



PRO TABLE 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.

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.





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.





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.



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 to toxic fumes that may be generated while plasma cutting. Certain chlorinated solvents such as perchloroethylene and trichloroethylene 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.



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



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

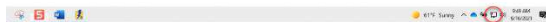
NFPA

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





Wireless internet connection



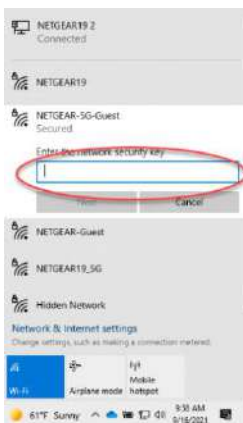
Select the wireless icon located in the lower right corner of the desktop.

Select the wireless network you wish to connect with.

Select **Connect**.



Enter the security key if the wireless network is encrypted (with WEP, WPA or WPA2). This will be stored for next time, so you'll only have to enter it once.



When it says that you're connected, open a web browser and visit a website to confirm that you can access the internet.

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



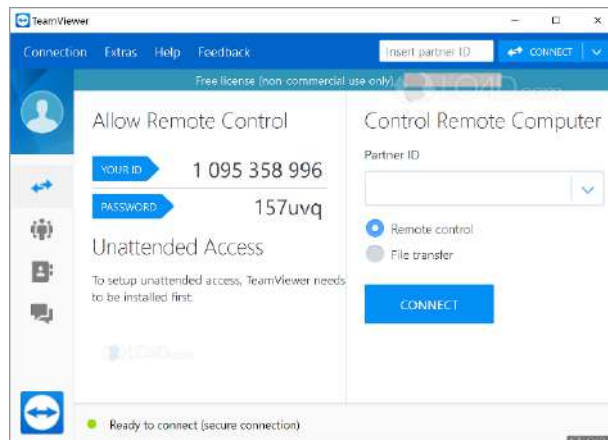


How to Fix Wi-Fi Connection Problems

If you have trouble connecting to a Wi-Fi network, there are several things you can check depending on your specific type of issue:

- If you can't find any wireless networks, make sure you have Wi-Fi enabled.
- If your wireless signal keeps dropping, you may need to get closer to the access point.
- If you have a wireless connection but no internet access, then the modem or router may need to be rebooted.
- If you have forgotten the password to your home network, your wireless security key may be found on the bottom of your router if you didn't change the defaults when setting up your network.

Teamviewer



Boss Tables uses Teamviewer to help with education and troubleshooting. Your Boss CNC Table will come preloaded with Teamviewer on its desktop. In order to make a start with Teamviewer's remote control functions, navigate to the Remote Control tab of the main interface. Here, you will find your Teamviewer ID and your temporary password, which you can change at any point. With this information, you can allow a partner remote control of your computer.

In order to do this in reverse and control another computer remotely, you simply enter the partner computer ID and choose between various connection modes such as remote control, file transfer or VPN. Additionally, as soon as one or more remote connections have been established, each session will be displayed in the title bar of the Remote Control window. For more information on how to establish a remote control connection, see the Teamviewer [manual for remote control](#) located at www.teamviewer.com





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.

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





Once everything is powered on open Flashcut on your desktop.



Your machine will open in CAD. To jog your machine and home it, left click on CNC at the top of the screen.



You can now jog your table around. **HOLDING Ctrl** using the **arrow** keys for left, right, up, and down. Use the **PgUp** and **PgDn** above the arrow keys to travel the Z axis up and down. You can also use the jog function on the lower right in Flashcut CNC.



Powering On


Begin by homing the machine. To home the machine, jog the table to the lower left side of the table. Do not ram the gantry into the stops. Stop 4 inches before the X and Y limit switch, select Home All and it will do both at the same time.

Note: Always home the machine after opening a new process of Flashcut.

Do not leave the gantry against the stops while parked. With the machine homed it is now able to run code and know its parameters.



Evaluation Mode


FlashCut CNC will run in evaluation mode until you click the **Connect** icon  in CNC or activate a license in a second copy. In evaluation mode, you can try out many features of the program. Some features will be disabled, while others will be limited. For example, you cannot communicate with the CNC controller, you will not be able to save files, and only 25 lines of G-Code will be generated when you send a CAM toolpath to the CNC workspace.

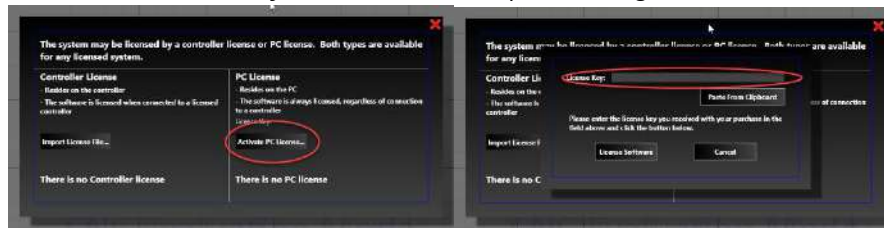
Activating a Second License

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





To enable the full functionality of the program, you must activate a PC license. Select the License button  in any workspace. Click on the **Activate PC License** icon and add the user key found on your controller. Click the **License Software** button to complete the registration.



Getting Familiar with CAD Menu & Ribbon

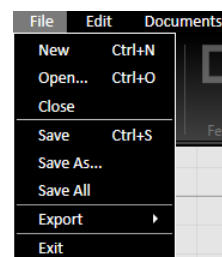
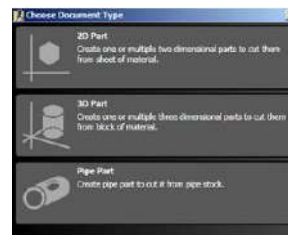
www.bosstables.com 563-380-1535 www.info@bosstables.com





File Menu

- **New** (Ctrl+N) Creates a new CAD/CAM drawing in a new FlashCut CAD window. When selected from the menu or entered from the keyboard, the New file command displays a dialog window that requires the user to select the type of file: 2D Part, 3D Part, or Pipe Part. The command does not close the current drawing. Note that you can view any open drawing by selecting it from the Documents menu.
- **Open** (Ctrl+O) Opens an existing CAD/CAM file for editing in a new FlashCut CAD window. It does not close the current drawing.
- **Close** Closes the current CAD/CAM drawing from the CAD editor. If the drawing is unsaved, the software prompts you to save the file before closing.
- **Save** (Ctrl+S) Saves the current CAD/CAM drawing using the existing file name and location. It will not save any other CAD/CAM drawing that is open. FlashCut will prompt you to create a file name and destination. FlashCut saves drawings to the CAD/CAM (file extension .cadcam) format. The Save command is unavailable when no recent changes have been made.
- **Save As** Saves the current drawing to a new file name or destination. It will not save any other CAD/CAM drawing that is open. FlashCut prompts you to create a file name and destination. Saved files use the extension .cadcam.
- **Save All** Saves all open drawings, including those in other windows. This command is unavailable when no recent changes have been made.
- **Export** Exports the file as a DXF/DWG file. There are two alternatives: Export DXF with Options or Export DXF. FlashCut CAD/CAM and CNC Control Software Page 27 Both commands bring up a Windows dialog box that allows you to select the location, filename, and specific file format.
- **Export DXF** Brings up the Windows save dialog. No parameters can be altered. The available file formats are DXF 2000 or DXF R12.
- **Export DXF with Options** Brings up the Windows dialog box, and provides a configuration panel in the Parameters area. After setting the parameters for the file, you may either Accept or Cancel the changes.
- **Export Curves as Polylines** When selected, curves will be saved as polylines in the DXF file. When not selected, arcs and circles will be maintained, but ellipses and splines will be saved as polylines. Polylines are drawing objects composed of multiple separate line segments.
- **Export Text as Polylines** When selected, text shapes will be saved as polylines in the DXF file. Polylines are drawing objects composed of multiple separate line segments. When not selected, text will be saved as a font.
- **Export Units** Choose from either inches or millimeters.
- **Exit** Closes the entire FlashCut application. If any drawings are unsaved, FlashCut prompts you to save these files or discard changes before closing.



Edit Menu





The Edit menu has the following commands: •Undo (Ctrl+Z) • Redo (Ctrl+Y) • Settings... • Cut (Ctrl+X) • Copy (Ctrl+C) • Paste (Ctrl+V) • Select All (Ctrl+A)



Undo (Ctrl+Z) Reverses the previous drawing action. Up to 20 actions can be reversed.

Redo (Ctrl+Y) Repeats the previous drawing action, or reverses the Undo stack.

Settings Displays controls for setting the Grid and Units for the drawing window. After changes are complete, you may either Accept or Cancel the changes.

-Grid

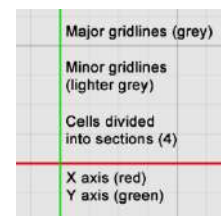
Visible This option will hide/display the gridlines in the workspace.

Dynamic Grid Toggles dynamic gridlines within the drawing window. When checked, these gridlines remain the same size on the screen despite zooming and panning motions. When unchecked, the dimension that these gridlines represent remain the same while zooming and panning.



Grid Major Spacing Changes the size of the major gridlines when Dynamic Grid is not enabled. These lines will change with zooming and panning. The number specified determines the numerical spacing between gridlines.

Sections Per Cell Determines the number of minor horizontal and vertical gridlines in between major gridlines. Range: 1-100.



-Units

Document Units Select the dimensional units of the CAD drawing. You may choose the following units: mm – millimeters, in – inches. FlashCut gives you the option to convert any existing parts (drawing objects) in the workspace.

Selecting Yes rescales the existing values into new units (i.e., a 1 inch circle is a 25.4mm circle). Selecting No reinterprets the values into new units (i.e., a 1 inch circle becomes a 1mm circle). *Note that changing the units for a drawing will clear all CAM data.*



Cut (Ctrl+X) Removes selected features and places them on the clipboard to be pasted. Note that objects are selected by clicking on them with the selection arrow (which becomes available by pressing the Esc key). To select all segments of a feature (chain select), hold down the Alt key. You can include other objects by holding down the Ctrl key while you select the objects that you want. You may also select multiple objects by creating a selection box with the selection arrow. Creating a selection box that goes from left to right will select all objects that it touches and a selection box that goes from right to left will select all objects that it completely envelopes.

Copy (Ctrl+C) Copies the selected features and places them on the clipboard to be pasted.





Paste (Ctrl+V) Pastes copied or cut features from the clipboard. The features will be centered at the cursor point and you will be able to maneuver them to a desired location. Clicking the mouse will paste the features permanently into the drawing.

Select All (Ctrl+A) Use the Select All command to select all geometry in the workspace.

Documents Menu

The Documents menu allows you to toggle between all open drawings in both FlashCut CAD and FlashCut CAM. The checked document is visible and available for editing. Note You may copy features from one drawing and paste them into another drawing.



CAD Ribbon



The ribbon features an assortment of command icons to create, modify, and transform elements such as points, curves and shapes in the drawing window. To use a tool, select it with the cursor. Pressing the Esc key enables you to exit out of any particular tool.

New (Ctrl+N)



Creates a new editable document in CAD.

Open (Ctrl+O)



Opens an existing CAD/CAM drawing for editing in a new FlashCut CAD window. It does not close the current drawing.

Save (Ctrl+S)



Saves the current CAD/CAM drawing using the existing file name and location. It will not save any other CAD/CAM drawing that is open. FlashCut will prompt you to create a file name and destination. FlashCut saves drawings to the CAD/CAM (*.cadcam) format. These files represent CAD drawings readable by FlashCut CAD version 6 or later. This command is unavailable when no recent changes have been made.





Import silhouette, centerline, or color image

Selecting the **Import Tools** icon from the ribbon brings up a menu with three different options: **Import silhouette image**, **Import centerline image**, and **Import color image**.



Import silhouette image – creates an outline of the silhouette of an image.

- **Use Arc Fitting**

When selected, **Arc Fitting Tool** will be applied to the imported image.

- **Fitting Tolerance**

The **Tolerance** can be set in drawing units.

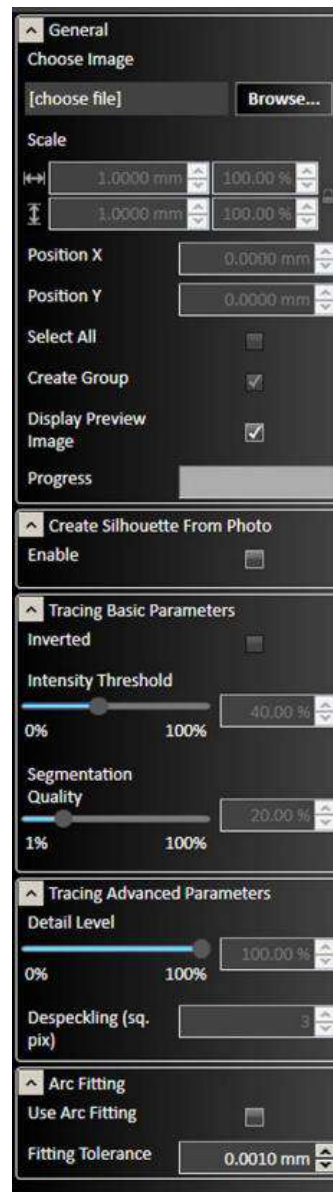
A lower number will increase accuracy.

Import centerline image – creates an outline of the centerline of features in an image.

Import color image – creates an outline based on the colored areas of an image.

Understanding silhouette and centerline images

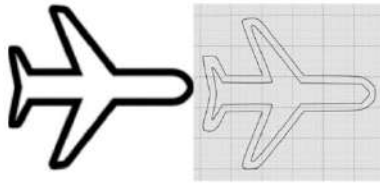
Below are three images: the original line drawing (left), the drawing imported as a silhouette (center), and the same drawing imported as a centerline image (right). When creating a silhouette, FlashCut attempts to identify the drawing by its contrast against the background. Note how both sides of the plane's outline have been reproduced. In the centerline image, FlashCut has translated the shape into a single line.





Import silhouette image

This tool takes an imported image and renders it as a series of closed line segments, forming a silhouette. Higher resolution images generally produce a silhouette with more precise edges and corners.



After changes are complete, you may either Accept or Cancel the changes.



Choose image Click **Browse...** to select an image file from the computer. Select **Open** to call up the desired image.

Scale Select how large or small the image will appear in the drawing relative to its original size. Scaling can be done as an absolute dimension or as a percentage of the original image size.

Position X Enter the value for the X coordinate of the lower left corner of the imported image.

Position Y Enter the value for the Y coordinate of the lower left corner of the imported image.

Select All When checked, the entire silhouette is selected for movement or reshaping after parameters are confirmed. When not checked, nothing will be selected.

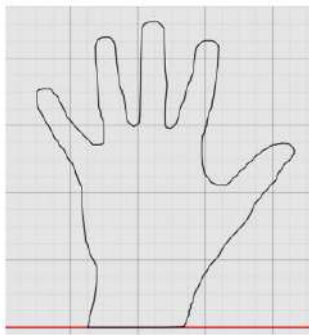
Create Group The Create Group checkbox allows the user to import the image as a Group instead of importing the image as individual elements.

Display Preview Image Superimposes a translucent preview of the imported image while the new drawing is being generated.

Progress Indicates the rendering progress after changes are made to the drawing. Making edits during rendering consumes more system resources. It is advised to wait for rendering to finish between edits.

Create Silhouette from Photo Finds the boundary between the background color of the image and any other color. This is beneficial for importing the profile of a part/item, when the part/item is taken in front of a solid color backdrop.





When you enable Create Silhouette from Photo, FlashCut presents these options:

- **Background color** –Select which corner of the image that is to be used to sample the background color. The color sampled from the specified corner will be set as the background color. When importing a photo, the edges of the object are determined by the color contrast between an object and the selected background color. The Color Tolerance setting is used to adjust the level of contrast.
- **Pick small details** – Increase the value if the algorithm missed some small details. Import performance may be lowered if this value is increased. 10% is a good default value.
- **Color Tolerance** – This tolerance is used to determine the contrast between the background and the silhouette. Pixels within this range will be considered part of the background. Range: 0- 100.

Tracing Basic Parameters

- **Inverted**, when checked, this option reverses the shapes enclosed by the curves of the silhouette.
- **Intensity threshold** Select how much detail FlashCut CAD transfers from the image to the final drawing. When increased, the program increases the number of features. Range: 0-100.
- **Segmentation quality** Select how finely the program will divide curves. FlashCut CAD automatically breaks curves into separate line segments. When segmentation quality is increased, the program divides curves into smaller segments, preserving more detail. This also increases the size of the drawing file, and the program's memory usage. Range: 1-100.

Tracing Advanced Parameters

- **Detail level** Select the degree of accuracy of the lines in the silhouette to the original image. A smaller detail level allows for more variation from the original drawing, while a larger level replicates the image more closely. Range: 0-100.
- **Despeckling** Reduces the number of small dots that appear in the image. It also can reduce the overall detail of the image being imported. The larger the number, the lower the dot tolerance, causing fewer dots to appear in the imported silhouette. Range: 0-100.

Arc Fitting

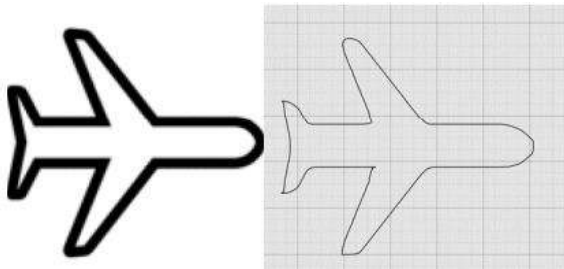
- **Use Arc Fitting** When selected, Arc Fitting Tool will be applied to the imported image.
- **Fitting Tolerance** The Tolerance can be set in drawing units. A lower number will increase accuracy.







Import centerline image

This tool imports a bitmap image and renders each feature as a single toolpath line down the center of the feature. Images with defined lines generally result in a cleaner drawing that requires fewer revisions.



After changes are complete, you may either Accept or Cancel the changes.  

Choose image Click Browse... to select an image file from the computer. Click Open to call up the desired image.

Scale Select how large or small the image appears in the drawing relative to its original size.

Position X Enter the value for the X coordinate of the lower left corner of the imported image.

Position Y Enter the value for the Y coordinate of the lower left corner of the imported image.

Segmentation quality Select how finely the program will divide curves. FlashCut CAD automatically breaks curves into separate line segments. When segmentation quality is increased, the program divides curves into smaller segments, preserving more detail. This also increases the size of the drawing file, and the program's memory usage. Range: 1-100.

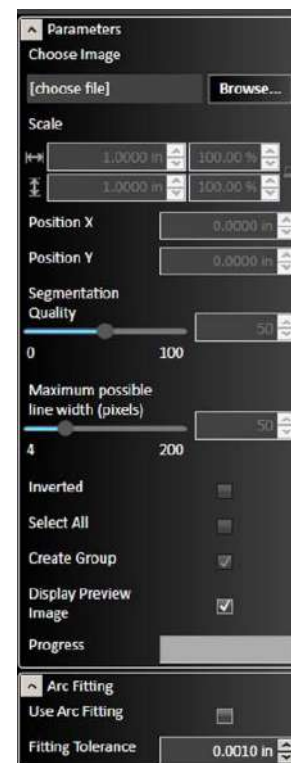
Inverted When checked, this option reverses the shapes enclosed by the curves of the image. The relationship is the same as that between a part and a cutout: if inverted, the object created from the imported image will behave like a cutout (below, right).

Select all When checked, the entire image is selected for movement or reshaping after parameters are confirmed. When not checked, nothing will be selected.

Create Group When checked, the imported drawing objects will be grouped together. **Display Preview Image** Superimposes a translucent preview of the imported image while the new drawing is being generated.

Progress Indicates the rendering progress after changes are made to the drawing. Making edits during rendering consumes more system resources. It is advised to wait for rendering to finish between edits.

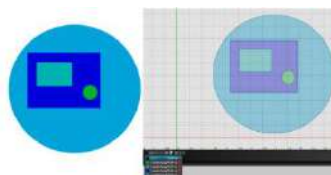
Arc Fitting When Use Arc Fitting is selected, Arc Fitting Tool will be applied to the imported image. The Fitting Tolerance can be set in drawing units. A lower number will increase accuracy.





Import color image

This tool creates shapes from the source drawing based on color. Boundaries between different colors define where the lines are drawn. Each new shape is placed on a unique layer, which is color coded. In the example below, the original bitmap image (left) is translated into four shapes (right), each occupying its own color-coded layer. See Layers for more information about manipulating drawing objects using layers. After displaying and changes are complete, you may either Accept or Cancel the changes.



Choose image Click Browse... to select an image file from the computer. Click Open to call up the desired image.

Scale Select how large or small the image appears in the drawing relative to its original size.

Position X Enter the value for the X coordinate of the lower left corner of the imported image.

Position Y Enter the value for the Y coordinate of the lower left corner of the imported image.

Select all When checked, the entire image is selected for movement or reshaping after parameters are confirmed. When not checked, nothing will be selected.

Create Group When checked, the imported drawing objects will be grouped together.

Display Preview Image Superimposes a translucent preview of the imported image while the new drawing is being generated.

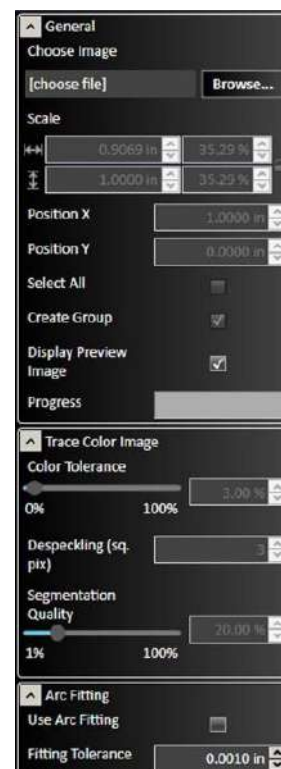
Progress Indicates the rendering progress after changes are made to the drawing. Making edits during rendering consumes more system resources. It is advised to wait for rendering to finish between edits.

Color Tolerance Defines color tolerance for grouping silhouettes by color. Range: 0-100.

Despeckling Reduces the number of small dots that appear in the image. It also can reduce the overall detail of the image being imported. The larger the number, the lower the dot tolerance, causing fewer dots to appear in the imported image. Range: 0-100.

Segmentation quality Select how finely the program will divide curves. FlashCut CAD automatically breaks curves into separate line segments. When segmentation quality is increased, the program divides curves into smaller segments, preserving more detail. This also increases the size of the drawing. Range 1-100.

Arc Fitting When Use Arc Fitting is selected, Arc Fitting Tool will be applied to the imported image. The Fitting Tolerance can be set in drawing units. A lower number will increase accuracy.





Import DXF/DWG image

This tool imports a two-dimensional DXF or DWG file into the drawing. After changes are complete, you may either Accept or Cancel the changes.

To choose an image Click **Browse...** to select a DXF file from the computer. Click

Open to call up the desired image.

Scale Select how large or small the image will appear in the drawing relative to its original size. X and Y values will be scaled equally. **Imported layers** Select which layers from the DXF file you want to appear in the drawing.

Position X Enter the X value of the program zero position here. FlashCut sets program zero based on the specified point on the imaginary rectangle that contains all DXF file entities.

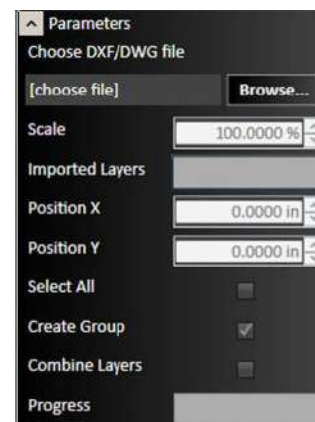
Position Y Enter the Y value of the program zero position here. FlashCut sets program zero based on the specified point on the imaginary rectangle that contains all DXF file entities.

Select All When checked, the entire image is selected for movement or reshaping after parameters are confirmed.

Create Group When checked, the imported elements are grouped together as a single object.

Combine Layers When checked, combines multiple layers in target drawing into one.

Progress Shows progress of file import.

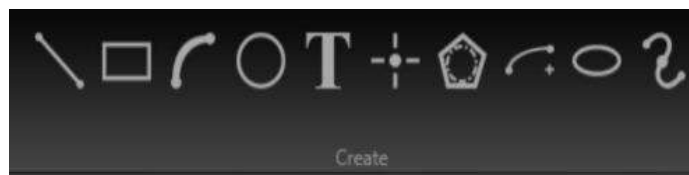


Create Tools

These tools add new elements to the drawing. After selecting a tool, move the cursor into the drawing window in order to begin constructing the element. You may use the mouse to place and size each element or you can edit

parameters for the element, such as dimensions and location, in the parameter window after creating the element. Click the shape or features and then edit the desired parameters. After changes are complete, you may either Accept or Cancel the changes.

Each tool (with the exception of **Point**) lets you define the Treatment parameter to determine whether the element is cut or only marked.



Select **Cut Element** to use the plasma cutter. This is the pre-selected option. These elements appear in the CAD window as solid black lines.

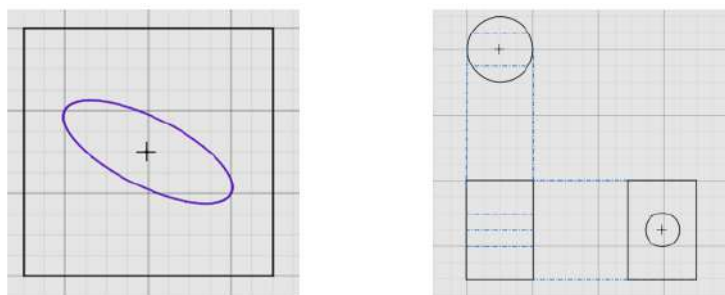
Select **Mark Element** to use the scribe. Marked elements appear in the CAD window as purple solid lines. If the drawing contains a marked element that is not contained within a part, FlashCut will notify you when you send the drawing to CAM.

In addition, you may select the **For Construction** option to indicate that the feature is a construction line and should not be cut or marked. Construction lines appear in the CAD window as blue dashed lines and are not used by FlashCut CAM and CNC.








The drawings below contain a marked ellipse inside a cut rectangle (left), and an orthographic projection using construction lines (right) to indicate the relationships between the three views.






Note that the behavior of all of the Create tools is influenced by the active Snap tools.


Line  Lines may be created either as **Continuous Lines** by adding segments and vertices with each mouse click, or as a simple **Two Point Line**. When creating a Two Point Line, the properties may be adjusted manually in the parameters window. Any single line segment (e.g., any segment that is part of a polygon or a multi-segment line) may be selected and its properties displayed. Continuous lines are used for reference lines in the construction selection.

Rectangle  FlashCut CNC offers several different options for constructing rectangles, including selecting **two corner points**; selecting a center and corner points; selecting **three corner points**; selecting a center and two outer points; and selecting **three points** to generate a parallelogram. Select which method will be used, and then click the rectangle points in the drawing window. You can manually enter point coordinates as well.

Arc  FlashCut CNC offers several different options for constructing arcs. The **Centerpoint Arc** prompts you to select a center point, a point on the arc, and the start and end angles of the arc. The **Tangent Arc** allows you to select an endpoint of a line or curve and then select a second point on the outer radius of the arc. This creates an arc tangent to the line or curve at this point. The **3 Point Arc** enables you to select two points on the curve, followed by a third point that determines the degree of curvature in between these points.


Circle  FlashCut CNC offers two different options for constructing circles. The **Center Circle** enables you to select a center point, followed by a point on the circumference of the circle. The **Perimeter Circle** prompts you to select three points on the circumference, through which the circle will be drawn.

Text **T** FlashCut CNC enables you to enter text into drawings. All TrueType fonts installed on the system are available. Text objects are scalable, and can be styled, aligned, etc. In the drawing field, click the desired point for the text. This point varies with the alignment setting of the text. Once the text is correctly configured choose the **green check mark**   or press **Enter** to Accept the changes. Note If you have the advanced text feature, there is no need to explode the text to create a toolpath, unless you are changing the text with tabs or welding.


Point  Click on the drawing to create a new point at that location.






Elliptic Arc  + Creates an arc that is a section of an ellipse.

Ellipse  Creates an ellipse.


Spline  Creates a complex curve in multiple segments. When complete, each point can be manipulated by using the blue control line, or by editing the parameters.


Shape Tool  FlashCut CNC offers a library of shapes for both simple and complex parts. Select the shape tool from the ribbon to load the library in the parameters area. To choose a specific shape, select it from the list. Shapes can be edited for size and characteristics.


Modify tools


The Modify tools on the ribbon affect a shape that is already in the drawing. First, select the features to be modified, and then click the tool to modify the feature. You may edit parameters for the modification in the parameter window.



Apply Chamfer  Select two intersecting lines or their intersection point, and then click **Apply Chamfer** to add a sloped chamfer to this edge. You can scale the size of the chamfer by dragging the mouse towards or away from the chamfer, or in the parameter window, type the distances from the intersection in both directions and the slope angle. Choosing the **Lock Angle** option causes both distances to change when one of them is edited, holding the angle measurement constant. Once the chamfer is correctly configured choose the green check mark or press ↵Enter.


Apply Fillet  + Select two intersecting lines or their intersection point, and then click **Apply Fillet** to add a rounded fillet to this edge. You can scale the size of the fillet by dragging the mouse towards or away from the fillet, or in the parameter window, type in the radius of the fillet. Once the fillet is correctly configured choose the green check mark or press ↵Enter.


Extend Tool  Click the **Extend Tool**, and then hover the selection cursor over the endpoint of the line or curve to be extended. A preview of the extension should appear, in blue, to show where the line or curve will intersect with another object in the drawing. Click the mouse to finalize the extension.


Trim Tool  Click the **Trim Tool**, and then hover the selection cursor over the line or curve to be trimmed. A preview of the cut should appear (in red, below) where the line or curve will be removed. Click the mouse to finalize the trim.








Scale Entities  Select a feature or group of features to be scaled and click **Scale Entities**. FlashCut CNC offers you two options for scaling. The **3 Points** scale prompts you to select a center point and a point close to the feature. Drag the mouse to increase or decrease the size of the features about the center point and click to apply the modification. The **Point and Factor** scale enables you to select a center point and type in a factor by which the features will be scaled.


Apply Offset  Select that feature and then click **Apply Offset**. You may then select parameters for the offset in the parameter window. The offset tool duplicates selected features and positions them at the specified distance away from the original feature. **Reversed** changes the direction of the offset distance. **Bidirectional** creates a second copy, opposite the first. When the Bidirectional option is chosen, the Close Opened Wires option appears. **Close Opened Wires** option closes the gaps created by the offset ends when chosen. **Use Arc Connectors** rounds out the corners of the offset by transforming them into arcs.


Fix Drawing  The **Fix Drawing** tool is used to correct mistakes in the drawing that could prevent accurate toolpaths from being created, such as line segments that do not touch. To identify issues and fix the drawing, select the icon, and choose a **Join Tolerance** or **Overlap Tolerance** larger than any of the errors in the drawing. Use the **Search Issues** button to identify issues with the drawing that could lead to mistakes in fabrication. Issues noted with a green circle can be fixed automatically by selecting **Fix All**. Issues noted with a red circle must be addressed manually.


Curve Text  The **Curve Text** tool modifies an existing text object to follow a curved shape. Start by creating both the text object and the curved shape you want to use. Select both the text object and the curve object in the viewport, then click **Curve Text**. The text will be automatically aligned with the curve. If the text object and curve object are not already selected. **The tool will provide instructions for how to select the text object and curve object.** The position of the text can be adjusted with the cursor and modified by changing the alignment parameter.

Cut Tool  Use the **Cut Tool** to divide an entity such as an arc or a line segment into multiple sections. Select the **Cut Tool** and click the mouse at the position on the entity where you want the cut to be made. You can verify the cut by hovering the cursor over the feature. Only a part of it will highlight if the cut worked properly. Explode Entities Transforms a feature that is solid and not composed of lines (such as a text object) into individual line segments. This is necessary for toolpath generation of any feature that is solid.

Explode  Transforms a feature that is solid and not composed of lines (such as a text object) into individual line segments. This is necessary for toolpath generation of any feature that is solid. Select a solid feature, and then click **Explode**. Exploding is necessary to **Bridge** fonts.

Group  The **Group** command combines selected objects so that they can be modified or transformed as a single unit. If you choose this command with no objects selected, you will be prompted to select the objects to be grouped, and to confirm the action by right-clicking.

Ungroup  The **Ungroup** command separates the selected grouped objects. If you choose this command without a group selected, you will be prompted to select the objects to be grouped, and to confirm the action by right-clicking.

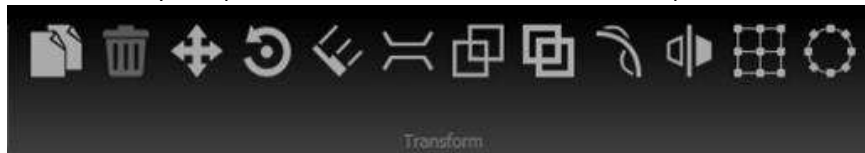
Ungroup All  The **Ungroup All** command can separate multiple sets of grouped objects in one operation. If you choose this command without a group selected, you will be prompted to select the objects to be grouped, and to confirm the action by right-clicking.








Transform Tools



These tools do not change the physical properties of the drawing features. Instead, they modify the size, scale, and number of features in the drawing. Select the features to be modified and select the specific transform tool to use. You may edit parameters for the transformation in the parameter window.





Copy Entities  Select the features to be copied in the drawing window, choose the Copy Entities button and then click a reference point about which the new features will be copied. Position the cursor at the paste location and click to paste the copied features. You may paste multiple features until you exit from the tool.


Delete Selected Objects  Select the features to be deleted, and then click the Delete Selected Objects tool. The features will be removed.


Move Entities  Select the features to be moved, and then click the Move Entities tool. Click the mouse to set a reference point, and then move the cursor to move the feature(s) in relation to that point. Click the mouse again to select the new location for the features.

Rotate Entities  FlashCut CNC offers two modes for rotating features in relation to a set point. The 3 Points rotation prompts you to select a center point, a point near the feature, and a final point to which the feature will rotate. The Point and Angle rotation requires you to select a central rotation point, and then enter a rotation amount in degrees. After changes are complete, click the green check mark or press ↵Enter to Accept changes or  the red X mark to Cancel changes.

Measure Entities  with Ruler You can measure entities in one of three modes: as a ruler between points or between entities, or as a protractor between points on an arc. The tool allows for measurement between any two points, and displays offset in X and in Y directions.

Bridge Entities  Connects two or more objects with a bridge that spans the gap between them. Can also be used to divide a single object into separate objects. The width of each bridge can be set individually. To create a bridge, select Bridge Entities. Use the cursor to draw a line. This line will become the bridge. You can modify the width of the bridge if desired. Select the green checkmark to Accept the change.

Perform Boolean Operation  on Entities Perform Boolean Operation can be used to modify one shape by adding or subtracting another shape to it. The three available operations are Weld (combine) solid areas, Weld Cutouts, or Subtract. Select Perform Boolean Operation, select the specific operation in the parameters window, then select the objects one at a time, right clicking to confirm each selection. The first object selected will be modified by the second. The example below shows the results of all three operations on the initial set of shapes.

Weld Selected Entities  To quickly combine multiple shapes into a single shape, first create the desired overlapping objects. Select the Weld Selected Entities tool, select the objects, and right-click to confirm the operation.





Arc Fitting Tool



Create arc and line segments for objects that are not already drawing objects, such as imported images. The Tolerance can be set in drawing units. A lower number will increase accuracy. Progress during recalculation is shown in the bar. When Show Graphics is selected, an outline lines and arcs will be displayed in the drawing window. This feature is also available when using the Import tools.

Mirror Entities



Select the features to be mirrored, select the Mirror Entities tool followed by a point through which the features will be mirrored. Clicking the mouse will move the features to a new position. When the Copy Object box is chosen, the features will be duplicated to the new position rather than only moved.

Linear Pattern Tool



The linear pattern tool duplicates selected features in a grid fashion. Select the feature to be duplicated, select the Linear pattern tool then specify parameters. Alternately, click and drag the blue points on the drawing screen to select the parameter for Total Distance. After changes are complete, click the green check mark or press \downarrow Enter to accept changes or the red X mark to cancel changes.

Circular Pattern Tool



The circular pattern tool duplicates selected features in a circle around a specified point. Select the feature to be duplicated, select the Circular Pattern tool, and then specify the parameters.

Pan and zoom tools

The pan and zoom tools are located at the top left of the workspace and are used for moving and magnifying the drawing in the drawing workspace. Click on a specific tool to change the cursor's function. Pan Click and drag anywhere in the drawing window to move the drawing around. Right clicking and dragging in the window allows you to pan without selecting the pan tool first. Zoom Extent Click and drag a box in which to zoom in. This tool allows to user to focus in on a specific section of the diagram. Zoom Click and drag anywhere to zoom in and out on the drawing. You can also use the scroll wheel to zoom; scroll up to zoom in and scroll down to move out. Zoom to Fit Click this button to adjust zoom automatically to best fit the drawing.



Snap tools

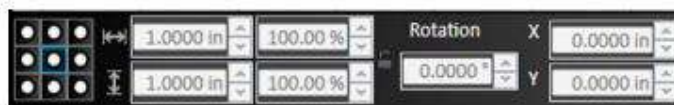
The snap tools affect how drawing tools interact with the background grid or with features already created.



Clicking a snap tool toggles it on or off. Snap tools cause a feature or segment to lock on to the closest active snap location when the cursor is dragged nearby. Note that more than one type of snap can be active at one time. Note that the snap tools can also be reviewed and selected by right clicking in the drawing window. This brings up a contextual menu, including a sub-menu to control snaps.

Free form stretching and manipulation tools

Any objects or set of objects in FlashCut CAD can be stretched, positioned, or rotated by manipulating a set of control points with the cursor, or by entering values using the toolbar. Note that different types of objects respond differently to scaling. For example, circles will expand, and straight-line segments will be extended, but ellipses and arcs will be divided into multiple line segments.



Layers

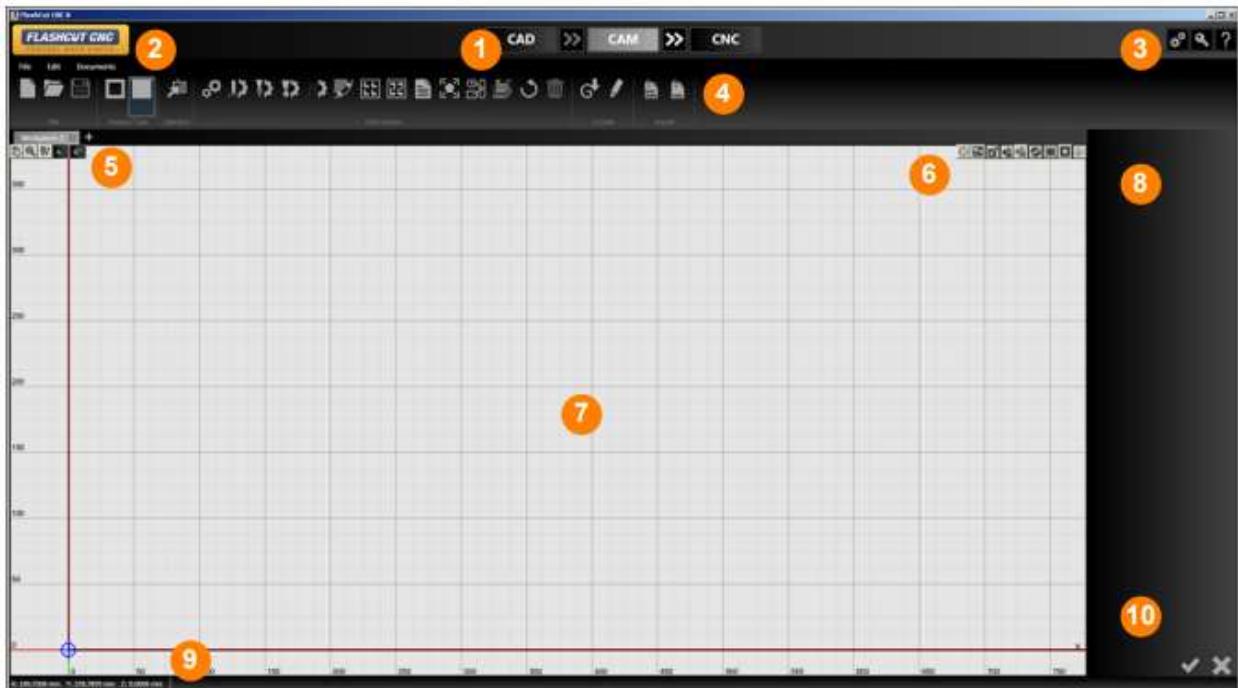
Drawings can have multiple layers. Each layer can have no objects, one object, or many objects. Layers can be used to selectively display objects or modify how they will be treated in Flash cut CAM.





FlashCut CAM

FlashCut CAM (Computer-Aided Manufacturing) generates an accurate toolpath from a CAD drawing. This toolpath is used to create the G-Code that will be used in FlashCut CNC. The FlashCut CAM main screen is shown here. An explanation of each area of the screen follows.



An explanation of each area of the screen is provided in these topics:

- | | |
|---|--|
| 1 Tabs | 6 Display options, tools, Part shading and Grid tools |
| 2 Menu bar | 7 Drawing workspace |
| 3 Configuration, License, and Help buttons | 8 Parameters area |
| 4 Ribbon | 9 Status bar |
| 5 Pan and zoom tools | 10 Accept / Cancel |





Getting Familiar with CAM Menu & Ribbon



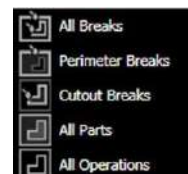
File tools There are two file tools on the ribbon: • Open (Ctrl+O) • Save (Ctrl+S)

Make Top Level Feature a Cut Out If this option is selected, the outermost level of the drawing will be understood as a cut out. For example, a simple shape (circle, rectangle, etc.) will be cut so as to make a precisely defined aperture in the workpiece. The lead in will begin inside the boundary defined by the outermost line, and the kerf will be placed inside the line. Note When this option is selected, you will not be able to use either grid nesting or true shape nesting. These options will be greyed out in the ribbon.

Make Top Level Feature a Part If this option is selected, the outermost level of the drawing will be understood as a part. For example, a simple shape (circle, rectangle, etc.) will be cut so as to preserve the material inside the line defining the boundary of the shape. The lead in will begin outside the boundary defined by the outermost line, and the kerf will be placed outside the line. Note This option is selected by default.

Selection When you click the **Selection** icon, FlashCut pops up the following choices for selection options: This tool allows you to select various objects in the drawing by type. Note Breaks are the points on the toolpath where any lead in and lead outs will be placed and is represented by a red diamond on the toolpath. For a discussion of breaks, see **Lead in/out settings**.

- All Breaks Selects all available breaks in the document, including perimeter breaks and cutout breaks.
- Perimeter Breaks Select this option to select all breaks that are on the outside of the part. When the toolpath is offset to the outside of the part the break on this toolpath is defined as a perimeter break.
- Cutout Breaks Select this option to select all breaks that are on the inside of a cutout. When the toolpath is offset to the inside of a feature or a cutout, the break is called a cutout break.
- All Parts Choose this option to select all of the parts in the viewport. This allows the user to drag and drop all parts to another position on the workpiece.
- All Operations Choose this option to select all of the operations used to fabricate the part.



When either All Parts or All Operations is selected, the following parameters can be set.

Tool

- Fabrication Head Change fabrication head, if desired.

General

- Override Kerf Width When selected, permits changing the default kerf width. See Kerf Width.

Custom Kerf Width

- Only displayed when the Override Kerf Width option is checked.

Feedrate factor

- Default is 100%.





Tabs

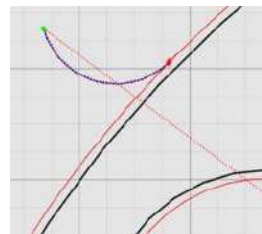
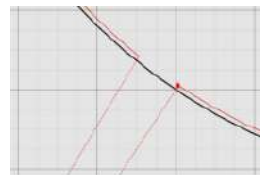
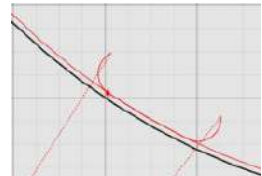
- Override Tabs When selected, permits changing the parameters used for any tabs included in the toolpath.
- Use Tabs When selected, tabs will be added to the toolpath.
- Tabs Width Specify the width of each tab, in document units.
- Tabs Placing Method Alternatives are Specify Spacing and Specify Number.
- Spacing Specify spacing between tabs, in document units.
- Minimal Tabs Count Specify value between 1 and 1000.
- Outside Corner Specify the behavior of tab placement at the outside corner. Alternatives include Avoid, Consider, or Prefer.
- Inside Corner Specify the behavior of tab placement at the outside corner. Alternatives include Avoid, Consider, or Prefer.

Lead in/out settings

Lead in and **lead out** lines are used to provide a way for the torch to ease into a shape. Usually, they are oriented in the same direction as the toolpath in order to ensure a clean cut of the final piece. The lengths of these lines are initially calculated from the thickness of the workpiece. The initial locations of these lines are automatically generated by an internal algorithm using basic rules. FlashCut CAM enables you to override these initial settings for each individual feature.

Editing settings To adjust the lead in and lead out settings, select the red diamond icon on the desired feature. This brings up the parameters window. If you want to change the location of the lead in and lead out, then you can simply drag the red diamond along the toolpath to the desired location. In the examples to the right, the top drawing shows a closed break with an overburn. The cut begins with an arc shaped lead in, and finishes with an arc-shaped lead out. The parameters for this break are shown below. The bottom drawing shows a tab break, no lead in or lead out.

Manual editing FlashCut CAM now permits manual adjustment of individual lead ins and lead outs in the toolpath. Click on the red diamond to make the line segment active. A blue dot appears at the end of the line. Hover over the blue dot until it turns green. This dot can now be used to drag the end of the lead in or lead out to the desired position. The example at right shows a single lead in with its end point selected (cursor not shown for clarity).



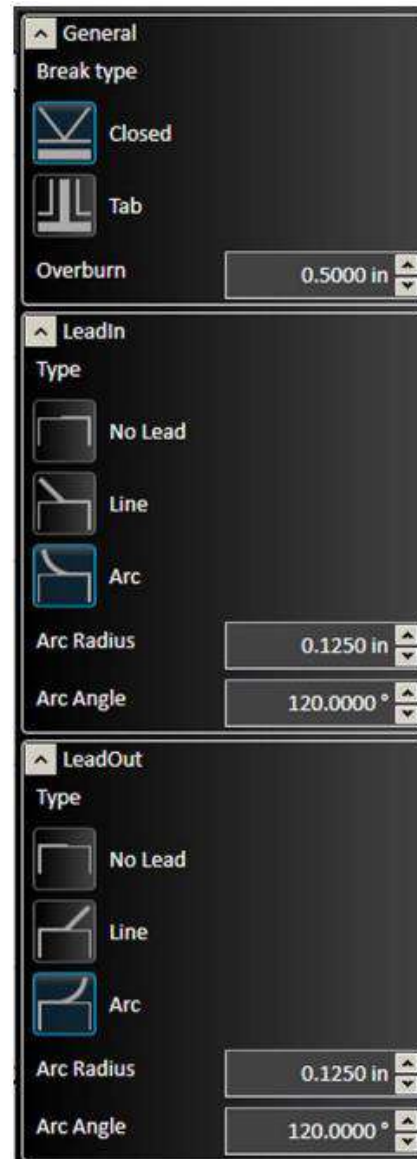


Break Type

- Closed When selected, the lead point and the end of the cut will overlap, closing the break between the beginning and end of the cut.
- Overburn To ensure a clean cut, you may set the torch to continue through a cut along the same toolpath. This feature allows you to determine how far this overburn will go. Lead out lines appear after the overburn.
- Tab When selected, there will be a gap between the lead point and end of the cut, creating a tab break between the beginning and end of the cut.
- Width The distance of the tab can be set here.

LeadIn/LeadOut Type Select the type of leading line to be used for the lead in and lead out lines.

- No Lead removes the respective line.
- Line results in a straight line. If you choose Line, the Line Length and Line Angle options below are displayed.
- Arc creates an arc tangent to the feature. If you choose Arc, the Arc Radius and Arc Angle options are displayed.
- Line Length Select the length of the leading line.
- Line Angle Select the angle that the leading line makes with the toolpath. The angle is measured clockwise.
- Arc Radius Select the radius of the circular arc.
- Arc Angle Select the angle that the arc travels through, measured from the edge of the toolpath.






CAM Actions

The CAM actions bar enables you to replicate and nest various features in the CAM drawing, as well as select various settings for lead in and lead out lines and the plasma torch. Clicking each option brings up numerous settings in the parameter window.



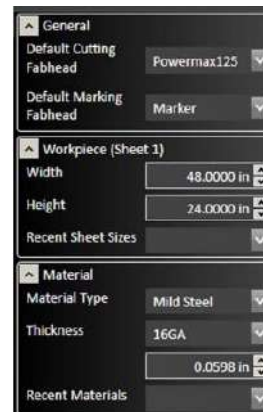
Project Settings  Configures options applied to the current CAM project. Click the Project Settings icon to bring up the following options in the Parameters window.

General

- Default Cutting Fabhead Select the default cutting fabhead. Multiple fabheads can be defined in the Configuration interface.
- Default Marking Fabhead Select the default marking fabhead.

Workpiece


- Width Specify the width of the sheet being cut. Changes to this value will change the size of the material in the drawing window as well as any nesting settings.
- Height Specify the height of the sheet being cut. Changes to this value will change the size of the material in the drawing window as well as any nesting settings.
- Recent Sheet Sizes Select the sheet size from a list of recently selected material types.



Material

- Material Type Specify the material type. The parameters will automatically adjust to match your selection.
- Thickness Specify the thickness of your material. The parameters will automatically adjust to match your selection.
- Recent Materials Select from a drop-down list containing recently selected materials.

Material thickness and type MUST match what is being cut!!!!

Change Project Tab Settings  When selected, tabs will be included in the toolpath for the drawing.

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

- Cutting at a lower amperage will yield the best results. Test cuts should be made to identify best quality setups.

Plasma Settings plasma fab head settings can be adjusted here. Click the icon to bring up options in the parameters window. You can override any of the recommended settings by simply typing in the respective field. After specifications are complete, you may either Accept or Cancel the changes. The values in the parameters are derived from Hypertherm's factory settings and are close with only minor tweaks necessary. Changes made here will apply to the specific drawing, but will not change the values set in the configuration window. The configured values for the fab head are in turn populated from the cut charts.





Amperages and settings set in Flashcut MUST match consumables and power unit settings!!! Failure to do so will end in incomplete and/or erratic cuts.

Nozzle Select your plasma torch nozzle. The parameters will automatically adjust to match your selection. Note that each of the following parameters will be automatically adjusted when the nozzle is specified. However, they can also be input manually

Quality Level Toggle whether the toolpath will be optimized for best quality or fastest cutting.

Feedrate Specify the default feedrate at which the machine will move while cutting.

Kerf Width Input the width of the kerf of the plasma torch. This will determine the thickness of the cut, and the toolpath will change to reflect the new size. The toolpath is automatically offset outside by $\frac{1}{2}$ the kerf width for parts, and automatically offset inside by $\frac{1}{2}$ the kerf width for cutouts.

Pierce Height Specify the height at which the torch will initially pierce before cutting along the toolpath. If your machine is set to do a touch off at the beginning of each cut, then the pierce height is relative to the last touch off location.

Cut Height Specify the height at which the torch will cut the part along its toolpath. If your machine is set to do a touch off at the beginning of each cut, then the pierce height is relative to the last touch off location.

Pierce Delay Specify the time between the command for the torch to fire and the motion of the machine. The delay allows the torch time to pierce completely through the material prior to any other machine motion.

Voltage This is the target voltage for the THC system in order to maintain a specified cut height. The set point value is determined by the cut chart for each plasma torch, and can also be found in the operator's manual of the plasma torch. It and is dependent on the material type, thickness, torch settings, and other parameters. When **sampling** is enabled, the system will automatically detect this voltage and track it throughout the cut. The higher the voltage the higher the torch will cut, lower the set point to bring the torch closer to the material during cut moves.

Amperage Specifies the operating amperage of the torch.

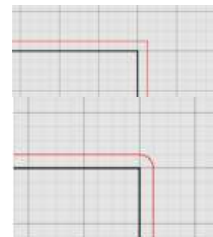
Pressure Specifies the air pressure of the torch in psi.

Note-consult your troubleshooting manual for adjustments to tweak cuts. Adjust minimal rates.

Corners Toggle whether the corners on the toolpath will be sharp or rounded. Different types of corners will result in more precise cuts depending on the qualities of the fab head and the material. Changes to the corner settings may take a short time to render the toolpath.

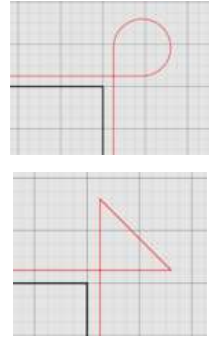
Edge Start When checked, indicates that the cut will start at the edge of the workpiece. The torch should sit as close to the material as possible without piercing.

- **Sharp corners** The toolpath follows the same angle as the part, maintaining a consistent distance from the edge.
- **Rounded corners** The toolpath describes an arc at the corner of the part, maintaining a consistent distance from the edge.





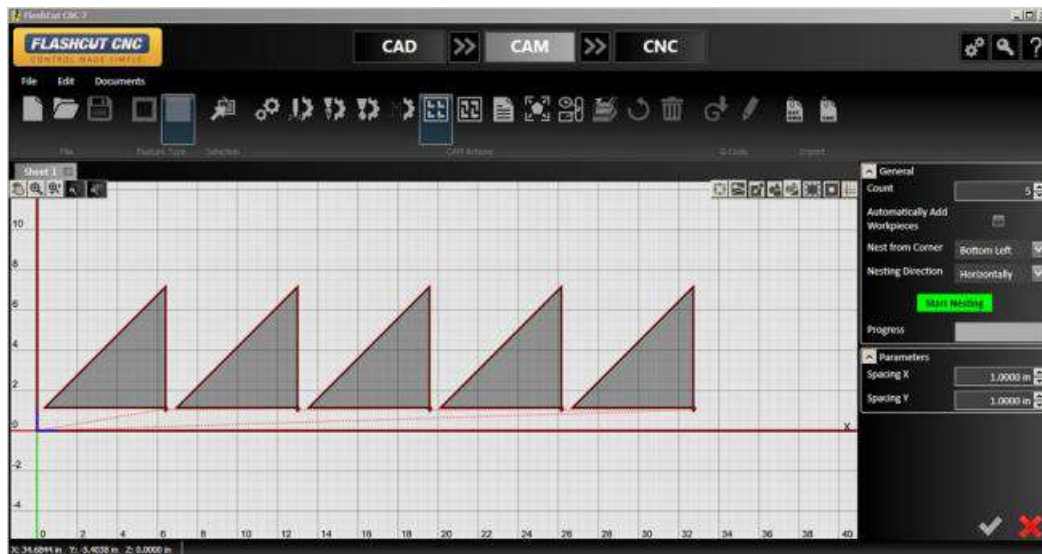
- **Rounded loops** The toolpath proceeds past the corner and describes a loop to reorient the fab head in the desired direction to cut the next section of the part. May result in a more precise cut at the corner.
- **Triangular loops** The toolpath proceeds past the corner and describes a triangle to reorient the fab head in the desired direction to cut the next section of the part. May result in a more precise cut at the corner.



Grid Nesting

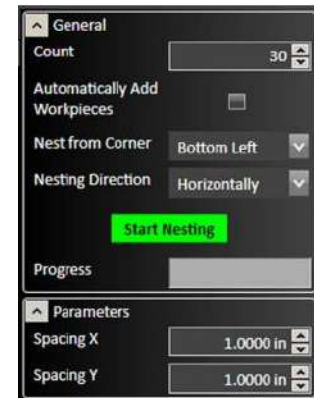



The grid nesting function duplicates a part in a grid pattern. In the parameter window, type in the count of parts that you want to cut along with the X and Y spacing of parts in the grid. The number of rows and columns is determined by the sheet size. Copies fill rows left to right, and when the edge of the material is reached, they will advance up a row. Grid nesting does not change the orientation of any of the parts being nested as illustrated in the screen below where 5 triangles are grid nested. The parameters for grid nesting are described below.

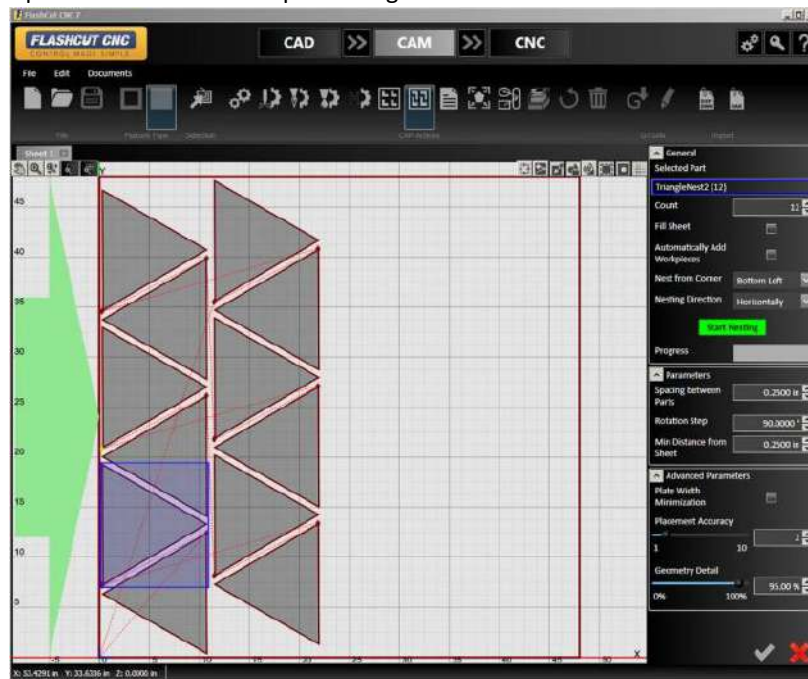




- Count Determines the total number of parts that will be nested, inclusive of the original.
- Automatically add workpieces Adds a new sheet to the drawing containing a new workpiece if the number of shapes to be cut exceeds the capacity of the current workpiece.
- Nest from corner Selects the starting point of the nesting operation.
- Nesting direction Selects the direction from the starting point in which new nested parts will be added.
- Start Nesting Starts the nesting operation
- Progress Shows the progress of the nesting operation. Complex nesting operations can take significantly longer.
- Spacing X Sets the horizontal spacing between parts.
- Spacing Y Sets the vertical spacing between parts.



TrueShape Nesting  The TrueShape nesting function will duplicate, rotate, and translate each part to create a nest that minimizes the amount of material used in a sheet. Twelve triangles are nested below using TrueShape Nesting. Notice the material usage is much lower for TrueShape nesting as it is for the same number of triangles using Grid nesting. The parameters for TrueShape Nesting are described below.





Selected Part If more than one part is in the CAD drawing, this option enables you to determine the count of each part that you need in the nest. A solid blue box will appear around the shape that is currently being replicated.

Count Determines the total number of parts that will be nested, inclusive of the original.

Fill Sheet When selected, parts will be added to fill the size of the current workpiece. Accepting the operation will change the count.

Automatically add workpieces Adds a new sheet to the drawing containing a new workpiece if the number of shapes to be cut exceeds the capacity of the current workpiece.

Nest from corner Selects the starting point of the nesting operation.

Nesting direction Selects the direction from the starting point in which new nested parts will be added.

Start Nesting Starts the nesting operation

Progress Shows the progress of the nesting operation. Complex nesting operations can take significantly longer.

Spacing between Parts Determines the minimum distance between parts. Takes compensated toolpath, lead ins and lead outs into account. Minimum: 0.0001

Rotation Step Specify how many part rotations will be tried by the algorithm. For example, if the value is set to 90 degrees, the system will try angles 0, 90, 180 and 270. A smaller step may produce a tighter nest, but it will also increase computation time. Setting the value to 360 means no rotation will be applied to parts. Range: 0-360 degrees.

Min Distance from Sheet Determines the minimum distance from the edge of the sheet to any feature on any part including kerf compensation, lead ins and lead outs. A value of zero corresponds to the edge of the material.

Plate Width Minimization Toggle in order to minimize the horizontal space taken by the nesting shapes. Vertical arrangements will take priority. Enabling this option can produce better nests at the cost of a slight increase in area.

Placement Accuracy Specify how accurately the algorithm will try to nest parts. Smaller values may lead to parts spacing larger than specified. Higher values will increase placement accuracy, but will also increase computing time. Range: 1-10.


Geometry Detail Specifies how much parts are simplified for nesting calculations. Decreasing this value may lead to faster nesting with a decrease in accuracy. Range: 0-100%.

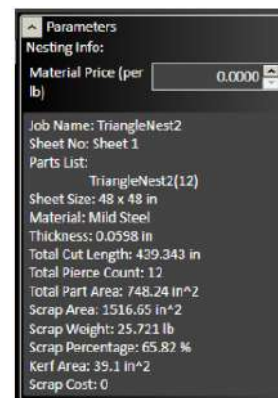
The screenshot displays the software's control panel, organized into three sections:

- General:** Shows the 'Selected Part' as 'TriangleNest2 (12)'. The 'Count' is set to 12. There are checkboxes for 'Fill Sheet' and 'Automatically Add Workpieces', both currently unchecked. 'Nest from Corner' is set to 'Bottom Left' and 'Nesting Direction' is set to 'Horizontally'. A green 'Start Nesting' button is visible, along with a 'Progress' bar.
- Parameters:** 'Spacing between Parts' is 0.2500 in, 'Rotation Step' is 90.0000 degrees, and 'Min Distance from Sheet' is 0.2500 in.
- Advanced Parameters:** 'Plate Width Minimization' is unchecked. 'Placement Accuracy' is a slider set to 2 (range 1-10). 'Geometry Detail' is a slider set to 95.00% (range 0-100%).








Nesting Information  Generates a report from the nested parts, estimating material use, waste, and cost. A sample generated from the nested triangle example (above) appears here (right). Note that material price can be input directly into the field. All other information is generated automatically from the drawing




Fit workpiece to the parts  Shrinks the workpiece to fit the area defined by the parts.


Sequence tool  When selected, the sequence tool will display the order in which parts will be cut or operations executed. The example below shows a trio of parts being cut from left to right. To change the order, click the edge of a feature and drag the arrow to the next feature you want to execute. When finished, select the green check mark to finalize changes.


Simulation Tool  Allows visualization and analysis of the material removal process. An example of a completed simulation appears below. The normal CAM toolpath window appears on the left. On the right, an isometric view of the cutter and the material is displayed. The controls (right) can be used to start, stop, pause, advance, or rewind the simulation. The slider with the arrows controls the speed at which the simulation plays

CAM Reset  Resets CAM page and clears all drawings as well as clearing the tool path.


Delete  Deletes the selected toolpath.


G-Code The G-Code tools on the ribbon allow you to create or edit a G-Code program for the CAM process for the current workpiece.

Create G-Code  file Click the Create G-Code file icon to create a G-Code program for the CAM process for the current workpiece.

Open editor  Click the Open editor icon to open the FlashCut CAM G-Code editor with the G-Code program for the CAM process for the current workpiece.

Import Files can be imported into the CAM drawing workspace. If other objects are added, new toolpaths will be generated. Note Objects imported as files will not appear in the CAD drawing space.

DXF/DWG  Imports DXF/DWG file to the CAM drawing workspace. The file selection dialog, and other options are shown in the Parameters window.

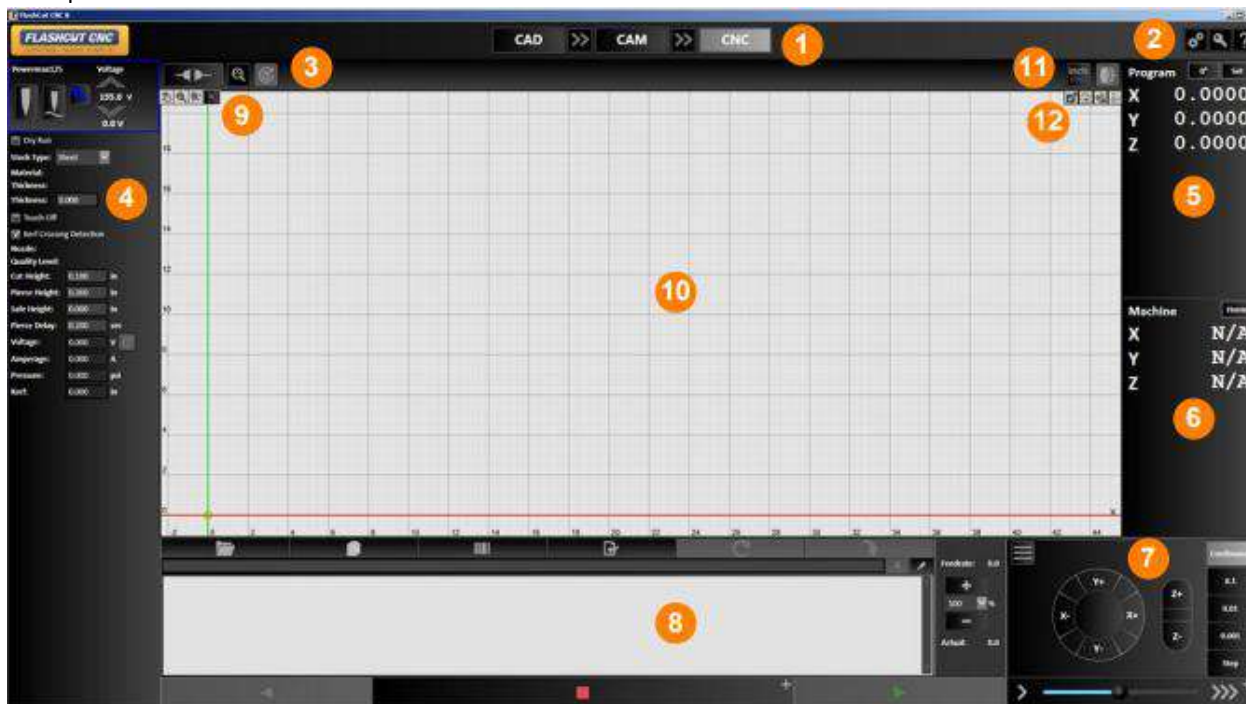
CADCAM  Imports a CAD/CAM file to the CAM workspace. The file selection dialog and a progress bar are shown in the Parameters window.





FlashCut CNC

FlashCut CNC (Computer Numerical Control) control is the final step in the cutting process and may be accessed at any point from the other two steps, or independently if you already have a file to import. The panel is used to program and control each axis on your machine via the FlashCut CNC Controller. For comprehensive information on programming and G-Code, see the Programming Reference. The main screen is shown below. An explanation of each area of the screen follows.



An explanation of each area of the screen is provided in these topics:

- | | |
|---|--|
| 1 Tags | 7 Jog and point control panel |
| 2 Configuration, License, and Help buttons | 8 G-Code window |
| 3 System Status, Connect, and Reset Motor Drivers | 9 Pan and zoom tools |
| 4 Fab head settings | 10 Drawing workspace |
| 5 Program coordinates panel | 11 Toggle Display Units
Toggle manual control |
| 6 Machine Coordinates Panel | 12 Material
Machine Envelope
Part Numbers
Show/Hide Grid |





System Status, Connect, and Reset Motor Drivers


System Status Click the magnifying glass icon to view the live status of the input and output lines. Input Lines If a normally closed (NC) input line is normal, it will be in the closed state and the blue LED will not be illuminated. If the machine is connected, these icons are live when a switch changes its state. If an NC switch is tripped then it will be in the open state and the blue LED will be illuminated. Normally, only input lines that have been defined are displayed. However, if the show all checkbox is selected, the status of all input lines will be displayed.

Output Lines The output line status icons are either white for off, or blue for on. You can also control the state of these switches by clicking on them here.

Controller The serial number and USB speed of the controller are also shown.

Connect Clicking this icon connects or disconnects the signal generator. When connected, the icon is illuminated and the two halves are in contact. Ensure that the signal generator (CNC controller) is securely connected via USB cable to your PC, and that the USB driver is installed. Click the Connect icon. FlashCut begins communication with the signal generator. If there is a problem with the connection, a dialog box appears. FlashCut CAD/CAM and CNC Control Software Page 116 When properly connected, the Connect icons will join and turn blue. Press Disconnect prior to unplugging the signal generator. When the signal generator is connected, all moves are performed by the machine tool. Before the unit connects, a safety reminder screen appears. It is imperative that you and anyone else near the machine understand, agree with and adhere to all of the safety guidelines. If the safety guidelines are not accepted, the software will not connect.

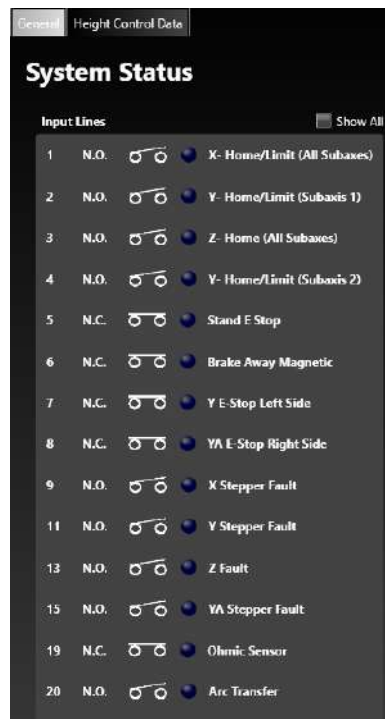
Note, please refer to your Troubleshooting Manual for System Status Alert codes.

Reset Motor Drivers  Toggles the enable line to reset all motor drivers controlled by the system.

Fab head settings Each fab head that has been configured and saved will be displayed here. See Configuring FlashCut for more information. If multiple fab heads are configured, the parameters for a specific fab head can be viewed and edited by selecting the appropriate tab.

Laser pointer control If a laser pointer is configured, it can be turned on and off by clicking the icon. The pointer can also be locked in the on position.

Marker control (Scribe) If a marker is configured, it can be turned on and off by clicking the icon.

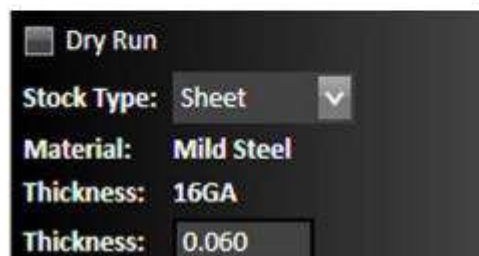




Plasma torch settings

Parameters are populated from Settings set in the CAM window. If an editable parameter is changed here, the settings in CAM will not be affected. Some controls will not be displayed unless they are enabled in the fabrication head configuration panel. See Plasma in Configuring FlashCut.

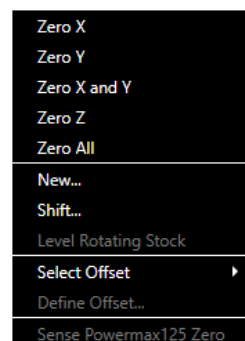
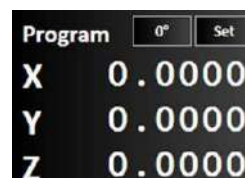
- **Torch On/Off** – Click to turn the torch on or off. Before turning on the torch, FlashCut CNC will display a warning dialog asking if it is safe to proceed.
- **Torch Height Control** – Click to turn THC on or off.
- **Use Sampling** – Click to turn sampling on or off.
- **Voltage** – Click up or down arrows to raise or lower the voltage.
- **Dry Run** – When selected, the machine will execute the operations, but the torch will not be lit, and cuts will not be made.
- **Material and Thickness** – These properties are visible here but are configured in the **CAM** window under **Project Settings**.
- **Touch Off** – The torch will touch the material in order to establish a zero point on the machine coordinates. Toggling Touch Off will make the torch touch down to reestablish the zero point between rapid and feedrate moves.
- **Kerf Crossing Detection** – When selected, helps prevent the torch from diving into the material where one cut crosses another.



Program coordinates panel

The program coordinates refer to the exact position of the tool with respect to Program Zero. The program coordinate system is referenced by the G-Code file as the set of absolute coordinates. Axes in the workspace represent the program coordinates. The **Set** dropdown menu sets program zero points along each axis, as selected from the dropdown menu.

- Zero X will zero the X axis.
- Zero Y will zero the Y axis.
- Zero X and Y will zero both X and Y axes.
- Zero Z will zero the Z axis.
- Zero All will zero each program axis.
- New sets the position of the tool relative to program zero.
- Shift sets the position of the tool relative to the last feed hold position
- Sense Powermax Zero will home and zero the torch head.



Program zero (X & Y) are set at reference 0,0 in cad unless otherwise edited. All drawings should be zeroed from the lower left corner of the material.





Machine Coordinates Panel

The machine coordinates refer to the exact position of the tool with respect to the machine home. The coordinates will read N/A until the tool is able to locate its home position. After connecting the signal generator to FlashCut, it is necessary to seek out and set the home point, or Machine Zero, of the tool. It is recommended that each axis is jogged first near the home switch before homing. Once machine zero is set, the machine tool envelope is redefined in the workspace.

The Home dropdown menu provides the following commands:

- **Home All** will reset all three axes to the home point.
- **Zero All** will set the current point as the origin of the machine coordinates.
- **Clear** will close the machine coordinate system. N/A will display on each axis. You will need to reestablish a machine zero point.

Your machine will need to be homed after first opening Flashcut

Jog and point control panel

Jog The jog control panel provides these controls for manually positioning all axes.

Axis jog buttons Pressing and holding an axis jog button (+X, +Y, +Z, -X, -Y, -Z) moves the machine tool exclusively on the selected axis. Ramping is used in cases where the jog rate is faster than the stop/start feedrate.

Diagonal jog buttons Pressing and holding a diagonal jog button (unlabeled) moves the machine tool evenly along two axes. Ramping is used in cases where the jog rate is faster than the stop/start feedrate.

If you want the move to be made at the maximum allowable speed, toggle the Plus icon and check the Rapid Move box.

Jog mode Determines the function of the jog buttons. Continuous jogging moves the tool at the assigned jog rate buttons are held down. Discreet distances move the tool incrementally the specified distance. Step jogging advances the motor exactly one motor step.

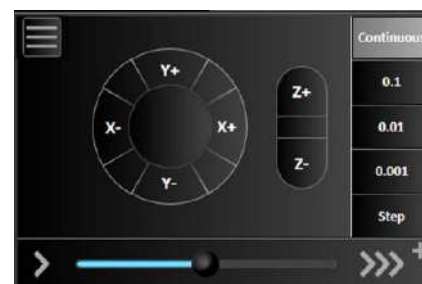
Jog rate Adjust the slider to change the speed at which continuous jogging will occur. Sliding to the left decreases the speed, while sliding to the right increases the speed. FlashCut CAD/CAM and CNC Control Software

Move to point allows you to move (jog) your tool to a specified exact point. You may specify the feed rate to be used for these moves by editing the value in the Feedrate field.

- Click the Program Zero button to move the tool to the program zero point shown on the Program coordinates panel.
- Click the Machine Zero button to move the tool to the machine zero point shown on the Machine Coordinates Panel.

If you want the move to be made at the maximum allowable speed, toggle the Plus icon and check the Rapid Move box

Note, machine can be moved by holding Ctrl and using the arrow keys to move the X and Y. PgUp and PgDn will raise and lower the Z axis.





Rip Cut is used to execute simple cuts without making a drawing or writing a G - Code program. Select the axes (X and/or Y) of the cut, the distance in each direction, and the federate. Press the green arrow to make the cut.



G-Code window



- | | | | |
|---|--|----|-----------------------------------|
| 1 | Open G-Code file
File name | 8 | G-Code workspace |
| 2 | Load all G-Code files in a folder | 9 | Feedrate override controls |
| 3 | Barcode | 10 | Run G-Code in reverse |
| 4 | Run CAD Import Wizard | 11 | Feed hold |
| 5 | Reset G-Code | 12 | Toggle G-Code run mode |
| 6 | Jump to Line | 13 | Run G-Code |
| 7 | Close G-Code file
Edit G-Code | | |

1. **Open G-Code file** When a CAD or CAM design is sent to be manufactured in FlashCut CNC, it appears in the G-Code workspace. However, to open an existing program manually, click the Open G-Code file button to browse to a G-Code file and open it in the G-Code workspace. A visual preview of the code will appear in the workspace. The name of the G-Code file is displayed in the G-Code workspace. This workspace displays the currently loaded G-Code file. While the program is running the current line of code is highlighted in real time, this way the user can track their progress throughout the operation. **File name** FlashCut CNC displays the file name of the currently displayed G-Code file here.
2. **Load all G-Code files in a folder** Prompts the user to select a folder. All G-Code files in the folder will be loaded into FlashCut CNC.
3. **Barcode** Allows the user to input a barcode representing a list of files to be cut.
4. **Run CAD Import Wizard** Allows the user to open DXF files directly into FlashCut CNC. Imported files will not be visible in the CAD and CAM windows.





5. **Reset G-Code** Click the Reset G-Code file button to reload the current G-Code file. This will not change the position of the tool. Jump to Line This button allows the user to jump or skip ahead to a specific line in the program.
6. **Close G-Code** file Click the Close G-Code file button to close the current G-Code file. You may replace it by loading a G-Code file, creating a part with FlashCut CAD or FlashCut CAM, or by writing a new G-Code manually with the G-Code editor.
7. **Edit G-Code** Click the Edit G-Code button to edit the currently displayed G-Code file with the G-Code editor.
8. **G-Code workspace** This workspace displays the currently loaded G-Code file. While the program is running the current line of code is highlighted in real time, this way the user can track their progress throughout the operation. Note Double-clicking in the G-Code workspace now brings up the G-Code editor.
9. **Feedrate override controls** Increases or decreases the defined feed rate by the given percent in the box. A setting of 100% override corresponds to zero change in feed rate. This function does not affect rapid move speeds, where feed rate is undefined.
10. **Run G-Code in reverse** This is useful when troubleshooting a program in simulation mode or dry run mode. Not typically used while cutting. Some commands are not possible to reverse through; For example, M106(fabhead change) and G611(advanced pierce options).
11. **Feed hold** Click the Feed hold button to pause execution of the G-Code file. The machine tool stops, ramping down if necessary. The slower the ramping rate, the longer it takes from the time the Feed hold button is clicked to the time the tool comes to a complete stop. This button pauses any motion including automatic changing, tool length sensing and so on.
12. **Toggle G-Code run mode** Click the Plus/Minus sign on the Feed hold button to toggle between continuous mode, and step or momentary mode.



- Run G-Code – solid arrowhead. G-Code will run to completion, unless Feed hold is pressed.
 - Run Step G-Code – striped arrowhead. One line of G-Code will be executed.
 - Run Momentary G-Code – outlined arrowhead. G-Code will run as long as button is depressed.
13. **Run G-Code** Click the Run G-Code button to begin execution of the current line of the G-Code file. When in step mode, execution stops automatically at the end of the current line, or when the Feed Hold button is clicked. When in continuous mode, execution continues until the end of the program, or until you click the Feed hold button. If the program has been stopped in the middle of a G-Code line, clicking the Run G-Code button begins execution exactly where the program stopped.

Progress meter When the Calculate Run Time for Progress Meter is selected, the G-code window will display a progress bar showing the total estimated time to complete the operation, and a countdown of the elapsed time. See G-Code.



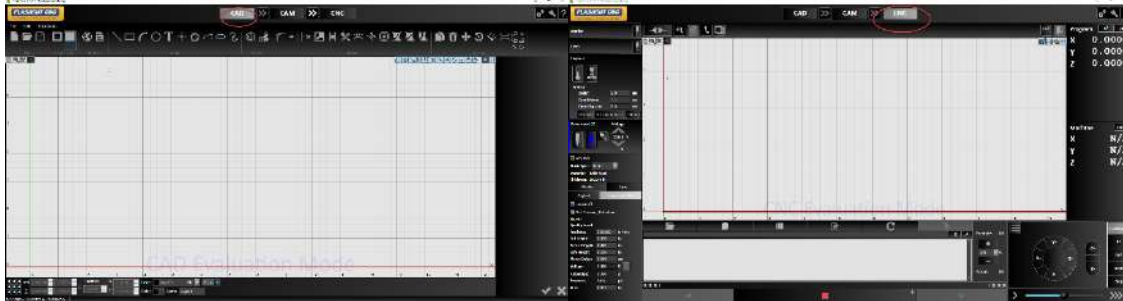


Operation



To Operate your Boss Table, begin by opening FlashcutCNC on your desktop.

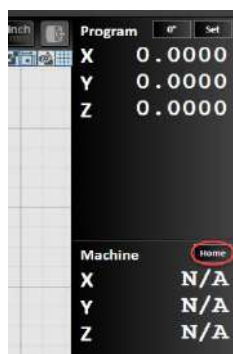
Flashcut will initially open CAD. Homing of the machine is required at this time. Select CNC to jog the machine.



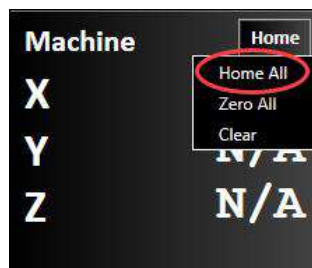
Note, Check the area of the table for obstacles and item on the rails. Remove items

Holding **Ctrl** and using the **Arrow** keys drive the machine to the lower left corner close to 4"-6" from the stop switches.

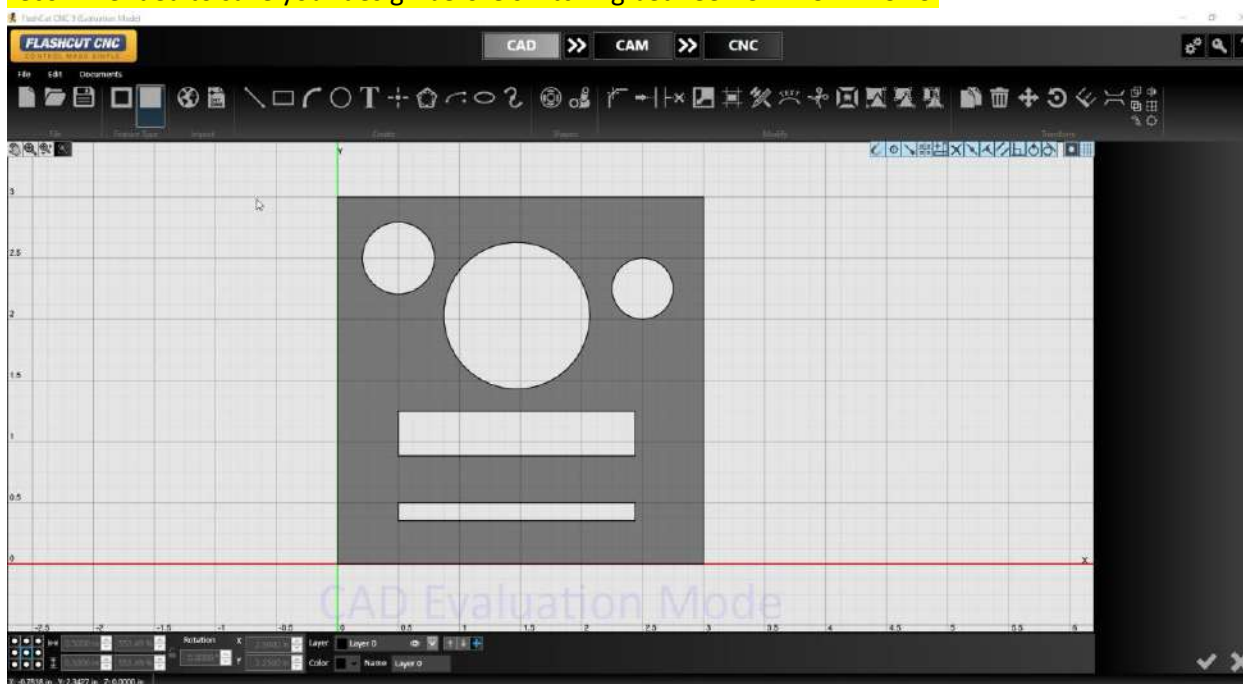






The stop switches and set the parameters for the table. **Your Boss Table requires homing after initially loading Flashcut after closing.**




Return to CAD and begin designing a drawing. To import a .DXF file into CAD select Import DXF/DWG File. Otherwise you can freely draw a design, open a previous cad/cam file or use one of the preloaded parts in flashcut. For this instance, we will use a 3"x3" square for our test cut. **It is recommended to save your design before switching between CAD>CAM>CNC.**

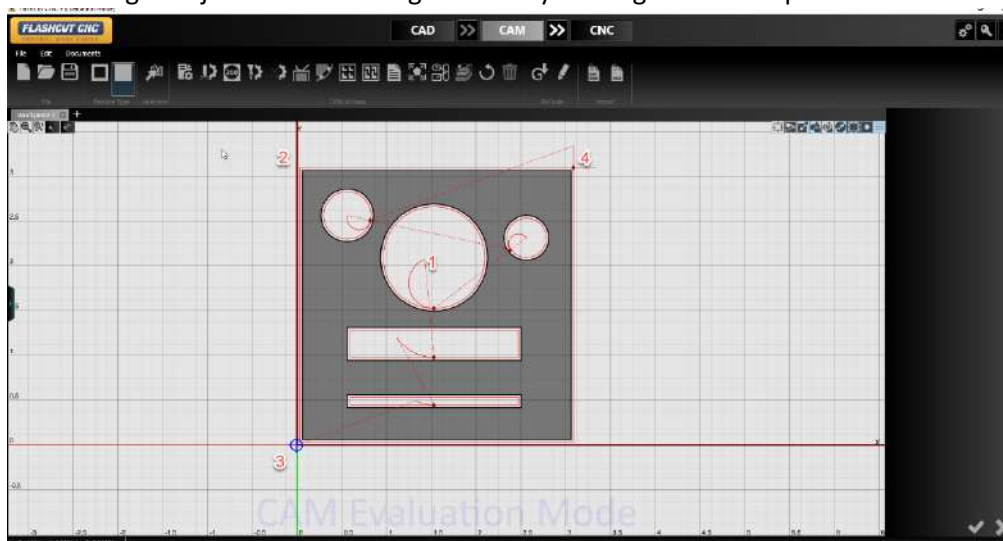




After completing an action select accept or cancel to continue.   Pressing **ESC** will remove the selected tool being used and revert to the previously used tool if not cleared. Once the design is finished, import it into CAM to setup tools and toolpaths. To do this click on the arrows between CAD and CAM.



Your design will be transferred to CAM. Toolpaths and Rapids will be automatically written but can be edited in the Change Project Breaks Settings  or by clicking on the toolpath.

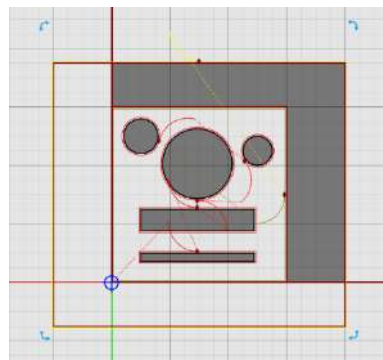





- 1- Shows an inside cut toolpath with an arched lead in. Lead in can be moved or sized by clicking the diamond associated with the lead in.
- 2- Cutting paths have clearance along the material edge. To move the design, highlight the whole drawing and move with the Arrow keys. Zooming in and out will affect the size of the movement steps.
- 3- This is the program's zero for the X and Y axis. All rapids after zeroing the machine will start from here.
- 4- Outside offset cut with a corner lead in. Lead ins can be moved by selecting the diamond associated with the lead in and dragging it along its path.








To flip lead ins from outside to inside simply draw a box around the design in CAD and import it into CAM. Select the boxes toolpath and press the **Delete** key. You will notice Flashcut has flipped lead ins to inside cuts.

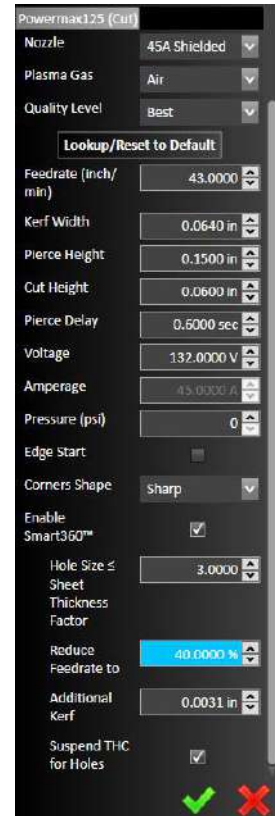


After editing the toolpaths, open the **Project Settings**  to edit the power source and material to be cut. **It is extremely important that the material and thickness selected matches what is being cut!!!** In this case we have selected a Powermax125, material size is 72"x144", mild steel is the type at 3/16" or .187" thickness. Finish by accepting or canceling  . Note, any issues in the project settings will prompt a message to appear in the upper project settings window.

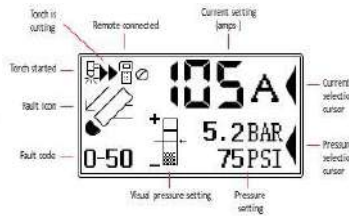
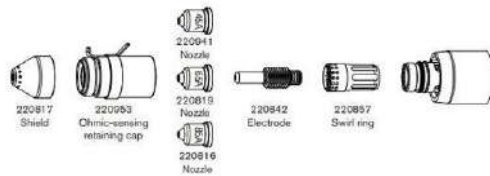




Next move to the Plasma Settings , this will be where the editing is done for the consumables and cut settings. Important, **The selected nozzle type must match what is loaded into the torch head!!! Example, 45 amp shielded is selected and 45 amp consumables are loaded into the torch. NO EXCEPTIONS!!** In this case we will be using a 45 amp Shielded tip. The recommended torch settings will automatically load. These numbers can be tweaked to achieve a superior cut. Please refer to the Troubleshooting Manual for instructions. By enabling Smart360 / Smart Hole Detection the torch will throttle down when cutting holes. In this instance the torch will throttle back to 40% when making a 3" or smaller cut with Torch Height Control (THC) turned off for holes. Finish by selecting accept or cancel.  



Mechanized shielded with ohmic consumables: Powermax85



All NEED TO MATCH!

- TIP
- AMP on Machine
- Setting in Controller

This is especially important as if you have chosen the improper tool you will have bad results.





Powermax85 (Cut)

Nozzle 45A Shielded

Plasma Gas Air

Quality Level Best

Lookup/Reset to Default

Feedrate (inch/min) 100.0000

Kerf Width 0.0610 in

Pierce Height 0.1500 in

Cut Height 0.0600 in

Pierce Delay 0.4000 sec

Voltage 133.0000 V

Amperage 45.0000 A

Pressure (psi) 0

EdgeStart

Corners Shape Sharp



Enable Smart360™

Hole Size ≤ Sheet Thickness Factor 3.0000

Reduce Feedrate to 14.0000 %

Additional Kerf 0.0107 in

Suspend THC for Holes

This is where you choose your tip size finecut, 45, 65, 85, Ect.

Gas is generally air

Best or production are the quality setting for the cut. Notice the feed change when adjusting.

You can change your feed rate manually and hit reset to default to reset.

Extend Pierce delay if cutting larger material and it does not pierce.

Voltage is your cut height, Higher the voltage the higher the cut.

PSI is changed manually on the machine.

Edge start is used when the material is too large to pierce.

You have the ability to change how the corner is cut.

Reduce federate for holes is especially useful to get better holes. 14% is low generally about 40-60 %

Suspend THC for holes should be checked if reducing federate for holes.

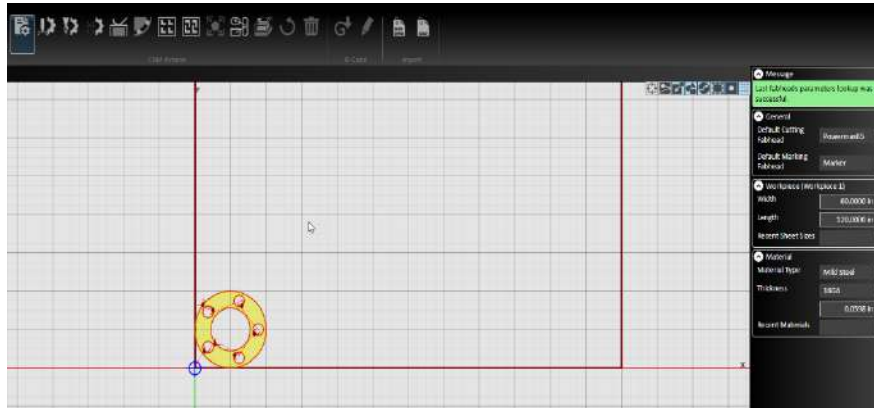
Path rules can always be left for holes. Holes sets the feed to a set percentage (50%) or choice of user from the chosen tool path. EX. 45 amp 3/16 steel cuts at 63 IPM. When the holes rule is applied the cut in the hole will travel at 31.5 IPM on the inside of the hole. This will greatly improve the kerf (taper) quality of you holes.



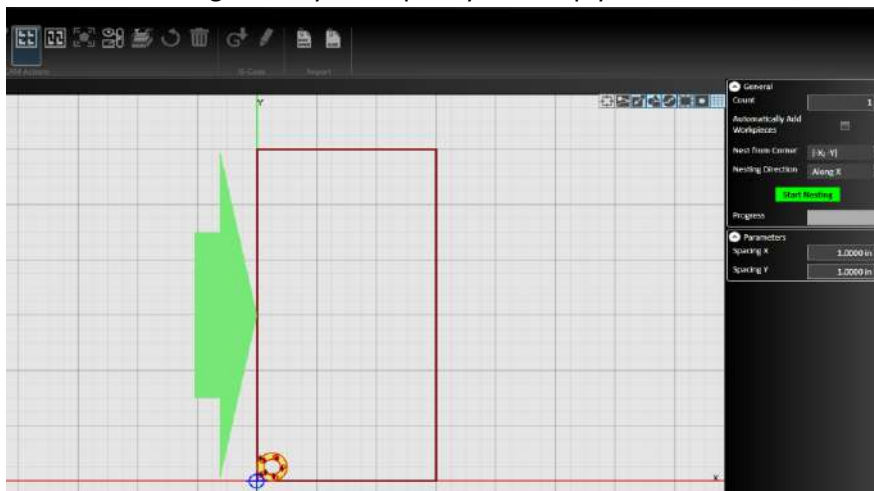


Nesting

Before preparing to nest it is important to enter the sheet size into Flashcut. This will ensure proper nesting amounts and accuracy.

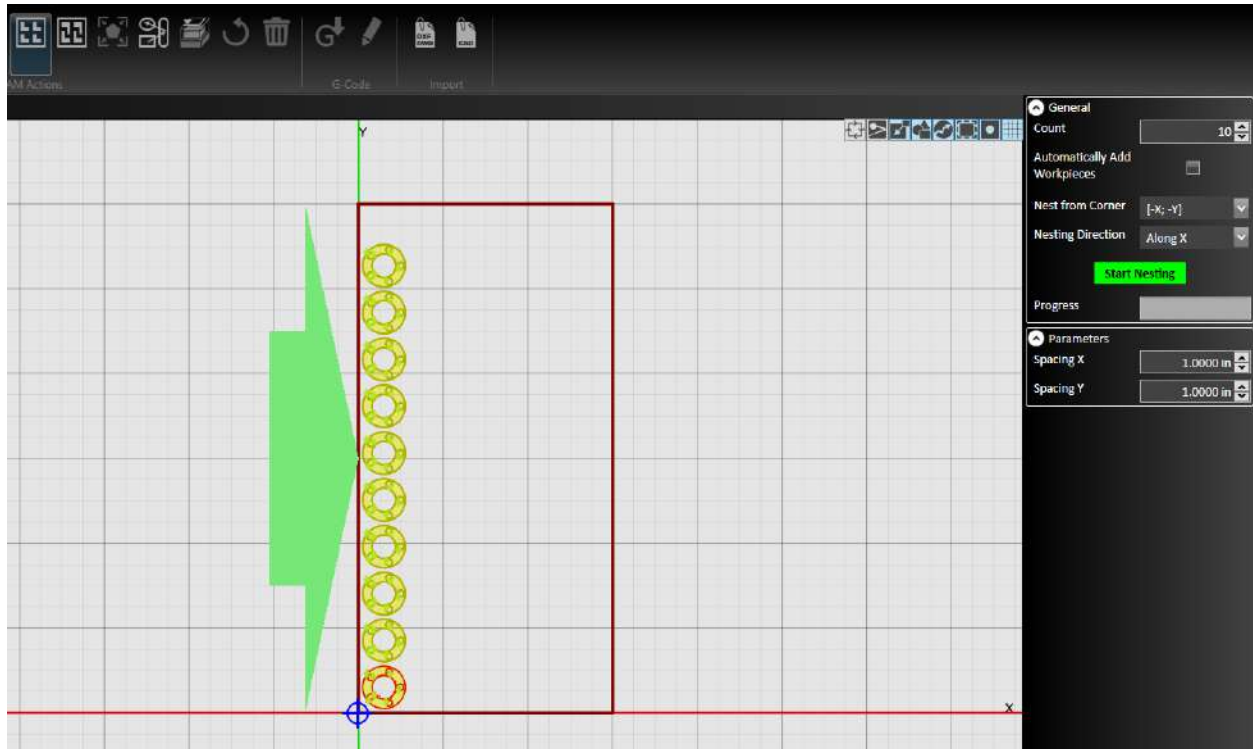


Grid Nesting allows you to quickly but simply nest out an item.



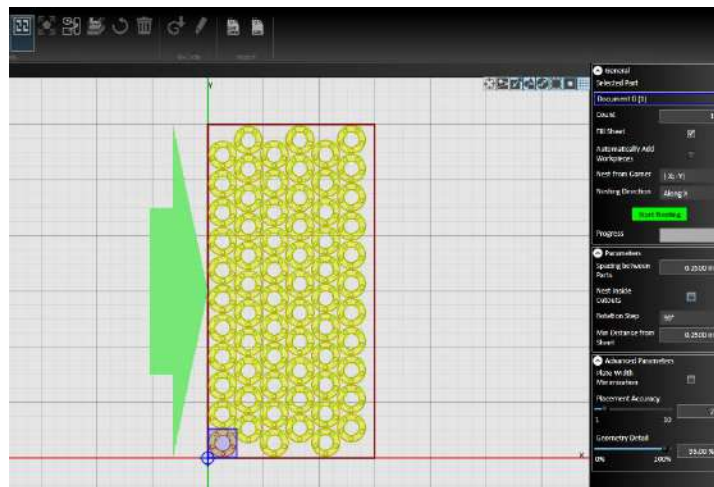
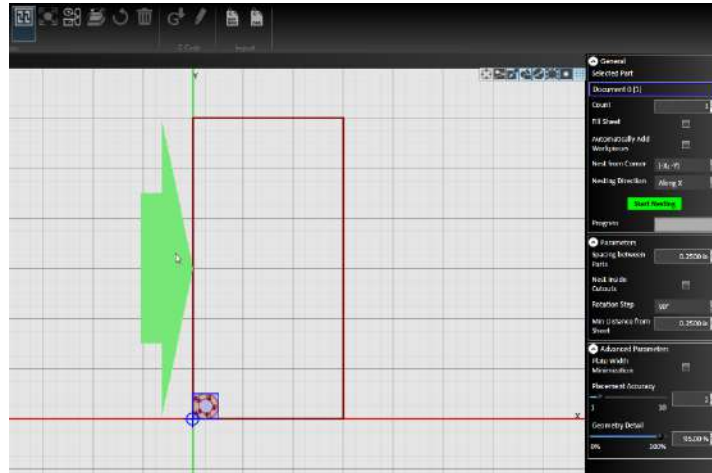


Select the number of parts to be nested and determine what orientation to nest them, the arrow next to the material box will guide you. Spacing can be entered as well. After or during nesting these can be changed. To nest click the Start Nesting button. If desired nesting is complete click the green accept check mark.



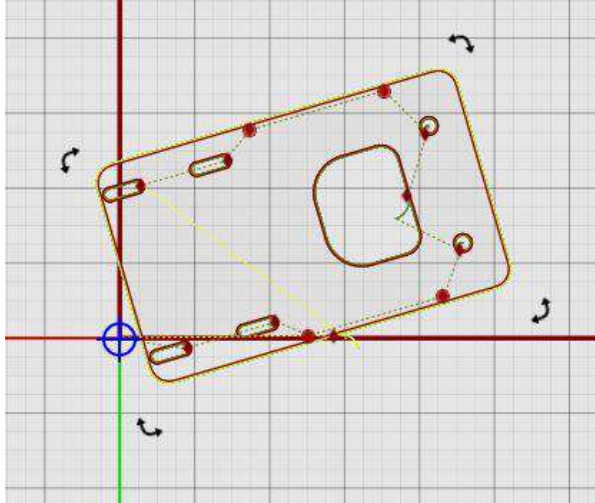


Trueshape Nesting works like grid nesting with the addition of more options. Trueshape Nesting allows you to nest multiple different parts in different rotations as well as simply filling a sheet. For a description of all the options see page 58.

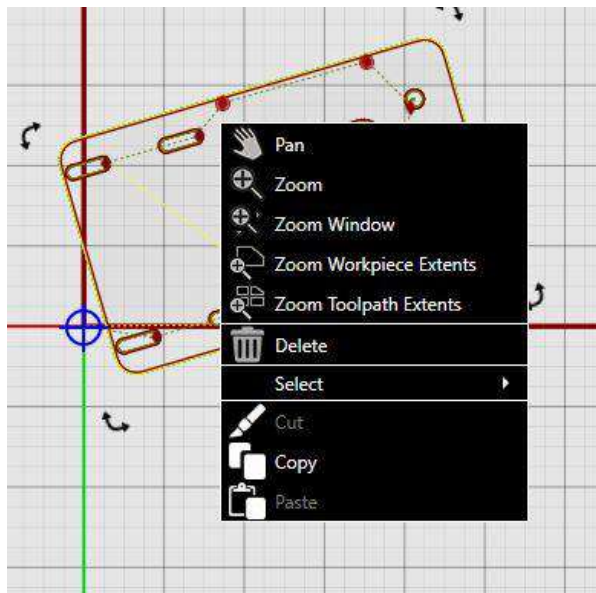




Your Part is imported now you need to nest your parts. Manual nesting can be performed by clicking on the profile of the part and dragging the part while holding the left click on mouse. You can rotate by dragging corners of parts like so.

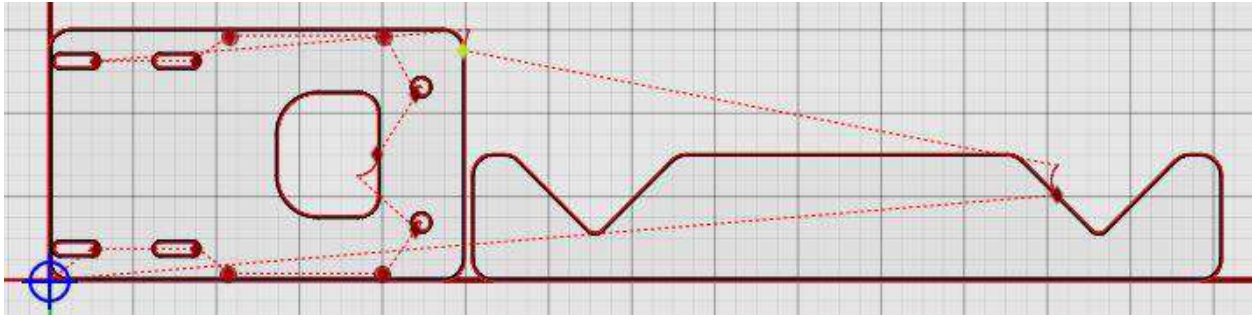


To Create Duplicates you can right click and hit copy or CTRL-C the Hit CTRL-V. They are the hot keys for cut and paste.

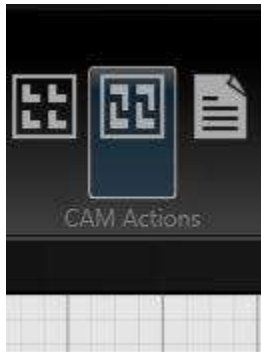




When you bring in another part it will automatically create a tool path like so.



This is the auto nesting button; it will allow you to choose the number part you want.



This is the auto nesting button. Click on it and you will open this window on the right-hand side.





▲ General
 Selected Part
 Brass bent 3-165 Holder (1)
 CLAMP BASE (1)
 Count
 Fill Sheet
 Automatically Add Workpieces
 Nest from Corner [-X; -Y] ▼
 Nesting Direction Along X ▼
 Start Nesting
 Progress

▲ Parameters
 Spacing between Parts
 Nest Inside Cutouts
 Rotation Step
 Min Distance from Sheet

▲ Advanced Parameters
 Plate Width Minimization
 Placement Accuracy
 1 10
 Geometry Detail
 0% 100%

The parts are listed in the top area. In the (#) is the total number of parts desired.

The number of parts needed are entered in the count. Highlighted part count are adjust in the count box and will change the total in the (#)

Fill sheet will automatically add parts

Automatically add work pieces will add more sheets as needed for great quantities of parts.

Nesting Corner will group parts in the specified corner

Direction will choose the side to hug parts to.

Part spacing will adjust the gap between parts AND THE LEAD IN. MOVE IF NEEDED TO NEST TIGHTER

Rotation step will change how many times it will rotate the parts at the determined angle. Smaller the step the more complicated parts will be nested but it will add time to the nesting process.

Distance from sheet will keep parts from the edge.

Placement accuracy 10 is the highest and will produce better nest but add time to process.

Geometry detail if raised to 100 will give more concise part definition

When finished hit start nesting and review nest.

Hit Check Mark to Save





Simulation Tool



Allows visualization and analysis of the material removal process.

An example of a completed simulation appears below. The normal CAM toolpath window appears on the left. On the right, an isometric view of the cutter and the material is displayed.

The controls (right) can be used to start, stop, pause, advance, or rewind the simulation.

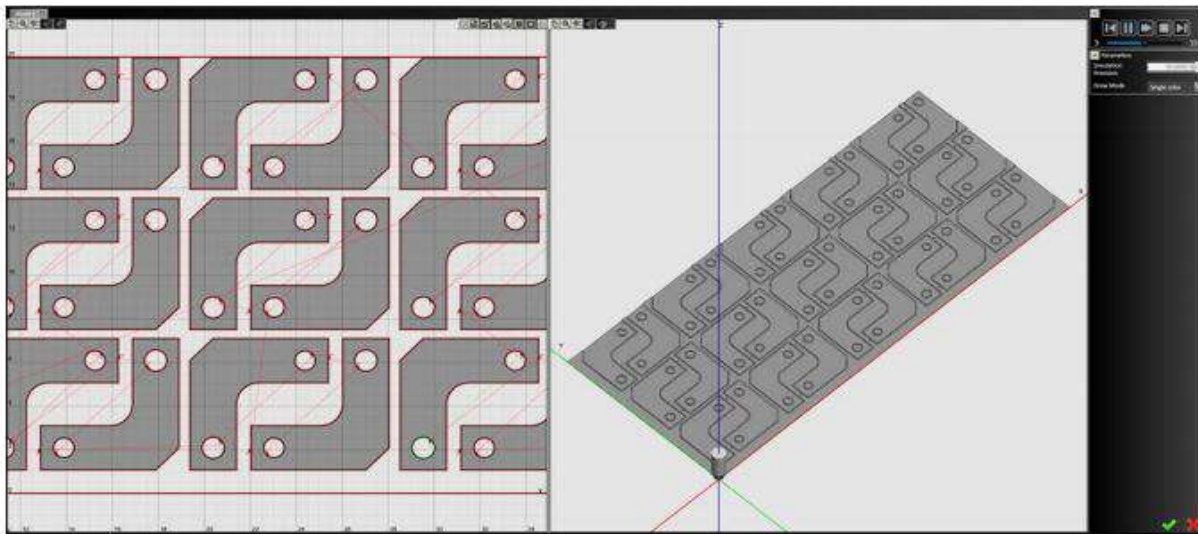
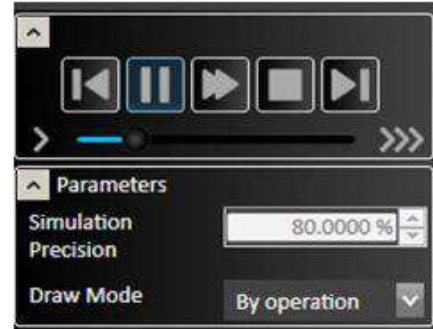
The slider with the arrows controls the speed at which the simulation plays

Simulation Precision

Range: 1-100%

Draw mode

Single color, or by operation.



CAM Reset



Resets CAM page and clears all drawings as well as clearing the tool path.

Once completed in CAM click the arrows between CAM and CNC to open CNC.

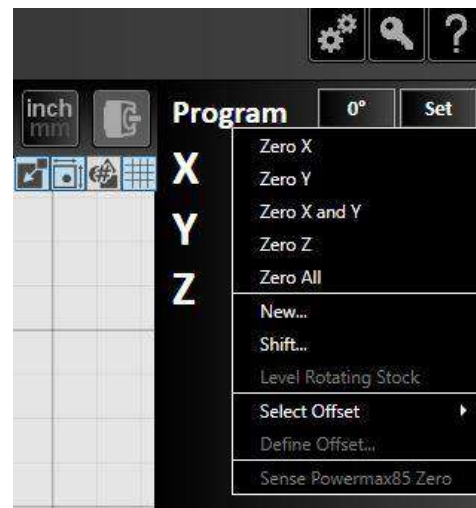
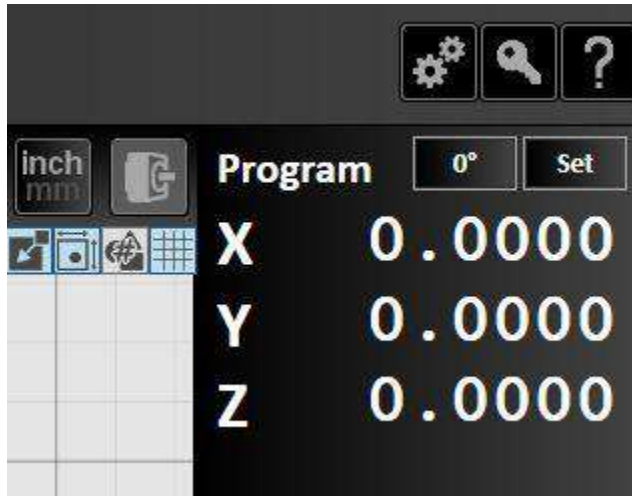
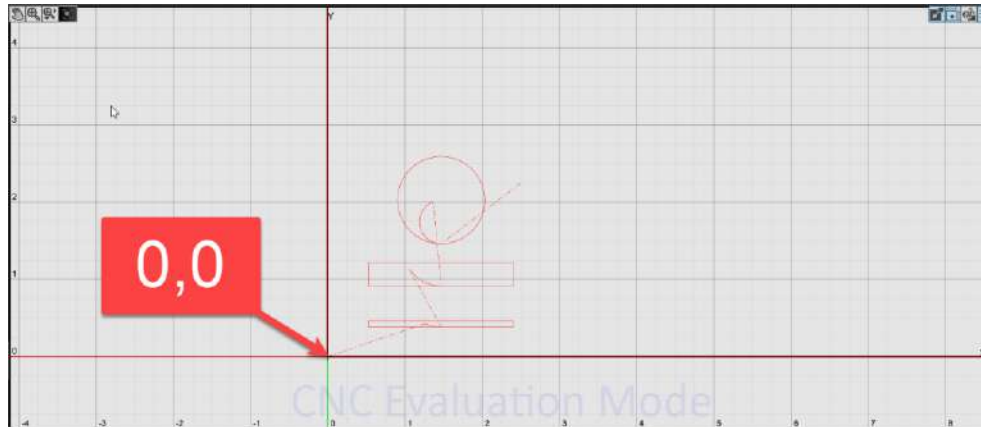




1. You have homed your machine but your image is located in a random location. You will need zero you part location.

Home sets the machine parameters, but it does not orientate the part on the machine. **Once the machine is homed you will not be able to run the machine into the stops or run it off of the table.**

Zero sets the part location located in the lower left of the screen oriented off of the torch.



- Bring torch to the lower left of the steel to be cut and hit Zero X, Zero Y and Sense PowermaxXX Zero. The part will be located off the torch tip. If you nested your part 4 inches away for the lower left corner your part will be offset 4 inches from the tip.
2. Your part is now nested on the steel to be cut. If desired depending on allotted room travel the torch around on the screen and verify that the part or parts are going to fit in allotted sheet. Used the arrow keys and the screen as a visual reference to see where the torch will travel.

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





3. **Shift Parts.** If you're going to cut outside of the allotted material by .25" then you can move the appropriate x and or y axis to a -.25 and rezero the part on the material. Shifting the parts nested by -.25. if you simply do not have enough room you will need to renest the parts on cam or resize the part.
 - **If you re-nest your parts you have essentially changed the g code needed to cut the appropriate job.**
 - **Tip- you thought the steel was 16 gauge but there is 14 on the table you do not need to start over. Go Back change the operation to 14 on ALL the parts and click the arrows to move new code into controller**
4. **Zero Z Axis.** Everything looks good? You have told the machine where **X and Y are oriented** but you now need to zero the Z height. Travel down close to material and hit Zero Z
5. Your part fits and your ready hit Run (Play button) in Lower center and you will be prompted to check settings. Hit ok and check settings. To start hit resume and your now cutting steel.
6. The torch will travel to the initial pierce point and touch off the steel again. Finding the height of the steel and offsetting the pierce height. Torch will fire and drop down to cut height and continue on path wrote by flashcut.
7. **Do not stare at the arc as it will damage your eyes.**

Plasma torch settings

Parameters are populated from **Settings** set in the **CAM** window. If an editable parameter is changed here, the settings in CAM will not be affected. Some controls will not be displayed unless they are enabled in the fabrication head configuration panel. See **Plasma** in **Configuring FlashCut**.

- **Torch On/Off** – Click to turn the torch on or off. Before turning on the torch, FlashCut CNC will display a warning dialog asking if it is safe to proceed.
- **Torch Height Control** – Click to turn THC on or off.
- **Use Sampling** – Click to turn sampling on or off.
- **Voltage** – Click up or down arrows to raise or lower the voltage.
- **Dry Run** – When selected, the machine will execute the operations, but the torch will not be lit, and cuts will not be made.
- **Material and Thickness** – These properties are visible here, but are configured in the **CAM** window under **Project Settings**.





Generally you want the Torch Height control on and the Sampling on to maintain and adjust your torch cutting height. Turned on when illuminated blue.

G-Code window

FlashCut supports ANSI standard G-Code to control machine tool movement and peripheral devices. This section describes how to create, open, and modify G-Code files, and the G-Codes supported. The G-Code window provides numerous options for creating, opening, editing, and running G-Code files. There are several ways to open or create G-Code files:

- Create a G-Code file through FlashCut CAD and FlashCut CAM. Clicking the scissors button in either of these modes generates lines of G-Code corresponding to the design specifications.
- Open an existing G-Code file created by FlashCut, another CAM program, or any other source.
- Write a G-Code program directly in the FlashCut G-Code editor.
- Double-click the G-Code window to launch the G-Code editor. The active file will be loaded.

This section describes these G-Code window features:



- | | | | |
|---|------------------------------------|----|----------------------------|
| 1 | Open G-Code file
File name | 8 | G-Code workspace |
| 2 | Load all G-Code files in a folder. | 9 | Feedrate override controls |
| 3 | Barcode | 10 | Run G-Code in reverse |
| 4 | Run CAD Import Wizard | 11 | Feed hold |
| 5 | Reset G-Code | 12 | Toggle G-Code run mode |
| 6 | Jump to Line | 13 | Run G-Code |
| 7 | Close G-Code file
Edit G-Code | | |

NOTICE # 12 When clicked this will appear Below





Run G-Code in reverse

Runs the program in reverse. This is useful when troubleshooting a program in simulation mode or dry run mode. Not typically used while actually cutting. Some commands are not possible to reverse through; For example, M106(fabhead change) and G611(advanced pierce options).

Feed hold

Click the **Feed hold** button to pause execution of the G-Code file. The machine tool stops, ramping down if necessary. The slower the ramping rate, the longer it takes from the time the **Feed hold** button is clicked to the time the tool comes to a complete stop. This button pauses any motion including automatic tool changing, tool length sensing and so on.

Toggle G-Code run mode

Click the **Plus/Minus** sign on the Feed hold button to toggle between continuous mode, and step or momentary mode.



- Run G-Code – solid arrowhead. G-Code will run to completion, unless Feed hold is pressed.



- Run Step G-Code – striped arrowhead. One line of G-Code will be executed.
- Run Momentary G-Code – outlined arrowhead. G-Code will run as long as button is depressed.

You can use this after you have had a collision. To reverse the G code and hit the “Play” or Start Button to restart the cut.

You have your tool paths created and your parts nested (lower Left as that’s your 0-X 0-Y location) -Start with something simple please-





Operating the Cut Interface

- **Touch Off** – The torch will touch the material in order to establish a zero point on the machine coordinates. Toggling **Touch Off** will make the torch touch down to reestablish the zero point between rapid and feedrate moves.
- **Start Height** – When **Touch Off at Start** is enabled with **Rapid Move to Start Height** selected, the start height will be displayed and can be set here. See **Plasma** in **Configuring FlashCut**.
- **Suppression Radius** – When **Touch Off at Start** is enabled with **Enable Suppression Radius** selected, the radius will be displayed and can be set here. See **Plasma** in **Configuring FlashCut**.
- **Kerf Crossing Detection** – When selected, helps prevent the torch from diving into the material where one cut crosses another.
- **Nozzle** – Displays information about the plasma torch nozzle. The nozzle type must be set correctly for FlashCut CNC to generate the proper toolpath.
- **Quality Level** – Specifies whether the cut will be optimized for speed or quality. Torch manufacturers often provide different cut parameter profiles.
- **Cut Height** – Specifies the distance between the torch tip and the stock material throughout a cutting move. Maintaining a precise cut height is vital to maintaining good cut quality. The cut height also provides the torch height control with a sampling height, if torch height control is active.
- **Pierce Height** – Specifies the distance between the torch tip and the stock material at the time when the torch is commanded to fire. The torch will maintain this height until the pierce delay expires, if a delay has been set.
- **Safe Height** – Specifies the height to which the torch is set during rapid moves. Also commonly referred to as the **clearance plane**, the safe Z height must be set so that the torch will retract above any parts, stock, fixtures, or other obstructions that may exist in the machine envelope. The safe Z height is called automatically before any rapid tool positioning command (G00).
- **Pierce Delay** – Specifies the delay between the command to fire the torch and the start of the next move. This delay allows the torch sufficient time to pierce the stock material completely. The delay begins when the torch fires, and there is no motion in any direction until the delay expires.
- **Voltage** – Specifies the operating voltage of the torch.
- **Amperage** – Specifies the operating amperage of the torch.
- **Pressure** – Specifies the air pressure of the torch in psi.
- **Kerf** – Specifies the width of the kerf used by the torch.

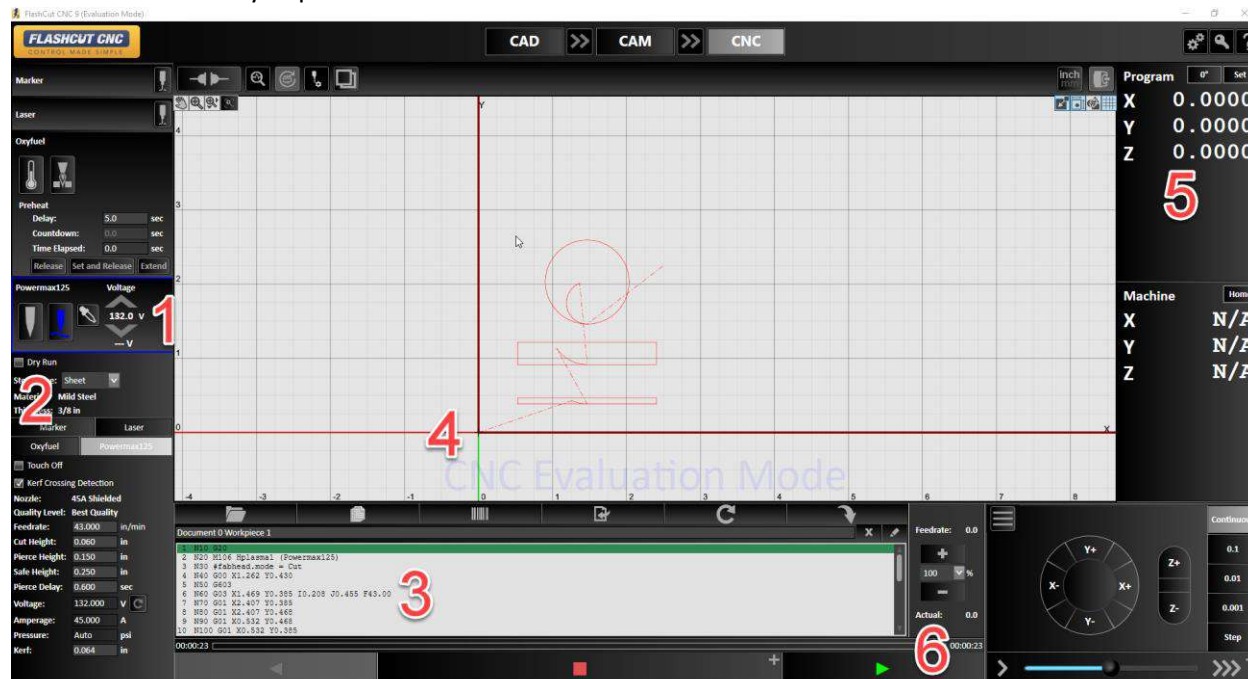
<input checked="" type="checkbox"/>	Touch Off		
	Start Height:	0.500	in
	Suppression Radius:	0.000	in
<input type="checkbox"/>	Kerf Crossing Detection		
	Nozzle:		
	Quality Level:		
	Cut Height:	0.180	in
	Pierce Height:	0.360	in
	Safe Height:	0.000	in
	Pierce Delay:	0.200	sec
	Voltage:	0.000	V
	Amperage:	0.000	A
	Pressure:	0.000	psi
	Kerf:	0.000	in





Prechecks before running a cut

It is good practice to do a quick precheck before running the machine. A few things to look for are proper amperage set and no errors on the Hypertherm power unit. Table is free of obstructions and all axis' are clear. Lastly a quick check in Flashcut as shown below.



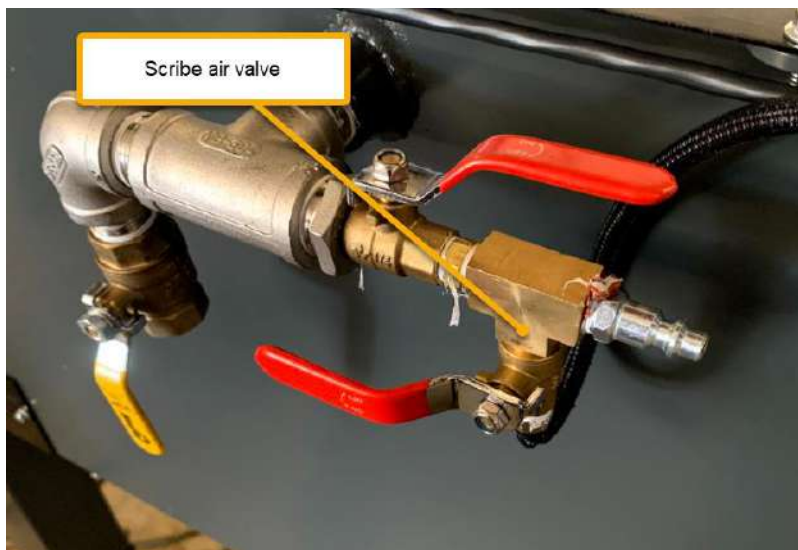
- 1- Voltage set to correct number as well as torch height control selected if desired.
- 2- Dry Run is unchecked and selected nozzle and amperage match Flashcut, machine and Hypertherm.
- 3- G-Code is at line #1.
- 4- Drawing is correct and true to the placement on the material sheet or plate.
- 5- Machine is zeroed
- 6- If all items look good hit Run G-Code!





Scribe Air Connection and Initial Setup

Your Boss Plasma Table will always require an air hookup of a minimum 90 psi when paired with a scribe. Boss Tables use a pneumatic scribe and are configured from the factory, so no adjustment is needed.

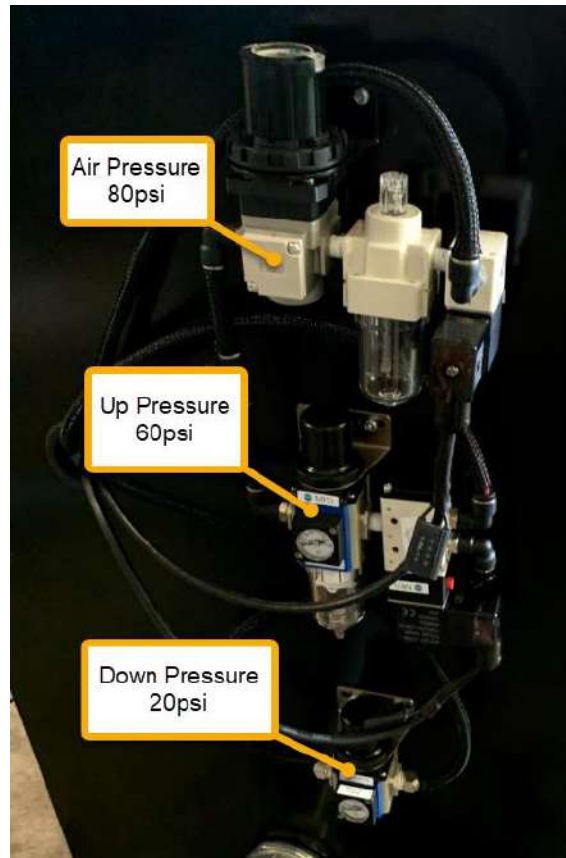


The air inlet valve is where the scribe gets it's air. A minimum of 90 psi is needed.

On the back of the control cabinet are the pressure valves for the air scribe. The correct PSI is already set from the factory so there is no need to adjust the valves. Maintain a level of air tool oil in the top valve as shown below.



Remove screw to fill the cup with air tool oil only.



Above are the air scribe valves located on the back of the Boss Tables control cabinet. The top valve with the glass cup receives the air tool oil.

Your air scribe is adjustable by turning the black band dial. This will increase or decrease the air pressure at the scribe head.



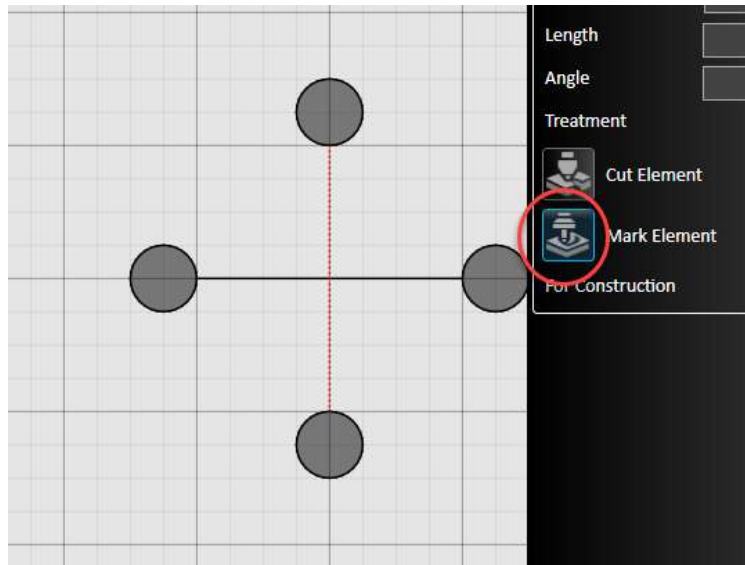
www.bosstables.com 563-380-1535 www.info@bosstables.com





Scribe Use

The scribe operation is set in the cad section of Flashcut. Simply select the object you want to scribe and select the **Mark Element** icon in the parameters to the right. The object will change from solid to dotted.



Setup from here will be the same as other cutting operations. Use the torch to zero like any other cut. The machine will compensate for the scribe offset automatically.





Router Setup and Operation

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.



Optional laser shown

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





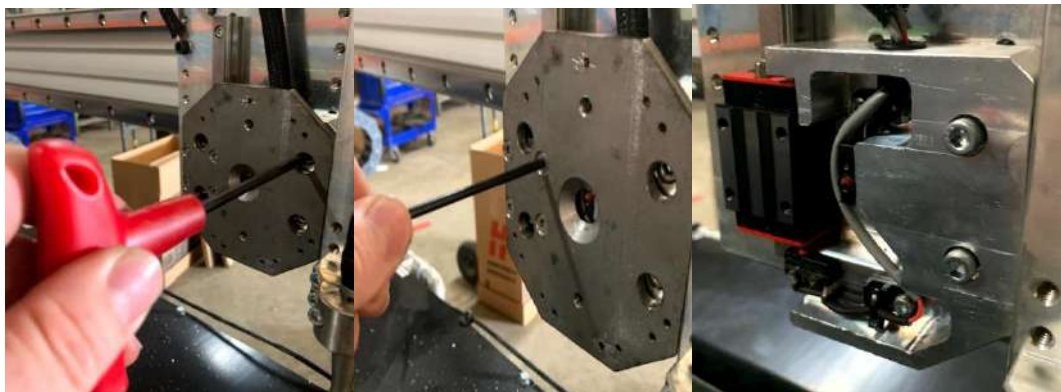
Some tables may require removal of several track clips. Using a needle nose pliers, remove these by unclipping them from the left side on the home side of the table.



Unscrew the quick connect located in the track.

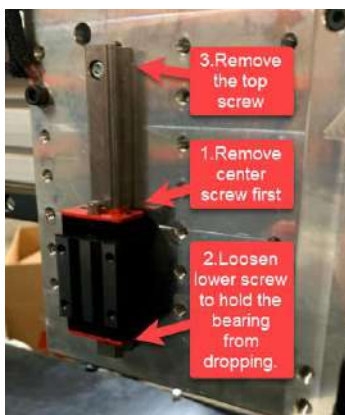


Using a **3/16" allen wrench**, remove the two screws in the torch holder and with a **T20 plus torx bit** remove the four screws on the torch holder. Older tables will have four allen screws to remove only.



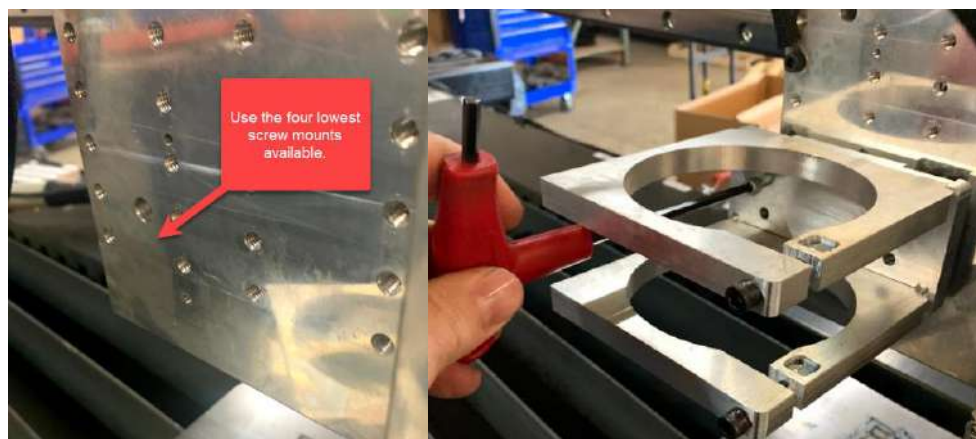


Remove the torch mount plate from the guide and set to the side. Use a T20 plus torques bit to remove the three screws holding the bearing guide with the bearing on the guide. Remove the middle screw and loosen the lower screw to hold the bearing from sliding off.



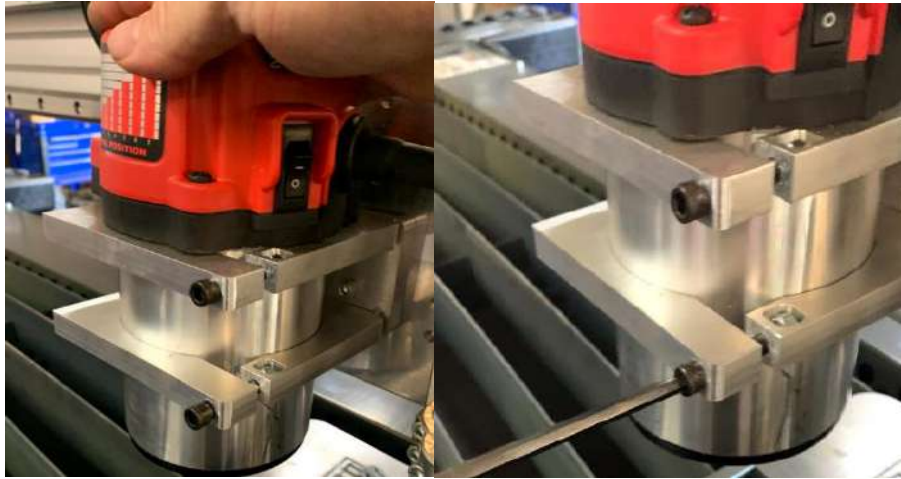
Installing Router and Mount

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.





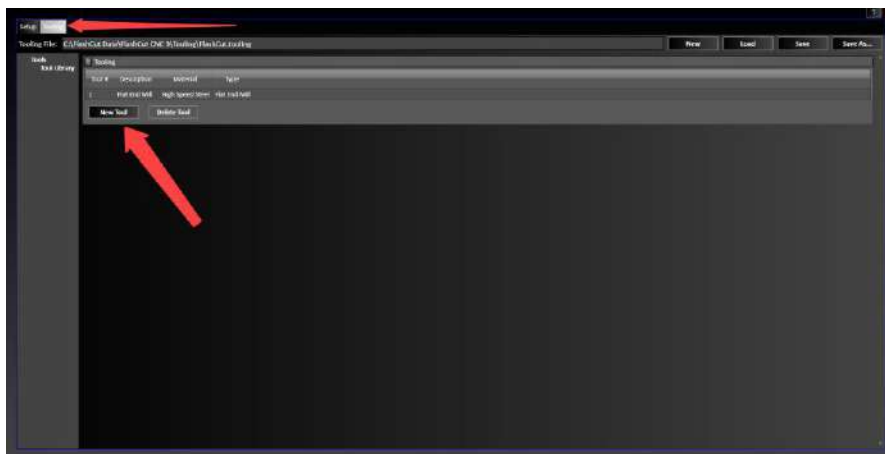
Loading a New Tool

To add a new tool or to edit a current tool select the configuration icon.

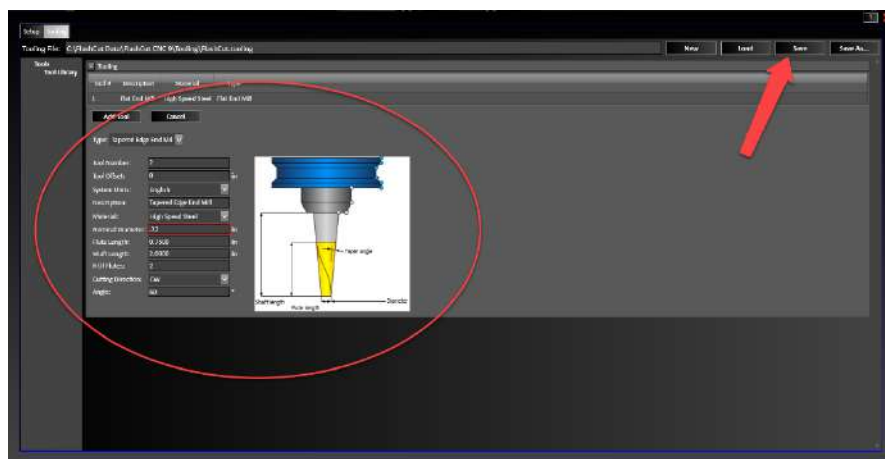




Next select tooling and New Tool. To edit a tool click on the desired tool to edit.



Insert the specifications of the tool to be added and save, then exit the tool library.



Router Software

With your computer on, open Flashcut. In the upper right corner select configuration.

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

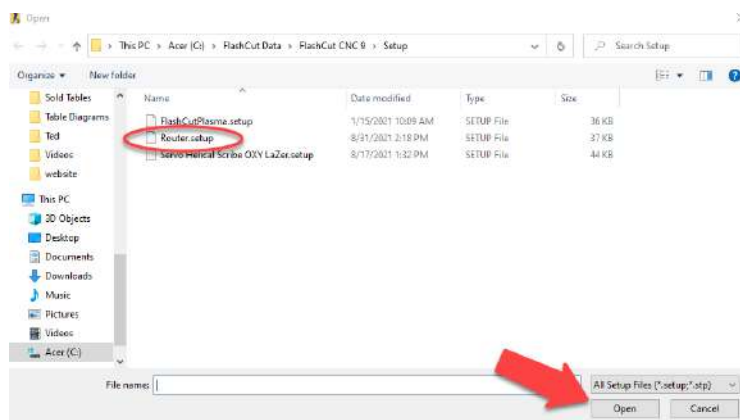




A setup screen will open. Next select Load.



You will need to load the Router.setup file. To do this click on Router.setup and click open. If your machine does not have this please call Boss Tables 563-380-1535.



And save.

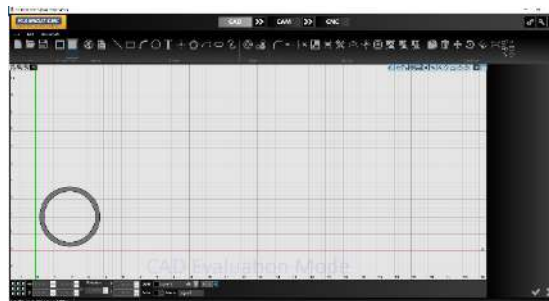


Now exit out of the setup screen.

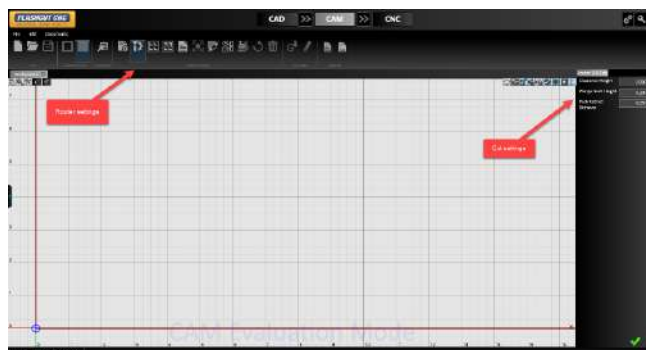
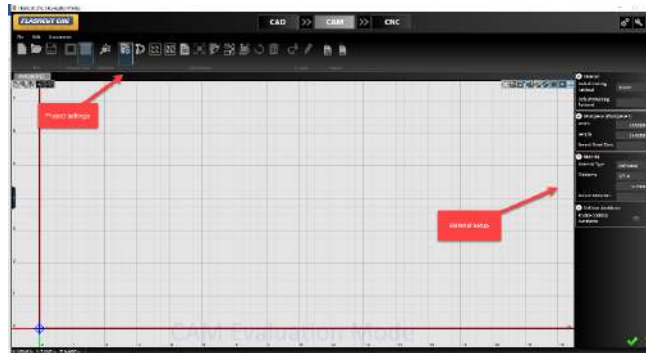




In CAD we will draw a simple round cutout. The first circle will have a 3" diameter and the second will be a 2.5" diameter.

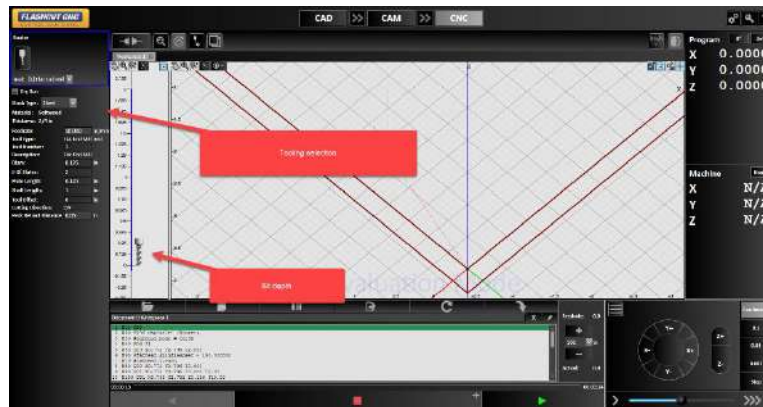


Moving to CAM we will set the material used as well as the tooling.





Once setup is accomplished, we can move to CNC. From here we will be able to select the specific tool to be used.



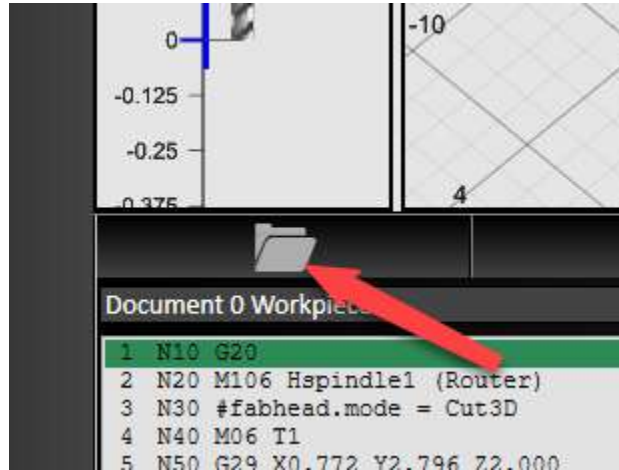
Securely fasten the materials to the table by either j hooks or by the use of a jig. Once ready, zero out the x and y axis in the desired area and manually dive the torch down to slight contact and zero the z, now click run to begin cutting.

Adding cut files with V Carve

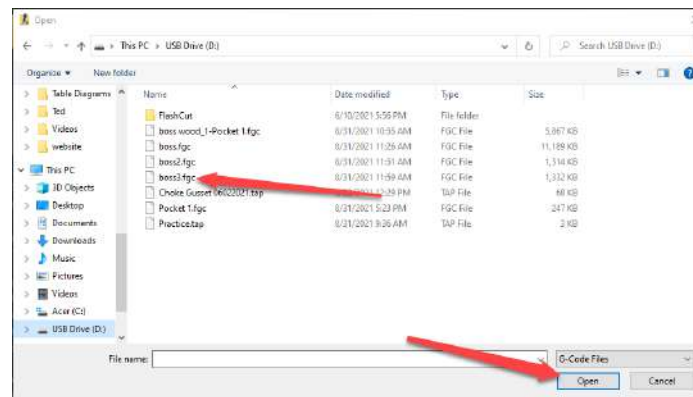
Adding a file from V Carve is done in Flashcuts CNC screen. All cut and tool settings made in V Carve apply in Flashcut. A generic tool will need to be created in Flashcut or an error will appear in the g code section.

When transferring a cut file from V Carve save the file to the computer or use a zip drive. Open the Load G-Code File icon.

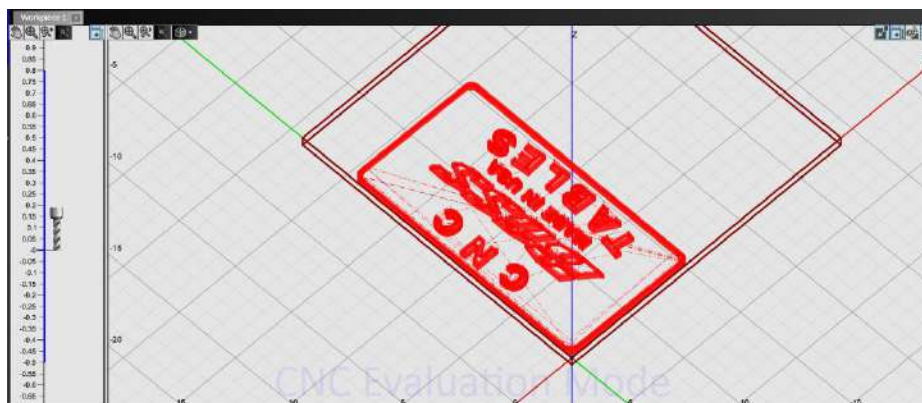




Select the desired file and open.



Now your ready to cut the file.



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





Router Bit Guide

If you want to do lettering or detailed sign making, you'll need to get a v bit. These are sometimes called v-carving bits, v-groove bits, or engraving bits. This is the only way to get a sharp grooved bottom on the inside of those roman numerals for your sundial. They are available in many sizes and angles. The most common and useful angles in order are 60deg, 90deg, and 30deg.



For intricate 3d carving, consider a tapered ball nose bit. The slight angle of the cutting edge helps reduce the appearance of tool marks parallel to your material surface.





If you are flattening large boards or are maintaining the spoil board on your CNC router, you'll want to have a spoil board cutter or flycutter bit. These bits are made to skim the surface and leave a smooth flat finish.



Endmills are used in making straight cuts in plywood. End mills come in many different diameters. You can choose an upcut or downcut.





Hypertherm cartridges	
Process	Cartridge
85 A	428934
65 A	428930
45 A	428925
FineCut® mechanized cutting	428926
Ohmic ring kit	428895 (contains 3 #420580 ohmic rings)

Mechanized cartridges can be used on hand torches if a stand-off is desired.

Capacity	Powermax65 SYNC		Powermax85 SYNC		Powermax105 SYNC	
	Thickness	Cut speed	Thickness	Cut speed	Thickness	Cut speed
Cutting						
Recommended	20 mm (3/4")	500 mm/min (20 ipm)	25 mm (1")	500 mm/min (20 ipm)	32 mm (1-1/4")	500 mm/min (20 ipm)
	25 mm (1")	250 mm/min (10 ipm)	32 mm (1-1/4")	250 mm/min (10 ipm)	38 mm (1-1/2")	250 mm/min (10 ipm)
Severance	32 mm (1-1/4")	125 mm/min (5 ipm)	38 mm (1-1/2")	125 mm/min (5 ipm)	50 mm (2")	125 mm/min (5 ipm)
Pierce*	16 mm (5/8")		20 mm (3/4")		22 mm (7/8")	
*Pierce rating for handheld use or with automatic height control						
Capacity	Metal removal rate	Groove profile	Metal removal rate	Gouge profile	Metal removal rate	Gouge profile
	Maximum removal gouging					
Typical gouge	4 kg (8.8 lbs.) per hour	2.9 mm D x 6.4 mm W (.11" D x .25" W)	8.2 kg (18.2 lbs.) per hour	3.5 mm D x 6.6 mm W (.14 D x .26 W)	8.6 kg (19.1 lbs.) per hour	6.6 mm D x 6.2 mm W (.26 D x .24 W)

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

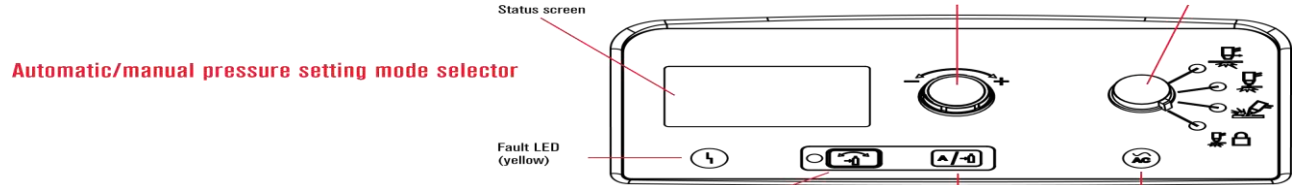




Non-continuous pilot arc Torch lock* Mode switch

Refer to your Operator Manual for instructions

***Not intended for Mechanized cutting applications**



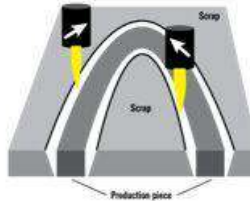
Step 4
Set the mode

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

	<p>1. Continuous pilot arc Expanded/punched metal</p>	<p>Mode switch</p>
	<p>2. Non-continuous pilot arc Plate/sheet metal</p>	<p>Mode switch</p>
	<p>3. Gouging</p>	<p>Mode switch</p>
	<p>4. Torch lock</p>	

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

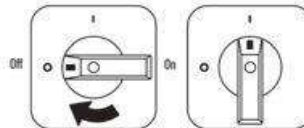
Due to the swirling action of the plasma gas, one side of the cut will always have more bevel angle. This is called the "scrap side" of the cut. The "good side" is on the right as the torch is traveling away from you. Refer to the picture on the right.



Step 5

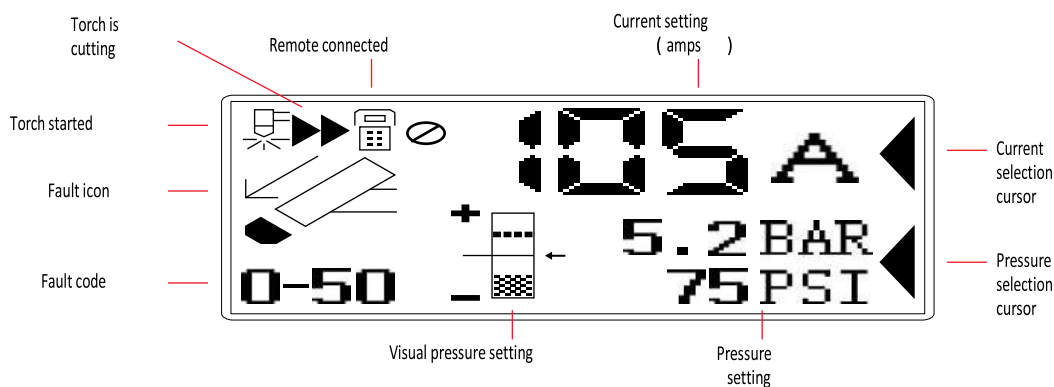
Turn on the power

- Position the power switch to ON as shown.
- Note: The cooling fan is automatic and will only operate when needed.**
- The power switch is found on the back of the system.





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



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

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

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





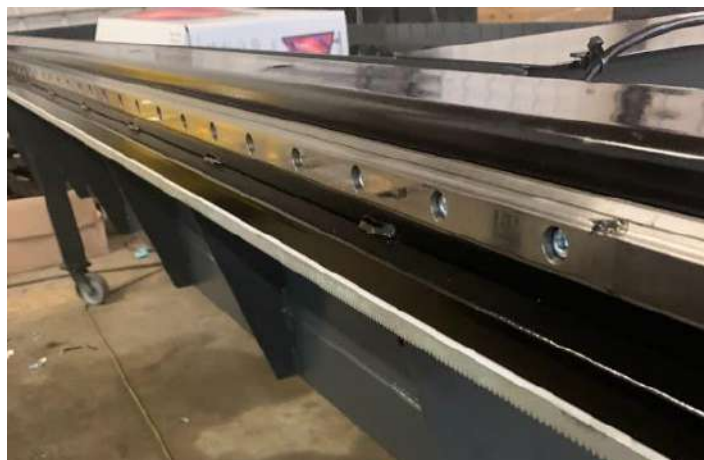
Boss CNC Table Maintenance

Care should be taken to extend the life and use of your Boss CNC Plasma Table. Periodic lubrication and cleaning is required.



Remove debris and clean gantry gear and track, spray with "Moly" Dry Film Spray Lubricant. This is located on the under sides of the machine along the rails.

Rails should be clean and free of debris at all times. Wipe with a clean rag and spray with a "Moly" Dry Film Spray Lubricant, grease Y Axis bearing zerts with EP1 grease.





Track tray should be free of debris and track should be blown off with compressed air.



Periodic greasing of the gantry is required. Grease zerks are located next to the gantry sides and z axis. **Warning do not over grease!** ¼ pump per bearing block then move the gantry and repeat until a film appears on the rail. Boss Table recommends Premium Grade lithium base lubricant.

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





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

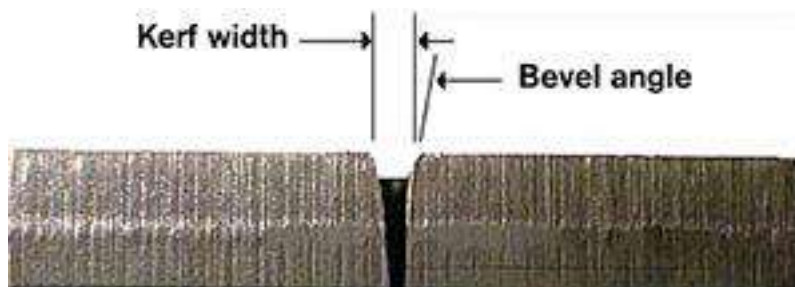




To judge the optimum cutting speed:

- 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.

When to use high-tolerance plasma

High-tolerance plasma uses a small nozzle orifice and intense gas swirl to constrict the arc. The result is an energy-dense arc with a very narrow kerf that can cut intricate details and very small holes. Conventional plasma systems can cut within 0.76 mm (.030") accuracy and produce cuts with 3–5° of bevel, sometimes as little as 1°. High tolerance systems can cut with 0.25 mm (0.010") accuracy and 0–3° of bevel. They can accurately cut holes as small as 4.76 mm (3/16").

Six rules for cutting bolt holes

1. Use the smallest nozzle size rated to pierce and cut the material
2. Make sure the pierce-delay allows full arc penetration before machine motion starts
3. Lock out voltage-regulated THC
4. Use a radiused or spiraled lead-in
5. Program a slower cutting speed
6. 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.

Refrigerated compressed air dryer



REFRIGERATED COMPRESSED AIR DRYERS

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.

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.

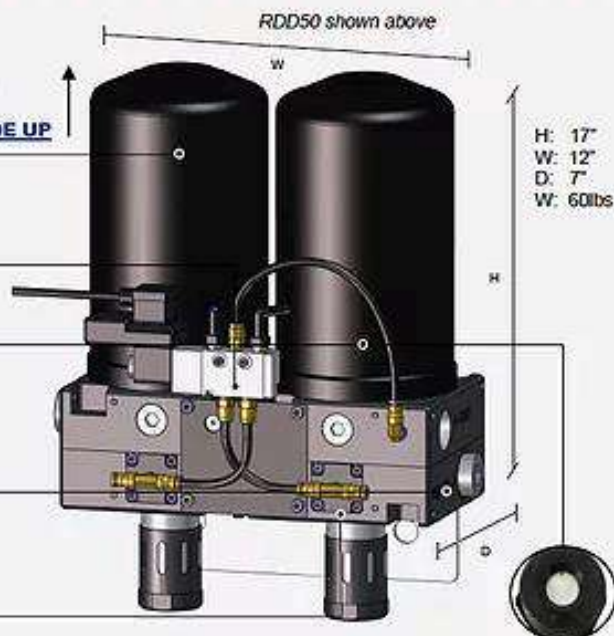
Regenerative Desiccant Air Dryers

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**

THIS SIDE UP

















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

Hypertherm Fault Codes





Fault code	Description	LED behavior			Solutions
None	The ON/OFF power switch is set to ON (I), but the Power ON LED does not illuminate.	 OFF			<ul style="list-style-type: none"> Make sure that the power cord is plugged into the receptacle. Make sure that the power is on at the main power panel or at the disconnect-power switch box. Make sure that the line voltage is not too low (more than 10% below the rated voltage for 1-phase models or 15% below the rated voltage for 3-phase models). See page 21 and page 28.
None	Low gas pressure	 ON	 ON	<ul style="list-style-type: none"> The gas pressure is below the minimum pressure for that process, mode, torch, and lead length. Check the input gas supply. See <i>Gas Pressure fault LED</i> on page 147. 	
None	No gas input	 ON	 Blinks	<ul style="list-style-type: none"> Connect the input gas supply to the plasma power supply. Turn OFF (O) then turn ON (I) the power supply. See <i>Gas Pressure fault LED</i> on page 147. 	
None	Torch stuck open (TSO) The nozzle and electrode are not touching after a start signal is received.	 ON	 Blinks slowly	<ul style="list-style-type: none"> Turn OFF (O) the power supply. Make sure that the consumables are installed correctly and that they are in good condition. See <i>Torch Cap fault LED</i> on page 148. 	
None	Torch stuck closed (TSC) The nozzle and electrode will not separate after a start signal is received.	 ON	 Blinks rapidly	<ul style="list-style-type: none"> Turn OFF (O) the power supply. Make sure that the consumables are installed correctly and that they are in good condition. See <i>Torch Cap fault LED</i> on page 148. 	
None	Power supply is over temperature or under temperature	 ON	 ON	 ON	<ul style="list-style-type: none"> The system may have overheated. Leave the plasma power supply ON to allow the fan to cool the internal components. See <i>Understand duty cycle to prevent overheating</i> on page 58. The system may be too cold to operate. If the internal temperature of the plasma power supply approaches -30°C (-22°F), move the system to a warmer location.
None	Retaining cap off	 ON	 ON	<ul style="list-style-type: none"> Turn OFF (O) the power supply. Make sure that the torch is connected to the power supply and that consumables are installed correctly. See <i>Torch Cap fault LED</i> on page 148. 	















Fault code	Description	Power LED	Fault LED	Fault icon	Solutions
0-12	Low input gas pressure or unstable gas pressure: Warning (the system continues to operate)	On	Off		<ul style="list-style-type: none"> Adjust the gas inlet pressure as needed.
0-13	AC input unstable: Warning (the system continues to operate)	Blinks (3 Hz)	Off		<ul style="list-style-type: none"> Correct the power source.
0-19	Power board hardware protection. One or more power board hardware faults (or noise) detected.	On	On		<p>The inverter shuts down and does not fire again for several seconds. If the fault is caused by electrical noise, the fault clears in a few seconds and the machine operates normally.</p> <p>If a true fault continues to occur, the 0-99 fault code appears on the operator screen. Service personnel can access the fault log in the service screen to identify the major fault.</p>
0-20	Low gas pressure	On	On		<ul style="list-style-type: none"> Check the input gas supply. Adjust the gas pressure to the acceptable range using Manual mode. See Section 4, Operation.
0-21	Gas flow lost while cutting	On	On		<ul style="list-style-type: none"> Restore the gas inlet pressure and restart the power supply. Check the torch lead for leaks or kinking.
0-22	No gas input	On	On		<ul style="list-style-type: none"> Connect the gas source and restart the power supply.
0-30	Torch consumables stuck This indicates either a "torch stuck open" or a "torch stuck closed" situation.	On	On		<ul style="list-style-type: none"> If the consumables became loose or were removed while the power supply is ON, turn OFF the power supply, correct the problem and then turn ON the power supply to clear this fault. Change consumables. If the consumables appear to be installed correctly, the torch may be damaged. Contact your Hypertherm distributor or authorized repair facility.
0-40	Over/under temperature	On	On		<ul style="list-style-type: none"> Leave the power supply on to allow the fan to cool the power supply. If the internal temperature of the power supply approaches -30° C (-22° F), move the power supply to a warmer location.









Fault code	Description	Power LED	Fault LED	Fault icon	Solutions
0-50	Retaining cap off	On	On		<ul style="list-style-type: none"> • Turn OFF the power supply. Verify that the consumables are installed and restart the power supply. • If the consumables appear to be installed correctly, the torch may be damaged. Contact your Hyperthem distributor or authorized repair facility.
0-51	Start/trigger signal on at power up This situation indicates that the power supply is receiving a start signal. It is sometimes referred to as a "stuck start."	On	On		<ul style="list-style-type: none"> • If the power supply is turned on while the torch trigger is pressed, the system is disabled. Release the trigger and recycle the power switch.
0-52	Torch not connected	On	On		<ul style="list-style-type: none"> • Plug a torch lead into the FastConnect receptacle on the front of the power supply and recycle the power switch.
0-60	AC input voltage error	On	On	 AC	<ul style="list-style-type: none"> • Phase loss: Check all input phases and fuses. • Over voltage: Check the line, decrease the voltage. • Under voltage: Check the line, increase the voltage.
0-61	AC input unstable: Shutdown	On	On		<ul style="list-style-type: none"> • The incoming line current is unstable. Power down and correct the line problem before continuing.
0-98	Internal communication failure	On	On		<ul style="list-style-type: none"> • Power down, wait 20 seconds, power up. • A qualified service technician must open the power supply case and check the ribbon cable between the control board and the DSP board.
0-99	System hardware fault — service required Indicates a major fault with the system.	On	On		<ul style="list-style-type: none"> • A qualified service technician must service the system. Contact your distributor or authorized repair facility.



Fault code	Description	LED behavior	Solutions
0-11-0	Remote controller mode invalid. Valid remote modes for this system: <ul style="list-style-type: none"> • 1 – Continuous pilot arc • 3 – Gouge 	 ON	There is a problem with the remote controller or the software interface to the system. The system cannot interpret the mode, output current, or gas pressure information coming from the controller. <ul style="list-style-type: none"> • Fix the controller. • Check the RS-485 interface cable. • Examine the programming code for incorrect process variables.
0-11-1	Remote controller current invalid. Valid remote current settings for this system: 10 – 45 A.	 ON	
0-11-2	Remote controller pressure invalid. Valid remote pressure settings for this system depend on the process, mode, torch, and torch lead.	 ON	
0-12-1	Output gas pressure low	 ON	The 0-12- <i>n</i> fault codes do not appear on the 2-digit display. They display only on a CNC via an RS-485 serial interface. An 0-12- <i>n</i> fault does not stop the system from operating. <ul style="list-style-type: none"> • Adjust the gas inlet pressure as needed. • Make sure none of the gas lines are kinked or blocked. • Run a gas test to see if the actual pressure is lower or higher than the set pressure. See <i>Run a gas test</i> on page 149. • Have a qualified service technician examine the system. Contact your distributor or authorized repair facility.
0-12-2	Output gas pressure high	 ON	
0-12-3	Output gas pressure unstable	 ON	
0-13-0	Alternating current (AC) input power unstable (system continues to operate)	 Blinks	<ul style="list-style-type: none"> • Perform a cold restart. • If applicable, disconnect the system from generator power. See <i>Generator considerations</i> on page 149. • If the fault does not clear, have an electrical technician correct the power source. See page 27.
0-51-0	Start/trigger signal on at power up This condition indicates that the power supply is receiving a start signal. It is sometimes referred to as a "stuck start."	 ON     Blinks alternately	<ul style="list-style-type: none"> • Hand torch: The torch trigger was being held in the "fire" position while the plasma power supply was being powered ON (I). Release the trigger and restart the power supply. • Machine torch: The plasma power supply was receiving a start signal when it was powered ON (I). Turn off the start signal and restart the power supply.



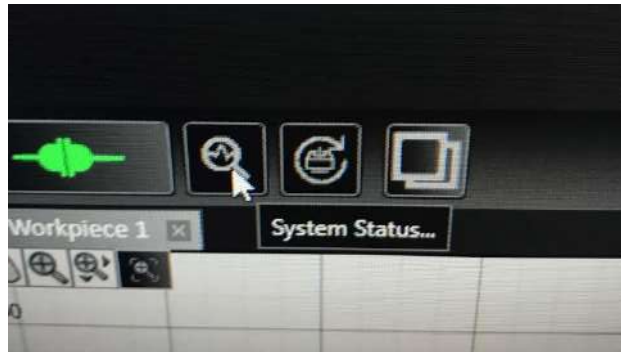
Fault code	Description	LED behavior	Solutions	
0-60-0	Alternating current (AC) input voltage phase loss	 Blinks	<ul style="list-style-type: none"> Have an electrical technician check all input phases and fuses/breakers for proper voltage at the power source and at the plasma system. If applicable, disconnect the system from generator power. See <i>Generator considerations</i> on page 149. 	
0-60-1	Alternating current (AC) input voltage too low	 Blinks	<ul style="list-style-type: none"> The input line voltage is too low (more than 10% below the rated voltage for 1-phase models or 15% below the rated voltage for 3-phase models). Have an electrical technician check the line and increase the voltage. See page 21 and page 28. If applicable, disconnect the system from generator power. See <i>Generator considerations</i> on page 149. 	
0-60-2	Alternating current (AC) input voltage too high	 Blinks	<ul style="list-style-type: none"> The input line voltage is too high (more than 10% above the rated voltage for 1-phase models or 20% above the rated voltage for 3-phase models). Have an electrical technician check the line and decrease the voltage. See page 21 and page 28. If applicable, disconnect the system from generator power. See <i>Generator considerations</i> on page 149. 	
0-61-0	Alternating current (AC) input unstable – system shutdown	 Blinks	<ul style="list-style-type: none"> The current from the incoming power line is unstable. Power down and correct the line resonance problem before continuing. Make sure the plasma system is not being used on a phase converter. If applicable, disconnect the system from generator power. See <i>Generator considerations</i> on page 149. 	
1- <i>nn-n</i> 2- <i>nn-n</i> 3- <i>nn-n</i>	Major fault	 ON	 ON	<ul style="list-style-type: none"> An internal component may be faulty. Restart the plasma power supply. In some instances, a restart can clear the fault condition. If restarting the plasma power supply does not clear the fault, a qualified service technician must service the system. Contact your distributor or authorized repair facility.





Flashcut Diagnostics

Flashcuts diagnostics (System Status) can be accessed by clicking the magnifying glass in the Flashcut program. From there you can view and address and active alerts on your system. Alerts will be have a lit up circle next to them.



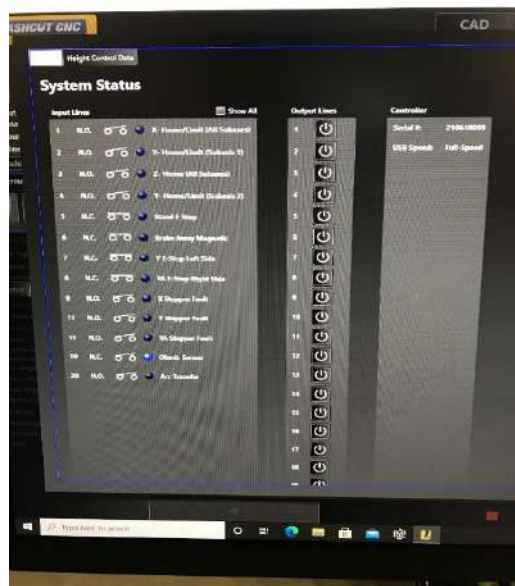
Common alert codes and fixes

Input 7 alert.....E-Stop button tripped on controller cabinet

Input 10 alert.....E-Stop button tripped on Y Axis

Input 11 alert.....E-Stop button tripped on Y Axis closest to wire tray

Input 12 alert.....Check magnetic torch breakaway



Flashcuts diagnostic screen shows the alerts and alert description. The diagnostics is showing a Ohmic Sensor alert in this case.

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





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 capacity 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

1

ACETYLENE



A LINCOLN ELECTRIC COMPANY

6290 Single Piece Cutting Tips

Medium preheat for clean surfaces

HAND/MACHINE CUTTING

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®



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®



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 WVCP & VVCM	20-24	40	12/8	4 oz. to 2 PSI	.05	#75
1/8-1/4	4/0 WVCP & VVCM	20-22	50	12/8	4 oz. to 2 PSI	.06	#68
1/4-3/8	3/0 WVCP & VVCM	18-22	75	25/8	4 oz. to 2 PSI	.07	#64
3/8-1/2	2/0 WVCP & VVCM	18-20	75	25/8	4 oz. to 2 PSI	.07	#62
1/2-3/4	0 WVCP & VVCM	15-18	90	25/8	4 oz. to 2 PSI	.08	#60
3/4-1 1/4	0 1/2 WVCP & VVCM	14-16	100	25/8	4 oz. to 2 PSI	.08	#58
1 1/4-2	1 WVCP & VVCM	13-15	100	25/10	4 oz. to 2 PSI	.09	#56
2-3	1 1/2 WVCP & VVCM	9-12	100	25/10	4 oz. to 2 PSI	.11	#54
3-4	2 WVCP & VVCM	7-9	100	25/10	4 oz. to 2 PSI	.12	#53
4-5	2 1/2 WVCP & VVCM	6-8	100	30/10	4 oz. to 2 PSI	.13	#51
5-6	3 WVCP & VVCM	5-7	100	30/10	4 oz. to 2 PSI	.14	#49
6-8	4 WVCP & VVCM	5-7	100	30/10	4 oz. to 2 PSI	.16	#45
8-9	5 WVCP & 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.

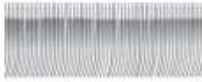
For additional gas flow information refer to our website at www.harrisproductsgroup.com and our equipment catalog.

Additional copies are available at www.harrisproductsgroup.com





MACHINE CUTTING GUIDE



PERFECT CUT - Regular surface with slightly sloping drag lines marks a perfect cut. A slight amount of scale at the top of the cut is caused by preheat flames and is easily removed. This surface can be used for many purposes without machining.



PRODUCTION CUT - Moderately sloping drag lines and a reasonably smooth surface characterize a production cut. For production operations a cut of this type represents the best combination of quality and economy.



DIRTY TIP - Dirt or scale in the tip will deflect the oxygen stream and cause one or more of the following problems: Excess slag on the steel, an irregular cut surface, pitting and undercutting.

CUTTING SPEED



EXTREMELY FAST - Rake angle of drag lines shows extremely fast cutting speed. Top edge is good and cut face is smooth. However, slag adheres to the bottom side and there is danger of losing the cut. Not enough time is allowed for slag to blow out of the kerf. Cut face often slightly concave.



EXTREMELY SLOW - Pressure marks indicate too much oxygen for cutting conditions. Either the tip is too big, cutting oxygen pressure too high, or speed is too slow as shown by a rounded or beaded top edge as in this case. As oxygen volume nears correct proportions, pressure marks appear closer to the bottom edge until they finally disappear.



SLIGHTLY TOO FAST - Drag lines incline backwards, but a "drop cut" is still attained. Top edge is good, cut face is smooth and slag free. Quality is satisfactory for much production work.



SLIGHTLY TOO SLOW - Cut is high quality although there is some surface roughness caused by vertical drag lines. Top edge is usually slightly beaded. Quality is generally acceptable, but faster speeds are more desirable.



The Harris Products Group • A Lincoln Electric Company
2345 Murphy Blvd., Gainesville, GA 30504
Phone: 1.800.241.0804 • FAX: 1.800.840.8684

PN: 9500593 REV. D
03/2014

TIP DISTANCE



TOO CLOSE - Grooves and deep drag lines caused by unstable cutting action. Part of preheat cone burns inside kerf where normal gas expansion deflects oxygen cutting stream.



TOO HIGH - Top edge is beaded or rounded, cut face is not smooth and often is slightly beveled when preheat effectiveness is partially lost due to the tip being held too high. Cutting speed is reduced because of the danger of losing the cut.

GAS ADJUSTMENT



TOO MUCH CUTTING OXYGEN - Pressure marks are caused by too much cutting oxygen. When more oxygen is supplied than can be consumed in oxidation, the remainder goes around the slag creating gouges, or pressure marks. Correct this fault by lowering cutting oxygen pressure, increasing speed, or using a smaller tip. As oxygen volume nears correct proportion, pressure marks appear closer to the bottom edge until they finally disappear.



TOO HOT PREHEAT - Rounded top edge caused by too much preheat. Excess preheat does not increase cutting speed. It only wastes gases.

WHAT TO LOOK FOR IN BEVEL CUTTING



GOOD QUALITY - Top edge is excellent and cut face extremely smooth. Slag should be easy to remove and the cut part dimensionally accurate. Cutting speed is slower than vertical cutting because preheat effect is partially deflected from plate.



POOR QUALITY - Gouging is the most common fault, and is caused by either speed too fast or preheat flame too mild. Another fault is a rounded top edge, caused by too much preheat indicating excessive gas consumption.





Notes/ Changes Made

