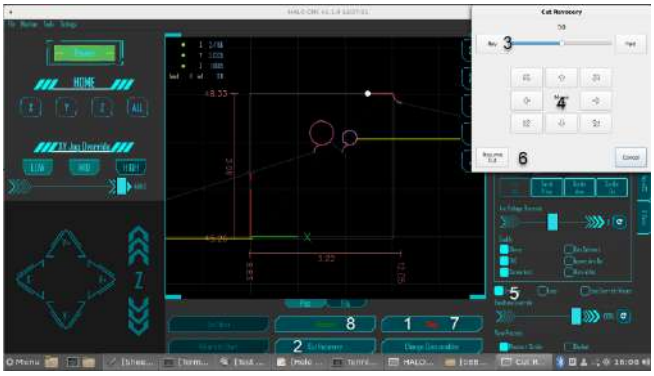
Restarting a cut:

Occasionally a mishap happens that will require the cut to be redone or completed. There are generally two issues, barring user error and improper tool selection, that cause torch to fail cutting. Improper touch off happens when a false zero is sensed. A false zero can happen because the torch forces the material down making the machine zero below the material. This will cause the torch to crash into the material. Another way is the shield or material accumulating slag from pierces. This slag will continue to build while the THC is calling for lower height causing the torch to collide with the material. If the pierce point is close to a cut line the slag from initial pierce will develop. As the torch travels and comes into contact with the slag it will bottom out and crash. Please regularly check your consumables for wear and blockage. Use the following options to do a cut recovery should the need arise.

Option 1

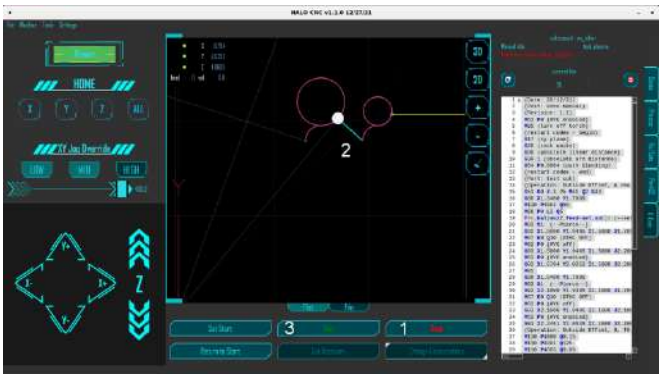
1. Click ***Stop*** twice to reset the cut process.
2. Select the rapid enroute to the lead in of the cut.
3. Select ***Run***

Your machine will travel to the rapid line, then to the proceeding cut. This option works well for missed cuts.



Option 2

1. Click ***Stop*** once
2. Select ***Cut Recovery***.
3. Click on ***Rev*** to reverse the path and pick up where the cut left off.
4. A second lead in/out can be made by clicking the arrows.
5. Click ***Dry Run***.
6. Click ***Restart Cut***.
7. Once the torch meets the end of the previous cut click ***Stop***.
8. Uncheck ***Dry Run*** and click ***Resume***.



This option works best for restarting incomplete cuts.

Router (optional)

The optional router attachment allows cutting and etching across a large spectrum of materials. A .dxf file is necessary to create a toolpath in SheetCam.

Installing the Router

Tools Needed

- 3/16" Allen Wrench
- T20 Plus Torx Bit
- Needle Nose Pliers

Removing The Torch

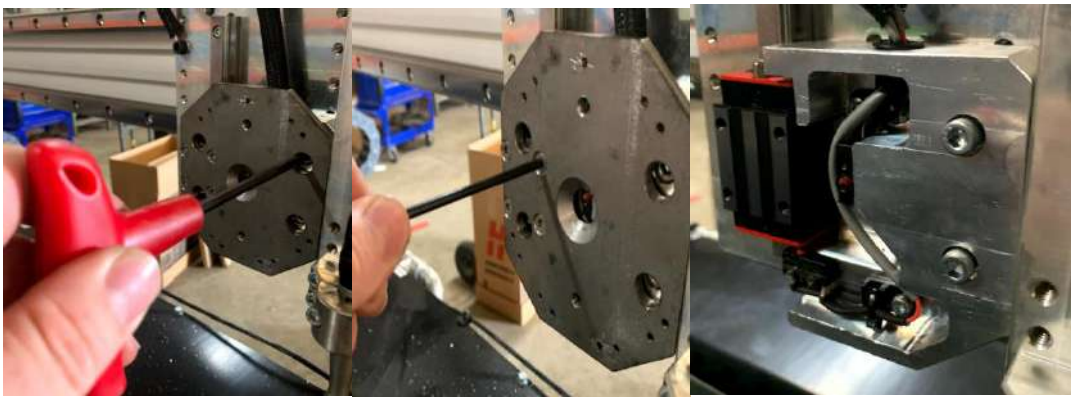
For ease of access drive the gantry to the end of the table. Start by removing the torch at the magnetic break away.

Mount the torch to the mounting bracket located on the side of the z axis.

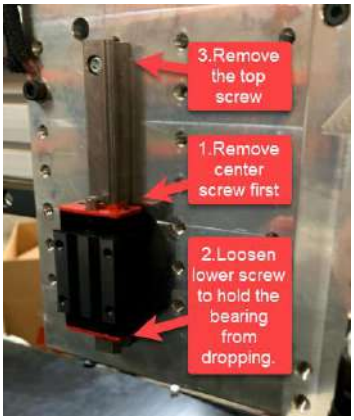
Some tables may require removal of several track clips. Using a needle nose pliers, remove these by unclipping them from the left side on the home side of the table.

Unscrew the quick connect located in the track.

Using a 3/16” allen wrench, remove the two screws in the torch holder and with a T20 plus torx bit remove the four screws on the torch holder. Older tables will have four allen screws to remove only.



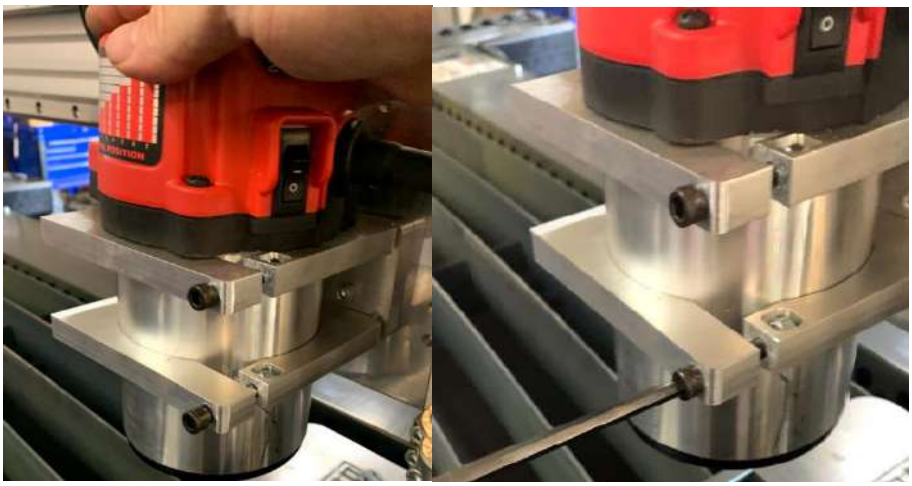
Remove the torch mount plate from the guide and set to the side. Use a T20 plus torques bit to remove the three screws holding the bearing guide with the bearing on the guide. Remove the middle screw and loosen the lower screw to hold the bearing from sliding off.



Installing Router and Mount

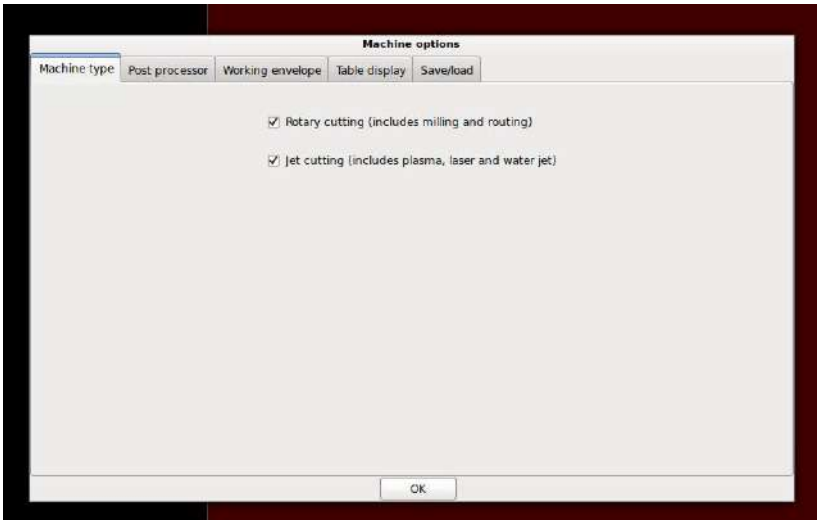
Using a 3/16” allen wrench fasten the router mount onto the z axis plate using the screws provided.

Slide the router into the mount. You may need to pry back by the screw to slide the router down the mount. Using a 3/16” allen wrench tighten the tensioning screws and plug the power cord in. Turn the switch on the router to the on position.

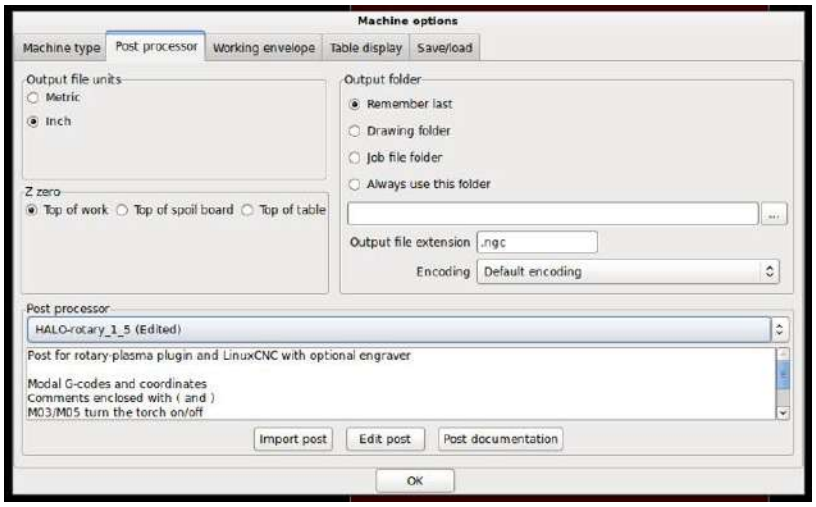


Sheetcam Process

Any .dxf files can be used with SheetCam for the router function. A few settings must be made in SheetCam to accommodate the router function.

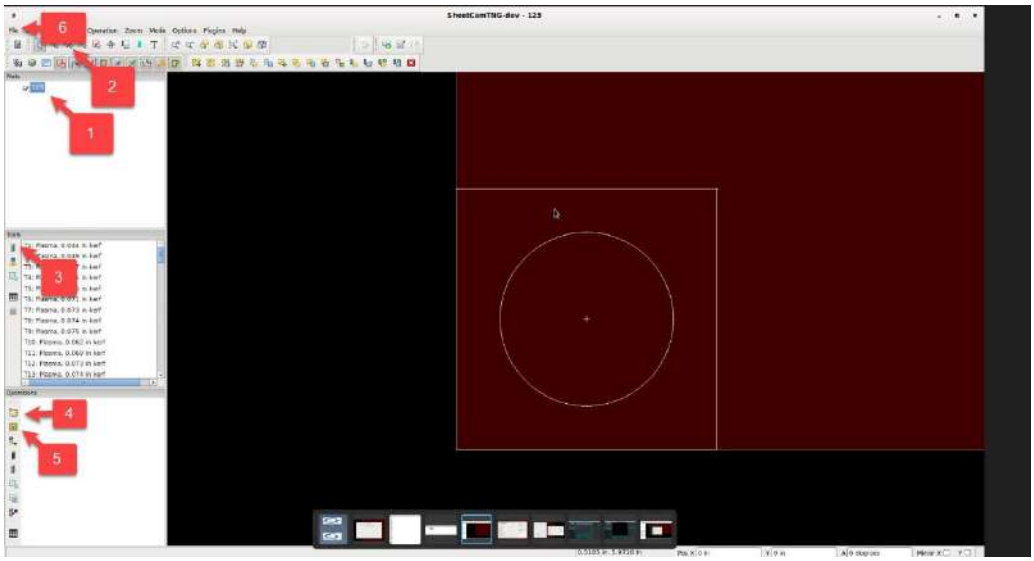


In SheetCam select ***options/machine options*** and check ***Rotary Cutting.***



While in ***Machine Cutting*** select ***Post Processor*** at the top of the menu. On the post processor drop down select the **HALO-rotary_1_5** post processor. To exit select ***OK.***

Once you have made the changes above, select a new part to import into SheetCam.



Some additional features will be available in SheetCam for router use.

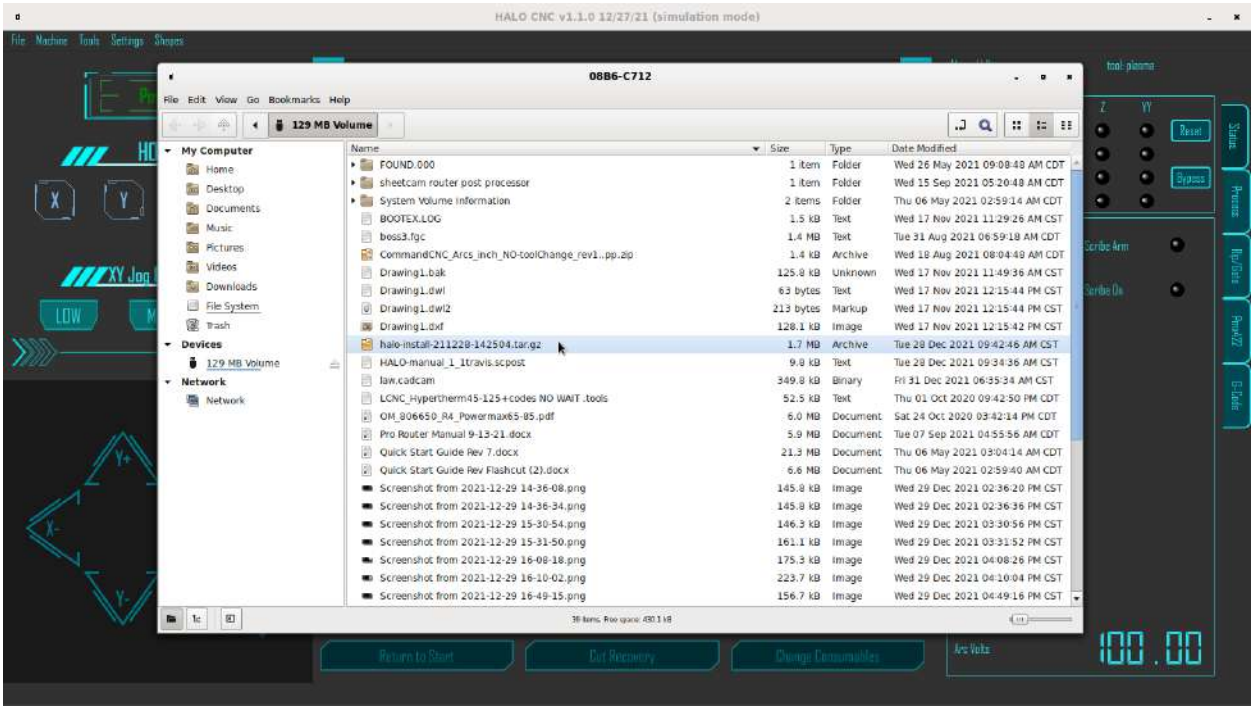
1. New Part adds a new drawing into SheetCam. The process is the same as adding a new part for plasma.
2. Tabs create a bridge between the material being save and the waste material. This is useful to keep parts from moving.
3. Create New Rotary Tool allows addition bits to be added to the cutting chart. Bit info is provided on the packaging.
4. Create a New Contour Operation is used to cut through or flute an item.
5. Create a New Pocket Operation is used to mill an item.
6. Once all operations are complete select ***file/run post processor*** to convert the file.

Loading a file in Halo works the same way as plasma. Select ***file***, and locate the file to be used then ***select***.

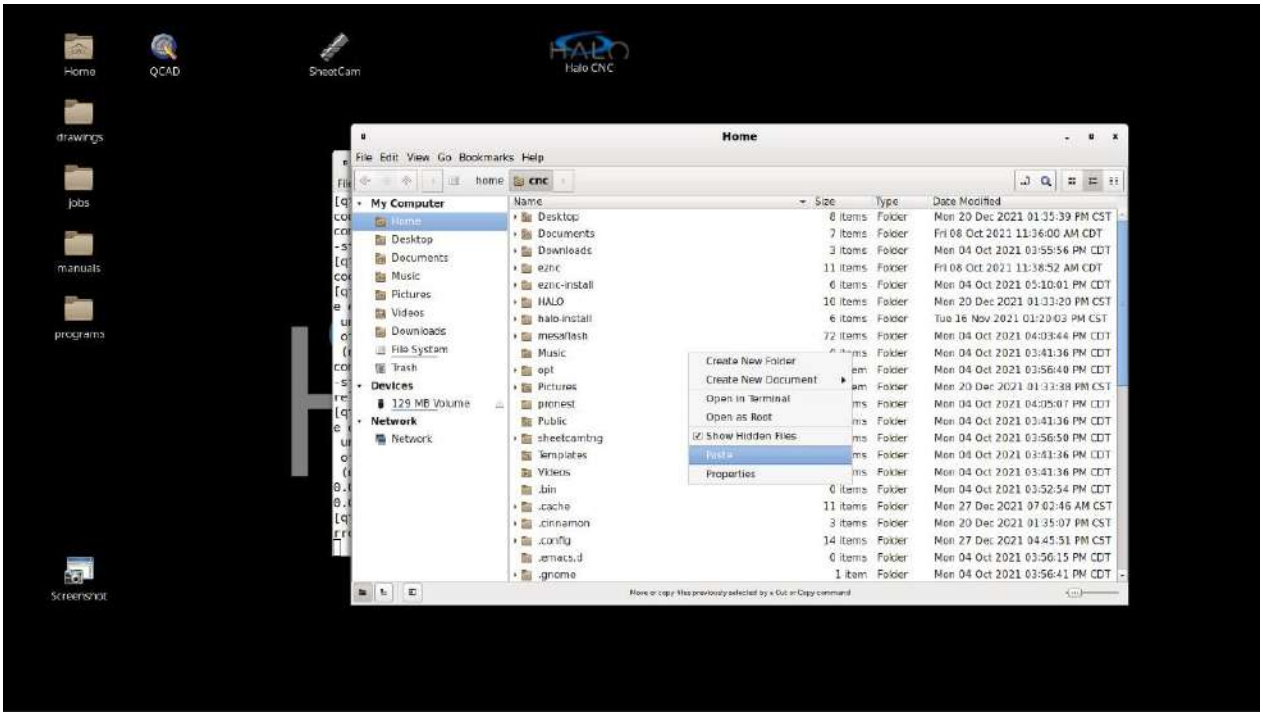
With the file loaded you will be required to zero the z axis and x/y axis independently. To do this lower the z axis down to the intended start height. Usually a paper thickness above the material then select the ***Process tab/SetZ0,*** this will zero the z axis. To zero the x and y select ***Set Start***. Your part is now ready to cut!

Halo Updating

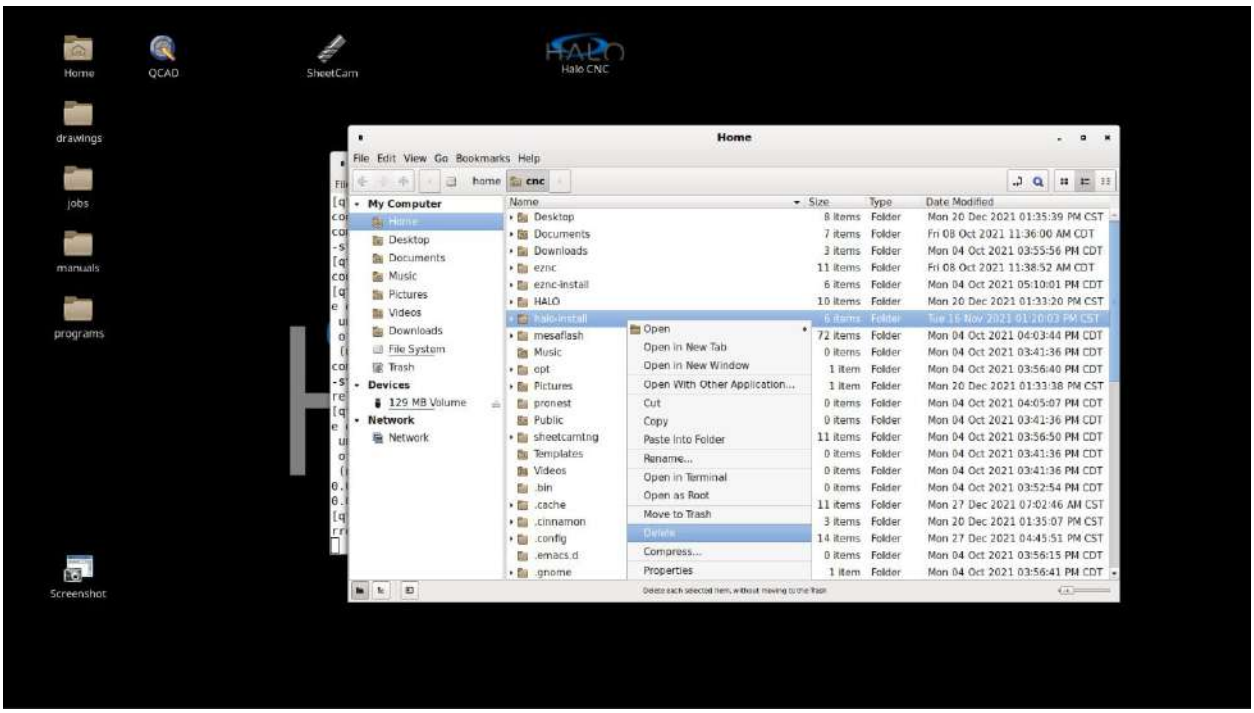
On occasion there will be updates to the Halo CNC software. A representative from Boss Tables will get in contact with you regarding the update process. To update Halo CNC:



Begin by locating the Halo update. You may either save the file to a USB or have it on the desktop.



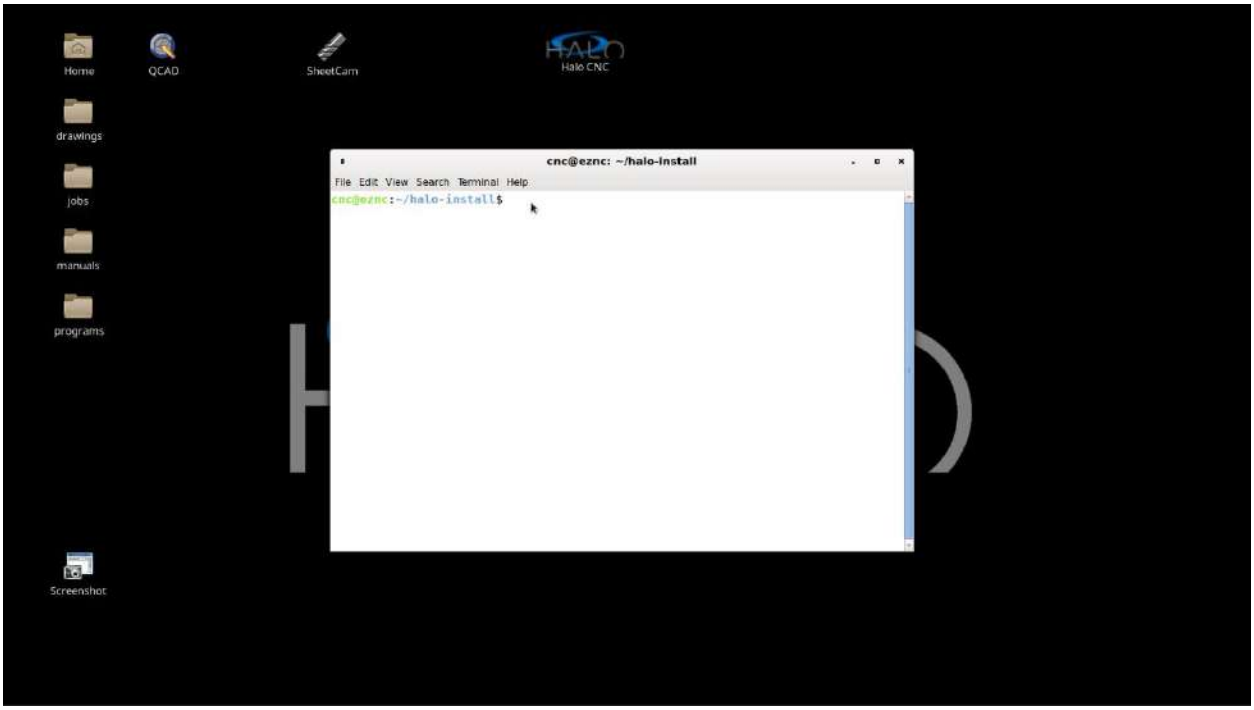
Select the update taking note of the update number, in this case the new update is .142504. Right click the file and **Copy/Paste** it into the **Home/CNC** Directory.



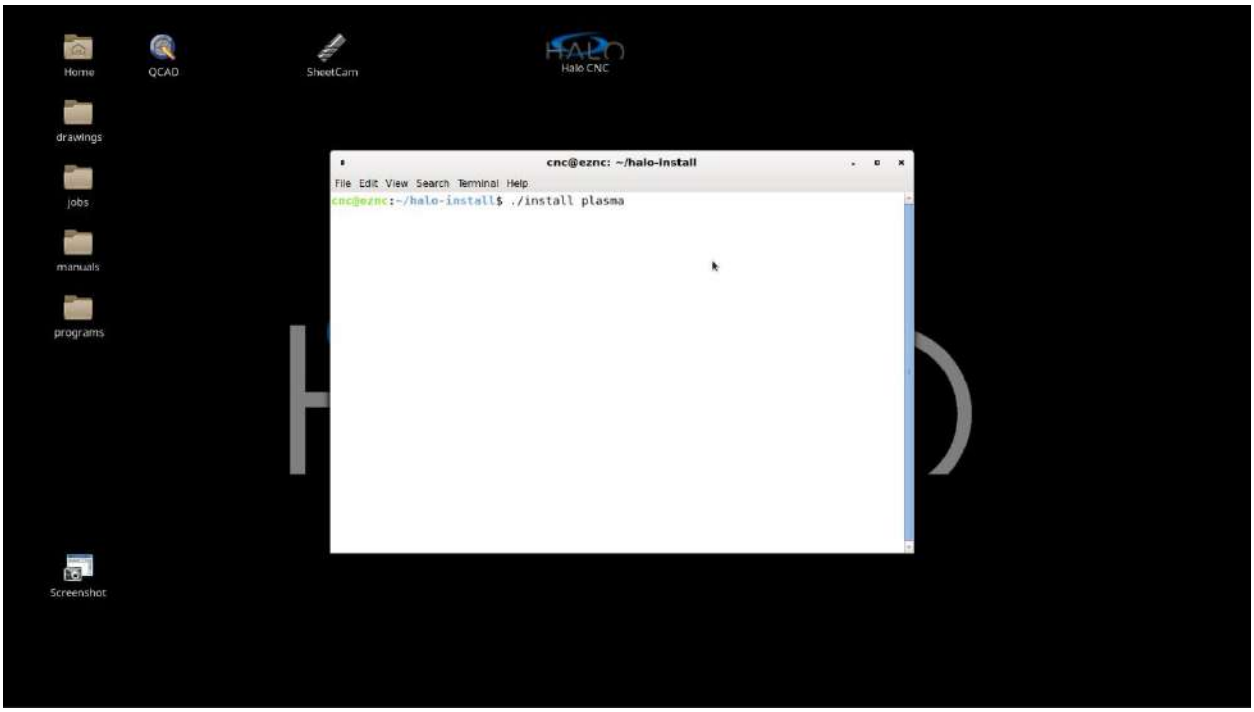
Select the old Halo-install file and right click/delete the file.

Move back to the new file and right click/extract now.

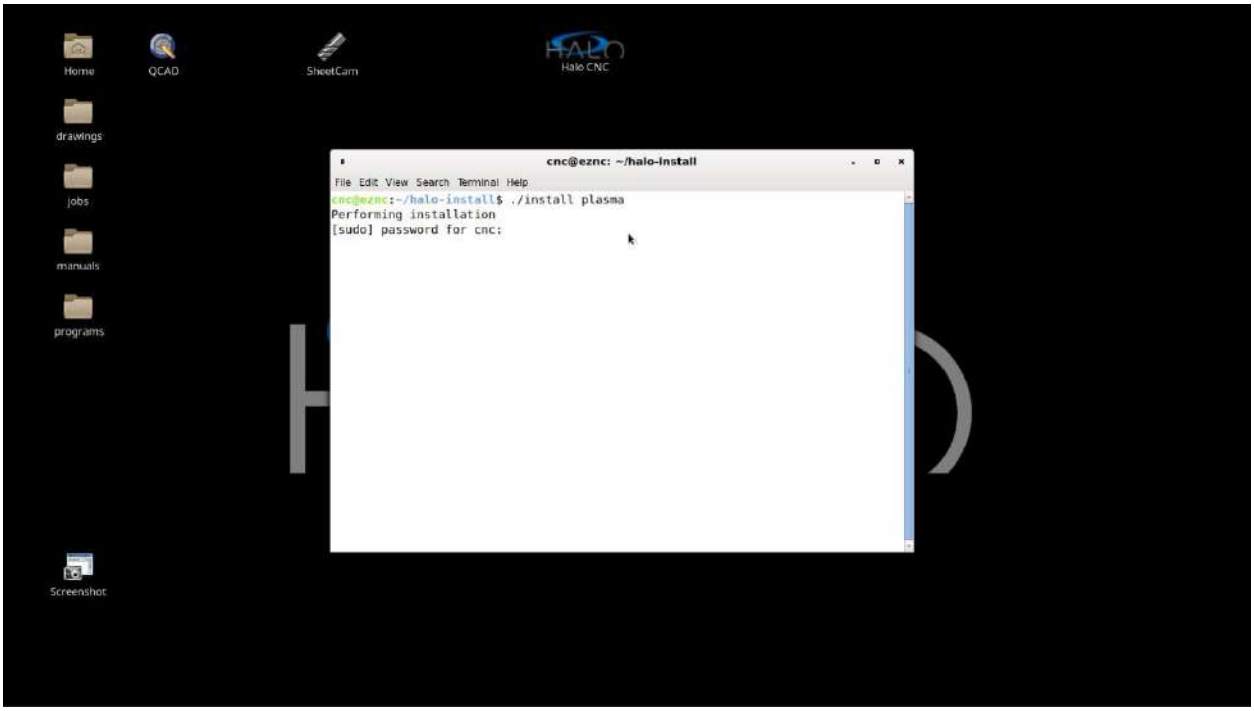
Next, select the **Halo-Install File** right click/open in terminal.



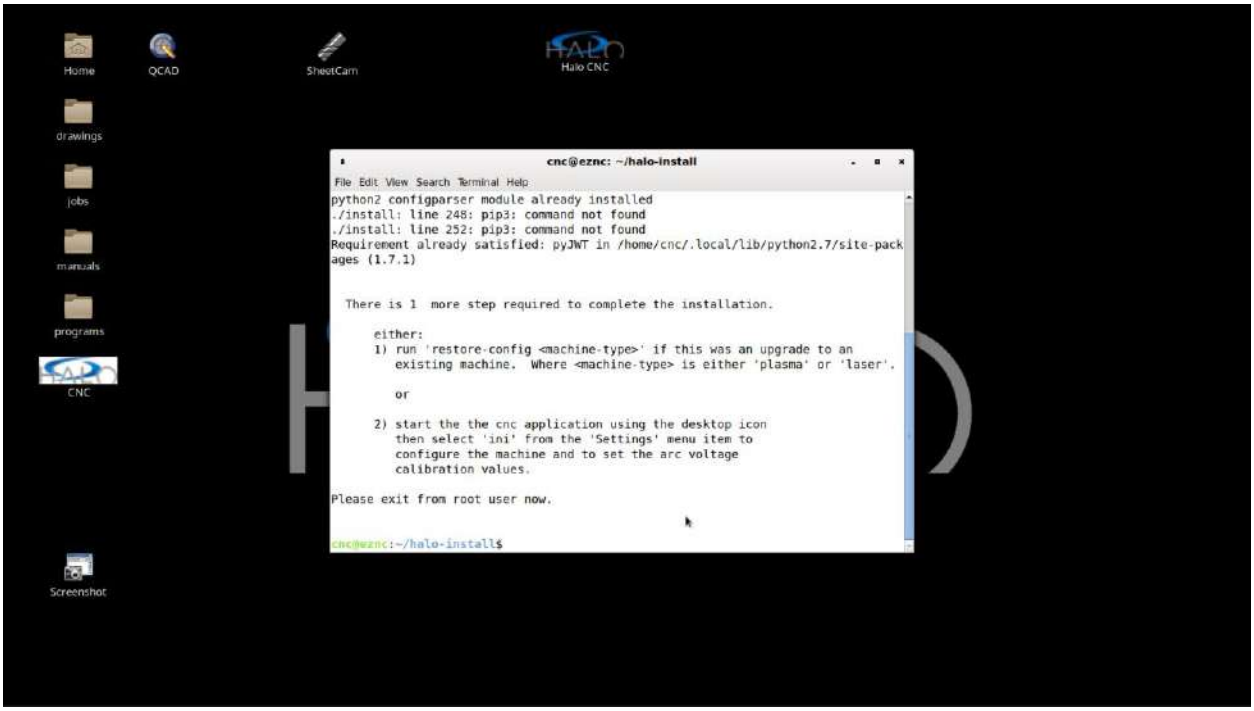
This will open the terminal emulator in Linux.



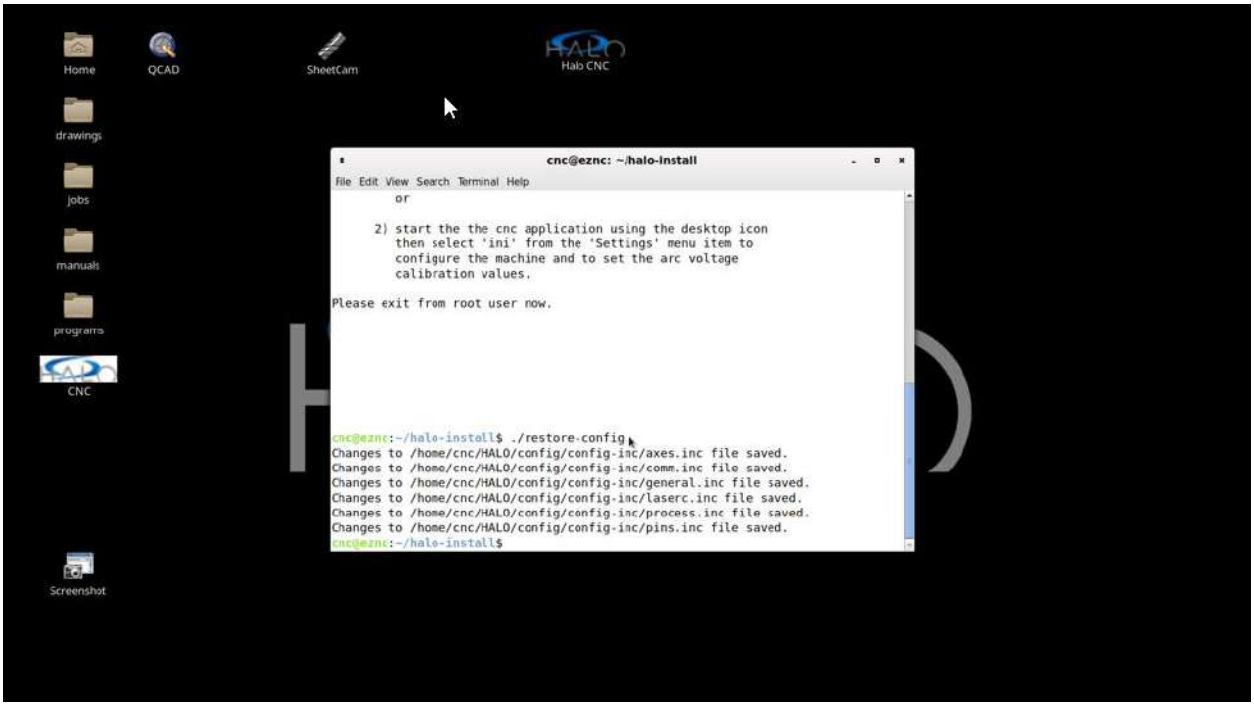
Type exactly ***./install plasma*** in the terminal.



You will be prompted to enter a password. The password is ***cnc.*** It will not be visible but if entered correctly you will proceed to the next step.



The program will install. After finishing the cnc@eznc line will be visible.



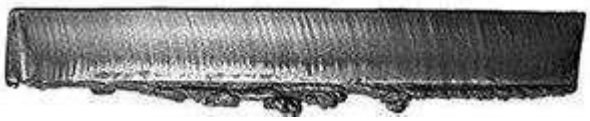
Type ./restore-config to complete the update. Your Halo CNC is now updated!

Troubleshooting & Tricks guide

Troubleshooting cut quality problems – parts have too much dross (slag)

Low speed dross

- Increase the cut speed in 5 ipm increments
- Increase the standoff in 1/16 increments or 5 volt increments
- Decrease the amperage in 10 amp increments
- If none of these measures improve the cut, consider a smaller nozzle size



High speed dross

- Check the nozzle first for signs of wear (gouging, oversize or elliptical orifice)
- Decrease the cutting speed in 5 ipm increments
- Decrease the standoff in 1/16 increments or 5 volts increments
- Increase the amperage (but do not exceed 95% of the nozzle orifice rating)

Top spatter dross

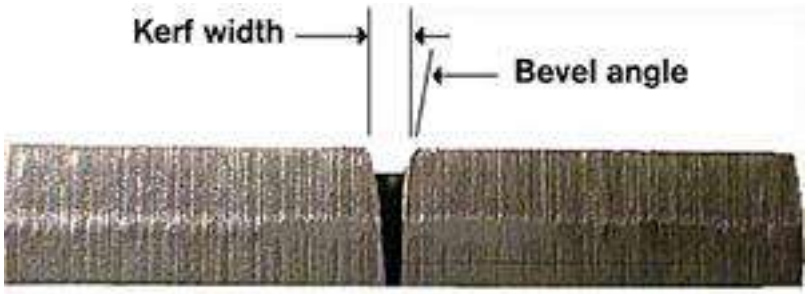
- Check the nozzle for signs of wear
- Decrease the cutting speed in 5 ipm increments
- Decrease the standoff in 1/16 increments or 5 volt increments

To judge the optimum cutting speed:

- Method 1: make a series of test cuts at various cutting speeds and choose the speed that produces the cleanest cut. Lag lines (small ridges in the surface of the cut) are a good indication of cutting speed. Slow cutting speeds produce vertical lag lines that are perpendicular to the plane of the plate. Fast cutting speeds make slanted s-shaped lag lines that run parallel to the plate along the bottom edge. By examining the lag lines the operator can determine whether an increase or decrease in speed is needed to find the dross free window. Many operators have the tendency to slow the machine down at the first appearance of dross, but often an increase in speed is necessary.

- Method 2: watch the arc (through the appropriate welding lens) during the cut and dynamically change the speed to produce the optimum arc characteristics. To do this, observe the angle of the arc as it exits the bottom of the work-piece. If you're cutting with air plasma gas, the arc should be vertical as it exits the bottom side of the cut. With nitrogen or argon/hydrogen, a slight trailing arc is best, and with oxygen plasma gas, the best cut speed is one that gives you a slight leading arc.

Troubleshooting cut quality problems – cut angularity



Kerf too wide (part too small)

This problem can be caused by a worn nozzle, high torch standoff (arc voltage), excessive amperage, inadequate gas flow, or low speed. Each of these variables will cause the arc column to grow, widening the kerf. An incorrect (small) kerf compensation value will also cause an undersized part. Kerf too narrow (part too big). This problem can be caused by low torch standoff (arc voltage), inadequate amperage, excessive gas flow, or high speed. These variables cause the arc column to shrink, narrowing the kerf. An incorrect (large) kerf compensation value will also cause an oversized part.

Bevel angle is the angle of the cut edge

A cut with 0° bevel is a straight cut, perpendicular to the plane of the material. Most plasma torches use a clockwise swirling flow of plasma gas, which produces a straighter cut on the right hand side of the kerf with respect to forward torch motion. Typical bevel angles for conventional plasma torches range from 1-3 degrees on the "good" side of the cut and 3-8 degrees on the "bad" side of the cut. High tolerance plasma cutting systems can achieve even lower bevel angles. Although some bevel is inherent in the plasma

process due to the shape of the gas jet as it exits the torch nozzle, it is possible to minimize it. Bevel angle greater than 5 degrees may indicate a problem with PAC machine parameters.

(Excessive) Positive bevel

Positive bevel - top of part smaller than bottom

This problem may be caused by a worn nozzle, high torch standoff (arc voltage), inadequate amperage, or excessive speed. All of these variables cause the arc to lag which causes more energy to contact the top of the kerf than the bottom. As a result, the kerf is wide at the top and narrow at the bottom. Improper cut direction around the part may also cause excessive positive bevel angle. A part with excessive positive bevel all around it may also have a hard bead of high-speed dross at its bottom edge.

Negative bevel

Negative bevel - bottom of part smaller than top, undercutting

This problem can be caused by low torch standoff (arc voltage), excessive amperage, or low speed. These parameters cause the arc to remove more material at the bottom of the plate. Usually, a consistent negative bevel around the part is accompanied by low speed dross.

Irregular bevel

Positive cut surface - positive and negative bevel on the same piece

This problem usually indicates that the nozzle has failed, the torch is out of square or the electrode and nozzle are misaligned. These variables cause the arc to deviate from a straight path through the material. Often one side of a square part will have a positive bevel and the opposing side a negative. The cross section of the part looks like a parallelogram rather than a rectangle. Sometimes the cut surface may not be flat, but rather concave on one side and convex on the other. These are all signs of severely worn or misaligned parts.

Incomplete cuts (not cutting through the material)

Common causes may include:

1. Worn out/damaged consumables
2. Cutting too fast
3. Incorrect torch height
4. Amperage is too low for the material thickness
5. Incorrect gas/airflow settings

Troubleshooting cut quality problems – hole quality

Bolt holes should be cylindrical

Hole diameter at the top and bottom should be nearly equal – in order to ensure a good fit with the bolt. One critical parameter that affects cylindricity of the hole is cutting speed. Programmers enter cutting speed as a lineal rate in inches per minute (in/min) or millimeters per minute (mm/min), but when cutting a circle, the torch must slow down to compensate for the natural lag of the plasma arc as it cuts. Most CNC controls automatically compensate for this phenomenon with an algorithm that factors down the velocity for hole cutting. Called centripetal limiting, this calculation accounts for the length of the radius, torch acceleration, and minimum corner speed to adjust the actual cutting speed around a circle. The programmer or operator may be able to adjust the lineal speed up or down to optimize actual circular-cutting speed for improved cylindricity. This would mean programming different, lower speeds for bolt holes than for straight cuts on the same part.

Cut height, or voltage setting

Cut height, or voltage setting, is another parameter that affects cut quality on bolt holes. For small holes, cut height should remain constant throughout the cut. With voltage regulated torch height control (THC), cut height is determined by an arc voltage setting of typically 100–180 V. Depending on the responsiveness of the system, using THC for small holes may worsen rather than improve cut quality. It may be necessary to lockout the THC during cutting of small parts to prevent the torch from cutting too high or low and to prevent the torch from diving at the end of the cut. The THC can be locked out by switching into manual mode after the pierce is complete or reprogramming the part to specify corner-slow-down – no THC – during hole cuts. Newer more responsive torch-height controls may help with defects caused by improper cut height.

Programming lead-ins and lead-outs

The type and size of lead-in and lead-out can significantly affect cut quality, particularly with bolt holes and slots. Two common defects are divots and bumps. A divot occurs when the arc removes too much material at the end of the cut. As the plasma arc crosses the lead-in kerf – the removed material from the beginning of the cut – it transfers to the saved part, causing a small indentation or, sometimes, a larger scooped-out region. This makes the hole out-of-round.

A bump occurs if the lead-in and lead-out do not adequately overlap. Some of the material in the hole is not completely removed, leaving a bump of uncut metal that prevents the hole from accepting a bolt.

Finding the appropriate lead-in and lead-out to minimize divots and bumps at start and end points can be challenging. Operators can use a trial-and-error process to find the appropriate combination. Generally, a radiused lead-in with a very small or negative lead-out (negative overburn) to the saved part will produce the best hole. Sometimes a short, straight lead-in works better with a small leadout (positive overburn). The outward-spiral lead-in is a special design that can be very effective for hole cutting. *(Note: This differs from the traditional locking lead-in used in oxyfuel cutting, typically not used for plasma cutting.)* The outward-spiral lead-in allows the machine to reach full speed and the arc to stabilize before cutting the hole perimeter, providing the smoothest machine motion throughout the cut.

Nozzle size and amperage

In general, a small nozzle with lower amperage and slower speed will produce a smaller kerf and a finer cut.

For example, with a 200-A plasma system, the highest power – 200 A, 2 mm (0.086") orifice, 3 mm (0.130") kerf) may not be suitable for cutting small bolt holes and intricate details.

Let us say you want to cut a precise 12 mm (1/2") hole in 12 mm (1/2") mild steel. A 100-A nozzle with a smaller orifice, 1-1/2 mm (0.059"), and kerf width, 2 mm (0.089"), cutting at a slower speed will produce a much finer cut.

To get the best cut from a given nozzle, always set amperage at 95 to 100% of the nozzle's rating. The downside: reduced consumable life and slower cutting speeds. The upside: a nearly finished part with minimal rework.

Six rules for cutting bolt holes

- Use the smallest nozzle size rated to pierce and cut the material
- Make sure the pierce-delay allows full arc penetration before machine motion starts
- Lock out voltage-regulated THC
- Use a radiused or spiraled lead-in
- Program a slower cutting speed
- Use a short or negative leadout to the saved part

Air Issues

Air Pressure Is Too High

If the pressure is too high this will dissolve the arc column and weaken the power of the plasma arc.

Things to check:

- Air compressor pressure
- The pressure between the Air filter and air compressor
- If the air filter relief valve is faulty or set too high

Air Pressure Is Too Low

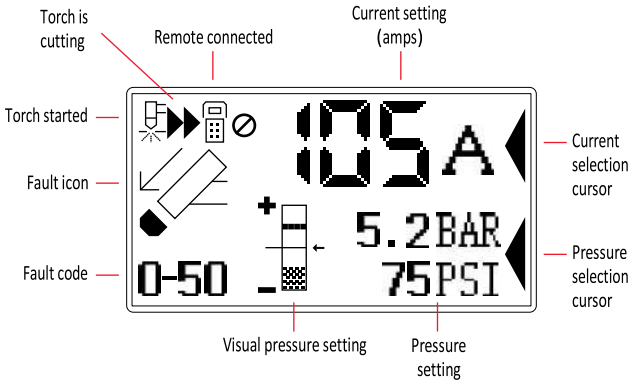
If this occurs, then an optimal plasma arc cannot be formed. This will result in a poor-quality cut and built-up slag.

Things to check:

- Is the compressor providing enough air?
- Are the air channels blocked?
- Is the air filter relief valve set correctly?

If you have a 4x8 or larger table, you need to be at a minimum 5HP 80 gallon tank or larger. Ideally your compressor should be able to output 1.5 to 2 times the CFM needs of your plasma cutter. Which is about 14 CFM at 90 psi.

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



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

Refrigerated compressed air dryer

Refrigerated Compressed Air Dryers use Freon to lower the temperature of the compressed air. This temperature reduction condenses the water vapor within the compressed air into droplets, allowing it to be eliminated through an onboard condensate drain.

Regenerative Desiccant Air Dryers

Minimum Pressure: 100 psi **Maximum Pressure: 175 psi**
Water, oil, and particulates must be removed prior to dryer in order to maximize dryer effectiveness and life of desiccant.

THIS SIDE UP

Stronger Construction

Aluminum billet and hard coat anodizing provides superior strength and corrosion resistance; eliminating casting porosity.

115v Control Time

Timer cycles the towers into regeneration mode every 2 minutes.

Customizable Performance

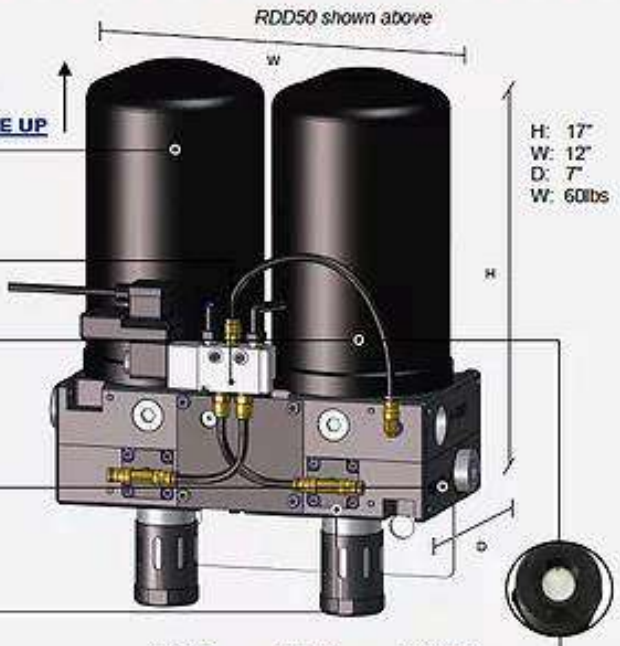
Cartridge style regeneration orifices provide the ability to control the amount of air used to dry the towers. Reducing air volume and increasing orifice size can provide dew points down to –40°F. Flow rates up to 100 CFM @ 175 psi. See orifice chart below.

Dual 1" Inlet / Outlet Ports

Dual inlet ports and outlet ports provide easier installation and allow air to come in and out of same side or in one side and out the other. This unique design allows for greater mounting flexibility.

Less Parts, More Reliable

Single piston spool per tower reduces the number of moving components; allowing for easier serviceability.



Regenerative Desiccant Air Dryers

Part # RDD50 2 Tower Dryer 0-50 CFM
Part # RDD100 4 Tower Dryer 51-100 CFM

Optional: 3/4" Dryer Mounting w/ 3-way Ball Valve
Part # DM75RD

RDD50 CFM Capacity	Part # Orifice Size	RDD100 CFM Capacity
0 - 10	RD .015	NA
11 - 25	RD .030	NA
26 - 40	RD .045	51 - 80
41 - 50	RD .060	81 - 100

Water Table Issues

Your Boss CNC Plasma Table is equipped with a water reservoir. Should your table bubble while filling simply add more water to the reservoir. Never fill the reservoir and bed to full compacity as air in the tank is needed to fill. An air connection is required to fill your table, as the air is inserted into the tank it pushes water up and out to the cutting bed. Closing the valves will stop the bed from filling and close off the tank. At this point the air line can be removed until further filling is required.

Ohmic Sensor and Wire

The wire coming down from the z axis to the torch head is for the ohmic sensor. There are two CammandCNC programs to choose from. The first is Plasma Feather Touch for new steel and the second is Plasma No Feather Touch for rusty or painted steel. The ohmic sensor works in conjunction with Plasma Feather Touch and uses resistance to find the material. No Feather Touch uses a switch located inside the torch mount to locate the material. Home the Z, on CommandCNC or Sense Powermax Zero, on Flashcut to set the Home Z. On CommandCNC zero the z after homing. Home the z on a sturdy area of the material to give a proper material location or the torch may break away during touchoff.

Oxy Torch Info

ACETYLENE

HAND/MACHINE CUTTING

HARRIS
A LINCOLN ELECTRIC COMPANY

6290 Single Piece Cutting Tips
Medium preheat for clean surfaces

6290-S Single Piece Cutting Tips
6290-AC Two Piece Cutting Tips
Heavy preheat for rusty or scaled surfaces

6290 & 6290-S TIPS FOR OXY-ACETYLENE

PLATE THICKNESS INCHES	6290 TIP SIZE	6290-S TIP SIZE	OXYGEN PRESSURE PSIG	ACETYLENE PRESSURE PSIG	CUTTING ORIFICE DRILL SIZES
Light gauge to 3/16	000	-	15-20	5-15	#68
3/16-3/8	00	-	20-25	5-15	#64
3/8-5/8	0	-	35-40	5-15	#60
5/8-1	1	1S	35-40	5-15	#56
1-2	2	2S	40-45	5-15	#52
2-3	3	3S	45-50	5-15	#48
3-6	4*	4S**	50-75	10-15	#42
6-8	-	5S**	65-80	10-15	#35
8-12	-	6S**	70-90	10-15	#30

6290-AC TIPS FOR OXY-ACETYLENE

PLATE THICKNESS INCHES	6290-AC TIP SIZE	OXYGEN PRESSURE PSIG	ACETYLENE PRESSURE PSIG	CUTTING ORIFICE DRILL SIZES
3/16-3/8	00AC	15-30	5-15	#64
3/8-5/8	0AC	20-35	5-15	#60
5/8-1	1AC	30-50	5-15	#56
1-2	2AC	40-65	5-15	#53
2-4	3AC	40-65	5-15	#52
4-7	4AC**	50-80	5-15	#42
7-10	5AC**	65-80	5-15	#35
10-12	6AC**	70-95	5-15	#31

** to provide required gas flow, use 3/8" I.D. hose for size 4 and larger.
Cleaning: Use Harris tip cleaner C-9 (P/N 9000156) for single piece tips.
E-9 (P/N 9000160) for two piece tips.

Additional copies are available at www.harrisproductsgroup.com

PROPYLENE/MAPP®

HARRIS
A LINCOLN ELECTRIC COMPANY

HAND CUTTING

6290-NXP Cutting Tips
6290-NXM Cutting Tips
Medium preheat for clean surfaces

6290-NXP TIPS FOR PROPYLENE

PLATE THICKNESS INCHES	6290-NXP TIP SIZE	OXYGEN PRESSURE PSIG	FUEL GAS LOW PRESSURE	FUEL GAS EQUAL PRESSURE	CUTTING ORIFICE DRILL SIZES
Light gauge to 3/16	000NXP	15-30	4 oz. to 2 PSIG	5-15 PSIG	#68
3/16-3/8	00NXP	20-30	4 oz. to 2 PSIG	5-15 PSIG	#64
3/8-5/8	0NXP	30-40	4 oz. to 2 PSIG	5-15 PSIG	#60
5/8-1	1NXP	35-50	4 oz. to 2 PSIG	5-15 PSIG	#56
1-2	2NXP	40-55	4 oz. to 2 PSIG	5-15 PSIG	#52
2-3	3NXP	45-60	4 oz. to 2 PSIG	5-15 PSIG	#48
3-6	4NXP	50-75	4 oz. to 2 PSIG	5-15 PSIG	#42
6-8	5NXP	65-80	4 oz. to 2 PSIG	5-15 PSIG	#35
8-12	6NXP	70-90	4 oz. to 2 PSIG	5-15 PSIG	#30

6290-NXM TIPS FOR MAPP® GAS

PLATE THICKNESS INCHES	6290-NXM TIP SIZE	OXYGEN PRESSURE PSIG	FUEL GAS LOW PRESSURE	FUEL GAS EQUAL PRESSURE	CUTTING ORIFICE DRILL SIZES
Light gauge to 3/16	000NXM	15-30	4 oz. to 2 PSIG	5-15 PSIG	#68
3/16-3/8	00NXM	20-30	4 oz. to 2 PSIG	5-15 PSIG	#64
3/8-5/8	0NXM	30-40	4 oz. to 2 PSIG	5-15 PSIG	#60
5/8-1	1NXM	35-50	4 oz. to 2 PSIG	5-15 PSIG	#56
1-2	2NXM	40-55	4 oz. to 2 PSIG	5-15 PSIG	#52
2-3	3NXM	45-60	4 oz. to 2 PSIG	5-15 PSIG	#48
3-6	4NXM	50-75	4 oz. to 2 PSIG	5-15 PSIG	#42
6-8	5NXM	65-80	4 oz. to 2 PSIG	5-15 PSIG	#35
8-12	6 NXM	70-90	4 oz. to 2 PSIG	5-15 PSIG	#30

** to provide required gas flow, use 3/8" I.D. hose for size 4 and larger.
Cleaning: Use Harris tip cleaner E-9 (P/N 9000160) for two piece tips.

Additional copies are available at www.harrisproductsgroup.com

PROPYLENE/MAPP®

HARRIS
A LINCOLN ELECTRIC COMPANY

MACHINE CUTTING

Series 6290-VVCP

Series 6290-VVCM

6290-VVCP TIPS FOR PROPYLENE

6290-VVCM TIPS FOR MAPP®

PLATE THICKNESS INCHES	6290 TIP SIZE	CUTTING SPEED IN/MIN.	CUTTING OXYGEN PSIG	PREHEAT OXYGEN PRESSURE HIGH/LOW	FUEL GAS PRESSURE PSIG	WIDTH KERF INCHES	CUTTING ORIFICE DRILL SIZE
1/16-3/16	5/0 VVCP & VVCM	20-24	40	12/8	4 oz. to 2 PSI	.05	#75
1/8-1/4	4/0 VVCP & VVCM	20-22	50	12/8	4 oz. to 2 PSI	.06	#68
1/4-3/8	3/0 VVCP & VVCM	18-22	75	25/8	4 oz. to 2 PSI	.07	#64
3/8-1/2	2/0 VVCP & VVCM	18-20	75	25/8	4 oz. to 2 PSI	.07	#62
1/2-3/4	0 VVCP & VVCM	15-18	90	25/8	4 oz. to 2 PSI	.08	#60
3/4-1 1/4	01/2 VVCP & VVCM	14-16	100	25/8	4 oz. to 2 PSI	.08	#58
1 1/4-2	1 VVCP & VVCM	13-15	100	25/10	4 oz. to 2 PSI	.09	#56
2-3	1 1/2 VVCP & VVCM	9-12	100	25/10	4 oz. to 2 PSI	.11	#54
3-4	2 VVCP & VVCM	7-9	100	25/10	4 oz. to 2 PSI	.12	#53
4-5	2 1/2 VVCP & VVCM	6-8	100	30/10	4 oz. to 2 PSI	.13	#51
5-6	3 VVCP & VVCM	5-7	100	30/10	4 oz. to 2 PSI	.14	#49
6-8	4 VVCP & VVCM	5-7	100	30/10	4 oz. to 2 PSI	.16	#45
8-9	5 VVCP & VVCM	4-6	90	30/10	4 oz. to 2 PSI	.20	#41

NOTE:

- Correct cutting oxygen pressure must be available at torch entry.
- Oxygen preheat pressures are for three hose torches.
- For two hose torches set same gas pressures for both high and low preheat.

Cleaning: Use Harris tip cleaner E-9 (P/N 9000160) for cleaning pre-heat holes and removing spatter from the tip face. When cleaning the preheat slots, do not brush across the slots as this motion can damage the slots. Always brush along the length of the slot to remove dirt or spatter.

For additional gas flow information refer to our website at www.harrisproductsgroup.com and our equipment catalog.

Additional copies are available at www.harrisproductsgroup.com

Notes/ Changes Made

Notes/ Changes Made

Notes/ Changes Made